DOCUMENT RESUME

ED 226 209 CE 035 109

TITLE	Machinist, 13-3. Military Curriculum Materials for Vocational and Technical Education.
INSTITUTION	Chanute AFB Technical Training Center, Ill.; Ohio State Univ., Columbus. National Center for Research in Vocational Education.
SPONS AGENCY	Office of Education (DHEW), Washington, D.C.
PUB DATE	78
NOTE	240p.
PUB TYPE	Guides - Classroom Use - Materials (For Learner) (051)
EDRS PRICE	MF01/PC10 Plus Postage.
DESCRIPTORS	Autoinstructional Aids; Behavioral Objectives; Hand Tools; High Schools; Individualized Instruction; Learning Activities; *Machinery Industry; *Machine Tools; *Machinists; Manufacturing; Metal Industry; Metal Working; Pacing; Postsecondary Education; Secondary Education; Tests; Textbooks; *Trade and Industrial Education; Workbooks
IDENTIFIERS	Military Curriculum Project

ABSTRACT

These three volumes with text and workbook materials for a secondary/postsecondary level course in basic and advanced machine shop practices comprise one of a number of military-developed curriculum packages selected for adaptation to vocational instruction and curriculum development in a civilian setting. The purpose stated for the individualized, self-paced course is to upgrade an apprentice semi-skilled machinist to a skilled level. The course is also intended for use in a laboratory or on-the-job learning situation. It can be used as independent or advanced study in machine shop courses. Each volume consists of (1) a coded text containing informative material, (2) a workbook detailing chapter objectives and providing review exercises where questions are keyed to the text (no answers are provided), and (3) a volume review exercise where questions are also keyed to the text, but no answers are available. The three volumes cover general shop management (precision tools; lubricants, coolants, and fluids; metallurgy; and stud, plug, screw, and insert, removal, and replacement), advanced machine work (drilling machines, lathes, milling machines, shapers, contour machines, and grinding operations), and tool design and fabrication. (YLB)

MILITARY CURRICULUM MATERIALS

The military-developed curriculum materials in this course package were selected by the National Center for Research in Vocational Education Military Curriculum Project for dissemination to the six regional Curriculum Coordination Centers and other instructional materials agencies. The purpose of disseminating these courses was to make curriculum materials developed by the military more accessible to vocational educators in the civilian setting.

The course materials were acquired, evaluated by project staff and practitioners in the field, and prepared for dissemination. Materials which were specific to the military were deleted, copyrighted materials were either omitted or approval for their use was obtained. These course packages contain curriculum resource materials which can be adapted to support vocational instruction and curriculum development.

The National Center Mission Statement

The National Center for Research in Vocational Education's mission is to increase the ability of diverse agencies, institutions, and organizations to solve educational problems relating to individual career planning, preparation, and progression. The National Center fulfills its mission by:

- Generating knowledge through research
- Developing educational programs and products
- Evaluating individual program needs and outcomes
- Installing educational programs and products
- Operating information systems and services
- Conducting leadership development and training programs

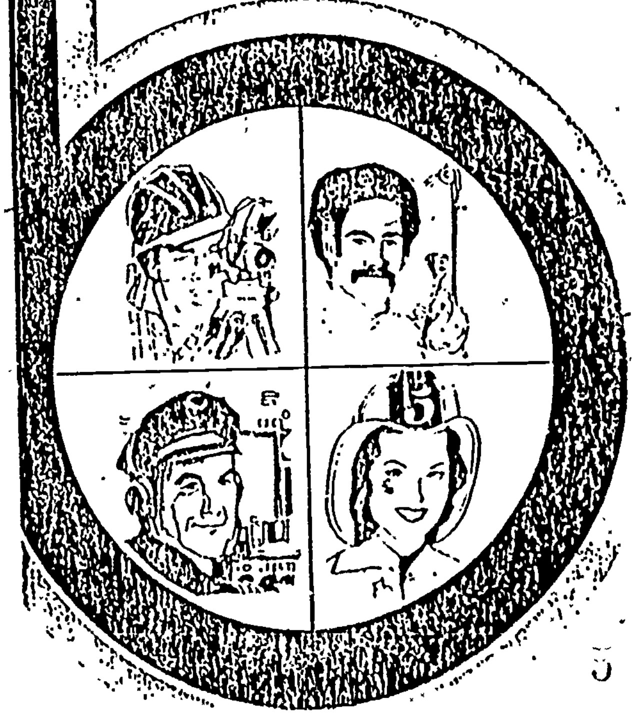
FOR FURTHER INFORMATION ABOUT
Military Curriculum Materials
WRITE OR CALL
Program Information Office
The National Center for Research in Vocational Education
The Ohio State University
1960 Kenny Road, Columbus, Ohio 43210
Telephone. 614/486-3655 or Toll Free 800/848-4815 within the continental U.S. (except Ohio)

THE NATIONAL CENTER
FOR RESEARCH IN VOCATIONAL EDUCATION

Military Curriculum Materials for Vocational and Technical Education

Information and Field Services Division

The National Center for Research in Vocational Education



Military Curriculum Materials Dissemination Is . . .

an activity to increase the accessibility of military developed curriculum materials to vocational and technical educators.

This project, funded by the U.S. Office of Education, includes the identification and acquisition of curriculum materials in print form from the Coast Guard, Air Force, Army, Marine Corps and Navy.

Access to military curriculum materials is provided through a "Joint Memorandum of Understanding" between the U.S. Office of Education and the Department of Defense.

The acquired materials are reviewed by staff and subject matter specialists, and courses deemed applicable to vocational and technical education are selected for dissemination.

The National Center for Research in Vocational Education is the U.S. Office of Education's designated representative to acquire the materials and conduct the project activities.

Project Staff:

Wesley E. Budke, Ph.D., Director
National Center Clearinghouse

Shirley A. Chase, Ph.D.
Project Director

What Materials Are Available?

One hundred twenty courses on microfiche (thirteen in paper form) and descriptions of each have been provided to the vocational Curriculum Coordination Centers and other instructional materials agencies for dissemination.

Course materials include programmed instruction, curriculum outlines, instructor guides, student workbooks and technical manuals.

The 120 courses represent the following sixteen vocational subject areas:

- Agriculture
- Aviation
- Building & Construction Trades
- Clerical Occupations
- Communications
- Drafting
- Electronics
- Engine Mechanics
- Food Service
- Health
- Heating & Air Conditioning
- Machine Shop
- Management & Supervision
- Meteorology & Navigation
- Photography
- Public Service

The number of courses and the subject areas represented will expand as additional materials with application to vocational and technical education are identified and selected for dissemination.

How Can These Materials Be Obtained?

Contact the Curriculum Coordination Center in your region for information on obtaining materials (e.g., availability and cost). They will respond to your request directly or refer you to an instructional materials agency closer to you.

CURRICULUM COORDINATION CENTERS

EAST CENTRAL
Rebecca S. Douglass
Director
100 North First Street
Springfield, IL 62777
217/782-0759

NORTHWEST
William Daniels
Director
Building 17
Airdustrial Park
Olympia, WA 98504
206/753-0879

MIDWEST
Robert Patton
Director
1515 West Sixth Ave.
Stillwater, OK 74704
405/377-2000

SOUTHEAST
James F. Shill, Ph.D.
Director
Mississippi State University
Drawer DX
Mississippi State, MS 39762
601/325-2510

NORTHEAST
Joseph F. Kelly, Ph.D.
Director
225 West State Street
Trenton, NJ 08625
609/292-6562

WESTERN
Lawrence F. H. Zane, Ph.D.
Director
1776 University Ave.
Honolulu, HI 96822
808/948-7834

MACHINIST

Table of Contents

MACHINIST

Correspondence Course 13-3

Developed by:

United States Air Force

Development and Review Dates

Unknown

Occupational Area:

Machine Shop

Cost:

Print Pages:

234

Availability:

Military Curriculum Project, The Center for Vocational Education, 1960 Kenny Rd., Columbus, OH 43210

Suggested Background:

Apprentice (semi-skilled level) machinist

Target Audiences:

Grades 12-adult

Organization of Materials:

Student workbooks containing lesson objectives, assignments, review exercises with answers and volume review exercises; text

Type of Instruction:

Individualized, self-paced

Type of Materials:		No. of Pages:	Average Completion Time:
Volume 1 —	*General Shop Management*	34	Flexible
	Workbook	30	
Volume 2 —	*Advanced Machine Work*	73	Flexible
	Workbook	30	
Volume 3 —	*Tool Design and Fabrication*	39	Flexible
	Workbook	23	

Supplementary Materials Required:

None

Expires July 1, 1978

Course Description

This course is designed to upgrade an Apprentice (semi-skilled) Machinist to a Machinist (skilled) level. It provides information about basic and advanced machine shop practices. The major duties of the machinist are as follows:

- Manufactures and reworks machined parts
- Assembles and fits machined parts
- Maintains hand and machine tools
- Supervises machine shop personnel

Machinist contains three volumes covering advanced operations and some supervisory aspects of the machine shop.

Volume 1 — *General Shop Management* discusses precision tools, lubricants, coolants, and fluids, metallurgy, and stud, plug, screw and insert removal and replacement. The first chapter dealing with shop administration was deleted because of its reference to specific military organization and forms.

Volume 2 — *Advanced Machine Work* discusses drilling machines, lathes, milling machines, shapers, contour machines, and grinding operations.

Volume 3 — *Tool Design and Fabrication* covers tool design practices, special tool design and fabrication, and fitting and assembly

The chapters in each volume are organized around objectives accompanied by a coded text, review exercises and answers to the exercises keyed to the text. The course was designed for student self-study and evaluation, but would be most useful in a laboratory or on the job learning situation. Volume review exercises with questions keyed to the text are provided, but the answers are not available. This course can be used as independent and/or advanced study in machine shop courses.

53150 01 7206

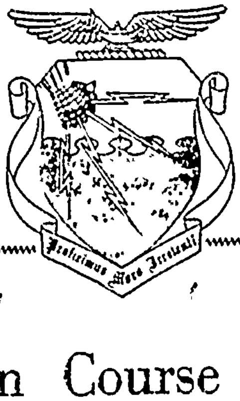
CDC 53150

MACHINIST

(AFSC 53150)

Volume 1

General Shop Management



13-3

Extension Course Institute

Air University

DEPARTMENT OF THE AIR FORCE
UNITED STATES OF AMERICA

PREPARED BY
DEPARTMENT OF WEAPON SYSTEMS SUPPORT TRAINING
3345th TECHNICAL SCHOOL (ATC)
CHANUTE AFB, ILLINOIS

EXTENSION COURSE INSTITUTE, GUNTER AIR FORCE BASE, ALABAMA

Page 9, para 3-18-m. Delete.
Para 3-18-o. Change "1098, Personnel Action Request" to "2095, Assignment/Personnel Action."
Para 3-18-p & q. Delete.
Para 3-18-r. Add "Retain VRE answer sheets."

Page 9, para 3-19, line 1. Change "Shop" to "Immediate." Lines 2 & 3: Change "a shop" to "the immediate." Line 9. Change "shop" to "immediate."
Line 11. Change "shop" to "immediate."
Para 3-19-a. Delete "the best."
Para 3-19-c. Delete.

Page 10, para 3-19-e. Delete.
Para 3-19-f. Delete "and Conducting OJT."
Para 3-19-g. Change "the program through daily supervision" to "the OJT program through daily contact."
Para 3-19-j. Delete.
Para 3-19-k. Change "For airmen" to "For assigned airmen."
Para 3-19-m. Delete first line and replace with "Initiate AF Form 2096 to." Line 3. Add "OJT" after "from)."

Page 10, para 3-20-c, d, e, f, g and h. Delete and replace with the following: "c. Teach theory and background information when required. d. Motivate and evaluate assigned trainees."

Page 10, para 3-23-a. Delete and replace with "Know where to locate and read his AFS description, his immediate training requirements and objectives, and his training folder and its contents."
Para 3-23-c. Delete "understand."
Para 3-23-e. Add "for future use."
Para 3-23-f. Change "trainer and supervisor" to "OJT administrator."

Page 11, line 1. Change "supervisor" to "OJT administrator."

Page 11, para 3-25-b, line 5. Change "freindly" to "friendly."

Page 12, para 3-31, line 4. Change "supervisor" to "administrator."

Page 13, para 3-31-c, line 4. Change "supervisor" to "administrator."

Page 13, para 3-33, lines 2 & 9. Change "supervisor" to "administrator."

Page 13, para 3-34, lines 2, 10, 12 & 14. Change "supervisor" to "administrator." Line 15. Change "wheterh" to "whether."

Page 14, para 3-37, line 5. Change "supervisor" to "administrator."

53150 00 S01 7505

CHANGE SUPPLEMENT

CDC 53150

MACHINIST

(AFSC 53150)

IMPORTANT: Make the corrections indicated in this change supplement before beginning study of Volumes 1, 2 and 3. This change supplement is printed on one side only to permit "cut and paste" posting of lengthy changes, in case you prefer to post these changes in this manner.

CHANGES FOR THE TEXT: VOLUME 1

Page iii, para 4, line 3. Change "(TTOC)" to "(TTOXC)."

Page iv, Contents, line 7. Change "53" to "52." Line 8. Change "60" to "59." Bibliography. Change "67" to "66."

Page 3. Delete Chart 1 and replace it with new Chart 1.

Page 8, para 3-13, line 1. Change "Consolidated" to "On-the-Job." Line 9. Delete "This would . . . duplicate training." Line 14. Change "consolidated" to "On-the-Job."

Page 8, para 3-15, line 4. Delete "although . . . location."

Page 9, line 1. Change "consolidated" to "on-the-job."

Page 9, para 3-16. Delete line 16.

Page 9, para 3-18, line 1. Change "Supervisor" to "Administrator." Line 2. Change "supervisor" to "administrator." Line 6. Change "Teach" to "Indoctrinate."

Para 3-18-d. Delete and replace with "Enroll trainee in CDC specified in ECI catalog."

Para 3-18-h. Delete.

Para 3-18-i. Change "consolidated" to "on-the-job."

Para 3-18-j. Delete.

Para 3-18-k & l. Delete and replace with "k. Make visits to individual sections at least every 90 days, and forward written report, through squadron commander, to sections visited and submit a copy to CBPO OJT unit."

6

Contents

5

Preface.

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

Note the chapter titles on the contents page for Volume 1. Chapter 1 covers shop administration; chapter 2, precision tools; chapter 3, lubricants, coolants, and fluids; chapter 4, metallurgy; and chapter 5, stud, plug, screw, and insert removal and replacement. 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 see 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 the preparing agency 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 21 hours (7 points).

Material in this volume is technically accurate, adequate, and current as of February 1972.

10

9

Page 14, para 3-38, lines 3 & 7. Change "supervisor" to "administrator."

Page 14, para 3-40, line 1. Change "supervisor" to "administrator."

Page 14, para 3-41-b, line 3. Change "supervisor" to "administrator."

Page 14, para 3-42, line 5. Change "supervisor" to "administrator."

Page 17, para 4-26, line 6. Change "fight" to "flight."

Page 19, para 4-35, line 2. Change "amd" to "and."

Page 21, para 4-55-a, line 1. Change "TI 0-1-02" to "TO 0-1-02." Line 3. Delete "TO 0-2-1."

Page 22, para 4-66, line 10. Change "eixst" to "exist."

Page 23, para 4-77, line 2, change "To" to "TO."

Page 24, para 5-2, line 2. Change "of it" to "or if."

Page 39, para 8-5, line 10. Change "minue" to "minus."

Page 48, para 10-7, line 6. Change "flim" to "film."

Page 57, para 18-2, line 8. Change "536XO" to "531X5."

Page 62, para 20-8, line 4 from bottom. Delete "the."

Page 62, para 21-2, line 3. Change "of" to "or."

Page 66, lines 2 thru 7. Delete.

Page 66, Department of the Air Force Publications, line 2. Change "Safeguarding Classified Information" to "Information Security Program." Line 4. Change "AFM" to "AFR" and "Industrial Safety" to "Ground" and delete "26 June 1970."

Page 66, Resident Courses. Delete entire paragraph.

CHANGES FOR THE TEXT: VOLUME 2

Page iii, para 4, line 3. Change "(TTOC)" to "(TTOXC)."

Page 4, para 2-6, line 8. Change "crill" to "drill."

Page 4, para 2-7, line 2. Change "still" to "steel."

Page 8, para 4-17, line 2. Change "tapes" to "tapers."

Page 36, col 2, line 4 from bottom. Add "co-" to end of line. Next to last line. Change "cos A" to "cot A."

Page 37, col 2, first line. Change "cos A" to "cot A."

Page 39, last formula. Change "ȼØ$" to "cos."

Page 69, Department of the Air Force Publications, line 2. Change "AFM" to "AFR" and "Industrial Safety" to "Ground" and delete "26 June 1970."

Page 69, Resident Course. Delete entire paragraph.

CHANGES TO THE TEXT: VOLUME 3

Page iii, para 4, line 3. Change "TTOC" to "TTOXC."

Page 28, para 9-37, line 2. Change "one one" to "on one."

Page 31, para 11-1, line 8. Change "536XX" to "531X5."

Page 35, Department of the Air Force Publications, line 2. Change "AFM" to "AFR" and "Industrial Safety" to "Ground."

Page 35, Resident Courses. Delete the entire paragraph.

11

MODIFICATIONS

Chapter 1 of this publication has (have) been deleted in adapting this material for inclusion in the "Trial Implementation of a Model System to Provide Military Curriculum Materials for Use in Vocational and Technical Education." Deleted material involves extensive use of military forms, procedures, systems, etc. and was not considered appropriate for use in vocational and technical education.

12

Precision Tools

YOU ARE BECOMING more proficient in the operation of various machine tools. There are other related skills in which you must become proficient, also. Among these skills are the use of machinist's special tools. Also, you must know how to use measuring devices and processes and how to care for precision measuring instruments. These skills will be very useful to you.

7. Machinist Special Tools

7-1. When you attended the resident machinist course, or as you studied Volume I of the 53130 CDC, you became familiar with many common handtools. You have learned to apply the knowledge you gained in the selection and use of the tools. Now, you will learn the purposes and uses of tools of a more special nature.

7-2. **Bench Plate.** The bench plate is a plate machined to a flat surface. Bench plates vary in size and are usually machined from cast iron. An average-sized bench plate may be about 18 inches square and about 1 1/2 inches thick. You use the bench plate on the bench as a base upon which to work. You use the bench plate for ordinary layout work when great accuracy is not required. You can lay the work directly on the plate, clamp it to angle plates, or hold it on "V" blocks.

7-3. **Surface Plate.** You use a surface plate when a flat surface with a high degree of accuracy is required. Surface plates are made of close grained cast iron, granite, or ceramic material. After being machined accurately, the surface is hand scraped flat and smooth. Surface plates are available in various sizes ranging from a few inches in width and length to 8 feet wide and 10 feet long. Ribs are cast into the underside of the plate to prevent warpage of the flat surface.

7-4. You use the surface plate for making accurate layout, as well as for ordinary layout work. You use it in conjunction with precision measuring instruments to check the accuracy of machined parts. You also use it when you check the accuracy of gages and measuring instruments. A good surface plate is an expensive piece of equipment. The cost of a particular surface plate depends upon the material from which it is made, its size, and its accuracy. You must give the surface plate proper care if it is to maintain its accuracy. The working surface must be kept clean and free of foreign matter. Do not drop any object on its surface. Do not hammer or pound on a surface plate. Keep the surface lightly oiled and covered to prevent damage when the plate is not in use.

7-5. **Parallels.** Parallels are accurately machined bars of steel or cast iron. Opposite surfaces are parallel with adjacent surfaces at right angles. You use parallels for a variety of work when the work must be raised from a surface while you maintain parallelism with the surface. Parallels are manufactured in a variety of types and sizes.

7-6. *Solid parallels.* Solid parallels are accurately machined from solid stock and are usually hardened. They are nonadjustable and come in pairs. Solid parallels may be purchased from most precision gage manufacturers. These parallels are hardened, ground, and lapped accurately to standard widths and thicknesses. Their accuracy is sufficiently close to permit you to use them in many precision measuring and gaging processes. Many times you will find you don't have the required sizes of parallels available. You can manufacture sets of parallels to meet specific needs. Often you can make parallels from soft steel to aid in the setup of many jobs. These parallels, however, would not have the accuracy required for precision measuring processes. You can manufacture parallels from tool steel and grind them to a fair degree of accuracy.

7-7. *Box parallels.* Box parallels are similar to solid parallels. The box parallels are larger and hollow in design. They are hollow to reduce the weight and to allow ease of

21

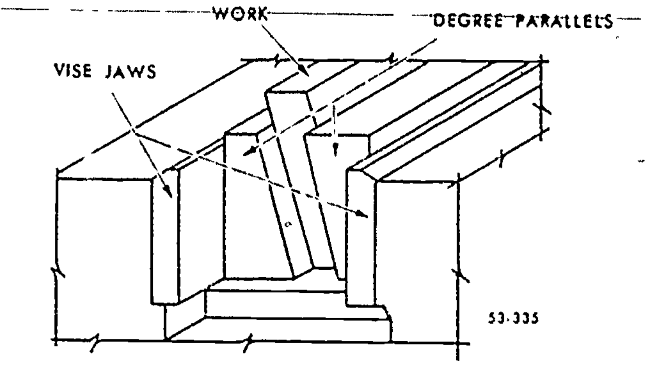


Figure 5 Degree Parallels.

handling. Again, they are manufactured with opposite surfaces parallel and adjacent surfaces square. You use box parallels for larger work or when the work has to be raised greater distances.

7-8. *Degree or angular parallels.* Degree parallels are similar to solid parallels, except that one side of each parallel is machined to an angle other than 90° to the adjacent sides. Figure 5 shows that degree parallels are manufactured in pairs with specific angles. Both parallels are manufactured with their faces at the same angle.

7-9. *Magnetic parallels.* Magnetic parallels are parallels designed to transmit magnetism. They are ground and lapped with the opposite sides square. Magnetic parallels are constructed of fused laminations of iron and brass. These laminations are in precision alinement to produce high permeability and low residual magnetism. These parallels are not magnetic in themselves but the high permeability allows a high degree of magnetism to be transmitted through the parallel. The low residual magnetism allows the parallel to quickly lose its magnetic properties when the source of magnetism is removed. You use magnetic parallels with various magnetic chuck applications. You would use magnetic parallels to provide positive grip for small, thin, or irregular work that cannot be held directly on the magnetic chuck.

7-10. *Adjustable parallels.* Adjustable parallels are composed of two wedges. The wedges are dove-tailed together so that one part may slide along the other. When one part slides along the other, the distance between the two opposite surfaces changes. The distance may be changed and locked at any micrometer setting within the limits of the parallel. You use adjustable parallels in connection with milling machine, shaper, and other machine tool vises when you are setting up work. This application is similar to the one using solid parallels. You would also use them when you are leveling work on various machine tool tables. Another use is that of an adjustable gage for checking slots.

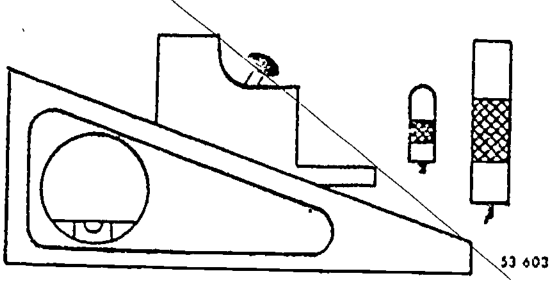


Figure 6 Planer Gage

7-11. **Planer Gage.** The planer gage, figure 6, like the adjustable parallel, has inclined planes. When the two parts are moved relative to each other the distances between the surfaces of the gage changes. The distance between the surfaces may be set to exact distances using a micrometer, surface gage, or vernier height gage. The planer gage was designed primarily to assist the planer or shaper operator in setting the cutting tool to the proper depth You could save much time and effort in operating the shaper by using this gage. It could eliminate the slow method of cut and try.

7-12. You set the gage to the finished thickness of the work with a micrometer or other gage, as mentioned. With the planer gage resting upon the machine platen, you then adjust the cutting tool in contact with the gage. The first cut can then be relied upon to give the proper thickness of work. You can also use the planer gage to check the width of slots and grooves. You can accurately check the width of any slot within the limits of the gage. Also, you can use the planer gage in conjunction with a sine bar when you are doing angular work.

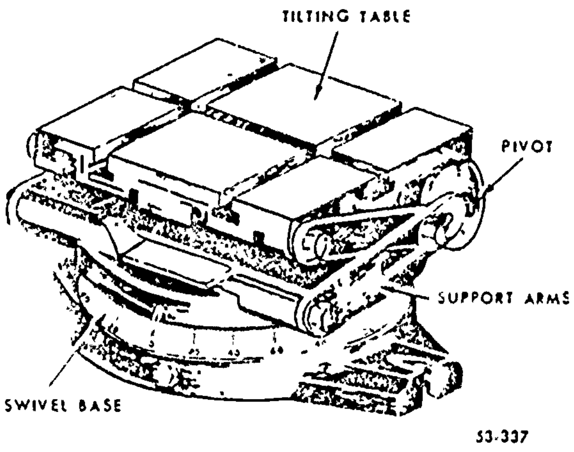


Figure 7. Adjustable angle Plate

7-13. **Angle Plates.** Angle plates are

Figure 8

composed of two members. The outer surfaces are machined flat. There are two types of angle plates: (1) the standard (nonadjustable) and (2) the adjustable. In the nonadjustable, the faces are machined to right angles with each other. The adjustable angle plate allows the angle between the two surfaces to be changed.

7-14 *Standard angle plate.* The standard angle plate has two or more surfaces machined flat and perpendicular to each other. This allows you to mount work on a machine table with the surface of the work at 90° to the table. You can then quickly and easily machine work to right angles. You can use the angle plate when you lay out work for machining. You set the angle plate upon a surface plate with the work to be laid out clamped to the angle plate.

7-15. *Adjustable angle plate.* The adjustable angle plate (also called the tool maker's knee) has an adjustable surface. You can raise the table on the adjustable angle plate as illustrated in figure 7. You can lock the table at any desired angle from the horizontal to the vertical positions. You can also rotate the angle plate on its base. This feature allows you to perform machining operations on two axes.

7-16 **Tool Maker's Buttons.** Toolmaker's buttons are hardened and ground cylindrical pieces of steel. The ends are ground flat or slightly concave. The buttons have a large hole through them to allow you to fasten them to the workpiece and to change the location of the button Toolmaker's buttons are normally available in sets of four with set diameters of .300. .400, or .500 inch. Usually, buttons are 1/2 inch long with one button 5/8 inch long. Toolmaker's button sets, shown in figure 8, are furnished with the screws and hardware for fastening the buttons to the work and for proper storage.

7-17. You use toolmaker's buttons to locate holes with precision. You would make the layout with ordinary layout tools. After the centers of the holes are located, they are drilled and tapped to receive the toolmakers' button screws The buttons are screwed lightly to the work. When you have two or more holes close together, use the long button for the hole you want to drill first. This will allow you to use the dial test indicator to center the work without interference from the other buttons. After you have the buttons accurately located, you then tighten them securely. When you require the buttons to be set with extreme accuracy, it is best to set them one button at a time. Let's assume that you need to bore two accurately located holes from the end and edge of the work with accurate center distances. You locate one button and then locate the other button relative to the first one.

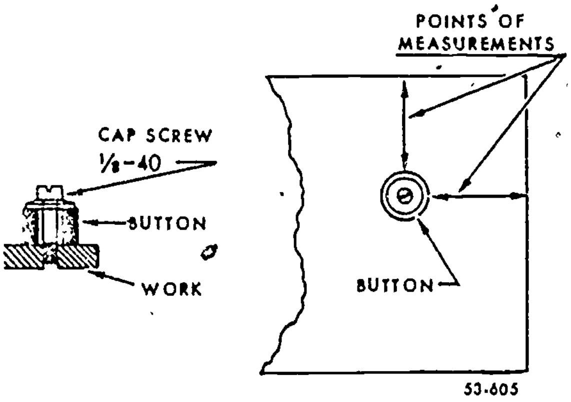


Figure 9. Toolmaker's Buttons and Their Application.

7-18. The first button is located the prescribed distance from the end and edge of the work. You could do this with the aid of a surface plate, angle plates, and gage blocks. Suppose the requirements are that the center of the hole is 1 inch from the edge of the part with its center 1 1/2 inches from the end. Select the proper sizes of gage blocks to give these distances. The correct amount of gage blocks is the required distances minus 1/2 the diameter of the buttons. Align the button, as shown in figure 9. When the button is accurately located, tighten the screw to hold the button securely.

7-19. You may then locate the second button by using gage blocks. You would use the same stack you used to locate the distance of the first button to locate the edge distance of the second

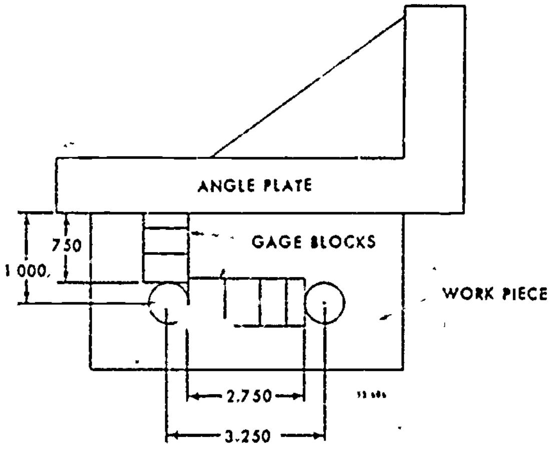


Figure 10 Setup for Toolmaker's Buttons.

button. Since the part requires accurate center distances between the holes, you locate the second button relative to the first. Suppose the center distance is 3.250 inches. You select the correct amount of gage blocks to fit between the buttons. Since the gage blocks are located between the buttons (buttons 1/2 inch in diameter), you subtract 1/2 the sum of the diameters of the buttons, as indicated in figure 10.

7-20. You may also use a micrometer to set the center distance between the buttons. You measure the distance across the buttons. Since the reading is across the buttons, you subtract 1/2 of the sum of the diameters of the buttons to the center distance to obtain the correct micrometer reading. The gage blocks may be used to check the edge distance of the second block. When you have the second button located, tighten the retaining screw securely.

7-21. You can use a dial test indicator to check the edge distance of straight line holes. After repeated checking of the locations of the buttons, you check to insure that they are secure to the work. Make a final check of the button locations after the screws are tightened. You now have the holes accurately laid out with the toolmaker's buttons. The part is ready for the boring operation. If the part is to be bored in a lathe, you mount the part on a face plate. You locate the part by using a dial test indicator on the buttons. When you have the button centered to within tolerances by lightly tapping the work, tighten the clamps securely. Recheck the reading after securing the work. You then remove the button and drill and bore the hole to specifications. Repeat the centering of the buttons for the remaining hole or holes and bore them to specifications. If you perform these operations carefully, you will be able to drill and bore holes with very accurate locations.

7-22. **Hand Scrapers.** Hand scrapers are tools which are used to remove high spots from machined surfaces. In order to produce an accurate, flat bearing surface you must locate and scrape off the high spots. You also scrape bearings for close running fits for shaft bearings.

7-23. *Hooked scraper (flat surfaces).* The hooked scraper is used for the "flowering" or "frosting" design you see on scraped surfaces. You also use the hooked scraper for scraping surfaces when it is inconvenient to use the flat scraper. An application of the hooked scraper would be the angle of a dovetail bearing surface. You may need to scrape round or curved surfaces to obtain good running or sliding fits, which demands another type of scraper.

7-24. *Half-round bent scraper.* The half-round bent scraper is the most common used scraper for curved surfaces. It has two cutting edges. The cutting stroke may be either toward or away from you.

7-25. *Hooked scraper (curved surfaces).* The hooked scraper is preferred by many craftsmen. It is used for scraping larger surfaces. You grind the cutting edge to conform to the curve of the work. Then you sharpen the scraper to cut on the pull stroke.

7-26. *Three-cornered scraper.* The three-cornered scraper is used to break (remove) the edges of holes or edges of other curved work. The three-cornered scraper may also be used to round the corners. The three-cornered scraper is usually made by grinding off the teeth of an old file. This produces three cutting edges. The three corners of the body back of the cutting edge should be well rounded to prevent cutting your hands.

7-27. **Hand scraping.** When you need to hand scrape work, you must first locate the high spots on the work. Do this by using venetian red or prussion blue. For flat work, rub a light coat of the red or blue on the surface plate. The surface to be scraped is then rubbed on the surface plate, and you can see the high spots plainly marked with the paint. High spots on bearing surfaces can be found in a similar manner. You apply the shaft to the bearing surface and the high spots in the bearing are indicated by the red or blue spots. Mating parts are used in the same way to find the high points. As you scrape the high points, the spots increase in number and decrease in size. When this occurs to the point at which the red or blue becomes difficult to use, you can then use turpentine to locate the high spots. The turpentine causes the high points to show up as bright spots. The turpentine also makes the scraper work better. When you use the flat scraper, the cutting stroke is the forward stroke. The stroke is seldom over 1/2 inch in length. As you make successive scrapings of a spot, the direction of the strokes should be parallel to each other. The operations using the hooked scrapers are quite similar. The cutting strokes on these scrapers are normally toward you.

7-28. While you are scraping, do not allow any oil, or even your fingers, to touch the surface. Be extremely careful that you do not allow any foreign matter to get on the surface plate or the parts. This will impair the checking of the surfaces. When you are roughing the surface, you should scrape fairly hard. As the spots become smaller, ease up on the chip. Dipping the scraper in turpentine will help you to scrape easier and faster. Keep the scraper sharp by occasional sharpening on the grinder and oil stone. A dull scraper will scratch the work.

7-29. **Trammels.** The trammel is simply a long beam compass or divider. The trammel has a long steel beam with two sliding heads. The two heads are designed to slide along the beam and then to be clamped securely in place for use. One head is equipped with a micrometer screw adjustment mechanism. This micrometer adjustment allows you to make precise adjustment of the distances between the points. The heads of the trammels have collet spring chucks. You can insert any of the various points and attachments into the heads. The interchangeable points extend the range of uses of the trammels. You use the trammel extensively for scribing circles and arcs with large radii. For this you use the trammels in the same manner as dividers and compasses. You can insert the ball points in one of the heads and scribe radii and circles from the center of holes. You can insert the caliper points into the heads and then use the trammels as large inside or outside calipers.

8. Measuring Devices and Processes

8-1. Much of your time is spent measuring and checking parts for accuracy. Perhaps there have been times when you wished for an easier or more accurate process of measuring or checking. As you study this section you will learn more of measuring devices and processes.

8-2. **Measuring Errors.** It is impossible to manufacture parts to mathematically exact dimensions. This is due in part to the fact that there are inherent errors in all measuring processes. No measuring device or process is entirely free of error. Checking and measuring parts are necessary to ensure that machined parts meet at least minimum tolerances. The tolerances permitted in most machining operations in Air Force shops pose no real problem Let us review some of the minute inherent errors you will find in checking parts for accuracy. A knowledge of these inherent errors will help you to machine parts with greater speed and accuracy.

8-3. *Instrument error.* Every measuring device has an inherent error of indication. Accuracy in the manufacture of measuring instruments affects the degree of accuracy of the reading. Like the parts you machine, the parts of measuring devices are not machined with absolute accuracy. These small inaccuracies result in tiny instrument errors. Wear also affects the accuracy of measuring instruments. As the instruments become worn they become less accurate. Temperature affects instrument accuracy If the part is not at room temperature, the instrument may give an erroneous reading. You can keep this error to a minimum by keeping parts and instruments at room temperature. This can be especially critical in making measurements to the fourth decimal. You can help minimize this error by making sure your instruments have been calibrated. Check the records of the dates of the PMEL (Precision Measurement Equipment Laboratory) schedules. Be sure the precision instruments are current on their inspection and calibration. Also, be sure that new instruments have been sent to the PMEL shop for calibration before you use them.

8-4. *Human element error.* Another source of error is the human element. Reading error is a part of the human element. Reading error may be a result of lack of skill. This may be caused by lack of sufficient light with which to clearly see the graduations. Good lighting would tend to reduce this error. Reading error may be due to the uncertainty of the human eye in preceiving fractional intervals on a scale. One person may have no difficulty in determining which division marks line up on a vernier scale. Another, even with the aid of a magnifying glass, may not be able to determine which divisions line up. Reading errors may be minimized with additional training and practice. Another human element error may be that of differences in "feel" or "touch." Obtaining extremely accurate measurements is an art. There are no indicating gages to tell you when you have just the right feel. For instance, when you measure a shaft with a micrometer there is no gage to tell you when you have turned the thimble just the right amount. You must develop this feel through practice.

8-5. *Judgment error.* Judgment error can be time consuming and costly. You must learn to use judgment quickly and easily in checking parts. Much time can be lost if you are unsure of your judgment of the feel and in reading instruments. If you are to turn a shaft to a tolerance of plus or minus 0.002 inch, it would not be practical to take additional time to machine the part to a tolerance of plus or minue 0.0002 inch. You must develop judgment in working to practical limits.

8-6. **Accessories for Graduated Rules.** Simple, but very effective, accessories are available for steel scales. These accessories greatly extend the usefulness of the scale.

8-7. *Sliding head.* The sliding head has a flat reference surface from which measurements are made. It has a thumb-screw clamp to secure the scale. Normally, the head's reference surface is held perpendicular to the scale by a hand slot or guide pins. The sliding head attachment allows you to use the rule as a depth gage. You would use this attachment to measure the depth of holes and grooves. You can measure the depth of holes and slots up to 5 inches with a 6-inch scale. You can substitute the scale with a 5/64 inch diameter rod for measuring depths greater than 6 inches. An angle index mark is engraved on one side of the conical rule clamp turret. This allows you to use the attachment as a combination protractor to measure approximate angles.

8-8. *Right angle clamp.* The right angle clamp is an accessory often used with the combination square. The right angle clamp holds a scale at a right angle to the blade of the combination square. This allows you to make measurements when both depth and lateral coordinate dimensions are required. When you use the assembly on the surface plate, you can use it as a height gage. You will find many more similar uses of the right angle clamp.

8-9. *Keyseat clamps.* When keyseat clamps are applied to the steel scale you have a keyseat rule. The clamps are easily put on and taken off the scale, which is usually a 6-inch scale. When so arranged they form a box square. The keyseat rule is a very useful tool when you apply it to cylindrical work. You have an excellent guide for scribing lines parallel to the axis of the cylindrical work. You can easily scribe parallel lines on cylindrical work. Also, this device is very convenient in making keyseat layouts on shafts.

8-10. **Caliper Rule.** The caliper rule is a type of measuring caliper. When you use a graduated rule, you have to judge the position of the rule datum edge. It is also necessary that you judge the relation of the graduation to the point being measured by eye. This judgment can lead to error. Greater accuracy is obtained if the measuring device makes contact with the two points being measured. The caliper rule does this. The caliper rule has a graduated scale, a fixed jaw, and a movable jaw. The movable jaw can be moved to make contact with the work. Like the micrometer or calipers, you must also develop a feel for using the caliper rule. When you have the feel of the caliper jaws to the work, you simply read the measurement indicated by the sliding jaw. Caliper rules are constructed to enable you to take both inside and outside measurements. There are two index marks on the movable jaw. Each is labeled as to its function, for outside or inside measurements. To read the dimension, measuring an outside diameter, you would read the dimension indicated by the index mark labeled "out." If you were taking an inside measurement, you would read the dimension indicated by the index mark labeled "in."

8-11. **Vernier Caliper.** The vernier caliper is quite similar to the caliper rule, except that it has a vernier scale. The vernier scale allows you to take measurements to within 0.001 inch. Vernier calipers have beams 6, 12, 24, 36, and 48 inches in length. Although slightly more bulky than the micrometer, one vernier can replace several sets of micrometers. For instance, a 12-inch vernier caliper has the same range of measurements as a set of 12 micrometers. Also, the vernier caliper is capable of making inside as well as outside measurements. The sliding jaw is thicker than the beam. This allows a vernier scale to be mounted on each side. One of the vernier scales is used for inside measurement while the other is used for outside measurement.

8-12. *Measuring with a vernier caliper.* When you make measurements with the vernier caliper, you move the sliding jaw in the direction to obtain the feel on the work. The fine adjusting screw allows you to make fine adjustments on the jaws. You do this by first setting the jaws to the approximate dimension and locking the nut carrier. Then, by turning the adjusting nut you can move the sliding jaw assembly very slowly in either direction. When you have the correct feel on the work, you can then lock the sliding jaw in position for reading the vernier or for holding the setting for subsequent measurements.

8-13. *Reading vernier scales.* When you have set the vernier caliper, you are then ready to read the scale. The vernier scale is a simple means of magnifying small differences in measurements so that they may be easily read on a linear scale. Although it would be possible to etch graduations of 0.001 inch on a steel scale, it would be almost impossible to read them. The addition of the vernier scale makes it a simple matter ro read the 0.001 inch graduations. The main scale is divided into inches the same as a machinist's scale. The inch division is broken into 10 divisions of 0.100 inch. These divisions are further divided into four parts of 0.025 inch each. Each graduation on the main scale, then, equals 0.025 inch. The vernier scale (the lower sliding scale) divides the 0.025 inch main scale into 25 parts equal to 0.001 inch. When the index of the sliding scale (marked "0") is in alinement with a main scale graduation, you simply read the main scale graduations. When the index is not in alinement with a main scale graduation, you read the main scale division to the left of the index and add to that the number of marks to the right of the main scale division in alinement with the vernier scale.

8-14. **Vernier Height Gage.** The basic types of height gages fall into two categories. They are height gages for direct linear measurement and height gages for comparison measurement.

You can use almost any height gage for both functions, depending upon the type of indicating system used. A variety of attachments are available to allow you to use the height gage with either system. The vernier height gage may appear to be quite different from the vernier caliper. However, the basic principle is the same. When you place the vernier height gage on a surface plate, the surface plate becomes the equivalent to the fixed jaw. The sliding arm of the vernier height gage differs from the sliding jaw of the vernier caliper only in minor mechanical details. The sliding arm of the height gage is designed for mounting accessories for easier height measurement. Vernier height gages are normally produced in height capacities of 10, 18, and 24 inches. This means that you can measure these heights with the respective models, even though the beam height is about 2 inches more. Like the vernier caliper, vernier height gages usually have 0.025 inch scale divisions and a vernier scale that permits you to read the scales to within 0.001 inch accuracy. The vertical beam or main scale of the vernier height gage is graduated in relation to the bottom of the base. The scale then reads the actual height of the gaging surface. The gaging surface is the top surface of the sliding arm.

8-15. *Scriber point.* You normally clamp the scriber point on the gaging surface of the height gage. The scale of the height gage permits you to make direct readings to the gaging surface of the scriber point. Essentially, this is a caliper-type reading when used for measuring work. The scribing tip of the scriber point is also at the height of the scale readings. You can scribe lines accurately at the indicated heights. The scriber point can be mounted to the bottom side of the sliding arm. Then the height gage can be used in the same manner as an inside caliper. When you are using the height gage in this way, you must subtract the thickness of the arm from the reading. The minimum height at which you can set the sliding arm gaging surface above the bottom of the base is usually one inch. When measuring heights less than one inch, use the offset attachment. The offset attachment lowers the contact point of the height gage one inch. Thus you can measure heights less than one inch. However, you must subtract one inch from the readings.

8-16. *Dial indicator.* Quite often you can use the height gage to advantage with a dial indicator attached to the sliding arm. You will observe that the contact point of the indicator is not at the same height above the plate as the gaging surface of the sliding arm. You cannot use the height gage for direct scale reading. Instead, the height gage is used to compare a height to a known master, such as a stack of gage blocks.

8-17. An important job you can quickly perform with the indicating height gage is that of checking shaft conditions. Suppose, for example, that you are required to check shaft diameters for concentricity. Suppose that one shaft diameter is to be concentric with another diameter on the same shaft to within a specified tolerance or at least within 1/2 the tolerance of the diameter. Several undesirable conditions of a shaft may affect a measurement of concentricity. The first check is for out of round. To do this you place the shaft on a right angle block. Place the height gage indicator point over the part and rotate the part slowly. Any deviation of the indicator needle will indicate the amount of out of roundness.

8-18. To check for a bent shaft, you place it on a "V" block. You place the indicator contact point at two points along the diameter. A difference in readings indicates a bent shaft if neither portion is tapered. Rotate the shaft one quarter of a turn and repeat the check. Any change in the readings indicates a bent shaft rather than taper or out of roundness.

8-19. To check for shaft curvature, you place the part on two "V" blocks with about 1/4 inch of each end of the shaft resting on a "V" block. Move the indicating gage over the part and set the dial for a zero reading. Then you take three more readings (90° apart) around the shaft. Repeat this at several stations along the shaft in order to separate bent shaft conditions from other conditions.

8-20. The final check is the measurement of eccentricity between one section of a shaft and another. You place the large diameter of the shaft in a "V" block. Place the indicator over the small section, close to the shoulder. Revolve the shaft in the "V" block and note any deviations. As an additional check, place the small diameter in the "V" block and repeat the check. Quite often a shaft will have a combination of conditions, such as out of roundness, crookedness, and curvature, as well as eccentricity on the same side of the shaft. Quite often these conditions will be mistaken for eccentricity as the practical effect on the function of the shaft at final assembly is nearly the same. Also, you may struggle in vain to eliminate eccentricity from a shaft when in reality the condition is something else.

8-21. Another example of the use you may find for the surface plate and indicating vernier height gage is that of checking the perpendicularity of a hole to the axis of a cylindrical part. You will need a "V" block and push-fit plug gage with the part mounted on the surface plate. The "V" block will hold the part perpendicular. Slowly pass the indicating height gage over one end of the plug gage until you obtain a zero reading. Pass the indicator over the other end of the plug gage and obtain that reading. The difference in the two readings indicates whether the hole is perpendicular to the axis or not.

8-22. **Vernier Bevel Protractor.** The vernier bevel protractor is somewhat similar to the bevel protractor you used with a combination set. Although there are differences in the arrangement of the parts, it serves the same purpose, the measurement of angles. Because of the vernier scale you can use the vernier bevel protractor to measure parts of degrees accurately. You can measure angles to an accuracy of 1/12 degree, or 5 minutes.

8-23. *Reading the vernier protractor scale.* We have discussed the reading of vernier scales in earlier paragraphs. Therefore, it should be easy to learn to read the protractor scale. The vernier principles are the same.. The basic difference is that the protractor scales are laid out in the form of a circle and graduated in degrees and minutes. The main scale is graduated into 360 divisions of 1° each. The scale is divided into four quadrants of 90° each. The degrees in each quadrant are numbered from zero to 90 so that there are two zeroes and two 90° positions on the scale. The zero line which is an imaginary line passing through both zeroes is perpendicular to the base of the protractor. The line passing through the 90° position is parallel to the base. The base, which is a straight line, represents a straight angle or 180°. Since all angles measured are referenced to the straight angle of the base, they are supplementary angles. For example, if one side of the blade makes an angle of 60°, the other side of the blade will make the supplement of 60°, which is 120°. Supplementary angles always add up to 180°. To determine which side of the blade represents the reading, you simply observe the angle. If it is an acute angle (less than 90°), you read the side of the blade that represents an acute angle. If the angle is an obtuse angle (over 90°) you read the side of the blade that represents the obtuse angle.

8-24. The vernier scale has a zero line in the center and 12 lines on each side. Since the vernier is used to read fractions of degrees, in terms of minutes, the scale consists of 60 minutes on each side of the zero index. The vernier scale graduations each represent 5 minutes. Every third graduation is identified by a number such as 15, 30, 45, and 60. When you read the scale, the vernier zero index line is the reference point. If the vernier zero index alines with a degree graduation on the main scale, you do not use the vernier scale. You then read the angle directly from the main scale at the vernier index. When the zero index of the vernier scale does not aline with a degree mark, you read the minutes from the vernier scale. If the vernier index is positioned to the right of the main scale zero, you use the right-hand vernier scale. The key to the use of the vernier protractor scale is the position of the vernier zero. You must note whether it is to the right or the left of the zero graduation on the main scale. Care must be taken in using the vernier protractor not to read or record the complimentary instead of the actual angle you are measuring. This is especially true of angles approaching 45°.

8-25. *Acute angle attachment.* The acute angle attachment is provided to permit a longer line of contact on parts that have small angles. Generally, the acute angle attachment is used for measuring angles less than 30°. For example, suppose you want to check an angle of 18° 15′. The reference plane is now the acute angle attachment. The reading on the protractor scale is now the supplement of the angle between the attachment and the blade.

8-26. **Precision Gage Blocks.** A precision gage block is a piece of alloy steel that has been carefully prepared in length to represent a particular dimension. A standard length of measurement is engraved on each block. Precision gage blocks come in sets consisting of many blocks, each with a different dimension. In addition to each individual dimension, you can construct other dimensions with a set of blocks.

8-27. *Gage block sets.* Gage block sets usually consist of from 5 to 103 blocks of various lengths. Each set is supplied with a fitted hardwood case. The case keeps the gage blocks neatly arranged for easy selection, as well as to protect them from damage. Gage blocks are manufactured in various degrees of accuracy. The highest grade is grand master or grade AA. These blocks are manufactured to an accuracy of two millionths (0.000002) of an inch to an inch of length. Grade A blocks have an accuracy of six millionths of an inch in blocks of one inch and longer and two millionths of an inch accuracy for blocks under one inch. The blocks you will most probably use are Grade B. They are known as the working blocks. These are manufactured within a length tolerance of plus ten millionths (0.000010") and minus six millionths (0.000006") tolerance. Each block has its deviation from the nominal size engraved on the block.

8-28. *Using gage blocks.* When you use the gage blocks, it is necessary for you to select the combination of blocks to give you the

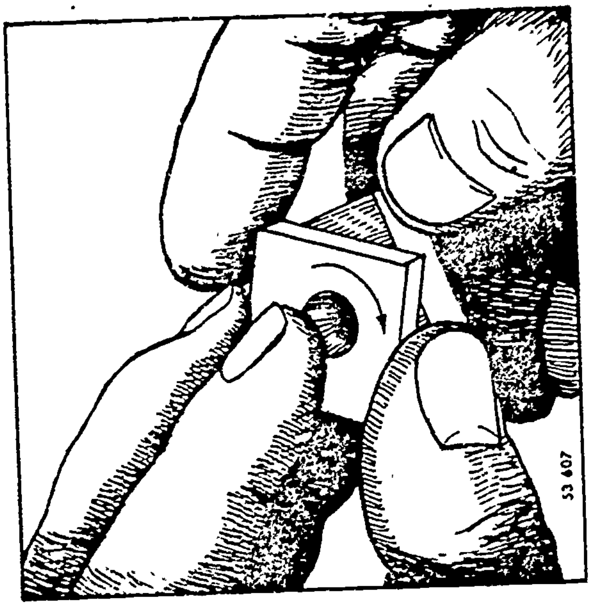


Figure 11. Wringing Out Gage Blocks.

dimension required. You simply select the proper blocks from the set that totals to the dimension you require. Usually, you start selecting from the larger blocks and add to the smaller ones to obtain the dimension, using as few blocks as you can. Once you have selected a stack of blocks, you must then properly stack them. Much of their precision depends upon the way they are stacked. The correct method of joining them together is called "wringing."

8-29. Be sure that the gage blocks are absolutely clean of any foreign matter when they are in use. Also, be sure that your hands are clean and free of oil or dirt. Grease-soiled hands are rarely free of grit. Grit is harmful to gage blocks. Avoid touching the gage block gaging surfaces with the hands as much as possible. Moisture in the hands contains an acid which induces rust. Keep surfaces and parts clean where they come into contact with the gage block gaging surfaces.

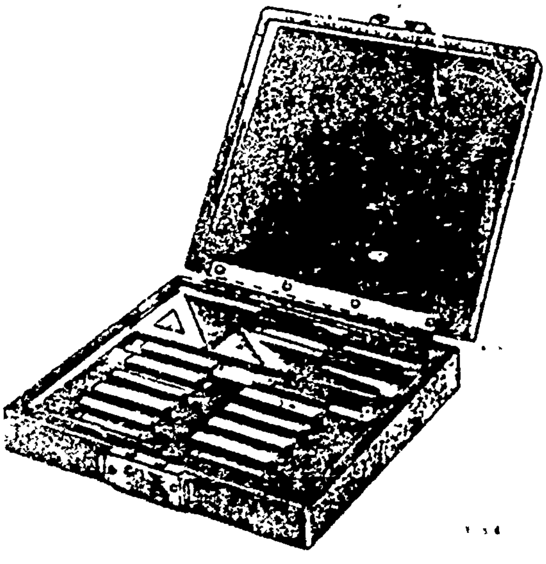


Figure 12. Precision Angle Block Set.

8-30. You are now ready to wring the stack of blocks together. Bring the blocks lightly together in a circular motion, as shown in figure 11. This will detect the presence of nicks or dirt. If a nick or dirt is present, it will result in a slightly gritty feeling rather than a smooth action. Further wringing should not be attempted until the condition is remedied. If the wringing produces a smooth action, slide the top block half way off the lower block. Apply a light pressure and follow by sliding the block into full contact under light pressure. In this position they are properly wrung together and ready for use. Properly wrung stacks of gage blocks will adhere to each other enough to defy the force of gravity.

8-31. Gage blocks have many uses in the machine shop. One of the primary uses is to check other gaging devices. You can use them as standards for checking the calibration of such instruments as a micrometer and vernier caliper and height gage. Also, you use them as the standard for setting comparison instruments. You will use them, too, for setting up toolmaker's buttons and sine bars.

8-32. **Precision Angle Blocks.** Precision angle blocks provide a means of precision angular measurement, much in the same manner as gage blocks for linear measurement. Precision angle blocks are manufactured to an accuracy of 1/4 second where this degree of precision is required. Working sets of precision angle blocks usually have an accuracy within 1 second arc of accuracy.

8-33. *Angle block sets.* A precision angle block set is illustrated in figure 12. A set of 16 precision angle blocks will make up to 356, 400 angles in steps of 1 second. The accuracy is measured to a millionth part of a circle. All precision angle blocks have three common dimensions. The length of the base is 4 inches. The width of the gaging face is 5/8 inch. The dimension at the small end is 1/4 inch.

8-34. *Use of angle blocks.* You can use the angle gage blocks to obtain desired angles. In selecting the angle gage blocks, you can select them to add or to subtract to obtain the desired angle. As mentioned earlier, you use the precision angle blocks much in the same manner as precision gage blocks. They are "wrung" together and adhere as effectively. Small angles are usually more quickly constructed and are more accurate than sine bars or sine plates. Also, for some angular

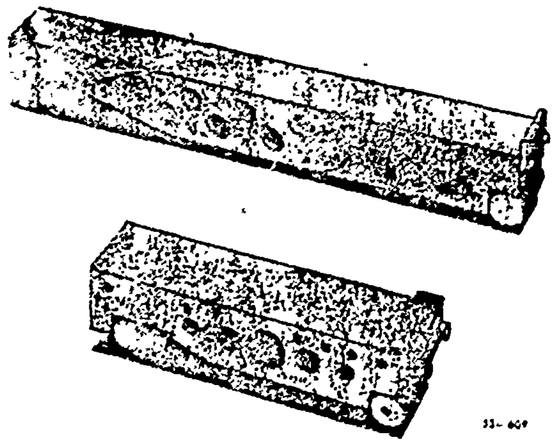


Figure 13. Sine Bars.

surface plate work, angle blocks can be used for precision indexing and checking angular areas not accessible for sine bars. You may also use angle gage blocks to check the accuracy of settings for sine bars and sine plates.

8-35. **Sine Bars.** A sine bar is a precisely machined tool steel bar which is used in conjunction with two steel cylinders. In the type shown in figure 13, the cylinders establish a precise distance of either 5 inches or 10 inches from the center of one to the center of the other, depending upon the model used. The bar itself has accurately machined parallel sides and the axes of the two cylinders are parallel to the adjacent sides of the bar within a close tolerance. Equally close tolerances control the cylinder roundness and freedom from taper. The slots or holes in the bar are provided for convenient clamping of work pieces to the bar. Although the illustrated bars are typical, there is a wide variety of specialized shapes, widths, and thicknesses.

8-36. *How to set up a sine bar.* The sine bar itself is very easy to set up and use. You do need to have a basic knowledge of trigonometry to understand how it works. When a sine bar is set up it always forms a right triangle. A right triangle is a triangle which has one 90° angle. The base of the triangle formed by the sine bar is the surface plate, as shown in figure 14. The side opposite is made up of the gage blocks that raise one end of the sine bar. The hypotenuse is always formed by the sine bar, as shown in figure 14. The height of the gage block setting may be found in two ways. The first method is to multiply the sine of the angle needed by the length of the sine bar. The sine bar of the angle may be found in any table of natural trigonometric functions. The second method is by using a table of sine bar constants. For example, if you had to set a 10-inch sine bar to check a 30° 5′ angle on a part, you would first go to a table of natural trigonometric functions and find the sine of 30° 5′. Then multiply by 10 inches: .50126 × 10 = 5.0126, which would be the height to the gage blocks.

8-37. *Care of the sine bar.* Although sine bars have the appearance of being rugged, they should receive the same care as gage blocks. Because of the nature of their use in conjunction with other tools or parts which are heavy, they are subject to rough usage. Scratches, nicks and burrs should be removed or repaired. They should be kept clean from abrasive dirt, sweat, and other corrosive agents. Regular inspection of the sine bar will locate such defects before they are able to affect its accuracy. When they are stored for extended periods, all bare metal surfaces should be cleaned and then covered with a light film of oil. Placing a cover over the sine or table will further prevent accidental damage and discourage corrosion.

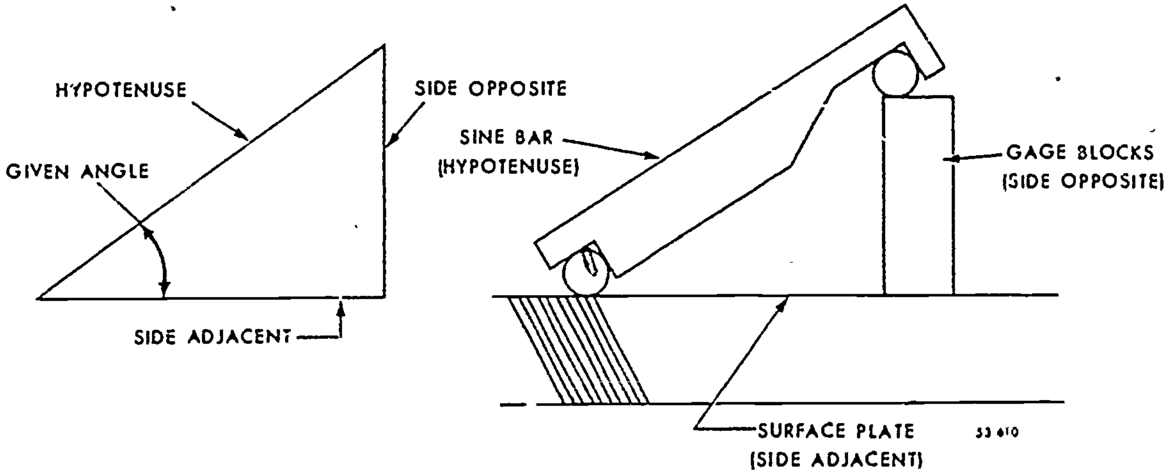


Figure 14. Setup of the Sine Bar.

22

9. Care of Precision Instruments

9-1. As modern production becomes more demanding with respect to accuracy, gages and measuring tools become more and more important in manufacturing. As tolerances or permitted variations in dimensions become smaller, gage tolerances, usually a small fraction of product tolerances, also become smaller. Not only does the higher accuracy demand gages of greater ability, but it requires also a more skillful use, meticulous care, and proper maintenance for a reasonable assurance of quality control. New and better ways of measuring and checking are incorporated as fast as they prove feasible. In the meantime, however, it is necessary that the present devices be made to the greatest possible accuracy and used in the most effective manner.

9-2. Instruments expected to perform such delicate and important tasks, as determining sizes and shapes within thousandths and sometimes millionths of an inch, certainly deserve the utmost care in respect to cleanliness. Although all gages are constructed of materials that provide the hardest contact surfaces for the greatest stability of accuracy, dirt remains the most prevalent cause of excessive wear. Most gages, to accomplish their purpose, make a physical contact with the surfaces being checked. In most all these contacts friction is present, such as sliding a snag gage over a shaft, or a plug gage into a hole. A close analysis of the film normally present on the surface of both the gage and the part being checked will show a large content of abrasives. Thus, through use in the presence of abrasive dirt, the surfaces of the gage are continually wearing away and destroying the accuracy with which it was credited when new. Obviously, if the abrasive laden dirt is eliminated, life and performance of each instrument are greatly preserved.

9-3. Measuring tools and gages should be provided with a storage area where dirt and dust will not accumulate. Also, to avoid rust, the temperature variations should not be excessive and relative humidity should never exceed fifty percent. If such an area is not available, all exposed steel surfaces of tools and gages should be covered with a corrosion preventive coating of lubricating oil, Specification MIL–I–35–3, when not in use.

Lubricants, Coolants, and Fluids

"WHEELS MAKE THE world go round." Although this is not true in the literal sense of the word, wheels do play an important role. In order for a wheel or any other object such as a gear, cog, shaft, or pulley to turn properly, it must have some form of lubrication.

2. Lubrication may be in the form of grease or oil for moving machine parts, hydraulic fluid for control surfaces of high performance aircraft, or a cutting and cooling oil for use in machining other metal parts. In this chapter we discuss each of these lubricants and their uses in an Air Force machine shop.

10. Lubrication

10-1. In every machine shop, bearings of one kind or another are without number. There are bearings which revolve high speed spindles and slow turning trunnions, and bearings which guide delicate mechanisms or support heavy masses, and they all have to be lubricated. In the machine shop the machinist is concerned with the lubrication of the equipment or of some mechanism he may be building. Therefore, the failure to properly lubricate the equipment may cause machine failures which, in turn, will hold up production and maintenance work. You can readily see that lubrication is highly important in the machine shop.

10-2. Bearings and Their Lubrication. Bearings, in the broad sense, are supports or guides for the moving part of machines, engines, and shaftings. The various types are known as plain bearings of the radial, thrust, or slipper types; and antifriction, consisting of both ball and roller bearings, each of which may be of the radial or thrust bearing type. The efficiency of all machine tools depends in a large measure upon the ability of these parts to operate without loss of power, excessive wear, or high maintenance cost. The most important cause of inefficiency is friction. Friction is defined simply as resistance to motion. Suppose for example, that two flat metallic surfaces rest one on the other, as shown in figure 15. It would appear that smoothly ground surfaces, such as these, would offer little or no resistance to the movement of one over the other. When these surfaces are greatly magnified, however, they are found to be made up of many microscopic irregularities, such as the hills and valleys shown on the flat plates in figure 16A. The interlocking of the irregularities produces a definite resistance to the motion of one surface over the other. The force necessary to start movement is great. After the movement is underway, the force required for continued motion is less because inertia prevents the moving surface from dropping into another interlocking position. This condition would exist in a slipper bearing operated without lubricant. The result would be the destruction of the rubbing surfaces. By placing a film of oil between the bearings, as shown in figure 16B,

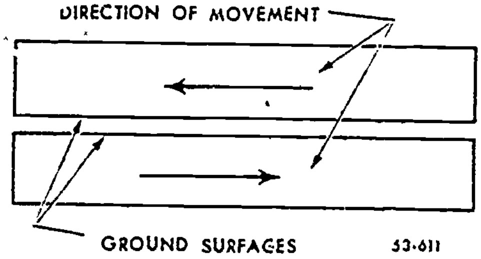


Figure 15 Ground Metallic Surfaces.

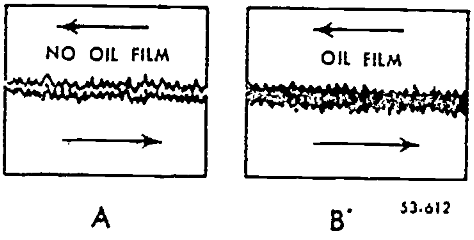


Figure 16. Magnified Ground Surfaces.

24

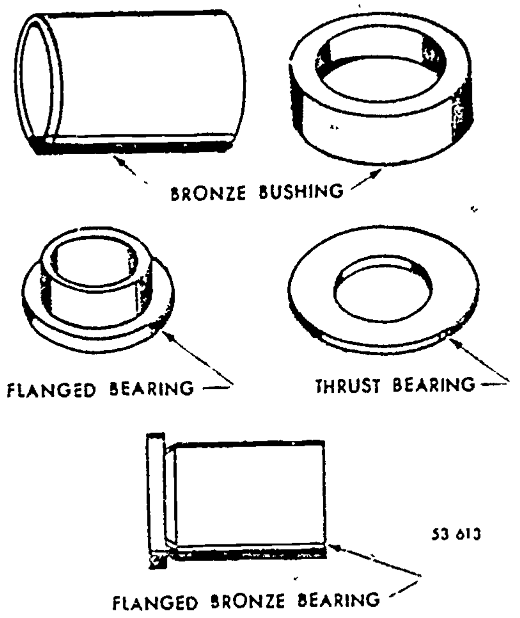
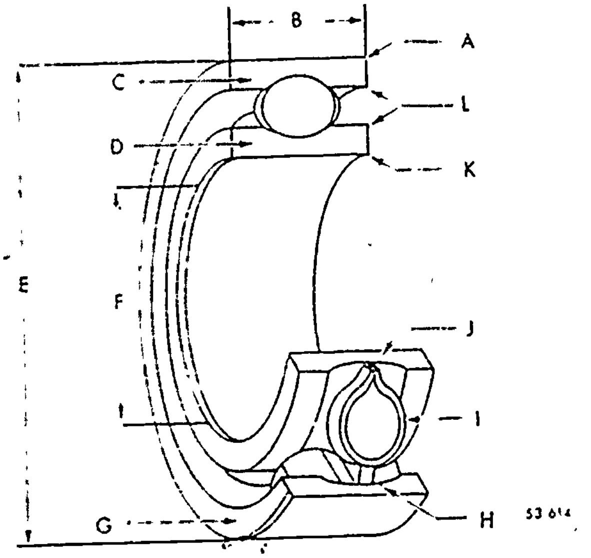


Figure 17 Plain Bearings.

you can see that the two surfaces are separated from each other, thus preventing the surface irregularities from interlocking and allowing one piece to slide over the other more easily.

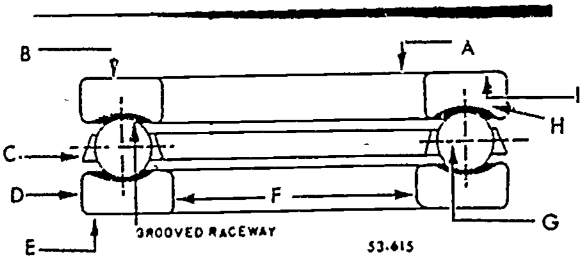


A	Corner Radius	G	Race
B	Width	H	Outer Ring Ball Face
C	Outer Ring	I	Separator
D	Inner Ring	J	Inner ring Ball Race
E	Outside diameter	K	Corner Radius
F	Bore	L	Shoulders

Figure 18 Antifriction Ball Bearings

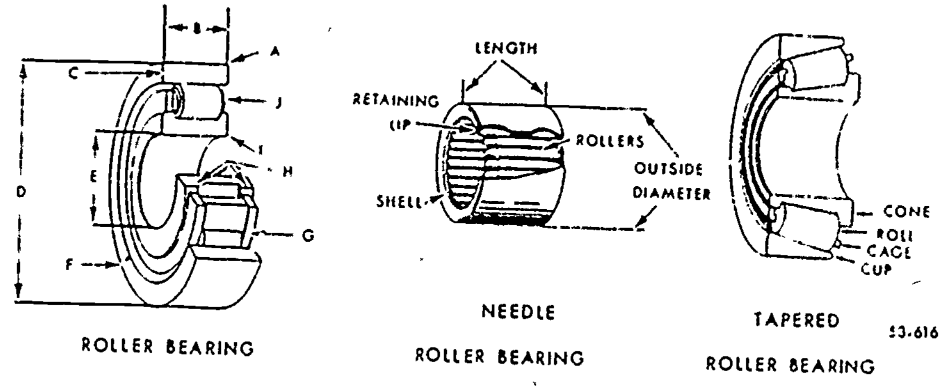
The primary purpose of lubrication in plain bearings is to introduce between moving metallic surfaces a fluid or plastic medium in the form of a lubricating film which separates the surfaces enough to prevent metallic contact. Figure 17 shows some of the more common types of plain friction bearings. They are used in machine tools, airframes, engines, trunnions, connecting rods, and control rods. The material they are made of is hard enough to take wear, yet softer than the shafts with which they make contact.

10-3. Ball and Roller Bearing Lubrication. The use of antifriction bearings is steadily increasing on all types of machinery. Antifriction bearings offer less resistance to starting motion. Their motion, instead of a sliding motion, is theoretically a rolling motion. A ball bearing consists of an outer race and an inner race separated by freely moving balls, as shown in figure 18. The outer race may be fixed to the bearing housing, in which case the inner race is mounted to the rotating shaft. Another type of antifriction bearing, which uses balls to roll on is a ball bearing thrust bearing, as shown in figure 19. The substitution of rolling motion for the sliding motion of plain bearings lessens the friction or resistance resulting from the contacting surfaces. In roller bearings, cylindrical or tapered rollers are employed instead of balls, as shown in figure 20. Antifriction bearings have a longer life than plain bearings. The problem in the lubrication of antifriction bearings is not the actual lubrication as much as it is the prevention of abrasion, corrosion, and the accumulation of harmful deposits. Abrasion is prevented by a grease that forms a seal between the shaft and bearing housing which excludes dust and grit. Antifriction bearing may be lubricated either by oil or grease. Bearings that are lubricated by



A.	Large Bore	E.	Face
B	Large Bore Washer	F	Small Bore
C	Bore	G.	Ball
D	Small Bore Washer	H	Raceway Depth
		I	Face

Figure 19 Ball Bearing Thrust Bearing



A. Corner Radius
B Width
C. Outer Ring
D. Outside Diameter
E. Bore
F. Face
G. Separator
H. Shoulders
I Corner Radius
J Roller

Figure 20 Roller Bearings

oil are open so that the oil can get to the balls or rollers and races. Usually, the type that is lubricated with grease is the sealed type in which the sides of the bearing have sealing rings that hold the grease in and around the balls and races.

10-4. **Types of Oiling Systems.** There are four basic types of oiling systems which we discuss: circulation, splash, bath, and ring and chain.

a. Circulation. This system is employed on practically all types of automotive and aircraft engines, as well as on a variety of machine tools. This type of oiling is used when lubrication is critical and when a large volume of oil is required for cooling, as well as for lubrication. The oil is pumped to the parts of the machinery by an oil pump, which is usually built into the machine.

b. Splash. This system is found on many types of machinery. The oil is contained in a gear housing on most machine tools and the gear teeth dip into the oil and splash it over the parts needing the lubrication. It then drains back to the sump or bottom of the housing where it is reused.

c. Bath. Frequently, this system is employed for the lubrication of thrust bearings or vertical step bearings. In this method the bearings are always submerged in oil, which provides cooling as well as lubrication.

d. Ring and chain. In this system the oil is carried from a reservoir in the bearing housing to the top of the bearing by means of rotating rings or chains which revolve loosely on the revolving journal. The ring and chain oiling system is often found on low speed machinery.

10-5. **Oiling Devices.** A number of different methods of getting the lubricant to the moving parts of the machinery may be used. We briefly discuss some of these methods.

a. Drop-feed cups. Drop-feed cups supply a regulated gravity feed of lubricant to parts of many machines.

b. Wick-feed. Wick-feed oilers use the principle of capillary action from a reservoir through the wick to the bearing surface.

c. Bottle. Bottle oilers differ from the ordinary drop-feed and wick-feed oilers in that they supply a more continuous, though extremely small, amount of oil to the bearing during the intervals when they are feeding.

10-6. **Characteristics of Good Lubricating Oils.** The following are some of the characteristics that go to make up good lubricating oils.

- High chemical stability.
- Rapid separation from water.
- Maximum resistance to detrimental influence of impurities.
- Maximum film strength and lubrication.
- Minimum variation in viscosity with temperature changes.

10-7. **Grease Lubricants.** Under certain conditions the plastic nature of a soap-thickened oil is preferable to the oil itself. Such a mixture of oil and soap make up greases. The resulting lubricant is capable of forming a plastic lubricating flim, thus minimizing friction, wear, loss of power, and leakage. Grease is used on various friction and antifriction bearings. Note some of the types of greases in the following paragraphs.

10-8. *Lime-soap greases.* These greases are commonly known as cup greases and they are recognized by their smooth texture. Every lime-soap grease is essentially an emulsion of oil and lime-soap These greases are excellent under conditions where the operating temperature is not high. where there is frequent application, and where there is no excessive churning action.

10-9. *Soda-soap greases*. These greases are noted for their fibrous texture. They can withstand higher temperatures and are less likely to separate from agitation or churning. They have good resistance to oxidation and deterioration. Soda-soap greases emulsify with water and their use is impractical in or around water.

10-10. *Metallic-soap greases*. These greases are specialty products of a very adhesive nature. Because they form an exceptionally sticky lubricant, they are suitable for service where only small quantities of lubricant may be used to prevent throwing, or dripping.

10-11. **Solid Lubricants.** Such solids as graphite, molybdenum disulfide, lead, babbit, silver, or metallic oxides are often used as lubricants. Though most of them are used in conjunction with fluid or grease, they can be and often are the only means of lubrication. They have certain inherent limitations that must be considered. First, is their inability to carry away heat. Second, they cannot be reused or replace themselves as do oils and greases. Third, they cannot be applied as oil and grease are. They must be bonded to the bearing surfaces by plating, fusing, or by chemical or thermal deposition.

10-12 **Machine Tool Lubrication.** All machine tools have definite lubrication requirements for such things as spindle bearings, work heads, ways, lead screws, gearing, and various bearings in general. All these require lubricants with the proper chemical stability, proper viscosity of oil or consistency of grease, and high film strength. Because there are so many oil companies refining lubricating oils and greases under so many different trade names, it is difficult to make any recommendations 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 on the lubrication needs of the machine. Usually, the lubricant recommendations are very general in the operator's manual, while each oil company recommends a specific brand of lubricant. It is up to the machinist to convert the general lubricant specification from the operator's manual to a corresponding specification as listed by the manufacturer. Oils and greases are also made by oil companies to meet certain specifications as set down by the government.

11. Hydraulic Fluids

11-1. More and more machine tool manufacturers are using hydraulic power to operate practically every type of machine. This is due largely to the flexibility, smoothness, and simplicity with which hydraulic power may be applied. Many complex controls on various machine tools may be greatly improved and simplified by the use of hydraulic actuation. For example, some of these controls are found in hydro-shift lathes, in which the speed may be changed while the machine is still running. Other advantages of hydraulic controls in machines are rapid tool approach, rapid tool return, and smooth vibrationless action, which is not affected by changes in loads. Cushioning effect, which often tends to improve surface finish, is still another advantage of hydraulically controlled machines. These are just a few of the many advantages of hydraulic power. These various machines employing hydraulic power need to be serviced. Extreme care should be used in changing or adding to the hydraulic fluids of these machines. Be sure to always check the operator's manuals for the proper type of hydraulic fluid. The use of improper fluids can cause severe damage to the seals in the machine and render the machine useless until repairs can be made.

11-2. **Types of Hydraulic Fluids.** There are several types of fluids which are used in hydraulic equipment. They are referred to as "hydraulic fluids" and "cleaning agents."

11-3. *Petroleum-base fluid*. One petroleum-base fluid is specified by MIL—H—5606. Petroleum-base fluid (5606) is dyed red for easy identification. It is generally supplied in 1-quart and 1-gallon containers and is available in one grade only. This one grade has an operating range of −65° F. (−53.9° C.) to +275° F. (135° C.). The advantage of this wide operating range is the ability of the fluid to perform adequately in summer and winter temperatures. The seals required with the petroleum-base fluid may be synthetic rubber, leather, or metal composition.

11-4. *Vegetable-base fluid*. Vegetable-base fluid, Specification MIL—H—7644, as it is presently known, was widely used in hydraulic systems using natural rubber seals. It has a bluish color, readily distinguishable from the petroleum-base type. However, this vegetable-base fluid is becoming obsolete.

11-5. *Synthetic-base fluid*. Some of the hydraulic systems used in high-speed supersonic aircraft require specially developed hydraulic fluids that have an operating range of abnormally high temperature. The main reasons for using this type of fluid are the close tolerances of the variable actuating units, the specially designed metering valves, and the overall heat generated by the supersonic speed of the aircraft.

11-6. **Storage of Hydraulic Fluids.** Hydraulic fluids should be stored in the same way as all flammable liquids. Care should be taken to prevent contamination. Cans that have been opened should be properly sealed to insure that the fluid will not be contaminated. Water, metal chips, dust, and other solids, if induced into the hydraulic system, could cause damage. Fluids should also be stored in properly marked containers.

12. Cutting Lubricants and Coolants

12-1. The primary purpose of a coolant is to carry away cutting tool heat. The various chemically active cutting fluids also reduce the cutting forces and thus ease the machining of metals. Operations that are performed at high speeds can benefit from a cutting fluid. High cutting speeds generate high temperatures at the tool point which causes the tool to lose its hardness. The use of coolants permits higher cutting speeds without increased tool temperature. In machining operations there are two methods of introducing cutting lubricants or coolants to the work. The most common method is the "conventional method" in which an abundance of coolant is flooded over the entire work and tool surfaces, as shown in figure 21. A disadvantage of this method in some cases is that the fluid cannot be applied close enough to the chip interface to perform its cooling function. Another method is the "jet cooling" application of cutting fluid to the work and tool, as shown in figure 22. The theory behind this method is that the coolant is forced to the cutting edge of the tool and interface of the chip to reduce tool tip temperature.

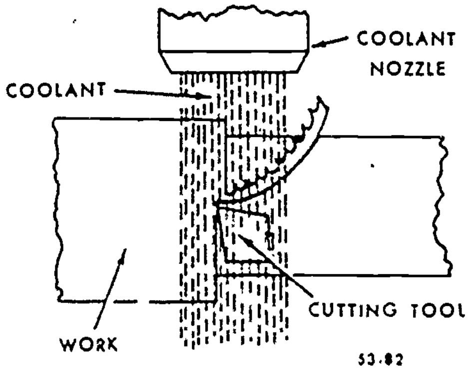


Figure 21 Conventional Method of Cooling a Cutting Tool.

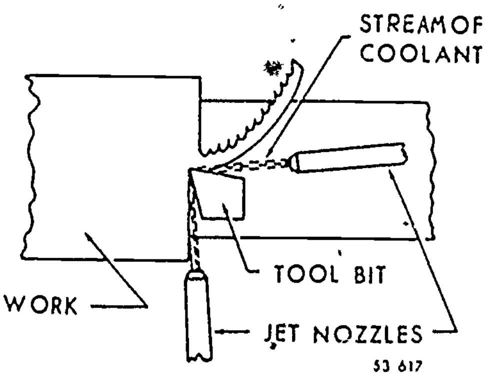


Figure 22 Jet Method of Cooling a Tool.

12-2. **Cutting Oils and Emulsions.** There are no hard and fast rules for the type of cutting fluid to use; an actual test with the tool or operation and material is by far the best test. The following general types of cutting oil and emulsions are used in most machine tool operations.

12-3. *Sulphurized oils.* The development of sulphurized oils was a great step forward in the development of cutting oils. One point about sulphurized oil that makes it useful as a cutting oil is that it helps to prevent the tendency toward welding of the chip to the tool tip and the consequent scuffing of work and tool deterioration.

12-4. *Chlorinated oils.* Chlorinated oils, like sulphurized oils, have a wide range of uses. Since they have low viscosity, many machinists feel that light bodied oils are better coolants and penetrants. These oils are not as well established as the sulphurized oils.

12-5. *Soluble oils.* Soluble oil products, when mixed with water for emulsions, have been widely used in machining operations. This type of coolant has proved very satisfactory for lathe and milling machine work where cooling is an important factor. Some machining operations benefit by a combination cutting oil and coolant where the reduction of tool wear and a cooling action are necessary. Most manufacturers of water-oil emulsion coolants add a rust inhibitor to the solution to prevent rusting of machine part due to the water in the coolant.

12-6. *Lard oils.* Lard oil was widely used in the past, but with the development of sulphurized and other treated oils, it has gradually lost out in popularity. It may still be preferable for some turning operations, such as threading, but any fluid fatty oil, such as peanut oil or olive oil, will do just as well. Also, it may be used as a coolant with water by adding sal soda to emulsify the solution, but then it has a disadvantage in that the coolant becomes rancid with age.

12-7. **General Recommendations for Cutting Fluids.** As was stated earlier, no hard and fast rules can be applied to the selection of cutting fluids. The selection of the type of fluid depends upon the machinability of the metal and various other factors. The following recommendations are made as a general guide in the selection of cutting fluids:

a. Steel: If soluble oil is suitable, use a mixture of 1 part of oil and 10 to 20 parts of water. If the soluble oil tends to corrode the work, mineral oil may be preferable. For steels that take a great deal of tool pressure to cut, a sulphurized oil may be used. In cutting hard steel a chlorinated oil may be preferable.

b. Brass. Soluble oil or a mineral-lard oil are used for brass. In some cases a cutting oil would be preferable because of its lubrication. Brass castings are usually machined dry

c. Monel Metal: Emulsions usually give a much longer tool life. Sulphurized oils tend to aid in breaking of the chip and this may be preferable in some cases.

d. Aluminum Alloys: For many machining operations soluble oil is good. There are many recommendations for cutting fluids to be used on aluminum.

Here are three recommendations:

(1) 90 percent kerosene and 10 percent mineral oil to 50 percent kerosene and 50 percent mineral oil, (2) 85 percent soluble oil and 15 percent kerosene, and (3) 50 percent kerosene, 40 percent lard-oil and 10 percent chlorinated solvent.

e. Cast iron: Cast is usually machined dry. However, soluble oil emulsions may be used to hold down the amount of dust around the machine.

CHAPTER 4

Metallurgy

THE SELECTION OF the proper metal is one of the most important steps in the manufacture of tools or parts. You must consider tooling, fabrication, and the available equipment, as well as the intended use or function of the tool or part. Metals may be selected for such mechanical properties as strength or toughness. Of course, the chemical composition of the metal controls its mechanical properties. These properties may be changed by heat treatment. In Chapter 4 we discuss the (1) properties and characteristics of metals, (2) identification, (3) effects of heat treatment, (4) hardness testing, (5) machinability of metals, (6) nondestructive inspection, and (7) corrosion control and plating.

13. Properties and Characteristics of Metals

13-1. The machinist is much more concerned with the mechanical than the physical properties of metals. In our discussion of the characteristics of metals, we limit our remarks to chemical composition.

13-2. **Properties of Metals.** The internal reactions of a metal to external forces are known as mechanical properties. The mechanical properties are directly related to each other. A change in one property usually causes a change in one or more additional properties. For example, if you increase the hardness of a metal, the brittleness usually increases and the toughness usually decreases.

13-3. We briefly review the mechanical properties of metals prior to discussing their composition:

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

b. Tensile strength. Tensile strength is the resistance that a substance offers to being pulled apart by a slowly applied load. Tensile strength increases or decreases as the hardness increases or decreases. The tensile strength of metal is usually stated as pounds per square inch of cross-sectional area.

c. Brittleness. Brittleness is the tendency of a material to fracture or break with little or no deformation, bending, or twisting. Brittleness is usually not a desirable mechanical property. Normally, the harder the metal, the more brittle it is.

d. Shear strength. Shear strength is the resistance to an action similar to the cutting of 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. Shear strength can be controlled in the same manner as tensile strength, i.e., by varying the hardness of the metal.

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

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

h. Stress. Stress is the reaction within a material to an externally applied force.

i. Strain. Strain is the change in the length per unit of length within a material subjected to a stress.

13-4. **Characteristics of Metal.** The composition of a metal (the chemical composition and the grain structure) determines the properties of the metal, the properties that can be changed, the amount of change that can be made, and the methods by which changes can be made. A knowledge of the composition of metal will help you determine the type of metal to use for a specific application and the heat treatment required to obtain the desired mechanical properties.

13-5. There is no way that a piece of copper can be made as strong as a piece of alloy steel, and there are limits to the properties that can be developed in any metal or alloy by heat treatment. A metal that permits the required properties to be obtained must be selected. The chemical composition refers to the elements, and the exact percentage of each element, that a metal consists of. The chemical composition is often referred to as the chemical analysis or, more simply, as the analysis. The chemical composition of a metal determines the grain structure and the properties which can be developed in the metal. Metal may be in the form of a (1) pure metal, (2) mechanical mixture, (3) solid solution, or (4) combination of a mechanical mixture and a solid solution.

13-6. *Pure metal.* Pure metals are rarely used outside of laboratories. A pure metal cannot be hardened by heat treatment because there is little change in its structure when it is heated.

13-7. *Mechanical mixture.* A mechanical mixture can be compared to concrete. Just as the sand and gravel are visible and held in place by the cement, the elements and compounds in a mechanical mixture are clearly visible and are held together by a matrix of base metal. An alloy in the form of a mechanical mixture at room temperature may change to a solid solution or to a partial solid solution when it is heated. When it is cooled to room temperature the alloy may return to its original structure, remain a solid solution, or form a combination of a solid solution and a mechanical mixture.

13-8. *Solid solution.* When two or more metals are absorbed, one into the other, they form a solution. You are probably most familiar with liquid solutions, but solutions may also be gaseous or solid. When an alloy is in the form of a solid solution, the elements and compounds forming the metal are absorbed into each other in much the same way that salt is dissolved in a glass of water. The separate elements forming the metal cannot be identified even under a microscope. The solubility of various metals often increases when the temperature is increased. Thus, a metal in the form of a mechanical mixture at room temperature often goes into solution when it is heated. When it is cooled, a metal may remain in a solid solution, form a combination solid solution and mechanical mixture, or return to a mechanical mixture.

13-9. *Combination solid solution and mechanical mixture.* An alloy which consists of a combination solid solution and mechanical mixture at room temperature may change to a solid solution when it is heated. When it is cooled it may remain a solid solution, revert to its original form, or form a complex solution.

14. Identification

14-1. Proper identification of a metal is necessary to insure that a locally manufactured item has the required properties. Unknown metals or metals that are not positively identified cannot be used. Various methods of identification and markings are used.

14-2. **Methods of Identification.** We discussed the numerical codes and the color code in detail in CDC 53130, Volume 1, Chapter 2. Spark testing an unknown metal is not accurate enough for the identification of metals.

14-3. The most positive means of identification is by Military or Federal Specification Number. When metal is received in the shop, it is normally identified by a specification number, or by a Federal Stock Number that can be cross-referenced to a specification number. To insure that the metal remains properly identified, each piece must be marked in accordance with Technical Order 42D-1-3.

14-4. **Methods of Marking.** One of the three marking methods which are listed in Technical Order 42D-1-3 must be used to mark metal that has not been marked in accordance with Federal or Military Standards by the manufacturer. In order of preference, the three approved methods are (1) stencil and paint, (2) color code, and (3) stamping with metal dies.

14-5. *Stencil.* A stencil and white or black paint, whichever shows up better on the metal being marked, should be used whenever the size of the metal piece permits. The Federal or Military Specification Number should be stenciled on the metal in vertically and horizontally alined rows. The distance between the vertical rows should not exceed 36 inches and the distance between the horizontal rows should not exceed 10 inches.

14-6. *Color code.* Several significant changes have been made in Technical Order 42D-1-3 pertaining to color codes. Two additional colors, gold and silver, have been added to the 10 colors which are used to represent numerals and letters. Aeronautical Material Specification and Military Specification metals of aircraft quality require an additional stripe that indicates the condition or quality of the metal. Gold is used to represent the letter N and indicates a normalized temper. Silver is used to represent the letter Q and indicates aircraft quality. The commercial designations, American Iron and Steel Institute (AISI), Society of Automotive Engineers (SAE), and Aluminum Association (AA) numbers are used as the designation numbers. The stripes need not extend completely around the circumference of a rod or bar; instead, dots may now be used, provided that the dots extend a third of the distance around the rod and that dots are used on two sides. The color code marking is no longer required in the center of the bar.

14-7. *Stamping.* Stamping the specification number into the metal is permitted only when it is impossible to use the stencil or color code methods. It is usually necessary to cut or eliminate the marked portion of the metal prior to using the material for work stock. Therefore, the marking should be located where waste will be held to a minimum. Gothic style numerals and letters should be used, the height may be 1/16 inch, 1/8 inch, or 1/4 inch, depending upon the size of the material being marked.

15. Effects of Heat Treatment

15-1. A machinist does not perform heat treating. He must, however, know the effects of heat-treatment and be able to specify the heat treatment required for metal parts. Most of the metals you work with in a machine shop are heat treated or will require heat treatment after machining.

15-2. Heat Treatment of Ferrous Metals. All heat treating operations involve the heating and cooling of metals. The common forms of heat treatment for ferrous metals are hardening, tempering, annealing, normalizing, and case hardening.

15-3. *Hardening.* A ferrous metal is normally hardened by heating the metal to the required temperature and then cooling it rapidly by plunging the hot metal into a quenching medium, such as oil, water, or brine. Most steels must be cooled rapidly to harden them. A few, however, can be hardened by cooling in air. Hardening increases the hardness and strength of the metal, but increases the brittleness.

15-4. *Tempering.* Steel is usually harder than necessary and too brittle for partical use after being hardened, and severe internal stresses are set up during the rapid cooling of the metal. Steel is tempered after being hardened to relieve the internal stresses and to reduce the brittleness. Tempering consists of heating the metal to the required temperature and then permitting the metal to cool The rate of cooling usually has no effect on the metal structure during the tempering operation. Therefore, the metal is usually permitted to cool in still air. The temperatures that are used for tempering are normally much lower than the hardening temperatures. The higher the tempering temperature used, the softer the metal becomes. High-speed steel is one of the few metals that becomes harder instead of softer after it is tempered.

15-5. *Annealing.* Metals are annealed to relieve internal stresses, soften them, make them more ductile, and refine their grain structures. Metal is annealed by heating it to a prescribed temperature, holding the metal at that temperature for the required time, and then cooling it back to room temperature. The rate at which the metal is cooled from the annealing temperature varies greatly. Steel must be cooled very slowly to produce maximum softness. This can be done by burying the hot part in sand, ashes, or some other substance that does not conduct heat readily (packing), or by shutting off the furnace and allowing the furnace and part to cool together (furnace cooling).

15-6. *Normalizing.* Ferrous metals are normalized to relieve the internal stresses produced by machining, forging, bending, or welding. Normalized steels are harder and stronger than annealed steels. Steel is much tougher in the normalized condition than in any other condition. Parts that will be subjected to impact and parts that require maximum toughness and resistance to external stresses are usually normalized. Normalizing prior to hardening is beneficial in obtaining the desired hardness, provided that the hardening operation is performed correctly. Low carbon steels do not usually require normalizing, but no harmful effects result if these steels are normalized. Normalizing is achieved by heating the metal to the required temperature (which is higher than either the hardening or annealing temperatures), soaking the metal until it is uniformly heated, and cooling it in still air.

15-7. *Case hardening.* Case hardening is an ideal heat treatment for parts which require a wear-resistant surface and a tough core, such as gears, cams, cylinder sleeves, etc. The most common case hardening processes are carburizing and nitriding.

15-8. Heat Treatment of Nonferrous Metals. There are only two types of operations for nonferrous metals: annealing and heat treatment. Heat treatment, when it is used with reference to nonferrous metals, means hardening.

15-9. *Annealing.* Most nonferrous metals can be annealed. The annealing operation consists of heating the metal to the required temperature, soaking, and cooling to room temperature. The temperature and method of cooling depend upon the type of metal.

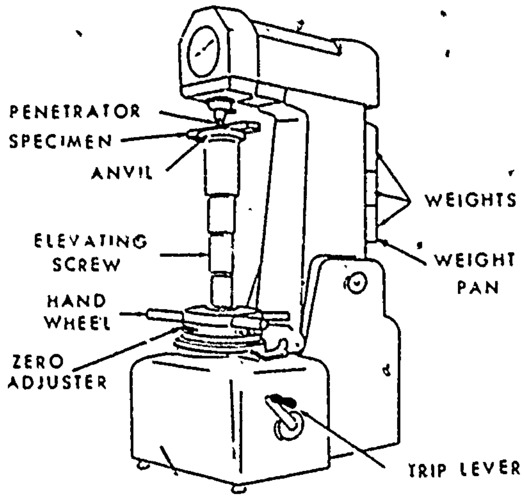


Figure 23 Rockwell Hardness Tester

15-10. *Heat treatment.* There are two types of heat treatment for nonferrous alloys: the solution treatment and the precipitation treatment. Some alloys require both types of treatment. Others require only the solution treatment

15-11 The solution treatment consists of heating the alloy to the temperature at which the elements or principle elements go into a solid solution, soaking until a uniform structure is obtained, and cooling at a rate fast enough to retain the solid solution at room temperature. The solution temperatures differ for various alloys.

15-12 A nonferrous alloy is in the form of a supersaturated solid solution after the solution treatment To obtain maximum hardness and strength, the hardening constituent in excess of that which is soluble at room temperature must precipitate or leave the solid solution. Precipitation takes place at room temperature in many alloys. Such alloys increase in hardness and strength upon standing, following the solution treatment. When precipitation is complete and the alloy obtains full hardness and strength, the alloy is said to be age hardened or aged. Other alloys do not precipitate at room temperature and must be given a second heat treatment called the precipitation treatment or artifical aging. Artificial aging is usually accomplished by heating the alloy to the required temperature, soaking until precipitation is completed, and then cooling in air. Precipitation, whether it results from artificial or natural aging, sometimes causes a decrease in corrosion resistance.

16. Hardness Testing

16-1. A machinist does not perform hardness testing of metals, but he should know the purpose and principle of Rockwell hardness testing. Hardness testing is a responsibility of the metals processing shop. The principle of Rockwell hardness testing is based on the distance that a penetrator will penetrate a piece of metal.

16-2. The Rockwell hardness tester has been selected by the Air Force 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 23 illustrates a Rockwell hardness tester, while figure 24 shows some of the attachments used with it. This tester uses the static principle and works on a leverage system When the lever shown on the right side of the base in figure 23 is tripped, a predetermined load or weight forces a penetrator into the metal being tested. The hardness value, which depends upon the depth of penetration between the minor and major load, is indicated on the dial. The shallower the penetration, the higher the hardness number. The penetrator can be

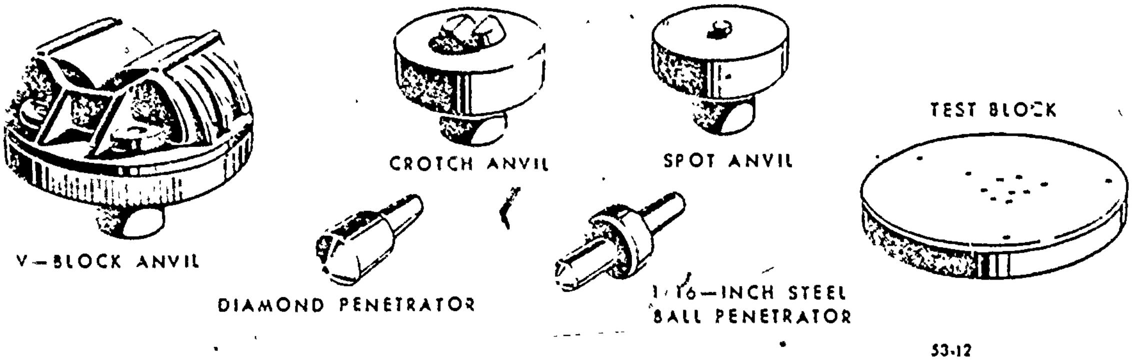
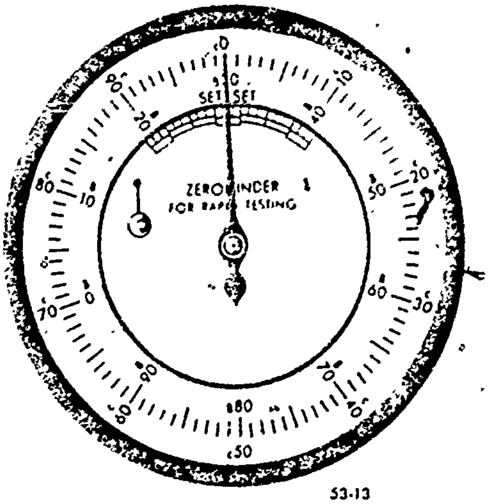


Figure 24 Rockwell Hardness Tester Attachments



53-13

Figure 25 Dial Face of a Rockwell Hardness Tester

either a diamond or a hardened steel ball. The diamond (Brale) has a sphero-conical shape (a cone with a rounded point) and is precision cut and polished. The standard steel balls are either 1/16 inch or 1/8 inch in diameter, while 1/4-inch or 1/2-inch balls are used for special tests.

16-3. Before the major load is applied, the test specimen 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.

16-4. The load that is applied to force the penetrator, shown in figure 23, 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 25 shows a dial face.

16-5. Let's suppose that a work order comes to your shop which required the fabrication of an aircraft replacement part. Your shop foreman turns the job over to you. You may work from a blueprint which specifies the required dimensions, type of metal, heat treatment, and hardness, or you may find all of these specifications in a technical order. In any 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.

17. Machinability of Metals

17-1. Many factors determine the machinability of a material, such as its composition and grain structure and the heat treatment it has undergone. Some materials, such as aluminum and magnesium, are naturally more machinable than others. When the composition of a metal is altered 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, and 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 relatively easy to machine, the most notable exceptions being titanium and nickel. The machinability of a material has a direct effect on the life of the cutting edge of a tool, and thus influences the design of the cutting tool.

17-2. Metals can be rated according to their machinability. This is sometimes done. Industry has not found it practical to develop machinability ratings as an aid to the machinist. From experience it has been found that different groups of metals should be machined at speeds which fall within different cutting foot speed ranges. The recommended cutting foot speed range is 80 to 110 (CFS) for low carbon steels, 60 to 80 (CFS) for medium carbon steels, and 50 to 60 (CFS) 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 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 which 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. Since there are so many variables, a machinability rating would not help you in determining the most efficient speeds and feeds. This is something which you can only learn through actual experience

18. Nondestructive Inspection

18-1. 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.

18-2. 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.

18-3. Nondestructive inspection (NDI) includes the preparation of metals and parts for testing. It also includes the use of the various methods magnetic particle, penetrant, ultrasonic, eddy current, conductivity meter, ultrasonic leak, and radiographic. Nondestructive inspection is concerned with the types, causes, and characteristics of discontinuities and defects. It is concerned with the conditions requiring inspection and the interpretation and evaluation of indications found by the various methods.

18-4. For many years NDI was practiced only in isolated cases and the methods used were crude and often unreliable. Critical parts were load tested, with an overload often applied. If a part could withstand a single cycle of such a load, the conclusion was that it would be safe enough to use in service.

18-5. Today, it is known that single, or static loads, are not always significant. Many parts are subjected to repeated loads and even reversing loads. Under such circumstances, failures are produced progressively. Each applied load, or cycle, causes a small crack to progress and grows into a larger crack. In time the cross section carries the applied load and the part fails. This type of failure is called fatigue failure. Such failures can take place at stresses 30 to 40 percent lower than the tensile strength of the part.

18-6. Fatigue cracks may develop from improper design of the part or from defects in the metal. It is also possible for fatigue failure to occur in parts that are well designed and made from sound material. This can occur because parts are sometimes subjected to accidental overloads. Such overloads may be caused by misuse or failure of other parts or components in the system which result in a redistribution of loads. The purpose of periodic NDI tests is to detect such fatigue cracks before complete failure of the part occurs.

18-7. Training is extremely important in nondestructive inspection. Present nondestructive inspection methods are becoming more and more exacting. The importance of the decisions of the person who performs these inspections cannot be overemphasized.

18-8. 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.

18-9. In field maintenance activities all parts that require inspection should be taken to an inspection laboratory. A central location prevents a 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.

19. Corrosion Control and Plating

19-1. Corrosion is of major importance to all Air Force maintenance personnel. Most metals are subject to corrosion, but corrosion can be minimized by the use of corrosion resistant metals, finishes, and plating.

19-2. **Corrosion Control.** Corrosion is the deterioration of a metal by reaction to its environment. The corrosion occurs because of the tendency for most metals to return to their natural state. For example, iron in the presence of moist air reverts to its natural state, iron oxide. Pure metals, such as gold and platinum, do not corrode since they are chemically uncombined in their natural state. Metals can also be corroded by the direct reaction of the metal to a chemical.

19-3. *Identification of corrosion.* Since corrosion always starts on the surface of a metal, you can often detect it visually. To assist you in the interpretation of corrosion, we list some of the most common types.

a. *Pitting* corrosion, the most common type found on aluminum and magnesium, results in a white or gray powdery deposit. When it is removed, small holes or pits are visible on the surface.

b. *Intergranular* corrosion is usually caused by imperfect heat treatment and attacks at the grain boundaries of a metal. All metal alloys are composed of minute grains, each having a different composition at the center than at the grain boundary. A small corrosion cell is established when an electrolyte is in contact with the metal surface, causing rapid corrosion at the grain boundaries. The intergranular corrosion appears as a bulky mass.

c. *Exfoliation* corrosion is a form of intergranular corrosion. It starts at the surface and progresses along the grain boundaries under the surface of the metal. The expanding corrosion causes the surface 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 establishes an electrochemical cell and corrosion occurs. This type may be recognized by the presence of a buildup of corrosion at the joint between the metals. The greater the electrical potential difference between the metals, the greater the resulting corrosion.

e. *Stress* corrosion cracking is caused by the simultaneous effects of tensile stress and corrosion. When a metal is stressed to a degree within the critical stress range for that particular metal, corrosion will occur at the point of stress concentration. The combination of the corrosion and the concentrated stress causes the metal to crack.

19-4. *Prevention of corrosion.* Scheduled inspection and preventive maintenance are essential for the early correction of corrosion. The principal corrosion preventive which is used in airframe structures is aluminum alloy sheet coated with commercially pure aluminum (alclad). Alclad is more corrosion resistant than the nonclad high strength alloys. Other metals commonly used in airframe structures, such as nonclad high-strength aluminum alloys, steel, and magnesium alloys, require special preventive measures to guard against corrosion. Aluminum alloys, for example, are usually anodized or chemically treated and painted. Steel, except most stainless steels, and other metals, such as brass and bronze, require cadmium or zinc plating, conversion coating, paint, or all three, for protection. Magnesium alloys are highly susceptible to corrosion attack, especially in marine environment. These materials require special chemical treatment and paint finishes.

19-5. The most vital factor in preventing corrosion, and one which can be controlled only by field personnel, is the removal of the electrolyte. The term "electrolyte" refers here to contaminating materials (moisture, scale, dirt, grease, fluids, etc) which come in contact with the metal surfaces. The amount of corrosion is determined by the composition of the contamination, the type of metal, and the length of time the contaminant and the metal remain in contact. Corrosion can be reduced by more frequent cleaning. Anodic and chemical treatments, as well as proper paint finishes, should be applied in the critical areas to protect aluminum and magnesium alloys. The purpose of these coatings is to provide a barrier to inhibit corrosion.

19-6. **Plating.** One means of protecting a metal from corrosion is to plate its surface with another more corrosion-resistant metal. In some cases, the plating gives the metal a hard wear-resistant surface in addition to protection against corrosion. This is particularly true of chromium plating. Other metals used for plating are cadmium, copper, nickel, silver, and tin.

19-7. Plating is applied by a process called electroplating. A pure metal and the part to be plated are placed in a liquid solution (electrolyte). With the pure metal as the positive electrode (anode) and the part to be plated as the negative electrode (cathode), electrical current is passed through the electrolyte. This causes tiny particles of the pure metal to be deposited on the surface of the part. The process is continued until the plating is of the required thickness. The thickness of the plating required varies, but in most cases it is extremely thin. The same surface may be plated with two or more of these metals to make the plating adhere better or to reduce cost. For example, in the chromium plating of steels, the steel is often plated first with copper, then with nickel, and finally with chromium. The various plating processes are similar. For more detailed information concerning any specific plating operation, consult either the appropriate technical order or metals finishing handbook, or consult the people in the metals processing shop.

19-8. The most common metals used in electroplating in the Air Force are cadmium, copper, chromium, and nickel. Electroplating is the responsibility of the metals processing shop.

CHAPTER 5

Stud, Plug, Screw and Insert Removal and Replacement

YOU FREQUENTLY have to remove and replace studs, plugs, screws, and inserts. Certain types of aircraft studs require frequent removal and replacement, especially those that are subjected to stress caused by the speed of modern aircraft. When a stud must be replaced as a result of damage caused by heavy duty, it is necessary to inspect the threads of the parent metal. A careful examination often reveals thread weakness. Repairing the threads prevents the threads from tearing out when a replacement stud is screwed in place. Stud failure may be due to several reasons other than stretch and intermittent loading. Vibration and exposure to heat are common causes of stud failure on aircraft engines. Disregard of technical orders and safety factors by the aircraft mechanic may result in stud failure. For instance, excessive torque on the stud or wear and tear from abuse of parts in dismounting and mounting may cause failure. Regardless of what caused the failure, it is the job of the machinist to remove the stud, plug, screw or insert without damaging the parent part from which it is removed.

2. In this chapter we discuss several methods of removal of studs, plugs, screws, and inserts. Then we discuss the types and uses of inserts. Finally, we describe methods of removal.

20. Methods of Removal

20-1. Removal may be easy or difficult, depending on the circumstances. Sometimes none of the methods we discuss will work and you will have to devise a method of your own.

20-2. **Use of Penetrating Oil or Heat.** A stud that must be removed because of breakage, stretching or other damage, may often-times be easily removed if it is loosened by heating the area surrounding it. For very thin walled aluminum sections, the use of a soldering iron for heating the area around the broken stud may be an effective aid in its removal. This method would be used in a shop or away from the aircraft due to safety regulations. Another possibility for loosening a stud is the application of penetrating oil to the stud and mating part. Penetrating oil sometimes breaks up the rust or corrosion between the threads and allows rather easy removal. It has been found that a light tapping with a hammer on the part to be removed sometimes adds to the effectiveness of the penetrating oil. In using penetrating oil, caution should be exercised that it does not soak any electrical wiring and damage or deteriorate the insulation.

20-3. **Use of Extractors and Stud Driver.** Stud drivers and screw extractors, such as the Ezy-Out, may be used to advantage for stud, plug, screw, or insert removal. If the part to be removed is not rusted or corroded, and an Ezy-Out is not available, a good stud driver can be ground from a high-speed tool. The tool is ground with a square tapered end, as in figure 26. The square tapered end is then lightly driven into the hole that is drilled in the stud. This cuts into the walls of the stud, and holds the tool when it is twisted with a wrench. Usually, if the stud is rusted or corroded, the tool will slip out when too much twisting action is applied. If a set of Ezy-Outs are available, it would probably be best to first try removing the stud with them.

20-4. A commercial Ezy-Out is a tool that may be used with a wrench or tap wrench. It has a tapered square with a left-hand twist, giving it somewhat the appearance of a left-hand thread, as illustrated in figure 27. The working principle of an Ezy-Out is that when it is inserted in the drilled hole in the stud and twisted in the direction for removing that part, it tends to screw in and lock itself into position. A stud or plug can usually be removed by locking an Ezy-Out into position and applying the proper amount of pressure to turn it out. However, the Ezy-Out will not always remove the part because of existing, unseen conditions. It is also easily broken by too much pressure. It is best to use the largest possible Ezy-Out on the part to be removed. For ordinary conditions, the makers of Ezy-Out screw

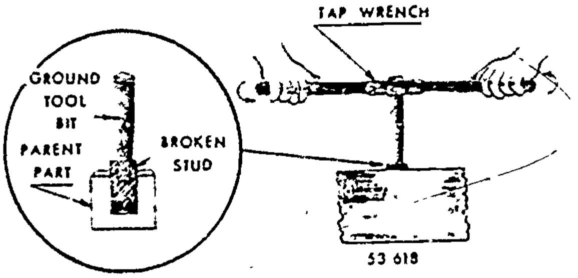


Figure 26 Stud Driver Made of High-Speed Tool Bit.

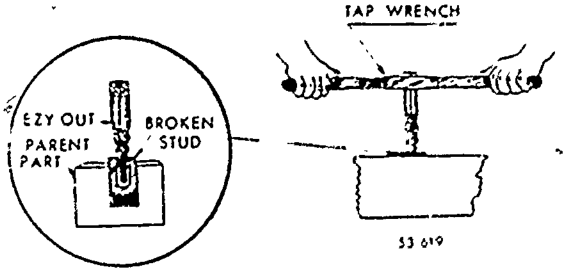


Figure 27 EZY-Out Screw Extractor

extractors recommend using the size drill shown opposite the Ezy-Out size indicated in table I Unusual conditions may require the use of larger drills depending on the length of the broken part or its depth in the hole. When you are drilling on an aircraft, consult the technical order or the line chief, to avoid drilling into an oil reservoir or jacket.

20-5. **Arc Welding.** Arc welding a nut on a worn or damaged stud is another method for

TABLE I
DRILL SIZES FOR EZY-OUTS SCREW EXTRACTORS

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

53150 3 4-63

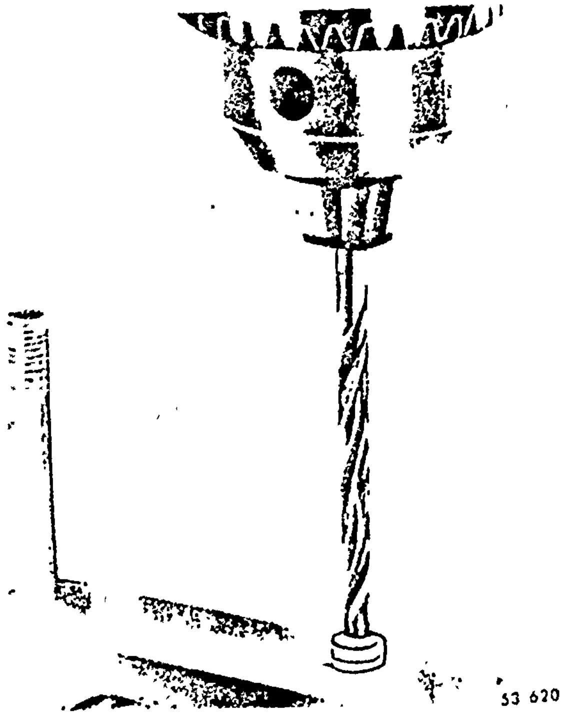


Figure 28 Removing a Stud by Drilling.

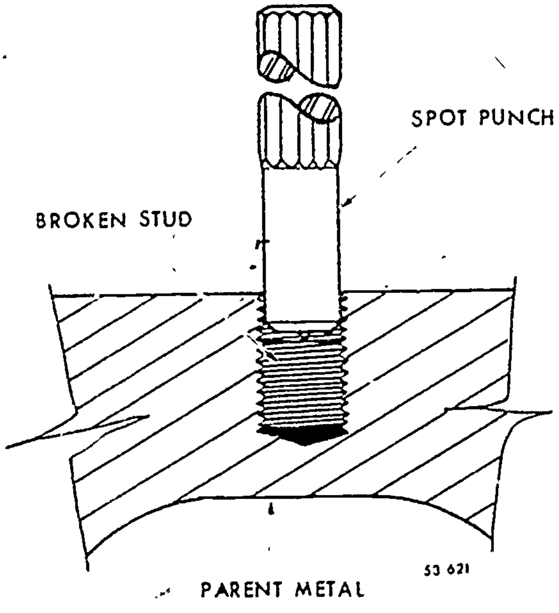


Figure 29 Spot Punching a Broken Stud.

the removal of studs. This is only possible when enough of the stud protrudes to weld on a nut. It is also possible to build up a stud that is broken off below the surface of the parent part by arc welding metal to it. Adding metal by welding builds up the stud so that a nut could be welded to it for its removal. This method, however, is not a job to be done by a machinist. If a welder is available to do the work, he should do it. Great caution should be exercised to protect the parent part if this method is used. Insulating material, such as sheet asbestos, should be used to protect the parent part from welding spatter. Also, the welder should be sure that no contact is made between the welding rod or nut and the parent part. Usually, the welder is called in to remove a broken stud as a last resort. Frequently, he can be of assistance when all methods have been exhausted. It is to be remembered that a welder may only work on an aircraft part that has been removed from the craft and brought into the shop. Welding equipment cannot be used around aircraft on the line.

20-6. **Removal of Faulty Studs by Drilling.** Broken or faulty studs may be removed by drilling out of the portion left in the parent part, as shown in figure 28. There are several ways of removing broken or faulty studs by drilling. Probably the best and most accurate method to accomplish the job is with the aid of a jig. Often, due to close working quarters, a jig cannot be used and the drilling will have to be done without a jig. Usually, you should try to spot punch the broken stud as in figure 29, and then drill for an extractor. Accuracy in drilling out a broken stud depends entirely upon your skill. After drilling a hole, try to remove the faulty stud by the use of extractors. If this method fails, you will have to drill it out. The object of drilling out a stud is to drill out the exact center and not touch the threads in the parent part. If a thin enough wall is left, it can usually be removed with the point of a scribe.

20-7. **Removal of Studs and Plugs by Trepanning.** A stud or plug may also be removed by trepanning. Trepanning is generally used when the parent metal around the stud has become damaged and it is impossible to save the threads. This should be used only as a last resort for stud removal, since it involves making a replacement with an insert or a stud having oversize threads. Since there are no regular commercial trepanning tools available, it becomes necessary to make a cutter when the trepanning method is used for stud removal. A trepanning tool may be made to fit almost any size stud and is illustrated in figure 30. A trepanning tool works like a hollow mill. The tool has teeth on the end and cuts down around the stud so that it may be removed. The disadvantage of this operation is that the

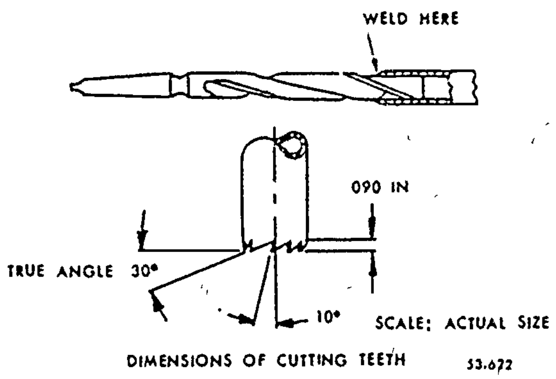


Figure 30. Trepanning Tool.

threads in the parent part are destroyed and oversize threads have to be tapped in the parent part. A stud with the same size thread or an insert is fitted to the tapped hole. This method of stud replacement is not recommended unless there is absolutely no other possibility of making the replacement with a standard stud.

20-8. **Other Methods of Stud Removal.** There is no set specific procedure that can be recommended for the removal of a stud, plug, screw or insert. Regardless of the method, however, the first concern in removing a stud is protecting the threads in the parent metal. When the thread in the metal is destroyed, it is either necessary to put in an insert or manufacture an oversize stud for replacement. The job of stud removal may be simple or it may be difficult. It is sometimes possible to file a square on a stud that protrudes above the parent metal, and with the aid of a wrench and some penetrating oil, to remove the stud easily. Occasionally, a stud that is broken off flush with the parent metal can be jarred loose with a punch and twisted out by hand. If there is a remaining thread on the stud, two nuts can be locked on the stud, and a wrench used to remove the stud. When a stud is broken off in an area where there are two adjacent studs, a jig can be made for drilling out the stud. The jig is so made that when it is clamped by the two adjacent studs, a guide hole lines up over the broken stud. This makes the drilling operation easy and accurate. Even though, in most cases, you cannot determine what has caused the stud failure and have no way of knowing how tight the part is in its parent metal, it will be up to you to remove the broken part. Removal may be accomplished by one of the above mentioned methods or one of your own. In conclusion, it should be remembered that the many studs, plugs, screws or inserts will have to be removed from the aircraft on the line. The ideal place for stud removal is the shop, provided that the part can be removed. The methods used most often are drilling and the use of commercial or handmade extractors when the work is done on the aircraft. Ingenuity is often a great help. A certain type stud may come out very easily using one method, and another identical stud, having undergone different conditions, may not come out at all by that method.

21. Types and Uses of Inserts

21-1. Inserts provide a necessary function in the repair and manufacture of many parts. As a machinist you are called upon to repair parts that require those inserts. We discuss the two basic types of inserts, Heli-Coils and solid inserts, and their uses.

21-2. **Heli-Coil Inserts.** Heli-Coil screw thread inserts are thread liners that are made of stainless steel of phosphor bronze wire. A cross section of the wire in one of these inserts has a diamond shape, as shown in figure 31. Heli-Coil inserts started as Aero-thread inserts that were invented by an aeronautical engineer. Aero-threads were distinguished by a pear-shaped cross section. They were widely used during World War II. Their shape proved to be a handicap because of the special thread shape. The cross section of the inserts was changed to a diamond shape and the name was changed to Heli-Coil inserts.

21-3. Heli-Coil inserts provide new threads that resist wear, corrosion, stripping, galling, and cross threading. On new equipment, these inserts prevent tapped thread failures before they start. They reduce maintenance costs, and because of their great strength, permit the use of smaller and fewer screws, smaller bases, and thinner flanges. Their use provides an easy, inexpensive way to repair a damaged or stripped thread. Another advantage is that the material in the inserts is harder than most screw threads that fit into them. Consequently, if damage occurs, it is the screw and not the insert that is damaged. Due to the smooth finish on the insert, there is less danger of seizure of a mating screw thread, especially around aircraft engines where there is heat. The use of these inserts reduces the effect of thread strains illustrated in figure 32. The use of Heli-Coil inserts has become wide and varied. Originally, they were used on aircraft, but now they are widely used for original application, repair and salvage in civilian and military equipment. This includes piston-engine and jet-engine aircraft, missiles, rockets, vehicles of all types, electronic apparatus, industrial machinery, machine tools, portable power tools, and contractor's equipment. Heli-Coil inserts are used on such materials as aluminum.

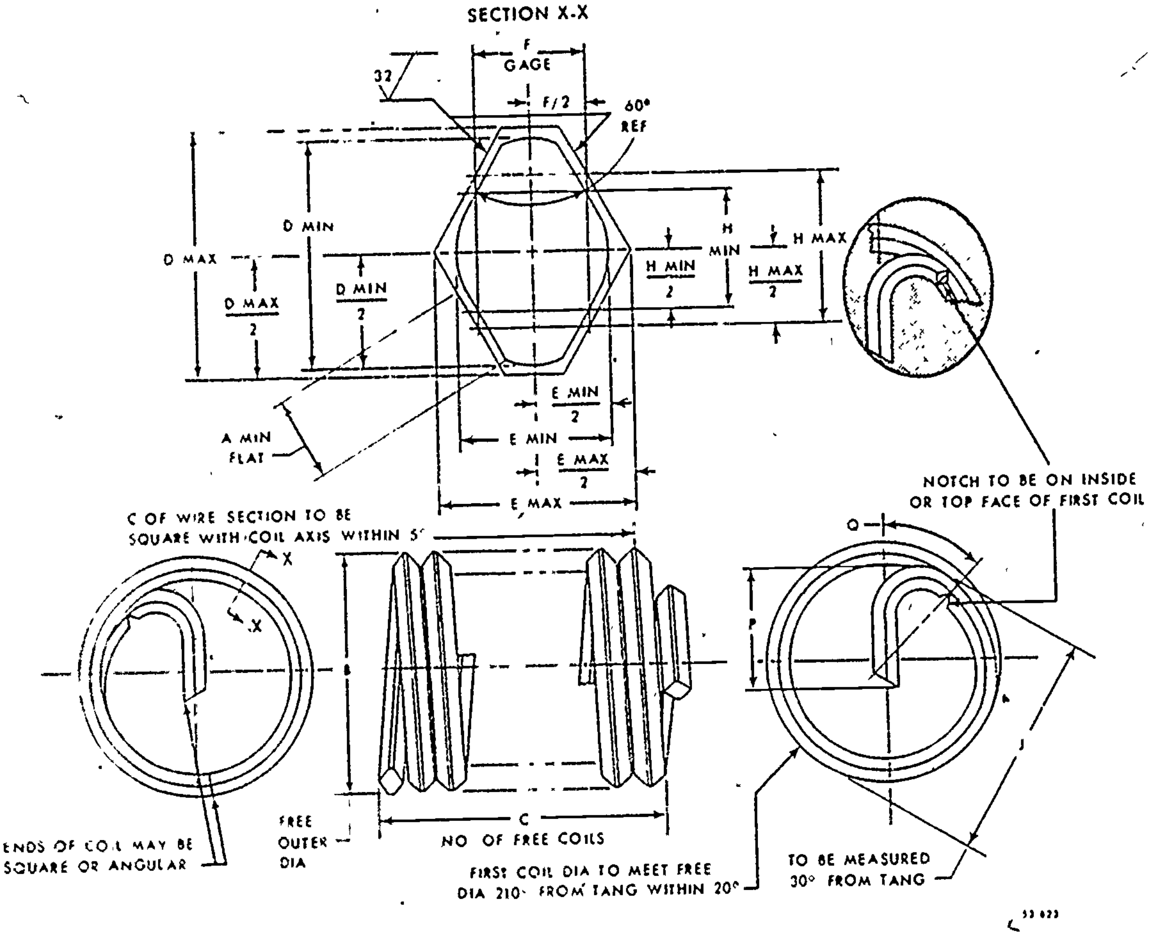


Figure 31 Diagram of a Heli-Coil Insert.

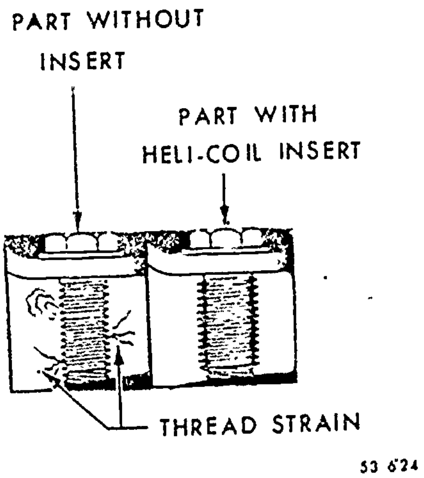


Figure 32 Thread Strains.

magnesium, titanium, ferrous metals, copper, brass, bronze, plastics, fiberglas, and wood. Heli-Coil inserts readily meet the present day demand for maximum performance of threaded assemblies with minimum space and weight requirements called for in military aircraft and related equipment. On aircraft they are especially beneficial when they are used in aluminum and magnesium. These inserts are being used more and more in engine manufacture.

21-4. **Solid Inserts.** Solid inserts are similar to Heli-Coils in that they also are used in the manufacture and repair of parts. It should be remembered that it takes a great deal more room to install a solid insert. Figure 33 shows the method of installing solid inserts.

21-5. In most cases, bronze is a suitable material for inserts, but if heavy or intermittent loads are encountered, it may be desirable to use stainless steel or nickel steel. A suitable design for smaller inserts is shown in figure 33. No dimensions are given because the various individual cases require different sizes. However, the pitch diameter of the threads or inserts should be from .002" to .0025" larger than the pitch diameter of the threads in the hole in which the insert is to be installed. This will make it a tight fit and keep the insert from working loose. If there is sufficient room, the flange of the insert should be made wide

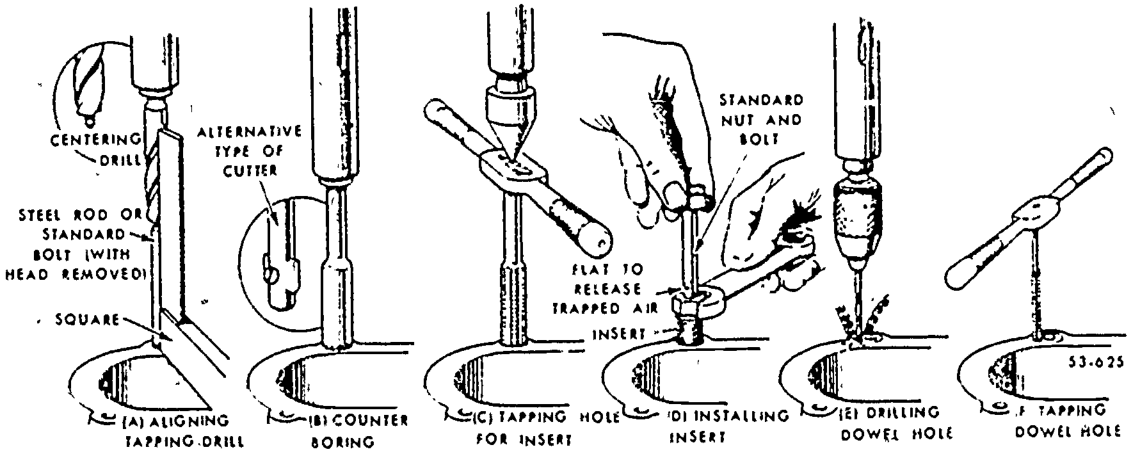


Figure 33. Method of Installing Solid Inserts

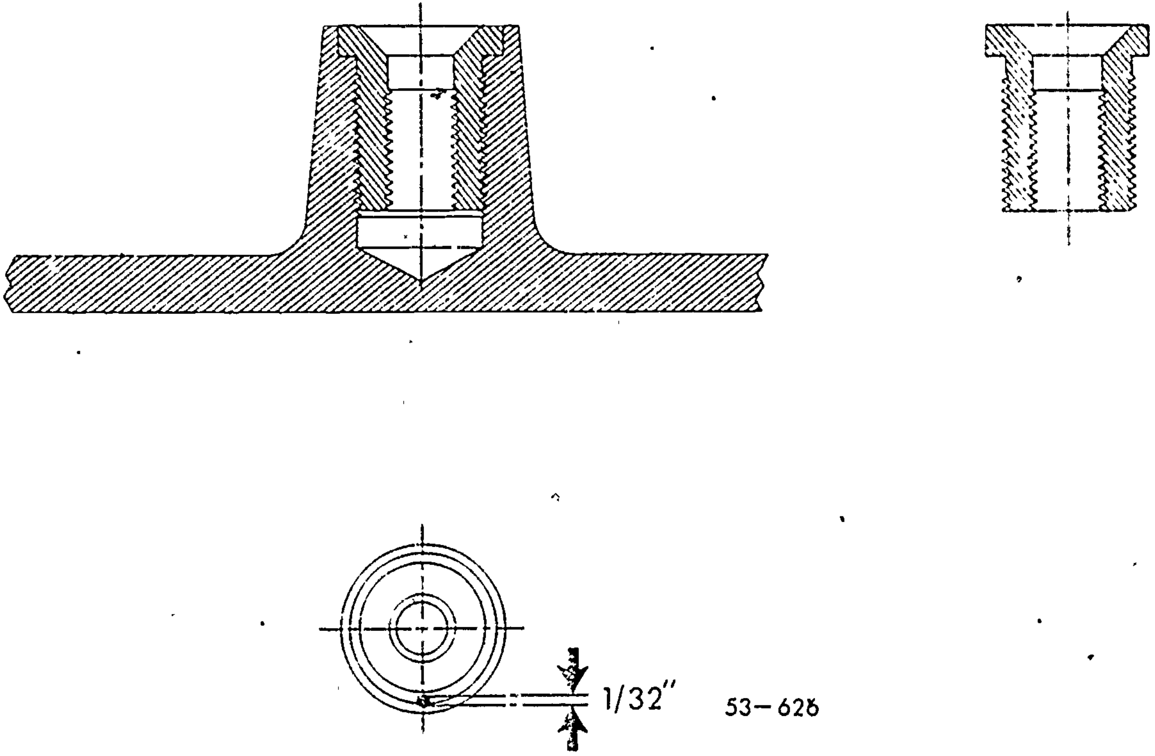


Figure 34 Solid Inserts

for smaller studs, either one or two holes for the dowel pins should be drilled and tapped to a size suitable for the installation of threaded dowels. The dowels are usually made of stainless steel rod about 1/16" to 3/32" in diameter or even larger in the case of larger inserts.

22. Methods of Replacement

22-1. Successful stud, plug, screw or insert replacement usually depends on the removal of the faulty part In many cases, if no trouble is encountered in the removal of the faulty part and the threads are not damaged in the parent part, it is only necessary to clean the internal thread with a tap and make the replacement. If trouble is encountered in removing a stud and the internal thread in the parent part is damaged, it is necessary to tap an oversize internal thread and install a replacement part to fit. It is also necessary to make a replacement with an oversize part when the part has been drilled out or trepanned. The reason is that

43

Bibliography

ECI Courses

CDC 53150, Volume 1, *Machine Shop Fundamentals.* Extension Course Institute, Gunter Air Force Base, Alabama.

CDC 53170, Volume 2, *Tool Design and Machine Work.* Extension Course Institute, Gunter Air Force Base, Alabama.

CDC 42152, Volume 1, *Career Fundamentals.* Extension Course Institute, Gunter Air Force Base, Alabama.

Books

BURGHARDT, Henry D.; Axelrod, Aaron; and Anderson, James. *Machine Tool Operation, Part I,* Webster Division, McGraw-Hill Book Company, New York, 1959.

BURGHARDT, Henry D.; Axelrod, Aaron; and Anderson, James. *Machine Tool Operation, Part II,* Webster Division, McGraw-Hill Book Company, New York, 1960.

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

OBERG, Erik, and Jones, Franklin D. *Machinery's Handbook. Eighteenth Edition.* New York: The Industrial Press, 1970.

Department of the Air Force Publications

AFR 205–1, *Safeguarding Classified Information,* 2 January 1968.

AFM 39–1, *Airman Classification Manual,* 29 December 1969.

AFM 66–1, *Maintenance Management,* 10 February 1970.

AFM 127–101, *Industrial Safety Accident Prevention Handbook,* 26 June 1970.

TO 00–5–1, *AF Technical Order System,* 15 June 1971.

TO 00–5–2, *Distribution and Storage of Air Force Technical Order System Publications,* 1 November 1971.

TO 00–110N–3, *Requisition, Handling, Storing and Identification of Radioactive and Radioactively Contaminated Material (Non-Fission Product),* 15 September 1970.

TO 1–1A–8, *Engineering H/B Series for Aircraft Repair—Acft Structural Hardware,* 15 June 1971.

TO 1–1A–9, *Engineering Series for Acft Repair—Aerospace Metals—Gen Data and USAGE Factors,* 15 June 1971.

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

TO 32–1–201, *Care, Use and Maintenance of Measuring Tools,* 1 July 1961.

TO 42D–1–3, *Method of Identifying Steel, Aluminum, and Copper Alloys in AF Stock,* 16 December 1969.

Resident Courses

Training Literature from Block I, 3ABR53130, *Machinist.*

Training Literature from Block IV, 3ABR53130, *Machinist.*

Training Literature from Block VII, 3ABR53130, *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*.)

50

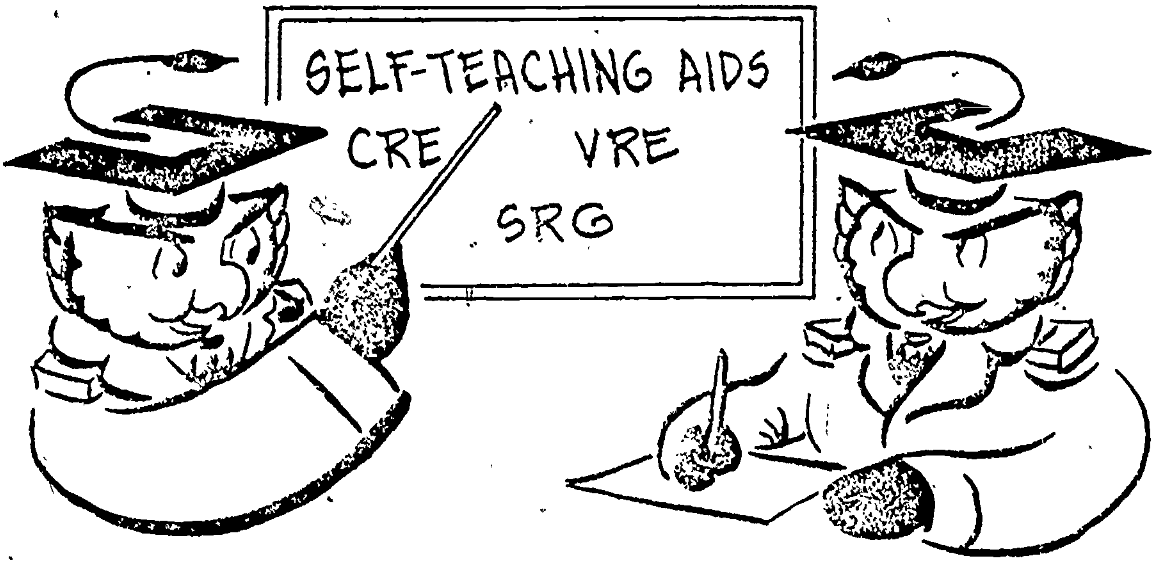
either drilling or trepanning removes the thread from the parent part. Even if the part were removed and the internal threads damaged very little, an oversize stud or an insert should be used. This is especially true on aircraft, where safety depends on perfect threads in every part.

In maintenance work on equipment other than aircraft, it is up to the machinist to make decisions about replacing the faulty part. When you work on aircraft parts it is always best to first check the TOs for specific procedures in the repair you are doing.

13-3 44

53150 01 24
WORKBOOK

GENERAL SHOP MANAGEMENT



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

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 100 through 111

100 Introduction, Duties and Responsibilities, pages 1–4

101 Supervision; pages 4–7

102 Training; pages 7–14

103 Technical Publications; pages 15–23

104 Supplies and Equipment; pages 23–29

105 Radiation Hazards and Flight Line Safety, pages 29–34

106 Introduction; Machinist Special Tools, pages 35–39

Guide Number

107 Measuring Devices and Processes; Care of Precision Instruments; pages 39–45

108 Introduction; Lubrication; Hydraulic Fluids; Cutting Lubricants and Coolants; pages 46–51

109 Introduction; Properties and Characteristics of Metals; Identification; Effects of Heat Treatment; Hardness Testing; pages 52–56

110 Machinability of Metals; Nondestructive Inspection; Corrosion Control and Plating; pages 56–58

111 Introduction; Methods of Removal; Types and Uses of Inserts; Methods of Replacement; pages 59–65

MODIFICATIONS

Page 3-8 of this publication has (have) been deleted in adapting this material for inclusion in the "Trial Implementation of a Model System to Provide Military Curriculum Materials for Use in Vocational and Technical Education." Deleted material involves extensive use of military forms, procedures, systems, etc. and was not considered appropriate for use in vocational and technical education.

49

Objectives. To show a knowledge of the selection and use of machinist's special tools in repair and fabrication. To be able to evaluate the use of measuring devices and procedures. To know how to care for precision instruments.

1. Compare bench plates and surface plates with regard to their construction and use. (7-2,4)

2. What is the primary use for parallels? (7-5)

3. For what kinds of work are magnetic parallels especially useful? (7-9)

4. Why is an adjustable angle plate more useful than a standard angle plate in mounting work on a machine tool table? (7-14,15)

5. When would you use toolmaker's buttons? (7-17)

6. What precision devices can be used to measure the center distance between toolmaker's buttons? (7-19,20)

7. Why are flat and curved bearing surfaces hand scraped? (7-22–25)

8. How are high spots located on a flat work surface that needs hand scraping? (7-27)

9. How can you tell that you are doing a good job of scraping? (7-27)

10. Over which of the three types of measuring errors do you have only minimum control? (8-3–5)

11. Which of the attachments for a graduated rule permits you to use the rule as a depth gage? (8-7)

12. Why is the caliper rule a more accurate measuring device than the graduated rule? (8-10)

13. What advantages do a vernier caliper have over either an outside or an inside micrometer? (8-11)

14. Explain how the basic principle of operation of the vernier caliper and the vernier height gage is the same. (8-14)

15. What is the maximum range of measurement for a 10-inch vernier height gage? (8-14,15)

16. What shaft conditions relating to concentricity can be checked with a vernier height gage and a dial indicator? (8-17–20)

17. How does the scale of a vernier bevel protractor differ from the scale of a bevel protractor used with a combination set? (8-23,24)

18. Compare Grade B with Grade A precision gage blocks for accuracy. (8-27)

19. What is one of the primary uses for precision gage blocks? (8-31)

20. Compare the use of precision angle blocks and precision gage blocks. (8-32)

21. What part of a right triangle is represented by a sine bar? (8-36)

22. What conditions should be avoided in storing precision instruments? (9-3)

CHAPTER 3

Objectives. To show an understanding of the types of lubricants and the lubrication of bearings and machine tools. To exhibit a knowledge of hydraulic fluids and their characteristics. To show a knowledge of the types of cutting lubricants and coolants and their uses.

1. How does oil reduce the friction between plain bearing surfaces? (10-2)

2. Why do antifriction bearings offer less resistance than plain bearings to starting motion? (10-3)

3. In which of the basic types of oiling systems is cooling, as well as lubrication, important? (10-4)

4. Which oiling device uses the principle of capillary action to provide oil to bearing surfaces? (10-5)

5. What are the advantages of using a grease lubricant? (10-7)

6. Match the greases in Column A with one or more characteristics in Column B.

COLUMN A

a. Lime-soap
b. Soda-soap
c. Metallic-soap

COLUMN B

1. Solid
2. Adhesiveness
3. Smooth texture
4. Frequent application
5. Fibrous texture
6. Ability to withstand high temperature

(10-8- 10)

7. How is it possible to make use of solid lubricants? (10-11)

8. Why is it always best to consult the operator's manual for the lubrication requirements for a machine tool? (10-12)

9. Why is it important to use the proper grade of hydraulic fluid for a machine tool? (11-1)

10. List the three types of hydraulic fluid and indicate which is best for high operating temperatures. (11-3–5)

11. Why can higher cutting speeds be used with a coolant? (12-1)

12. What must be added to soluble oils to make them satisfactory for use as coolants? (12-5)

13. What types of cutting fluids are best for machining brass? (12-7)

CHAPTER 4

Objectives. To show a knowledge of the properties and characteristics of metals, to be able to identify the various types of metals, to exhibit an understanding of the effects of heat treatment on metals, to be able to specify the required hardness to be obtained in hardness testing, to exhibit an understanding of the machinability of metals, to show a knowledge of nondestructive inspection, and to show a knowledge of corrosion control and plating.

1. What are the mechanical properties of a metal? (13-2)

2. Identify the following phrases with the appropriate mechanical property.

   a. Resistance to being pulled apart.
   b. Resistance to deformation or penetration.
   c. Tendency to fracture with no deformation, bending, or twisting.
   d. Ability to be elongated without breaking.
   e. Ability to absorb sudden shock without breaking.
   (13-3)

3. What is meant by the chemical composition of a metal? (13-5)

4. List the forms in which a metal may be found. (13-6 9)

5. What is the most positive identification of a metal? (14-3)

6. List in order of preference the approved methods of marking metal. (14-4)

7. What would a gold or silver stripe in a color coded piece of steel indicate? (14-6)

53

8. What does a machinist need to know about heat treating? (15-1)

9. Why is steel tempered after being hardened? (15-4)

10. Compare annealed and normalized steels. (15-5,6)

11. What characteristics in steel are obtained by case hardening? (15-7)

12. What are the two types of heat treatment operations which are used in the heat treatment of nonferrous metals? (15-8)

13. State the principle of operation of the Rockwell hardness tester. (16-1,2)

14. Name some of the factors that determine the machinability of material. (17-1)

15. What happens to the machinability of metal when its composition is altered by adding alloying elements? (17-1)

16. What is nondestructive inspection? (18-1)

17. Name some of the various methods of nondestructive inspection. (18-3)

18. At what percent of the tensile strength can fatigue failure take place? (18-5)

19. Name two causes of fatigue cracks. (18-6)

20. What is the purpose of periodic NDI tests? (18-6)

21. What is corrosion and why does it occur? (19-2)

22. List the four most common types of corrosion. (19-3)

23. What is the most important factor in the prevention of corrosion? (19-5)

24. List some of the common metals used in plating. (19-6)

CHAPTER 5

Objectives. To be able to evaluate the conditions affecting the removal of defective studs, plugs, screws and inserts and to determine the best method of removal, to identify the types and uses of inserts, and to evaluate the methods of replacing studs, plugs, screws, and inserts and to determine the best method to use.

1. How does penetrating oil aid in the removal of a broken stud? (20-2)

2. Why is it easier to remove a stud with a commercial Ezy-Out than with a stud driver? (20-3,4)

3. What are the two main precautions to observe when a nut is arc welded to a damaged stud? (20-5)

4. What is the most accurate method of drilling a stud? (20-6)

5. What is the main disadvantage of the trepanning method of stud removal? (20-7)

6 What is the most important precaution to observe in stud, plug, screw, or insert removal? (20-8)

7 Of what value are Heli-Coil inserts in replacing a damaged stud? (21-3)

8. List three metals that are suitable for solid inserts? (21-5)

9 What repair is necessary when the threads of the parent metal are damaged in stud removal? (22-1)

MODIFICATIONS

Pages 16-18 of this publication has (have) been deleted in adapting this material for inclusion in the "Trial Implementation of a Model System to Provide Military Curriculum Materials for Use in Vocational and Technical Education." Deleted material involves extensive use of military forms, procedures, systems, etc. and was not considered appropriate for use in vocational and technical education.

ANSWERS FOR CHAPTER REVIEW EXERCISES

CHAPTER 2

1. Both are constructed to provide flat surfaces upon which work is placed. The surface plate is constructed with much more precision and is used for performing tasks which require more precision.

2. To raise work above a surface and keep the work parallel with the surface.

3. To hold small, thin, or irregular work which cannot be held directly on a magnetic chuck.

4. Work mounted on an adjustable angle plate can be positioned at any desired angle from the horizontal to the vertical positions. The adjustable angle plate can also be rotated on its base. The standard angle plate permits mounting the work only at 90° to the machine table.

5. When it is necessary to locate holes with a very high degree of precision.

6. Gage blocks and a micrometer.

7. To reduce the friction between the bearing surfaces.

8. Rub a light coat of venetian red or prussian blue on the surface plate. Then rub the work surface over the surface plate. Paint will adhere to the high spots.

9. The paint spots increase in number and decrease in size.

10. Instrument error.

11. The sliding head.

12. Because it makes contact with the work being measured.

13. A vernier caliper can replace several inside or outside micrometers, and it can take either inside or outside measurements.

14. The surface plate, upon which the vernier height gage rests, corresponds to the fixed jaw of the vernier caliper and the sliding arm of the vernier height gage corresponds to the sliding jaw of the vernier caliper.

15. 9 inches.

16. Out of round, bent shaft, curvature, and eccentricity.

17. The scale of the vernier bevel protractor is graduated into increments of $1/12^\circ$ or 5 minutes. The scale of the bevel protractor used with a combination set is graduated into increments of 1°.

18. The length of Grade B precision gage blocks are accurate plus ten millionth (0.000010") and minus sixth millionth (0.000006"). The length of Grade A precision gage blocks are accurate within six millionth (0.000006") for blocks 1 inch and longer and within two millionth (0.000002") for blocks under 1 inch.

19. They are used as standards for checking the calibration of precision measuring devices.

20. Precision angle blocks are used to obtain precision angular measurements. Precision gage blocks are used to obtain precision linear measurements.

21. The hypotenuse.

22. Dust or dirt in the storage area, extreme variations in temperature, and relative humidity in excess of fifty percent.

CHAPTER 3

1. The oil acts as a film which separates the surfaces enough to prevent metallic contact.

2. Because the motion in antifriction bearings is a rolling motion rather than the sliding motion on plain bearings.

3. Circulation.

4. Wick.

5. A grease lubricant minimizes friction, wear, loss of power, and leakage.

6. *a.* 3, 4.
   b. 5, 6.
   c 2.

7. They must be bonded to the bearing surfaces.

8. Because oil companies refine oils and greases under so many trade names that it is difficult to recommend any of them.

9. To prevent damage to the seals.

10. Petroleum-base, vegetable-base, and synthetic base. Synthetic-base hydraulic fluid is best for high operating temperatures.

11. Because the coolant reduces the temperature of the cutting tool and permits it to retain its hardness.

12. A rust inhibitor.

13. Soluble oil or a mineral-lard oil.

CHAPTER 4

1. The internal reactions to external forces.

2. *a.* Tensile strength.
   b Hardness.
   c. Brittleness.
   d. Ductility.
   e. Toughness.

3. The number of elements and the exact percentage of each element that a metal possesses.

4. Pure, mechanical mixture, solid solution, and combination of solid solution and mechanical mixture.

5. Military or Federal Specification Number.

6. (1) Stencil and paint, (2) color code, and (3) stamping with metal dies.

7. Gold would indicate that the piece of steel was normalized, silver would indicate that it was of aircraft quality.

8. He must know the effects of heat treating and be able to specify the heat treatment that a piece of metal requires.

9. To relieve the internal stresses set up by hardening and to reduce the brittleness.

10. Normalized steel is harder and stronger than annealed steel.

11. A wear-resistant surface and a tough core.

12. Annealing and heat treatment.

13. The principle of operation is based on the distance that a penetrator will penetrate a piece of material under a given amount of pressure.

14. Composition of the material, grain structure, and the heat treatment it has undergone.

15. The machinability may changed.

16. Any form of inspection in which the part inspected is not damaged or destroyed.

17. Magnetic particle, penetrant, ultrasonic, eddy current, conductivity meter, ultrasonic leak, and radiographic.

18. At about 30 to 40 percent of tensile strength.

19. Improper design of parts and defects in the material.

20. To detect fatigue cracks before complete failure of the parts occurs.

21. Corrosion is the deterioration of a metal by reaction to its environment. Corrosion occurs because of the tendency of most metals to return to their natural state.

22. Pitting, intergranular, exfoliation, and galvanic.

23. The removal of electrolyte.

24. Cacmiun, copper, nickel, silver, tin, and chromium.

CHAPTER 5

1. By breaking up the rust or corrosion between the threads of the stud and the parent metal.

2. The Ezy-Out digs in the walls of the drilled stud better when pressure is applied because the tapered square end has a left-hand twist.

3. To protect the parent metal from welding spatter and to prevent any contact by the welding rod between the nut and the parent metal.

4. Drilling with the aid of a jig.

5. The threads of the parent metal are destroyed.

6. To protect the threads in the parent metal from damage.

7. Heli-Coil inserts provide new threads that resist wear, corrosion, stripping, galling and cross threading.

8. Bronze, stainless steel, and nickel steel.

9. Tap an oversize thread in the parent metal and install a stud with oversize mating threads to fit.

STOP— 1. MATCH ANSWER SHEET TO THIS EXERCISE NUMBER. 2. USE NUMBER 1 PENCIL.

53150 01 24

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 numerical sequence on answer sheet alternates across from column to column.

3. Use only medium sharp #1 black lead pencil for marking answer sheet.

4. Circle the correct answer in this test booklet. 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 with a #1 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 Text Page Numbers for these first three items.

1. The form number of this 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. (001) A duty that belongs to the machinist specialty is

   a. troubleshooting of difficult metal machining, design, and production problems.
   b. instructing in metals machining techniques and maintenance of machinery and equipment
   c. planning and organizing of metalworking activities.
   d. manufacturing and reworking of machined parts.

5. (002) The design and manufacture of precision tools, gages, dies, and jigs is a responsibility of the

   a. apprentice machinist.
   b. machinist.
   c. machine shop technician.
   d. metalworking superintendent.

6. (002–004) The duties and responsibilities of the metalworking superintendent do *not* include

   a. assembling and fitting machined parts.
   b. directing metalworking activities.
   c. inspecting and evaluating metalworking activities.
   d. establishing OJT for metalworking personnel.

7. (004) During the initial phase of setting up a machine shop, the supervisor is most likely to concentrate on the principle of

a. organizing.
c. planning.
b. coordinating.
d. controlling.

8. (004-005) Authorizing the assignment of a welder to duty in a machine shop would be a violation of the principle of

a. logical assignment.
c. delegation of authority.
b. unity of command.
d. span of control.

9. (005) A discussion initiated by a machine shop foreman with his two machinists to plan a work project is an example of

a. horizontal coordination.
c. lateral direction.
b. vertical coordination.
d. projected direction.

10. (006) Inspecting the work and evaluating worker performance in accordance with established objectives are good principles of

a. directing.
c. coordinating.
b. planning.
d. controlling.

11. (007) Under the dual-channel OJT concept, airmen achieve job proficiency by

a. performing on the job under the supervision of a skilled trainer.
b. studying a formal ATC resident course.
c. working under the supervision of airmen with a higher skill level.
d. studying a course developed and conducted at base level.

12. (008) The competence with which a machinist performs the tasks required for his position on the manning document are measured by using

a. AF Form 623.
c. a management guide.
b. STS 53230/50/70.
d. a Job Proficiency Guide.

13. (009) Which Air Force manual listed below provides information for making entries on AF Form 623, Consolidated Training Record?

a. AFM 50-25.
c. AFM 50-23.
b. AFM 50-24.
d. AFM 50-20.

14. (010) Who is responsible for a trainee's satisfactory completion of the career knowledge portions and the assigned job proficiency portions of the Specialty Training Standard?

a. The squadron OJT supervisor.
c. The shop supervisor.
b. The individual trainee.
d. The trainer.

64

15. (010) Which of the following is *not* a responsibility required of the trainee?

   a. To understand his AFS description, the training requirements of his STS and JPG, and the contents of his training folder.
   b. To accept and react favorably to the motivational effort of his OJT trainer and supervisors.
   c. To understand, accept, and perform all of the training requirements and assignments in an efficient manner.
   d. To maintain his consolidated training records in accordance with AFM 50–23.

16. (012) The training device containing a list of key points that a trainer should emphasize when he is teaching a trainee to perform a job is called a

   a. job breakdown.
   b. Job Proficiency Guide.
   c. Job Training Standard.
   d. Specialty Training Standard.

17. (013) The most effective use of CDC chapter review exercises (CREs) is for the student to

   a. read each CRE and immediately check the answer supplied in the workbook.
   b. write the answers for the CREs and submit them to ECI for grading.
   c. write the answers for the CREs on each chapter and to check them against the answers in the workbook.
   d. answer the CREs after studying the entire volume and check the answers against those in the workbook.

18. (014) Who has the most opportunity with his personal interest to promote the importance of the OJT program?

   a. The OJT squadron supervisor.
   b. The trainee.
   c. The trainer.
   d. The base commander.

19. (015) The index that contains lists of equipment that are cross-referenced to categories is the

   a. alphabetical index.
   b. cross-reference table TO 0–4–1.
   c. cross-reference table TO 0–4–2.
   d. numerical index and requirement table.

20. (016) To find a listing of each of the different categories of equipment, first look in

   a. TO 0–1–1.
   b. TO 01–1–1.
   c. TO 0–1–01.
   d. TO 1–1–1.

21. (016–017) The alphabetical index can be used to the best advantage when you need to

   a. locate specific technical orders.
   b. requisition technical orders.
   c. find the category to which an item of equipment belongs.
   d. insure locating a current technical order.

22. (017–018) In the number TO 0–1–1–3, an NI&RT, which part of the number indicates that the tech order is a sectionalized TO?

   a. The 0.
   b. The first –1.
   c. The second –1.
   d. The –3.

23. (018) Upon receipt of an urgent action TCTO that specifies you have 6 days to accomplish the work, the symbol you would enter in the maintenance form is a

a. red X.
b. red diagonal.
c. red dash.
d. black circled X.

24. (019) The type of TCTO which is issued as an electrically transmitted message is referred to as

a. immediate action TWX.
b. urgent TWX.
c. interim TCTO.
d. record message TWX.

25. (019) The disposition you make on incomplete TCTO kits is to

a. keep them until spare parts are ordered.
b. turn them in within 10 days if parts have not been located.
c. store them as extra parts.
d. turn them in immediately to the base supply officer.

26. (019) Methods and procedures technical orders (MPTOs) are issued to establish

a. policies, information, and instructions.
b. inspection workcards.
c. lubrication charts.
d. inspection checklists.

27. (021) A methods and procedures type TO is identified by which of the following numbers?

a. TO [illegible]
b. TO [illegible] 25A-247.
c. TO 1F-100A-1-7.
d. TO 21M-LGM30A-678.

28. (021) The method of modifying a technical order into a new edition with a new basic date is referred to as a

a. change notice.
b. revision.
c. recision.
d. supplement.

29. (022) Which of the following TO numbers identifies a tech order that contains information to correct work stoppage?

a. 1B-52G-[illegible]
b. 1C-[illegible]35B-6-SC-1PI
c. 1F-104B-1-SS-3.
d. 1F-106A-1-S-2.

30. (022, 023) The type of tech order file that a machinist uses more than any other is the

a. shop.
b. limited.
c. school.
d. aircraft.

31. (024) Which of the following forms is used to request issue [illegible] for the machine shop?

a. AF Form 601a.
b. AF Form 601b.
c. DD Form [illegible]
d. AF Form 1297.

66

32. (024) If a precision measuring gage is not on your tool list for issue, which form listed below would you sign at the tool crib to check out the gage?

a. AF Form 1295.
b. AF Form 1296.
c. AF Form 1297.
d. AF Form 1298.

33. (024) When a supply custodian finds an item of equipment during inventory that does not belong on his supply account, he forwards

a. an AF Form 601b to EMO.
b. a DD Form 1150 to Research.
c. a DD Form 1150 to Claims Investigation.
d. a DD Form 1150 to Materiel Control Section.

34. (025) The contents of an original container should be inspected and retagged when

a. it is shipped from a depot.
b. it is shipped from one base to another.
c. the label or tag has been damaged.
d. the container is shipped from one command to another.

35. (026) The DD Form 1577-2 is used to indicate that the item to which it is attached is

a. serviceable, with a listing of the missing items on the reverse side of the tag.
b. serviceable, but has suspected materials.
c. unserviceable and nonreparable.
d. unserviceable, but reparable.

36. (027) Who should inspect a piece of equipment that is sent to the shop from EMO?

a. The supply office.
b. The supply inspector.
c. The equipment management office.
d. The equipment inventory supply office.

37. (029) Airman Jones pays the $22 repair cost for a piece of equipment he damaged through neglect with a

a. Cash Collection Voucher.
b. Report of Survey.
c. personal check attached to a statement of charges.
d. supply voucher.

38. (033) If a placard attached to a crate in base supply has a yellow background with a three-bladed magenta-colored insignia on it, this indicates to you that the contents of the crate are

a. serviceable.
b. radioactive.
c. reparable.
d. explosive.

39. (033) The minimum safe distance behind an operating jet engine is

a. 150 feet.
b. 200 feet.
c. 250 feet.
d. 300 feet.

73

40. (033-034) Which of the following would be the greatest danger in the compressor bleed areas?

a. Temperature and velocity.
b. Noise and careless workers.
c. The turbine wheels.
d. The intake areas.

Chapter 2

41. (035) Bench plates are usually made of

a. stainless steel.
b. wrought iron.
c. cast steel.
d. cast iron.

42. (036) For which of the following work setups would you *not* be able to use a solid parallel?

a. Support work in a vise.
b. Raise work from a surface plate.
c. Mount work on a magnetic chuck.
d. Mount work on a rotatory table.

43. (036) The primary use of the planer gage is to

a. set the cutting tool.
b. check planed surfaces.
c. check the width of slots.
d. set up a sine bar.

44. (037) A precision tool that may be used in the work setup for accurately planing surfaces square to one another is

a. sine bar.
b. an angle plate.
c. a box parallel.
d. a surface plate.

45. (037) Using toolmaker's buttons (.500 inch in diameter) to accurately locate two holes with a center distance of 3.105 inches, what would be the distance between the toolmaker's buttons?

a. 1.605 inches.
b. 2.605 inches.
c. 2.855 inches.
d. 3.605 inches.

46. (037) When complete accuracy is necessary in the setup of toolmaker's buttons, which of the following precision tools should you use to obtain the required accuracy?

a. Gage block.
b. Sine bar.
c. Micrometer.
d. Vernier height gage.

47. (038) Which of the following types of hand scrapers should be used to scrape the angular portion of a dovetail bearing surface?

a. Half-round bent scraper.
b. Half-round scraper.
c. Hooked scraper.
d. Three-cornered scraper.

48. (039) Which of the following should be used for the layout of large circles and arcs?

a. Dividers.
b. Compass.
c. Template.
d. Trammels.

49. (039–040) Which graduated rule accessory enables you to use a rule as a depth gage?

a. Sliding head.
b. Keyseat clamps.
c. Caliper rule.
d. Hooked attachment.

50. (040) What is the smallest measurement which can be made with a vernier caliper?

a. 0.0001 inch.
b. 0.001 inch.
c. 0.010 inch.
d. 0.100 inch.

51. (041) The minimum height you can set the sliding arm gaging surface above the bottom of the base of the vernier height gage is usually

a. 1/2 inch.
b. 1 inch.
c. 1 1/2 inches.
d. 2 inches.

52. (041) To check a shaft for curvature, how many gage readings are necessary and how many degrees apart should they be made?

a. Two at 90°.
b. Four or more at 180°.
c. Three or more at 90°.
d. Three at 180°.

53. (042) The acute angle attachment is generally used for measuring angles less than

a. 90°.
b. 60°.
c. 45°.
d. 30°.

54. (043) One of the primary uses of gage blocks is to

a. set up precision work.
b. check other gaging and measuring instruments.
c. check machined parts for accuracy.
d. make linear measurements.

55. (043–044) Which of the following precision measuring devices can be used to check the accuracy of a sine bar setup?

a. Master sine bar.
b. Gage blocks.
c. Angle blocks.
d. A vernier bevel protractor.

56. (045) The most prevalent cause of excessive wear of precision gaging tools is

a. improper handling.
b. gaging surfaces not hard enough.
c. dirt and abrasive materials on both the part and the gage.
d. constant physical contact with parts being gaged.

57. (046) The most important cause for inefficient machine tool operation is

    a. friction.
    b. improper use.
    c. electrical failure.
    d. too much lubrication.

58. (047) Which of the following bearings offer the least amount of resistance to movement?

    a. Plain
    b. Antifriction.
    c. Thrust.
    d. Slipper.

59. (047) Which of the following is *not* a problem in the lubrication of antifriction bearings?

    a. Abrasion.
    b. Corrosion.
    c. Getting the lubricant to the bearing.
    d. Accumulation of harmful deposits.

60. (048) Which type of oiling system is used when cooling as well as lubrication is necessary?

    a. Circulation.
    b. Splash.
    c. Plunge
    d. Ring and chain.

61. (048) The bath oiling system is often used for the lubrication of

    a. plain bearings.
    b. thrust and vertical step bearings.
    c. slipper bearings.
    d. antifriction bearings.

62. (048-049) Lime-soap greases are excellent for use when

    a. the operating temperature is low.
    b. excessive churning is present.
    c. infrequent application is needed.
    d. the operating temperature is extremely high.

63. (049) The best source of information with reference to the lubrication needs of a particular make of machine is found in the

    a. TOs.
    b. MIL Standards.
    c. operator's manuals.
    d. oil company recommendations list.

64. (049) For easy identification, petroleum-based hydraulic fluids are dyed

    a. blue.
    b. red.
    c. yellow.
    d. brown.

65. (050) The primary purpose for cutting lubricants is to

a. improve work finish.
b. wash chips away from the tool.
c. lower work temperatures.
d. prevent work warpage.

66. (050) Which type of cutting oil would you most likely use for lathe and milling machine work when cooling is an important factor?

a. Soluble oils.
b. Lard oils.
c. Sulphurized oils.
d. Chlorinated oils.

67. (051) When a machining operation requires high-cutting pressure, the proper cutting lubricant to use is

a. chlorinated oil.
b. lard oil.
c. soluble oil.
d. sulphurized oil.

68. (051) In machining monel metal, a cutting lubricant preferred because of its tendency to aid in breaking of the chip is a

a. chlorinated oil.
b. lard oil.
c. sulphurized oil.
d. soluble oil.

Chapter 4

69. (052) When you increase the hardness of metal, which of the following takes place?

a. Brittleness is decreased.
b. Brittleness is increased.
c. Toughness is increased.
d. Shear strength is decreased.

70. (052) The hardness of a metal is described as the resistance it offers to

a. being pulled apart by an applied load.
b. cutting or abrasive actions.
c. sudden shock without breaking.
d. deformation or penetration.

71. (053) The form of a metal in which the elements and compounds are clearly visible and are held together by a matrix of base metal is called a

a. mechanical mixture.
b. solid solution.
c. pure metal.
d. combination of solution and mechanical mixtures.

72. (053) The preferred method of identifying metal pieces in the machine shop stock rack that do not bear an Air Force approved manufacturer's mark is by

a. tagging.
b. color code.
c. stencil and paint.
d. stamping with metal dies.

73. (054) The heat treatment operation in which a ferrous metal is heated to the required temperature and rapidly cooled is referred to as

a. annealing.
b. tempering.
c. normalizing.
d. hardening.

74. (054) The heat treatment that steel undergoes to release its internal stresses and to reduce its brittleness is called

a. normalizing.
b. annealing.
c. tempering.
d. hardening.

75. (054) If you are required to manufacture a part which must have a hard wear-resistance outer surface and a tough inner core, which type of heat treatment listed below should you use?

a. Hardening.
b. Tempering.
c. Normalizing.
d. Case hardening.

76. (055) The term "age hardened" is used to describe a nonferrous metal that has been given

a. the solution treatment.
b. the precipitation treatment.
c. maximum pliability.
d. maximum corrosion resistance.

77. (056) Which of the following elements, when added to carbon steel, tend to increase its machinability?

a. Chromium.
b. Sulphur.
c. Nickel.
d. Tungsten.

78. (057) The most important part of the visual inspection is its reliability which depends upon

a. the ability and experience of the inspector.
b. a good lighting system.
c. the correct methods and procedures employed.
d. the good judgment used in making decisions.

79. (057 058) The type of corrosion most commonly found on aluminum and magnesium is

a. intergranular.
b. exfoliation.
c. pitting.
d. galvanic.

80. (058) What is the most vital factor in the prevention of corrosion?

a. Identification of corrosion.
b. Removal of the electrolyte.
c. More frequent inspections.
d. Frequent painting.

Chapter 5

81. (059) A primary purpose of using penetrating oil in stud removal is to

a. clean the area being worked on.
b. lubricate the mating threads.
c. penetrate to the cutting edges of the drill.
d. break up corrosion between the threads.

82. (060–061) In the arc welding method of stud removal, the parent metal should be protected by using

a. thin pieces of metal around the stud.
b. sheet asbestos around the stud.
c. wet rags around the stud.
d. wood around the stud.

83. (061) Drilling for an extractor is preferable to completely drilling out a damaged stud because the former

a. can be done in less time.
b. does not require as much skill.
c. does not require the use of a drill jig.
d. is less likely to damage the parent part.

84. (061) The most important thing to remember in drilling out a stud is to

a. use the correct size of drill.
b. maintain a constant pressure on the drill.
c. use penetrating oil during drilling.
d. keep the drill as nearly on center as possible.

85. (061–062) The *least* preferred method of stud removal is

a. trepanning.
b. using a punch and hammer.
c. welding a nut on the stud.
d. drilling out the stud with a drill jig.

86. (062) With reference to stud removal, the greatest help will be your

a. hammer and punch.
b. drill.
c. EZY-out.
d. ingenuity.

87. (062) The cross section of the wire used in the manufacture of Heli-Coil inserts is shaped like a

a. diamond.
b. pear.
c. rectangle.
d. spring wire.

88. (062) Which of the following is considered the best reason for using Heli-Coil inserts?

a. Low resistance to wear.
b. No resistance to corrosion.
c. High resistance to stripping.
d. Low resistance to cross threading.

89. (063 064) When heavy or intermittent loads are encountered, solid inserts should be made of

a. bronze.
b. stainless steel.
c. manganese steel.
d. manganese bronze.

90. (064-065) The successful replacement of studs, plugs, screws, and inserts usually depends upon the

a. skill of the machinist.
b. type of aircraft involved.
c. removal of the faulty part.
d. where the job is located on the aircraft.

53150 02 7206

CDC 53150

MACHINIST

(AFSC 53150)

Volume 2

Advanced Machine Work

13-3

Extension Course Institute

Air University

76

Preface

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

Note the chapter titles on the content page for Volume 2. Chapter 1 covers drilling machine work; Chapter 2, lathe work; Chapter 3, milling machine work; Chapter 4, shaper work, Chapter 5, contour machine work; and Chapter 6, grinding machine 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.

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 (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 18 hours (6 points).

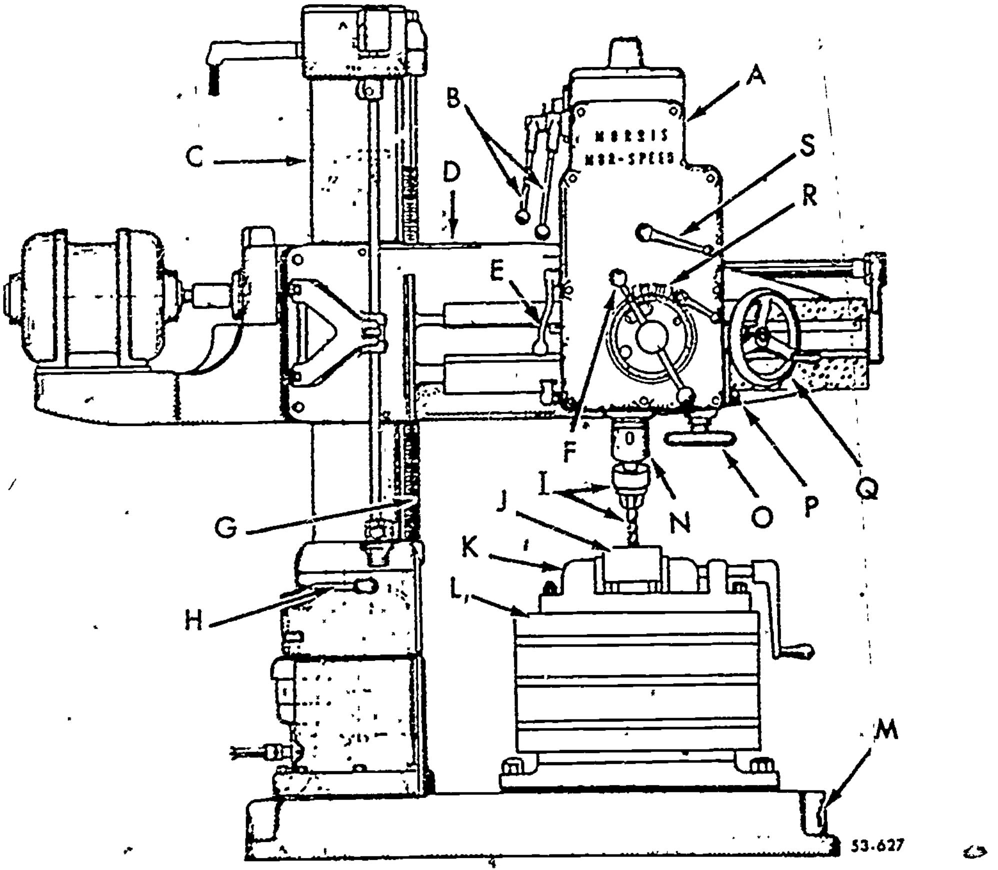
Material in this volume is technically accurate, adequate, and current as of March 1972.

Contents

Drilling Machine Work

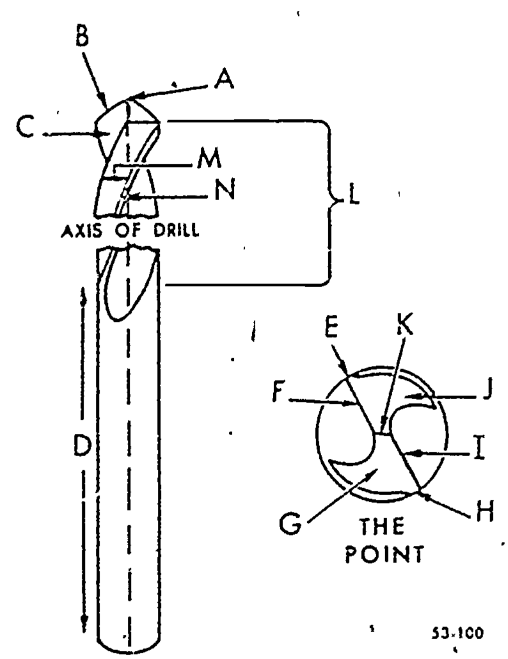
THE DRILL PRESS is used primarily to cut round holes through some type of material. It employs a variety of cutting tools, of which the twist drill is the most common. Drilled holes may be finished by reaming, boring, counterboring, countersinking, spot facing, and tapping on the drill press.

2 The accuracy of work done on a drill press varies with the operation performed. Holes which do not need to be concentric or smooth may be drilled without further finishing. When reaming and boring operations are performed, the holes must be finished to size and within close limits. Also, the holes must be concentric and in the exact location. Such operations may also be performed in various types of drilling and boring jigs and fixtures. In this chapter we will discuss the radial drill press, drill grinding, and drill press maintenance.



A	Head	F	Feed clutch lever	K	Vise	P	Start, stop, and reverse lever
B	Speed change levers	G	Elevating screw and lever	L	Table	Q	Head traverse hand wheel
C	Column	H	Arm clamp lever	M	Base	R	Feed dial
D	Arm	I	Chuck and drill	N	Spindle	S	Feed change lever
E	Head clamp lever	J	Work	O	Feed hand wheel		

Figure 1 Radial drill press



A	Dead center	H	Margin
B	Lip of cutting edge	I	Cutting lip
C	Flute	J	Lip clearance
D	Shank	K	Dead center
E	Margin	L	Body
F	Cutting lip	M	Body clearance
G	Lip clearance	N	Margin

Figure 2 Twist drill

1. Radial Drill Press

1-1. Radial drill presses, such as the one shown in figure 1, have the head mounted on an arm instead of directly on the column of the machine. You can raise, lower, and swing the arm to the right and to the left, and move the head along it. This allows you to position the spindle over the work—a great advantage when you are working with heavy, bulky items. Power feeds and reversible spindles, usually provided, increases the capability of the machine. Radial drill presses are good all-around machines. They are suitable for both light- and heavy-duty work, and are capable of doing highly accurate work. The size of a radial drill press is designated by the length of the arm. This is the distance from the center of the spindle to the edge of the column when the head is located as far out as possible on the arm. For example, a radial drill press designated as having a 3-foot arm will drill to the center of a 6-foot circle, and the length of the arm will actually be greater than 3 feet.

1-2. **Parts of the Radial Drill Press.** In order to understand the operation of a radial drill press and its advantages you should be able to recognize the basic parts of the machine. There are four basic parts, (1) base, (2) column, (3) arm, and (4) head, as shown in figure 1. All of the features of the radial drill press are included in these four parts.

1-3. *Base.* The base of the radial drill press is the large cast section located at the bottom of the machine (fig. 1,M). The base has several functions. It provides a foundation on which the rest of the machine is mounted. It provides a place to mount the work table. The base also has a reservoir for cutting lubricants and coolants.

1-4. *Column.* The column is the upright portion of the radial drill press (fig. 1,C). The column is a large precision ground shaft which supports the arm, column clamp lever, and the arm locking lever.

1-5. *Arm.* The arm of the radial drill press has many functions and advantages. It can be moved around the column 360°. This is a great advantage in locating the drill over large work. The arm gives support to the spindle drive motor and to the head of the machine. It provides the ways along which the head is moved.

1-6. *Head.* The head (fig. 1,A), houses many parts of the radial drill press. It houses the speed change gearbox, feed change gears, and the spindle.

1-7. **Workholding Devices Used on Radial Drill Presses.** Successful drill press work depends to a great extent upon the manner in which work is held. Properly mounted work requires various holding devices, such as the drill press vise, V-blocks, clamps, and straps. The practice of holding work by hand on the drill press is dangerous and can easily result in damaged work, broken drills, and injury to the operator.

1-8. *Vise.* The vise is the most commonly used holding device for drill press work. It usually has slots to receive T-slot bolts and can easily be secured to the machine table. The work is normally supported on parallel bars to prevent drilling holes in the vise. Most vises are constructed with a movable jaw which is operated by a single screw. Vises are available in a variety of sizes and designs.

1-9. *V-blocks.* V-blocks are easily adaptable for the support of cylindrical work. When you use V-blocks to support the work on a drill press table, you normally clamp the blocks directly to the table surface. Then clamp the work in the V-block. When work cannot be held in a vise or clamped to a V-block, you can clamp or strap it to the table surface. A strap can be easily made from a flat piece of steel with a hole drilled near the clamping end. The size and shape of these straps are governed by the nature of the work to be drilled. Clamps, on

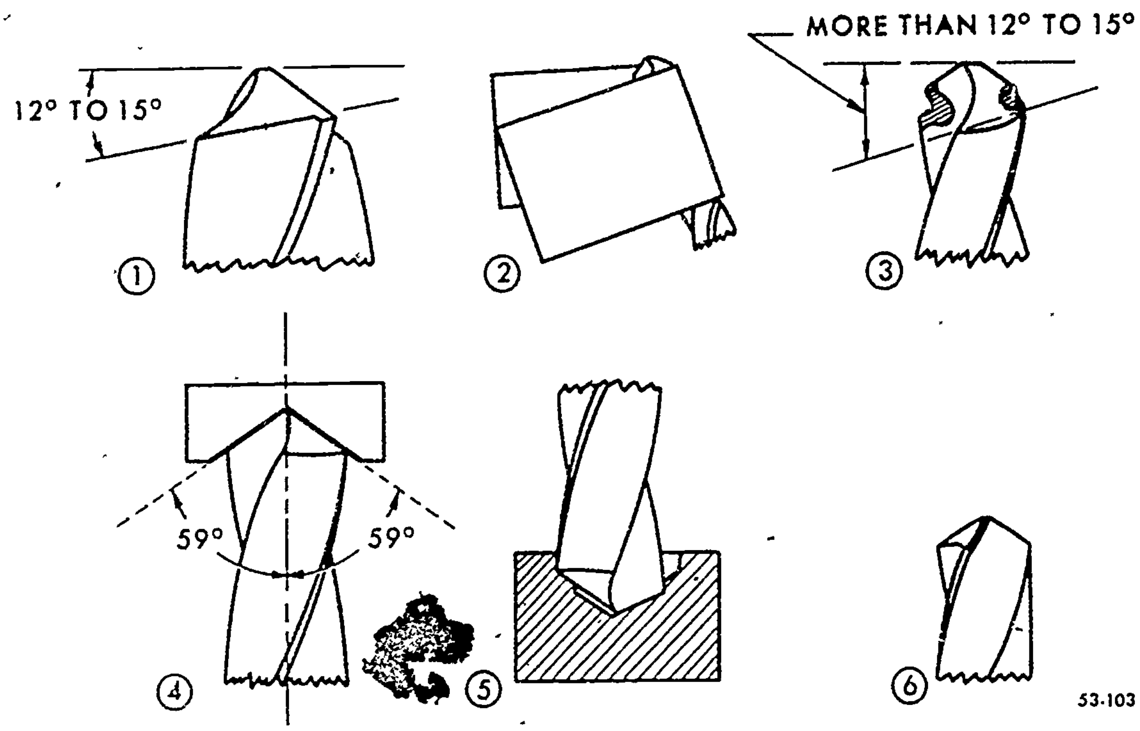


Figure 3. Drill grinding hints.

the other hand, usually have a specific shape, such as the flat, gooseneck, pin, and V-clamp.

2. Drill Grinding

2-1. A correctly sharpened twist drill is a necessity. A sharp twist drill performs more efficiently and you produce a higher quality of work with less pressure on the drill. Success in correctly grinding drills depends upon your skill and experience in using various techniques. Before we go into the actual grinding of a twist drill we will discuss the parts of a drill and their functions.

2-2. **Parts of a Drill.** There are three main parts to a drill: shank, body, and the point, as shown in figure 2. We will discuss these parts of the drill.

2-3. *Shank.* The shank (D) is that part of a drill behind the flutes which is used to hold the drill in a chuck, collet, or spindle. Several types of shanks are available. The most common types are the straight and taper shanks. Most drills larger than 1/2 inch in diameter are made with Morse taper shanks. The tang, which is common to the taper shank drills, is that flattened portion on the shank end of the drill that assists in driving the drill.

2-4 *Body.* The body of a drill (L) is that portion of a drill between the point and the shank. There are several parts that make up the body of a drill, such as flutes, web, and margin. Each of these parts performs some important function, but the drill must be sharp for the parts to complete their tasks.

2-5. *Point.* The point, as shown in figure 2, is the cone-shaped portion on the cutting end of the drill. It is this portion which is ground to form the cutting edges. The dead center (K) or chisel edge is the portion connecting the bottom of the flutes (C) at the extreme end of the drill. It is formed by the intersection of the cone-shaped surfaces of the point. It should always be in the exact center of the axis of the drill. The cutting lips (F) are the actual cutting

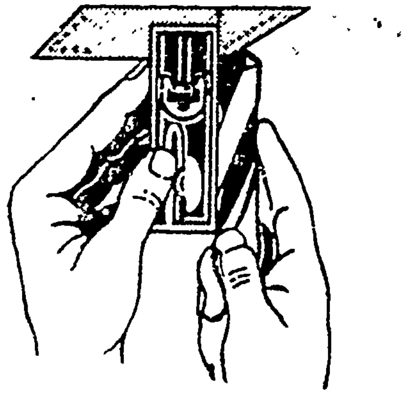
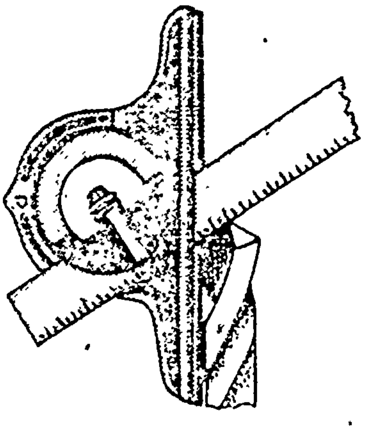


Figure 4. Drill grinding gage.



CHECKING DRILL POINT
WITH PROTRACTOR

53—108

Figure 5 Protractor head and blade

edges of the drill. They extend from the chisel edge to the periphery of the drill.

2-6 **Grinding a Twist Drill.** The greatest difficulty encountered in drilling is caused by incorrectly ground drili lip clearance. Three things should be considered in grinding a drill lip clearance, the length and angle of the lips, and the chisel edge in relation to the axis or centerline of the drill. All three must be correct before a crill can function properly.

2-7. The correct lip clearance for a drill used for the general purpose drilling of still is 12° to 15°. Any change in this clearance angle will change the cutting characteristics of the drill. For example, a drill ground at 12° or slightly less would be best suited for drilling harder carbon and alloy steel. This angle may be checked with sufficient accuracy by using as a gage a strip of paper 3 inches wide and 8 1/2 inches long. Place a mark on the margin of the left end of the strip 1 3/4 inches from the lower side. Fold the strip around the drill and place the upper right-hand corner on the marginal mark, as shown in figure 3(2). Then compare the clearance angle with the diagonal formed by the edge of the strip. If the clearance angle is too small, it will result in possible drill breakage. A clearance angle too large will result in chipping the cutting edges, as shown in figure 3(3). The length of the cutting edges must be the same. Otherwise, a greater torsional strain is placed on one cutting edge, causing the drill to cut a hole larger than its diameter, as shown in figure 3(5). The cutting edge angle (included angle) for carbon and alloy steels is 118°, or 59° on each side of the centerline, as shown in figure 3(4 and 6). This angle is checked for accuracy with a bevel protractor or drill grindling gage, as shown in figures 4 and 5

2-8. When drills are manufactured they are made so that the web of the drill is thicker at the shank end than at the point. This is done to provide strength. As the drill wears and is ground several times it may become necessary to thin the web. Thinning the web is done to achieve maximum cutting efficiency, or ease of penetration and minimum wear. The web should be thinned to approximately its original thickness. This is usually done with a round-faced abrasive wheel, as shown in figure 6.

2-9. Another method of thinning the web of a drill, known as notching, is used to improve the performance of a drill used in hand feed operations, such as in crankshaft drilling. This type of thinning works well when the drill is to be used in a hand drill. Notching is similar to thinning, as shown in the right half of figure 6, except that the sharp-cornered hand abrasive wheel is used instead of a round-faced wheel.

2-10 Since there are so many types of metals in use today, no one drill point will perform satisfactorily in all of these metals. Figure 7 illustrates some of the various types of drill points and the types of metals for which they are best suited.

3. Drill Press Maintenance

3-1 The drill press, like all other machine tools, requires some maintenance When the

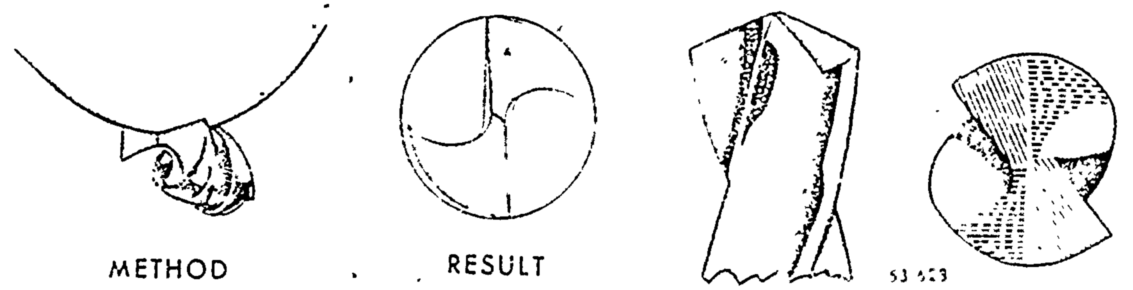


Figure 6 Methods of thinning the web

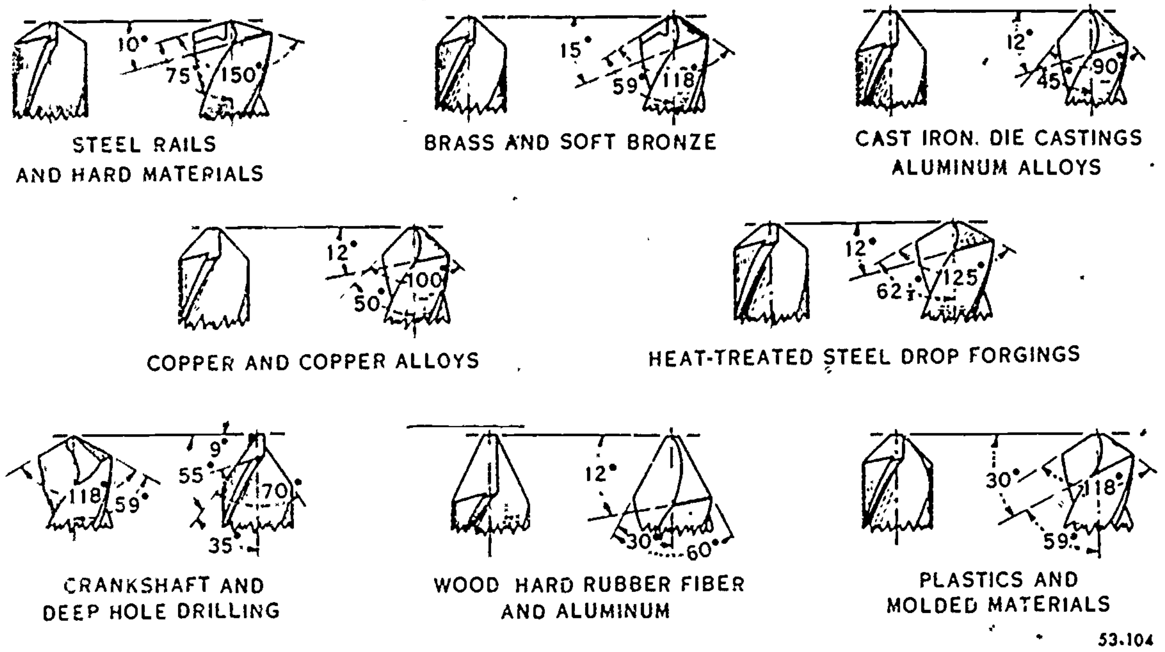


Figure 7 Shapes of drill points.

drill press is installed, it should be leveled and lagged to a solid foundation. This foundation should be solid enough to support the weight of the machine. Great care should be taken in leveling the drill press. If the machine is not properly installed, you cannot hope for the machine to perform with the accuracy for which it was designed.

3-2. The oil cups and bearing surfaces should be oiled daily before the drill press is used. Oiling should be progressive, starting at one point and moving around the machine, making certain that all points are lubricated. All surfaces not covered with paint should be lightly oiled to prevent corrosion. The drill press gearbox should receive periodic oiling and oil changes should be made according to the manufacturer's specifications. Be sure to check manufacturer's specification for the type, viscosity, and amount of oil to use. Remember, that in some cases, too much oil can cause severe damage to machine seals.

CHAPTER 2

Lathe Work

THE LATHE IS the most useful machine in the machine shop. More operations can be done on it than on any other machine tool. In this chapter we will not discuss the more common operations; we will limit the discussion to (1) special lathe operations, (2) attachments, (3) special threading operations, and (4) lathe maintenance.

4. Special Lathe Operations

4-1. You will be called upon from time to time to perform various special lathe operations. A special lathe operation is an operation that is performed, normally, on a lathe with special tools or attachments. Such operations as radii and form turning, taper turning, toolpost grinding, parting, spring winding, filing, polishing, and knurling fall into this category. We will discuss these operations.

4-2. **Radii and Form Turning.** 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.

4-3. *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 movement of the carriage and cross-slide as you observe the cutting action.

4-4. *Forming tool.* You may grind the forming 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 forming tool is in machining several duplicate pieces, since the machining of one or two pieces would not warrant the time spent in grinding the tool. You can use forming tools to machine either concave or convex radii. A *concave radius* is hollow in shape and a *convex radius* is spherical or ball shaped. Figure 8 shows some typical shapes you can produce with forming tools.

4-5. *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 filing and polishing. Figure 9 shows a template and pointer being used to produce a contoured surface.

4-6. *Radius rod.* When you do 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 10. 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.

4-7. *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 11.A, or a con-

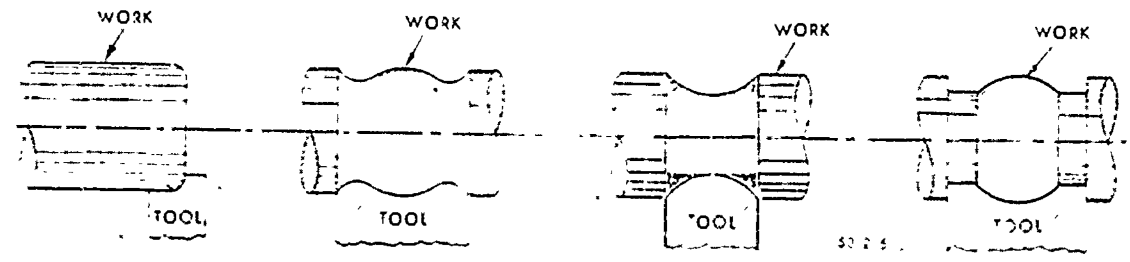


Figure 8. Forming tools.

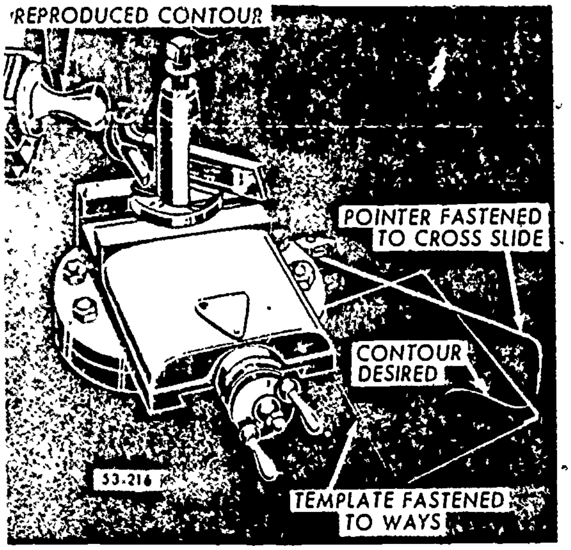


Figure 9. Template and pointer.

vex radius by positioning the tool behind the pivot point, as shown in figure 11,B.

4-8. **Taper Turning.** 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 checking tapers; and taper turning by means of the compound rest, tailstock offset, and taper attachment.

4-9. 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: (1) self-holding tapers and (2) self-releasing tapers.

4-10. *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° and the shank of the tool is so firmly seated in the socket that there is considerable frictional resistance to any force tending to turn it in the socket. There are several different types of self-holding tapers.

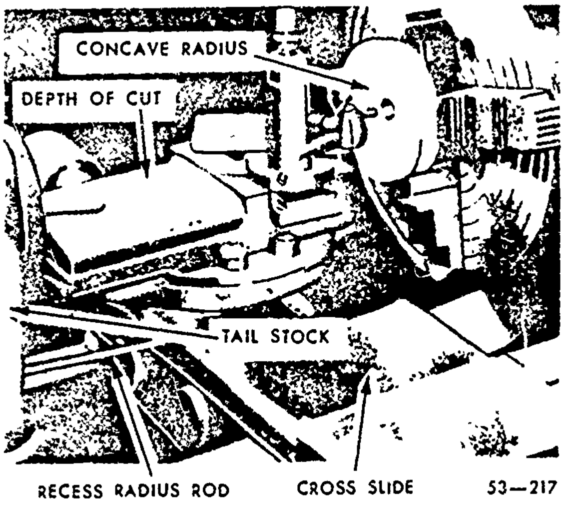


Figure 10. Use of a radius rod

4-11. Morse taper. There are eight different sizes of Morse tapers. The taper for each is slightly different, but it is approximately 5/8 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.

4-12. Brown and Sharpe taper. There are 18 different sizes of Brown and Sharpe tapers. The taper is approximately 1/2 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.

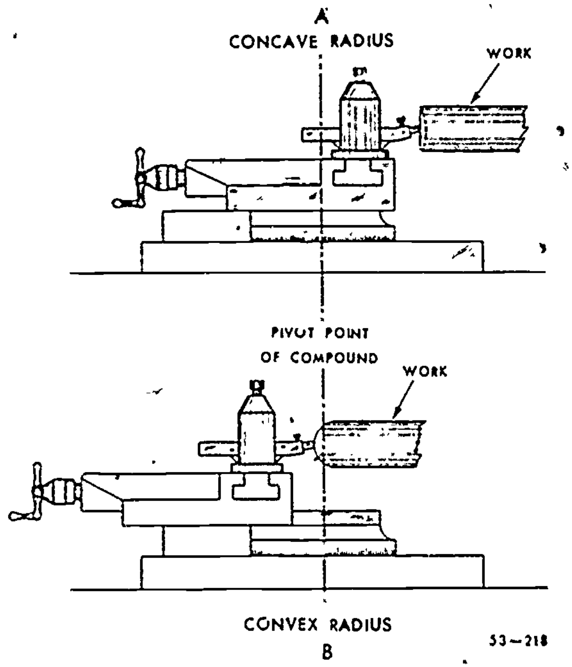


Figure 11. Matching a radius with the compound rest.

4-13. The 3/4-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 Morse tapers stop in the American Standard Self-Holding Taper Series.

4-14. 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 three sizes of the Brown and Sharpe, all eight sizes of the Morse, and all 11 sizes of the 3/4-inch-per-foot taper.

4-15. 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 number 7 Jarno taper has a 7/8 inch large diameter, a 7/10-inch smaller diameter, and a 7/2 inch, or 3 1/2 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

4-16. Taper pins and reamers. Taper pins have a tape of 1/4 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 pinholes.

4-17. Other tapers. There are a number of other tapes, but they are used to such a limited extent that full tables are not given in 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: (1) standard and (2) 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 keyseat the whole length of the taper but no tang.

4-18. *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 3 1/2 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 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 3 1/2 inches per foot. Note, in figure 12, 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 Machinery's Handbook.

4-19. 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 13 shows how a protractor head and blade are used to check a steep taper.

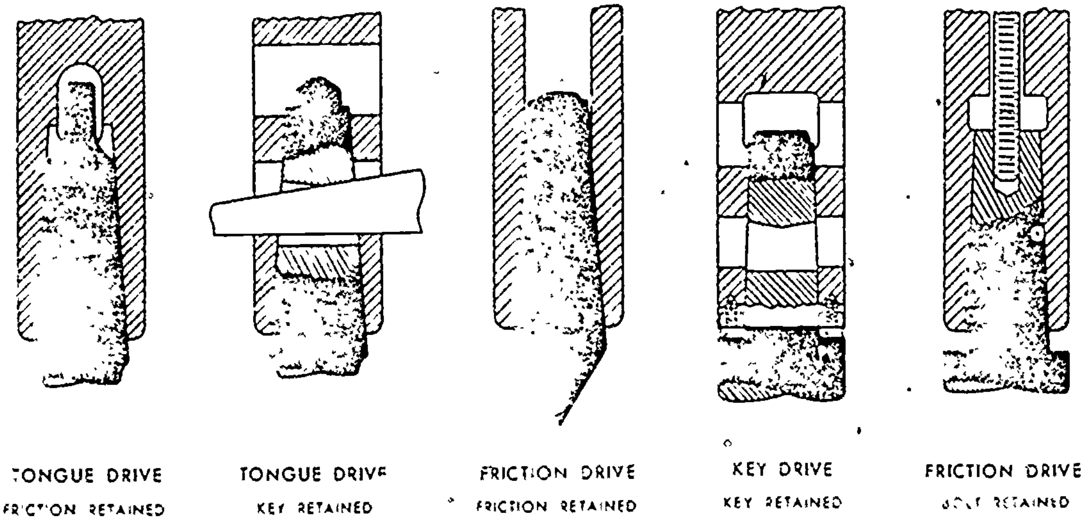


Figure 12. Types of standard taper drives.

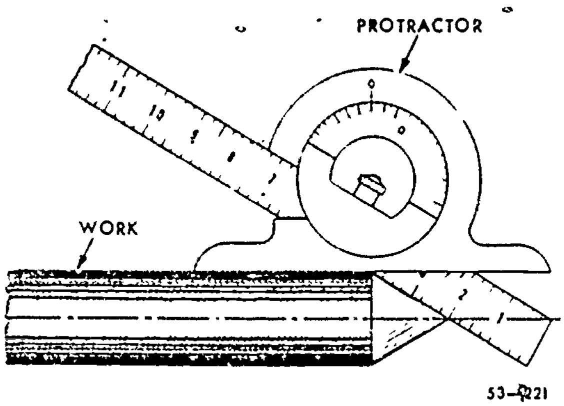


Figure 13 Checking taper with a protractor head

b. Tapered ring gages are used to check tapers, as shown in figure 14. Insert the tapered part in the gage and wiggle it. 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 15. Careful layout of the lines and alinement of the micrometer will help to insure accurate results when you use this method.

4-20 Taper Turning with the Compound Rest. Both external and internal tapers can be turned with the compound rest. You 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 centerline of the work, figure 16,A, or from a line perpendicular to the centerline of the work, figure 16,B. For example, the 40° angle in figure 16,B, is measured from a line perpendicular to the centerline of the work. In order to machine this angle, you must first position the compound rest perpendicular to the centerline, 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.

4-21. 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:

$$\text{TPI} = \frac{\text{LD} - \text{SD}}{\text{L of T}}$$

$$\text{Tangent of the angle (tan } \angle \text{)} = \frac{\text{TPI}}{2}$$

4-22. Let us apply the formulas above, first, when the TPI is given, and, second, when only the dimensions of a taper are given:

a. Suppose that a TPI of 0.800 is specified:

$$\tan \angle = \frac{\text{TPI}}{2} = \frac{0.800}{2} = 0.400$$

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 a table of trigonometric functions (often called *trig tables*). This table, which is several pages in length, may be found in machinists' publications, such as *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 numerals. Thus, 0.39997 is the tangent for 21° 48 You would set the compound rest for 21 3/4° 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.)

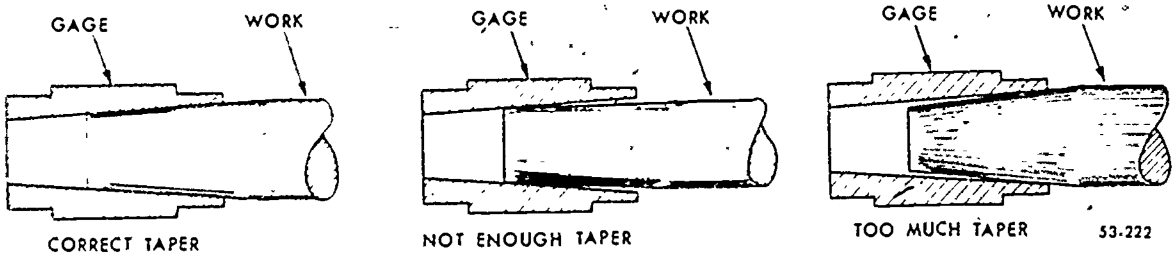


Figure 14. Checking taper with a tapered ring gage

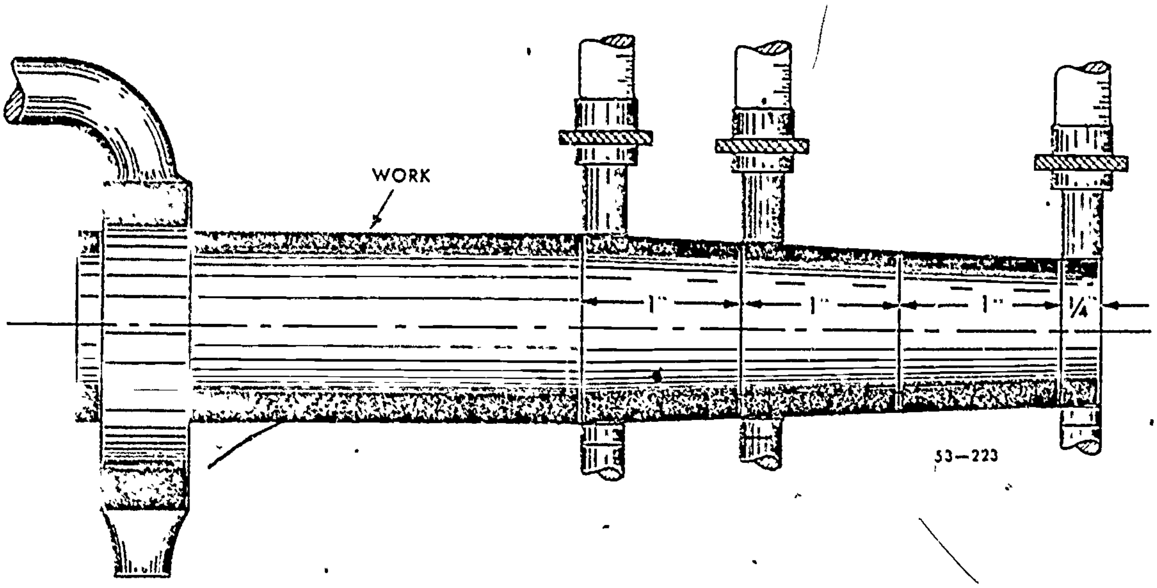


Figure 15— Measuring taper per inch with a micrometer.

b. If the dimensions of a taper are given, as in figure 17, 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′.

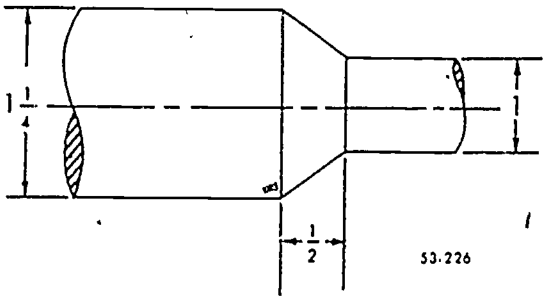


Figure 17. Dimensions of a taper

4-23. **Taper Turning by Offsetting the Tailstock.** You will use the *offset tailstock method* frequently to turn long slow tapers because of the limitations of the compound rest method. Also, you will find that every lathe is not equipped with a taper attachment. When you offset the dead center by moving the tailstock out of alinement, the centerline of the work and the line of travel of the turning tool are no longer parallel. A taper will be turned on the work, as shown in figure 18. 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.

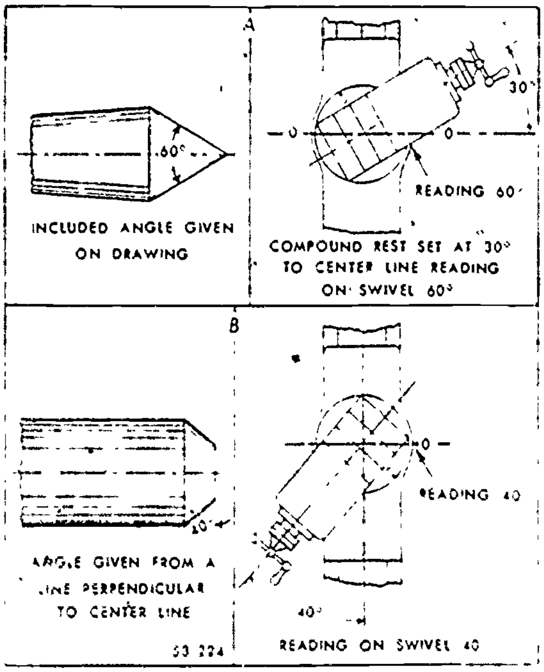


Figure 16. Setting the compound rest

4-24. *Calculating offset.* You can calculate the offset when you have either (1) the taper per inch and the length, or (2) the included angle and the length.

a. 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.

$$\text{Tailstock offset (TO)} = \frac{\text{taper per inch}}{2} \times \text{length of work}$$

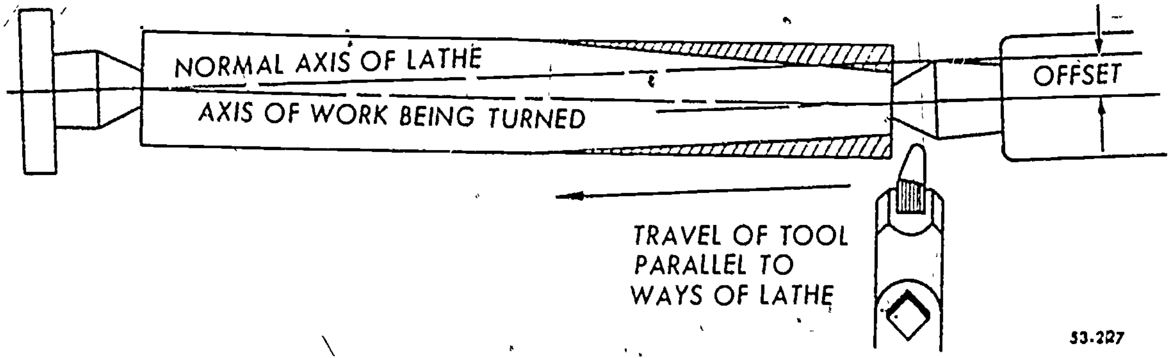


Figure 18. Effects of tailstock offset.

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 19, the tailstock offset would be calculated as follows:

$$TO = \frac{TPI}{2} \times LW$$

$$TO = \frac{0.050}{2} \times 10 = 0.025 \times 10$$

$$TO = 0.250 = 1/4 \text{ inch}$$

b 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.

$$\text{Angle} = \frac{\text{included}}{2}$$

(2) Now, determine the offset by using the following formula:

Tailstock offset = tan ∠ × length of work
TO = tan ∠ × LW

Example To cut a taper with an included angle of 7° on a piece of work 12 inches long, as shown in figure 20, the calculations would be as follows:

$$(1)\ \text{Angle} = \frac{7^\circ}{2}$$

$$\text{Angle} = 3\ 1/2$$

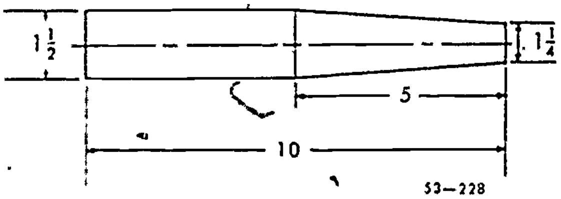


Figure 19 Calculating tailstock offset when the dimensions are given

$$(2)\ TO = \tan 3\ 1/2^\circ \times 12$$

$$TO = 0.06116 \times 12$$

$$TO = 0.7339 = \frac{47}{64} \text{ inch}$$

NOTE: You should note that the length of taper is 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 of the overall length of the work and the included angle of the taper.

4-25. When you determine the proper offset for the tailstock, remember that any change in the length of work between centers will necessitate 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.

4-26. *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 21. Or, you may measure the lateral distance between centers with a machinist's rule, as shown in figure 22. On work requiring a great deal of accuracy, you may measure the amount of offset (1) with an inside caliper, (2) with the dial test indicator, (3) by using the crossfeed calibrated collar, or (4) by using the cut and try method.

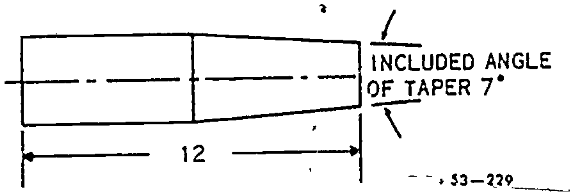


Figure 20 Calculating tailstock offset when the included angle is given.

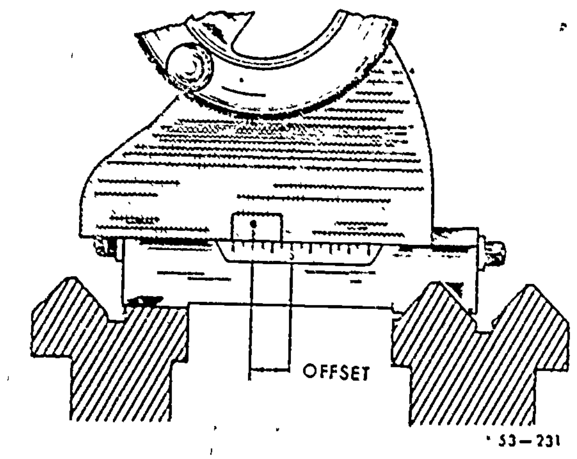


Figure 21 Tailstock cricket marks

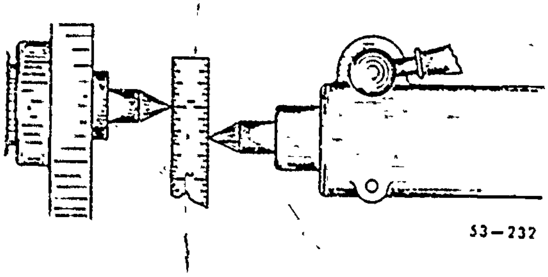


Figure 22 Measuring offset with a steel rule

4-27. When you check the offset with an inside caliper, bring the toolpost to bear lightly against the side of the tailstock spindle, figure 23,A, and then back it away from the tailstock spindle a distance equal to the predetermined caliper setting, figure 23,B. Then offset the tailstock until the tailstock spindle again contacts the toolpost, as shown in figure 23, C

4-28. When you use the dial test indicator method, mount the instrument on the toolpost, position the indicator plunger to bear lightly against the side of the tailstock spindle, and read the amount of offset on the indicator as you move the tailstock laterally.

4-29. When you use the crossfeed (cross-slide) calibrated collar method, position the side of the toolpost near the tailstock spindle. Then move the toolpost *away* from the tailstock spindle with the cross-slide This will eliminate the backlash. Now set the cross-slide graduated collar at zero. Move the toolpost toward the tailstock spindle with the compound rest until you feel a slight drag on a strip of paper between the tailstock spindle and the toolpost when you pull on the paper. 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 toolpost when you pull on the paper. At this point, you have obtained the desired offset.

4-30 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 tailstock, and take more trial cuts until you obtain the desired taper per inch.

4-31. While the offset 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 scored 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 the centers; therefore, only external tapers may be turned. The degree of taper that can be turned is governed by the offset of the tailstock and the length of the work. The range of offset of the tailstock varies from approximately 1/2 inch on small lathes to 1 1/2 inches on larger lathes.

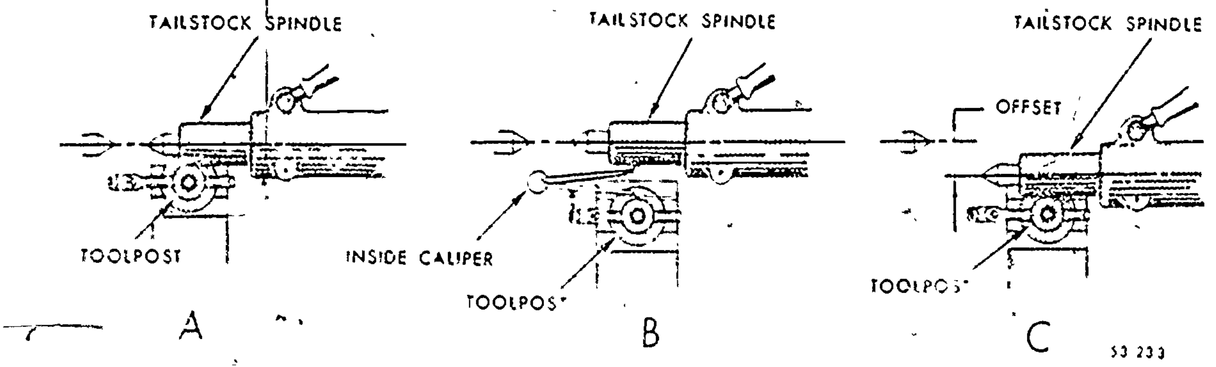
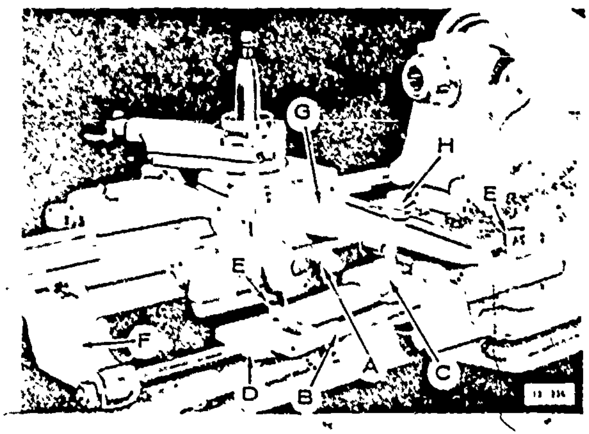


Figure 23 Measuring offset with inside calipers



A	Carriage bracket	E	Clamp screw
B	Guide bar	F	Bed bracket
C	Guide block	G	Draw bar
D	Guide block base	H	Clamp

Figure 24. Taper attachment.

4-32. 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. It is more desirable to machine a taper with the taper attachment than with the offset tailstock method because the lathe centers are in alinement. The wear on the centers and center holes is not as great, and duplicate tapers may be machined on pieces of different lengths without changing the taper setting.

4-33. *Description*. The essential parts of the taper attachment, shown in figure 24, and the purpose each serves are as follows:

a. The carriage bracket (A) is attached to the saddle of the lathe and supports the attachment.

b. 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.

c. The shoe or guide block (C) travels on the guide bar and is attached to the cross-slide.

d. The guide block base (D) supports the guide bar.

e. The clamping screws (E) clamp the guide bar to its base.

f. The bed bracket (F) clamps the guide bar to the lathe bed.

g. The guide block gibs (not shown) take up the wear between the guide block and the guide bar.

h. 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.

4-34. *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 desired degree of taper and clamp it in position. As the carriage travels 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.

4-35. 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 the 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:

a. Move the carriage and tool slightly past the start of the cut, then return the carriage and tool to the start of the cut.

b. Move the cross-slide and the tool in the same direction in which the taper runs, i.e., 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.

4-36. Taper attachments equipped with a drawbar do not have backlash. The drawbar connects the cross-slide directly to the guide block and takes the push and pull off the crossfeed screw and the nut. You set the compound rest at a right angle to the ways and take the depth of cut with the compound rest, since the drawbar locks the cross-slide directly to the guide block, 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 25. To set the guide bar for the proper taper when the taper per inch is given, multiply by 12 to get the taper per foot and set the graduated scale to the nearest fractional setting.

4-37. 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: Find the taper per inch of a number 2 Morse taper in a machinist publication, such as *Machinery's*

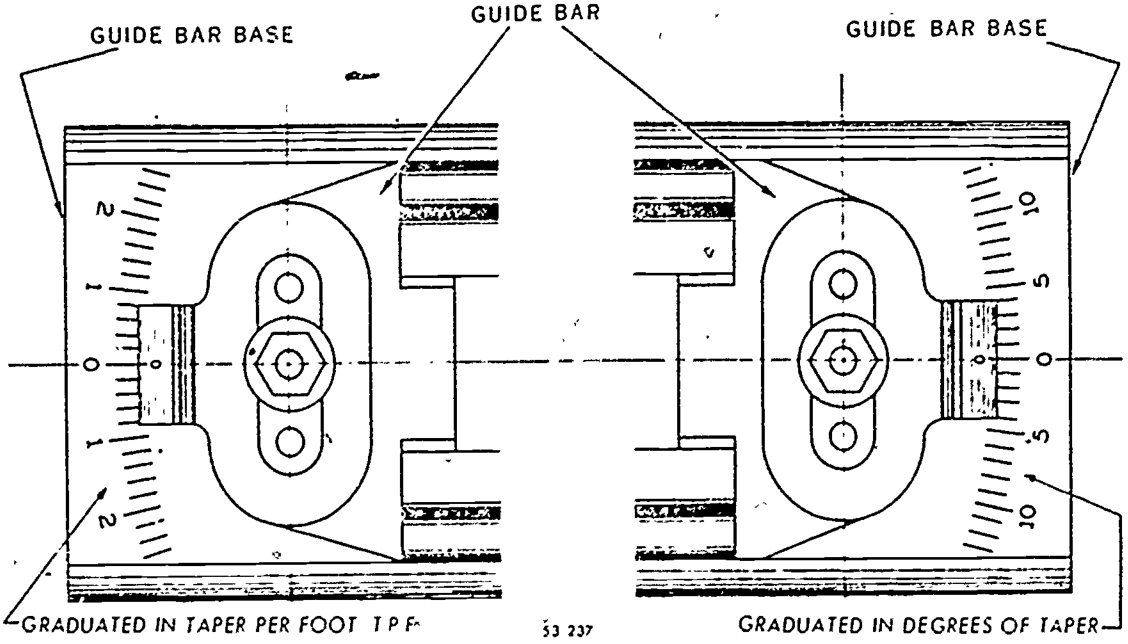


Figure 25. Guide bar graduations

Handbook. (TPI = 0.04995.) Convert the taper per inch into taper per foot.

$$TPF = TPI \times 12$$
$$TPF = 0.04995 \times 12$$
$$TPF = 0.5994$$

Convert the decimal to a fraction, using a decimal equivalent table:

$$0.5994 = \frac{19}{32} \quad (0.5937)$$

Set the taper attachment to obtain a 19/32-per-foot taper. NOTE: Taper attachments are often graduated in 1/8-inch-per-foot increments; therefore, you may have to estimate settings that fall between the 1/8-inch graduations. You can obtain sufficient accuracy for the first trial cuts by setting the taper attachment for a taper of slightly less than 5/8 inch per foot.

4-38. If you had decided to use the degree graduations, you would determine the degree setting in the following manner:

Find the tangent of the angle $(\text{Tan}\angle = \frac{TPI}{2})$

$$\text{Tan}\angle = \frac{0.04995}{2}$$

$$\text{Tan}\angle = 0.02497$$

Look in a table of trigonometric functions and find the angle represented by the tangent you have calculated.

$$\text{Tan}\ 0.02497 = 1^\circ\ 26'$$
or
approximately $1\ 1/2^\circ$

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°

4-39. 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 and 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 26 shows a typical

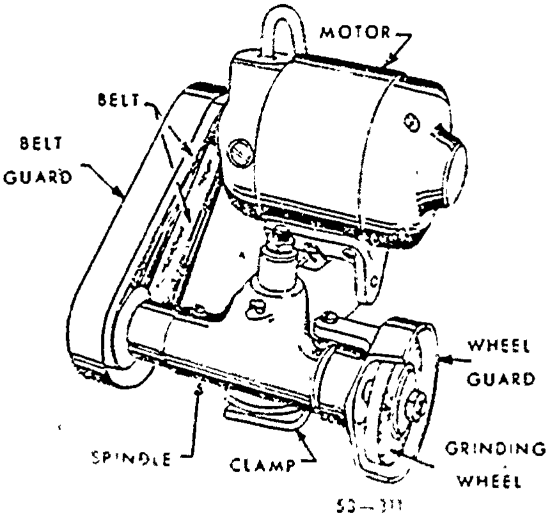


Figure 26. Tool post grinder

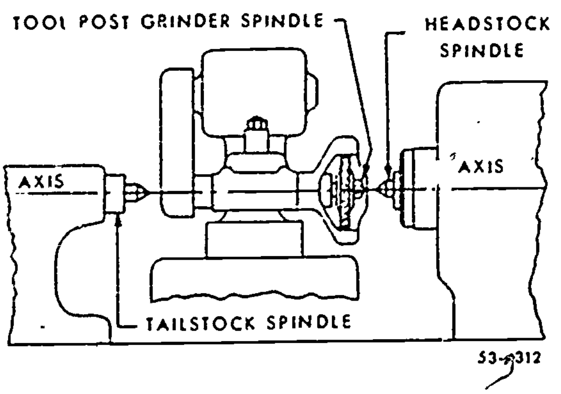


Figure 27. Mounting the grinder at center height

toolpost grinder. The grinder must be set on center, as shown in figure 27. The centering holes located on the spindle shaft are used for this purpose. The grinding wheel takes the place of a lathe tool bit. It can perform most of the operations that a tool bit is capable of performing. Cylindrical surfaces, tapered surfaces, and internal surfaces can be ground with the toolpost grinder. Very small grinding wheels are mounted on tapered shafts known as *quills* in order to grind internal surfaces.

4-40. 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. (CAUTION. Grinding wheels are safe for operation at a speed just below the highest recommended speed. A higher than recommended speed will result in the increase in centrifugal force which may cause the wheel to break This is one reason that they offer protection in case you select a higher speed by mistake. Avoid such a mistake by careful checking.)

Wheel guards are furnished with the toolpost grinder 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.

4-41. 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. Set the point of the diamond at center height and at a 10° to 15° angle in the direction of the grinding wheel rotation. 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 into contact with the diamond by carefully feeding the cross-slide in by hand. Move the wheel clear of the diamond and make a cut by means of the cross-slide. The maximum depth of cut is 0.002 inch. Move the wheel slowly by hand back and forth over the point of the diamond. Move the carriage if the face of the wheel is parallel to the ways of the lathe, or 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.

4-42. Rotate the work at a fairly low speed during the grinding operation. The recommended surface speed is 60 to 100 feet per minute (fpm). The depth of cut depends upon the hardness of the work, the type of grinding wheel, and the desired finish. Avoid taking grinding cuts deeper than 0.002 inch until you gain experience. Use a fairly low rate of feed. You will soon be able to judge whether the feed should be increased or decreased. NOTE: Never stop the rotation of the work or the grinding wheel while they are in contact with each other.

4-43. 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 the tailstock spindle must be removed in order to perform other operations, a mark placed on the headstock spindle sleeve and center will enable you to install them in the same position they were in when the center was ground and thus insure the greatest degree of accuracy for future operations involving turning work between centers.

4-44. **Parting.** *Parting,* also called *cutting off,* is the process of cutting a groove around revolving work in order to sever part of it from the piece held in the lathe. Parting is used (1) to cut off parts that have already been machined in the lathe, (2) to cut off tubing and bar stock to required lengths, (3) to machine necks and grooves in material, and (4) to cut off material that would be impractical to saw in power hacksaws.

4-45. *Parting tools.* Three types of tools are commonly used for parting: the (1) solid, forged type; (2) high-speed steel cutter blades which are held in patented toolholders, and (3)

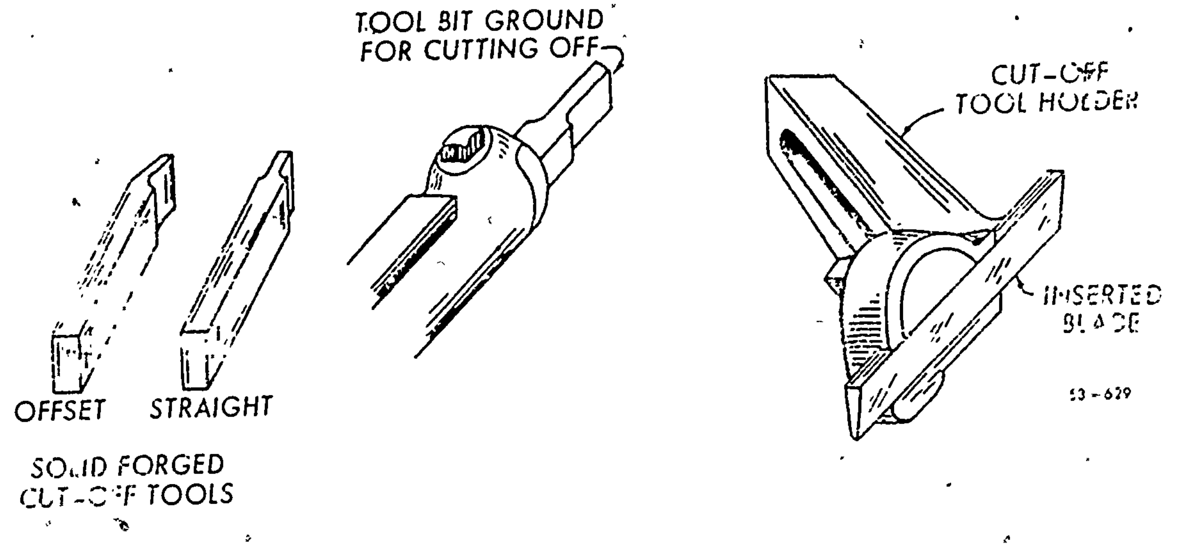


Figure 28 Types of parting tools

tools ground to shape from tool bits. Examples for these three types are shown in figure 28.

4-46 The *solid, forged type parting tool* is forged from either carbon steel or high speed steel and is ground to meet the requirements of the job at hand. It may be either straight or offset. This tool is being rapidly replaced by the *inserted cutter blade type parting tool*

4-47. The inserted blade type parting tools are made in a variety of sizes and are held in special toolholders. The toolholders are available in straight and offset types, as shown in figure 29. You can sharpen the blades repeatedly and replace them when they are too short to hold safely in the toolholder Some blades are manufactured with the necessary flank and side relief angles ground in them, only the end relief is ground when you sharpen them. *Flank relief* is the clearance ground on both sides of the parting tool behind the cutting edge. Flank relief helps to prevent the sides of the parting tool from contacting the sides of the groove that is formed during parting.

4-48. Parting tools ground from tool bits, as shown in figure 30, are used mostly on small work. The cutting edge may be ground straight or offset, as shown in figure 30.A and B. The top of the tool is ground down, as shown in figure 30.C, to eliminate the excessive back rake created by the toolholder.

4-49. *Parting tool geometry.* The general shape of parting tools is the same for all three types They are ground so that the cutting edge is the widest part of the tool. Grind both sides of the tool with 1° to 2° of back clearance or flank relief, figure 31.A, and 2° side clearance, figure 31.B. Back rake is usually eliminated. The end relief should be approximately 15° for parting soft metal, such as brass (fig. 31.C). For steel and harder metals, a back rake of about 5° gives free-cutting action and helps to curl the chip, and the end relief should be approximately 10° (fig. 31.D). In order for the tool to have maximum strength, the length of the cutting portion of the blade should be slightly greater than half the diameter of the work to be parted.

4-50. *Parting tool setup* Work to be parted should be held in a chuck, preferably the four-jaw type, with the point at which the parting is to occur as close as possible to the chuck jaws Always make the parting cut at a right angle to the centerline of the work and feed the tool into the revolving work with the cross-slide until the tool completely severs the work. A power feed of approximately 0.002 inch per revolution

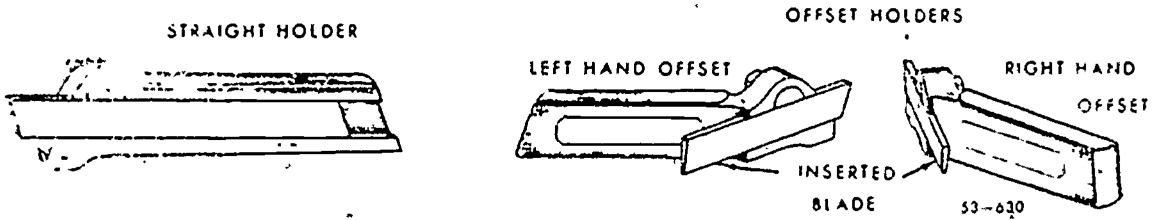


Figure 29 Inserted blade parting tools

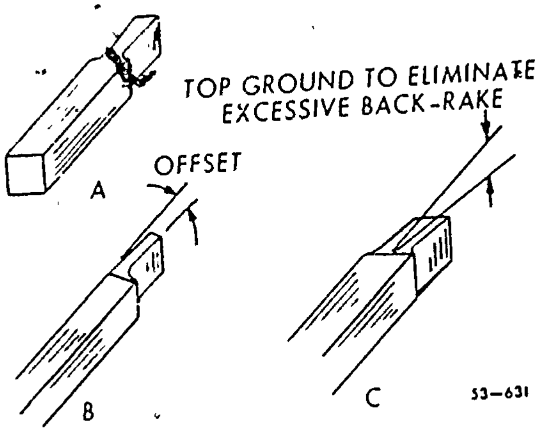


Figure 30 Parting tools ground from tool bits

may be used. However, you have better control of the tool if you feed it by hand.

4-51. Cutting speeds for parting are comparable to turning speeds. You should use a feed that will keep a thin chip coming continuously 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.

4-52. On large diameter jobs where there is danger of the tool binding in the groove, you should use the step-parting method. In step parting you feed the tool into the work a short distance. Then you withdraw the tool from the groove, move the carriage slightly to one side, and feed the tool in again. This leaves only one side of the tool in contact with the groove and prevents binding. Take alternate cuts until the work is cut off.

4-53. 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. Move the carriage until the desired length is obtained. You may also aline the parting tool to a layout line scribed on the work. NOTE: Always lock the carriage in position to prevent it from moving while you are taking the parting cut. CAUTION: Never attempt to catch the piece that has been parted off.

4-54. **Spring Winding.** When a small number of springs of the helical or coil form are needed in connection with repair work it is common practice to wind them on the lathe. When springs are manufactured in large quantities, special spring winding machines are used. Single coil springs are made on a lathe using the simple setup shown in figure 32.

4-55. Coil springs may be of the tension or the compression types (fig. 33). A tension spring, figure 33,A, is one in which the coils lie one against the other as in a screen door spring. When tension or pull is exerted on the spring, the coils are spread apart and the tendency of the spring is to pull back to its original form. A compression spring, figure 33,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 an example of 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 form.

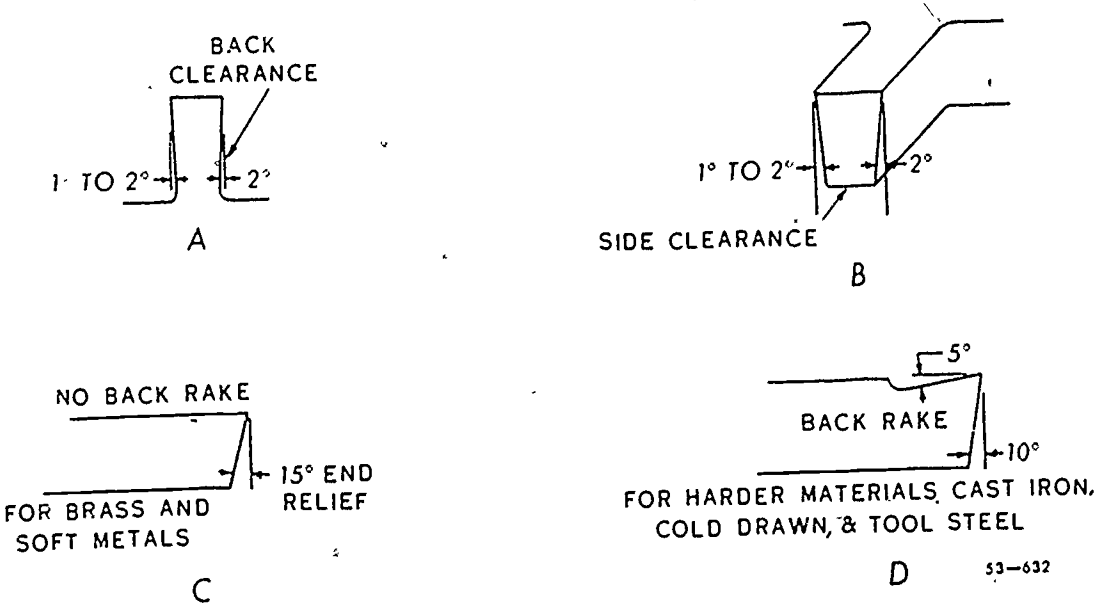


Figure 31 Parting tool rake and clearance angles

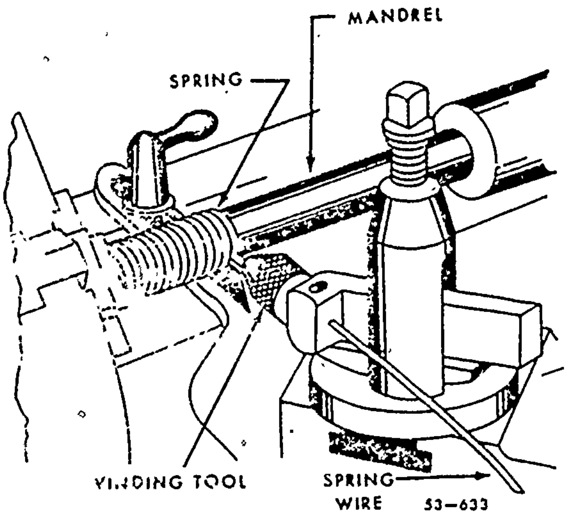


Figure 32. Winding springs in a lathe.

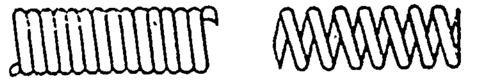


Figure 33 Coil springs.

4-56. *Spring wire material.* Different grades and types of spring wire material are used for winding springs because of the different requirements, such as resistance to fatigue, corrosion, temperatures, etc. Steel containing about 1 percent of carbon and comparatively free from phosphorus and sulphur, known as spring steel, is ordinarily used for springs. For small springs, music wire is used to a great extent and is the best material obtainable for this purpose. Music wire ranges from .004 inch to .046 inch in diameter. The carbon content of music wire varies from 0.70 percent to 1.00 percent, and manganese from 0.25 percent to 0.40 percent. Music wire is more expensive than ordinary spring wire, but can be subjected to higher stresses. Phosphorus bronze wire may be used if steel wire would corrode rapidly. It usually contains about 5 percent tin, a trace of phosphorus to prevent brittleness, and the remainder copper. Brass wire is inferior to phosphorus bronze wire but may be used when the cost of material is an important factor. Monel and inconel have excellent corrosion resistant properties and will withstand abnormally high temperatures.

4-57. [illegible] *of springs.* You can wind [illegible] on mandrels of the proper size [illegible] wire of the required material and [illegible]. Pass the wire between wood or brass blocks or through a spring wire holder. Fasten the end of the wire between a chuck jaw and the mandrel. The other end of the mandrel is supported by the tailstock center. Mandrels having a diameter greater than 3/8 inch are held in a lathe chuck, while mandrels less than 3/8 inch in diameter may be held in a drill chuck. 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. When the spring is wound to the desired length, relieve the tension, clip the wire off, and remove the spring from the mandrel.

4-58. Springs are wound on the lathe by causing the lathe carriage and the wire holder to move a distance equal to the lead of the required spring for each revolution of the spindle. The speed at which the lathe lead screw turns in relation to the speed of the lathe spindle determines the distance the carriage and wire holder will advance when the half nut is closed over the lathe lead screw. The motion of the spindle is transmitted to the lathe lead screw through a train of gears. The carriage is caused to move by the closing of the split nut on the revolving lead screw. The lathe is geared to the pitch or lead of the spring to be wound.

4-59. Nearly all modern lathes are equipped with a quick change gearbox. Changes to accommodate a wide range of leads can be made quickly by simply sliding the gears in the quick change gearbox by means of a lever On older type lathes, to increase the range of leads, the change gears are changed by hand to establish various speed ratios between the spindle and the lead screw. Each time a different lead is required, the change gears must be changed.

4-60. The number of coils per inch or lead produced depends upon the number of turns the work makes while the lead screw moves the carriage and wire holder 1 inch. As an example, a lead screw having four threads per inch will make four revolutions to move the carriage and wire 1 inch along the mandrel. If the ratio of the change gears were 1 to 1 so that the number of revolutions of the spindle and the lead screw were the same, then four coils per inch would be wound. If the spindle revolved twice as fast as the lead screw, then the spindle would make eight turns while the carriage moved an inch and eight coils per inch would be wound. This difference in speed ratio of lead screw and spindle may be established by using a combination of change gearing.

4-61. *Filing.* When you file work in the lathe, hold the point 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

95

101

before the stroke is completed. Exert less pressure than for ordinary bench filing, as 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 so that the metal that lodges between the teeth of the file will not scratch the work or impair the cutting action of the file. CAUTION. Keep your hands and arms clear of the chuck or other work-driving device to avoid injury, and never use a file without a file handle.

4-62. You will usually use a *mill file* when you file work in a lathe. Use a *bastard cut mill file* for rough filing and 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.

4-63. The *spindle speed* for filing ferrous metals is four to five times faster than the *rough turning speed*. For nonferrous metals the speed is two to three 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, causing it to dull rapidly and to glaze the work. The amount of stock left for filing is 0.002 inch to 0.005 inch.

4-64. **Polishing.** Polishing the work with abrasive cloth or sandpaper 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 or sandpaper to produce a very fine finish. Use abrasive cloth for ferrous metals and sandpaper for nonferrous metals.

4-65. The spindle speeds for polishing are very high, the recommended surface feet per minute SFM is 5000. A great deal of heat is generated because of the high speed and the friction of the abrasive cloth against the work. The work will expand rapidly because of the heat. Therefore, you should lubricate and adjust the tailstock center frequently when you are polishing between centers. To determine the spindle speed, use the following formula.

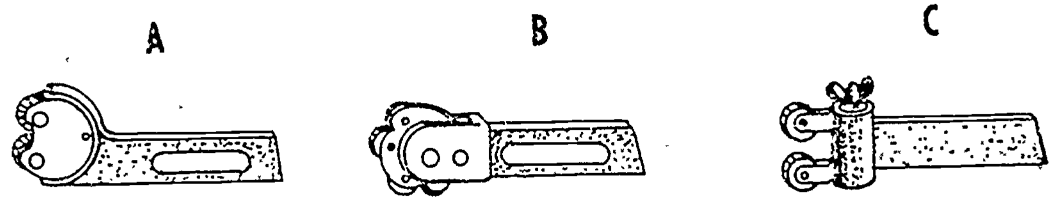
$$\text{rpm} = \frac{4 \times 5000}{\text{work diameter}}$$

4-66. Keep the ends of the strip 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 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.

4-67. **Knurling.** *Knurling* is the process of rolling or squeezing impressions into the work by means of hardened steel rollers that have teeth milled in their faces. Knurling provides a gripping surface on the work; it is used also for decoration. Knurling increases the diameter of the work slightly.

4-68. *Knurling tools.* The three common types of knurling tools are the *knuckle joint*, shown in figure 34,A, the revolving head, shown in figure 34,B, and the *straddle*, shown in figure 34,C. The revolving head type has three sets of rolls, while the knuckle and straddle types each have one set of rolls. NOTE: The straddle-type 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.

4-69. There are two patterns of knurls, *diamond* and *straight line*, and three pitches, *fine*, *medium*, and *coarse*, in each pattern. The diamond is the most common pattern, and the medium is the most common pitch. The coarse



KNUCKLE JOINT TYPE (SINGLE PAIR OF ROLLS)

REVOLVING HEAD TYPE (FITTED WITH THREE PAIRS OF ROLLS)

STRADDLE TYPE

Figure 34 Types of knurling tools

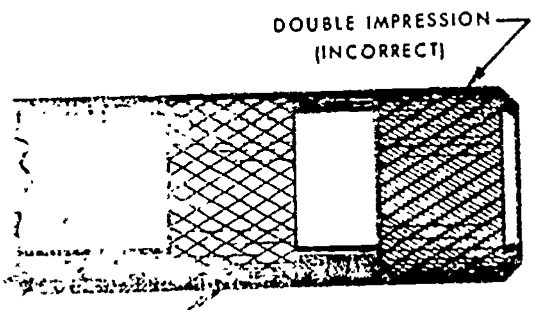
pitch is used for large diameter work and the fine pitch for small diameter work.

4-70. *Knurling setup.* The knurling tool is positioned with the faces of the rolls parallel to the surface of the work and with the upper and lower rolls equally spaced above and below the work axis. The spindle speed should be approximately half the rough turning speed, but do not exceed the highest speed permitted for cutting. the feed reverse mechanism on the lathe. The feed should be between 0.015 inch and 0.02[illegible] inch. The 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.

4-71. *The knurling operation.* You perform the knurling operation in two ways: (1) for ordinary knurling and (2) for knurling between layout lines.

4-72. Ordinary knurling is performed in the following manner. Set up the work and lay out the length of the portion to be knurled. Set up the knurling tool. Set the lathe for the correct spindle speed and the correct feed. Check the knurling tool to insure that the rolls revolve freely and that the revolving head is free to move also. Oil the rolls and the pins on which the rolls revolve by flooding the knurling tool head with oil. Apply oil to the work and the knurling tool generously during the knurling operation. Apply oil to the surface of the work opposite the knurling tool with an oilcan (keep the spout well away from the knurling tool rolls) or with a brush. CAUTION: Keep rags, brushes, and your fingers away from the knurling tool during the knurling operation.

4-73. Position the carriage so that a third to a half of the face of the rolls extends beyond the end of the work. This eliminates part of the pressure required to start the knurl impression. Force the knurling rolls into contact with the work. Engage the spindle clutch. Check the knurl to see if the rolls have tracked properly, as shown in figure 35, by disengaging the clutch after the work has revolved three or four times and by backing the knurling tool away from the work by means of the cross-slide. If the rolls have "double tracked," as shown in figure 35, move the carriage to a new location and repeat the operation. If the knurl is correctly formed, engage the spindle clutch and the carriage feed. Move the knurling rolls into contact with the knurled impressions. (The rolls will aline themselves with the impressions.) Allow the tool to knurl to within 1/32 inch of the layout line.



Knurled impressions

4-74. Disengage the carriage feed and, with the work revolving, feed the carriage by hand to extend the knurl to the layout line. Force the knurling tool slightly deeper into the work, reverse the direction of the carriage feed, and engage the carriage feed. Allow the knurling tool to feed to the right until approximately half of the rolls extend beyond the end of the work. Never allow the knurling tool to feed entirely off the end of the work. Repeat the knurling operation until the diamond-shaped impressions converge to a point. If additional passes are made after the pattern has completely formed, the points may be stripped away from the surface. Move the knurling tool away from the work and, with the work revolving, clean the knurled work surface with a file brush. Remove any burrs, which may have formed at the end of the work due to the knurling pressure, by machining or by filing.

4-75. You use a slightly different method to start the knurling operation when you knurl between layout lines. You can perform this operation in the following manner. Set up the work, lathe and knurling tool as usual. Swing the compound rest to the right 5° to reduce the starting pressure. Position the knurling tool slightly to the left of the right-hand layout line and start the knurl by forcing the corner of the roll into the work surface. Move the carriage by hand, if the knurl is tracking properly, to extend the knurl to the right-hand layout line. Move the knurling tool clear of the work and position it parallel to the work surface by swinging the compound rest back to the left 5°. Continue the knurling operation in the same manner as knurling to the end of the shaft.

4-76. Regardless of the method of knurling that you use, 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 36. 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

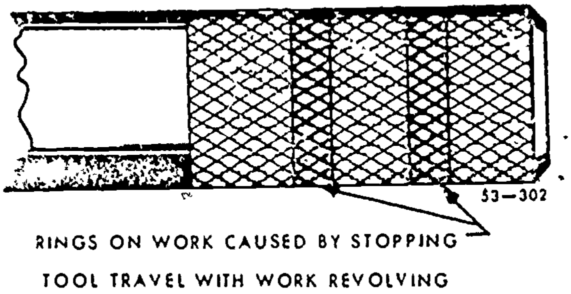


Figure 36 Rings on a knurled surface

work well oiled throughout the operation. Check the adjustment of the tailstock center frequently. The pressure of the knurling operation may cause the centers to loosen slightly.

5. Lathe Attachments

5-1 Lathe attachments play an important role in the production and repair of aircraft and equipment parts. They increase the output of the shop by taking the place of machines which can do only a few jobs. This helps to cut operating costs. We will discuss some of these attachments.

5-2. **Turret Attachment.** The turret attachment is a cylinder or head arranged to turn and slide on the ways of a lathe. It may be fitted with sockets or chucks to receive the various cutting tools. This attachment can be of great help in limited production work.

5-3. **Thread-Chasing Dial.** The thread-chasing dial, shown in figure 37, is a device that indicates the position at which to close the split nut so that the tool will always follow in the same groove as the previous cut. It is of great assistance in cutting threads in the lathe, especially when the thread to be cut is not a common division or multiple of the lead screw pitch. The dial is attached to the carriage and driven by means of the lead screw. An instruction plate, which is generally attached to the apron of the lathe, gives information about the use of the thread-chasing dial for various threads. Carefully observe it before you start the threading operation. Some types of lathes are not equipped with thread-chasing dials, but have a reversing lever to take its place. 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 and it is not necessary to use the thread-chasing dial.

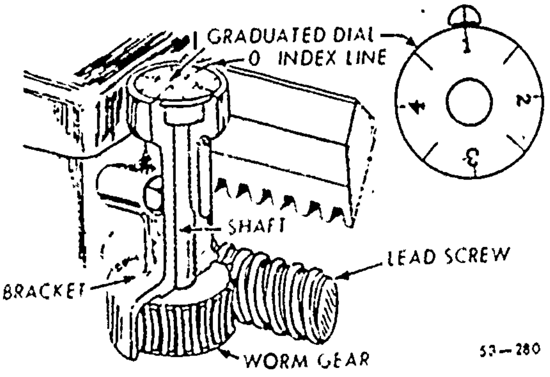


Figure 37 Thread-chasing dial

5-4. **Center Rest.** The center rest consists of a frame and three adjustable jaws, figure 38, which support the work at some point along the axis of the work. One purpose of the center rest is to prevent springing or deflection of slender flexible work. Another is to furnish auxiliary support to permit taking a heavier cut. Still another is to support work for drilling, boring, or internal threading. An overarm is provided so that work may be removed and replaced without disturbing the jaw adjustment in order to permit machining duplicate pieces.

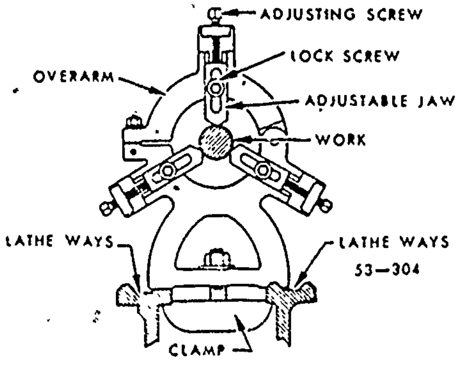


Figure 38 Center rest

5-5. The center rest may be used on work held in a lathe chuck or on work held on the headstock center. It is first necessary to machine a concentric bearing surface on the work, at the place where the jaws are to be applied. After the work is mounted and the center rest is clamped in place to the ways of the lathe, the jaws are carefully adjusted to the surface of the work. A slight clearance must be left between the jaws and the work to allow for lubrication in order to prevent scoring of the work and the center rest jaws. When you are adjusting the center rest jaws to ground work, copper shims may be used between the jaws and

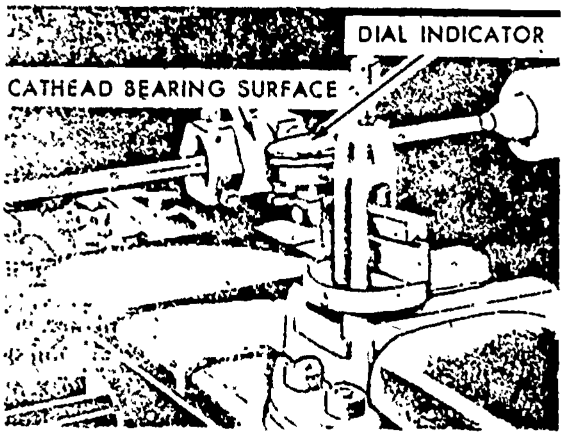
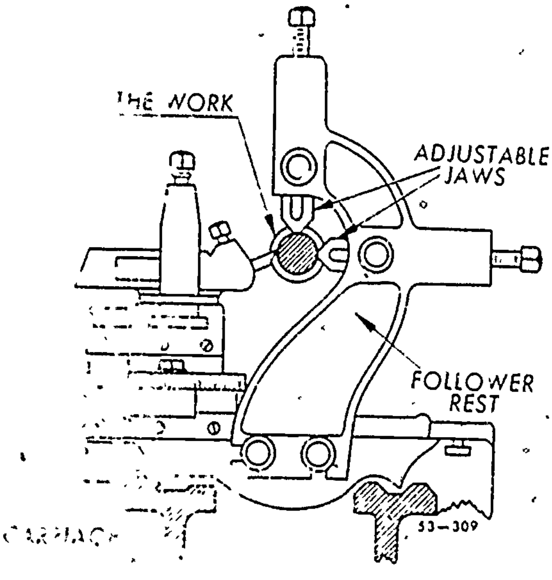


Figure 39 Cathead.

work to prevent marring the finished surface of the work.

5-6. **Catheads.** The cathead, figure 39, consists of a concentric bearing surface and a series of adjustable set screws. The cathead is used to provide a bearing surface for the center rest jaws on work of an irregular nature, hexagonal and square stock, or on long slender work that is too small for the center rest jaws. When you use the cathead, the bearing surface is adjusted to the work by means of the adjusting screws until it runs true. A dial test indicator may be used to check the trueness of the bearing surface as the work is revolved by hand. The center rest jaws are then adjusted to the bearing surface of the cathead.

5-7. **Follower Rest.** Long shafts that are likely to be sprung out of alignment by the thrust of the cutting tool often require the support of a follower rest, figure 40. A follower rest mounts on the carriage of the lathe and hence moves with the tool, backing up the work opposite the point of tool thrust. Follower rests have two adjustable supporting jaws. One holds the work down to prevent the tendency to climb on the tool. The other is behind the work to counter the thrust of the tool.



Follower rest

5-8. The cutting tool may be set to precede the jaws of the follower rest for the first cut and then set to follow the jaws for the second cut. This eliminates the necessity of adjusting the jaws for each cut. The jaw adjustments are made the same as the center rest.

5-9. A follower rest is useful in turning and threading long work of small diameter. Since the diameter of the work does not change, one adjustment of the jaws is sufficient. When it is used in a threading operation, it is necessary to remove any burrs by filing. Burrs which may bear against the follower rest jaws may cause the work to revolve out of alinement.

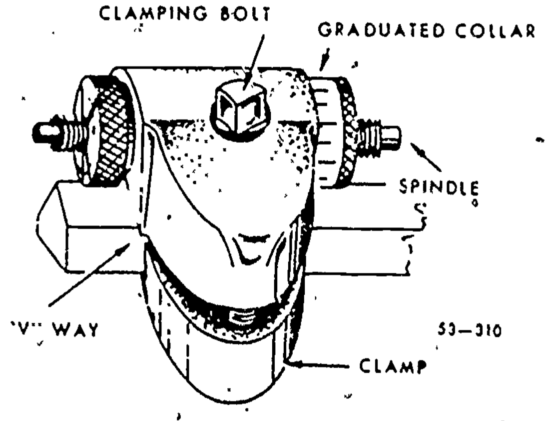


Figure 41 Micrometer carriage stop

5-10. **Micrometer Carriage Stop.** The *micrometer carriage stop*, shown in figure 41, is used to accurately position the lathe carriage. Move the carriage so that the cutting tool is approximately positioned, and 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 thousandths of an inch) make it possible for you to accurately set the spindle. Next, bring the carriage into contact with the micrometer spindle again. The carriage can be accurately positioned within 0.001 of an inch, which is very useful when you are facing to length or machining shoulders to an exact

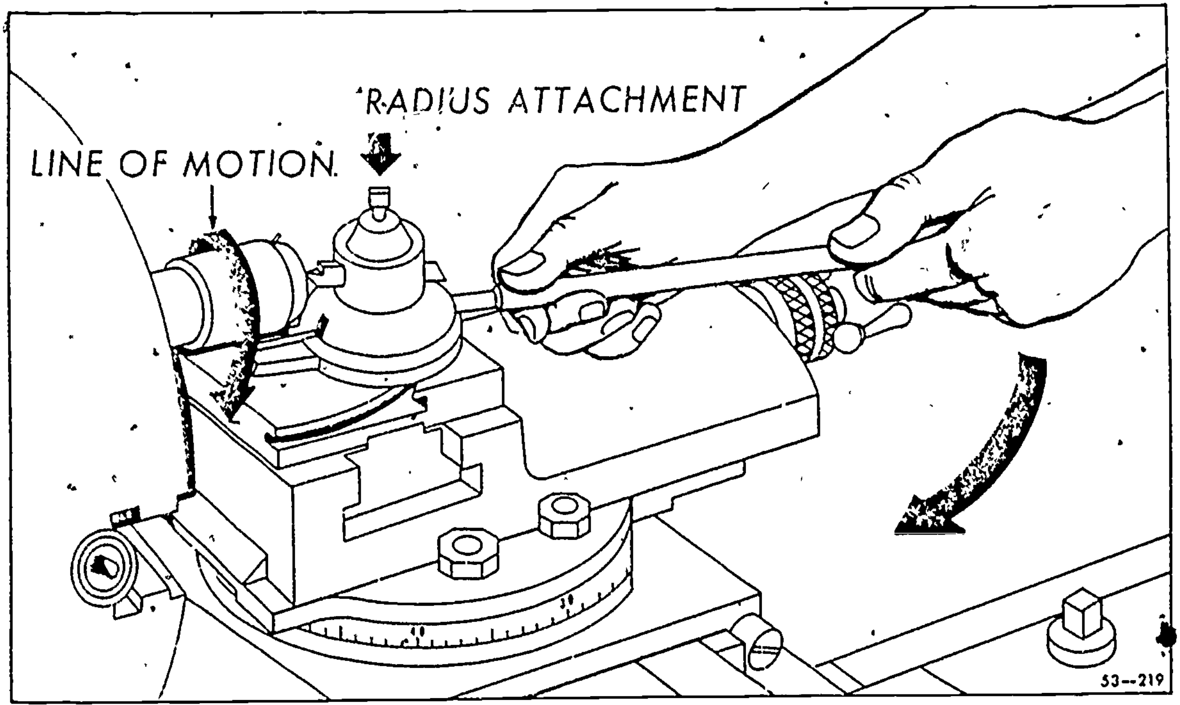


Figure 42.- Radius turning attachment

length. 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 hold to take measurements and then reposition it to take additional cuts. NOTE. Always bring the carriage into contact with the stop by hand. Use power feed to bring the carriage within 1/32 inch of the stop, and move the carriage by hand the remaining distance

5-11 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 42 shows a hand-operated radius attachment being used to machine a convex radius.

6 Special Threading Operations

6-1. You will occasionally be required to do special threading operations, such as the cutting square threads. Acme threads, multiple threads, metric threads, and odd size threads. The following information will assist you in the identification and machining of these types of threads.

6-2. Cutting Square Threads. The principle uses of square threads are for jackscrews and for some machine tool feed screws. Square threads have been replaced to a large extent, however, by Acme threads, because of the difficulty encountered in machining the square threads.

6-3. A square thread is a thread whose sides are parallel. The depth of the thread is equal to the width of space between the teeth. This space is, theoretically, equal to one-half of the pitch. It is necessary, in practice, however, to make the space in the nut a trifle wider than the thread to permit a sliding fit. The threads in the screws are made exactly according to the theoretical standard. The width of the point of the tool for cutting screws that require only an ordinary degree of finish and cut with one tool is, therefore, exactly one-half of the pitch. The width of the point of the tool for cutting taps which are to be used for threading nuts is slightly less than one-half the pitch. The width of an inside thread tool for threading nuts is slightly more than one-half the pitch.

6-4. Since square and Acme threads are usually cut with a relatively coarse pitch, a large helix angle is produced. Therefore, careful attention must be given to the clearance of the tool bit. In order to check accurately the amount of clearance necessary to prevent binding of the bit in the groove, a gage may be made, as shown in figure 43. To make such a gage, you should scribe a line at 90° to one side of a small rectangular piece of sheet metal and on it lay off a distance equal to the cir-

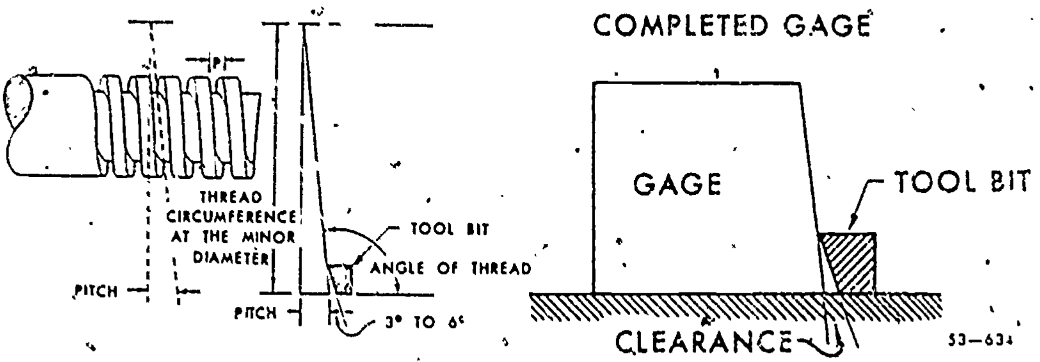


Figure 43. Gage for checking threading tool.

cumference of the thread to be cut at its minor diameter. At one end of this line and at right angles to it, lay off a distance equal to the pitch of the thread to be cut. Connect the ends of these two lines, and the angle of this third side is the angle of the required thread at its minor diameter. The sheet metal is then cut off along this last line and the tool bit is gaged as illustrated in figure 42. The tool bit is considered correctly relieved when its side clears the gage from 3° to 6°.

6-5. For square external and internal threading, the compound rest is set parallel to the axis of the machine in order that material may easily be removed during finishing cuts. For precision thread cutting, a tool bit having 0° side rake with respect to the helix angle and ground from 0.002 inch to 0.008 inch undersize is used for the roughing cut while finishing is accomplished with tool bits having left and right rake. The front and back sides of the thread are finished by moving the compound rest parallel to the axis.

6-6. **Cutting Acme Threads.** The American Standard Acme thread is a thread whose depth equals one-half the pitch plus an allowance for clearance, and the sides of the thread form an included angle of 29°, as shown in figure 44. The allowance for clearance for a single depth is .010 inch on threads of 10 per inch and less, and .005 inch on threads of more than 10 per inch. Formulas accompanying figure 44 for depth and root values are for threads having .010 inch clearance (1 to 10 threads per inch, inclusive). The pitch is equal to one divided by the number of threads per inch and is written P = 1/N. (For example, if a feed screw has eight threads per inch, the pitch is 1/N or 1/8 inch.) Since the depth of thread is 1/2P plus clearance, the pitch formula can be transformed into the depth formula as follows. Depth = (1/2N) + .010" = (.500/N) + .010, as shown in figure 44.

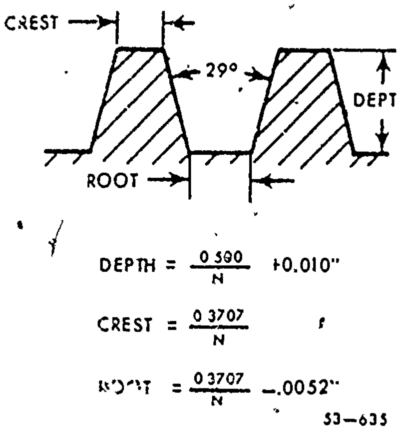


Figure 44. Acme thread.

6-7. The Acme thread is generally used on feed screws where it is rapidly replacing the less durable square thread. An Acme thread is slightly weaker but wears less rapidly than the square thread and may be cut with a die more readily than a square thread. When an Acme thread is engaged by a half-nut, as in a lathe apron, engagement or disengagement is more readily made than with a square thread. An adjustable split nut may be used in connection with an Acme screw thread to compensate for wear and to eliminate backlash or lost motion because of its angular sides. The lathe setup for cutting the 29° thread is the same as that used for cutting a square thread.

6-8. The Acme thread is cut on the lathe with a tool which has been ground to fit the correct pitch of a 29° Acme thread gage. If the same tool is to be used for both roughing and finishing operations, it is ground to an included angle of 29°, a side clearance of 3° to 6° with respect to the thread helix, 0° side and back rake, and for a thread one pitch smaller than the thread being cut. After the thread is cut to the correct depth, the cutting edge of the roughing tool is sharpened and reset in the same position for finishing each side of the thread individually by feeding the compound rest (set parallel to the ways) to the right and left for the depths of cut This method will prove satisfactory for most of the finer pitches and for threads requiring an ordinary degree of finish. For precision Acme threading, the roughing tool is used as previously described and the sides of the thread are individually finished by the use of two separate finishing tools, each having side rake with respect to the side being finished. Their cutting edges are individually set horizontally at center height and to the tool setting gage similar to the precision finishing operations used for American Standard threads and a square thread.

6-9. For the coarser pitches that are to be finished by individual finishing tools, the roughing tool may be ground with 0° side rake with respect to the helix angle of the thread in order to produce a free cutting action. During the roughing operation, the depths of cut for large threads may be alternated between the cross-slide and compound rest in order to eliminate gouging that may occur when cutting with all three edges of a large Acme threading tool. Another method sometimes used for roughing out the thread is cutting the Acme thread to the proper depth using a square tool having a width equal to the width of root of the Acme thread. This method is less desirable because the square tool is not as strong as the Acme tool and more time is required to complete the thread.

6-10 **Cutting Multiple Threads.** A multiple thread, as shown in figure 45, is a combination of two or more threads, parallel to each other, progressing around the surface into which they are cut. If a single thread is thought of as taking the form of a helix, that is, of a string or cord wrapped around a cylinder, a multiple thread may be thought of as several cords lying side by side and wrapped around a cylinder. There may be any number of threads, and they start at equally spaced intervals around a cylinder. Multiple threads are used in cases where rapid movement of the nut or other attached parts is desired and any weakening of the thread is to be avoided. A single thread having the same lead as a multiple thread would be very deep in comparison to the multiple thread.

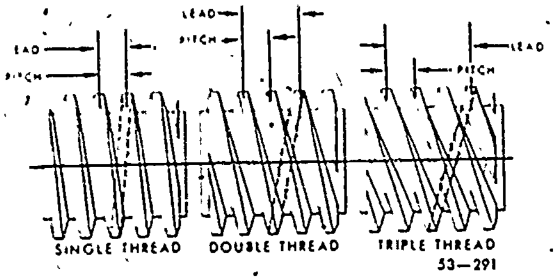


Figure 45 Comparison of single and multiple-lead threads

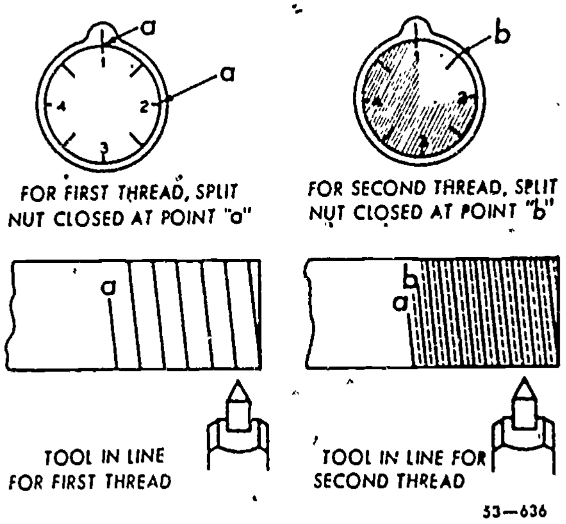


Figure 46. Cutting multiple threads.

6-11. The tool selected for cutting multiple threads has the same shape as that of the thread to be cut and is similar to the tool used for cutting a single thread except that greater side clearance is necessary. The helix angle of the thread increases with an increase in the multiple thread. The general method for cutting multiple threads is about the same as for single screw threads, except that the lathe must be geared to the number of single threads per inch, or with reference to the lead of the thread, and not the pitch, as shown in figure 45. Provisions must also be made to obtain the correct spacing of the different thread grooves. This may be accomplished by using the thread-chasing dial, setting the compound parallel to the ways, using a multiple driving plate, or using the stud and box gear break up.

6-12. The use of the thread-chasing dial is the most desirable method for cutting 60° multiple threads. With each setting for depth of cut with the compound, successive cuts may be taken on each of the multiple threads so that the use of thread micrometers is made possible. To explore the possibility of using the thread-chasing dial, it is first necessary to find out if the lathe can be geared to cut a thread having a lead equal to that of one of the multiple threads For example, if it is desired to cut 10 threads per inch, double threaded, it is necessary to divide the number of threads per inch by the multiple (in this case 10/2) to obtain the number of single threads per inch (in this case 5). The lathe is then geared for the number of single threads per inch. To use the thread-chasing dial on a specific machine: you

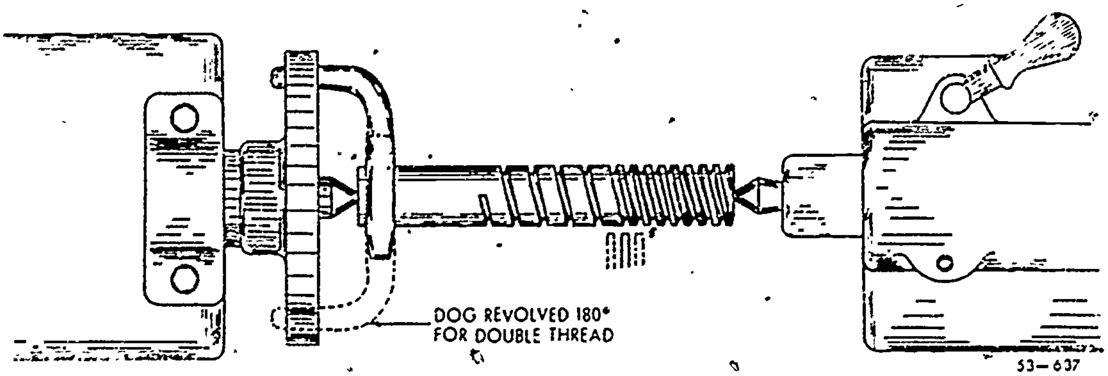


Figure 47. Use of a slotted drive plate.

should refer to instructions usually found attached to the lathe apron. If, for 5 threads per inch, you should engage the half nut at any numbered line on the dial, the same thread would be cut at positions 1 and 2 on the dial, as shown in figure 46. If the dial is then covered with the hand, leaving the part uncovered between those adjacent positions that cut the groove, positions 1 and 2 in figure 46, a check should be made to see if there is a point of engagement midway between positions 1 and 2 for the second thread. The second groove of a double thread lies midway of the flat surface between the grooves There is a point of engagement in this case, position "b" in figure 46. For the same depth of cut, the half nut is engaged first at one of the "a" positions, then at "b" position so that alternate cuts bring both thread grooves down to size together. In the event that positions 1 and 2 would indicate the engagement place for groove of a triple thread, it would be necessary to have two positions of engagement, equally spaced, between positions 1 and 2 in order to cut the other grooves of the triple thread.

6-13 Cutting multiple threads by positioning the compound parallel to the ways should be limited to square and Acme external and internal threads, since that is the normal position of the compound for cutting those threads. The compound rest is set parallel to the ways of the lathe and the first thread is cut to the finished size. The compound and tool is then fed forward parallel to the thread axis a distance equal to the pitch of the thread and the next thread is cut, etc. Any desired multiple thread may be cut in this manner, provided that the lathe is geared to the lead of the multiple thread.

6-14. The multiple driving plate method of cutting multiple threads involves changing the position of the work between centers for each groove of the multiple thread. One method of accomplishing this is to cut the first thread groove in the conventional manner. Then the work is removed from between centers and replaced with the tail of the dog in another slot of the drive plate, as shown in figure 47. Two slots are necessary for a double thread, three slots for a triple thread, etc. The number of multiples that can be cut by this method depends upon the number of equally spaced slots in the drive plate. Special drive or index plates are obtainable, so that a wide range of multiples may be accurately cut by this method.

6-15. Another method of cutting multiple threads is to disengage either the stud or spindle gear from the gear train in the end of the lathe after cutting a thread groove. Then turn the work and spindle the required part of a revolution, and reengage the gears for cutting the next thread. If it is necessary to cut a double thread on a lathe having a 40-tooth gear on the spindle, the first thread groove is cut in the ordinary manner. Then one of the teeth on the spindle gear that meshes with the next driven gear is marked and the mark is carried onto the driven gear, in this case the reversing gear. The tool diametrically opposite the marked spindle gear tooth (the 20th tooth of the 40-tooth gear) is marked also. The tooth next to the marked tooth should be counted as tooth number one. The gears may then be disengaged by placing the tumbler (reversing) gears in neutral position, the spindle is turned one-half revolution or 20 teeth on the spindle gear, and the gear train is reengaged. The stud gear may be indexed as well as the spindle gear. However, if the lathe does not have a 1 to 1 ratio between the spindle and stud gears, the stud gear instead of being turned as when geared for a 1 to 1 ratio would be given a proportional turn depending upon the ratio of the gearing. The method of indexing the stud or spindle gears is only possible when the number of teeth in the gear indexed is evenly divisible by the multiple desired. Some of the newer type lathes are equipped with a sliding sector gear that can be readily engaged or disengaged with the gear train by shifting a lever. Graduations on the end of the spindle show when to disengage and reengage the sector gear for cutting various multiples.

6-16. **Change Gears for Threading.** In preparing the lathe for thread cutting, the levers on the quick change gearbox are set to correspond to the number of threads per inch desired. Generally there are two or more levers on the box, along with a chart giving the position of these levers for the various threads to be cut. Change gears on older type lathes had to be set up in the gear train by hand to give required ratios between the spindle and lead screw to produce the desired leads. Change gears are those gears whose position and size may be changed to establish various ratios between the speed of the spindle and the speed of the lead screw. A set of change gears usually ranges in size from 16 teeth to 100 teeth by steps of 4 teeth, plus a few odd sizes for cutting certain odd thread pitches.

6-17. *Metric change gears.* Transposing gears are gears used to cut metric threads on a lathe. When metric threads are cut, the lead of the thread is given in millimeters, instead of in the number of threads per inch. To find change gears for cutting metric threads it is necessary to determine the number of threads per inch corresponding to the given lead in millimeters. As an example, suppose a thread is to be cut with a pitch of 2 millimeters on a lathe with a lead screw of 6 threads per inch. There are 25.4 millimeters per inch. The number of threads per inch is 25.4 divided by 2. Place the lead screw constant in the numerator and the desired number of threads in the denominator:

$$\frac{6}{\frac{25.4}{2}} = 6 \div \frac{25.4}{2} = \frac{6 \times 2}{25.4} =$$

$$\frac{12}{25.4} = \text{ratio to obtain desired lead}$$

This then represents the ratio between the change gears necessary to cut the metric thread. The 25.4 shown in the denominator must be converted into a whole number and the smallest whole number by which it can be multiplied to get a whole number is 5. Thus, 25.4 x 5 = 127. One gear with 127 teeth is always required to cut metric threads with a lathe, and the other gear required in this example has 60 teeth, as shown below:

$$\frac{12}{25.4} \times \frac{5}{5} = \frac{60}{127} = \frac{\text{Driving Gear}}{\text{Driven Gear}}$$

6-18. *Increasing threading ranges.* To increase the range of threads leads that can be cut on a lathe with a quick change gearbox, the following formula for calculating gears may be used:

$$\frac{A \times B}{C} = X$$

Where

A = number of teeth on the spindle or spindle stud gear
B = number of threads per inch to be cut.
C = number nearest to B on the quick change gearbox that will make X an even number.
X = number of teeth on the spindle replacement gear.

As an example, assume it is desired to cut 27 threads per inch and the gear on the spindle or spindle stud has 32 teeth. By trial it is found that 24 threads per inch on the quick change box is the nearest to 27 which will make X come out to an even number.

$$\frac{A \times B}{C} = X \quad \frac{32 \times 27}{24} = 36$$

The 32-tooth gear on the spindle stud is replaced by the 36-tooth gear. Then, with the lathe set to cut 24 threads per inch, 27 threads per inch will be cut instead. NOTE: Always replace the original spindle gear after cutting a thread not included in the quick change gearbox, because the lathe will not cut the threads listed until the original gear is replaced.

7. Lathe Maintenance

7-1. Lathe maintenance is important and must not be neglected. Good maintenance makes it possible for you to get the best results in lathe operations and lengthens the life of the machine. A lathe that is improperly maintained soon wears out. To keep the lathe in the best operating condition, it is necessary to make frequent inspections and various adjustments. A periodic check should be made of such things as levelness, spindle bearing condition, clutches, gibs, crossfeed and lead screws, gearing, and lubrication. Adjustments should be made only when it is necessary. Since the different makes of lathes vary in their construction, it is always best to follow the manufacturer's directions on making these adjustments.

7-2. **Gib Adjustments.** Gibs may be either tapered or flat metal bars for taking up wear between bearing surfaces, such as the dovetailed surfaces of the cross-slide, compound rest, or carriage. Gibs are provided with thrust screws by means of which the necessary adjustments are made. In making gib adjustments, first loosen the lockscrew. Now tighten the gib screw until a smooth snug fit is obtained. Then lock the adjustment. If gibs are adjusted too tightly, binding will result. Gibs on the compound slide should be fairly tight when the compound is not being used for cutting angles.

7-3. Headstock Spindle. The spindle bearings of the newer type are generally of the antifriction or taper roller bearing type. They are properly adjusted at the factory and need not be adjusted for long periods of time. When spindle bearing adjustments are necessary, they may be adjusted by means of a thrust nut which is provided for that purpose. The adjusting nut is generally located on the rear end of the spindle, outside the headstock to allow for easy adjustment When you make spindle bearing adjustments, place the headstock gearing in neutral so that the spindle revolves freely. Remove the back gearing guard and release the locknut or set screws that hold the thrust nut in place. With a spanner wrench provided for that purpose, turn the thrust nut clockwise until no end play is detected and the spindle can still be rotated freely by hand. A drive plate should be placed on the spindle so that the spindle may readily be rotated by hand. Adjustments on other types of bearings are essentially the same CAUTION: Before making any spindle adjustments make certain the trouble does not lie elsewhere by checking and making other adjustments.

7-4. Driving Clutches. The types of clutches incorporated by various lathe manufacturers may vary. Consequently, the method of clutch adjustment is not the same. The more expensive precision lathes generally have the friction type of clutch similar to that of an automobile. To make adjustments of the friction type clutch, remove the clutch guardplate, pull back the adjusting pin, and rotate the adjusting yoke or ring to the right or clockwise until the pin slips into the next hole or notch. Proper adjustment is made when the clutch level snaps in and out of engagement.

7-5. Apron Feed Clutches. Apron feed clutches vary somewhat in design. They may consist of two friction cones or two serrated plates held together under cam pressure and released under spring tension. A thrust screw in the clutch shaft makes whatever adjustment is necessary To adjust, turn the thrust screw clockwise until the clutch level snaps in and out of engagement

7-6. Lead Screw. The lead screw is adjusted for end play by removing the cap from the end of the screw and tightening the thrust collar Adjustment for end play can be checked by engaging the half nut and moving the carriage. by hand, back and forth along the ways.

7-7. End Gearing. To make adjustments on end gearing, remove the guard and loosen the stud nuts of the gear quadrant and mesh the gears until a slight clearance is obtained between the mating teeth. Tighten all nuts securely. Proper adjustment is made when a smooth action is obtained. No adjustment is complete until all guards have been properly replaced.

7-8. Periodic Oil Changes. When a lathe is run daily the oil should be changed in the headstock reservoir about every 6 months. A good grade of machine oil of SAE 20 or 30 should be used. The operator's instruction manual will state the grade of oil that should be used for the various machines. When changing oil in the reservoir, the plugs should be removed and the reservoir flushed with kerosene before refilling The machine should be left running during the flushing process. All bearings fitted with oil cups should be oiled daily or as often as necessary. The performance of a lathe depends on the attention it receives. During the first 3 or 4 days or "the breaking in period," all bearings should be carefully oiled and watched to see that none run hot.

Milling Machine Work

MANY YEARS AGO a machinist, whose name is unknown to us now, placed a mandrel upon which he had mounted a cutter between the centers of a lathe, fastened the object he was machining to the lathe cross-slide, and invented the milling machine! From this simple invention has evolved one of the most versatile and important metal cutting machines in existence, the modern milling machine. In this chapter we discuss special milling applications, helical milling, single point milling, milling machine attachments, gearing and gear cutting, and maintenance of the milling machine.

8. Special Milling Applications

8-1. You will be called upon from time to time to do repair and fabrication jobs, which require special milling operations, such as gang milling, slotting, boring, and milling cams. The following information will help you to understand these operations and how they are done

8-2. **Gang Milling.** Gang milling involves the use of two or more cutters mounted on the same arbor. All cutters may perform the same type of operation or each cutter may perform a different type of operation. For example, several workpieces may need a slot, a flat surface, and an angular groove cut in them. The best method to accomplish this would be gang milling, as shown in figure 48. All the completed workpieces would be the same. Remember to check the cutters carefully for proper size.

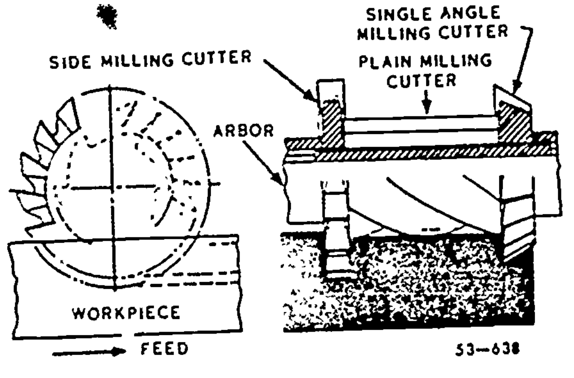


Figure 48 Gang milling setup

8-3. Gang milling setups can save both time and labor when several identical parts must be machined on a milling machine. If you think in production terms when you receive a production-type work order, you may be able to devise cutter setups which will enable you to do the necessary work with a minimum amount of time and effort.

8-4. **Slotting.** Slotting is often regarded as a shaper operation, but it can be done on a milling machine with a slotting attachment. The slotting attachment is capable of internal slotting operations, such as the machining of squares, hexagons, 12-point sockets, keyways, splines, and gear teeth, and is especially useful for tool and die work.

8-5. Some of the more common shapes of holes are often slotted on a milling machine with the slotting attachment. A square hole is often machined to mate with a square surface on the end of a mating member. This assures a positive drive between two members, such as a socket wrench and handle, and some boring tools and boring bars. Square sockets may extend part way or all the way through the piece. In order to slot a socket that extends part way into the bore, you must recess the bore to clear the slotting tool point and provide a space for chip clearance. Slotting hexagonal shapes, or even 12-point holes for socket wrenches, is often done in Air Force machine shops. A 12-point socket is in reality two internal hexagons of the same size with one hexagon machined within the other. Since 12 equally spaced cuts are required, you would use the index head to space the work. Here, you can use the direct index plate to advantage. Use a hole circle equally divisible by 12. When you slot work that must be equally spaced, mount the work in the index head chuck and aline the bore of the work concentrically to insure accuracy. In a hexagon. the length of one side is equal to one-half its diagonal. Determine the diagonal of the hexagon by multiplying the distance across flats by the constant 1.115. Grind the width of the edge of the slotting tool to equal the length of the side of the hexagon and round it to clear the bore. Swivel the slotting attachment to a horizontal or vertical position.-The index head must be swiveled so that the position of its spindle corresponds to the position of the slotting attachment ram. Usually, it is best to hold the work vertically in the index head chuck to provide better vision. However, if the work is too long to be held vertically but can be extended through the index head spindle, it can be held horizontally.

8-6. **Angular Milling.** Angular milling is the milling of a flat surface that is at an angle to the cutter axis. Angular milling cutters may also be used in this operation, as shown in figure 49. These angular milling cutters are made to a predetermined angle, such as those made for cutting dovetails. Angular milling may be done in several ways. It may be done by using the vertical head on the milling machine. This is done by tilting the head to the required angle and using an end mill or a shell milling cutter. The toolmaker's knee may be used for angular milling operations. The toolmaker's knee affords a rapid and convenient method of setting up angular work on a milling machine.

8-7. Angular parallels may be used for angular milling also. These parallels are accurately machined to a predetermined angle. With a full set they may be set to any angle needed. Care should be taken in using angular parallels not to damage their surfaces. The vise and work should be kept clean at all times to insure that chips do not damage the parallels

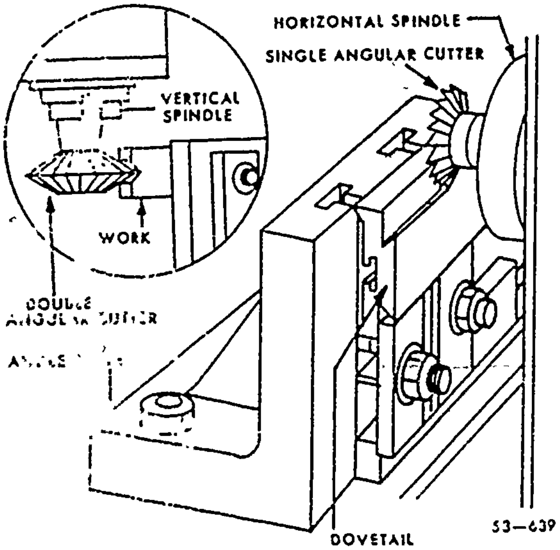


Figure 49. Angular milling.

8-8. **Milling Cams.** The method you select to machine a cam depends upon the design of the cam and the accuracy required. If the design of the cam and the accuracy permit, it is sometimes possible to saw the cam out on a contour machine or to machine the cam on a shaper. Usually, however, you will use a milling machine to machine a cam, such as cylindrical constant rise, and changing rise, especially when a high degree of accuracy is required.

8-9. *Cylindrical cams.* Cams that are grooved on the periphery of a cylinder are set up and machined in the same manner as for a helical groove. Calculate the lead of the cam in the same manner as for a helix and gear the index head to the table feed screw by means of the end gearing to produce the desired helix head. Usually, you will use an end mill, mounted in a vertical milling attachment, to machine the cam groove.

8-10. *Constant rise cams.* A constant rise cam is machined by gearing the index head by means of the end gearing to produce the desired lead. The index head should be in the vertical position. Use the side surface of an end mill to machine the cam.

8-11. *Changing rise cams.* Some cams are designed so that the rise changes for various portions of the cam. For example, a cam may have a rise of 0.500 inch in 80°, a rise of 0.750 inch in the next 70°, and then drop to the original starting point during the remaining portion, a decrease of 1.250 inches during the last 210°. To machine a cam, such as the one just described, you must determine the lead for each rise and then machine each rise (or cam segment) separately. You can determine the lead of each cam segment by using the formula:

$$\text{Lead of segment} = \frac{360^\circ}{\text{included angle of segment}} \times \text{rise in segment}$$

If, for example, a cam segment has a rise of 0.500 inch in 80°, the lead would be determined as follows:

$$\text{Lead of segment} = \frac{360}{80} \times .500$$

$$\text{Lead of segment} = 2.250 \text{ inches}$$

8-12. You could machine each segment by changing the end gearing for each segment; however, there is a simpler method. Gear the index head to obtain a lead slightly greater than the largest cam segment lead. You can now obtain the lead for each cam segment by changing the angular position of the index head and the universal milling attachment with reference to the table, provided that the axis of the index

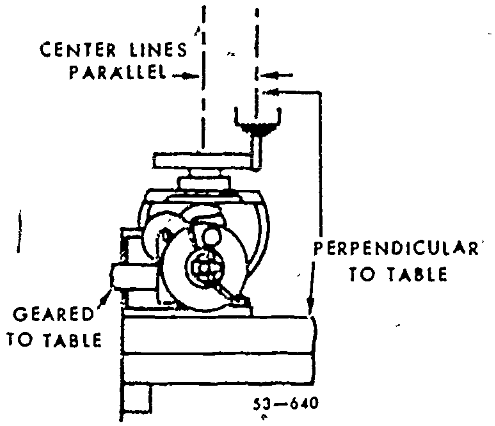


Figure 50 Cam and cutter in vertical position

head and the axis of the universal milling attachment are kept parallel. If the index head and cutter are parallel to the table no metal will be removed when the cam is rotated as the table moves forward. However, if the index head and cutter are perpendicular to the table, as shown in figure 50, the cutter will cut a spiral lobe equal in lead to the lead for which the machine is geared. Thus, if you position the index head and cutter at an angle to the table, as shown in figure 51, the lead produced will be between zero and the lead for which the machine is geared. Therefore, the lead produced can be controlled by changing the angular setting of the index head and cutter. You can determine the angle at which the cutter and index head must be positioned by dividing the desired lead of the cam segment by the lead for which the machine is geared (slightly greater than the largest cam segment lead). The result of the division will be the sine of the angle. You then find the angle in a table of trigonometric functions You can determine the angular settings and the end gears required for various leads from tables in various machinist publications, such as *Machinery's Handbook*.

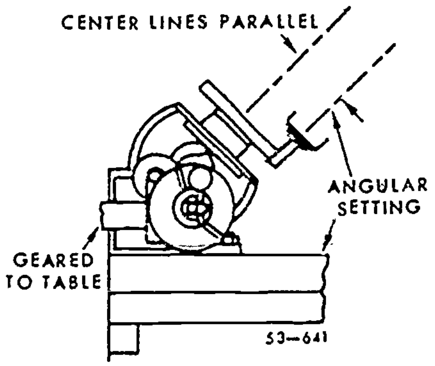


Figure 51 Cam and cutter in angular position

9. Helical Milling

9-1. A helix is a curve or path formed by the progressive rotation of a point around the surface of a cone or cylinder, such as the thread of a screw. The helix is formed on the milling machine by causing the work to rotate at the same time that it is fed longitudinally to the cutter. The rotation of the work is accomplished by gearing the table feed screw to the index head. As the table feed screw revolves, the work revolves at the same time it is fed to the cutter in a line parallel with its axis.

9-2. Helical work includes the milling of helical tooth milling cutters, helical reamers, twist drills, helical gears, some types of cams, etc. Milling a helix is similar to cutting a thread on the lathe. In cutting a thread, the tool moves a certain distance while the work makes one revolution. This distance is the lead of the thread and is governed by the gear ratio between the lathe spindle and its feed screw. This is also true in milling a helix. However, the distance it would have to feed in order to revolve once is termed the lead. The lead of a thread is usually short in proportion to its diameter and length while the lead of a helix may be long in proportion to its diameter and length. For example, a 1-inch single threaded screw having 8 threads per inch has a 1/8-inch lead, while the lead of a helix on a 1-inch reamer may be as much as 18 inches. The helix may make several turns around the work or it may make a fraction of a turn around the work and still have the same lead, depending upon the length of the helix. For instance, on a reamer flute 6 inches in length having an 18-inch lead, the helix would only make one-third of a turn around the piece. If the piece were 18 inches in length and had a helix with an 18-inch lead, the helix would make a complete turn around the work. In both cases the lead is the same. A helix may be either right hand or left hand, as the threads on a screw. A right-hand helix runs off to the right as it is viewed from the end. A left-hand helix runs off to the left.

9-3. The lead of a helix is the distance the helix advances in one complete turn around the work, measured on a line parallel with the axis of the work. If the gearing between the table feed screw and the index head worm were such as to cause the index spindle to revolve once as the table is fed 10 inches, then the lead would be 10 inches. The length of the work or the length of the cut makes no difference. To find the lead of a helix, multiply the diameter of the helix by 3.1416 and divide the product by the tangent of the helix angle.

$$\frac{D \times 3.1416}{\text{Tan of helix angle}} = \text{Lead}$$

EXAMPLE. A piece 2 inches in diameter is to have a helix whose angle is 22°. What is its lead?

$$\frac{2 \times 3.1416}{.40403} = 15.55'' \text{ lead}$$

The lead of the helix is expressed in inches, as [illegible]-inch lead, 14-inch lead, 2.5-inch lead, etc.

9-4. The angle of helix is the angle formed by a line tangent to the side of the helix and a line parallel with the axis of the helix The helix may be represented by a right angled triangle, as shown in figure 52. The base of the triangle represents the circumference of cylinder. The vertical leg of the triangle represents the lead of the helix, and the hypotenuse of the triangle represents the length of the helical path. The angle of the helix is the angle formed by the vertical leg and the hypotenuse of the triangle. The tangent of the angle of helix may be found by multiplying the diameter of the cylinder by 3.1416 and dividing this product by the lead of the helix:

$$\text{Tan of helix angle} = \frac{D \times 3.1416}{\text{Lead}}$$

EXAMPLE: Find the angle of a helix whose diameter is 3 inches with a lead of 12 inches.

$$\frac{3 \times 3.1416}{[illegible]} = .7854 = 38° 9'$$

9-5. A chart is usually available with each milling machine showing the number of gear combinations to use for most leads, or this information may be found in a machinist handbook. In the absence of a table of leads, ratios for various leads may be calculated.

10. Single-Point Milling

10-1. Single-point milling, or fly cutting, as it is sometimes called, is one of the most versatile milling operations. It is done with a single-point cutting tool shaped like 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. Formed cutters are expensive. There will be times when you will need a cutter for a very limited number of cutting and boring operations. In these situations it would be more economical to manufacture the cutter than to buy it.

10-2. The single-point or fly cutter can be used to great advantage in gear cutting. All that is needed is enough of the broken gear to grind the cutting tool to the proper shape. It can also be used in the cutting of splines and standard and special forms.

10-3. Boring, an operation that is too often restricted to a lathe, can easily be done on a milling machine. On a milling machine, you can bore very accurate holes with an offset boring head. Boring can be done with the aid of an offset boring head and a tool that is adjustable by means of a graduated adjusting screw mounted in the head. However, in the average machine, you will probably use a fly cutter for most boring operations.

10-4. Let's assume that your shop has received a large engine block that requires a

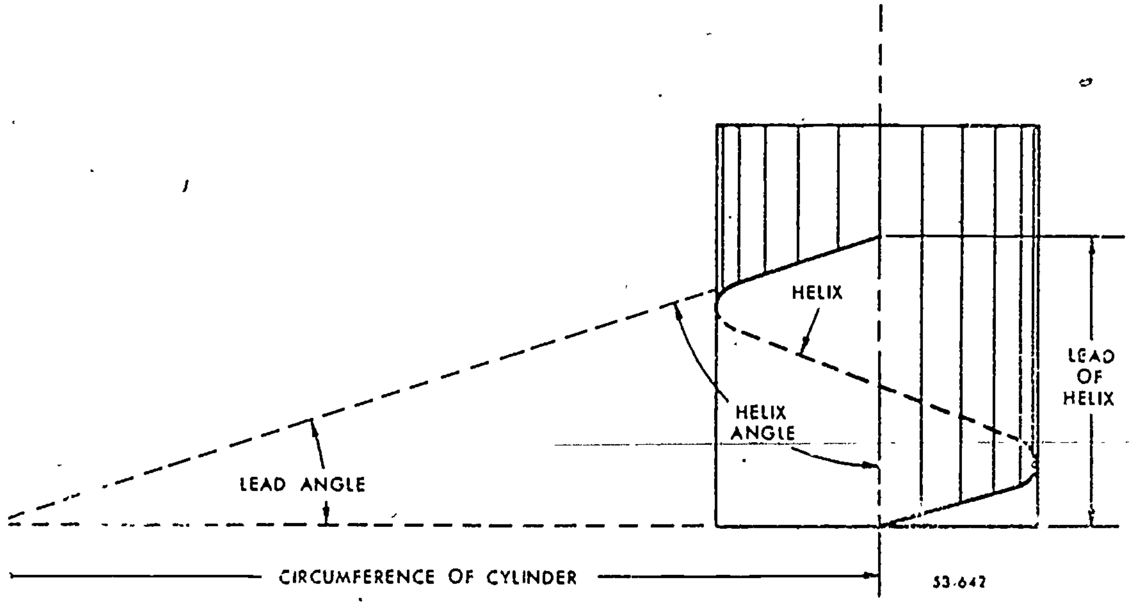


Figure 52 Development of a helix

bored hole for mounting a water pump. You could produce the desired bore, using a fly cutter and a milling machine, in the following manner. Mount the engine block on the milling machine table. Aline the spindle and the center of the hole to be bored by manipulating the crossfeed, longitudinal, and vertical feed cranks and by using a center finder or dial indicator. Once the center has been located, lock the longitudinal and vertical movements. If the part does not already have a hole, center drill it, lead drill it, and then use a drill slightly smaller than the size desired. Mount the offset boring head and set it to remove enough metal to true the hole. Start the machine and bore about 1/8 inch into the hole. Stop and measure the diameter of the bore to be sure that it is not oversize. If the hole is still undersize, continue boring through the hole. Remove the boring tool from the hole and readjust the tool bit for a finish cut. Start the machine and bore about 1/8 inch into the hole, and then check for proper size. If the size is correct, finish boring the hole.

11. Milling Machine Attachments

11-1. 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. 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 attachment you can perform milling operations at higher speeds than those for which the machine was designed. These attachments will make complex jobs easier.

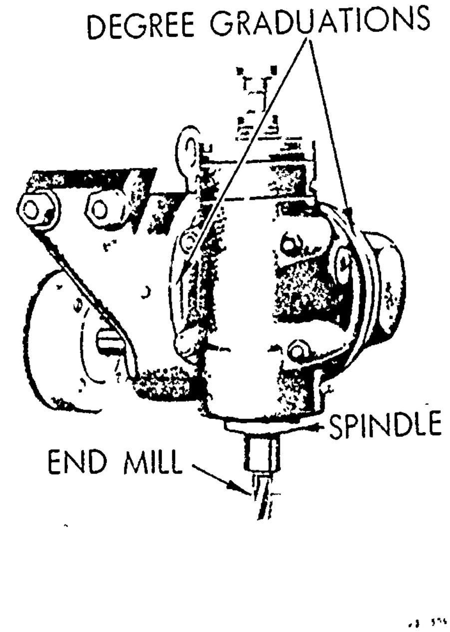


Figure 53. High-speed universal milling attachment.

11-2. **High-Speed Universal Attachment.** This device is clamped to the machine and is driven by the milling machine spindle, as you can see in figure 53. 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 obtain an efficient cutting action.

11-3. **Circular Milling Attachment.** This device, shown in figure 54, is a circular table which is mounted on the milling machine table. The circumference of the table is graduated in degrees. Smaller attachments are usually equipped for hand feed only and larger ones are equipped for both hand and power feed. The work is mounted on the circular table. 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.

11-4. **Rack Milling Attachment.** The rack milling attachment, shown in figure 55, is used primarily for cutting teeth on racks, although it can be used for other operations. The cutter is mounted on a spindle which extends through the attachment parallel to the table T-slots. An indexing arrangement is used to space the rack teeth quickly and accurately.

11-5. **Right-Angle Plate.** The right-angle plate, as shown in figure 56, is attached to the table. The right-angle slot permits mounting the index head so that the axis of the head is parallel to the milling machine spindle. With this attachment you can make work setups which are off center or at a right angle to the table T-slots. The standard size plate T-slots make it convenient to change from one setting to another for milling a surface at a right angle.

11-6. **Raising Block.** Raising blocks, as shown in figure 57, are heavy-duty parallels, which usually come in matched pairs. They are mounted on the table and the index head is mounted on the blocks. This arrangement raises the index head and makes it possible to swing the head through a greater range to mill larger work.

11-7. **Toolmaker's Knee.** The toolmaker's knee, as shown in figure 58, is a simple but useful attachment for setting up angular work,

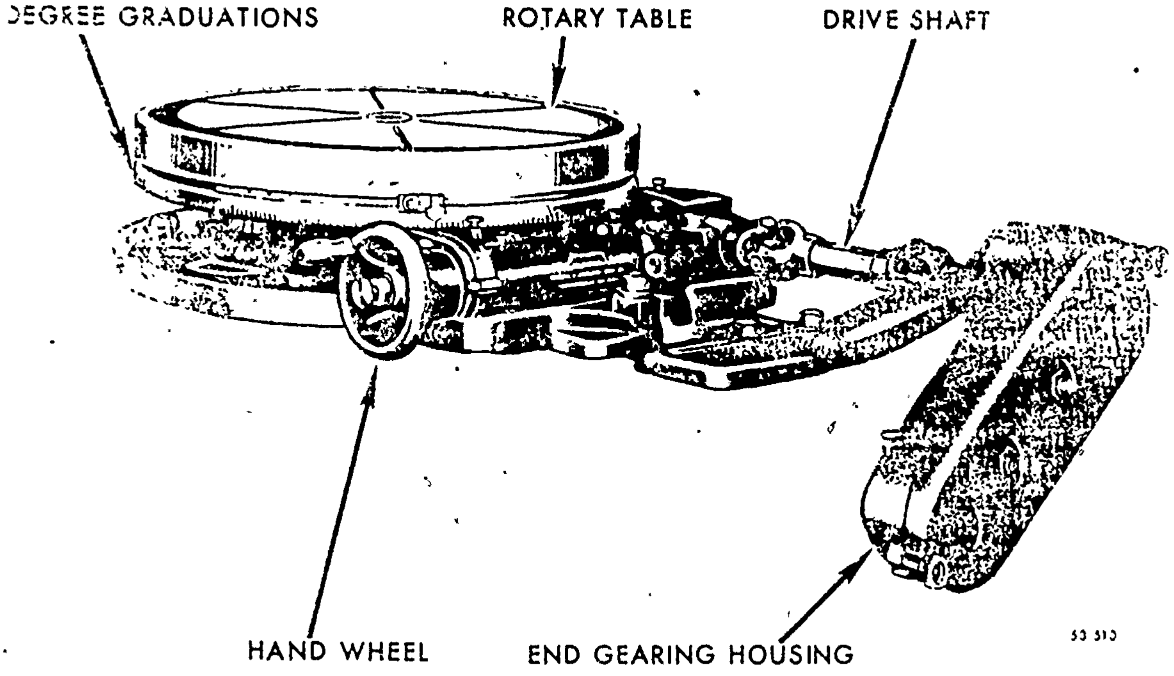


Figure 54. Circular milling attachment

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 table with either a built-in protractor head graduated in degrees for setting the table or a vernier scale for more accurate settings.

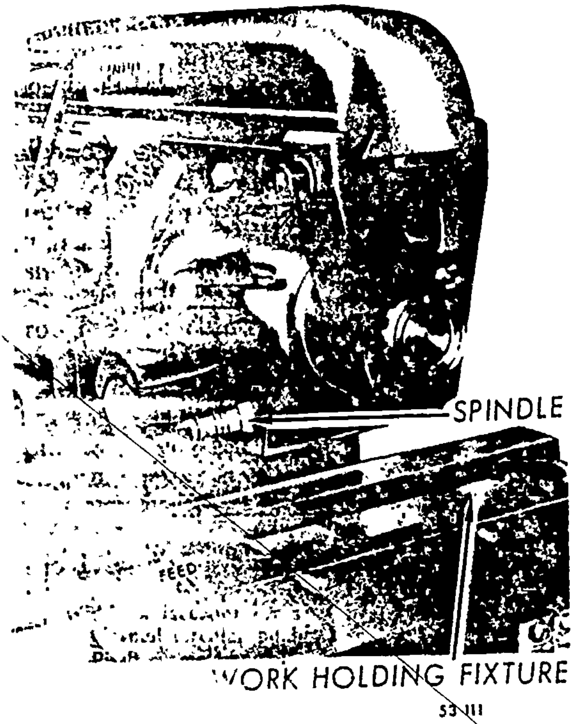


Figure 55. Rack milling attachment.

11-8. **Slotting Attachments.** You can often perform internal slotting operations, such as the machining of squares or hexagons, 12-point sockets, keyways, splines, internal gears, and various other shapes, with a slotting attachment, such as the one shown in figure 59. The slotting attachment is fastened to the milling machine column and driven by the spindle. A bull wheel and arm changes the rotary motion of the spindle to a reciprocating motion. You can vary the length of the stroke by loosening the nut which clamps the pivot shaft nearer or farther away from the center of the bull wheel. 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.

12. Gearing and Gear Cutting

12-1. Gears are produced for industry in large quantities, both by the shaping and the hobbing methods. There are still many uses for the older type of gear cutter using the single milling cutter and indexing from tooth to tooth, as in the milling machine methods. This is especially true in large gear work where several

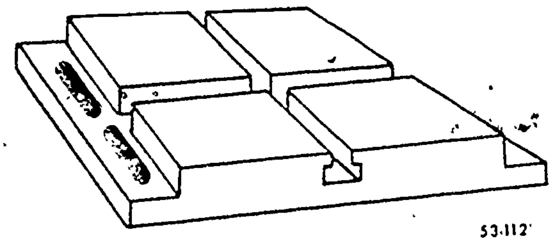

53-112

Figure 56 Right-angle plate

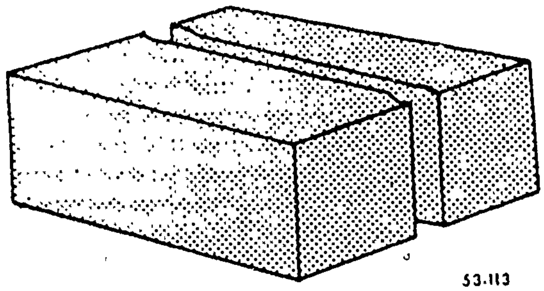
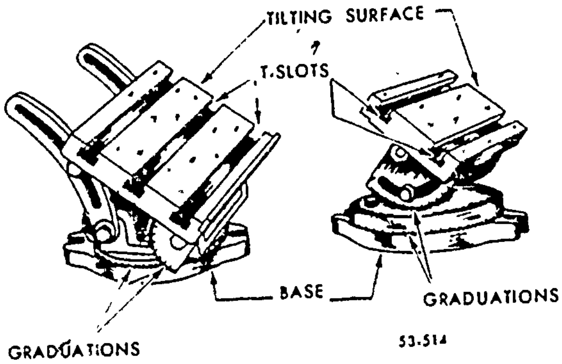

53-113

Figure 57 Raising block.



53-514

Figure 58 Toolmaker's knees

gear blanks may be mounted side by side and the gear teeth are cut at the same time by a single cutter. In a machine shop, such as is found in the Air Force, the number of gears that have to be made does not warrant the need for special machines for gear cutting.

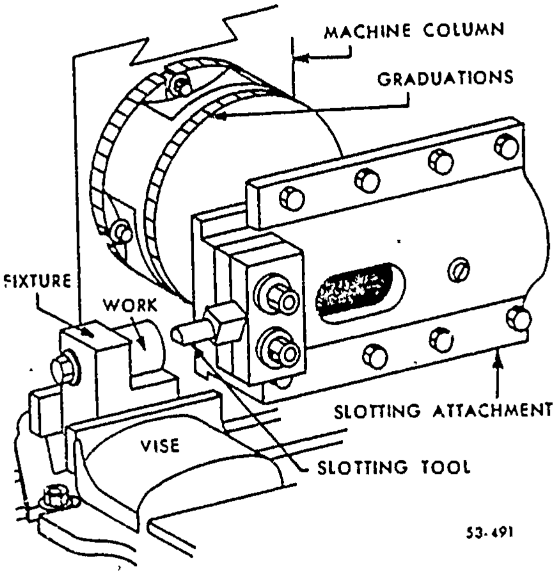
12-2. **Helical Gears.** Helical gears, as shown in figure 60, are coming into wider use in nearly all kinds of machinery. While their use dates back to over a half century, the difficulty of cutting them by the old method with a single-tooth form cutter retarded their use. With the development of the gear-hobbing machine and the reciprocating gear cutter or the gear shaper, helical gears can be machined about as easily as a spur gear. The quietness of the helical gears because of the simultaneous contact of several teeth make them desirable for automobile and many other uses. On a production basis, helical gears are cut on special gear cutting machines. For repair purposes where only one or a few are needed, they are best cut on a universal milling machine.

12-3. Helical gears have been called by several names, "spiral" being the most common. They have even been called "skew" gears in some sections. In late years there has been an effort made to call them by their right name which is "helical" not "spiral." Strictly speaking there are no spiral gears. The nearest approach to a spiral gear is a scroll, such as that used on the back of a universal chuck.

12-4. Helical gears differ from spur gears in that the teeth of a helical gear are inclined at an angle to the axis of the gear. Each tooth is a part of a thread or helix that winds across the outer rim of the blank. Helical gears operate smoothly and have great strength because of their sliding action and because of the number of teeth in contact at one time.

12-5. Helical gears may be used on shafts that are either parallel or at an angle to one another. When mounted on parallel shafts the angle of helix must be equal for both gears, but of the opposite hand. When mounted on shafts that are at right angles, the gears must have helix angles of the same hand.

12-6. Helical gears are ordinarily cut in the universal milling machine by using an index head which is geared to the swivel table and feed screw. This process involves many of the principles used for cutting helical forms. Helical gears may also be cut on a plain milling



53-491

Figure 59 Slotting attachment

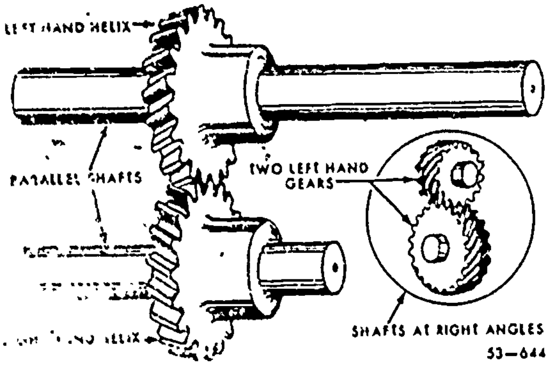
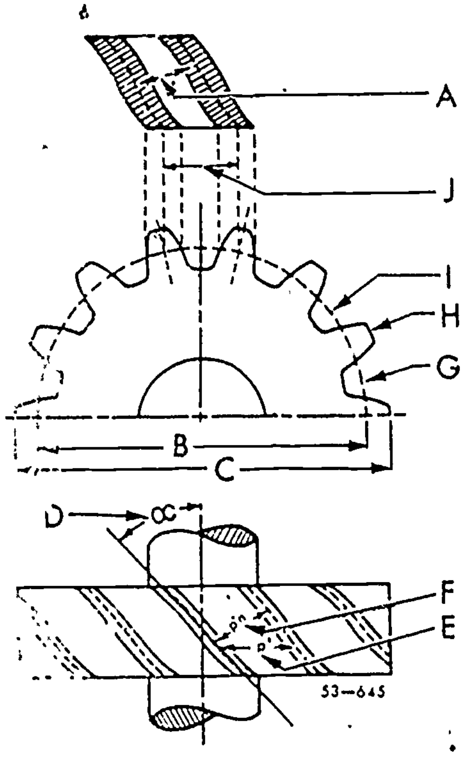


Figure 60. Helical gears

machine. This involves the use of a universal milling attachment, so that the form cutter may be set to the correct helix angle. Gears having an angle of helix greater than 45° may also be cut with this attachment. Another form of helical gears is herringbone gears. These are used on shafts that are parallel to one another. Herringbone gears virtually consist of two single helical gears of opposite helix angles. The primary object of herringbone gears is the elimination of end thrust which is the chief objection to the use of single helical gears.



A	Normal circular pitch	F	Normal circular pitch
B	Pitch diameter	G	Pitch circle
C	[illegible]	H	Addendum
	[illegible]	I	Dedendum
E	Circular pitch	J	Circular pitch

Figure 61. Helical gear terms.

12-7. The nomenclature of helical gears is the same as that of spur gears with a few exceptions. The calculations for determining tooth part values of helical gears differ somewhat from those of spur gearing because of the position of the teeth.

12-8. In the calculations and formulas used for helical gearing, reference is made to "circular pitch." The circular pitch of a helical gear is the distance from a point on one tooth to a corresponding point on an adjacent tooth measured along the pitch circle, as in the case of a spur gear.

12-9. Because helical gear teeth are not cut squarely across the face of the blank, a line drawn from a point on the pitch surface of one tooth to a corresponding point on an adjacent tooth at right angles to the tooth is termed "normal circular pitch." Figure 61 will help you understand the following helical gear terms:

Normal circular pitch (pn). The distance from a point on the pitch surface of one tooth to a corresponding point on an adjacent tooth at right angles to the tooth. The normal circular pitch equals the circular pitch times the cosine of the helix angle. $pn = p \times \cos A$

Normal diametral pitch (PN). The diametral pitch of the cutter. The normal diametral pitch equals pi (3.1416) divided by the normal circular pitch. $PN = \frac{3.1416}{pn}$

Number of teeth that the cutter selection is based on (N'). If the number of teeth actually on the gear is used to select the cutter, the teeth will be too narrow. The cutter selection is based on the number of teeth calculated by dividing the number of teeth actually on the gear by the cosine of the helix angle cubed.

$$N' = \frac{N}{(\cos A)^3}$$

Exact lead (L). The lead of the helix. The exact lead is used to determine the end gearing setup used to rotate the gear blank. Exact lead equals pi times the pitch diameter times the tangent of the helix angle.

$$L = 3.1416 \times D \times \cos A$$

NOTE: The following terms have the same

meaning as in spur gearing, but because of the helix angle, the formulas are different:

Pitch diameter (D). $D = \dfrac{N}{PN \times \cos A}$

Addendum (a). $a = \dfrac{1}{PN}$

Thickness of tooth (T) $T = \dfrac{pn}{2}$

Whole depth of tooth (W) $W = \dfrac{2.157}{PN}$

Outside diameter (OD). $OD = D + 2a$

12-10. The calculations which are necessary to machine a helical gear that has 20 teeth, a normal diametral pitch of 10, and a 22 1/2° helix angle are as follows:

Number of teeth (N) (given) 20
Normal diametral pitch (PN) (given) 10
Helix angle (A) (given) 22 1/2°

Pitch diameter (D) $D = \dfrac{N}{PN \times \cos A}$

$\dfrac{20}{10 \times 0.92388} = 2.1648$ inches

Addendum (a) $a = \dfrac{1}{PN} = \dfrac{1}{10} = 0.100$ inch

Outside diameter (OD) $OD = D + 2a = 2.1648 + 0.200 = 2.3648$ inches

Tooth thickness (T) $T = \dfrac{pn}{2} = \dfrac{0.3141}{2} = 0.15705$ inch

Whole depth of tooth (W) $= \dfrac{2.157}{PN} = \dfrac{2.157}{10} = 0.2157$

Circular pitch (p) $p = \dfrac{D \times 3.1416}{N} = \dfrac{2.1648 \times 3.1416}{20} = 0.340$ inch

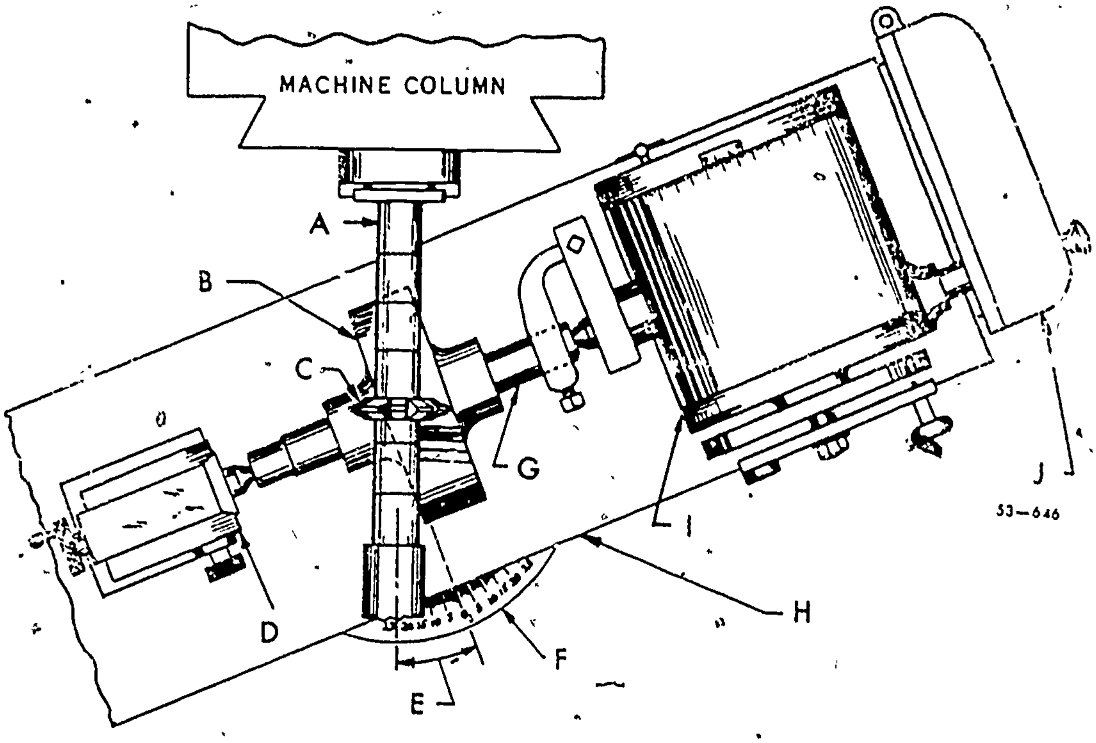
Normal circular pitch (pn) $pn = p \times \cos A = 0.340 \times 0.92388 = 0.3141$ inch

Number of teeth to base cutter selection on (N') $N = \dfrac{N}{(\cos A)^3} = \dfrac{20}{(0.92338)^3} = \dfrac{20}{0.78858} = 25$ teeth

Exact lead (L) $L = 3.1416 \times D \times \cos A = 3.1416 \times 2.1648 \times 2.41421 = 16.4188$ inches

12-11. You can mill the teeth on a helical gear in the following manner. Perform the necessary calculations for the helical gear and cutter. Make the necessary calculations to determine the end gearing required to obtain the correct lead. Set up the machine in the same manner as for milling helical reamer flutes. Make the work and cutter setup in the same manner as for machining spur gear teeth. Figure 62 shows the work, machine, and cutter setup required to machine helical gear teeth. Machine the gear teeth to the required size by combining the spur gear and helical fluting methods. Always lower the table before returning the gear blank to the starting point of the cut to prevent backlash from causing the cutter to drag on the gear teeth. Additional information pertaining to gearing can be obtained from machinists' reference sources, such as *Machinery's Handbook*.

12-12. **Bevel Gears.** It has been found that motion can be transmitted between two parallel shafts by friction between wheels mounted on the shafts. Then, if teeth of the correct shape are cut on these wheels, a positive uniform motion can be transmitted. These wheels with teeth are called spur gears. Motion can also be transmitted between two shafts that are at an angle to each other by the friction of two rolling cones. If teeth of the correct shape are cut on the faces of the cones, a positive uniform motion can be transmitted. The two cones become a pair of bevel gears. The machining of bevel gears involves problems and terms that are not found in spur gearing. We will discuss these problems and terms and the calculations that are necessary to machine bevel gears. Bevel gears may be mounted on shafts that are at any desired angle. However, the most common use of bevel gears is to transmit motion between shafts that are at right angles to each other. If the bevel gears are mounted on two shafts at a right angle to each other and the gears are of the same size, they are called miter gears. If one gear is larger than the other, the larger is called the gear and the smaller of the two is called the pinion. We will use a miter gear that is mounted on a shaft at a right angle to another shaft as an example for our calculations. For other angles, nomenclature, and symbols which are used for bevel gears, refer to machinists' publications, such as *Machinery's Handbook*. A knowledge of the nomenclature and symbols, figure 63, used for bevel gears will help you to understand the calculations. Let's assume that you are manufacturing a pair of miter gears having 16



A. Arbor
B. Helical gear blank
C Gear cutter
D. Foot stock
E. Helix angle
F. Table graduations
G. Mandrel
H. Machine table
I Index head
J. Gear train box.

Figure 62. Helical gearing setup.

teeth, a 5 diametral pitch, and a pitch cone angle of 45°. The gears are to be mounted on shafts that are 90° to each other. You could calculate for the dimensional values of these gears by using the following formulas. You should use these formulas in the order they are given, because the values are progressive from one problem to the next:

$$\text{Diametral pitch (P)} = 5$$

$$\text{Number of teeth (N)} = 16$$

$$\text{Pitch cone angle} = 45^\circ$$

$$\text{Pitch diameter (D)} = \frac{N}{P} = \frac{16}{5} = 3.200 \text{ inches}$$

$$\text{Addendum at large end of tooth (al)} = \frac{1}{P} = \frac{1}{5} = 0.200 \text{ inch}$$

$$\text{Clearance (c)} = \frac{0.157}{P} = \frac{0.157}{5} = 0.0314 \text{ inch}$$

$$\text{The whole depth of tooth (W)} = \frac{2.157}{P} = 0.2314 \text{ inch}$$

$$\text{Dedendum at large end of tooth (dl)} = \text{al} + \text{c or } \frac{1.157}{P} = 0.4314 \text{ inch}$$

$$\text{Thickness of tooth at pitch line (T)} = \frac{1.571}{P} = 0.3142 \text{ inch}$$

$$\text{The pitch cone radius (C)} = \frac{D}{2 \times \sin a} = \frac{3.200}{2 \times \sin 45^\circ} = \frac{3.200}{2 \times 0.707} = 2.263 \text{ inches}$$

$$\text{Width of face for gears up to 3 inches pitch diameter (F)} = \frac{C}{3} = \frac{2.263}{3} = 0.754 \text{ inch}$$

$$\text{Width of face for gears 3 inches to 20 inches in pitch diameter (F)} = \frac{C}{4} = \frac{2.263}{4} = 0.5657 \text{ inch}$$

116

$$\text{Addendum at small end of tooth (as)} = \text{al} \times \frac{C - F}{C} = 0.200 \times \frac{2.263 - 0.5657}{2.263} = 0.150 \text{ inch}$$

$$\text{Thickness of tooth at small end (t)} = T \times \frac{C - F}{C} = 0.314 \times \frac{2.263 - 0.5657}{2.263} = 0.2355 \text{ inch}$$

$$\text{Tangent of the addendum angle} = \frac{\text{al}}{C} = \frac{0.200}{2.263} = 0.0883 = 5° \ 3'$$

NOTE. You will need a table of trigonometric functions for conversion to degrees

$$\text{Tangent of the dedendum angle} = \frac{\text{al} + c'}{C} = \frac{0.2314}{2.263} = 0.10225 = 5° \ 50'$$

$$\text{Face angle} = \text{pitch cone angle} + \text{addendum angle} = 45° + 5° \ 3' = 50° \ 3'$$

$$\text{Cutting angle} = \text{pitch cone angle} - \text{dedendum angle} = 45° - 5° \ 50' = 39° \ 10'$$

$$\text{Angular addendum (K)} = \text{al} \times \cos \text{pitch cone angle} = 0.200 \times 0.707 = 0.1414$$

$$\text{Outside diameter (OD)} = D + 2K = 3.200 = 0.2828 = 3.482 \text{ inches}$$

$$\text{Number of teeth used to select cutter (N')} = \frac{N}{\cos \text{ of pitch cone angle}} = \frac{16}{0.707} = 22.6$$

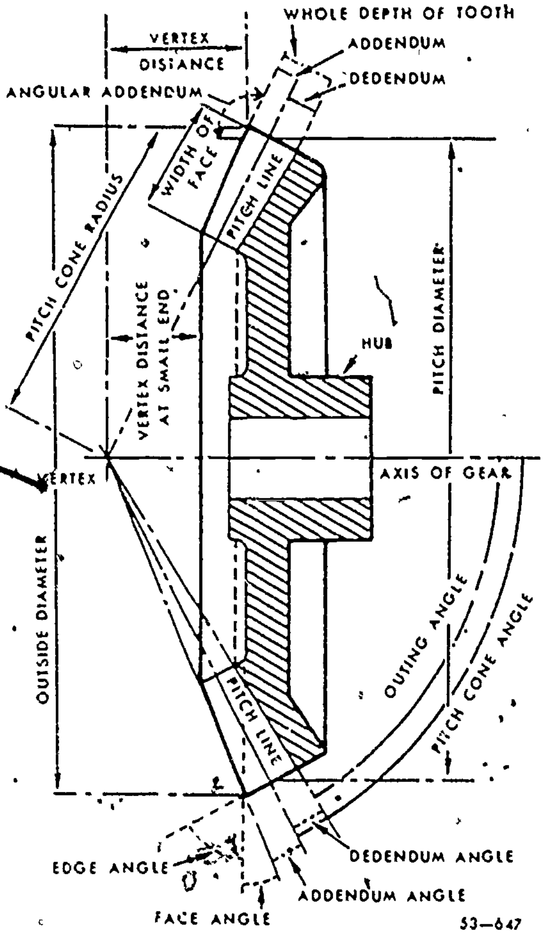


Figure 63 Bevel gear nomenclature.

12-13. The cutter selected is for the outer end of the gear teeth, but the small end of the gear teeth has a curvature which is too straight. This can be compensated for by filing the teeth at the small end above the pitch line, as shown in figure 64. The selected cutter will produce the correct width of tooth space at the small end of the teeth, but will leave the teeth too thick at the large end. Trim the sides of the teeth to correct this error by setting the gear off center and rotating the gear blank until the desired tooth thickness is obtained, as shown in figure 65. Using a table of setover factors, which can be found in machinist publications, such as *Machinery's Handbook*, calculate the amount of setover required. To select the factor from the table, you must first determine the ratio of the pitch cone radius to the width of the tooth face. The setover factor listed in the table at the intersection of the column corresponding to the number of the gear cutter you are using and the column nearest the ratio you have calculated is the factor used. Use the formula

$$\text{Table setover} = \frac{\text{thickness of cutter}}{2} - \frac{\text{factor}}{\text{diametral pitch}}$$

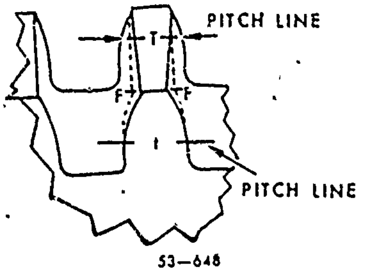


Figure 64 Bevel gear tooth.

117

to determine the amount of setover required. The thickness of the cutter is measured at the pitch line and must *actually be measured.* Otherwise, identical cutters may have different thicknesses due to sharpening, slightly different clearance angles, etc. The setover for a 5-pitch, 16-tooth gear with a 2.263-inch pitch cone radius and a width of face of 0.5657 inches that is to be cut with a number 5 cutter is calculated in the following manner:

$$\text{Ratio} = \frac{2.263}{0.5657} = \frac{4}{1}$$

The setover factor corresponding to a 4 to 1 ratio and a number 5 cutter is 0.295. The measured thickness of the cutter selected at the pitch line is 0.2317 inch. Using these values, you would calculate the setover as follows:

$$\text{Setover} = \frac{0.2317}{2} - \frac{0.295}{5} =$$

$$0.116 - 0.059 = 0.057 \text{ inch}$$

Thus, the setover required would be 0.057 inch.

12-14. The actual setup and machining of a bevel gear is a very simple operation. The cutter is mounted on an arbor and the gear blank is held in the index head by a chuck. The chuck jaws usually grip the gear blank on the gear hub but in some cases when the blank has a hole through it a special arbor is made to hold the gear blank in place. After the center of the cutter and the center of the gear blank are brought into line, the index head is tilted to the calculated cutting angle and locked into position. At this point, pick up the surface of the gear blank with the cutter. Then gash all the teeth to their full depth of tooth. Once the correct number of teeth have been cut to their full depth, the next operation is to offset and rotate the gear blank, as shown in figure 65. (NOTE. The direction of offset is always opposite the direction of rotation.) After the offset and rotation have been completed, trim the one side of all the gear teeth, then repeat this for the opposite side of the teeth. The final operation is to hand file all the teeth of the gear above the pitch line, as shown in figure 64.

13. Milling Machine Maintenance

13-1. Proper care and maintenance are probably the two most important factors in the use of the milling machine. These in turn will enable you to get the maximum efficiency from the machine. Certain periodic adjustments must be made to retain the accuracy permitted by the construction of the machine. Periodic changes of oil in the reservoirs of the column and knee must be made. Spindle bearings, gibs, clutches, and feed screws must be kept in adjustment if the machine is to function accurately and efficiently.

13-2. **Installation and Leveling of the Machine.** Occasionally a milling machine is moved in a shop or a new one installed. If it is to be placed in an area where there are other machines, a space check should first be made before it is permanently placed. Usually a plan dimension drawing is furnished with the machine. The plan lists the dimensions for the amount of space needed. It is best to allow ample space over and above the exact dimensions of the machine. The longitudinal table movement is variable on a milling machine and enough space should be allowed for its movement without interfering with the machine

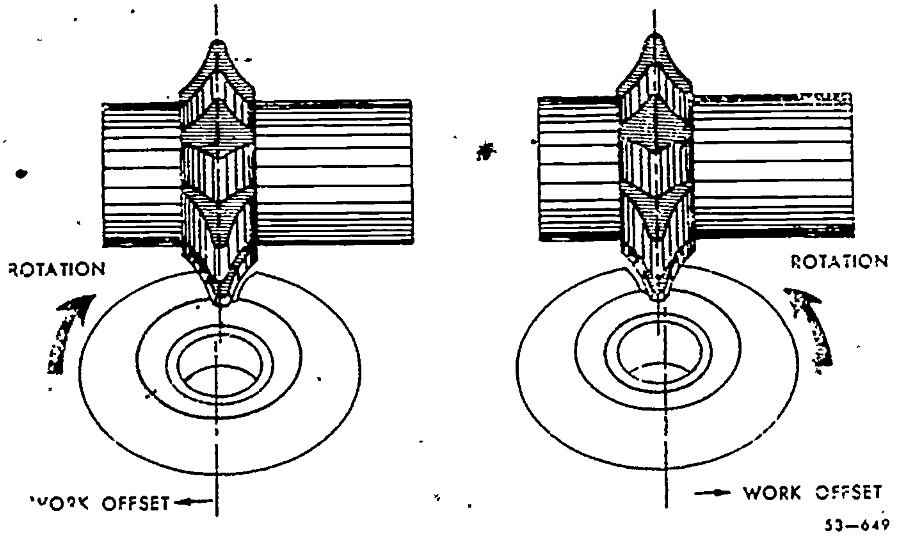


Figure 65. Offset and rotation of bevel gear blank

123

next to it. Another factor to consider is the floor on which the mill is set. It should be installed on a level floor. Hardwood wedges, shingles, or shim stock may be used in leveling the machine. The machine is set level by means of a precision level placed on the machine table.

13-3. Alinement of Table, Knee, and Column. The alinement and accuracy of the table, knee, and column surfaces can be checked by mounting an arbor in the machine spindle and attaching a dial test indicator to the arbor so that the indicator plunger bears on the table. Checks are taken by moving the table lengthwise and crosswise and noting any variation on the indicator dial. These checks should be taken with the knee set at various heights. Misalinement is corrected by adjusting gibs, rescraping the bearing surfaces of the table, knee, and column or refinishing the top of the work table.

13-4. Gib Adjustment. Gibs are usually of the headless taper type and are provided with adjusting screws. To adjust the gibs of the table, saddle, or knee, the screw is loosened at the small end of the gib and tightened by the adjusting screw at the large end There is another type of tapered gib used on some machines The only difference between this gib and the one described above is that it only has one adjusting screw on the large end. The set screw and gib end are designed so that the screw moves the gib in or out to tighten or loosen the gib. Correct adjustment is made when the slides move snugly by hand Proper adjustment is of vital importance. Adjust the gibs too tight and the slides may become scored Leave them too loose and they cause chatter and vibration, as well as undue wear on the machine ways. This condition soon results in an inaccurate machine.

13-5. Adjustment of Spindle Bearings. Spindle bearings on the newer type of milling machines are the nonfriction or taper roller bearing type. They are properly adjusted at the factory and need not be adjusted for long periods of time. If it ever becomes necessary to adjust spindle bearings, place the spindle gearing in neutral position so that the spindle can be rotated freely by hand. The various makes of machines may have the thrust bearing located at a different place along the spindle. One machine may have the thrust bearing behind the rear bearing or it may be located behind the front bearing. Regardless of where it is located the adjustment is the same. The locking set screw or nut is first loosened and the thrust nut tightened just enough so that the slack is removed. With proper adjustment the spindle can still be turned by hand without too much effort. The check may accurately be made by chucking a rod or bar in the spindle and using it as a lever to move the spindle back and forth along its axis. If a dial test indicator is used with the plunger of the indicator placed against the nose of the spindle, only about .003 inch end play should be permitted.

13-6. Checking the Accuracy of the Spindle. The spindle may be tested by placing a test arbor in the spindle. A dial test indicator is then set on the table with the indicator plunger contacting the circumference of the arbor near its outer end. With the spindle rotating, the table and indicator are moved crosswise and any variation on the indicator needle is noted. If the spindle is inaccurate, the spindle should be removed and the tapered hole reground to restore it to its original accuracy.

13-7. Driving Clutch Adjustment. Several milling machine driving clutches are of the plate disc type. They may be the single plate or multiple plate type or they may be the dry disc type or they may operate in oil. The drive is similar in operation in that all depend on friction. If the clutch slips under a normal load it should be adjusted as soon as possible to prevent excessive wear. When the clutch slips it means that the plates do not come together tight enough when the clutch is engaged. To adjust the clutch, the plunger lock is pulled out and turned in the right direction to tighten. This direction is usually clockwise, or to the right. Proper adjustment is made when full engagement of the starting lever can be accomplished and when the clutch cone contacts the fingers to compress the driveplates. The main object is not to set the clutch too tight. A great amount of pressure in engaging the starting lever places strain on the clutch fingers and may cause them to break.

13-8. Adjustment of the Table Feed Screw. The table feed screw is mounted in brackets at each end of the table. The right-hand bracket is provided with two antifriction bearings and an adjusting collar nut for taking up looseness in the bearings. To adjust, remove the cover from the right-hand end of the screw and turn the adjusting collar nut clockwise until one of the ears on the lockwasher of the collar nut slips into place in one of the notches of the adjusting nut.

13-9. Adjustment of the Crossfeed Screw Bearings. To adjust the bearing of the crossfeed screw, remove the hand wheel, power feed lever, and graduated dial. A locknut and thrust nut are then exposed. Loosen the locknut and turn the thrust nut to the right. Replace the locknut after the adjustment has been made. On the vertical feed screw, no provision has been made for the adjustment. The reason for this is that the weight of the knee prevents "back lash" between the screw and its nut and driving members.

13-10. **Periodic Oil Changes.** If the machine is run daily, the oil of the column and the reservoir should be changed about every 4 months. A good grade of machine oil SAE 20 to [illegible] should be used. Proper attention in keeping a milling machine cleaned and oiled increases the accuracy and efficiency of the machine and prolongs its life. Neglect may ruin it.

13-11. Working parts exposed to dust, dirt, and chips should be frequently cleaned and oiled. Chips should not be allowed to collect upon the surface of the table until they fall over the sides on the flat bearing surfaces on top of the knee. Care should also be taken to prevent chips and dirt from getting between the column and knee, causing scoring of these flat surfaces and throwing the knee out of alinement. Oil tubes and channels may at times become clogged due to the accumulation of dirt and oil. This dirt and oil can be removed without damage to the bearing surfaces by flushing the tubes and channels with a flushing oil.

13-12. Oil completely before starting the machine. Many machines of recent design are equipped with automatic lubrication systems which insure a constant supply of lubricant at important points. Check all oil gages to see that the gages of the column, knee, and saddle reservoirs show oil at a safe level. If the oil in the reservoir is low, the oil pump will not work properly. Too much oil will cause leaking and impair working conditions. Usually there will be several points on a milling machine that are hand oiled. It is well to remember in applying oil that an ordinary bearing can hold only a certain amount of oil at a time and that this amount applied at regular and frequent intervals is far more beneficial than a flood of oil applied at irregular intervals. To prevent rust from accumulating on the working and machine surfaces of the milling machine, keep a light film of oil on these surfaces at all times when the machine is idle.

120

Shaper Work

ARE ALL manufactured items cylindrical in shape? Of course not. The shape of many items is rectangular, square, 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, and since you will be machining objects with flat surfaces, we will discuss the machining of flat and angular surfaces, the machining of shoulders, corners, and grooves, the machining of irregular shapes, and the maintenance of the shaper

14. Flat and Angular Planing

14-1. In this section we will discuss the planing of flat and angular surfaces. The term "planing" refers to machining operations done on planers and shapers. The production of parallel and square surfaces is a fundamental planing operation and may best be done by following the proper procedures. When planing parallel and square surfaces it is also important to have the correct setup. Among the numerous operations that you may perform on a shaper is angular planing. Various attachments may be used to produce an angular surface.

14-2. **Flat Planing.** Flat surfaces are machined on a shaper by feeding the work below the ram or by feeding the tool past the work by using the toolslide. The direction of the feed is usually from left to right in machining horizontal surfaces and from top to bottom in machining vertical surfaces. By using these directions of feed you have an unobstructed view of the work surface and the cutting action of the tool.

14-3. A roughing tool and roughing feed should be used to remove excess stock. A finish cut should be taken with the roughing tool, using a finish feed. This should aid in the removal of taper or spring caused by the heavy cut. NOTE. Use the cutting lubricant that is recommended for the metal you are machining during all cutting operations. You can apply the lubricant to the 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.

14-4. You can machine surfaces that are perpendicular to each other by machining them horizontally in the proper sequence. The following information will help you to understand how to machine a parallel so that the opposite surfaces are parallel and the adjacent surfaces are perpendicular. Figure 66 shows the position of the work as the various sides are machined.

14-5. Obtain the proper size and type of material for the job. Prepare the machine by oiling the necessary points. Then aline the stationary jaw and the bottom of the vise so that they are parallel to the travel of the ram. Mount the work in the vise on parallels of the correct size. The work must extend above the jaws far enough to permit the removal of the necessary material. If the work has a rough surface, use shim stock between the work and both vise jaws and between the work and the parallels. This protects the vise jaws and the parallels from the rough material and also gives the vise greater holding power. Smooth or machined surfaces require shim stock only against the movable vise jaw to permit the work to seat on the parallels and also to prevent damage to the finished surface. To properly 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.

14-6. The top of the clapper box should be tilted in the opposite direction to that which the tool is to cut, except when a roundnose tool is used. Then the clapper box should be perpendicular to the work surface. Mount the toolholder and tool. Calculate and set the proper length of stroke and position the ram properly in relation to the work. Then calculate the amount of material to be removed from one side. Be sure to leave enough material for a

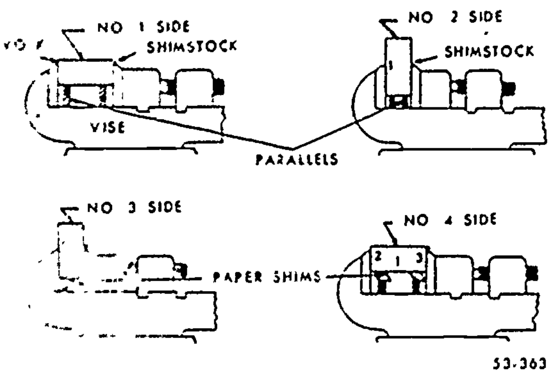


Figure 66 Squaring a block

finishing cut. Set the shaper for the proper speed for the material which you are machining. Set the feed for the roughing cut and rough machine the first side of the work. After roughing the first side, replace the roughing tool with a shear tool for finishing. The roughing tool can be used if an exceptionally good finish is not required. Set the shaper for the finishing speed and feed and finish machine the first side.

14-7. The second side is machined much the same as the first. Place the work in the vise so that side one contacts the solid jaw of the vise. This positions side two in an upright position for machining. Calculate the amount of material to be removed and repeat the roughing and finishing operations the same as for side one.

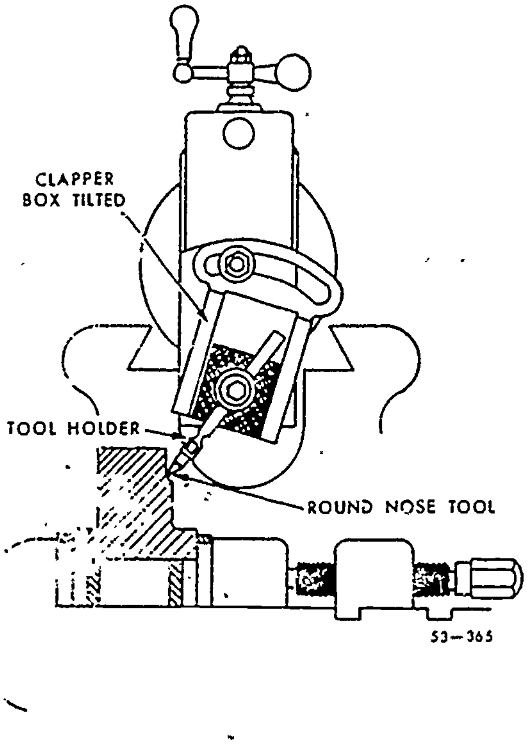


Figure 67 Tilting clapper box for down cutting

14-8. The work piece now has two sides which are square or perpendicular to one another. To machine side three parallel with side two and perpendicular to side one, place the work in the vise so that side one contacts the solid jaw of the vise and side two rests on parallels. Paper shims should be used to ensure that the work seats properly. Rough and finish machine side three so that the distance between side two and side three is correct. For side four, place the work in the vise with side one resting on the parallels and side two contacting the stationary jaw of the vise. Rough and finish machine side four so that the distance between side one and side four is correct.

14-9. When you machine *vertical surfaces*, such as the vertical surfaces of shoulders or the ends of work, position the toolhead perpendicular to the table or vise. Position the vise either perpendicular to or parallel to the travel of the ram, depending upon the location of the surface 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 test indicator. 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 67. 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 machine 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.

14-10. **Angular Planing.** You can machine angular surfaces by (1) swiveling the vise, (2) swiveling the toolhead, (3) swiveling or tilting the table, or (4) by mounting the work either on an adjustable angle plate or on a fixture.

14-11. *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. Then feed the tool vertically by means of the toolslide handcrank. Use the same tooling setup that you use to machine vertical surfaces.

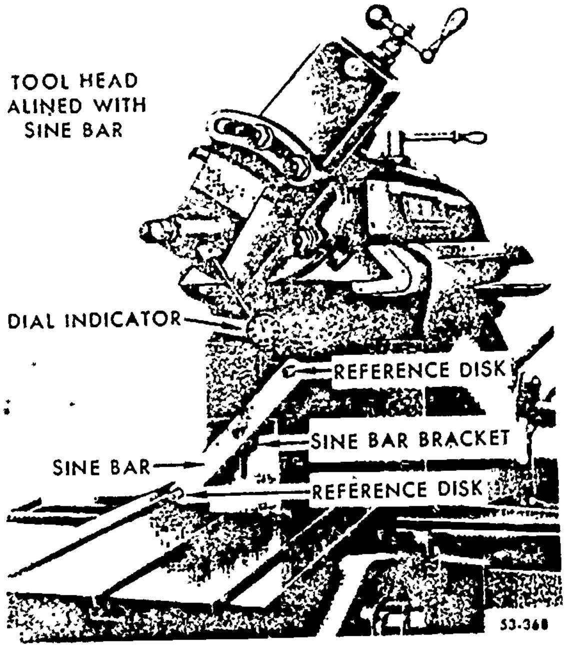


Figure 68. Alining toolhead with a sine bar

14-12. *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 feed the tool by hand by means of the toolhead crank. Use the toolhead graduations for angles requiring average accuracy. Aline the toolslide with a dial test indicator and a sine bar, as shown in figure 68, when you are machining angles that must be highly accurate.

14-13 A sine bar is an extremely accurate bar of metal that has two cylindrical reference discs attached to it. The centers of the reference discs are either exactly 5 or 10 inches apart. You can position the sine bar at any desired angle by elevating one end of the sine bar and placing the reference disc on a stack of gage blocks of the correct height. Gage blocks are extremely accurate blocks of metal of varying thicknesses. You can clamp the sine bar to a support bracket at the position you desire so that you can aline the toolhead parallel to it by means of a dial test indicator You can find the height of the gage blocks necessary to obtain a given angle (in minutes of a degree) in machinists' publications, such as *Machinery's Handbook*

14-14 *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 tabletop Use the graduations on the table for average accuracy, or a sine bar, as shown in figure 69, for extreme accuracy. Set up the tool and feed the work in the same manner that you use to machine a horizontal surface.

14-15. *Using an adjustable angle plate.* You can mount work on an adjustable angle plate in order to machine the work at the angle, or combination of angles, that you desire. You can machine angles using an adjustable angle plate. They will be the same as those you can machine by swiveling the vise, swiveling the table, or by tilting the tabletop. However, you should use the adjustable angle plate for light-duty applications only.

14-16. *Using a fixture.* You can mount work on a fixture, as shown in figure 70, in order to machine an angular surface in the same manner that 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.

15. Shoulders, Corners, and Grooves

15-1. You will be required to machine shoulders, corners, and grooves. In this section we will discuss the machining of shoulders corners, and grooves in the shaper.

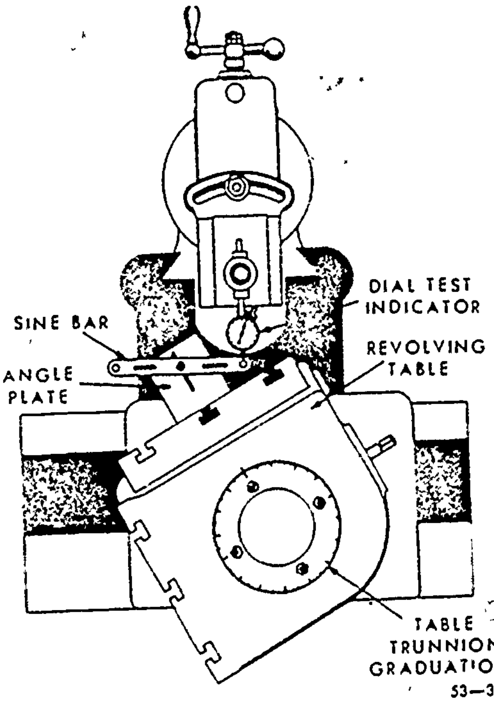


Figure 69 Alining table with a sine bar

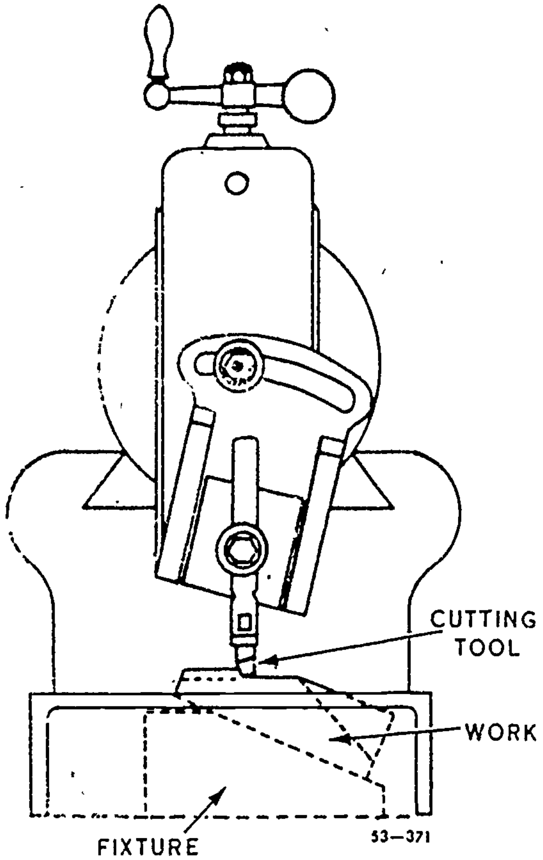


Figure 70. Angular planing with a fixture.

15-2. **Shoulders and Corners.** Shoulders and corners are machined by removing excess material with a roundnose tool and then finishing with a side-finishing tool or a squaring tool.

15-3. 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 both directions. This will save time, since you will not have 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 toolholder set vertically. Set up the tool, as shown in figure 71, when you are machining 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.

15-4. The following information will help you to understand how a square shoulder, such as the one on the forming die, shown in figure 71 is roughed out. Mount a roundnose tool in a straight toolholder. Set the machine for the length and position of stroke and the correct speed. Check to insure that the ram and toolslide will clear the work and vise. Pick up the cut with a piece of paper and set the toolslide dial to zero. Make the necessary calculations to determine the depth from the top of the work to the bottom layout line. Begin the cut approximately 1/32 inch from the vertical layout line. Use a roughing feed. You can use the power feed to machine within 1/8 inch of the layout line and then feed by hand to within 1/32 inch of the line. Move the work back to the starting point of the cut and set the toolslide for an additional depth of cut. Engage the power feed and allow the table to feed until the tool is within 1/8 inch of the shoulder. Feed the work by hand until the tool is within 1/64 inch of the preceding cut. (CAUTION: Be sure that the toolholder clears the work as the depth of the shoulder increases.) Repeat the operations until you obtain the depth you desire. NOTE: You can measure the height of the shoulder with a depth micrometer but 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.

15-5. Swing the clapper box away from the shoulder and position the tool at a 30° to 40° angle with the shoulder. Pick up the vertical surface of the shoulder and set the crossfeed graduated collar at zero. Position the work and

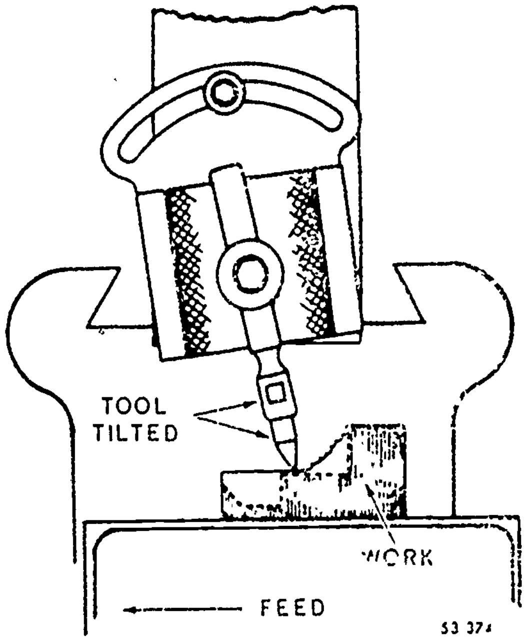


Figure 71 Roughing excess to one shoulder

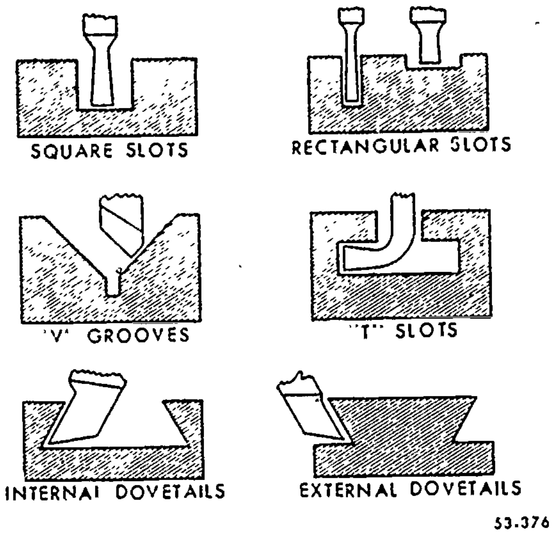


Figure 72 Slots and grooves.

the tool bit with the crossfeed and toolslide so that the tool bit contacts the horizontal surface of the work. Then, set the toolslide graduated collar at zero. Rough out the stepped material that is adjacent to the shoulder to within 0.005 inch of the vertical layout line. (NOTE: 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 with the crossfeed, and move the tool bit downward with the toolslide for the feed.) Feed the tool downward to within 0.005 inch of the horizontal line. Feed the work away from the tool bit by engaging the power feed.

15-6. 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 bit is not objectionable and the finish produced by the roundnose tool is acceptable, you can machine the shoulder to the finished dimensions. If you require a better finish and a square shoulder, you can machine the vertical surface with a side finishing tool and the horizontal surface with a squaring tool or both the vertical and horizontal surfaces with a squaring tool. You obtain the best finish when you use both the side-finishing tool and the squaring tool.

15-7. *Grooves*. You can machine small slots and grooves with form tools, as shown in figure 72, or you can rough out large slots and grooves with a roundnose tool and finish machine them with finishing tools, as shown in figure 73. Use square-nose tools to finish square and rectangular slots and 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. Grind shaper parting tools identical to the squaring tool 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.

15-8. *Keyways*. A square or rectangular keyway is in reality nothing more than an external or internal slot or groove that is parallel to the axis of a shaft or hole and which is fitted with a key. The terms "keyway" and "keyseat" are often used interchangeably. The cutting tool that you use to machine a keyway is similar to the square-nose tool, 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. Prior to machining a keyway, you should accurately lay out its location, width, depth, and centerline. Extend the centerline of the keyway down the end and through the axis of the work. 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 keyways can be machined with a shaper. First, we will discuss the information you will need to

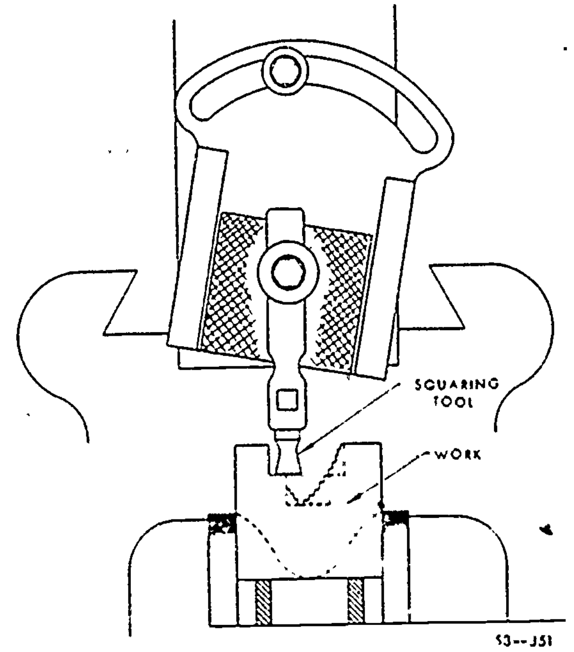


Figure 73 Machining a wide groove.

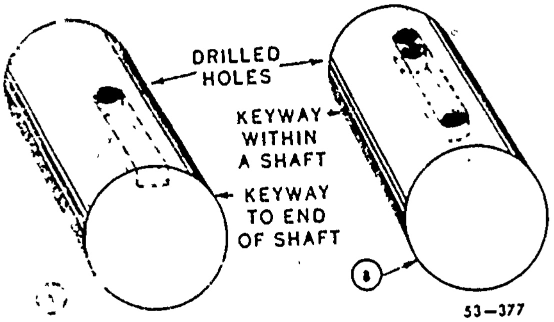


Figure 74. Drilled holes for keyseats.

machine external square keyways and then the information pertaining to internal square keyways.

15-9. *External keyways.* If the keyway you are to machine does not extend the full length of the shaft, drill a hole at the point where the keyway will terminate, as shown in figure 74,A. The diameter of the drill should be equal to the width of the keyway. The depth of the drilled hole, excluding the conical point of the drill, should be equal to the depth of the keyway. The hole prevents chips from building up in front of the cutting tool and permits machining the keyway to its full length. If both ends of the keyway you are to machine terminate on the shaft, drill holes at both ends of the keyway, as shown in figure 74,B. Drill two adjacent holes on the end of the keyway where the cut will originate, as shown in figure 75. Remove the metal between the holes with an end mill or by chiseling and filing. The elongated hole will permit the cutting tool to drop into position. Grind away the back portion of the tool bit, as shown in figure 75, 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!* Measure the depth of the keyway 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 keyways for shafts of various diameters in machinists' publications, such as *Machinery's Handbook.*

15-10. You can machine an external keyway in the following manner: Lay out the keyway, extending the centerline over the end of the shaft. Drill holes at the ends of the keyway. Mount the work on the shaper, alining the centerline with a machinist's square, as shown in figure 76. Position the clapper box in the vertical position and mount the cutting tool and toolholder, alining the cutting edge horizontally. Position the keyway under the cutting tool. Set the machine for the proper length of stroke and speed. Carefully position the stroke checking to insure that the tool will not overrun the drilled holes. Pick up the top surface of the shaft. Machine the keyway to the required depth. NOTE: The total depth of cut can be determined by the following formula:

$$D = \frac{W}{2} + t$$

when

D = depth of cut
W = width of key
t = height of arc

(The height of arc can be found in machinists' publications, such as *Machinery's Handbook.*) Insert a key of the proper size in the keyway and measure over the key and the diameter of the shaft to insure that the keyway is the required depth. NOTE: The micrometer reading over the key and the shaft can be determined by the following formula.

$$M = D + \frac{W}{2} - t$$

when

M = micrometer reading
D = diameter of shaft
W = width of key
t = height of arc

Deburr the keyway and remove the work from the shaper.

15-11. *Internal keyways.* Internal keyways are machined with a setup like the one shown in figure 77. Note that the clapper box is in the vertical position and locked to keep it from moving and that the tool is held in an extension toolholder and fed upward for the depth of cut. Internal keyways are machined in much the same manner as external keyways.

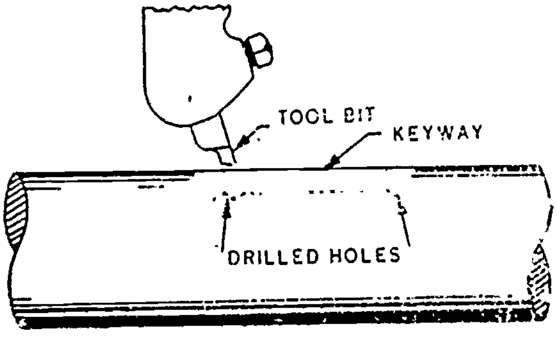


Figure 75. Keyseat slotting tool

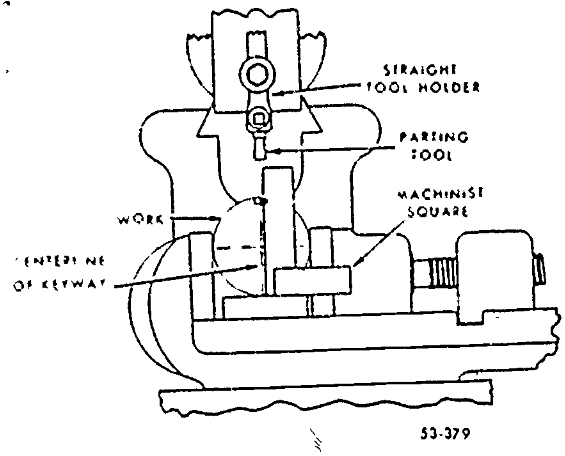


Figure 76 Alining work with a machinist square.

16. Irregular Shapes

16-1. In addition to straight 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 78.

16-2. You can machine contour 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 or by moving the work and tool bit. You normally use form tools to machine several identical items. If the contour is too large to be formed by one tool bit, you can grind tool bits 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 prior to using the form tools. Contours that are not practical to produce with form tools can be machined by moving the work and the tool bit simultaneously so that the tool bit 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.

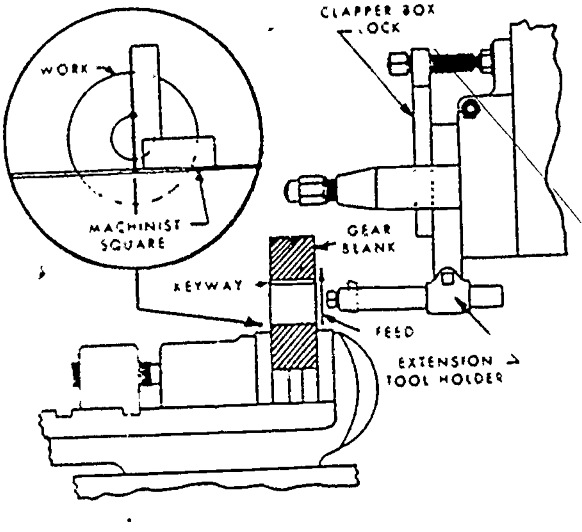


Figure 77 Alining gear and positioning extension toolholder

17. Shaper Maintenance

17-1. A shaper is usually shipped from the factory in a single crate or box and is normally almost entirely ready for use as soon as it is placed in position and properly lubricated. The machine is normally covered with a preservative, such as a heavy grease, to protect it from corrosion and rust. In this section we discuss (1) installation, (2) cleaning, (3) adjustments, (4) lubrication, and (5) troubleshooting.

17-2. **Installation.** A shaper should be installed on a concrete foundation. If this is not available or practical, it could be set on a solid wood floor and held down by heavy lag bolts. When you are installing a shaper you must pay particular attention to the amount of floor space needed. Ample room must be allowed for the ram travel, and there must be ample room for an operator to move around the ends and sides of the machine. The manufacturer usually furnishes a dimension plan for reference when you locate the machine in your shop.

17-3. **Cleaning.** Before operating a newly installed shaper, you must clean the preservative from the machine. Many types of preservatives are used, but the most common type is a heavy grease. You can remove the grease and most of the other preservatives with a cleaning solvent. You must exercise caution in the use of cleaning solvents as most of them are flammable and should be used only in well-ventilated areas.

17-4. **Adjustments.** To keep the shaper in the best operating condition, make periodic inspections and minor adjustments. The periodic inspections should include checking the levelness of the machine, condition of the clutch and brake assembly, and the adjustment of the gibs, belts, and the rail clamp.

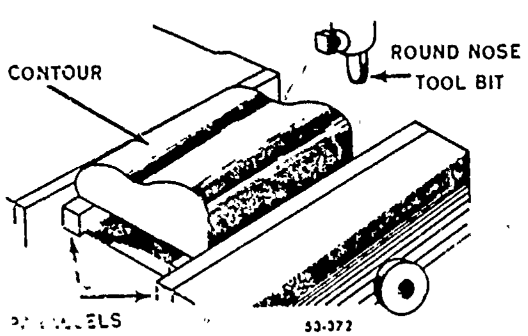


Figure 78. Machining a contour.

17-5. *Leveling*. The shaper is leveled with a precision level. Place the level crosswise on the table and lengthwise on the top of the column. The crossrail must be clamped to prevent it from moving. The machine should be level to within 0.001 of an inch per foot. The leveling is done by placing shims under the supporting points of the machine.

17-6. *Clutch and brake assembly*. After a long period of time, the clutch and brake assembly may require adjustment. If this adjustment ever becomes necessary, you should refer to the applicable technical order or the manufacturer's handbook on the care and maintenance of the specific machine on which you are working. The normal operational test for action is made with the ram stroking and the clutch engaged; a light push on the clutch lever should stop the machine automatically.

17-7. *Gibs*. Most shapers are gibbed throughout to maintain proper clearance between working parts. Correct adjustment of gibs is necessary for accurate and smooth operation, and also, to eliminate excessive chatter during cutting operations. Gibs are usually adjusted with a minimum of clearance; they should be drawn up snugly, and the adjusting screw backed off to provide a clearance of about 0.002 inch between the bearing surfaces.

17-8. *Belts*. The adjustment of the motor drive belts is usually not an involved operation. You can do it on most machines by loosening the nuts holding the motor base and turning an adjusting screw at the bottom of the base. Whether you want to tighten or loosen the belts determines the direction to turn the adjusting screw. The belts should be tight enough so that they do not slip under a normal load but not so tight that they stretch and possibly break.

[illegible] *Rail clamp*. To adjust the clamp on the right end of the rail, apply a wrench to the clamp shaft to loosen the mechanism and then traverse the table to the left end of the rail. Turn the adjusting screw tight against the toggle and back it off about one-half turn. Reverse this procedure for the clamp on the left end of the rail. You may have to make additional minor adjustments for the best clamping condition, depending upon the variation of limits and unequal wear on the mechanism.

17-10. **Lubrication.** Oiling the shaper is often neglected during high production. When the machine is operated continuously, oil it at least once each day. Proper attention to lubrication will greatly increase the life of the shaper and hold maintenance to a minimum. The oiling should be done progressively. Shapers of recent design are equipped with an automatic lubricating system which insures a constant supply of clean lubricant at important points. Check the oil gages to insure that the oil is at a safe level. If the oil is low in the reservoir, the oil pump will not operate properly. There are many points on the shaper that must be hand oiled. A small amount of oil applied at regular intervals is better than a flood of oil at irregular intervals. Periodic oil checks and changes are necessary for preserving the life of the machine, and thereafter once every 12 months per shift of operation. When you change the oil, consult the applicable technical order or the manufacturer's operator manual for the specific oil and the amount that should be used for the particular make or model of shaper that you have in your shop.

17-11. **Troubleshooting.** Machining troubles on the shaper are caused almost entirely by the setup. The work must be held solidly and securely while it is being machined. The following is a representative list of possible causes of trouble, some of which may even result in damage to the machine:

a. Tool loose in the toolholder.
b. Toolholder loose in the toolpost.
c. Work set too high, causing the ram to strike the work instead of passing over it
d. Work not securely clamped to the table
e. Improper use of jigs and fixtures.
f. Improper feed—excessive feed causes chatter or dulls and breaks the tool.
g. Incorrect speed—excessive speed overheats the tool, causing it to dull or break.
h. Changing gears with the machine running will probably cause the gears to lock.
i. Lack of lubrication will cause an early breakdown of any machine.
j. Allowing dirt and chips to accumulate causes excessive wear on ways and sliding surfaces of the machine.

CHAPTER 5

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 work it produces. The contour machine is primarily a metal-cutting bandsaw, but you can also be filing and polishing 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. In this chapter we will discuss angular sawing and filing; three-dimensional, stack, and difficult materials sawing; the attachments used on the contour machine; and maintenance of the contour machine.

18. Angular Sawing and Filing

18-1. Sawed surfaces are normally 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 angular clearance must be given to a punch and die. The table can be tilted forward or backward up to 10°, to the left up to 10°, and to the right up to 45°. Most contour machines have two sets of holes in the keeper block for the mounting of the lower saw guide. When an angle of 20° or less is to be sawed, the lower saw guide should be mounted in the upper set of holes in the keeper block. To saw an angle of more than 20°, use the lower set of holes. The lower set of holes is provided to ensure that the saw guide does not limit the tilting of the table. You tilt the table by loosening lock bolts located below the table and then set it to the desired angle. The settings are indicated on graduated plates located below the table. You may check the angular setting by measuring the angle formed by the post and the table top with a protractor head and blade. Tighten the table lock bolts after you have positioned the table to the desired angle. From this point on angular sawing is done in the same manner as straight and contour sawing. Feeds and speeds for angular sawing are the same as for straight and contour sawing. Both internal and external sawing can be performed.

18-2. Angular filing is done for the same purpose as for other filing operations. Selection of bands, speeds, and the setup procedures are the same as those for all filing operations. After the sawing operations, you file 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.

19. Three-Dimensional, Stack, and Difficult Material Sawing

19-1. Previously, we discussed the basic operations of straight and contour sawing. In this section we will discuss the more difficult operation employing straight and contour sawing, such as three-dimensional, stack, and difficult material sawing.

19-2. **Three-Dimensional Sawing.** Three-dimensional sawing, filing, or polishing are operations that produce two or more surfaces which may be at an angle to each other. You can produce many three-dimensional shapes more easily on the contour machine than on other machine tools. The reason is that you produce many shapes having radii and compound angles on other machines by a series of tooling setups on the surface of the work. On the contour machine you can cut many of the irregular shapes and angles directly. The most important factor in the sawing of a three-dimensional object is the layout prior to the actual cutting. Scribe the layout lines on two or more sides of the stock prior to cutting the irregular shapes. Use extreme care to make sure that the layout corresponds as to size and shape from one side to the other. You must decide the sequence of cuts necessary on each side, and the layout must correspond to this sequence. You must also decide which waste pieces can be sawed off immediately and which must be retained to support following cuts. When support and reference sides are needed, partial cuts can be made, and you can finish the cuts after the work has been completed to a point at which support and layout are no longer necessary. Surfaces that are at right angles to each other are produced by setting the work table at zero or perpendicular to the post and [illegible]. You can produce most other angular surfaces by tilting the table to the desired angle. The only limitation to cutting angles by tilting the table is the range of the table movement. If angular surfaces are required, the normal method is to complete one surface, such as sawing, filing, and polishing, before going to the next surface. You should use this method, unless it is not practical, because the position of the table for a given angle does not change. Finish one angular surface to specifications and then set the table for another angle.

19-3. **Sawing Difficult Materials.** Modern manufacturing processes have made it possible to produce many new metals, some of which are extremely difficult to machine or form into various shapes. Probably the most common of these metals are the nickel-alloyed stainless steels and the titanium alloys. These alloys are hard to cut with any machine tool, but cutting them with the contour machine is probably the most difficult because it uses a saw band as a cutting tool. The teeth of a saw band are generally quite small, very hard, and brittle. They are easily broken. A slight error in selecting or calculating feed and speed can very quickly ruin a band. When you are required to cut these alloys you should pay careful attention to the selection of the saw band, considering such factors as type of material, type of sawing operation, pitch of the teeth, set of the teeth, temper of the band, speed and feed applicable for the material, and the machine setup for the band. If the material to be cut is exceptionally hard, you can sometimes use a method, called friction sawing, to advantage.

19-4. *Friction sawing.* In this method of sawing, you use a fairly heavy feed pressure in conjunction with a high velocity of the saw band, from about 7000 SFS to about 14,000 SFS, depending upon the material being cut. For this type of cutting, the saw band does not need to be sharp. For most operations a dull saw band actually works best because the material ahead of the band during the cut must reach a high temperature. Under certain conditions it may reach [illegible]0° F. The cut is so quickly completed that the heat at the cut is dissipated quite rapidly. The saw band does not usually overheat because the velocity creates an air-cooling effect, except directly at the point of contact. With the correct band selection, velocity, and feed, you can cut hard alloyed materials up to 1 inch quite easily. Friction sawing should be done only on a machine capable of producing the high velocity needed—a heavy-duty type, which usually uses the roller-type saw guide insert. Never use this method to saw combustible materials.

19-5. *Sawing titanium.* Titanium is a difficult metal to machine, although it is not usually very hard. Titanium is a poor heat conductor, and the heat generated by a cutting action does not dissipate easily. Therefore, the heat is concentrated on the cutter face and edge. Titanium has a tendency to gall and weld, which rapidly destroys a cutting tool. Workpieces fabricated from titanium have a tendency to move away from the cutting tool, making a heavier feed necessary.

19-6 You cannot change the machining properties of titanium but you can minimize their effects by using the following simple rules:

- Use low cutting speeds.
- Maintain a high feed rate.
- Use a high volume of coolant
- Use sharp tools and keep them sharp.
- Never stop the feed while tool and work are in contact.

19-7. Titanium is one of the relatively few combustible metals. Under certain conditions fine titanium residue burns with intense heat. Using a liberal coolant flow and keeping your tools sharp can eliminate most of the danger. To further minimize the possibility of fire, you should practice good housekeeping. This should be standard procedure in any metalworking shop. If a fire should occur, use dry powders (such as dolomite, which is powdered limestone) that have been developed specifically for this type of fire. For maximum safety, containers of dry powder should be within easy reach of machinists working with titanium. CO_2 extinguishers and chlorinated hydrocarbon extinguishers are not recommended for titanium fires. NEVER APPLY WATER DIRECTLY TO A TITANIUM FIRE.

19-8. **Stack Sawing.** Stack sawing is the sawing of several pieces of sheet material of the same shape in one operation, as shown in figure 79. This method of duplicating identical parts saves time, especially when not too many pieces are needed and the job does not warrant the

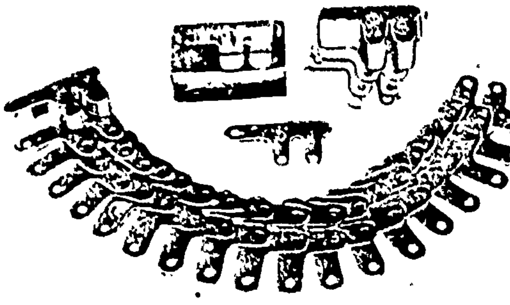


Figure 79 Stack sawed parts

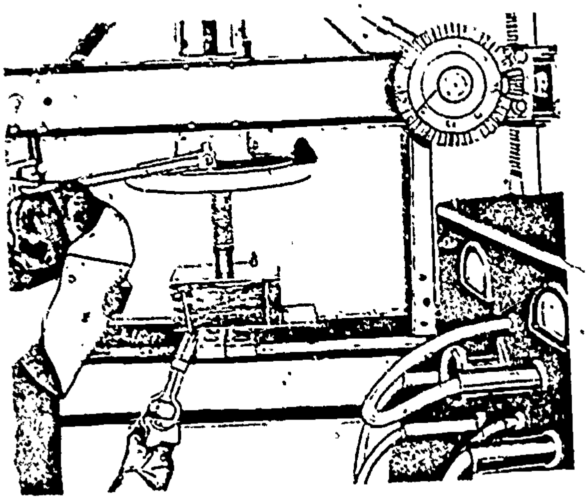


Figure 80 Compressing a stack of pieces in an arbor press and welding them together

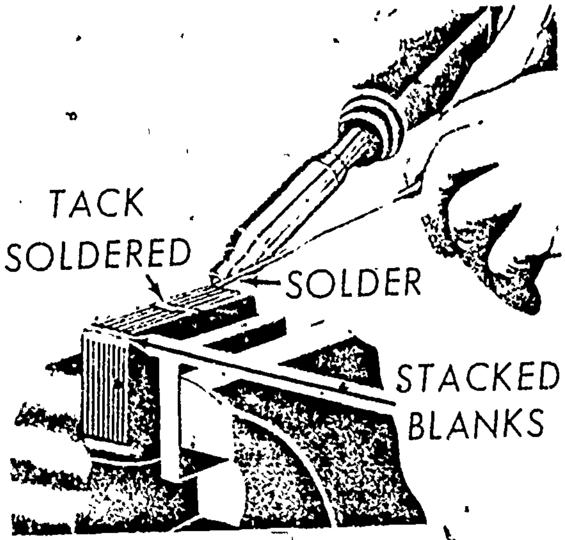


Figure 81 Compressing and soldering small objects

manufacture of a blanking die. The number of pieces that may be stack sawed is limited by the capacity of the machine. The capacity of the machine for sawing duplicate pieces is the same as for sawing solid material. This operation can be best applied to flat pieces of sheet stock since it is necessary to stack one piece on another. After the material has been stacked it must be fastened together to hold it during the sawing operation. For larger work, rather than relying on the weight of each piece for the close contact desired, it is best to place the stack in an arbor press, as shown in figure 80. Holes may be drilled on the waste section of a stack and the stack bolted or welded together to hold the work for the sawing operation. Small stacks for small parts can be placed in a vise and fastened together by soldering, as shown in figure 81. You must decide from the nature of the work which method is best suited for fastening the pieces together. If bolts are used, the heads may interfere and not allow the workpiece to lay flat on the machine table. The heads should be countersunk flush with the bottom piece of work to prevent chatter. (NOTE: Keep the thickness of the stack in proportion to the size of the base so that it will not become top heavy, difficult to handle, or dangerous to the operator.) The thickness of the stack should be no greater than the width.

20. Contour Machine Attachments

20-1. The use of various attachments will enable you to perform certain operations with greater ease and accuracy. In this section, we will discuss some of the attachments.

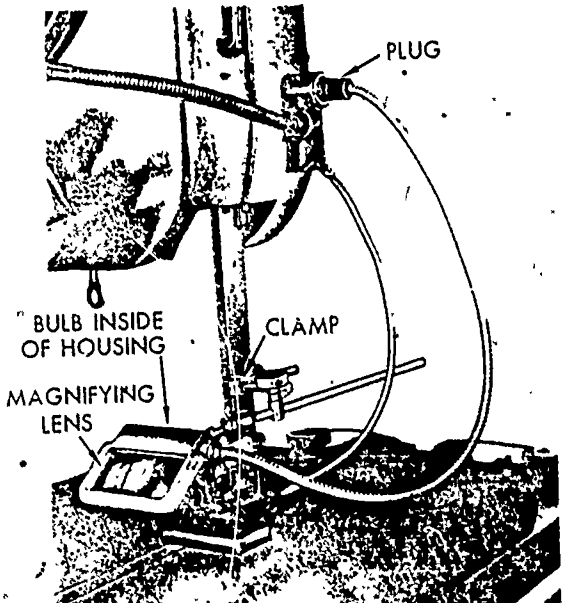


Figure 82. Magnifying attachment

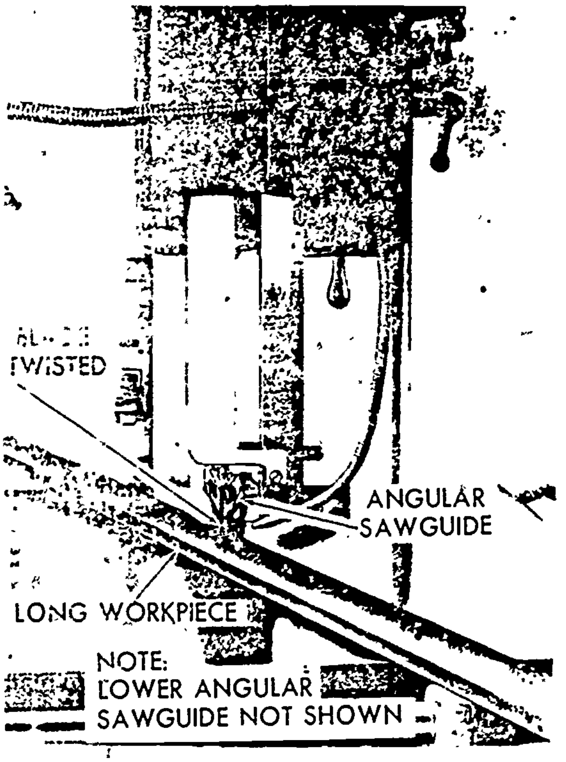


Figure 83. Angular saw guides.

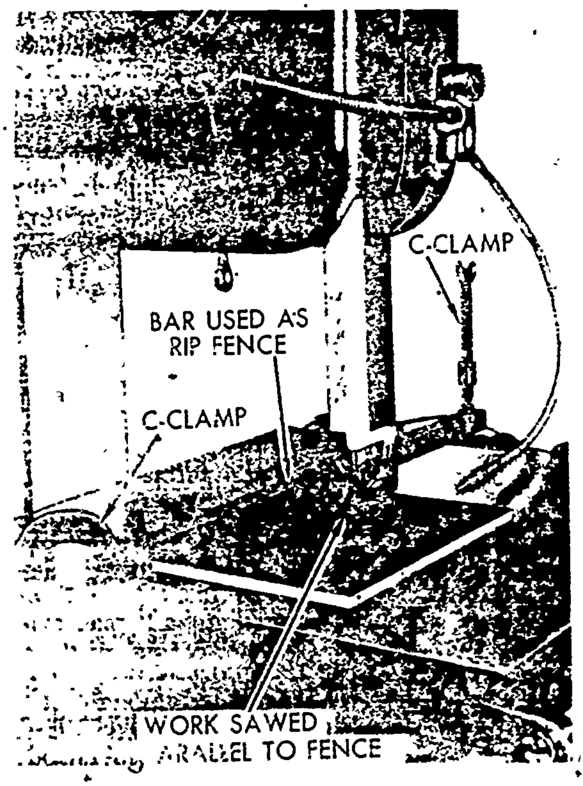


Figure 84. Improvised rip fence.

[illegible] **Magnifying Attachment.** The *magnifying attachment*, shown in figure 82, consists of a 5-inch rectangular lens mounted in [illegible] housing. The *housing* contains a light socket for a 15-watt candelabra-type 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 both sawing and filing. A *special plug connector* on the extension cord connects with the *outlet jack* located on the front of the machine above the table light outlet. This outlet is fused for [illegible] ampere; you should not use it for any other light extension where more than 15 watts will be consumed. You use the magnifying attachment when precision sawing and filing to close tolerances are required.

20-3. **Angular Saw Guides.** *Angular saw guides* twist the saw band to a 30° angle and allow work that would normally be too long [illegible] saw to be machined; for example, the bar shown in figure 83. The tension must be less than normal to permit the saw band to twist without causing the inserts to wear excessively.

20-4. **Rip Fence.** You use the *rip fence* to cut stock so that opposite sides are parallel. The fence must be set parallel to the table slot and located the desired distance from the saw band. A rip fence can be improvised by clamping a metal bar to the table, as shown in figure 84. Accurate results can only be obtained if the saw is properly set up and a sharp saw band is used.

20-5. **Cutoff and Mitering Attachment.** You use the *cutoff and mitering attachment*, shown in figure 85, for cutting off, mitering, and ripping 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 when identical lengths are sawed. The attachment is swung on the slide rod and allowed to hang below the tabletop when it is not in use.

20-6. **All-Purpose Mitering Attachment.** You can perform three operations with the *all-purpose mitering attachment* shown in figure 86. You can use it with hand or power feed for (1) ripping, (2) cutting off, or (3) 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 guide rail on the front edge of the table. The attachment has a graduated plate with an adjustable work stop on the mitering bar and a lockscrew on the miter head, which enables you to set the attachment at any desired angle.

20-7. **Disc-Cutting Attachment.** You can use the *disc-cutting attachment* to saw internal or external circles and dis[illegible]

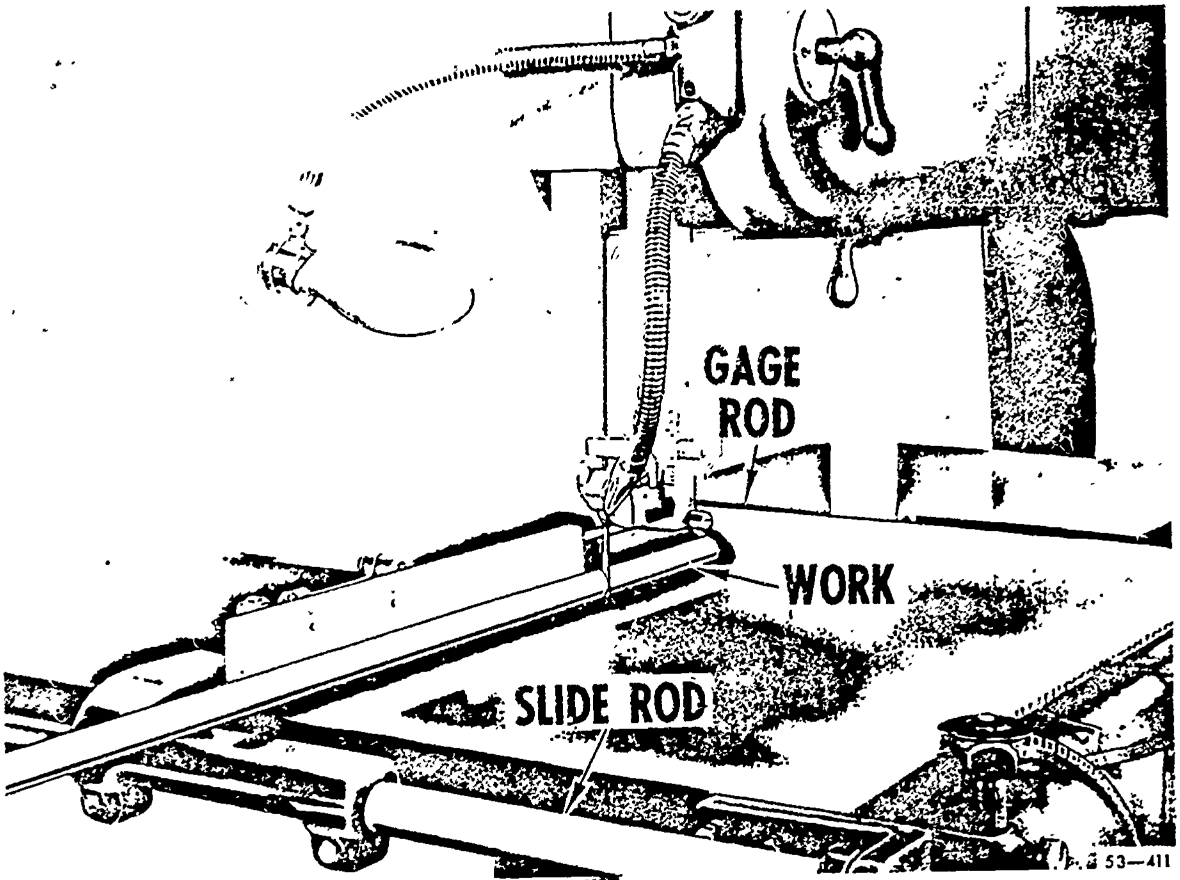


Figure 85 Cutoff and mitering attachment.

the circle which you can cut is limited to the length of the cylindrical bar on the attachment or to the throat depth of the machine. The disc-cutting attachment consists of three parts. (1) a clamp and cylindrical bar. which can be fastened to the saw guidepost, (2) an adjustable arm that slides on the cylindrical bar. and (3) a pivot or centering pin—all shown in figure 87. The disc must be laid out and the center 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. NOTE 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 sawteeth and at the desired distance from the saw band.

20-8. **Power Feed Attachment.** The *power feed attachment*, shown in figure 88, 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 on the front of the machine. 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 88. Position the movable pulleys in line with the sides of the work for straight sawing and slightly outside the edges of the work for contour sawing. NOTE: You can use the *workholding jaw*, shown in figure 88, to hold the work. Place the power feed chain around it as shown in figure 88.

20-9. You engage the power feed by releasing the foot pedal. Unlock the foot pedal by depressing it and moving it slightly to the left. Do not allow the foot pedal to rise too rapidly, or the work may be jerked into the saw band. Disengage the power feed by depressing the foot pedal when you are approaching the end of the cut.

133

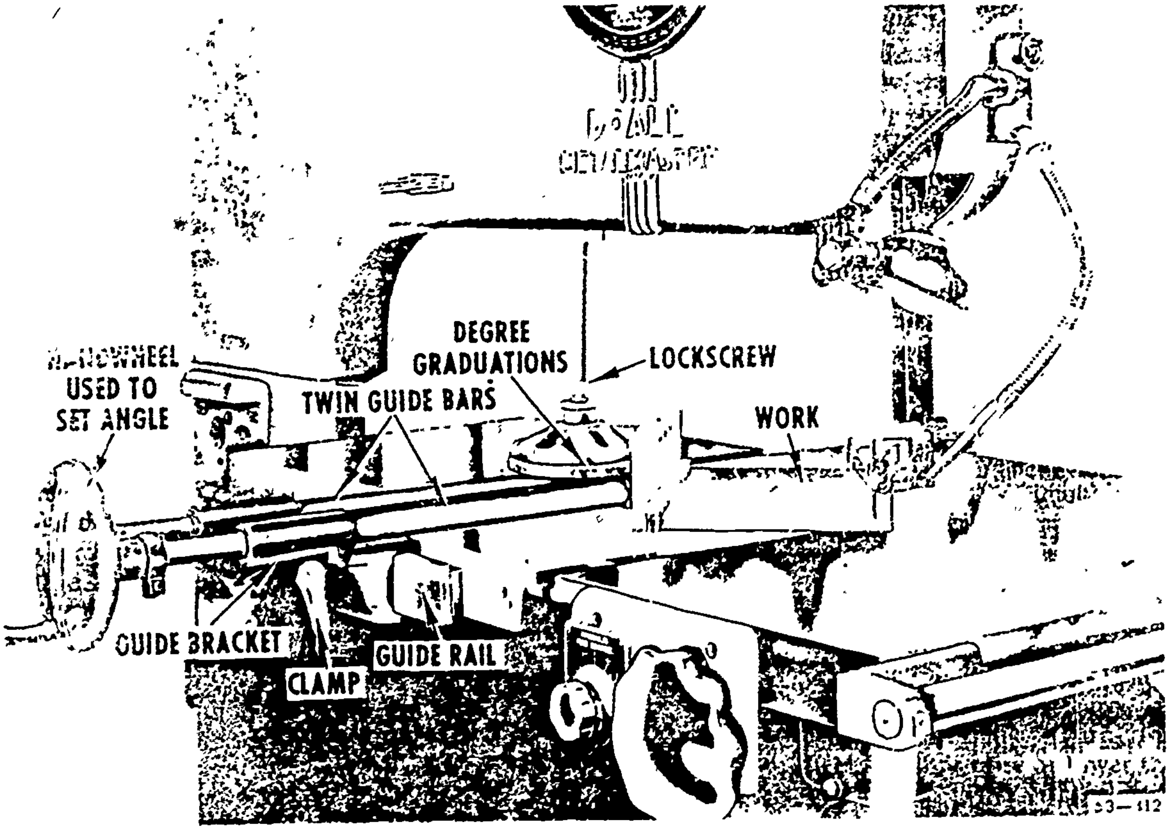


Figure 86. All-purpose mitering attachment.

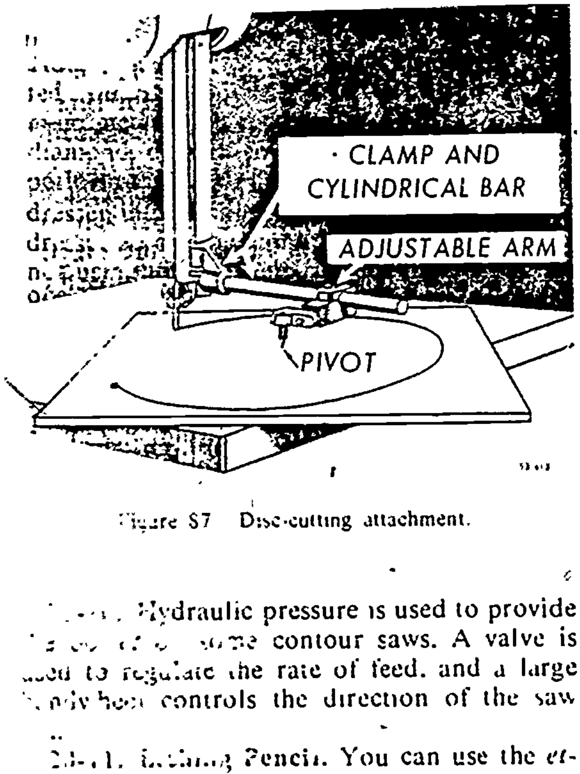


Figure 87 Disc-cutting attachment.

[illegible] Hydraulic pressure is used to provide [illegible] some contour saws. A valve is used to regulate the rate of feed, and a large handwheel controls the direction of the saw

[illegible] Etching Pencil. You can use the *etching pencil*, shown in figure 39, with the butt w[illegible] mark metal; such as tools, jigs, and fixtures. The attachment consists of two cables with a cord-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 jaws, also prevents any movement of the jaws when the welding level 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.

21. Contour Machine Maintenance

21-1. The contour machine is one of the easiest machine tools in the machine shop to maintain. It is a very simple machine with only a few moving parts, all of which are easily accessible. In this section we discuss (1) installation and alinement, (2) lubrication, and (3) troubleshooting and adjustments

21-2. Installation and Alinement. The contour machine should be installed so that the light strikes the table from over the right shoulder of the operator whe[illegible]

139

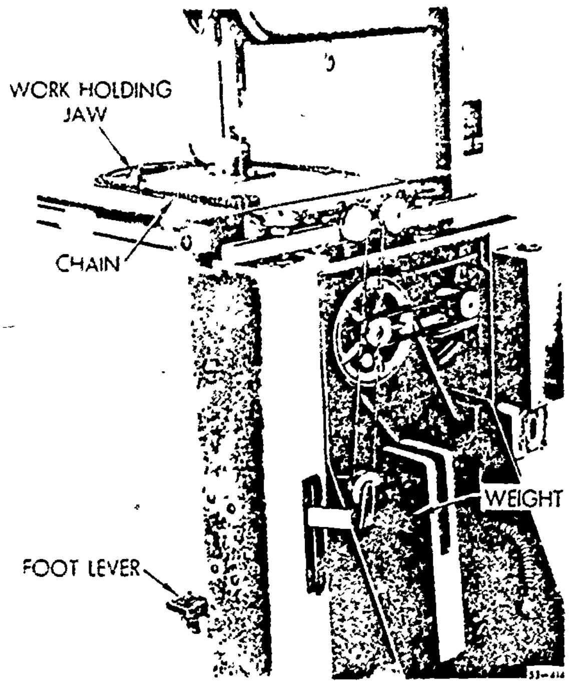


Figure 88 Power feed

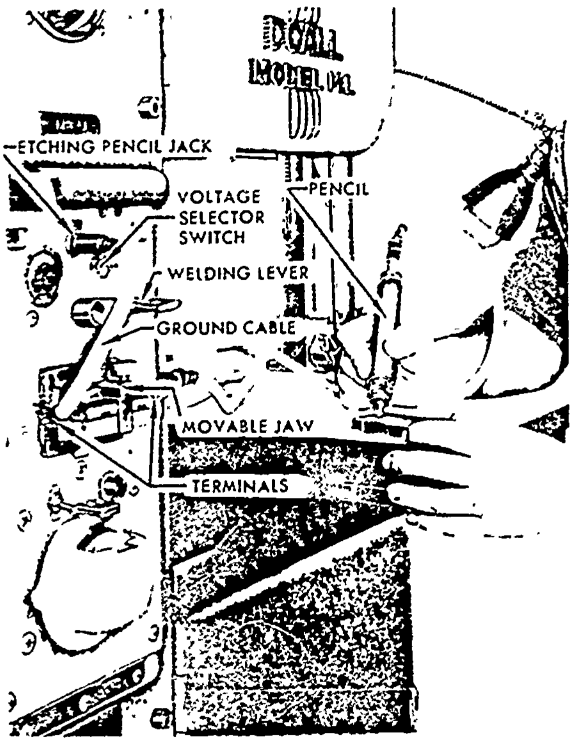


Figure 89 Etching pencil

for sawing and from over his left shoulder when he is in position for filing or polishing. The machine should be rigid and level on a solid floor to insure that the frame will not spring out of alinement. After the machine is in place, shim under the base until the post and keeper block aline with each other. You can aline the post and keeper block with the aid of a machinist square. When alinement is obtained, bolt the machine securely to the floor with lag screws through the holes in the flanged base of the machine.

21-3. **Lubrication.** When you are lubricating the contour machine you should use a lubrication chart to insure that all parts will be lubricated as recommended by the manufacturer. The lubrication chart usually includes the following items:

a. Transmission. The transmission should be oiled monthly. Use a good grade of transmission oil and fill the case until the oil appears in the filler pipe elbow.

b. Variable speed pulley. The variable speed pulley should be oiled about once a month with a good grade of spindle oil. This unit should not be overoiled, because excess oil causes the belt to slip.

c. Drive motor. The drive motor has wool-packed sleeve bearings and should be oiled about once a month with a good grade of machine oil.

d. Grinder motor. The grinder motor can be oiled through the two spring cap oil fillers on the butt welder panel. A few drops of machine oil about once a month are sufficient.

e. Moving parts. Moving parts, such as the slide rod, slide screw, thrust bearing of the upper wheel, power feed screw, and speed change screw, should be oiled occasionally to assure free movement.

21-4 **Air Pump.** The air pump has plastic vanes and should not be lubricated with oil. If it becomes necessary to lubricate the air pump, pour powdered graphite into the air intake while the pump is operating.

21-5. **Troubleshooting and Adjustment.** Since the contour machine is a simple machine with few moving parts, only a few problems normally occur and not many adjustments are required. We will discuss some of the most common problems and the steps to take to correct them.

21-6. *V-belt slippage.* V-belts tend to stretch after continued operation. You can adjust for this condition by increasing the counterbalance. This is done by loosening the nut on the base equalizer spring. This results in placing more of the dead weight of the motor on the drive belt. However, too much weight causes unnecessary wear on the bearings and shaft. Proper counterbalance is attained when the motor mounting base just touches the base of the machine when the motor is allowed to drop from a height of 1 inch.

21-7. *Butt Welder failure.* Only certain adjustments can be made on the butt welder. The movement of the welder jaws cannot be changed. The switch cutoff can be adjusted in or out to regulate the timing of the weld. A clockwise rotation of the cutoff switch adjustment screw usually causes a slower breaking of the welding circuit to give more heat at the point of weld, while a counterclockwise adjustment gives less heat. If the failure is electrical, you should not attempt the repair yourself but should have the repairs made by an electrician.

21-8. *Incorrect tracking.* Incorrect tracking of the saw band results in excessive pressure on the guide block thrust rollers, causing the saw to wear out the bearing cap. When this occurs, remove the worn cap from the bearing and press on a new one.

21-9. *Air pump stoppage.* Sticking plastic vanes in the pump are caused by dirt and oil and will result in air pump failure. Clean the vanes with a dry brush or rag and lubricate them sparingly with powdered graphite.

21-10. *Worn guide inserts.* Worn or scored inserts on the surface that make contact with the sawband should be reground to a 45° angle. After grinding the inserts, remove the sharp edges with an oilstone.

21-11. *Grooved tires.* Saw band slippage causes wear and grooves on the tire of the lower wheel, excessive tension on a small, narrow saw band causes wear and grooves on both the lower and upper wheels. If the wear causes the saw band to ride in the grooves, it is impossible to track the saw band properly. Remove the tires, turn them inside out, and replace them on the wheels. If the wear is excessive and the grooves are deep, install new tires.

CHAPTER 6

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. precision, in which you grind work with a machine to close tolerances; and hand, in which you hold the wheel to the work or the work to the wheel and grind by hand manipulation. In this chapter we will discuss cylindrical, taper, face, and form grinding; surface grinding operations, tool and cutter grinding, grinding machine attachments and grinding machine maintenance.

22. Cylindrical, Taper, Face, and Form Grinding

22-1. You can perform many types of operations on a grinding machine. In this section we will cover cylindrical, taper, face, and form grinding.

22-2. **Cylindrical Grinding.** Cylindrical grinding is the grinding of cylindrical surfaces to remove the warpage caused by heat treatment, reduce the work to exact size, and improve the finish. The work can be held and rotated by mounting it (1) between centers and driving it with a drive dog, (2) in a chuck and supporting it with a center rest, (3) on a live center and supporting it with a center rest, and (4) on a faceplate. The revolving grinding wheel provides the cutting action which takes place in the area of contact. The area of contact varies when the dimensions of the wheel or the work are increased or decreased and when the depth of cut is increased or decreased.

22-3. The wheel and the work are usually set to revolve in opposite directions at the area of contact, as shown in figure 90, to provide 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 spread, work speed, and table traverse.

22-4. *Preparing work for grinding between centers.* Before you can grind work which is to be held between centers you must lap the center holes to insure obtaining precise limits for roundness, straightness, and concentricity and to increase the life of the work centers. Lapping removes the scale and distortion left by heat treating and corrects inaccurately or roughly drilled holes.

22-5. If your shop does not have a center lapping machine you can lap the center holes in a lathe or drill press. To do this, mount a round piece of hard wood, one end of which is turned or filed to a 60° angle, in a chuck. Cover the tapered 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.

22-6. Lapping a center hole by machine, as shown in figure 91, is a simple operation. You hold one end of the work on an adjustable center and, by pulling down on a hand lever, 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

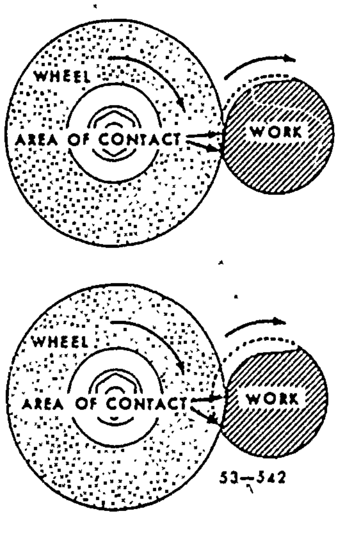


Figure 90. Area of contact

137

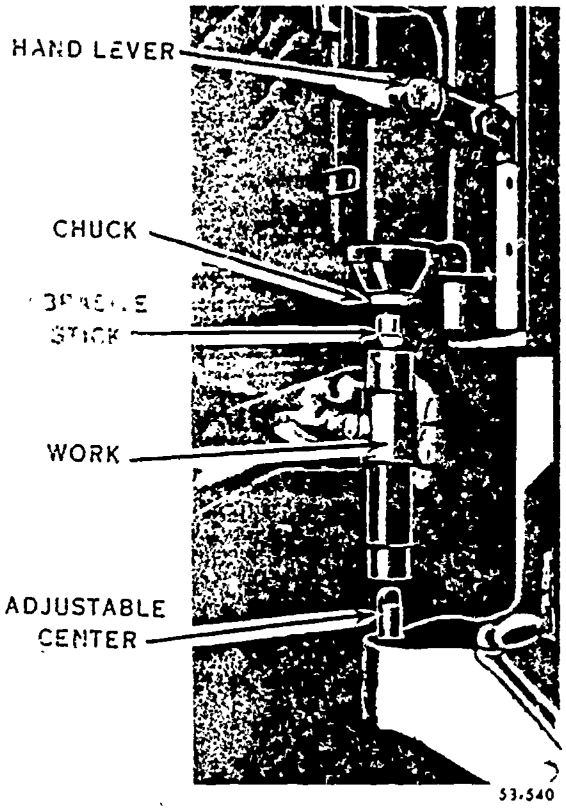


Figure 91. Lapping setup.

[illegible] up and down on the ways to accommodate a maximum work length of 36 inches. The maximum width which can be accommodated is [illegible] 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 steel spindle. It should be dressed frequently. A loaded or blackened stone does not cut freely, and generates excessive heat. Lapping stones are often treated [illegible] to improve their cutting action. Some [illegible], however, prefer to use untreated stones and apply their own cutting lubricant, such as [illegible] kerosene, or spindle oil.

22-7. [illegible] *cylindrical grinding.* Assume that you have the job of grinding a straight-[illegible] reamer to a specified size. After [illegible] the center holes, you could cylin-[illegible] the reamer on the universal tool and cutter grinder in the following manner: Select the proper wheel and mount it on the wheel flange assembly. Mount the wheel and wheel flange assembly on the wheelhead spindle and tighten the spindle nut. Place the wheel guards in position to adequately cover the wheel. Position the column so that the graduations on the base and on the front of the table indicate a ZERO setting and tighten the lock bolts. When the ZERO setting is indicated on the column and table, the wheelhead is parallel to the table. Place the diamond dresser and holder on the machine table with the diamond point properly positioned in relation to the wheel face and tighten the lock bolt. Calculate the wheel speed and the work speed. Turn on the spindle drive motor to start the wheel rotating and let it run continuously throughout the grinding operation. The continuous running of the spindle and the wheel keeps the spindle bearings and the grinding wheel in balance.

22-8. Turn on the coolant pump motor and position the nozzle to supply an adequate flow of coolant to the area of wheel contact. Place the splash guards in position to return the coolant to the reservoir tank. Turn the elevating handwheel to either raise or lower the wheelhead spindle until the lower cricket mark on the vertical slide corresponds with the cricket mark on the wheelhead. This alines the center of the spindle and the wheel with the headstock center. Bring the wheel forward by turning the crossfeed handwheel until the revolving grinding wheel lightly touches the diamond dresser and dress and true the wheel. The depth of cut for dressing and truing should not exceed 0.001 inch for each pass across the face of the wheels. Move the wheel away from the diamond dresser to allow safe access to the table and remove the dresser and holder and splash guards from the table.

22-9. Mount the headstock on the left end of the table. Aline the graduated base of the headstock to the ZERO setting and secure the lock bolts. Attach the proper size of drive dog to the fluted end of the reamer. Place the reamer between the headstock and footstock centers with the end with the attached drive dog positioned on the headstock dead center. Do this in such a manner that the dead center drive stud will be in the crotch of the drive dog. Support the shank end of the reamer with the footstock center under spring tension. Secure the footstock to the machine table. Position the table trip dogs to allow minimum table traverse. The wheel should run off the shank end of the reamer into the footstock half center. Not more than one-half the wheel width should run off the shank into the undercut between the [illegible] and the flutes. Bring the wheel to [illegible] proximately 1/8 inch of the reamer shank and engage the work head clutch to revolve the reamer. The grinding wheel and the reamer should be revolving in opposite directions at the area of contact, as shown in figure 90.

22-10. Pick up the cut by hand manipulation of the wheel crossfeed and the table traverse. If any warpage is evident from heat treatment, pick up the cut at the highest point of warp. Place the splash guards in position to return the coolant to the reservoir tank. Calculate the table traverse feed for roughing, and set the table traverse speed levers to correspond as closely as possible to these calculations. Use the formula: Table traverse = width of wheel × fraction for finish × rpm of work. Turn on the table traverse power feed and grind the shank of the reamer, using light depths of cut until you have a clean ground surface from end to end. Stop the table traverse with the wheel positioned off the shank in the half center of the footstock. Remove the reamer from between the centers.

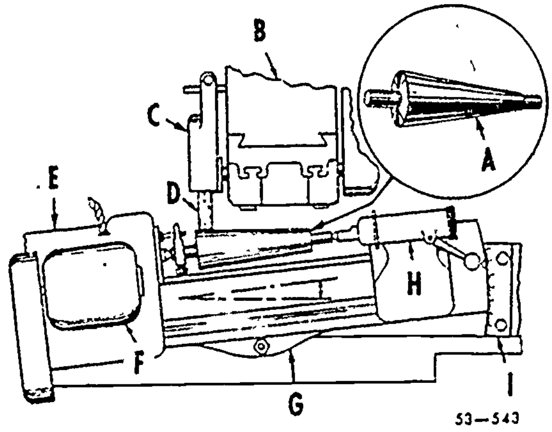
22-11. Use a vernier scale outside micrometer to measure the shank on each end for any taper that may be present. Adjust for any taper by loosening the five locking bolts on the table and swivel the table by turning the fine adjustment screw. You can compensate for any taper up to 3 inches of taper per foot on the fine adjustment scale. CAUTION: Any time the table is swiveled, you must back the wheel away from the work and reestablish the cut, because 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 breaking the wheel and destroying the workpiece.

22-12. Tighten the table lock bolts and rough grind the reamer shank to within 0.001 inch of the finished size. Remember that the body size of the shank of a hand reamer is 0.005 inch less than the nominal size of the reamer. This clearance enables the shank of the reamer to pass through the reamed hole without binding. Remove the reamer from between the centers and prepare the machine for finish grinding. Calculate the table traverse feed for finish grinding and position the traverse speed levers accordingly. Replace the reamer between the centers and pick up the cut for finish grinding. Finish grind the reamer shank to size, using light depths of cut. Remember that only 0.001 inch of metal remains to be removed. Move the wheelhead away from the work and remove the reamer from the centers. Remove the drive dog from the fluted end of the reamer and place it on the shank end.

22-13. Prepare the machine for rough grinding the flutes. Use the same procedures and calculations that you used to prepare the machine for rough grinding the shank. Place the reamer between the centers and position the table trip dogs to accommodate the length of the fluted end of the reamer. Rough grind the reamer flutes, using the same steps that you used to rough the reamer shank. Prepare the machine for finish grinding the reamer flutes, using the same procedures and calculations that you used to finish grind the reamer shank. Finish grind the reamer flutes to the designated size of the reamer, using the same procedures that you used for finish grinding the reamer shank.

22-14. **Taper Grinding.** Taper or conical grinding applies to the grinding of round 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 taper grinding is that flat work may also be ground with a taper. You can grind either external or internal work conically to any length.

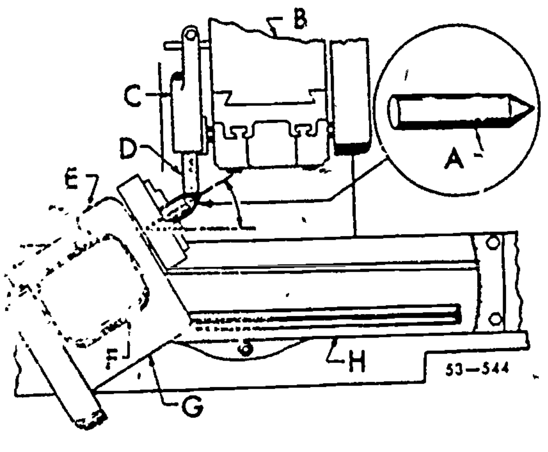
22-15. You grind conical tapered work in a manner similar to grinding straight cylindrical work provided that the taper is not too steep or abrupt. After placing the work between the centers of the grinding machine, swivel the table to the required taper by means of the graduations on the end of the table. The correct work setup is illustrated in figure 92. This setup locates the axis of the work at an angle with the line of motion of the table. As the work moves across



A	Work	E	Headstock
B	Wheelhead	F	Motor
C	Wheel guard	G	Swivel table
D	Grinding wheel	H	Tailstock

I Taper scale

Figure 92. Conical grinding setup for slow tapers.



A. Work
B. Wheel head
C. Wheel guard
D. Grinding wheel
E. Head stock
F. Motor
G. Swivel head stock
H. Table

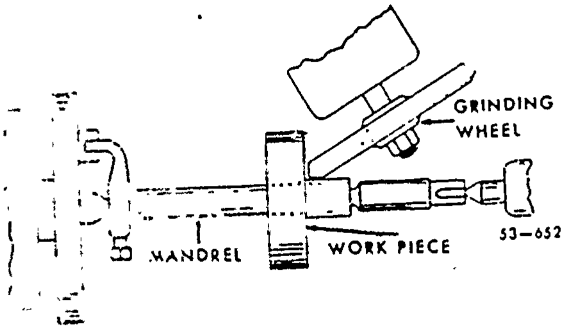
Figure 93. Conical grinding setup for steep tapers.

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.

22-16. You can usually grind steep tapers on a universal machine by swiveling the headstock to the desired angle of the taper, as shown in figure 93. 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 on the universal grinding machine 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. The face of the wheel is always true and parallel to the ways regardless of the angle to which you swivel the table.

22-17. There are many methods of checking tapers. The two most common methods are to measure the taper per inch with a micrometer or with a gage. When you measure a taper with a tapered 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. You will usually grind long tapers by setting the swivel table to correspond to the inches of taper per foot or to the desired number of degrees of taper specified for the work.

22-18. **Face Grinding.** Face grinding is often necessary on heat treated or hard parts to obtain the correct length and finish. In making the work setup you must be careful to insure that the face to be ground is square with the axis of the work. If it is possible, you should do all the necessary grinding operations, such as cylindrical, shoulder, and face grinding, at one setting to insure maximum accuracy. There are three methods of face grinding: angular wheel, cup wheel, and straight wheel. You normally use the angular wheel method, shown in figure 94, when you grind two or more surfaces, such as a cylindrical surface, shoulder, and the face. A disadvantage of the angular method is that the wheel spindle axis is not perpendicular to the face of the work.



Face grinding with an angular shaped wheel

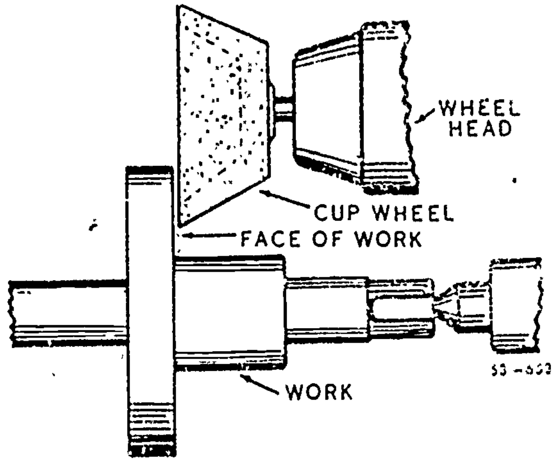


Figure 95 Face grinding with a cup wheel

22-19. You can use the cup wheel method, shown in figure 95, for either external or internal face grinding by selecting a cup wheel of proper size. The area of contact between the

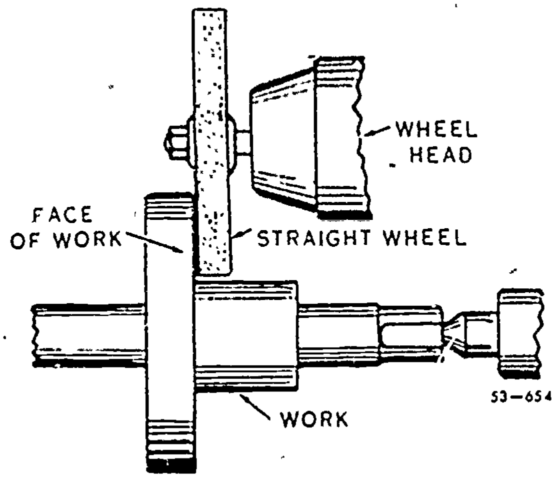


Figure 96 Face grinding with a straight wheel.

wheel and the face of the work is, as you can see, quite small. This is a desirable feature.

22-20. The straight wheel method, shown in figure 96, is seldom used in Air Force shops. You would normally use this method for rough grinding only. Grinding with the side of the wheel is not efficient, since the area of contact is too large. If you must use this method, you can obtain better results by recessing the wheel.

22-21. **Form Grinding.** Form grinding is performed in much the same manner as form turning on a lathe or form milling on a milling machine. The form that is to be ground on the surface is cut into the face of the wheel. This type of grinding should be limited to small parts because of the large surface contact of the wheel to the work. An example of this type of form grinding would be the grinding of a convex surface. The wheel for grinding a convex surface would be trued to a concave shape of the proper radius and the surface is then ground to the proper size. Extremely light cuts should be made to reduce the possibility of burning the work or causing it to develop grinding cracks. Form grinding can be done on the surface grinder or on a cylindrical grinding machine

23. Surface Grinding

23-1. Surface grinding is the grinding of flat surfaces In actual use the surface may be in a horizontal, vertical, or angular position. You can compare surface grinding to machining a flat surface on a shaper if you remember that a shaper uses a single-point cutting tool and a grinder uses a grinding wheel There are two types of horizontal spindle surface grinders. In this section we will discuss types of surface grinders, the action of the wheelhead and table, and a typical surface grinding operation.

23-2. Horizontal spindle surface grinders are designed for efficient production of accurate flat surfaces where precision, fine finish, and rapid removal of stock are of equal importance. They may be divided into two classes 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. Wheel feed takes place at each end of the table movement. Figure 97 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 98. The wheel moves in a horizontal plane across the work from the outer to the inner circumference and back.

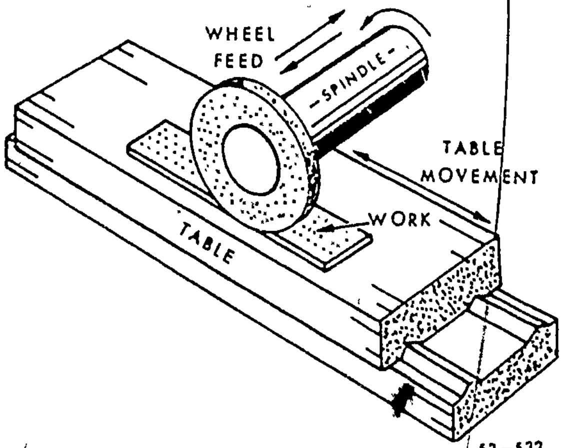


Figure 97. Reciprocating table.

23-3. Assume that you have a hardened steel parallel to grind to a specified size. You could perform this operation in the following man-

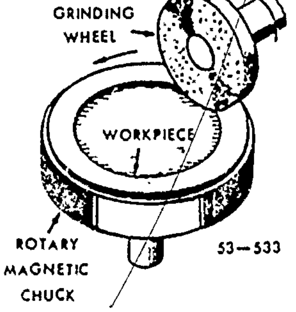


Figure 98 Rotating table

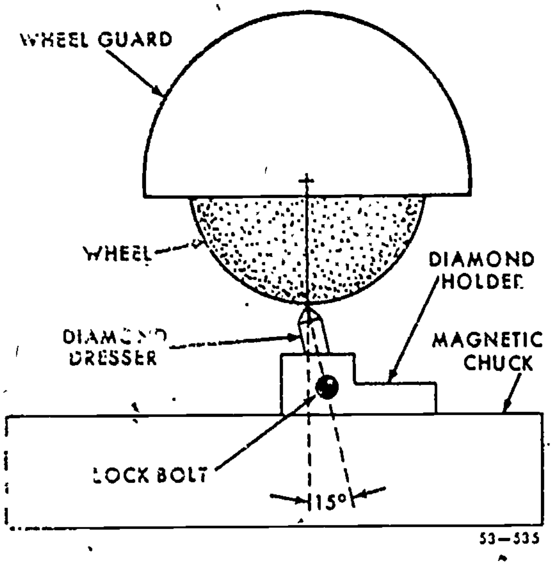


Figure 99 Surface grinder wheel dressing setup

ner. Mount the proper wheel on the wheel flange assembly. Mount the wheel and wheel flange assembly on the wheelhead spindle. Tighten the spindle and flange nuts. Place guards over the wheel. Mount the diamond dresser and holder, as shown in figure 99, on the magnetic chuck and turn the chuck switch to the ON position. (NOTE: Tilt the diamond in the direction of wheel rotation.) Position the wheel directly over the diamond. Start the wheel rotating and bring the wheel down until it touches the diamond. Turn on the coolant and dress and true the wheel by using the hand crossfeed. Move the wheelhead assembly away from the table enough to allow safe and easy access to the chuck face. Position the magnetic chuck switch to the OFF position and remove the diamond and holder from the chuck.

23-4. Clean the chuck face thoroughly to remove all abrasive residue. Place the parallel on the magnetic chuck and shim if necessary. Turn the magnetic chuck switch to the ON position. Position the table trip dogs so that the wheel will run off the parallel at both ends. Position the wheelhead trip dogs so that the wheel will run off sides of the parallel. Turn on the machine, hydraulic system, and coolant pump. Check the power feed and the wheelhead and table for wheel overrun and make any necessary adjustments. You could use the continuous power feed for these settings. Using the hand feed, position the wheel directly over the parallel and pick up the cut while the table is in motion. Position the coolant nozzle to supply an adequate volume of coolant to the wheel and the parallel.

23-5. Turn on the wheelhead power feed and rough grind the parallel. The downfeed depth of cut should not exceed 0.002 inch. If the wheel loads during rough grinding, repeat the dressing operation. Stop the table motion and move the wheelhead assembly away to allow safe access to the parallel. Turn the magnetic chuck switch to the OFF position and remove the parallel. Clean the chuck thoroughly. Replace the parallel in the same position on the chuck with the ground side down. Turn the magnetic chuck ON and rough grind the second side. Turn on wheelhead power feed and finish grind the side of the parallel. The downfeed depth of cut for finish grinding should not exceed 0.0005 inch. Deburr the parallel to remove the rough edges and check all dimensions for accuracy.

24. Tool and Cutter Grinding

24-1. The working efficiency of a cutter is largely determined by the keennes of its cutting edges. Therefore, it is important to sharpen a cutter at the first sign of dullness. A dull cutter not only leaves a poorly finished surface, but continued use of such a 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 cutting 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 cutters cylindrically, cutting tool clearance, grinding form cutters, grinding shell end mills, and helical milling cutters.

24-2. **Grinding Cutters Cylindrically.** Various types of cutting tools, such as reamers and milling cutters, are ground cylindrically.

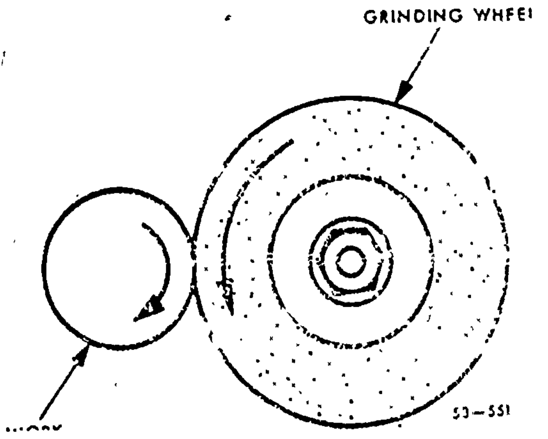


Figure 100 Wheel and cutter rotation for cylindrical grinding.

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 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 100. 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 grain and a medium grade grinding wheel for the cylindrical grinding of hardened steel and high-speed steel cutters.

24-3. 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 101 illustrates two methods of straight grinding wheel setup. In method A, the 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. This method has a tendency to draw the temper from the metal. In method B, the wheel rotation is from the cutting edge toward the body of the tooth. In this method there is less danger of burning the tooth, but you must exercise great care to hold the cutter on the tooth rest. If the cutter turns while you are grinding it, the tooth will be ruined. Cup wheels, as shown in figure 102, are also used extensively to grind cutters and reamers. Cup wheels are used in a manner similar to that used with straight wheels.

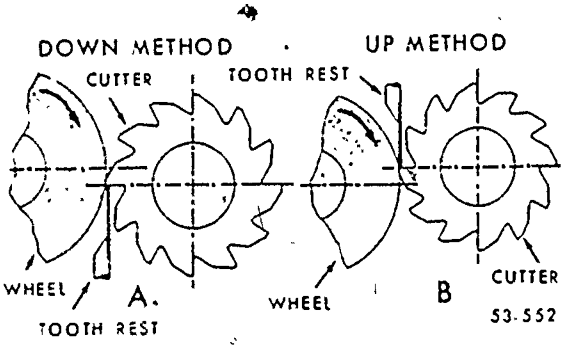


Figure 101. Cutter sharpening positions.

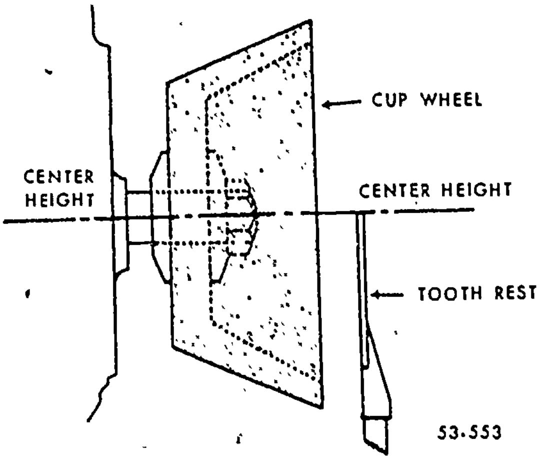


Figure 102. Alinement of tooth rest with axis of cup wheel.

24-4. **Cutting Tool Clearance.** Correct clearance back of the cutting edge of any tool is 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 103 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. The width 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 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

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 [illegible]
Soft bronze	10 to 12	30	Large cutters [illegible]
Medium bronze	6 to 7	30	
Hard bronze	4 to 5	30	
Copper	12 to 15	30	
Aluminum	10 to 12	30	

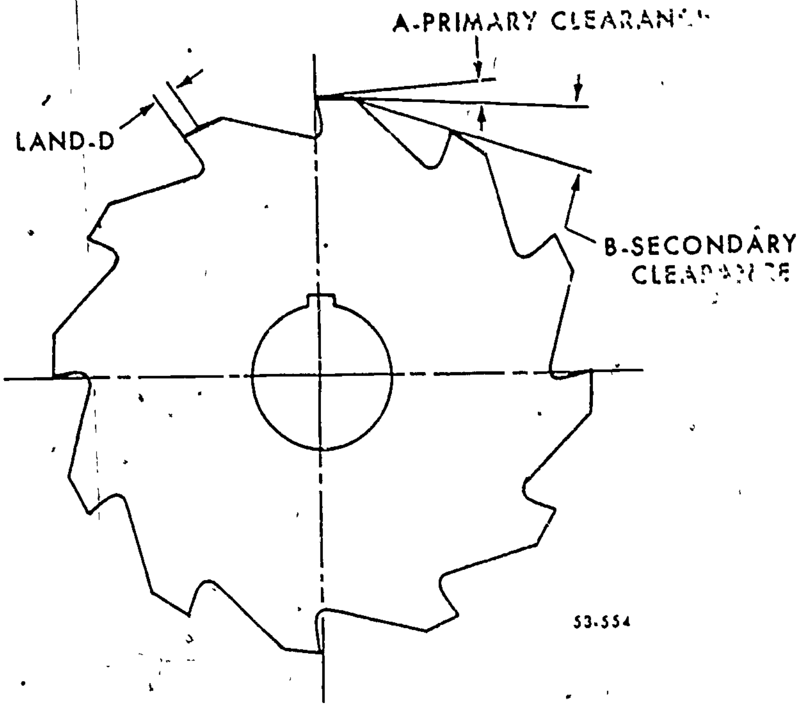


Figure 103. Recommended clearance angles

[illegible], multiply the clearance angle of the cutter in degrees times the wheel diameter in inches times the constant 0.0088. The constant 0.0088 is derived by measurement of the distance of 1° on the circumference line of a 1-inch circle. The result will be the amount in ten-thousandths of an inch that you raise or lower [illegible]er 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.

24-5. The tooth rest is normally fastened to the table when you grind straight-toothed cutters. When you grind helical-toothed 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 *Machinery's Handbook*, which normally have charts from which you can obtain the recommended clearance angles for most cutters.

24-6. **Grinding From Cutters.** Formed or eccentrically relieved cutters, such as gear cutters and convex or concave cutters can be sharpened in the same manner as the profile type cutters. Form cutters have a definite shape that must be retained through many sharpenings. To retain this shape, you grind the face of the tooth. You grind the face on th[illegible] with a radial rake. Excessive positive or negative 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 form cutters by using the infeed type of form 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 form cutter on a mandrel or a stub arbor and grind the cutting edges by using the index method in a universal head.

24-7. **Grinding Shell End Mills.** Shell end mills are sharpened by mounting the cutter on a stub arbor or shell end mill arbor and placing the arbor in the universal attachment in the same manner that it is supported in the milling machine. Place a cup wheel, either straight or flaring cup, on the spindle and with an abrasive stick dress the cutting edge of the wheel. To prevent the end mill from dragging, the teeth are usually slightly tapered to the middle or center of the cutter. This is done by swiveling the universal head about 1/2° to 1°. In order to obtain the proper clearance angle on the teeth, swivel the head vertically to the prescribed clearance angle. The micrometer tooth rest is usually used in this sharpening operation because it is easily set to the proper height.

24-8. **Grinding Helical Milling Cutters.** A helical milling cutter is ground by mounting the cutter on the arbor or mandrel and placing it between the right- and left-hand footstock. Select and mount a cup wheel. Swivel it several degrees so that the back cutting edge of the wheel does not come in contact with the cutter being ground. Adjust the tooth rest to the proper height to achieve the desired clearance angle. This setting may be calculated using the following formula:

$$\text{Tooth rest setting} = 0.0088 \times \text{clearance angle} \times \text{dia of cutter}$$

The cutter clearance may be found in machinist publications, such as *Machinery's Handbook*.

24-9. The cutter should be held lightly against the tooth rest, with just enough pressure to maintain contact. When you use the down method, the grinding wheel aids in maintaining this pressure. Care should be taken to maintain contact with the tooth rest when you return the cutter to the starting point. If the cutter is not in contact with the tooth rest when it is returned to the starting point, damage to the tooth will occur. The cutter may be traversed across the wheel face by the movement of the table or by sliding on an arbor. The life of cutter teeth depends on keeping the peripheral cutting edges concentric. When the teeth are out of round, the resulting pounding action soon breaks down the cutting edge, thus shortening the life of the cutter. This condition is due largely to the wearing of the grinding wheel during the sharpening operation. The out-of-round condition can be corrected by an equalizing operation. The equalizing operation is simply grinding around the cutter and then rotating the cutter 180° and starting the second cut around the cutter on the tooth just opposite the first tooth. Light cuts should be used to aid in the reduction of wheel wear. This operation is repeated until the cutter has been completely sharpened.

25. Grinding Machine Attachments

25-1. There is a large assortment of attachments which are designed to perform some operation in conjunction with the basic machine. When these attachments are used for the jobs for which they are designed, they make the grinding operations much easier. We will discuss some of the more common attachments which may be used on most general purpose grinding machines.

25-2. **Headstock.** Although the headstock is sometimes considered a part of the basic machine, it is actually an attachment. The headstock is normally used with the universal grinding machine; it provides a means of holding and driving a workpiece in relation to the cutting action of a grinding wheel.

25-3. **Work Head.** The work head differs from the headstock in that it normally has no provision for power. It is primarily a holding device on which a chuck, collet, or faceplate can be mounted to hold work securely for grinding operations. It is especially adaptable for the sharpening of cutters, such as gear cutters, reamers, and end mills.

25-4. **Surface Grinding.** You can use the surface grinding attachment for grinding flat forming tools, lathe tools, flat thread chasers, chisels, and work of a similar nature. The attachment consists of a vise which you can swivel in two planes. By placing the regular work head support between the vise support and the base, you can adjust the attachment in three planes. Using this attachment makes it possible to grind almost any flat tool without removing it from the vise. Therefore, you can maintain greater accuracy between the ground surfaces.

25-5. **Gear Cutter.** Since the cutters used to produce gear teeth are form cutters, they are not sharpened or ground in the conventional manner. To properly sharpen a form cutter it must be ground on the face of the tooth. The gear cutter attachment is designed to hold a cutter on an arbor in such a manner that it produces a rotary or circular motion of the cutter teeth in relation to the grinding wheel. On most cutter grinders the grinding wheel runs in a plane perpendicular to the table. Therefore, most grinding is vertical. The gear cutter attachment provides a horizontal approach for the cutter to the grinding wheel.

26. Grinding Machine Maintenance

26-1. Maintenance is an extremely important part of your job as a machinist. It is unfortunate, but true, that many machinists do not realize the importance of proper machine maintenance. This is especially true of grinding machines. When you operate a precision machine, it will continue to produce precision work only as long as it is properly maintained. In this section we will discuss (1) installation, (2) preventive maintenance, (3) adjustment of gibs and spindle bearings, and (4) troubleshooting of grinding machines.

26-2. **Installation.** The installation of grinding machines does not usually require a special foundation. Any substantial floor of wood or concrete that is flat and heavy enough to support the weight of the machine is satisfactory. You should never place a precision machine close to any other machinery that has a tendency to vibrate—for example, a punch press—because outside vibration will usually result in a poor finish on the work surface. The grinding machine should be leveled when it is installed in the shop. Place an accurate [illegible]ometer-type precision level on the table or in the ways in at least two directions. Also, check the levelness of the machine periodically, since it will lose its levelness through constant operation.

26-3. **Preventive Maintenance.** Preventive maintenance costs far less than corrective maintenance. The prevention of a costly breakdown is quite easy if you will follow a few simple rules concerning cleanliness, lubrication, and corrosion control.

26-4. *Cleanliness.* Dirt that is allowed to accumulate on a machine will invariably find its way into the bearings, slides, and sensitive electrical units and cause breakdown. Thorough cleaning at regular intervals will practically eliminate this hazard.

26-5. *Lubrication.* Proper lubrication will save time and money. Refer to the recommended lubrication schedule and follow it. When a grinding machine fails because of improper lubrication, it is too late to oil and [illegible]. The same rule applies to coolant. When it becomes rancid or dirty, it is time to change it. Always keep an adequate supply in the reservoir. You should clean and refill the coolant tank at regular intervals regardless of machine usage.

26-6. *Corrosion control.* Grinding machines are especially susceptible to corrosion. This is because most of them use water-soluble types of coolant. When water is left for a period of time on a machine surface, it will cause rust. You should keep all machine surfaces clean and painted. Not only does a clean and painted machine look better but its working life is prolonged.

26-7. **Adjustment of Gibs and Spindle Bearings.** The adjustments that can be made on the mechanical parts of a grinding machine are restricted almost entirely to the gibs and spindle bearings.

26-8. *Gibs.* The various gibs on the grinding machine should be checked and adjusted at regular intervals. Correct adjustment is made when the slides move snugly by hand. If you tighten a gib too tight, the slide may become scored or galled. Loose gibs, on the other hand, will cause vibration, as well as undue wear on the machine ways, and will result in the machine's becoming inaccurate. The correct adjustment to the gibs in the sliding member of the machine is necessary. Otherwise, the machine will not produce accurate work.

26-9. *Spindle bearings.* Spindle bearings on most grinding machines are of either the plain or the antifriction ball bearing type. The surfaces of plain bearings are hardened, ground, and lapped. Some machines have bronze boxes which are provided with spring shoes that automatically compensate for wear. Others are equipped with setscrews by means of which wear and end play may be eliminated. Extreme care should be taken in making any adjustments. The bearings freeze easily if they are not correctly adjusted. 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. This type of bearing has sealed lubrication and does not require adjusting or oiling. Grinding machines with plain spindle bearings require a special, thin spindle oil because of the close tolerances with which they are built. Failure to use the recommended spindle oil in the wheelhead bearings will eventually result in shaft freezing. Excessive spindle oil is harmful to the machine, because the oil overflows, gets on the drive belt, and causes it to slip. In turn, the slippage on the spindle drive pulley causes it to heat and to expand the spindle on its bearings. Because of the close fit, there is danger of the spindle's [illegible]

Bibliography

Books

OBERG, Erik, and Jones, Franklin D. *Machinery's Handbook*, 18th ed. New York: The Industrial Press, 1970.

Department of the Air Force Publications

AFM 127–101, *Industrial Safety Accident Prevention Handbook*, 26 June 1970.

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

TO 34–1–10, *Fundamentals of Ordnance Corps Machine Tools*, 2 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

Taining literature from Block II, AFR53130, *Machinist*.
Training literature from Block III, ABR53130, *Machinist*.
Training literature from Block IV, ABR53130, *Machinist*.
Training literature from Block V, ABR 53130, *Machinist*.
Training literature from Block VI, ABR 53130, *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*.)

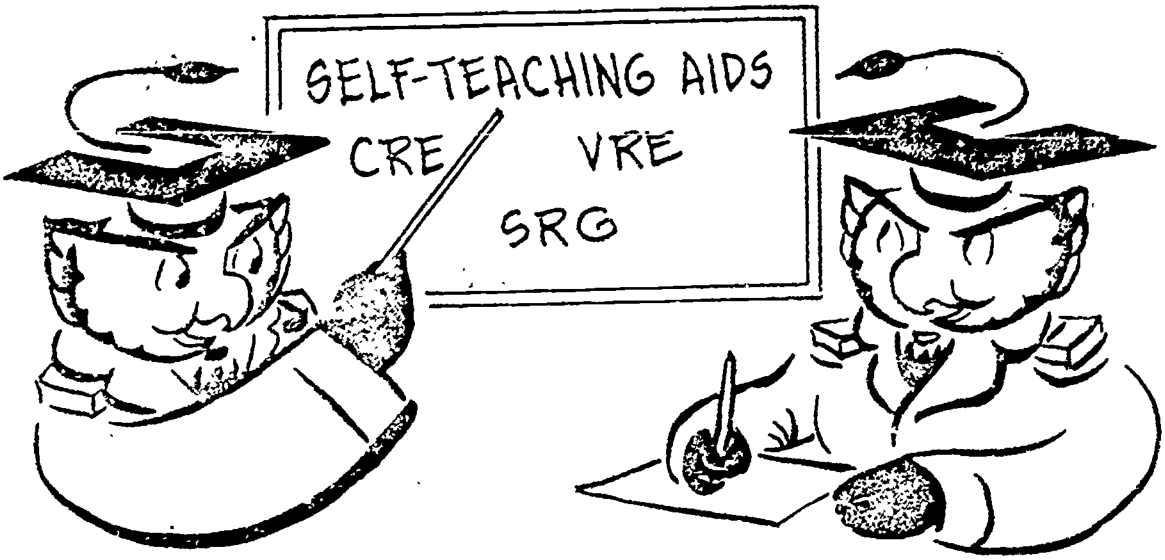
147

13-3

53150 02 23

WORKBOOK

Advanced 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

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 200 through 212*
200	Introduction. Radial Drill Press. pages 1–3
201	Drill Grinding: Drill Press Maintenance. pages 3–5
202	Introduction; Special Lathe Operations; Radii and Form Turning, Taper Turning, Taper Turning With the Compound Rest, pages 6–10
203	Special Lathe Operation: Taper Turning by Offsetting the Tailstock, Taper Turning with Taper Attachments, Toolpost Grinder, pages 10–15
204	Special Lathe Operation; Parting, Spring Winding, Knurling. pages 15–21
205	Lathe Attachments, Special Threading Operations; Lathe Maintenance, pages 21–28
206	Introduction, Special Milling Applications; Helical Milling: Single-Point Milling, pages 29–33
207	Milling Machine Attachments; Gearing and Gear Cutting, pages 33–40
208	Milling Machine Maintenance, pages 40–43
209	Introduction: Flat and Angular Planing; Soulders, Corners, and Grooves; Irregular Shapes; Shaper Maintenance, pages 43–50
210	Introduction; Angular Sawing and Filing; Three-Dimensional, Stack, and Difficult Material Sawing; Contour Machine Attachments; Contour Machine Maintenance, pages 51–58
211	Introduction; Cylindrical, Taper, Face, and Form Grinding, pages 59–63
212	Surface Grinding; Tool and Cutter Grinding; Grinding Machine Attachments; Grinding Machine Maintenance, pages 63–68

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 understanding of radial drill press construction and operation; to show a knowledge of drill grinding, and to show an understanding of drill press maintenance and how it is done.

1. Why is it easier to position the spindle over the work with a radial than with a standard drill press? (1-1)

2. How can the arm of a radial drill press be moved? (1-1, 5)

3. Why is work manually mounted on parallels on a drill press? (1-8)

4. How are the shanks of the drill held in the spindle? (2-3)

5. What is the greatest problem in sharpening a drill? (2-6)

6. What effect will grinding the lip clearance of a drill too small and too large have on drilling? (2-7)

7. When is thinning the web of a drill desirable? (2-8)

8. Why should a drill press be level and solid? (3-1)

CHAPTER 2

Objectives. To show an understanding of special lathe operations, how attachments are used in lathe operations and how special threading operations are performed. To show a knowledge of lathe operator maintenance.

1. Which method of form turning would be most desirable to machine several duplicate pieces? (4-4)

2. When you use the compound rest for radius turning, how is the size of radius controlled? (4-7)

3. What type of taper is generally found in twist drills and in the spindle and tailstock of most lathes? (4-11)

4. What are the dimensions of a Number 8 Jarno taper: large diameter, small diameter, and length of taper? (4-15)

5. List three methods of measuring taper. (4-19)

6. List four methods of measuring the amount of offset. (4-26)

7. Which method of machining tapers would be used when duplicate tapers are to be turned and length of the pieces vary? Why? (4-32)

8. When you use a toolpost grinder, what enables you to use small diameter wheels for internal grinding? (4-39)

9. What is likely to happen when grinding wheels are operated at a speed higher than the recommended speed? (4-40)

10. Why do parting tools have flank relief? (4-47)

11. Where should the point at which the parting occurs be in relation to the chuck? (4-50)

12. List two methods of measuring the length of the portion to be parted. (4-53)

13. What type of spring wire can be substituted for steel when there is danger of corrosion? (4-56)

14. In filing work in a lathe, how should the file be positioned? (4-61)

15. How much stock is usually left for filing in the lathe? (4-63)

16. What is the recommended surface feet per minute for polishing? (4-65)

17. For what reasons are parts knurled? (4-67)

18. What purpose does the thread-chasing dial serve on a lathe? (5-3)

19. Why is the center rest equipped with an overarm? (5-4)

20. For what is the cathead used? (5-6)

21. For what reason are square threads being replaced by Acme threads? (6-2)

22. How is the compound rest set when you machine square threads? (6-5)

23. How is the compound rest set for finishing Acme threads? (6-8)

24. What is the difference between the threading tool used to machine single-lead threads and the threading tool used to machine multiple-lead threads? (6-11)

25. Using a lathe with a lead screw constant of 6, what are the number of teeth in the driving gear and in the driven gear for cutting a 3 mm lead? (6-17)

26. In lathe maintenance what should be periodically checked to insure good operation? (7-1)

27. How are apron feed clutches adjusted? (7-5)

CHAPTER 3

Objectives To show a knowledge of special milling applications. To exhibit an understanding of helical milling. to show a knowledge of single point milling, to evaluate a milling machine requirement and select the proper attachment, to show a knowledge of gearing and gear cutting; and to show how to perform milling machine maintenance.

1. When is it economical to use gang milling? (8-3)

2. How is it possible to perform slotting on a milling machine? (8-4)

3. What type of index plate is best suited for milling a 12-point socket? (8-5)

4. List four methods of performing angular planing. (8-6, 7)

5. What type of cutter is usually used to mill a cylindrical cam groove and how is the cutter mounted? (8-9)

6. Using the formula

$$\text{Lead of segment} = \frac{360^\circ}{\text{included angle of segment}} \times \text{rise in segment}$$

what is the lead of a segment if it has a rise of 0.750 inch in 60°? (8-11)

7. What kind of work does helical milling include? (9-2)

8. Compare the lead of a thread and the lead of a helix. (9-2)

9. Define the lead of a helix. (9-3)

10. When is it more economical to make a fly cutter than to buy a form cutter in single-point milling? (10-1)

11. What type of cutting tool is usually used for boring on a milling machine? (10-3)

12. What determines the cutting speed of the attachment spindle of the high-speed universal attachment? (11-2)

13. Compare the feed control of small and large circular milling attachments. (11-3)

14. What are raising blocks and how are they used? (11-6)

15. For what type of milling operation is a toolmaker's knee used? (11-7)

16. Why do helical gears run more quietly than spur gears? (12-2)

17. How do helical gears differ from spur gears? (12-4)

18. How are helical gears cut on a milling machine? (12-6)

19. What is the circular pitch of a helical gear? (12-8)

20. What is the most common angle between shafts on which bevel gears are mounted? (12-12)

21. Why is the plan dimension, provided by the manufacture, useful in installing a milling machine? (13-2)

22. Why is correct milling machine gib adjustment important? (13-4)

23. How is spindle end play adjusted? (13-5)

24. What is the danger of adjusting the driving clutch too tight? (13-7)

25. How should oil tubes and channels clogged with dirt and oil be cleaned? (13-11)

CHAPTER 4

Objectives To demonstrate a working knowledge of flat and angular planing, to show an understanding of the techniques and procedures for shaping shoulders, corners, and grooves, to show a knowledge of the techniques and procedures for shaping irregular shapes, and to demonstrate a working knowledge of shaper operator maintenance.

1. How are flat surfaces machined on the shaper? (14-2)

2. How should the clapper box be positioned? (14-6)

3. How is the toolside alined when great accuracy is required? (14-9)

4. List the four methods of planing angular surfaces. (14-10)

5. What is the most accurate method of alining the toolslide on a shaper for cutting an angle? (14-13)

6. What type of cutting tool is ground for machining between shoulders? (15-2)

7. Why is it *not* practical to cut deep grooves parallel with vise jaws? (15-7)

8. When cutting keyways on a shaper, why is the centerline extended through the axis of the work? (15-8)

9. How are large irregular shapes, that would be impractical to machine with a form tool, produced? (16-2)

10. To what must particular attention be given in installing a shaper? (17-2)

11. To what degree of accuracy is the shaper leveled? (17-5)

12. How much clearance should the gibs of a shaper have? (17-7)

CHAPTER 5

Objectives. To demonstrate a working knowledge of the techniques of angular sawing and filing, to show an understanding of three-dimensional, stack, and difficult materials sawing; to show a knowledge of how to use contour machine attachments, and to show an understanding of contour machine operator maintenance.

1. How many degrees and in what directions may the table of the contour machine be tilted? (18-1)

2. For what is the lower set of holes in the keeper block used? (18-1)

3. What is the most important factor in the sawing of a three-dimensional object? (19-2)

4. What are the two most common difficult-to-machine metals? (19-3)

5. What keeps the saw band from getting too hot in the friction sawing operation? (19-4)

6. What makes titanium a difficult material to machine? (19-5)

7. What is stack sawing? (19-8)

8. When would you use a magnifying attachment? (20-2)

9. For what reason are angular saw guides used? (20-3)

10. List the three operations which can be performed by the all-purpose mitering attachment? (20-6)

11. What limits the diameter of a circle that can be cut with the disc-cutting attachment? (20-7)

12. How should the contour machine be installed with regard to lighting? (21-2)

13. Where may information about the lubrication of the contour machine be found? (21-3)

CHAPTER 6

Objectives. To demonstrate a knowledge of cylindrical, taper, face, and form grinding; to show an understanding of surface grinding operations, to exhibit a knowledge of tool and cutter grinding; to demonstrate a knowledge of grinding machine attachments; and to show an understanding of grinding machine maintenance.

1. For what reasons are various parts cylindrically ground? (22-2)

2. Why do the wheel and the work revolve in opposite directions? (22-3)

3. Why must the center hole be lapped before grinding work between centers? (22-4)

4. In grinding a conical taper, if the wheel is positioned above or below the center of the work, what will be the results? (22-15)

5. What are the two most common methods of checking tapers for accuracy? (22-17)

6. List the three methods of face grinding. (22-18)

7. Which of the three methods of face grinding is least efficient? Why? (22-20)

8. What type of grinder would be used to produce accurate flat surfaces when fine finish and rapid removal of stock are equally important? (23-2)

9. When you are cylindrically grinding milling cutters, why are the grinding wheel and the work rotated in the same direction at the area of contact? (24-2)

10. After sharpening a cutter, it is discovered that the heel of the teeth is dragging. What would cause this? (24-4)

11. When is the tooth rest normally fastened to the table? (24-5)

12. What would be the result if a formed milling cutter was sharpened with excessive negative or positive rake? (24-6)

13. What attachment is needed to sharpen the cutting teeth of a shell end mill? (24-7)

14. What grinding machine attachment would be necessary for grinding the end of an end mill? (25-3)

15. What should always be avoided in installing a grinding machine? (26-2)

16. When is correct gib adjustment achieved? (26-8)

ANSWERS FOR CHAPTER REVIEW EXERCISES

CHAPTER 1

1. Because the head of a radial drill press is mounted on an arm instead of on the column of the machine. This permits you to swing the head to the right or left or to move it along the arm. Thus it is possible to position the spindle over the work.
2. It can be raised or lowered or moved 360° around the column.
3. To prevent drilling holes in the vise.
4. In a chuck, collet, or directly in the spindle.
5. Grinding the correct lip clearance.
6. If the lip clearance is too small, the drill may break. If it is too large, the cutting edges of the drill will be chipped.
7. After it has been sharpened several times to improve its cutting efficiency.
8. Drilling will not be accurate if the drill press is not level and solid.

CHAPTER 2

1. The forming tool method would be most desirable to machine several duplicate pieces.
2. By feeding the tool in or out on the compound slide.
3. Morse.
4. 1 inch large diameter, .8 inch small diameter, and 4 inches in length.
5. (1) Protractor, (2) tapered ring gage, and (3) scribing equally spaced lines and measuring with a micrometer.
6. (1) With an inside caliper, (2) with the dial test indicator, (3) by using the crossfeed calibrated collar, and (4) by using the cut and try method.
7. The taper attachment. Because the length of the work has no bearing on the taper attachment.
8. Taper shafts which are known as quills.
9. There is a good chance that the wheel will explode or break.
10. So that the point of the tool is the widest point. This prevents the tool from binding.
11. As close as possible to the chuck jaws.
12. (1) Place the end of the steel rule against the side of the parting tool and move the carriage until the desired length is obtained. (2) Aline parting tool to scribed line.

13. Phosphorus bronze wire.

14. At an angle of approximately 10° toward the tailstock.

15. 0.002 inch to 0.005 inch.

16. 5000 SFM.

17. To form a gripping surface and for decoration.

18. To aid in the cutting of threads which are not a multiple of the lead screw pitch.

19. To allow the easy removal and replacement of the work.

20. To provide a bearing surface for the center rest jaw on work of an irregular shape.

21. Because of the difficulty encountered in the machining square threads

22. Parallel to the axis of the machine.

23. Parallel to the ways of the machine.

24. Greater side clearance is necessary.

25. $\frac{6}{\frac{25.4}{3}} = 6 \div \frac{25.4}{3} = \frac{6 \times 3}{25.4} = \frac{18}{25.4} \times \frac{5}{5} = \frac{90}{127}$

26. Levelness, spindle bearing, clutches, gibs, crossfeed, lead screw, gearing, and lubrication.

27. A thrust screw is turned clockwise until the clutch lever snaps in and out of engagement.

CHAPTER 3

1. When several identical parts must be machined on a milling machine.

2. By using an attachment.

3. A direct index plate.

4. By using angular milling cutters, by tilting the vertical head and using an end mill or a shell milling cutter, by using a toolmaker's knee, and by using angular parallels.

5. An end mill mounted in a vertical milling attachment.

6. 4.5 inches.

7. The milling of helical tooth milling cutters, helical reamers, twist drills, helical gears, and some types of cams.

8. The lead of a thread is usually short in proportion to its diameter and length, [illegible] helix may be long in proportion to its diameter and length.

9. It is the distance the helix advances in one complete turn around the work.

10. When a very limited number of cutting and boring operations are necessary.

11. A fly cutter

12. The ratio between the milling machine spindle and the attachment spindle.

13. The small attachment has only hand feed. The large attachment has both hand and power feed.

14. They are heavy duty parallels which usually come in matched pairs. They are mounted on the table and the index head is mounted on them.

15. Angular milling.

16. Because several, rather than two, teeth make contact simultaneously.

17. The teeth of helical gears are inclined at an angle to the axis of the gear. The teeth of a spur gear are parallel with the axis of the gear.

18. By using an index head which is geared to the swivel table and feed screw.

19. The distance from a point on one tooth to a corresponding point on an adjacent tooth measured along the pitch circle.

20. 90°.

21. The plan dimension gives the length, width, and height of the machine. Additional space should be provided so that the longitudinal table movement does not interfere with the operation of other machines.

22. If the gibs are too tight, the slides may become scored. If they are too loose, chatter and vibration will occur.

23. By loosening the locking set screw or nut and tightening the thrust nut just enough to remove the slack

24. The starting lever will place strain on the clutch fingers and may cause them to break.

25. They should be removed by flushing the tubes and channels with a flushing oil.

CHAPTER 4

1. By feeding the work below the ram or by feeding the tool past the work by using the [illegible]

2. It should be tilted opposite the direction of the cut.

3. By using the dial test indicator

4. (1) Swiveling the vise, (2) swiveling the toolhead, (3) swiveling or tilting the table, and (4) adjustable angle plate or fixture

5. Using a sine bar and a dial test indicator

6. Roundnose.

7. The pressure of the vise may cause the groove to close and bind the cutting tool.

8. The extended centerline helps to line up the work and the cutting tool.

9. By moving the work and the cutting tool simultaneously.

10. The amount of room around the shaper.

11. .001 inch per foot

12. .002 inch between bearing surfaces.

CHAPTER 5

1. Forward and backward 10°, to the left 10°, and to the right 45°.

2. When angles of more than 20° are sawed, the lower set of holes must be used.

3. The layout prior to the actual cutting.

4. Nickel-alloyed stainless steel and titanium alloys.

5. The velocity of the saw band creates an air-cooling effect.

6. Titanium is a poor conductor of heat and tends to overheat the cutting tool.

7. Stack sawing is the sawing of several pieces of sheet material of the same shape in one operation.

8. When you are doing precision sawing and filing to close tolerances.

9. To allow work to be sawed that would normally be too long.

10. Ripping, cutting off, and mitering.

11. The length of the cylindrical bar or the throat depth of the machine.

12. The light should strike the table of the machine over the right shoulder of the operator when he is in the position for sawing and over the left shoulder when he is in position for filing and polishing.

13. Lubrication chart.

CHAPTER 6

1. To remove warpage, reduce work to exact size, and to improve the finish.
2. To produce a shearing-type of cutting action between the wheel and the work.
3. To insure precise limits for roundness, straightness, and concentricity and to increase the life of the work centers.
4. The taper will be different from that which the table setting indicates.
5. A micrometer or a taper ring gage or plug gage.
6. Angular wheel, cup wheel, and straight wheel methods.
7. The straight wheel method. The area of contact is too large.
8. A horizontal spindle surface grinder.
9. To produce a slight amount of clearance.
10. Insufficient clearance would probably be the biggest factor.
11. To sharpen straight-toothed cutters.
12. It would change the form of the cutter.
13. Universal work head.
14. A work head.
15. Placing it near machinery that may have a tendency to vibrate.
16. When the slides move snugly by hand.

165

STOP—

1. MATCH ANSWER SHEET TO THIS EXERCISE NUMBER.
2. USE NUMBER 1 PENCIL.

53150 02 23

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 numerical sequence on answer sheet alternates across from column to column.
3. Use only medium sharp #1 black lead pencil for marking answer sheet.
4. Circle the correct answer in this test booklet. 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 with a #1 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.

166

Multiple Choice

Chapter 1

1. (002) Which of the four main parts of the radial drill press is the most useful for large work?

   a. Arm.
   b. Head.
   c. Base.
   d. Column.

2. (002) Of the workholding devices listed below, which is the most commonly used?

   a. V-blocks
   b. Vises.
   c. Clamps.
   d. Straps.

3. (003) Drills larger than ½ inch are usually made with which type of shank?

   a. Straight shank.
   b. Taper shank.
   c. Flatted shank.
   d. Combination straight and taper shank.

4. (004) The cutting edge angle (included angle) used for general purpose drilling carbon and alloy steels is

   a. 39°.
   b. 62½°.
   c. 118°.
   d. 125°.

5. (005) Information about the type, viscosity, and amount of oil to use when you need to lubricate the drill press should be obtained from

   a. TOs.
   b. the *Machinery's Handbook*.
   c. MIL standards.
   d. the manufacturer's specifications.

Chapter 2

6. (006) The most practical method of machining several small pieces of the same shape and size is to use a

   a. forming tool.
   b. radius rod.
   c. hand manipulation.
   d. template and pointer.

7. (006) When you use the radius rod for form turning, the length of the rod should be equal to

   a. one-fourth the radius to be turned.
   b. one-half the radius to be turned.
   c. the same as the radius to be turned.
   d. twice the radius to be turned.

8. (007-008) Which of the following tapers would you be most likely to find in the [illegible]

   a. Mosre taper.
   b. Jarno taper.
   c. Brown and Sharpe taper.
   d. American [illegible]

172

9. (009) Which of the methods would be best to use for machining a short steep taper?

a. A taper attachment.
b. The compound rest.
c. An offset tailstock.
d. An offset tailstock and taper attachment.

10. (009) You are given a job that requires you to make a shaft with a tapered section. You are given the large diameter of 2.250 inches, the small diameter of 1.000 inch, and the length of taper of 3 inches. What is the taper per inch if $TPI = \frac{LD - SD}{L \text{ of } T}$?

a. 0.6066 inch.
b. 0.5666 inch.
c. 0.4166 inch.
d. 0.3666 inch.

11. (013) It is more desirable to machine a taper with the taper attachment than with offset method because

a. the cutting tool does not have to be on center.
b. the attachment can be set up more easily.
c. duplicate parts can be turned with varying lengths without any effect on the taper.
d. with the taper attachment method of taper turning there is no lost motion.

12. (013–014) You are to machine a number 2 Morse taper, having a taper per inch of 0.04995, with a taper attachment. Before the taper attachment can be set, the TPI must be converted to a taper per foot. Using the formula TPF = TPI X 12, what would the taper per foot of a number 2 Morse taper be?

a. 0.4668.
b. 0.5994.
c. 0.6998.
d. 0.7408.

13. (015) The surface feet per minute of a toolpost grinding wheel is controled by the

a. drive and wheel pulleys.
b. types of grinding wheels.
c. belts on the grinder.
d. rpm of the work.

14. (017) What procedure should be used to part large diameter work when there is danger of tool binding?

a. Use increased amounts of coolant.
b. Decrease the speed.
c. Use the step-parting method.
d. Increase the feed.

15. (018) You may use phosphorus bronze wire rather than music wire to manufacture a spring when

a. steel wire would corrode rapidly.
b. it is used near high temperatures.
c. more strength is needed.
d. the cost of material is an important factor.

16. [illegible] on the quick change gearbox in winding a constant spring should be

a. one-half the wire diameter.
b. the diameter of the wire

c. twice the wire diameter.
d. the desired spacing between the coils.

17. (018, 019) In filing work in a lathe, the file tip should point toward the

a. [illegible] at about 20° angle.
b. headstock at about a 20° angle.
c. [illegible]
d. headstock at about a [illegible] angle.

18. (019) The amount of stock that is normally left for filing is

a. 0.002 inch to 0.005 inch.
b. 0.005 inch to 0.010 inch.
c. 0.010 inch to 0.015 inch.
d. 0.015 inch to 0.020 inch.

19. (019) The primary purpose of polishing is to

a. reduce the dimensions of the work.
b. improve the finish of the work.
c. aid in accurately fitting mating parts.
d. remove high spots from the work.

20. (019) The purpose of moving the abrasive cloth slowly back and forth along the work in polishing is to prevent the

a. abrasive cloth from gouging the work.
b. generation of excessive heat.
c. removal of too much metal.
d. formation of polishing rings on the work surface.

21. (020) At what speed should knurling be done?

a. Half the finish turning speed.
b. The same as the rough turning speed.
c. The highest speed that shifting the spindle into reverse permits.
d. The highest speed that shifting the feed into reverse permits.

22. (021) What is the purpose of the center rest when equipped with an overarm?

a. Make vertical adjustment of the work easier.
b. Clearance between the jaw and work is more closely controlled.
c. It enables duplicate pieces to be removed and replaced without jaw readjustment.
d. It compensates for slight irregularities on the surface of the work.

23. (021) The lathe attachment that would be most useful in the limited production of parts is the

a. thread-chasing dial.
b. center rest.
c. cathead.
d. turret attachment.

24. (022) When the follower rest is set up, you mount it on the

a. cathead.
b. lathe ways.
c. carriage.
d. compound rest.

25. (022 023) A lathe attachment that is used to accurately space grooves or turn work to a shoulder is the

a. follower rest.
b. thread-chasing dial.
c. micrometer carriage stop.
d. micrometer cross-slide stop.

26. (023) One of the principal uses of square threads is for

a. jackscrews.
b. water valve stems.
c. vise screws.
d. clamp screws.

27. (024) What type of thread will permit compensation for wear by seating split nuts deeper in the thread groove?

a. Buttress.
b. Straight.
c. Pipe.
d. Acme.

28. (025) Which of the methods listed below would be most desirable in machining a 60° multiple thread?

a. Setting compound parallel to the ways.
b. Using a multiple drive plate.
c. Thread-chasing dial.
d. Using the stud and box gear breakup.

29. (027) You are to cut a 3-millimeter lead thread on a bolt using an American-made lathe with a lead screw having 6 threads per inch. If you use the following formula, number teeth on driving and driven gear = $\frac{6}{\frac{25.4}{3}}$, what will be the number of teeth on the driving gear and on the driven gear to obtain a 3-millimeter lead?

a. Driving 100; driven 127.
b. Driving 90: driven 127.
c. Driving 80; driven 127.
d. Driving 70; driven 127.

30. (028) Headstock spindle bearings on the lathe should be adjusted only when

a. lubricating oil is changed in the headstock.
b. the bearings are to be repacked.
c. the lathe has had 6 months of operating time.
d. it is evident that the trouble is in the bearings.

31. (028) A lathe drive spindle clutch is adjusted properly when the clutch

a. adjusting collar is tightened securely.
b. lever snaps in and out of engagement.
c. will slip under heavy loads
d. collar pin slips into place.

32. (028) Proper adjustment of lathe end gearing is made when

a. the gears are tightly meshed.
b. the teeth barely contact each other.
c. a clattering noise is heard at high speed.
d. smooth action is obtained.

Chapter 3

33. (029) The milling machine operation which is best adapted for milling several grooves in the end of a square bar is

a. straddle milling.	c. gang milling.
b. slotting.	d. form milling.

34. (029–030) When you machine an internal hexagon on the milling machine, the slotting tool should be ground to a width equal to

a. one-half the distance across the flats.
b. one-half the length of one side.
c. one-fourth the distance across the flats.
d. the length of one side of the hexagon.

35. (030) Which of the following methods of manufacturing a cam is preferred when a high degree of accuracy is required?

a. Machine the cam on a milling machine.
b. Grind the cam on a grinding machine.
c. Saw the cam on a contour machine.
d. Machine the cam on a shaper.

36. (030) You are to machine a changing rise cam on a milling machine. Using the formula lead of

$$\text{segment} = \frac{360^\circ}{\text{included angle of segment}} \times \text{rise in segment},$$

what would be the lead of a segment having a rise 0.500 inch in 80°?

a. 1.500.	c. 2.250.
b. 1.750.	d. 2.500.

37. (031) The distance the helix advances in one complete turn around the worm, measured parallel to the axis of the worm, is known as the

a. helix angle.	c. development of the helix.
b. lead of the helix.	d. helical ratio.

38. (032) Which of the following operations is considered to be the most versatile?

a. Plain milling.
b. Angular milling.
c. Face milling.
d. Single-point milling.

39. (032-033) An advantage of performing boring operations on a milling machine is that the

a. boring tool and holder can be supported rigidly.
b. calibrated feed screws may be used to locate holes accurately.
c. work can be held in a vertical position.
d. cutting speeds and feeds can be faster.

40. (033) Which of the following milling attachments would be used to increase the range of the index head to mill larger work?

a. A rack.
b. A raising block.
c. A toolmaker's knee.
d. A right-angle plate.

41. (035) The greatest advantage of helical gears over spur gears is that helical gears

a. wear longer.
b. operate more quietly.
c. can be mounted on shafts that are parallel.
d. operate more smoothly and have greater strength.

42. (035-036) In manufacturing an assembly containing gears, you find that the elimination of end thrust is very important. What type of gear would best serve that purpose?

a. Herringbone gears.
b. Helical gears.
c. Bevel gears.
d. Spur gears.

43. (037-039) The pitch cone angle of a miter gear is

a. 180°.
b. 90°.
c. 45°.
d. 35°.

44. (037-039) Using the formula $D = \frac{N}{P}$, what is the pitch diameter of a bevel gear having 15 teeth, a diametral pitch of 4, and a pitch cone angle of 45°?

a. 2.121 inches.
b. 2.667 inches.
c. 3.000 inches.
d. 3.750 inches.

45. (041) Excessive end play in the spindle of a milling machine would indicate that the

a. draw bolt needs tightening.
b. spindle bearing needs adjusting.
c. arbor support bearing is loose.
d. arbor is not seated properly.

46. (041) The drive clutch on a milling machine should be adjusted or parts replaced when the

a. driving clutch slips under normal load.
b. machine has been used for 6 months.
c. cutter chatters during the cut.
d. starting lever will not stay engaged.

47. (041 042) Excessive backlash in the milling machine table and crossfeed screw may be [illegible] by

a. tightening the calibrated dial.
b. cleaning and reoiling the screw.
c. tightening the adjusting collar nut and the thrust nut.
d. replacing the feed screw.

Chapter 4

48. (043 044) The sequence for machining the sides of a parallel, exclusive of the ends, is to make the

a. rough cut for all sides, and then the finish cuts.
b. rough and finish cuts for the top and bottom, and then for the adjacent sides.
c. rough and finish cuts on one side, its adjacent sides, and then the opposite side.
d. rough and finish cuts on the adjacent sides in succession.

49. (045) The method of machining angular surfaces in which hand feed is always used is swiveling the

a. vise.
b. toolhead.
c. table.
d. adjustable angle plate.

50. (045) When you are to machine a quantity of identical angular surfaces, which of the following would you be most likely to use?

a. vise.
b. The table.
c. A fixture.
d. The toolhead.

51. (046) When you use a roundnose tool to machine between shoulders and the toolholder is set vertically, the clapper box should be positioned

a. centrally.
b. to the left.
c. to the right.
d. away from the surface being machined.

52. (047) If you are [illegible] to manufacture a V-block using the shaper, which [illegible] tool should you use to finish the angular sides of the V-block?

a. A shear tool.
b. A roundnose tool.
c. Any forming tool.
d. A side-finishing tool.

53. (048) The type of toolholder normally used to machine internal keyways is the

a. [illegible]
b. extension toolholder.
c. [illegible] toolholder.
d. universal toolholder.

54. (049) Which of the following methods requires the greatest degree of skill to [illegible] shapes on the shaper?

a. Hand manipulation.
b. [illegible]
c. Form cutting tool.
d. [illegible]

55. (049) With reference to a shaper, which statement listed below is correct?

a. A newly installed shaper is ready for immediate use as neither cleaning nor lubrication is required.
b. A shaper is factory shipped in properly numbered boxes ready for easy assemblying.
c. A shaper can be installed anywhere as no particular consideration to floor space is needed.
d. Before operating a newly installed shaper, the preservative protecting it from rust and corrosion must be removed.

56. (050) All of the following are possible causes of trouble likely to result in damage to the shaper *except*..

a. lack of lubrication.
b. work securely clamped to the table.
c. incorrect speed.
d. toolholder loose in the toolpost.

Chapter 5

5" (051) With reference to a contour machine. it would be necessary to mount the lower saw guide in the lower set of holes when you are sawing angles of more than

a. 10°.	c. 30°.
b. 20°.	d. 40°.

58. (051-052) After laying out the work for three-dimensional sawing, the next step is to

a. plan the sequence of cuts for adequate support for each cut.
b. drill a hole at a corner of the work.
c. make a partial cut to one corner of the work.
d. saw the longest surface first to insure leverage.

59. (052) Titanium is more difficult to saw than stainless steel because titanium is

a. more elastic.	c. a better conductor of heat.
b. a very hard metal.	d. a poor conductor of heat.

60. (052-053) A condition necessary in order to insure accurate stack sawing is

. slower sawing speed than for a solid object.
b. a layout line scribed on each piece.
c. work pieces securely fastened together.
d. work pieces identical in shape.

61. (054) Which of the following contour machine attachments should be used to aid in the sawing operation when great accuracy is required?

a. The magnifying attachment.	c. The mitering attachment.
b. The rip fence.	d. The all-purpose mitering attachment.

62. (054 055) Which of the following is *not* considered a part of the disc cutting attachment?

a. A cylindrical bar.
b. An adjustable sliding arm.
c. A centering pin.
d. A gage rod.

63. (056 057) Good lighting conditions are particulary important when you are locating a contour machine in the machine shop because

a. the operator stands in a different position for sawing than for filing and polishing.
b. operating a contour machine is more dangerous than operating other machine tools.
c. the operator must use his hands to feed the work to the saw band.
d. more hand manipulation is required in contour machining operations.

64. (057) Use a lubrication chart to insure that all parts of the contour machine will be lubricated as recommended by the

a. supervisor.
b. maintenance shop.
c. manufacturer.
d. section foreman.

65. (057 058) The corrective action to take for drive motor V-belt slippage on a contour machine is to

a. replace the V-belt.
b. reposition the drive motor on its mounting base.
c. tighten the nut on the base of the equalizer spring.
d. increase the amount of motor dead weight on the drive belt.

Chapter 6

66. (059) The work and the wheel are rotated in opposite directions in order to

a. eliminate chatter.
b. avoid climbing by the grinding wheel.
c. retard wheel wear.
d. provide a shear-type cutting action.

67. (059) If a center lapping machine is unavailable, which of the following could be used to the greatest advantage?

a. A piece of hard wood and lapping compound.
b. A lathe center and lapping compound.
c. A lathe center and emery paper.
d. A piece of brass and lapping compound.

68. (061–062) To correctly grind a conical taper, the wheel axis should be

a. above the work axis.
b. below the work axis.
c. at an angle to the axis of the work.
d. at the same height as the work axis.

69 (062) When you are face grinding work that must have two or more other surfaces ground, you would normally use the

a. cup wheel method.
b. straight wheel method.
c. angular wheel method.
d. dish wheel method.

70. (062) Which type of grinding listed below would be required to grind the spline drive on the end of an axle shaft?

a. Cylindrical.
b. Internal.
c. Face.
d. Form.

71. (063) The wheel feed during surface grinding takes place at

a. one end of the table movement.
b. both ends of the table movement.
c. one end of the wheel movement.
d. both ends of the wheel movement.

72. (064-065) The work and the grinding wheel are rotated in the same direction in cylindrically grinding cutting tools when there is a need to

a. improve the cutting action of the wheel.
b. improve the finish.
c. produce a slight amount of clearance.
d. retard wheel wear caused by shape edges.

73. (065-066) Using the formula, clearance angle times cutter diameter times 0.0088, what would be the tool rest setting for a 3-inch diameter cutter with a clearance angle of 4°?

a. 0.10056 inch.
b. 0.1056 inch.
c. 0.207 inch.
d. 0.2106 inch.

74. (066) When would the tooth rest be fastened to the table and *not* to the wheelhead?

a. To grind helical-tooth cutters.
b. To grind straight-tooth cutters.
c. To grind helical cutters with a universal workhead.
d. The tooth must always be mounted on the wheelhead.

75. (067) If you get a job that requires the grinding of a compound angle and the only machine available for use is a plain tool and cutter grinder, you can grind the required angles by using the

a. surface grinding attachment.
b. internal grinding attachment.
c. radial grinding attachment.
d. toolpost grinder.

76. (068) Proper gib adjustment in the sliding members of grinding machines is highly important because

a. it insures accuracy and prevents wear.
b. it allows proper lubrication of moving parts.
c. strain on the feeding mechanism is eliminated.
d. less power is required to run the machine.

53150 03 7208

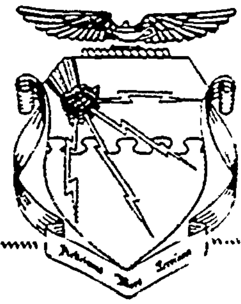
CDC 53150

MACHINIST

(AFSC 53150)

Volume 3

Tool Design and Fabrication



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 machinist skill level. The emphasis will be on broadening your technical knowledge and preparing you to perform the duties of a machinist.

Note the chapter titles on the contents page for Volume 3. Chapter 1 covers tool design practices; Chapter 2, special tool design and fabrication; Chapter 3, jig and fixture design and fabrication; Chapter 4, die design and fabrication; and Chapter 5, fitting and assembly. 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.

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 (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 April 1972.

179

CONTENTS

184

180

Tool Design Practices

IF YOU HAVE ever watched a skilled surgeon operate, you noted that he used each tool strictly for its intended purpose. As a skilled machinist, you too must use each tool for its intended purpose. In your career field you have one advantage that the surgeon doesn't have. You have the capability to change the design of a tool from time to time as the need arises. In order to fully understand how you can use each tool to its maximum, you must know how to use freehand sketching, tolerancing, and dimensioning. Other information you need is how to use standard parts and fabricating processes, and the relation of part design to heat treatment.

1. Freehand Sketching, Tolerancing, and Dimensioning

1-1. In this section we will discuss freehand sketching, the tools you need to do freehand sketching, and the techniques used in developing freehand sketches. We will also discuss the tolerancing system and the dimensioning of drawings.

1-2. **Freehand Sketching.** In CDC 53130 you learned some basic information about blueprints. That information applies to freehand sketches as well as to the mechanical drawings from which blueprints are made. To include information on a sketch or omit it is the decision of the person who makes the sketch. He must include as much information as he feels is necessary to serve his purpose. The information, such as dimensions or notes, that he supplies should conform to good drafting standards and practice. Mechanical drawings are made with mechanical devices, such as the pencil compass, triangle, and T-square. A sketch is usually considered to be freehand, although in practice it is often made on squared paper or with the help of a rule and pencil compass. Usually a sketch is made of an existing object, but it can also be an "idea" sketch of something only thought about, or a combination of both. It can be drawn pictorially, so that it actually looks like the object or it can be an orthographic sketch of the object with different views, usually front, top, and side. It can be either an assembly sketch or a detail sketch. An assembly sketch, as the name implies, shows two or more parts fastened together to form a unit. A detail sketch shows one single part of an assembly in detail.

1-3. *Freehand sketching tools.* Some of the value in freehand sketching, in addition to the fact that it is an excellent way to present your ideas to someone else, lies in the fact that so few tools are necessary. If you have a stub of a soft pencil (HB or F) and a scrap of paper handy, you are ready to go. However, a pencil long enough to permit a relaxed but stable grip improves your sketching. For most sketching, you hold the pencil exactly as you do when you are writing. If you are sketching a circle, you may find it easier with the pencil below your hand and held against your four fingers with your thumb. If erasing is needed, the eraser at the end of some pencils is, of course, convenient and satisfactory for limited use. The soft end of a pencil-and-ink eraser, however, is better, and artgum and pink pearl erasers are best. They do a cleaner job of removing pencil lines from paper.

1-4. If you need to use a pencil compass, the inexpensive kind costing about a quarter at stationery stores is all you need. Almost any kind or size of rule can be used as a straightedge. As your ability to sketch improves, you may find that you use the compass and straightedge less and less until you no longer need them to produce neat and effective sketches quickly. This is the ideal situation, which some people reach before others.

1-5. Just as preparing rough sketches without instruments saves time, the use of cross-section paper also saves time. Cross-section paper, which is usually ruled in one-inch squares, is especially useful when you sketch to scale. These squares can then be subdivided into one-eighth or one-tenth inch squares. Note that the person who made the sketch in figure 1 has indicated that each square represents one-fourth inch. A specially ruled isometric paper, shown

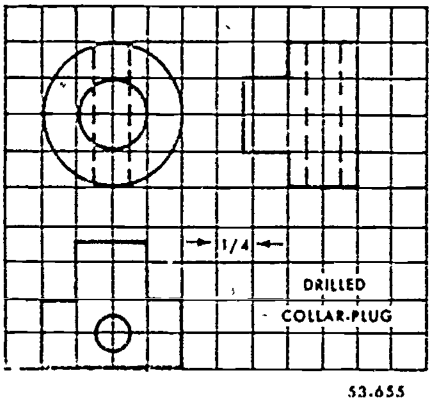


Figure 1. Cross-section paper.

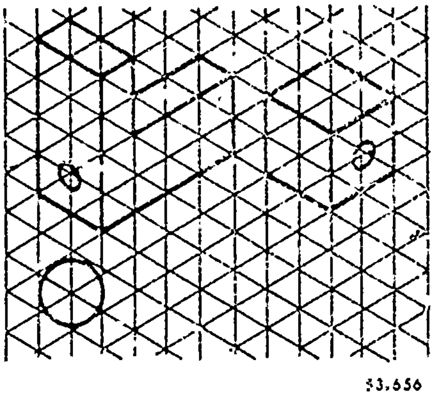


Figure 2. Specially ruled isometric paper.

in figure 2, is used to make isometric and oblique sketches. This type of ruled paper is helpful in making other shapes, such as those shown in figure 2, and is often a great help in developing the ability to sketch well.

1-6. *Technique of sketching.* Hold the pencil from three-quarters to an inch from the point so that you can see what you are doing. Strive for a free and easy movement, rather than a cramped finger and wrist movement. In freehand pencil sketching, draw lines with a series of short strokes, instead of trying to draw each line with one stroke. If you use short strokes, you can better control the direction of the line and the pressure of your pencil on the paper. In sketching lines, place a dot where you want the line to begin and another where you want it to end. In sketching long lines, place one or more dots between the end dots. Then swing your hand in the direction the line should go, and back again a couple of times before you touch the pencil to the paper. In this way, you get the feel of the line. Then use these dots to guide your eye and your hand as you draw the line. Try drawing several light horizontal lines and, after each one is drawn, examine it for straightness, weight, and neatness. If it is too light, you need either a softer pencil or a little more pressure. Vertical lines are usually sketched downward on the paper. The same suggestions for locating dots and using a free movement of the entire arm apply to vertical lines as well as to horizontal lines. Slanting lines can be drawn from either end toward the other.

1-7. With only horizontal lines, vertical lines, and slanting lines, it is possible to make any number of complete and acceptable freehand sketches, depending, of course, on the item or job to be sketched. Keep your freehand sketch neat. To do this, first sketch the line lightly. Lines that are not essential to the drawing can be sketched so lightly that it is not necessary to erase them. Darken essential lines by running the pencil over them with more pressure after you have first drawn them lightly.

1-8. Freehand sketches frequently include many circles and arcs. You don't need to be gifted with artistic talent to draw good circles if you follow these suggestions. In A of figure 3, observe how the pencil is held beneath the four fingers with the thumb. This grip produces a "soft" or "easy" motion for sketching large

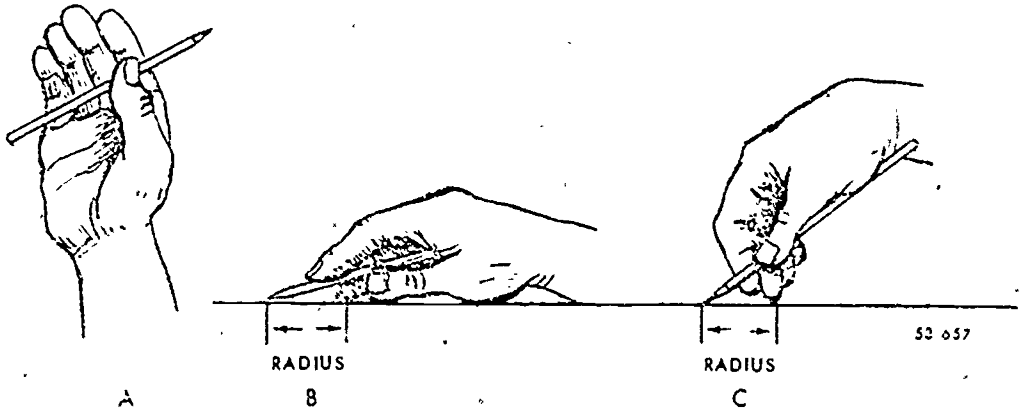


Figure 3. Circles and arcs

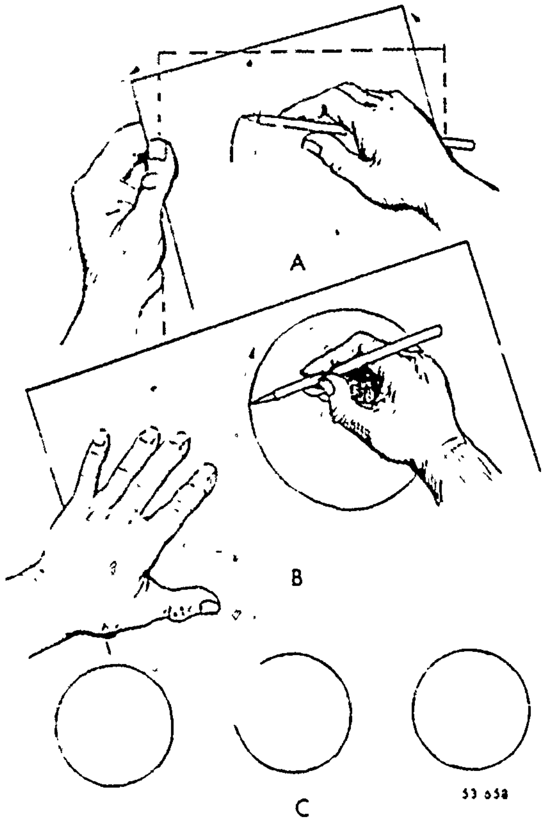


Figure 4 Sketching a circle

circles, as shown in figures 3.B and 3.C. Note in figure 3.B. that the second finger rests at the center of the circle and forms a point about which the pencil lead can swing. The distance from the finger tip to the pencil lead determines the radius of the circle. For smaller circles, a somewhat different grip on the pencil is necessary, as shown in figure 3.C, but the principle is the same. Figure 3 shows the proper way to grip the pencil, figure 4 shows how to draw the circles using these grips

1-9. As shown at A, in figure 4, the first step in sketching either large or small circles with the grips shown in the previous figure is placing your second finger on the paper at the center of the proposed circle. Then, with the pencil lightly touching the paper, use your other hand to rotate the paper to give you a circle that may look like the one in figure 4.B To correct the slight error of closure shown in 4.C, erase a substantial section of the circle and correct it by eye as shown in figure 4.C. You now have complete and round circles, but with a very light line that must be made heavier Do this as shown in 4.B Note that you do not pivot on the second finger during this step Rest your hand on its side, keep it within the circle, and trace over the light line with your hand pivoting naturally at the wrist. As you work around the circle in this way, rotate the paper counterclockwise so that your hand can work in the most natural and easy position. With smaller circles you cannot, of course, work with your hand within the circle, but the same general approach can be used with success.

1-10. **Dimensioning.** In the first years of machine manufacturing, it was the duty of the craftsman to do the designing and manufacturing. If drawings were used, they were design drawings of the assembly type and were scaled by the craftsman to obtain the basic dimensions. The proper functioning of the product depended upon the skill and judgment of the craftsman. It was not necessary or working drawings to carry the detailed size and description of the object to be manufactured. The development of automatic machines and devices for precision measuring made working to a given size easier for the craftsman. His chief duty now became production. The engineer, instead of the craftsman, took over designing and accuracy control. Size control is now exercised by the engineer through working drawings. The craftsman no longer exercises judgment in engineering matters, but is concerned only with properly following the instructions given on drawings. You, the craftsman, must have a thorough understanding of the symbols used in order to carry out these instructions. In your previous study, you were interested in the first step of reading a blueprint, which was to determine the shape of an object. Now we are going to take up the second step, to find the size of an object. This information is given by a definite set of lines and symbols that have been accepted as standard practice in dimensioning.

1-11. A dimension line is a fine solid line that bears a numeral giving the distance indicated. It is fine enough to show a distinct contrast with the object lines. It has a small break near the center for the placing of the numeral. Dimensions should not be placed within the outline of a view, nor should a dimension line end on an object line. Avoid crossing dimension and extension lines to prevent confusion. No line of any kind ever passes through a numeral. Arrowheads on the end of dimension lines assist the eye in determining the extent of the dimension.

1-12 An extension line is a fine solid line that "extends" from a point on the object to which the dimension refers. It should be the same width as a dimension line Leave a space of about 1/16 inch at the point where the extension line would join the object, to clearly distinguish the extension line from the object line. Extension lines should extend about 1/8 inch beyond the points of the arrowheads.

1-13. In the following paragraphs, we will discuss types of dimensioning, tolerance, and allowance. The three types of dimensions are detail, position, and overall. A detail dimension shows one length or dimension necessary to express the size of the structure. A position dimension locates the centers of circles or radii necessary to fabricate structures to exact dimensions. An overall dimension is a total dimension, to give the entire length, height, and width of an object or structure. It is generally a summation of smaller dimensions, and is placed on the outside of detail and position dimensions.

1-14. *Tolerance.* Mass production has brought about the need for interchangeable parts. Mating parts may be manufactured and assembled in entirely different factories. To make parts that can be interchanged, sizes must be specified in such a way that machine operators in widely separated shops can produce parts that are interchangeable. This would be a simple task if it were possible to manufacture parts exactly to the dimensions intended in the design. It is the problem of the designer to specify the allowable amount of error that can exist and still permit the parts to function satisfactorily.

1-15. "Tolerance" is the amount of variation permitted in the dimension or surface of machine parts. Tolerances are stated in several ways. They may be stated as limits. For example, a piece is to be made 1.950 inches long with a maximum dimension .002 inch larger than 1.950 inches with a minimum dimension .003 inch less than 1.950 inches. The extreme dimensions are indicated by $1.950 \begin{smallmatrix}+.002\\-.003\end{smallmatrix}$ The maximum dimension is 1.952 inches, and the minimum dimension is 1.947 inches.

1-16. "Unilateral tolerance" is related to the basic size or dimension in one direction only. For example, if the size of a shaft is 1.000 inch and the tolerance is expressed as 1.000 +.001 and −.00, this is a unilateral tolerance. If the tolerance is expressed as partly plus and partly minus, it is a "bilateral tolerance." The size of the shaft stated as 1.000 inch $\begin{smallmatrix}+.001\\-.001\end{smallmatrix}$ because the total tolerance is given in two directions. When you use unilateral tolerance, use one of the three following methods to express it.

a. Specify limiting dimensions

Hole size 3.250, 3.252

Shaft size 3.249, 3.247

b. Specify one limiting size

Hole size 3.250 + .002, − .000
Shaft size 3.249 + .000 − .002

c. Specify nominal size for both showing both allowance and tolerance

Hole size 3¼ + .002 − .000
Shaft size 3¼ − .001, − .003

Bilateral tolerance should be specified with a plus and minus tolerance, usually of an equal amount.

1-17. *Allowance.* Allowance is an intentional difference in the dimensions of mating parts and may be positive or negative. The allowance is positive if there is a clearance between the external part and the internal part. The allowance is negative if the internal part is larger than the external part.

2. Use of Standard Parts and Fabricating Processes

2-1. You should, whenever possible, use standard fastening devices and bearings in the construction of any item that you design. Standard parts are usually readily available. Therefore, the number of parts to manufacture is reduced. The following information will help you select the proper types of fastening devices and bearings for manufacturing the tool that you are designing.

2-2. **Fastening Devices.** There are many types of fastening devices. The most common devices for fastening parts together are bolts, screws, nuts, and pins.

2-3. *Bolts.* Bolts are used with nuts and threaded holes to fasten parts together. Some of the more common types of bolts are as follows

a. Aircraft bolts are specifically designed to be used on aircraft. Markings on the bolthead indicate the type of material that the bolt is made of and whether or not it is a special or a close tolerance bolt. Figure 5 shows the markings on the head of an aircraft bolt and typical dimensions and tolerances.

b. Capscrews are available in diameters of 1/4 inch to 1 inch and in lengths of 1/2 inch to 6 inches. They are made of steel or brass. The longer capscrews can be used with nuts, but they are usually used to fasten parts to other parts that have threaded holes.

c. Socket head capscrews are used for the same purposes as common capscrews. However, instead of hexagonal, the head of a socket head capscrew is round and has a hexagonal socket. Hexagonal or Allen wrenches are used to tighten and loosen socket head capscrews. The heads normally fit into counterbored holes in the part being fastened.

d. Setscrews are used to fasten parts, such as pulleys, gears, etc., to shafts. They are usually

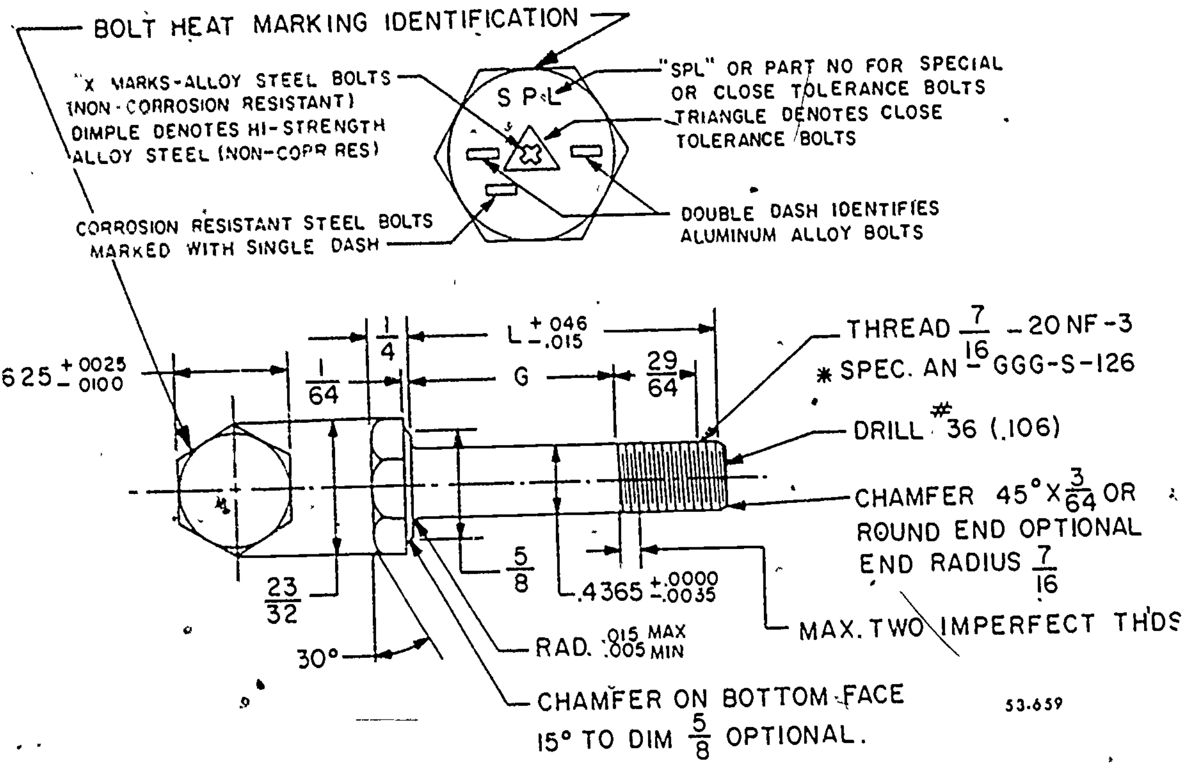


Figure 5. Bolt markings

hardened Setscrews have slotted heads, square heads, or socket heads.

e Machine bolts are available in a variety of sizes. They are usually used with nuts to fasten parts together They have either square or hexagonal heads and are made of steel or brass.

f. Carriage bolts have round heads above a square section. The square section pulls into the wood and keeps the bolt from turning when the nut is tightened.

g. Stove bolts have slotted round or flat heads, and are available in diameters of 1/8 inch to 1/2 inch and in lengths of 3/8 inch to 6 inches.

h. Studs are bolts that are threaded on both ends. One end, usually the coarse-threaded end, is threaded into the parent part of a mechanism and another part is fastened to the fine-threaded end by means of a nut

2-4 *Screws.* The screw is a common form of threaded fastening device. Screws are usually identified by the shape of the screwhead. Sheet metal screws are made of hardened steel and are threaded directly into holes drilled in sheet metal. The heads of sheet metal screws are usually round or flat. Machine screws are made of steel, brass, or aluminum alloy. Machine screws are designed to be threaded into a tapped hole or to be used with a nut. Some of the types of machine screws are given below:

a. Fillister head screws are commonly used in light mechanism assemblies and are available with or without a drilled head.

b. Flat head screws are used in countersunk holes. They provide a surface face free of projections and are used for streamlining.

c. Round head, button head, and washer head screws are used where the projection head is not objectionable. The washer head has a large contact area on the head.

d. Phillips screws and Reed and Prince screws have cross-slot type heads. A Phillips screwdriver must be used with Phillips screws, and a Reed and Prince screwdriver is used with Reed and Prince screws.

2-5. *Nuts.* A nut is threaded internally to mate with the threads of a bolt, screw, or stud. The threads of a nut must correspond to the external threads to eliminate jammed or stripped threads. Nuts are made of steel, aluminum alloy, or brass.

a. Plain aircraft, or hexagon, nuts have a limited use in aircraft and require a locking or safety device, such as a checknut or lockwasher

b. Checknuts are thin nuts used as a locking device for nuts, setscrews, threaded rod ends, and other devices.

c. Castle nuts are used with bolts that have a drilled hole in the threaded end to receive a cotter pin or lock wire for safetying

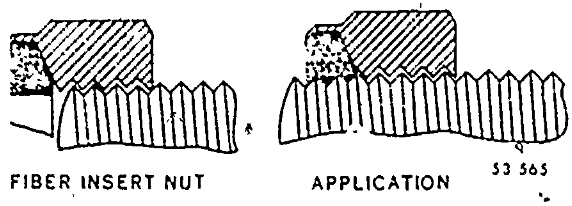


Figure 6. Fiber insert nut

d. Shear nuts are used with such devices as clevis bolts, in which there is no tension on the bolt.

e Plain and slotted engine nuts are high-strength nuts and are designed for use on engines. They require a locking device.

f. Locknuts are used for locking all external engine nuts, except those used with safety wire or a cotter pin.

g. Wingnuts are used where the desired tightness can be obtained with the fingers.

h. Fiber insert nuts are the most common of the self-locking nuts. They have a fiber insert that is smaller in diameter than the bolt. When the nut is threaded on a bolt, the fiber insert expands, as shown in figure 6, and tightens the nut on the bolt. The fiber insert nut should not be used in a location subjected to rotational movement or near excessive heat (above 250° F.).

2-6. *Pins.* The three types of pins, shown in figure 7, that are most frequently used in assemblies are flat head, taper (plain and threaded), and straight.

a. Flat head pins, commonly called clevis pins, are made of steel and are used on tie rod terminals or on secondary controls. The pin is safetied with a cotter pin.

b. Taper pins are used to pin or to connect two parts (such as the hub of a part to a shaft) where it is essential to eliminate any looseness or play. They are made of steel and some taper pins have a hole in the large end to receive safety wire. The common plain taper pin does not have a drilled hole in the large end. Another common use of the taper pin is to aline parts in an assembly. Threaded taper pins are used for the same purposes as common taper pins, when a more positive means of securing the pin is required. Threaded taper pins are frequently used/for fastening aircraft parts. A taper pin washer and a nut are used with threaded taper pins.

c. Straight pins are used primarily to fasten or secure mating parts and to aline parts.

2-7. **Bearings.** Bearings are supports or guides for the moving parts of machines, engines, and shafting. There are two types of bearings: plain and antifriction.

a. Plain bearings. Plain bearings can be made of various metals, but are quite frequently made of bronze. ~~The material~~ must be hard enough to wear well, yet softer than the material they contact.

b. Antifriction bearings. 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 consist of an outer and an inner race, separated by freely moving balls. They are primarily intended to resist radial thrust.

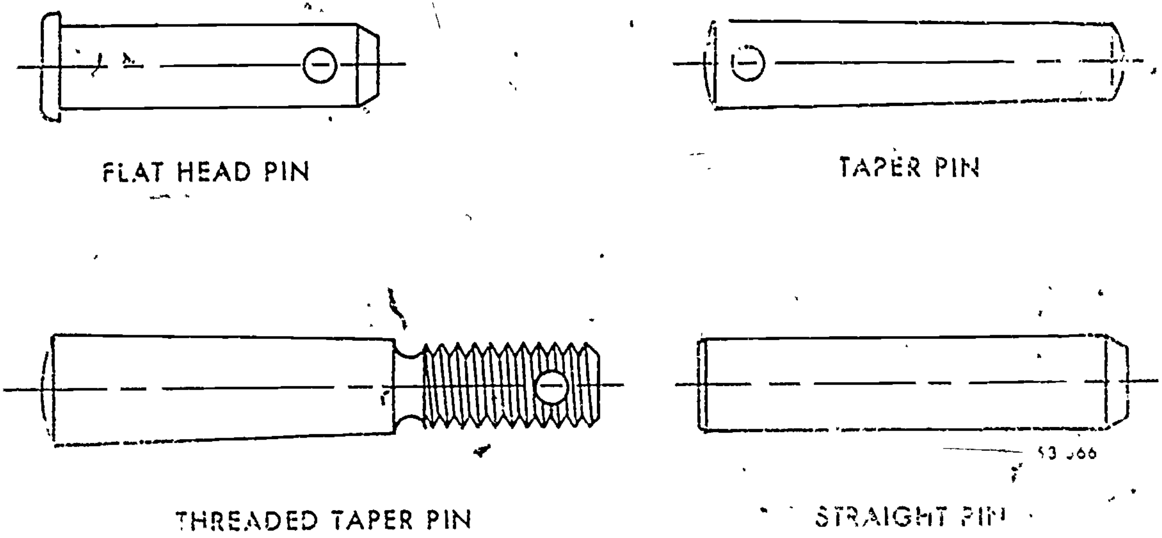


Figure 7 Pins

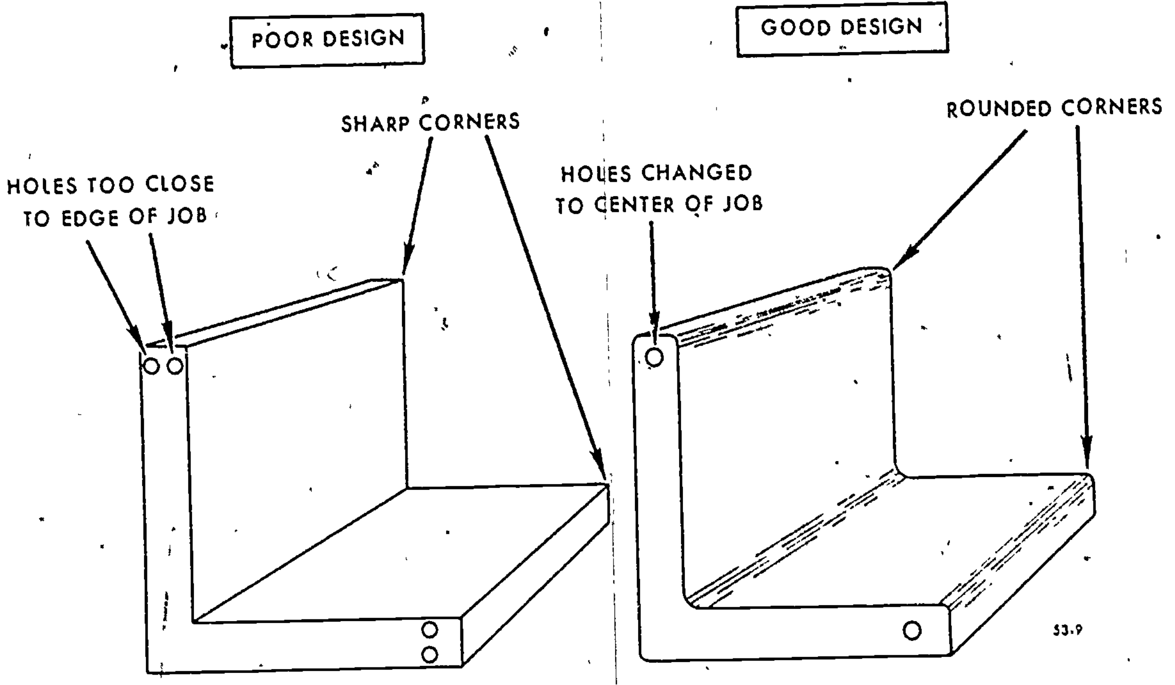


Figure 8. Corner and fillet design.

(2) Roller bearings are similar to ball bearings except that rollers are used instead of balls. The rollers are cylindrical, tapered, or barrel-shaped. Bearings with rollers that are relatively long compared to the roller diameter are needle bearings. Tapered roller bearings are usually used to combat extremely high end thurst.

NOTE: Additional information about plain and antifriction bearings can be found in machinists' publications such as the *Machinery's Handbook*.

3. Design Related to Heat Treatment.

3-1. Heat treating is a series of operations involving the heating and cooling of metals in the solid state. Its purpose is to change a metal's mechanical property or combination of mechanical properties so that the metal will be more useful, serviceable, and safe for a definite purpose. Heat treating can make a metal harder, stronger, and more resistant to impact. Heat treating can also make a metal softer and more ductile. No one heat-treating operation can produce all of these characteristics. In fact, some properties are often improved at the expense of others. As a result of hardening, for example, a metal becomes brittle.

3-2. As a machinist, you should know the effects of heat treating and the results that you can expect from each heat-treating operation. In many cases, the metal and the heat treatment are specified. Make no substitutions of materials unless the substitution is authorized by technical orders or other authoritative sources. In other cases, you may have to specify the heat-treating process. If so, you must consider the use to which the part will be put and the mechanical properties required. If you overlook the design of the part or the metal of which it is made, the part may warp badly or crack, or the required mechanical properties may not be obtained during heat treatment.

3-3. The rate of cooling is affected not only by the difference in mass or size but also by the shape and the surface finish. When a part is removed from a furnace to be quenched, even though it is uniform in cross section, its shape may cause uneven cooling. Particular attention should be given to contour to avoid such abnormalties. Avoid protruding corners on parts, such as angle plates and V-blocks, whenever possible, by slightly rounding the corners and the edges. Design affects the serviceability of a tool or machined part. The failure of a part, in most cases, can be traced to improper design or to improper heat-treating procedures. Figures 8 and 9 illustrates good and poor part design. Improper design or heat treatment of a part can cause cracking, warping, or internal strains, which render a part unfit for service.

3-4. Two forces that may combine to break steel apart are the residual stresses set up during machining operations and the heat treatment of the part. There is also the force applied to the

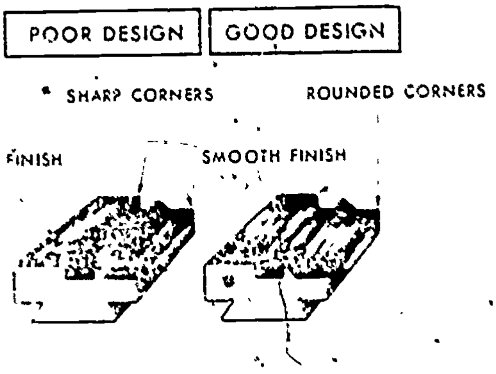


Figure 9. Corner and surface design.

part when it is put into service. Internal stress in a part can equal as much as 90 percent of its strength and, therefore, can cause it to break under a comparatively small load.

3-5. The most serious cause of residual stress is the hardening operation. The right method of cooling depends not only upon the chemical makeup of the metal but also upon the size and shape of the part. The failure of a part is usually caused by the use of a design without any consideration for the heat treatment or for the use of the part. Designs with abrupt changes in mass or size, will, when quenched, set up massive stresses in the part and will cause warping. Other parts even if they are heat-treated satisfactorily, will fail in use if they are improperly designed.

3-6. Errors in design not only cause residual stress in a part but they can also cause a concentration of stress, which, when combined with service stresses (the stresses caused by an actual load in use), can cause the part to fail. Some designs are almost impossible to harden because of the part's change in size. Sharp angles and uneven balance of mass concentrate stress. A piece, properly designed in relation to heat treatment, heats and cools evenly. Perfection in this respect is unattainable, because the surface of a well-balanced part cools faster than the center. A machinist should design a part so that it will cool as evenly as possible and will have a low concentration of stress under load.

Special Tool Design and Fabrication

THE DESIGN and fabrication of practical tools are important tasks of a machinist. Some tools are simple and are quite easy to develop. Others are complicated and require more careful development. A tool can be defined as any device that is capable of working a material into a desired shape, holding a material while it is being worked, or measuring it after the work is completed. In the Air Force, as well as in industry, tool design and fabrication are important processes that are necessary for the economical production of parts. In this chapter we will discuss certain necessary factors in the design and fabrication of tools.

4. Design of Hand and Cutting Tools

4-1. "Tool design" has become recognized as meaning the design of special tools for the economical production of parts. In the Air Force as well as in industry, certain factors that influence the design and fabrication of special tools must be considered In this section we will discuss the general aspects of tool design and fabrication.

4-2. In every operation the tools must be adapted to the material, the machine, other tools, and the operator. When you choose a machine for an operation, its capacity and range are important items to consider. After you have selected the machine, the tools that you use on it must perform well on that particular machine. For example, the length to be machined must fall within the travel of the table, and provision must be made for mounting the tools. The tools must have enough bulk and backing to prevent their destruction during a machining operation. We will apply these ideas as they relate to (1) tool planning and (2) types of tools.

4-3. **Tool Planning.** Tool planning is concerned with the processes to use, the operations required, and the locating points for each operation. It also involves the dimensions and tolerances to be held and the machine tools needed to make the part. Tool planning must be thorough and usually involves the following three steps:

(1) Determine what the tool must do. This step tells you what type of tool you need, what operation it is to perform, and how it must be held or fastened to the machine.

(2) Select or invent a device to meet the requirements. Your knowledge of the tools and holding devices that are available from standard sources will be of great value at this point. If there are none available, then you must invent your own tools for this purpose.

(3) Construct the device to perform the required task most efficiently. When you have decided upon a general form of tool, you must arrange and position it to do a specific job. Consider these main ideas at this time: economy, kind of operation, accuracy, and safety.

4-4. **Types of Tools.** Machine tools are powered tools, not normally held by the hand. The true machine tools are the lathe, milling machine, drill press, planer, and shaper. Some modifications of the basic types, such as the turret lathe and boring mill, are accepted in the same class. Other tools supplement the basic machine tools. It is not uncommon for the tool designer to design specific tools or to modify standard ones. We will discuss some of the more common tools that can be used as standard tools or can be modified.

4-5. *Jigs.* A jig is a device for holding and locating a workpiece, and for guiding the tool that performs an operation. Though jigs are usually movable, they are sometimes fastened in alinement with the cutting tool. They are frequently designated by their type of operation, such as drilling or boring, but are more often classified according to their construction, such as template, plate, angle plate, and ring.

4-6. *Fixtures.* A fixture is a device for holding work while an operation is being performed. Fixtures are usually classified according to the kind of operations for which they are used. Some of the more common types are the lathe, turret lathe, and milling machine.

There are often many fixtures for each job or operation, with each fixture of a different construction. Fixtures are frequently given a name descriptive of their design or features, such as vise, faceplate, and indexing fixtures.

4-7. *Dies.* There are many kinds of dies, but the most common are those used for cutting and shaping sheet metal. These dies are called press dies to distinguish them from others, such as forging and moulding dies. Dies that cut sheet metal include stamping dies for blanking and piercing, for cutoff and trimming, and for shaving. Dies used for shaping metal include those for drawing, forming and bending. At times, several operations are performed with one die. These dies are of the progressive, compound, or indexing design.

4-8. *Cutting tools.* Cutting tools are divided into two classes with respect to design. One class includes single point tools, such as those used in turning, facing, boring, shaping, and planing. The second class includes multitooth cutters, such as drills, reamers, milling cutters, and saws. Cutting tools are usually named for the operation they perform, such as drill, lathe tool, and milling cutter.

4-9. *Gages.* Gages are devices for investigating the dimensional fitness of a part. They are usually classified into three general groups. Fixed-size gages are used in production for both large and small work. Indicating gages are usually used in an inspection department. Combination gages are used to check more than one dimension on a workpiece.

5. Design of Special Gages

5-1. Modern machine work requires extensive use of gages for shop inspections and references. In this section, more detailed information about the design of gages will be discussed. A gage is defined as a device for determining whether or not the dimensions of a manufactured part are within specified limits.

5-2. **Steels Used in Gages.** The types of steels used for gages are machine steel, plain carbon steel, and special alloy steels, with machine steel used most often. The carbon content of machine steel for gages usually ranges from 0.15 percent to 0.25 percent, although it may be as high as 0.50 percent, especially for ring or plug gages. A 0.20 percent carbon steel, containing 1.00 percent manganese and about 0.05 percent phosphorus and sulphur, is considered very satisfactory. This general class of steel is extensively used for snap gages. Steel for gages should not contain silicon, since this causes warping in the hardening operation. Plug gages, ring gages, and other forms that can be ground easily after hardening are often made of a steel containing about 0.50 percent carbon.

5-3. High carbon steel is sometimes preferred to machine steel because it can be hardened in much less time than is required for carbonizing and hardening machine steel gages. The high carbon steel used for gages usually contains about 0.90 percent carbon. Special alloy steels have been developed, which are adapted to fine gage work partly because changes due to hardening are very slight.

5-4. **Gage Tolerance.** The size of a gage is determined by the tolerance of the part to be gaged. According to the practice of prominent manufacturers of gages, a tolerance of 10 percent of the work tolerance is generally allowed on ordinary working and inspection gages. For example, if you were to make a ring gage to gage a 1.00-inch diameter shaft and the tolerance of the shaft was plus or minus 0.005 inch, the tolerance of the gage would be plus or minus 0.0005 inch. Since the work has a tolerance of plus or minus 0.005 inch, the tolerance would then be 0.0005 inch for both working and inspection gages. This amount is subtracted from the minimum and maximum size of the shaft. There is a difference between the minimum and maximum size of working and inspection gages. The minimum size of the working gage is 10 percent of the tolerance *larger* than the minimum size of the inspection gage. The maximum size of the working gage is made 10 percent of the tolerance *smaller* than the maximum size of the inspection gage. The reason for this difference is that, if the working gage and the inspection gage were made the same size, the working gage, which wears faster, would become larger than the inspection gage.

5-5. **Fixed Gages.** Fixed gages are the most common and are used for both large and small production work. As an Air Force machinist, you are most likely to use this type of gage. Fixed gages are subdivided into these general classes: ring, receiving, plug, pin, snap, thread, and form gages. The following will help you understand the use of these gages.

a. Ring gage. A ring gage has a circular hole, which is ground accurately to the specified size. Ring gages are more often used in pairs. The difference in the hole size 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 can go into the larger gage but can not go into the smaller gage, it is within tolerance. Any part that fits into the smaller gage is too small and is not acceptable. Any part that can not enter the larger ring gage is too large and must be re-machined.

b Receiving gage. Receiving gages are similar to ring gages but are used to check the size and contour of noncircular parts. They are used quite extensively to check splined shafts.

c. Plug gage. A plug gage has an outside gaging surface to fit a hole. It may be round, tapered, or irregular in shape. It may have either an integral or a replaceable handle.

d. Pin gage. Pin gages are used for measuring large holes when a plug gage is too heavy. Place the pin gage lengthwise across the hole, and take the measurement as you would with an inside micrometer. Pin gages may also be used to measure the width of slots and grooves.

e. Snap gages. Snap gages have inside measuring surfaces for checking diameters, length, thickness, and other similar dimensions.

f. Thread gage. Thread gages are made in the form of ring or plug gages. They have a definite limitation for very precise measurement. Single full-form plugs or ring screw gages cannot be used to check all the thread elements. At least three sets of plug and ring gages are needed to check a thread completely. One set checks pitch diameter, the second, the major diameter, and the third, the minor diameter.

g. Form gage. As a rule, the ring and plug gages are used for checking ordinary work. When great accuracy is required, such methods as contour projection, are used. Form gages are specially designed to check the form or contour of a workpiece. An example of a form gage is the gage used for checking lathe tool angles and forms.

6. Fabricating Processes

6-1. The fabrication of tools is probably the most important part of your work as a machinist. After a tool has been designed to perform a certain function, it is extremely important that it be constructed properly. After the tool is fabricated, it must be given a functional tryout to determine whether or not it will perform the job for which it was designed. In this section we will limit our discussion to the more critical areas of tool fabrication. These include (1) tool materials, (2) selection, (3) layout, (4) serviceability of machined parts, and (5) heat treatment.

6-2. **Tool Materials.** The material from which a tool is made is the most critical factor in its fabricating. Tool design engineers, through years of experience and trial and error, have set up standards for fabricating most of the cutting tools, such as lathe tools and milling cutters, and for most of the hand tools. For our purpose in this section, we will center our discussion on punch and die manufacture. The many variables in the design and fabrication of punches and dies make it impossible to recommend specific standards. But there are certain factors that must be considered in the manufacture of all punch and die sets.

6-3. The many materials from which punches and dies can be constructed include zinc alloys, rubber, and thermosetting phenolic and epoxy plastic. Most die sets, however, are made of steel in one of its many forms.

6-4. *Plain carbon steel.* Plain carbon steel is the least expensive of all steels and the most widely used. Its carbon content ranges from about 0.7 to 1.5 percent. This steel is sometimes modified for some uses by the addition of small amounts of chromium and vanadium. Although they are frequently used in the manufacture of tools, carbon steels are prone to be erratic in their response to heat treatment. This is due largely to their differences in grain size and ability to harden. Even steels of the same composition and steels made by the same manufacturer are similarly erratic.

6-5. *Nondeforming tool steel.* Nondeforming tool and die steels are used extensively for intricately shaped tools, where heat-treating distortion must be held to a minimum. These steels generally contain substantial amounts of manganese—about 1.5 to 1.75 percent. The manganese content is lower when chromium and tungsten are added. With the proper proportions of alloying elements, this type of tool steel can possess good hardening quality and resistance to wear.

6-6. *High-speed tool steel.* High-speed steels contain large amounts of tungsten and smaller amounts of chromium, vanadium, and in some cases, molybdenum and cobalt. They are valuable in the manufacture of cutting tools, because they retain hardness and strength at high-operating temperatures. This type of tool steel, when properly heat-treated, is hard and has good strength.

6-7. *Nonferrous tool materials.* Nonferrous tool materials include such metallic elements as tungsten, tantalum, titanium, columbium, and cobalt in their carbide forms. These materials can be used for machine tools for blanking, drawing, shaping, and spinning dies, as well as for many other applications. Tools made of these materials perform beyond the limits of steel tools and are quite suitable for use on hard, abrasive materials and hard alloys. The hardness that can be attained by heat-treating these materials is much higher than can be attained in tool steels by any known method of heat treatment.

6-8. **Selection.** When you are designing or manufacturing a tool, there are several factors that you should consider in selecting the material.

6-9. *Use of the tool.* The use of the tool governs the selection of the material from which it should be made. For example, milling cutters should be made of highly refined tool steel that has been tested for both internal and external defects. Defects in the material used is one of the most important causes of tool breakdown. Use only materials that can withstand the forces to which they will be subjected. Material defects are usually found in the low-grade steels. The manufacturer did not intend for these steels to be used to fabricate tools and machines. Many defects in low-grade steels are caused by the manufacturing process, such as seams, laminations, blowholes, erratic grain size, and uneven tensile strength. With these things in mind, you should select only high-grade materials for such tools as jigs, fixtures, die sets, and cutting tools. High-grade steels are made specifically for items of this type. The manufacturers have been very critical in the refinement of the high-grade tool steels. However, there is a remote possibility that even the best tool steel may have a defect.

6-10. Assume that you are fabricating a die set for a production run of several thousand parts, and you have selected the very highest grade of tool steel. You would be wise to have the material tested for defects by the personnel of a nondestructive inspection laboratory. You can minimize tool breakdown by selecting the proper tool material and having the material tested before you use it.

6-11. *Wear.* Wear causes tools to become inaccurate. You should always use hard, wear-resistant materials in areas subject to wear, such as on jigs, fixtures, and gages. Keep the wearing surfaces as small as possible without sacrificing durability. Small surfaces are also easier to keep clean and are cheaper to build. A good example of the effect of wear is a drill bushing in a drill jig. This type of bushing is subjected to abrasive wear from the revolving drill bit. If the drill bushing is too soft or made of improper material, normal usage will cause it to wear and become inaccurate. You can minimize wear by using a material that can be heat-treated to an extremely hard state.

6-12. *Deflection.* Deflection is usually present in machining operations. It cannot be totally eliminated, but it can be reduced to a minimum. For example, a dull tool causes several times as much deflection as a sharp tool. Forces that cause deflection come from handling, clamping, and cutting actions. When you design and manufacture cutters, jigs, or fixtures, provide ample support for the work. This reduces the problem of deflection to a minimum.

6-13. *Thermal expansion.* Thermal expansion must be taken into consideration especially when you are machining or welding dissimilar metals. For example, aluminum expands about twice as much as steel for every degree of change in temperature. The best way to keep thermal expansion to a minimum is to keep the tools sharp and to use an ample volume of coolant.

6-14. *Load.* The type of load to which a tool will be subjected is a prime consideration. There are two general considerations involved in load. First, you must determine whether machines or tools are capable of withstanding known external loads without failure. Second, you must determine the size and shape of a tool that can withstand the known external load forces.

6-15. **Layout.** Jigs, fixtures, and die sets have two features in common. Each requires accurate layout and accurate machining to insure the production of interchangeable parts. Probably the most difficult task in all layout is the location of holes, especially in the manufacture of a jig or a fixture. When extreme accuracy is desired, it is virtually impossible to do the job with the common hand layout tools. Common layout tools produce measurement errors. If you use dividers for laying out the location of holes, the first error occurs when you try to transfer an exact measurement from a rule to the points of the divider. You cannot make measurements in thousandths of an inch with dividers. If the intersection of center lines is center-punched, another error is likely to occur because of the human element. If you drill a hole with a full sized drill, which you line up as nearly as possible with the center punch mark, a larger error will probably result. These are just a few of the errors that are possible when you use such layout tools as dividers, a steel rule, a scriber, and a center punch. A jig boring machine is ideal for locating, drilling, and boring holes, but this machine is not normally found in an Air Force machine shop. If extreme accuracy is required in the manufacture of tools, you can locate and produce accurate holes by one of several methods. These are some of the more common methods of layout:

6-16. *Toolmaker's buttons.* The use of toolmaker's buttons is a very accurate method for locating holes. These buttons are precision-ground, cylindrical pieces of hardened tool steel with a hole through the center. The ends of the buttons are slightly concave so that the outside circumference bears on the flat surface of the piece being laid out. A convenient size is the 0.500-inch diameter, 0.500-inch length, with a 0.250-inch diameter hole. This size uses a 5-40

192

screw to clamp each button in place.

6-17. *Disc.* For machining holes in small precision work, the three-disc method is sometimes used to aline the respective holes with a lathe spindle. The diameter of each disc is machined to the exact size that will cause the disc centers to coincide with the positions of the holes to be machined when the discs are touching each other. This method requires calculating the respective diameters of the discs before any machining can be done.

6-18. *Dial indicator* The dial indicator method can be used to advantage in locating straight line holes. It is usually used with toolmaker's buttons if the shape of the work permits it to be attached to an angle plate or placed on a surface plate. You must exercise care in handling a dial indicator to assure accuracy in its use. Dial indicators must be calibrated frequently because they are easily damaged.

6-19. *Toolmaker's knee.* The toolmaker's knee affords a rapid and convenient method of setting up angular work on the milling machine, shaper, drill press, or grinder. It is a valuable tool when you are building tools and dies, and when you are machining holes or other surfaces at compound angles.

6-20. **Machining Requirement.** An area of prime consideration in fabrication is the determination of machining requirements. You will constantly be faced with problems of machining methods and procedures. Whether or not you are working from your own or another design, you must determine the machining operations, and how best to perform them. You should first study the design and the parts to be produced. You must fix thoroughly in your mind the function the completed tool is to perform. Before starting fabrication, develop a plan of machining requirements. The following areas to be considered will help you develop your plan.

6-21. *Materials, parts, and tools list.* A material and parts list is usually a part of the design or drawings. However, you should check this list or make a new list to make sure that the material and parts you need are available. Also you need to know what hand and measuring tools are available to you. You can then take action to acquire these materials and tools. Also, if you find that all the listed materials and parts are not available, you may be able to use suitable substitutes. In the absence of the required tools, you may be able to improvise or use alternate methods or techniques.

6-22. *Machining methods and techniques.* This is a vital area of consideration. Economy and quality of workmanship depend often upon the methods and techniques used. As you have probably found, there are often several ways to perform a certain machining operation. Any one of these ways may be sound and good shop practice. But there is usually a best method for performing an operation under certain conditions. For example, drilling, boring, and reaming can be done on a drill press, lathe, or milling machine. You must decide which machine and which technique is best for a given situation. If hole location is not critical and several holes are to be machined, a drill press can be used to advantage. If hole location is critical, you can use the milling machine feed screw dials to great advantage. If large holes are to be bored, recessed, and threaded, you can use the lathe to advantage.

6-23. As far as is practical, group and perform all like machining operations at the same time. This practice keeps work and tooling setup time to a minimum. If you find that you have a number of flat pieces to machine and planing is the best method, perform the planing operation on as many pieces as possible while you are at the shaper. On the other hand, if milling is the best approach, perform as many milling operations as possible while you are at the milling machine. Also, you should perform machining operations on mating parts, in pairs and at the same time, when practical. This technique insures accurate location and alinement when the parts are assembled.

6-24. Your machining requirements plan should be brief, yet complete. It should be legible and understandable to fellow workers and helpers. It should contain notes or references to the publications, such as TOs, manuals, machinist handbooks, etc., that you may need to use. In short, your plan should enable you to accomplish an economical and quality job.

THE PRIMARY PURPOSE in the design and manufacture of jigs and fixtures is to aline the tool and the workpiece properly during machining operations. Jigs and fixtures usually have some type of device for guiding, supporting, clamping, or gaging, to insure the accurate production of parts.

2. The use of jigs and fixtures helps to reduce the cost of parts manufactured in large quantities. Jigs and fixtures are used to great advantage when the interchangeability and accuracy of the finished products are important factors. In some cases, the use of jigs and fixtures is even justified in low or limited production jobs if extreme accuracy can be achieved only by their use. One of their greatest advantages is that relatively unskilled labor can accomplish the job using these special tools.

7. Types, Uses, and Design of Jigs and Fixtures

7-1. Jigs and fixtures are so closely related that the terms are often confused. The term "jig" should be applied to those devices that hold, support, and locate a workpiece while they guide the cutting tool. The term "fixture" should be applied to the devices that hold, support, and locate the workpiece in relation to the cutting tool. They are usually fixed to the machine. In this section we will discuss the types and uses of jigs and fixtures, and some of the factors in their design.

7-2. **Jigs.** As we stated earlier, jigs hold, support, and locate the part to be machined, in addition to guiding the cutting tool. Jigs can be divided into two general classes: drill jigs and boring jigs. Because of limited equipment, drill jigs are used more by the Air Force machinist than boring jigs. For this reason, we will limit our discussion to drill jigs.

7-3. *Template.* The template jig is used for limited production. It is used more for accuracy in layout than for labor or time saving. The template jigs, shown in figure 10, were designed for drilling out three pins that hold a grease seal. These pins could not be drilled out accurately by any other method.

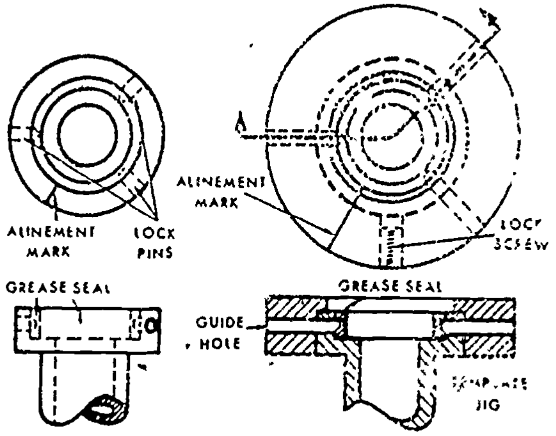


Figure 10 Template jig.

7-4. *Plate.* The plate jig, shown in figure 11, gets its name from the fact that the largest part of it is a plate. The other essential parts are locating pins, drill bushings, and a clamping device. A jig of this kind can be used for drilling holes in a flat workpiece.

7-5. *Channel.* The channel jig, shown in

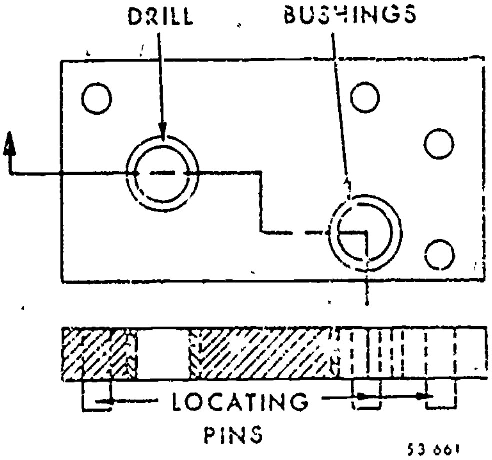


Figure 11 Plate jig

figure 12, is made for parts with a simple symmetrical shape. The channel jig is hollow, and holds the part inside its walls by means of a locking device, such as the lockscrew illustrated

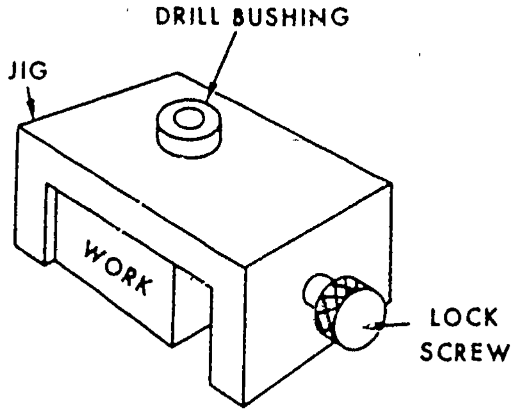


Figure 12 Channel jig

7-6. *Angle plate.* The angle plate jig, shown in figure 13, is especially adaptable for locating and drilling holes for setscrews on such parts as collars, pulleys, and gears. This type of jig can easily be constructed so that it can locate and guide a drill or cutting tool in an angular plane other than vertical.

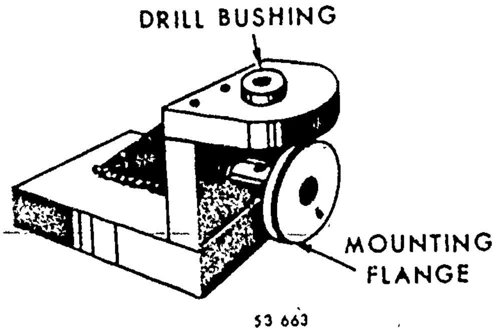


Figure 13 Angle plate jig

7-7 *Closed or box.* Closed or box jigs are designed to completely or partially enclose the part to be machined. They are used on parts in which holes must be drilled from several directions. To firmly support the jig, it must be equipped with four feet or legs. These supports are located on the opposite side of the box from the drilling surfaces. Closed or box jigs sometimes have a leaf or cover, which swings back to allow loading and unloading of the parts. Devices for accurately locating and clamping the workpiece are usually permanently attached to the jig body.

7-8. *Indexing or rotary.* Indexing or rotary jigs are used for drilling holes that must be located at angles to each other. This type of jig may be partially or fully enclosed. By means of locators and clamps, the work is mounted on a swiveling or rotating drum (trunnion) so that the workpiece can be indexed to the proper location in line with a stationary drill bushing. This type of jig may be simple or complex, depending upon the accuracy required and the number of parts to be manufactured. It can also be designed to hold several identical parts and be manually or automatically operated.

7-9. A special type of indexing jig is often built for large parts that are too heavy to handle. These are generally box type jigs, which are mounted on bearings or trunnions. Mounting these jigs on bearings makes it quite easy to move them into position for the drilling operations.

7-10. *Combination.* Combination jigs are used when more than one operation is to be performed on the same hole, such as drilling, reaming, boring, tapping, countersinking, and counterboring. The use of combination jigs is possible because of their slip renewable bushings. After the part is placed in the jig and properly located and clamped, the hole is drilled, and the bushing is removed and replaced by a different bushing for subsequent operations.

7-11. **Types of Jig Bushings.** Jig bushings are made of hardened steel. Since a jig bushing serves as a guide for a drill in locating a hole, the dimensions of the hole in the bushing must be extremely accurate. Standard bushings are available and can be used in production work. A jig is often fabricated for limited production. In this situation you must fabricate the bushing, as well as the jig. There are five main types of jig bushings.

7-12. *Press fit.* Press-fit bushings are permanently pressed into position. Used only for limited production, they are put into simple jigs that are used for only one machining operation, such as drilling. There are two types of press-fit bushings, plain and shoulder. Plain bushings can be set closer together. It is better to use them when their location in a jig requires a flush surface or when the holes in the jig plate are closely spaced. Shoulder bushings are better for general use because there is less danger of their becoming dislodged by the cutting tools.

7-13. *Fixed renewable.* A fixed-renewable bushing fits into an outer sleeve, which is pressed into the jig plate. It is kept in place until it is worn out. Then it is replaced without changing the dimensions of the sleeve in the jig plate.

7-14. *Slip renewable.* The slip-renewable bushing also fits a sleeve. It makes possible the drilling of several holes because it can be moved from hole to hole. Also, bushings of different sizes can be used in the same sleeve to facilitate drilling, reaming, and boring. This type of bushing must be clamped to keep it from rotating with the drill or cutting tool, and from rising from the sleeve. There are many ways of clamping a bushing in place, as shown in figure 14.

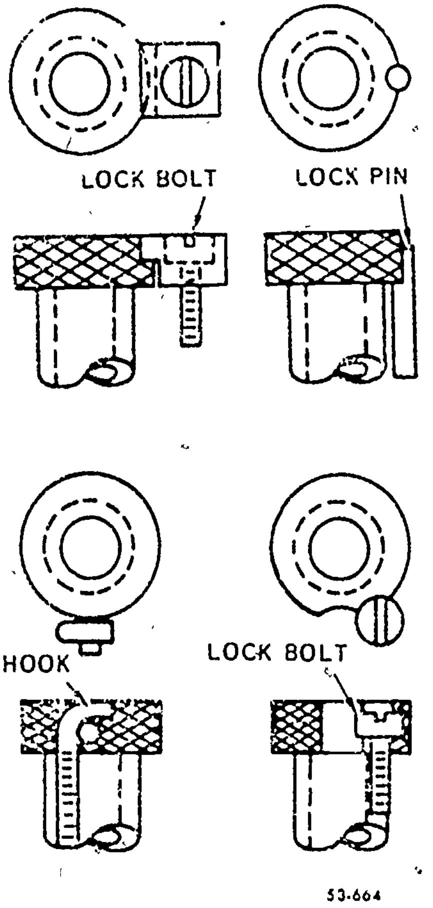


Figure 14. Bushing clamps.

7-15. *Screw.* The screw bushing, shown in figure 15, performs well for light work with large tolerances. These bushings not only guide the tool but clamp the work firmly and eliminate the need for other holding devices. A disadvantage of this type of bushing is that when the thread becomes worn, it is inaccurate. A screw bushing must have a head that can be turned by a wrench. For the several different types of heads, shown on figure 16, you use (from left to right) an end wrench, round-tipped spanner, square-tipped spanner, socket or box end, and a special pin wrench, respectively. The use of the bushing determines the type of head you select.

7-16. *Special.* Special bushings can be designed and made according to the task they must perform. Your skill and ingenuity are your only guides. If an operation requires a bushing that is not of a normal configuration—for example, when the holes to be drilled are too close together to use three separate bushings—you must design a single bushing to accommodate the three holes

7-17. **Fastening and Alining Devices.** For every type of jig or fixture designed and used in any machine shop, there must be some means of clamping the workpiece to either the jig or the fixture. Also, some provision must be made for alinement. The design of clamping devices is limited only by your imagination. Some of the more common clamps are the screw, cam, hook wedge, toggle, and rack and pinion. You can use one or all of these in one form or another. The function of the jig will guide you in selecting the type to use for clamping. Alining devices are many and varied. Here again, the size, shape, and operational need governs the type of alining device you should use:

7-18. **Fixtures.** There are several classes of fixtures that can be subdivided into many types. The class of a fixture is determined by the machine on which it is used. The milling machine, planer, lathe, boring mill, and turret lathe are a few examples of the machines on which you use fixtures. You can also design these fixtures to be used on more than one machine. We will limit our discussion to milling fixtures.

7-19. The type of fixture selected depends on the kind of milling operation to be performed and upon the type of cutter used. Milling fixtures aid in the performance of many milling operations: form milling, angular milling, T-slot cutting, straddle milling, and many more

7-20. *Auxiliary vise jaw.* One of the simplest

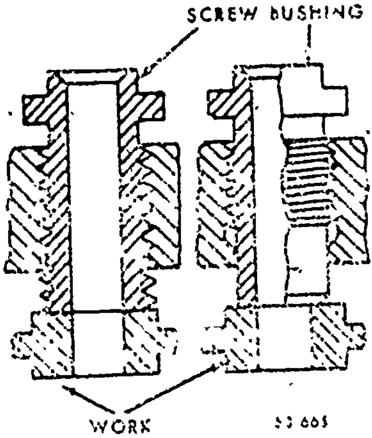


Figure 15. Screw bushings

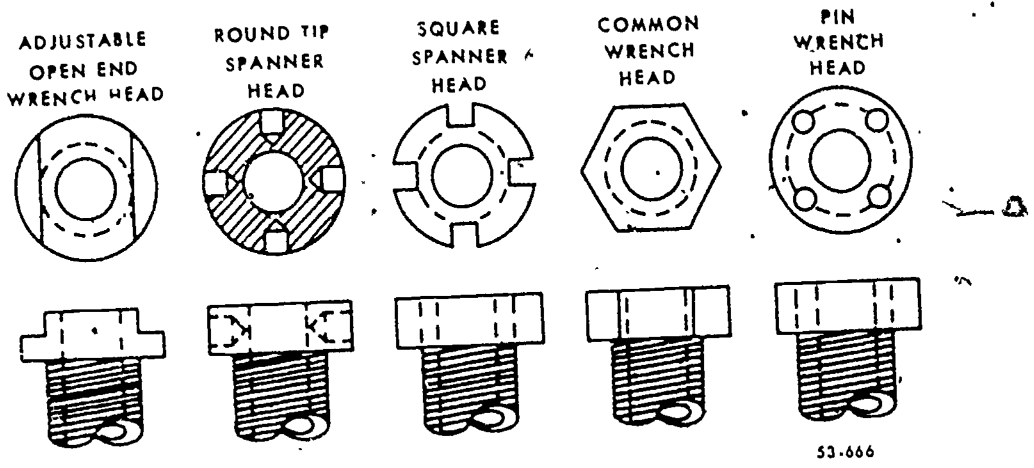


Figure 16. Screw bushing heads.

and most widely used fixtures is a set of auxiliary vise jaws. These vise jaws are built to replace standard vise jaws and are used for simple milling operations if the shape and the size of the workpiece permit. They are usually made to fit only one part. An exception is the V-block vise jaw. The governing factors of this special vise jaw are the size of the vise and the depth to which the V-block is cut. The auxiliary jaw usually replaces the stationary jaw in the vise, and the movable jaw only holds the part in place. Auxiliary vise jaws are usually made of low carbon steel and are case-hardened. For short runs or low production jobs, they need not be case-hardened.

7-21. *Angle plate fixtures.* Angle plate type fixtures are designed to hold a part for slotting or face milling. This fixture is built on a base plate with another plate at an angle to the base. The angle plate milling fixture is made like an angle plate It can be made at any desired angle. The workpiece is located and securely clamped on the face of the angle plate.

7-22. *Multiple and duplex fixtures.* Fixtures that hold two or more like parts are gang or multiple fixtures. With a little change however, they can be converted to continuous milling fixtures if provisions are made for clamping each part separately. Duplex fixtures are designed so that, as one part is completed, it can be removed while another part is being machined. Also, two like fixtures can be mounted at opposite ends of the indexing base. This enables you to load or unload one fixture while the part in the other fixture is being machined After machining the workpiece in one fixture, the fixture on the other end is swung into position for the machining operation.

7-23 In some cases, you may need a fixture with which you can perform more than one operation without removing the part. This is done in machining castings that must have subsequent operations performed on them. In order to maintain accuracy, the parts must remain in the fixture, and the fixture moved to the next machine. Some castings are very difficult to reset in another fixture or even to reset in the same fixture because of their rough and uneven surfaces.

8. Fabrication of Jigs and Fixture

8-1. There are several methods of fabricating jig and fixture bodies. We will discuss the three most common types of body construction that are used by tool makers: builtup, welded, and cast or one piece.

8-2. **Builtup Type.** The builtup type, usually fabricated by fastening together steel plates with screws and dowel pins, is a convenient and economical method. Dowel pins must be used to aline the parts and to keep them in alinement. The screws simply hold the parts of the tool body together. Remember, however, that this type of tool body lacks the rigidity of the cast and welded body types. Its accuracy can be distorted by careless handling and loosening the screws that hold the parts together.

8-3. **Welded Type.** The welded type of body construction is an outgrowth of the builtup type. The difference is that the builtup type is put together with screws and dowel pins and the welded type is welded together. The welded bodies are often preferred because of their greater strength and rigidity. Another advantage of the welded body type is that it is easily altered, permitting its adaptation to a part other than the one it was designed for. The surfaces of welded type tool bodies, which must serve as bearing surfaces, base lines, or surfaces on which accessories are mounted, must be machined after welding. This minimizes distortion and other imperfections caused by the welding, normalizing, and sand blasting processes.

8-4. **Cast or One-Piece Type.** The cast or one piece type of tool body is often required in the construction of special tools. It can be molded into any necessary size and shape and can be designed to require a minimum amount of machining. Cast bodies can be easily lightened by coring out material without reducing their strength or rigidity. You will be more likely to work with the one-piece body. The only difference between the cast type and the one-piece type is that the latter is machined from a solid piece of stock. It has the same strength and rigidity as the cast type. If possible, avoid the use of the one-piece type because of the cost of manufacturing it.

8-5. **Locating Points.** In the manufacture of a large number of parts, tools and machines are arranged to carry out definite routines. Since aircraft construction requires accuracy, all the parts in an operation must be presented to the tools in as nearly the same position as possible. Each part must be located on enough surfaces to give it a definite place in which it can be held securely. Common surfaces have the form of planes, cylinders, and cones. Some parts have special surfaces and are more complicated to locate. Many means of locating have been devised to cope with all types of surfaces, but the key to all of them is in the principles of locating.

8-6. Confinement alone does not insure location. For instance a part may be held in a vise securely and not be located. A free body moves in any direction, but restriction of movement in three axial directions prevents motion in any direction. A free body also rotates in any direction, but you can bring a body to any desired position by revolving it about its three axes, as shown in figure 17. The three linear movements and the three rotations are the six elementary movements that must be restricted to confine an object completely.

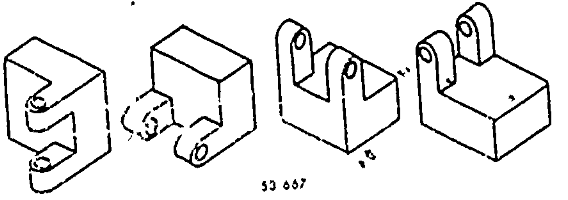


Figure 17. Revolving an object to the desired position

8-7. The locating points are used to position the part. They should not be confused with clamps. Clamps are the devices that hold the part firmly against the locating points, which we will designate as rest buttons.

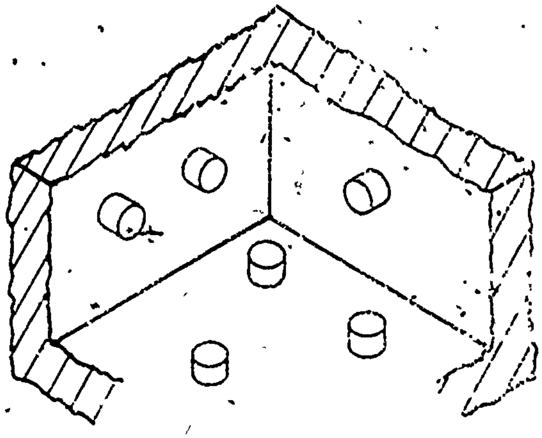


Figure 18. Location of rest buttons

8-8. The 3-2-1 principle of location is usually adopted in building a jig or fixture. Three locators on the base keep the part from rocking in any direction. Two contact points are on the vertical member, next to one of the long sides of the part, as shown in figure 18. They keep the part from rotating and also prevent linear movement in one direction. One rest button is placed on the vertical member next to an end of the part to complete the location of the part. These six locators, with the help of the clamps, hold a part rigidly in position. It is a poor practice to put locators in other than the horizontal and vertical planes. Figure 19 illustrates how an error is increased when a rest button contacts the slanting side of the part. The locator also has a tendency to lift the part when the part is clamped, and a wedging action takes place between the locator and the slanting edge of the part. Because of its name, the 3-2-1 principle is easy to remember when you need it during the construction of jigs, fixtures, and dies.

8-9. **Surface Conditions.** The types of locators and the number of locators needed to position a part depend upon the finish of the

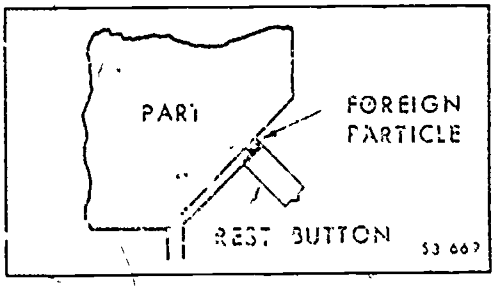


Figure 19. Position error

locating surfaces on that part. The three kinds of surfaces that are usually found on parts are finished, semi-finished, and rough.

8-10. *Finished surfaces.* Finished surfaces have been machined until they are smooth and true. They can be located on flat surfaces, which make full contact. However, if any metal chips from the previously machined parts are left on the location surfaces, the next part can not be located accurately. The use of rest buttons minimizes this problem. More than three locating points on one surface do not improve location but may be used to give the needed rigidity to parts with finished surfaces. If more than three rest buttons are used on one surface, the buttons must be ground to the same height to prevent the piece from rocking. If a part were clamped on locators of uneven height, it would be machined in a warped position, producing an untrue machined surface.

8-11. *Semi-finished surfaces.* Semi-finished surfaces have been rough-machined but still contain some warpage and a slight amount of roughness. If more than three locators are used on one surface, the additional locators should be adjustable to allow for the variations in the different parts.

8-12. *Rough surfaces.* Rough surfaces are unmachined and have wide variations. To cope with rough surfaces, keep the locating buttons as far apart as possible to minimize the difference in position of the parts that are being machined, as shown in figure 20. Also, use the minimum number of locating points necessary to hold the part rigidly.

8-13. **Types of Locators.** The shape of a part determines the type of a locator that is best suited for the job. Each type of locator excels the others when it is positioning the shape of a part for which it is best adapted. The types of locators and their uses will be covered in the following paragraphs.

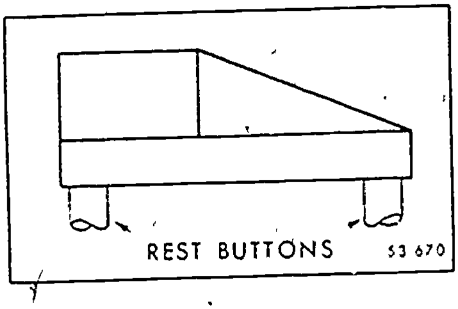


Figure 20 Wide separation of rest buttons

8-14 *V-locators* V-locators are best for holding cylindrically shaped objects. When a cylinder is placed on a V-block, it loses all but two degrees of freedom—it can rotate about its own axis and it can move lengthwise in the V-slot. If a stop is put at one end of the V-block, the cylinder can no longer move lengthwise. If the cylinder is clamped to the V-block, the cylinder loses its last degree of freedom, rotation about its axis. A rough, cylindrically shaped object should have a V-locator near each end instead of one V-block to hold it over its entire length. This minimizes rock or spring in the part. V-locators are also used for parts other than cylinders. An example of the V-blocks that are used as centralizers for parts with radial ends is shown in figure 21. The placement of a V-locator is important. For drilling through the diameter of rods of different sizes, position the V-block so that the center of any sized rod is in line with the axis of the drill bushing.

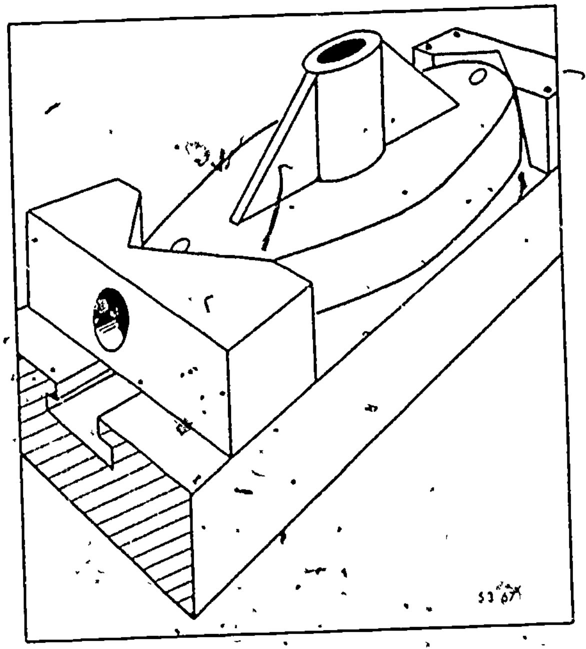


Figure 21 V-block centralizers.

8-15. *Cylindrical locators.* Parts that have had holes drilled in them during a previous machining operation can be located for subsequent machining operations by cylindrical locators placed to fit into the holes. There are also many other uses for cylindrical locators. For instance, the assemblies in a jet engine, such as the compressor and the turbine, are cylindrical in shape and are held by cylindrical locators during machining operations.

8-16. *Conical locators.* In many cases, conical locators are more adaptable than cylindrical ones. If there is a tolerance for the hole sizes in a part, the cylindrical locator must be smaller than the smallest hole size allowed. The lateral movement of the part with the largest hole diameter within tolerance on this same locator can be excessive. You can use a conical or tapered plug to overcome the variation of the placement of the part in the fixture. A common use of conical locators is holding work between centers on a lathe. Another tapered locator is the mandrel, which can be pressed into a centrally located, straight-machined hole of a part so that the part can be turned between centers. Another application of the conical locators is illustrated in figure 22. A rough blank is located from its hub by an internal cone in the end of a screw bushing.

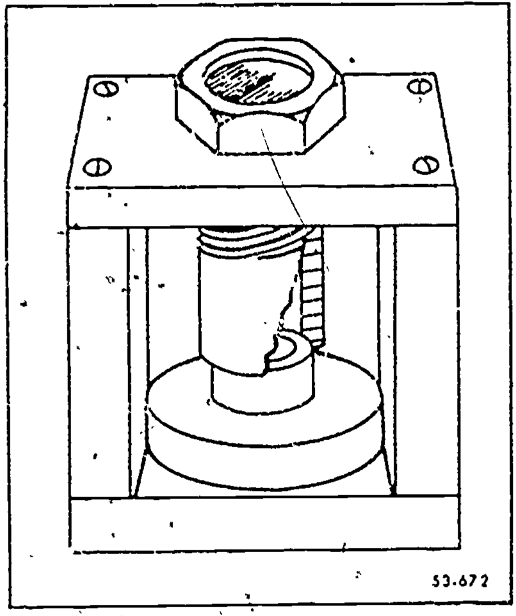


Figure 22. Internal cone locator

8-17. A part, located by means of a cylindrical plug, often needs to rest against another locator to keep it from rotating during the machining operation. Note how this is shown in figure 23 The pin used to prevent the part from revolving should be placed so that its point of contact with the part is as far as possible from the fulcrum point. A part located by two pins is considered as deriving most of its location from the base of the fixture and one pin. The second pin only keeps the part from rotating around the first pin. The second pin is ground into a diamond shape leaving cylindrical segments on the two ends of the diamond. By being relieved as it is, the diamond pin locator allows for the slight deviation in measurement between the two respective holes. A closeup of the top view of the diamond locator is shown in figure 24 The included angle at each end of the diamond is 60°. The distance across the flats is three-fourths of the diameter of the locator.

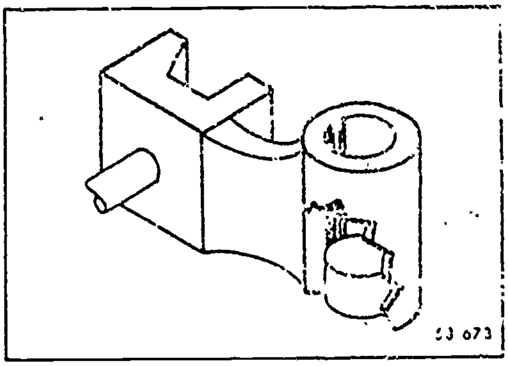


Figure 23. Radial locator.

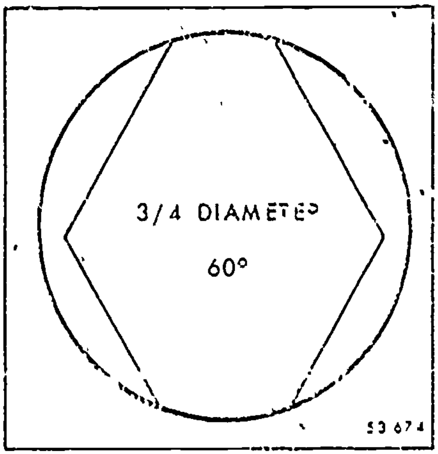


Figure 24 Top view of a diamond locator

Die Design and Fabrication

THE DESIGN AND fabrication of metal cutting and forming dies are very important parts of your job as a machinist. These punches and dies can save you many hours of work. In this chapter we will discuss the basic press operations, and the design and fabrication of punches and dies.

9. Types, Uses, and Designs of Dies

9-1. Press tools provide one of the major methods of producing metal parts. Their range and application have become almost unlimited. Many articles that were formerly produced by machining operations have become products of the press department. In earlier years, press work was limited to small items, but now there is apparently no limit to the size and variety of parts that can be made on the press, varying from small metal eyelets to automobile body parts, fenders, and similar pieces of large dimension. Press tools may be divided into groups based on their effect upon the structure of the metal to which they are applied. Some tools act upon the metal by some form of shearing or cutting action, such as blanking, piercing, or shearing. Some tools work the metal in the sense that they cause it to flow. Forming and drawing tools are typical of this group.

9-2. **Types of Die Operation.** A large variety of press-work operations, some of which are very complex, can be reduced to simple fundamentals.

9-3. *Blanking.* Blanking is the operation of cutting out a part with a punch and die. The material used is called the stock. During the working stroke, the punch goes through the material. After the material is cut, it drops through the die; and the punch returns to its original position. Figure 25 illustrates the action of a blanking die. The work is fed by hand or by some type of feed device. Stop pins are usually used to gage the stock so that maximum stock is used.

9-4. *Piercing.* The piercing operation consists of the punching of holes. It differs from blanking in that the punched out material is the scrap. Both flat-ended and spiral-cutting piercing punches are used. A flat-ended punch is illustrated in figure 26. It is often thought that piercing dies produce round holes principally. But they are almost as frequently used for making openings of other shapes, such as square, oblong, irregular, curved, and slotted.

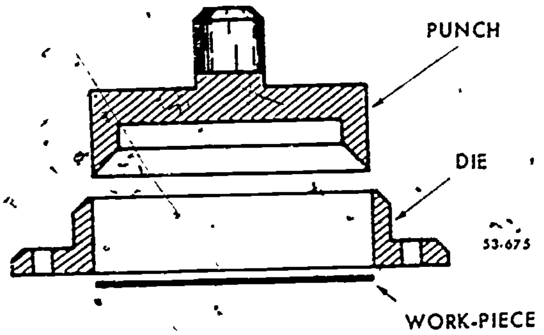


Figure 25. Blanking die.

9-5. *Bending and forming.* Bending and forming dies are made in great variety and operate on all classes of work. As the name

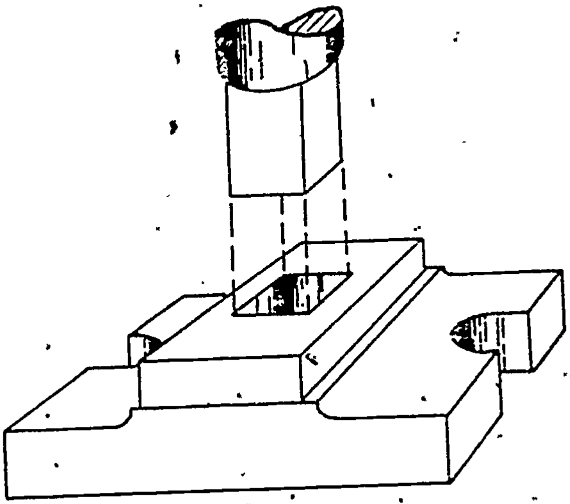


Figure 26. Piercing die.

implies, this operation forms or bends the blanks. A simple form of bending die is illustrated in figure 27. The outline of the bend, which is to be imparted to the blank, is formed on the punch and die. Quite frequently, when more intricate forms are required, the work is passed through two sets of dies, one for starting the outline and the other for completing the work.

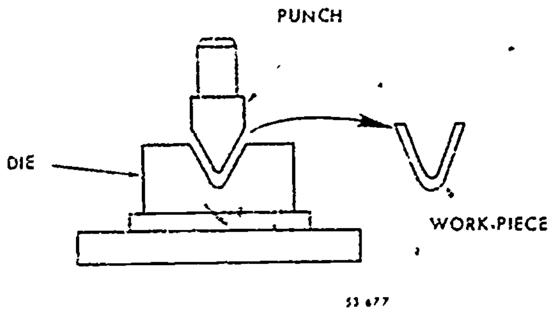


Figure 27. Bending die.

9-6. *Drawing.* The production of cups, shells, boxes, and similar articles from metal blanks is called drawing. In this process, a piece of flat stock (brass, steel, or other metal) is pushed through a round die by a dull-ended punch that cannot cut through the stock. An example is shown in figure 28. The shell that is pushed through the die is removed from the punch on the upward stroke by catching on the stripping edge.

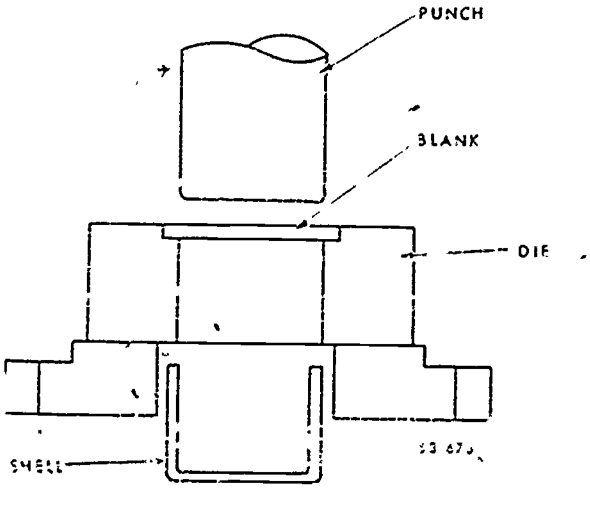


Figure 28. Drawing die

9-7. *Crimping.* Crimping means pinching or squeezing the sides of a shell in order to hold an object. An example is the crimping of a cartridge shell to hold the bullet in place.

9-8. *Coining.* As is implied by the name, coining is forcing metal into dies for the purpose of making letters and similar marks are in relief.

9-9. *Deep drawing.* Designing die sets for deep drawing is a very complicated process and requires considerable skill and experience. We will discuss only the general principles. Deep drawing is possible only by dividing the work into several stages or draws. Between each draw, the workpiece is annealed by heat because cold working the metal makes it hard and brittle. It is not possible to make long deep draws in one single stage because the workpiece, even if made of the softest material, would split and crack.

9-10. Deep drawing is often accompanied by an operation called ironing. Ironing is reducing the wall thickness of a shell by forcing it through a tight die. The walls of the shell are both lengthened and made thinner but the thickness of the shell bottom is not changed during the ironing operation. The space between the punch and die must be less than the thickness of the stock. Dies and other working parts used for ironing operations must be hardened, ground, and highly polished.

9-11. **Location of Blanks in Stock.** The die designer must consider the spacing of blanks on the stock. There are two factors to consider: the best location of blanks to save material and the best location to secure good bending where bending is needed. These matters require a great deal of good judgment. It is often wise to cut a few pieces of paper to the required outline of the punching. Then, by arranging them in different ways, you can determine the most economical pattern. Figure 29 illustrates the right way to punch stock. The gage pin is so located that there is sufficient stock left between each pair of holes, after the strip passes entirely through the press, to allow it to be reversed and passed through once more. This punches out most of the metal that remains between holes after the first punching. By arranging the operations to take place in this way, you can get a great many more punchings from the same amount of material. However, in choosing this method, you must weigh the extra labor cost against the cost of material saved.

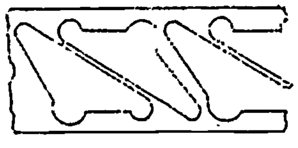


Figure 29. Location of blanks

9-12. **Selection of Economical Size of Stock.** Many times some savings can be made by the wise selection of the proper width of stock. In figure 30, you can see that by using stock wide enough to punch staggered holes, less material is needed for a given number of punchings than you need when using a narrower strip. You can see that the wide stock may not be twice as wide as the narrow stock, but it gives nearly twice the number of parts.

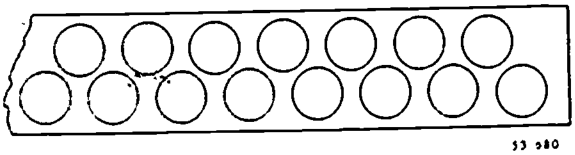


Figure 30 Method of locating blanks in stock

9-13. Relation of Grain to Bending. If you must consider bending the blanks after they are produced, you must understand that strip steel has a definite grain, somewhat like wood. The grain of sheet metal always runs in the direction of the length of the sheet. Bends should be made across, or at right angles to the grain, and not parallel to it. The figure 31 illustrates a part whose shape lends itself to an economical layout at a 45° angle to the grain of the metal. This brings the bend, as illustrated by the dotted line, at 45° to the grain. This angle seldom causes trouble unless a very sharp bend is made or a poor grade of material is used. Bends should not be laid out at less than 45° to the grain. The nearer they come to being directly across the grain, the less trouble will be experienced later from breaking in the formed part.

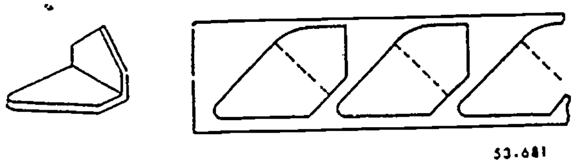


Figure 31 Blanks that are to be bent

9-14. Die Sets and Parts. Standardized die sets are used very extensively in many different sizes and styles. When a tool designer has a product for which he must design a die, he studies the job carefully and selects a suitable die set upon which he can mount his punches and dies. Standard die sets have two parts, the punch holder and the die shoe, as shown in figure 32. Figure 33 shows the other parts of a standard die set. They are the shank by which the punch holder is fastened to the press, the guide pins which insure that the punch is in accurate alinement with the die, and the bushings. The guide pins and bushings are made of hardened steel.

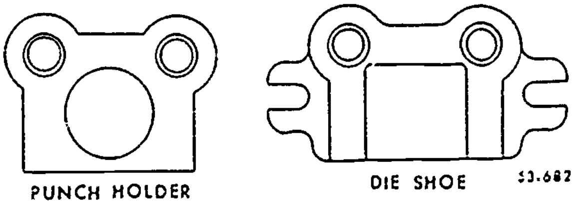


Figure 32. Die parts.

9-15. There are other standard parts that can be purchased, such as springs, dowels, and stripper bolts. Five types of standard die sets are available, but the one most frequently used is the back-pin set, shown in figure 33. In this type of die set, the pins are at the back of the set, leaving clear space for hand-feeding the blanks. It also permits a good view of the moving parts, free from obstructing pins and bushings. The center-pin type die set is used when the load is heavy and is fed from the front. The guide pins are in alinement with the load along the transverse centerline of the set. This leaves the front clear but prevents end-feeding either by hand or automatically.

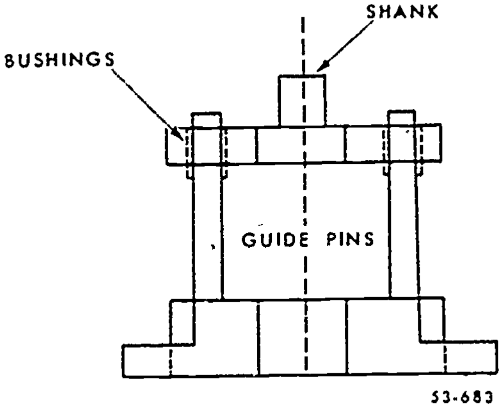


Figure 33 Assembled standard die parts.

9-16. The diagonal-pin type set is usually used if very heavy loading is required. In this arrangement, one guide pin is placed at the front of the set to avoid the overhang of the back-pin type. The load is in alinement with the pins along a diagonal line and leaves the end clear for feeding. The left pin is generally in front, but if the designer wishes to feed from the left, he can put the right pin forward. The final decision is based on the arrangement that gives the best view of the stop pins and is more adaptable to standard guards.

9-17. Round-die sets are made especially for coining and shaving operations. The guide pins are usually along the back, but can be at the center if needed. The punch holders and die shoes are usually round or oval-shaped.

9-18. The four-pin type die set is used for roll-fed operations, especially for progressive figure 30, you can see that by using stock wide enough to punch staggered holes, less material is needed for a given number of punchings than you need when using a narrower strip. You can see that the wide stock may not be twice as wide as the narrow stock, but it gives nearly twice the number of parts.

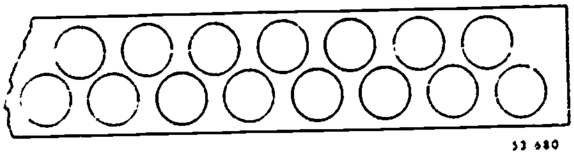


Figure 30 Method of locating blanks in stock

9-13 **Relation of Grain to Bending.** If you must consider bending the blanks after they are produced, you must understand that strip steel has a definite grain, somewhat like wood. The grain of sheet metal always runs in the direction of the length of the sheet. Bends should be made across, or at right angles to the grain, and not parallel to it. The figure 31 illustrates a part whose shape lends itself to an economical layout at a 45° angle to the grain of the metal. This brings the bend, as illustrated by the dotted line, at 45° to the grain. This angle seldom causes trouble unless a very sharp bend is made or a poor grade of material is used. Bends should not be laid out at less than 45° to the grain. The nearer they come to being directly across the grain, the less trouble will be experienced later from breaking in the formed part.

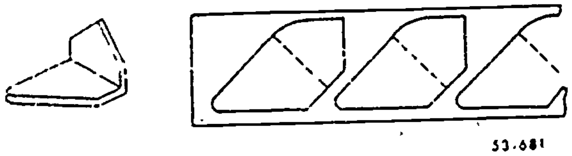


Figure 31 Blanks that are to be bent

9-14 **Die Sets and Parts.** Standardized die sets are used very extensively in many different sizes and styles. When a tool designer has a product for which he must design a die, he studies the job carefully and selects a suitable die set upon which he can mount his punches and dies. Standard die sets have two parts, the punch holder and the die shoe, as shown in figure 32. Figure 33 shows the other parts of a standard die set. They are the shank by which the punch holder is fastened to the press, the guide pins which insure that the punch is in accurate alinement with the die, and the bushings. The guide pins and bushings are made of hardened steel.

9-15 There are other standard parts that can be purchased, such as springs, dowels, and stripper bolts. Five types of standard die sets are available, but the one most frequently used is the back-pin set, shown in figure 33. In this type of die set, the pins are at the back of the set, leaving clear space for hand-feeding the blanks. It also permits a good view of the moving parts, free from obstructing pins and bushings. The center-pin type die set is used when the load is heavy and is fed from the front. The guide pins are in alinement with the load along the transverse centerline of the set. This leaves the front clear but prevents end-feeding either by hand or automatically.

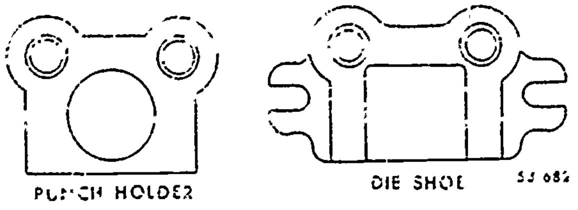


Figure 32. Die parts.

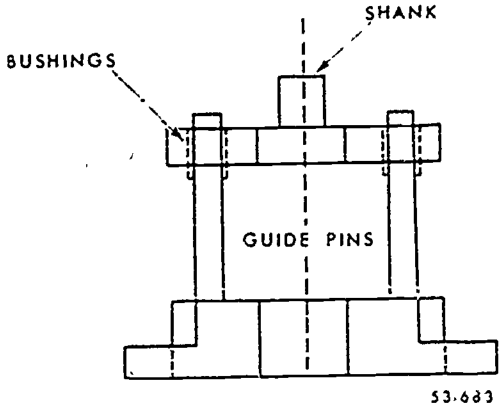


Figure 33 Assembled standard die parts

9-16 The diagonal-pin type set is usually used if very heavy loading is required. In this arrangement, one guide pin is placed at the front of the set to avoid the overhang of the back-pin type. The load is in alinement with the pins along a diagonal line and leaves the end clear for feeding. The left pin is generally in front, but if the designer wishes to feed from the left, he can put the right pin forward. The final decision is based on the arrangement that gives the best view of the stop pins and is more adaptable to standard guards.

9-17. Round-die sets are made especially for coining and shaving operations. The guide pins are usually along the back, but can be at the center if needed. The punch holders and die shoes are usually round or oval-shaped.

9-18. The four-pin type die set is used for roll-fed operations, especially for progressive

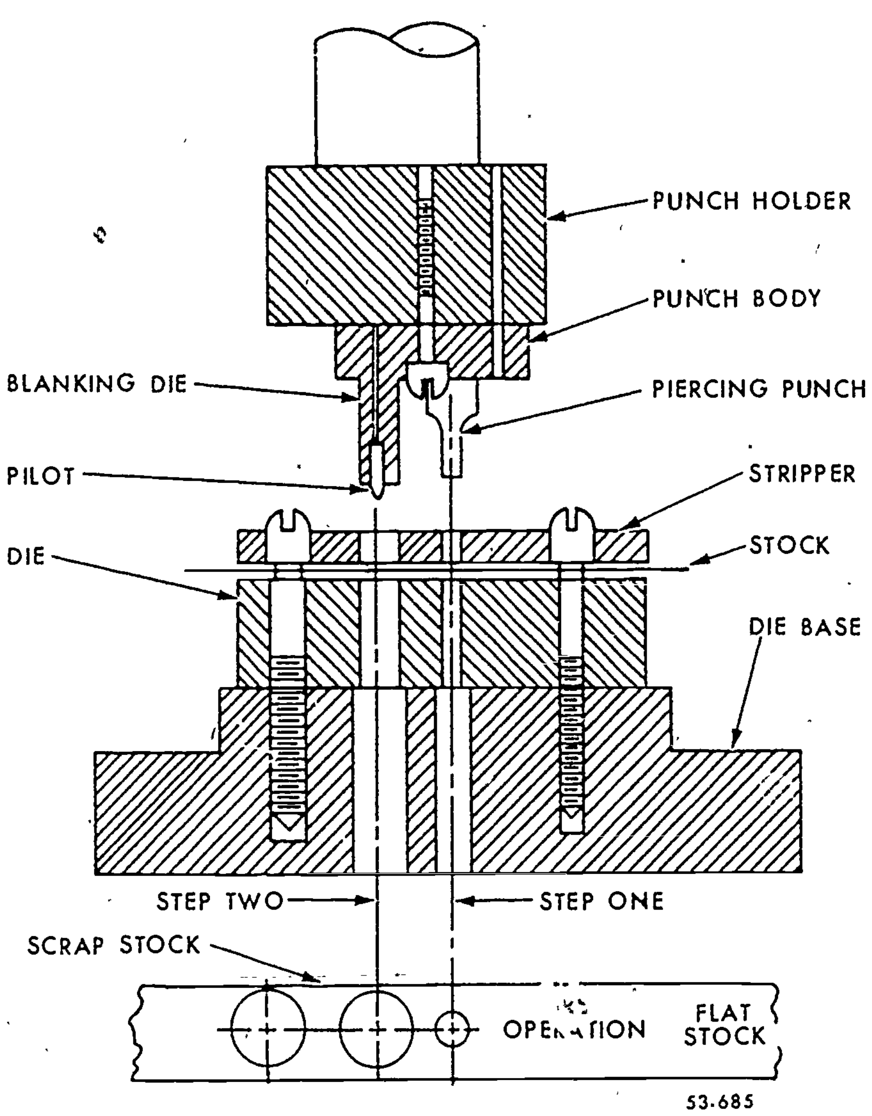


Figure 35. Progressive die.

9-25. *Compound.* A compound die is one that performs several operations during one stroke of the press at one station. To do this, both the upper and lower members of the die set carry the punch and die elements, together with the necessary strippers or ejectors. Figure 36 illustrates a simple compound die. This die blanks and draws a shell. Determine its action from figure 36.

9-26. The compound die action is such that its product is very accurate, and a die that is made correctly will perform accurately throughout its working life. Compound dies are usually more expensive to construct than the plain die of two-stage design, which is the simplest of the progressive types of press tools. On long runs, the initial high first cost is justified, and on certain classes of work, the compound die eliminates the necessity for shaving operations. The compound die was originally adapted for small circular blanks of small tooth wheels, special washer-shaped parts, pierced instrument elements, and other similar parts. It was later enlarged for almost every kind of work where its application is advantageous economically or mechanically.

9-27. The usual arrangement of a compound punch and die is to locate the blanking die in the punch head. The piercing punches are then fitted in the blanking die above, and the piercing dies are drilled or formed in the blanking punch below. There are a number of reasons for this practice, such as when there is no knockout for the upper die, or owing to the size of the work, when it is not feasible to introduce a knockout into the upper die shank. With the blanking punch fitted below to the die shoe, the piercings or slugs pass down through the die shoe as with a plain piercing die.

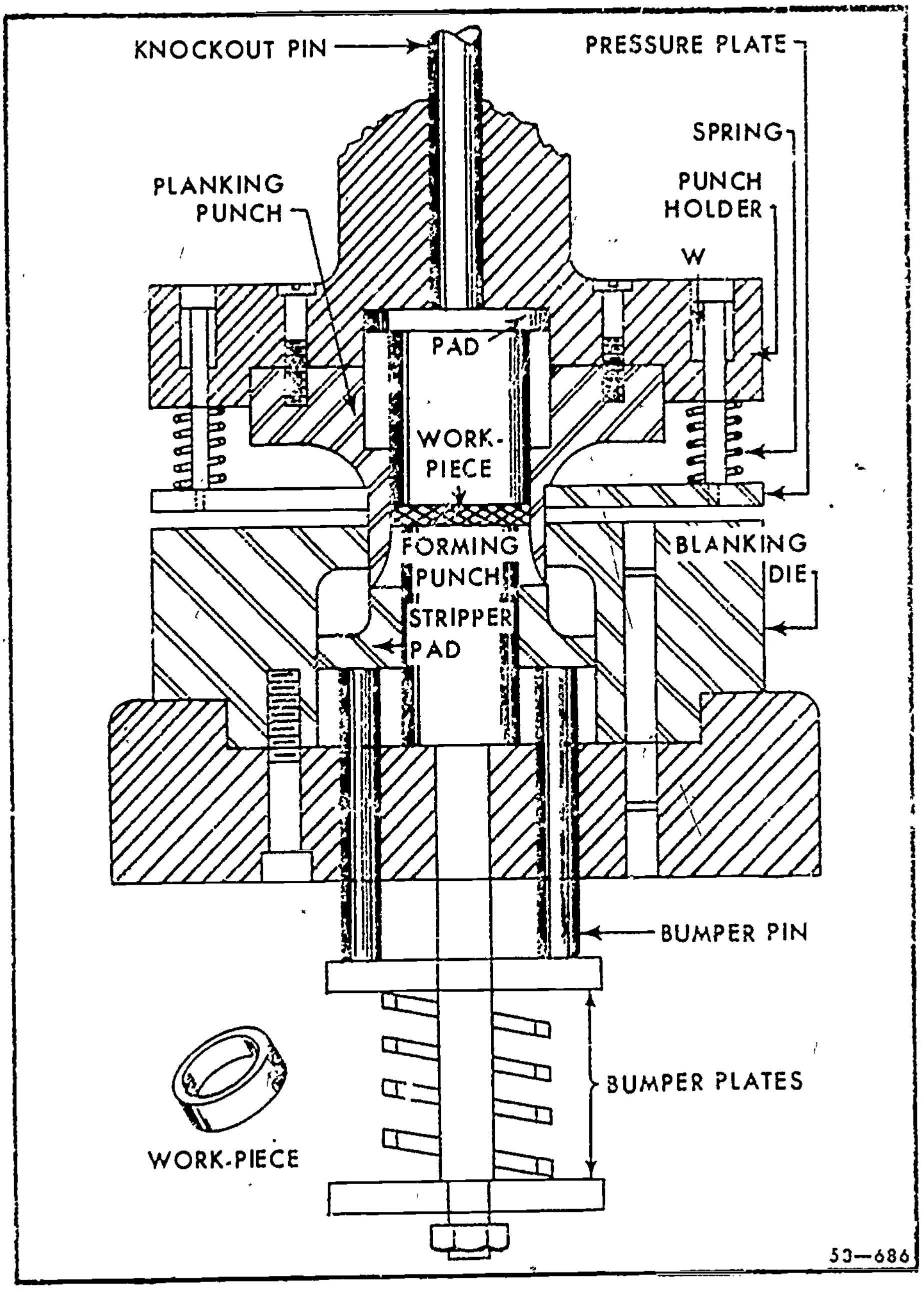


Figure 36. Compound die.

9-28. Compound dies operate more slowly than progressive dies, but they have advantages for certain jobs, especially where close tolerances must be held. Some of these advantages are listed below:

a. The action of the pressure pad assures flatness of the blank.

b. A pierced hole in the blank can be held to close tolerance with the edge. This is important when the hole must be concentric with the edges of a part.

c. Larger parts can be blanked in a smaller press if compound rather than progressive dies are used.

d. Long strips of material are needed with progressive dies. Sometimes scrap blanks are available. These can be hand-fed to a compound die when the savings in material offset the cost of the labor.

9-29. **Stock Stops.** When stock is fed into a press, some method must be used to locate the stock in the proper place. The simplest method is the use of a stop pin, but it has the disadvantage of requiring considerable skill in the operator. Trip dogs are frequently used to determine the amount of the material that should move forward. Figure 37 illustrates a typical trip dog arrangement. As the stock is fed

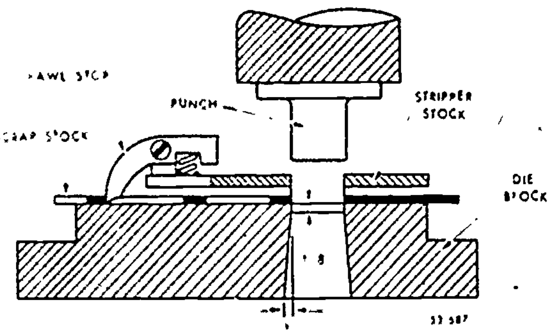


Figure 37 Trip stop.

forward, the pawl rises on the ratchet principle; then the stock is pulled back, and the pawl drops and locates the stock against the vertical surface of the pawl. Automatic stock stops are extensively used. They are controlled by the action of the punch as it descends or rises. There are many applications of automatic stops, designed for specific jobs. If the workpiece has the same width as the stock, and the feed is from one side without having a skeleton of scrap material pass out the other side, a shoulder stop is the most effective type. This stop is used on progressive dies where the last operation is a cutoff or trimming one. Figure 38 illustrates one of several shoulder stops. Note the indicated use.

9-30. **Strippers and Pressure Pads.** Strippers are used to remove the stock from the punch after a blanking or piercing operation. A channel-type stripper is often used. An example is shown in figure 39. A pressure pad is used frequently as a stripper, thus serving a dual purpose. The pressure pad is mounted on the punch holder and is held down by stiff springs. As the punch ram descends, the pressure pad holds the work in place while the punch passes into the die. On the upstroke, the pressure pad is held down momentarily, acting as a stripper. This action wipes the stock from the punch. Figure 39 illustrates this action. In simple forming operations, pressure pads are not necessary for stripper purposes, but they are used extensively to prevent edgewise movement of the blank after it has been moved from the locating nest by preliminary bending movements. While a blank is being bent or formed, it has a tendency to move sideways because strip steel has a tendency to resist bending more in certain areas than in others.

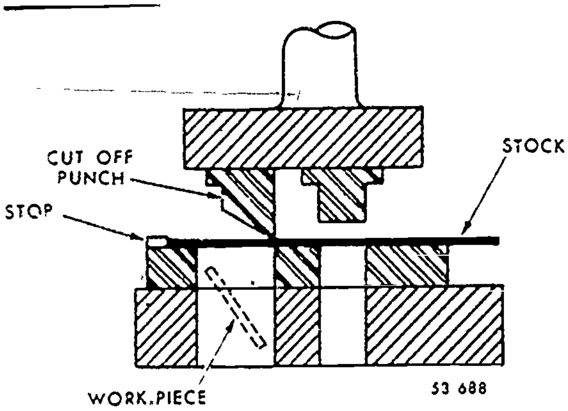


Figure 38 Shoulder stop

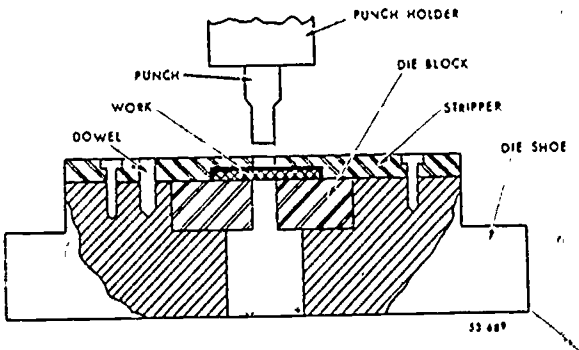


Figure 39. Channel-type stripper.

9-31. **Stripper Design.** Simplicity of design reduces cost. With this in mind, the designer must consider simple channel strippers first. These can be built up with a rectangular plate that is mounted on a single strip to form a backstop or guide. If the strip must be guided to close tolerances, a second strip, placed below the stripper, provides a groove through which the strip is passed. The width of the groove should be at least 0.005-inch larger than the width of the stock, and the height of the groove should be 1 1/2 the thickness of the stock. Strippers for large die sets can be made of one-piece construction and the groove can be machined out, as shown in figure 39. Strippers must be alined with dowels to insure accurate alinement on the die holder.

9-32. *Stripper bolt.* Commercial companies now market special shoulder bolts for pressure pads. You can find the dimensions for these bolts in catalogs and handbooks. The heads formerly had screwdriver slots, but now almost all of them contain recessed hexagon sockets of the Allen type. These specially made stripper bolts have an advantage over the cap screws in the definite shoulder against which the stripper plate can be fastened. This shoulder positions the stripper plate exactly, when the die set is new. However, when the punches are ground for resharpening and are thus shortened, the stripper plate will not be flush with the cutting edge of the punch. To adjust this, you must place washers under the head of the bolts at W, shown in figure 36.

9-33. Some designers prefer to use cap screws to avoid this lengthy adjustment. The heads of the cap screws are staked to prevent turning. You can retract the stripper plate by turning the cap screws after grinding the face of the punch, but releasing the staked head is difficult. Cap screws are frequently used with spacers made of steel tubing or pipe. The modern development of self-locking nuts has encouraged the use of nut and bolt construction.

9-34. *Pressure pad design.* Pressure pads for dies are spring- or cushion-actuated. Rubber cushions are satisfactory for short runs if the deflection is not too great and the possibility of oil splash is small. Select springs with an inside diameter only slightly larger than the stripper bolts. If the diameter of the spring is too large compared with that of the bolt, it may be advisable to counterbore holes in both the stripper and the punch plate to prevent the spring from "cocking" sideways and breaking. The number and placement of springs must be determined by the designer. There is no standard for this, but good judgment and experience are the determining factors. It is important to consider, when you are selecting springs, that they are subject to many repetitions of stress and are liable to fatigue failure.

9-35. As mentioned before, it is difficult to determine the pressure needed for stripping. It is evident that, if the stock around the punch is frail and stretches easily, little pressure is needed to remove the scrap from the punch. If there is a substantial amount of material around the punch or several punches, the stripping pressure can be as much as ten percent of the cutting pressure. You must consider the amount of pressure needed to hold the stock firmly while it is being cut, and also when you are choosing or designing springs.

9-36. **Clearance and Angular Relief.** There must be a definite amount of clearance between the punch and the die for blanking or piercing. The amount of clearance is controlled by the thickness and type of material to be blanked. For thin material with a low tensile strength, such as brass, for example, the clearance is very small. If too much clearance is applied, the blank will have ragged or burred edges. Heavy stock needs more clearance than thin stock. It requires greater clearance to lessen the possibility of breaking the punch or die and to reduce the pressure required to complete the blanking operation. The clearance is designated by two methods. The first is to designate the space between the punch and the die on *one side only*, or one-half the total difference between the sizes of the punch and the die. This method of designating die clearance is most useful when you are working with parts of nonsymmetrical forms or irregular contours. The second method is the total difference between the sizes of the punch and die. This method works best and is less confusing in connection with symmetrical parts. Specify the method of designating clearance that you used. This helps to eliminate confusion and error.

9-37. The clearance usually allowed for brass and soft steel on most dies, one one side, is equal to the stock's thickness multiplied by 5 or 6 percent. For some classes of work, one-half of this clearance is preferred. For some piercing operations, a clearance equal to the stock's thickness multiplied by 10 percent gives the cleanest fracture. This clearance may be used in such an operation as punching holes in ductile boiler plate.

9-38. If blanks are to pass through a die, as in figure 37, an angular relief is needed to keep the blank from jamming in the passage. The amount of relief ordinarily given a blanking die varies from 1/4° to 2°. However, dies to be used for a relatively small number of blanks are sometimes given a relief of 4° to 5° to facilitate making the die quickly.

9-39. There are two methods of applying angular relief to a die. The first method is to extend the angular relief from the bottom of the die to the top surface or to the cutting edge. This method is best suited for thin, soft materials. The second method is to leave a straight section below the cutting edge of the die. This straight section should be about 1/8 inch in width. The second method is best suited for harder materials. The straight section, called the "land of the die," permits many sharpenings of the die with out changing the size of the die cavity.

10. Fabrication of Dies

10-1. The fabrication of tools is another important part of your job as a machinist. After you have designed the tool for a certain operation, such as blanking or forming it is extremely important to fabricate it properly In this section we will limit our discussion to the more critical areas of die fabrication.

10-2. **Die Thickness, Length, and Width.** Some general rules for calculating the thickness of small dies should be remembered For blanks with a perimeter of 3 inches or less, use a die block thickness of 3/4 inch. For blanks with a perimeter 3 inches to 10 inches, use a 1-inch die block thickness. For blanks with a perimeter of over 10 inches, use a 1¼-inch die block thickness. For example, if you were to manufacture a round die to blank a 7/8-inch disc, the perimeter of that part is calculated by multiplying *pi* (π) times the diameter, which is $3.1416 \times .875 = 2.7489$. As you can see, the perimeter is less than 3 inches so the die block thickness should be 3/4 inch.

10-3. There should be a margin of 1¼ inches around the die opening. This margin is left around the die opening to insure that the die does not break during the blanking operation. The margin also provides enough material for cap screws and dowel plans. To calculate the length and width of the die, add 2½ inches to the length and to the width of the part to be manufactured. For example, the length and width of the die block for a die to blank a rectangle that measures 2.375" by 4.875" is calculated as follows. 2.375 + 2.500 = 4.875" width, and 4.875 + 2.500 = 7.375" length.

10-4. **Clearance.** The application of clearance to the die block or punch is a most important step in the fabrication of dies. If clearance is not properly calculated and applied to the proper die members, the parts produced will probably not meet blueprint specifications. Remember that the die cavity controls the size of the blank and the punch controls the size of the hole. From this simple rule, you must decide to which die member the clearance should be applied. If the die block is the member that receives the clearance, the amount of clearance calculated must be added to the size of the die opening. If the punch receives the clearance, the amount of clearance calculated is subtracted from the size of the punch.

10-5. **Blanking Pressure.** Blanking pressure depends upon the material and the area to be sheared, together with the percent of penetration and the amount of shear on the punch. For round holes, the pressure required equals the circumference of the hole times the thickness of the stock times the shearing strength. The formula is:

B.P = L × T × S
B P = Blanking Pressure
L = Length of cut in inches
T = Thickness of material
S = Shear strength of material in p.s.i (pounds per square inch)

To allow some excess pressure, the tensile strength can be substituted for the shearing strength. The tensile strength of common materials is roughly assumed as follows: mild steel, 60,000 pounds per square inch; wrought iron, 50,000 pounds; bronze, 40,000 pounds; copper, 30,000 pounds, aluminum, 20,000 pounds; zinc, 10,000 pounds, and tin and lead, 5,000 pounds.

10-6. You can reduce the amount of blanking pressure needed to cut a workpiece as much as 50 percent by placing suitable shear on the punch die. Figure 40 shows an example of shear applied to a punch. When shear is placed on a die member, the amount of pressure required to perform the operation is reduced. This not only reduces the size of the press needed but also adds to the life of the punch. If the blank is the workpiece, the shear should be on the die, and the punch should be flat because the shear angle has a tendency to distort the metal. If the blank is scrap and the strip must be flat, the shear should be on the punch. Shear should be applied to the die member that contacts the scrap. The amount of shear added to the punch or die should be equal to a taper, across the face of the punch or die, of 1½ times the thickness of the stock to be blanked. It may be preferable to use a double angle starting at the center of the punch or die. This double angle helps maintain symmetry and prevents the setup of lateral forces.

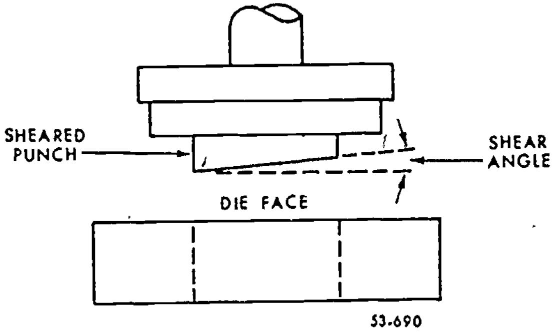


Figure 40. Shear applied to punch.

10-7. **Contour Sawing of Dies.** Internal contour sawing is well-suited to producing blanking dies and other straight-through die openings. It is the only process that permits both the die and punch to be machined from the same piece of steel. This procedure, shown in figure 41, is not complicated because every work step

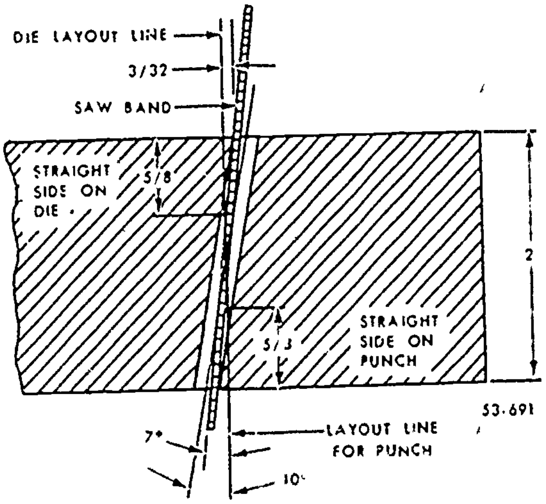


Figure 41. Contour sawing of dies

TABLE 1

DIMENSIONAL DATA FOR CONTOUR DIE MAKING

Die Thickness Inch	Angle Of Saw Starting Hole Degree	Angle For Saw Cut, Degree	Distance From Die Layout Line To Center Of Saw Kerf Inch	Distance From Die Layout Line To Center Of Starting Hole Inch	Diam Of Drill Inch	Width Of Starting Saw, Inch	Amount Of Straight Sides On Punch and Die, Inch
1/2	21	18	5/64	3/32	1/8	3/32	3/16
3/4	18	15	3/32	1/8	1/8	3/32	9/32
1	14	11	3/32	1/8	9/64	1/8	3/8
1-1/4	12	9	3/32	1/8	9/64	1/8	15/32
1-1/2	11	8	7/64	9/64	9/64	1/8	9/16
2	10	7	1/8	3/16	13/64	3/16	13/16
3	9	6	5/32	1/4	17/64	1/4	1-1/8
4	8	6	7/32	9/32	17/64	1/4	1-5/8
5	7	6	1/4	5/16	17/64	1/4	2-1/4
6	6	5	1/4	5/16	17/64	1/4	2-1/2

and related dimension is based on the thickness of the die block, as indicated in table 1. Three operations are involved:

a. The starting hole is drilled at an angle that varies with the thickness of the material. It is started inside the die layout line and emerges from the die block on the opposite side of the layout line.

b. With the table tilted at an angle slightly less than that of the starting hole, the slug is sawed out of the die block. The path of the saw on the surface is entirely inside the layout line, and the proper cutting angle at the bottom of the slug allows for excess material outside the die layout line.

c. With the slug or "punch" removed from the die, the die is band-filed to the layout outline, and the slug is band-filed to fit the die. This filing operation removes all traces of the starting hole on the finished surfaces of both punch and die.

Fitting and Assembly

FITTING AND ASSEMBLY is another important part of the machinist's job. After you have manufactured the parts, in many cases you will have to assemble them. Before the parts are assembled, you should carefully inspect them for accuracy and, if necessary, rework them so that they fit properly. After the parts are assembled, you must perform a functional tryout and inspect the end product. These are the topics of discussion in this chapter.

11. Inspection, Fitting, and Assembly of Machined Parts

11-1. The term "inspection" has come to have a very broad meaning in recent years in the metalworking field. For example, the concept of inspection now includes all types of destructive and nondestructive inspection. The testing and inspection of materials have become so involved and complex that a new job specialty, Nondestructive Inspection, AFSC 536XX, was created in September 1966 to train specialists to work in this field. This specialty is manned by highly skilled personnel who have been trained in the various methods of inspection, such as X-ray, ultrasonic, and magnaflux. If you are fabricating a replacement part that is to be subjected to critical stresses, you should take it to the nondestructive inspection specialist for testing. Also, if there is a question about the adequacy of any mechanical property, such as hardness and tensile strength, you should take the part to the metals processing specialist for testing.

11-2. Today in the Air Force, the nondestructive inspection specialist and the metals processing specialist perform many of the inspection functions formerly required of the machinist. As a result of this change, the machinist now thinks of inspection primarily in terms of checking the physical dimensions and the finish of machined parts to insure that they fit together properly. We will, therefore, discuss (1) The use of measuring tools and devices to check machined parts for accuracy, and (2) The classification of machine fits.

11-3. **Checking Machined Parts.** A machinist is often required to machine a replacement part. Suppose you have the task of machining a gear to replace one that has been damaged. You could measure the damaged gear and make a duplicate, but it would be much more satisfactory, to obtain a blueprint and work from that. Because of wear, the dimensions of the damaged gear may not be correct. It is impossible to machine a part to mathematically exact dimensions. Furthermore, it is impossible to manufacture a measuring device that is entirely free of error. It is also impossible for a machinist, even with the aid of a magnifying glass, to read a measuring device with absolute accuracy. These observations are not intended to discourage you. We call them to your attention only to remind you that difficulties do exist and that checking is required to make sure that machined parts are within tolerance. The tolerances permitted in most machining operations in an Air Force machine shop are not so precise that they pose a serious problem to a skilled machinist. We should review some of the minute, inherent errors in checking the accuracy of a machined part. A knowledge of these errors will help you to machine parts with greater accuracy.

11-4. *Instrument error.* Every measuring instrument has an inherent "error of indication." The accuracy of a micrometer depends mainly upon the amount of error in the lead of the spindle thread. A new micrometer of good quality is accurate within 0.0002 inch in the range of spindle travel. The accuracy of a dial gage depends mainly upon errors of graduation and of eccentricity and friction in the transmission gearing.

11-5. As a machinist, you cannot do anything about inherent error of indication. You can, however, insist that the people in your shop treat measuring tools with the respect they deserve. Do not tolerate any abuse in the handling and use of measuring tools. The work of a highly skilled machinist is wasted if he is forced to use inaccurate measuring tools. Keep records of the dates on the PME (precision measurement equipment) schedule, where each item of equipment is listed for delivery to the PME shop for inspection and calibration. Make certain that new measuring tools are sent to the PME shop to be checked before using them in the shop.

11-6. *Error in use of tools.* Obtaining an extremely accurate measurement with a micrometer is an art. When a machinist "mikes" the outside diameter of a shaft, there is no precision indicating gage to tell him when he has turned the micrometer thimble just the right amount. Proficiency in the use of measuring tools can be developed only through on-the-job training.

11-7. *Reading error.* Reading error is often caused by lack of skill. For example, the trainee's first attempt to take a reading on a vernier scale can be quite confusing. Reading error from lack of skill can be reduced by additional training. Technically, "reading error" is the uncertainty of the human eye in perceiving fractional intervals on a scale. One machinist has no difficulty determining which division marks line up on a vernier scale: another, even with the aid of a magnifying glass, can not determine which marks line up. There is no remedy for deficient perception, though poor eyesight can be corrected.

11-8. **Classification of Machine Fits.** The fit between two mating machined parts is determined by the engineer who designed the equipment. In selecting the type of fit, he considers many factors, such as length of engagement, bearing load, speed, lubrication, temperature, and humidity. The tolerances permitted in the diameter of a hole and the diameter of a shaft are intended to produce a particular type of fit.

11-9. The type of fit between a machined part and its mating part is determined by the purpose the parts are intended to serve. If, in the operation of the assembly, there is relative motion between the parts, a running or sliding fit is required. Sometimes, the purpose of a fit is simply to locate a part with reference to other parts in an assembly. Another general purpose is to hold the parts tightly together.

11-10. One system of classifying the types of fits, called the American Standard Fits System, is described in the *Machinery's Handbook*. It divides fits into general groups: running and sliding, locational, and force fits. Each category is divided into several classes. For the sake of brevity, we will discuss only the force fit system.

11-11. A force fit (FN) is a special type of interference fit in which constant bore pressures are normally maintained throughout the range of sizes. Therefore, the interference varies almost directly with the diameter. The difference between its minimum and maximum value is small in order to maintain the resulting pressures within reasonable limits. The American Standard Fits System divides force fits as follows:

a. FN 1. Light drive fits that require light assembly pressure and result in a relatively permanent assembly. They are suitable for thin sections and for long fits and for assembling a part in a cast-iron external member.

b. FN 2. Medium drive fits that are suitable for ordinary steel parts and for a shrink fit on light sections. They are the tightest fit that can be used in assembling a part in a high-grade, cast-iron external member.

c. FN 3. Heavy drive fits that are suitable for heavier steel parts and for a shrink fit in assembling medium sections.

d. FN 4 and FN 5. Fits that are suitable for parts that can be highly stressed, and for shrink fits in which it is impractical to use heavy pressure to assemble the parts.

11-12. Producing a force fit is more than just fitting two parts together tightly. Different applications call for different degrees of tightness, for different degrees of clearance in running and sliding fits, and for different degrees of tightness or clearance in locational fits. Assembling parts with clearance fits is no problem. Technical orders prescribe the method for assembling parts with interference fits (light blows with a rawhide mallet, pressing into place with an arbor press, etc.), which will prevent damage to the parts. Suppose that, in two different assemblies in which the interference tolerances are identical, the technical order prescribes forcing the inner part into the outer part in one assembly and prescribes shrinking the inner part in the other assembly. Why? Because the resistance to slippage (whether axial or rotational) in a shrink fit is about three times greater than in a force fit.

12. Rework of Machine Parts

12-1. Even though you took pains to machine all parts to the required tolerances and specifications, you find that they just don't fit. Don't be discouraged. This happens to the best of machinists. At times like these, parts need to be reworked. In the following paragraphs we will discuss some of the conditions that require the reworking of parts, and some possible ways to do it.

12-2. You cannot accurately machine a part on a lathe unless the lathe is in good operating condition, the work setup and the tool setup is rigid, and the cutting tool is sharp. Thin sections must be rigidly supported to avoid deflection. A new three-jaw universal chuck may have a runout of 0.003 inch. A grinding allowance is specified for some parts, and even though a grinding allowance is not specified, grinding, in some cases, is necessary to obtain the desired dimensions or finish. For some jobs, a finishing operation, such as honing, lapping, polishing, or buffing, is specified in the technical order. Some little imperfection in the metal, such as a high spot in the bore of a cylinder, may make honing necessary.

12-3. Reworking for final fitting and assembly often involves bench work operations. If you find that a part binds because of insufficient clearance, you can work the part to fit by hand, by filing, scraping, or polishing. Hardened parts require hand-grinding or the use of abrasive cloths and polishing compounds. If large amounts of material have to be removed, you may find it necessary to remachine the part. On occasion, you will find it more practical to remake the part. You should fit all parts before heat treatment if possible. Hardened parts are difficult to work.

12-4. Your proficiency in reworking machined parts depends, to a large extent, upon having in the shop an adequate number of measuring tools in good condition. It also depends upon your skill in using and reading measuring devices, and upon machine tools that are in good operating condition. You will gain further proficiency through the practice you receive in on-the-job training.

13. Functional Tryout and Inspection of End Products

13-1. Checking assemblies for proper operation is usually the final part of your job. After you have fabricated the parts, they must fit together correctly so that they will do the job they were intended to do. Assume that you have designed and made the parts for a die set. A die set has a number of component parts that must be assembled, such as the punch, die shoe, guide pins, springs, and various items of hardware. The only way that you can determine whether or not the die set will work is to put it together, mount it on the applicable machine, and perform a functional tryout. If everything performs correctly during the functional tryout, you have a good operational assembly. If the die set does not perform correctly, you must find the cause of the trouble and correct it. Perhaps the guide pins are not alined; and you must rework or reposition them. Perhaps the clearance between the punch and the die is incorrect, and you must recalculate the clearance. Careful checking during the making of each individual part reduces the possibility that your work must be done over.

13-2. **Troubleshooting Machine Operations.** One of your most important duties as a machinist is troubleshooting machining problems that arise during the manufacture of parts. We will list some of the more common troubles, causes, and possible remedies as follows:

a. Tool chatter. Cause—improperly ground tools or excessive speed and feed. Remedy—regrind the tools correctly or recalculate the speed and feed for the material being machined.

b. Inaccurate parts. Cause—thermal expansion, dull cutting tools, or incorrect machine adjustment. Remedy—use a coolant, sharpen the tools, or correctly adjust the machine.

c. Failure of the punch to complete a blanking operation. Cause—incorrect die clearance, press not large enough, incorrect alinement of the punch and die shoe. Remedy—recheck the clearance, recalculate the press tonnage for the job, or check the die set alinement.

d. Blanked parts stick to the punch. Cause—lack of a stripper or lack of lubricant. Remedy—providing a stripping device to remove the parts from the punch or provide a means of lubricating the material being blanked.

e. Tooling marks on formed parts. Cause—machine toolmarks on die parts. Remedy—buff or polish all marks from both the male and female members of the die set.

f. Die set breaks down during operation. Cause—improper design, incorrect material, improper heat treatment, or incorrect alinement. Remedy—redesign the die set, select a better material for the die set, make certain that it has been properly heat-treated, or recheck the alinement.

13-3. **Serviceability of the End Product.** The production of serviceable end products is the final test of your workmanship. You must select the proper material for the part, and design and fabricate a tool or tools to produce accurate parts. Parts are serviceable only if they perform the job they were made for. The failure of parts in use or the production of inaccurate work can usually be traced to the selection of the wrong material, to improper design, or to inaccurate layout of either the part or of the tools used in the part fabrication. You can minimize the production of unserviceable parts by paying careful attention to material selection, design, and the layout of both the part and the tools that are used to make the part.

Bibliography

Books

OBERG, Erik, and Jones, Franklin D. *Machinery's Handbook*, 18th ed. New York: The Industrial Press, 1970.

WILSON, Frank W. (Editor). *Fundamentals of Tool Design*. Englewood Cliffs, New Jersey: Prentice-Hall, Inc., 1962.

WITZEL, Ewald L. (Editor). *Jib and Fixture Design*, Volume 1. Albana, New York: Delmar Publishers, Inc., 1947.

Department of the Air Force Publications

AFM 127-101, *Industrial Safety Accident Prevention Handbook*, 26 Jan 70.

Resident Courses

Training Literature from Block I, ABR53130, *Machinist*.

Training Literature from Block VII, ABR53130, *Machinist*.

Training Literature from Block II, 3AZR53150-1, *Machinist*.

Training Literature from Block III, AZR53150-1, *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*.)

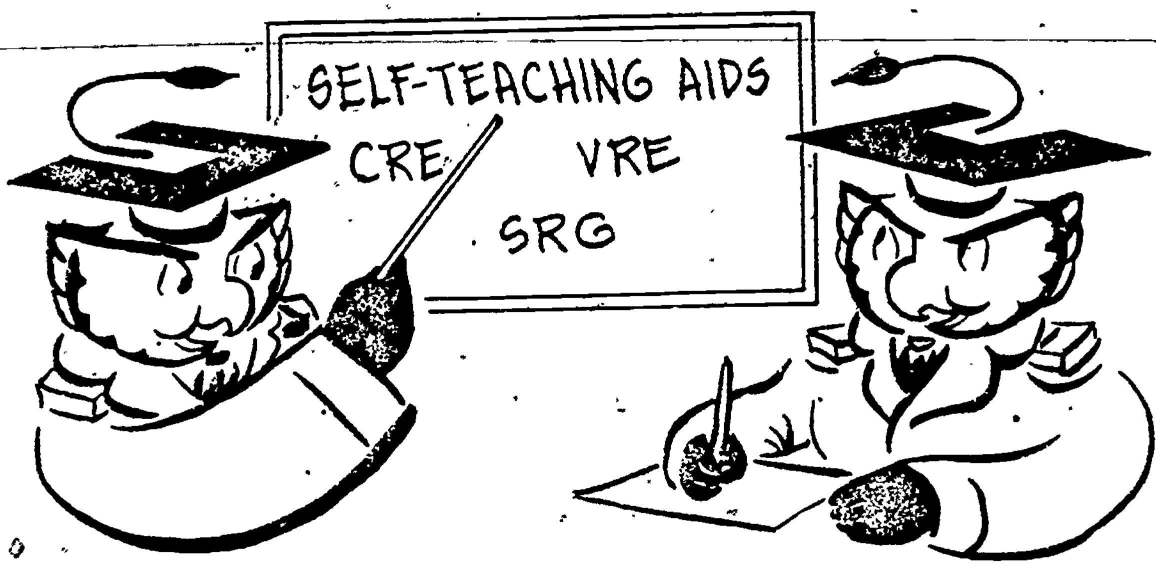
215

13-3

53150 03 23

WORKBOOK

Tool Design and Fabrication



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

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 308

300 Introduction. Freehand Sketching, Tolerancing, and Dimensioning; pages 1–4

301 Use of Standard Parts and Fabricating Processes: Design Related to Heat Treatment; pages 4–8

302 Introduction, Design of Hand and Cutting Tools; Design of Special Gages; pages 9–11

303 Fabricating Processes; pages 11–13

304 Introduction; Types, Uses, and Design of Jigs and Fixtures, pages 14–17

Guide Number

305 Fabrication of Jigs and Fixture, pages 17–20

306 Introduction; Types, Uses, and Designs of Dies; pages 21–28

307 Fabrication of Dies; pages 28–30

308 Introduction; Inspection, Fitting, and Assembly of Machined Parts; Rework of Machine Parts: Functional Tryout and Inspection of End Products; pages 31–34

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 knowledge of freehand sketching, tolerancing, and dimensioning, the use of standard parts and fabricating processes, and the effects of design on heat treatment.

1. How much information is required on a freehand sketch? (1-2)

2. What is the best type of eraser to use for erasing a sketch? (1-3)

3. Cross-section paper is ruled into squares of what size? (1-5)

4. How should lines be drawn in freehand sketching? (1-6)

5. What is the first step in reading a blueprint? (1-10)

6. How much space should you leave between extension lines and object lines? (1-12)

7. List three types of dimensions. (1-13)

8. Define work tolerance. (1-15)

9. Define allowance. (1-17)

10. List four of the most common fastening devices. (2-2)

11. How are castle nuts used? (2-5,*c*)

12. If you must specify the heat-treating process for a tool, what factors must you consider? (3-2)

13. What effect will an abrupt change in mass or size have on a part that is being heat-treated? (3-5)

CHAPTER 2

Objectives. To show an adequate knowledge of the design of hand and cutting tools, of the design of special gages, and of fabricating processes.

1. List the three steps in tool planning. (4-3)

2. List five types of common tools that you may be called upon to design. (4-5—9)

3. What are gages? (5-1)

4. What is the most common type of gage? (5-5)

5. List four types of gages. (5-5)

6. List two nonmetallic materials from which punches and dies are made. (6-3)

7. What type of tool steel should be used when distortion must be held to a minimum? (6-5)

8. List some of the more common nonferrous tool materials. (6-7)

9. What is one of the most important causes of tool breakdown? (6-9)

10. Thermal expansion must be especially taken into consideration under what condition? (6-13)

11. What are toolmaker's buttons used for? (6-16)

12. What is the toolmaker's knee used for? (6-19)

13. What is the first step toward developing a machining requirements plan? (6-20)

14. Why should your machining requirements plan contain a materials, parts, and tools list? (6-21)

15. Why is it good practice to group and perform like machining operations at the same time? (6-23)

CHAPTER 3

Objectives: To show an adequate knowledge of the types, uses, and design of jigs and fixtures, and their fabrication.

1. Explain the difference between the terms "jig" and "fixture." (7-1)

2. For what type of work would a template jig be used? (7-3)

3. What type of parts would you use a channel jig to drill? (7-5)

4. What type of jig would be used on a part in which holes must be drilled from several directions? (7-7)

5. How can you make it easier to position a heavy and hard-to-handle part for the drilling operation? (7-9)

6. What type of jig can be used for more than one operation, such as drilling, reaming, and countersinking? (7-10)

7. What type of bushing should be used for limited production and for only one operation? (7-12)

8. What type of bushing should be used if several operations are to be performed, such as drilling, reaming, and counterboring? (7-14)

9. What is one disadvantage of the screw-in type bushing? (7-15)

10. What determines the class of a fixture? (7-18)

11. What is the simplest type of fixture? (7-20)

12. Upon what portion of an angle plate fixture is the workpiece clamped? (7-21)

13. List the three most common types of body construction used in jig and fixture work. (8-1)

14. List two advantages of welded body type jigs or fixtures. (8-3)

15. In how many linear directions must a part be clamped in order to secure complete confinement? (8-6)

16. Explain the 3-2-1 principle of rest button location. (8-8)

17. What governs the type and number of locators to be used? (8-9)

18. What can be done to minimize the effect of variations of rough unmachined parts? (8-12)

19. How do you determine which type of locator will be best suited for the job? (8-13)

20. What type of locator is best suited to locating from holes with large tolerances? (8-16)

CHAPTER 4

Objectives: To show an adequate knowledge of the types, uses, design, and fabrication of dies.

1. List the seven general types of die operations. (9-3–9)

2. What two factors must be considered with regard to location of blanks in the stock? (9-11)

3. Why should a part be bent as nearly as possible to a right angle to the grain? (9-13)

4. List the five standard die sets. (9-15–18)

5. Give two advantages of the inverted die. (9-21)

6. What is the main advantage of a progressive die? (9-23)

7. State some advantages of the compound die. (9-26–28)

8. Explain the use of stock stops. (9-29)

9. What is the primary function of a stripper? (9-30)

10. Why should you counterbore holes in both stripper and punch plate when you use springs with too large a diameter? (9-34)

11. How is clearance specified for a die that is to be used for blanking a symmetrical part? (9-36)

12. How much angular relief should be given a die used for a relatively small number of blanks? (9-38)

13 Why should a die have a margin of 1¼ inches around the die opening? (10-3)

14. To which die member should clearance be applied if the blanked part is to be the workpiece? Why? (10-4)

15. How is blanking pressure reduced? (10-6)

CHAPTER 5

Objectives To show an adequate understanding of the inspection, fitting, and assembly of machined parts, the reworking of machined parts, and the functional tryout and inspection of end products.

1. What specialist in the metalworking field would you consult to determine (a) whether or not a piece of metal contains internal flaws, and (b) whether or not a piece of metal meets the specified requirement for hardness? (11-1)

2. Why is it impossible to machine a part to mathematically exact dimensions? (11-3–7)

3. What can a shop foreman do to insure that machined parts are accurately measured? (11-5–7)

224

4. How is the type of fit determined for mating machined parts? (11-9)

5. Which will result in a greater resistance to slippage–driving a hub into the hole in a gear blank with a rawhide mallet or shrinking the hub to fit the hole? (11-12)

6. Give the best method of removing a small imperfection, such as a high spot from the bore of a cylinder (12-2)

7. What is the primary cause of tool chatter? (13-2,*a*)

8. What produces tooling marks on formed parts? (13-2,*e*)

9. What determines when a part is serviceable? (13-3)

223

225

ANSWERS FOR CHAPTER REVIEW EXERCISES

CHAPTER 1

1. As much as is necessary to serve the purpose for which the sketch was made.

2. Artgum or pink pearl.

3. One inch. The one-inch squares are subdivided into one-eighth or one-tenth inch squares.

4. Draw lines with a series of short strokes instead of trying to draw each line with one stroke.

5. To determine the shape of an object.

6. 1/16 inch.

7. Detail, position, and overall.

8. The amount of variation permitted in the dimensions or surfaces of machined parts.

9. The intentional difference in the dimensions of mating parts.

10. Bolts, screws, nuts, and pins.

11. They are used with bolts that are drilled to receive a cotter pin or lock wire for safetying.

12. Consider the future use of the tool and the mechanical properties it needs.

13. It may cause serious stresses to develop during heat treatment.

CHAPTER 2

1. (1) Determine what the tool must do. (2) Select or invent a device to meet the requirements. (3) Construct the device to perform the required task most efficiently.

2. Jigs, fixtures, dies, cutting tools, and gages.

3. Devices for determining whether or not the dimensions of a manufactured part are within specified limits.

4. Fixed gages.

5. Ring, receiving, plug, pin, snap, thread, and form.

6. Rubber and thermosetting plastics.

7. Nondeforming tool steel.

8. Tungsten, tantalum, titanium, columbium, and cobalt in their carbide forms.

9. Material defects.

10. When you are machining and holding dissimilar metals.

11. Toolmaker's buttons are used for the accurate location and layout of holes.

12. For the machining of angles and compound angles.

13. Study the design and parts to be produced.

14. To identify needed and available items, and to enable you to substitute and improvise or use alternate methods and techniques.

15. To enable you to keep work and tooling setups to a minimum.

CHAPTER 3

1. The basic difference between jigs and fixtures is that the jig guides the cutting tool while the fixture only holds work in relation to the cutting tool.

2. For limited production and accuracy in layout.

3. The channel jig is made for parts of simple symmetrical shape.

4. Closed or box types.

5. Mount the jig on trunnions or bearings.

6. Combination jig.

7. Press-fit bushing.

8. Slip renewable bushing.

9. When the threads become worn, the screw-in bushing becomes inaccurate.

10. The machine on which it is used.

11. Auxiliary vise jaws.

12. The workpiece is located and securely clamped on the face of the angle plate.

13. Builtup, welded, and cast or one-piece types.

14. They are strong and rigid, and are easily altered.

15. Restriction of movement in three directions will secure complete confinement.

16. Three rest buttons are located on the base to keep the part from rocking. Two buttons are placed on the vertical member next to one of the long sides of the part. One button is placed on the vertical member next to the end of the part.

17. The finish of the locating surface of the part.

18. Keep the locating points as far apart as possible.

19. The shape of the part determines the type of locator to be used.

20. Conical type locators.

CHAPTER 4

1. Blanking, piercing, bending, drawing, crimping, coining, and deep drawing.

2. The best location to save material and the best location to secure good bending where required.

3. The nearer the part comes to being directly across the grain, the less trouble will be experienced later from breaking in the formed part.

4. Back-pin, center-pin, diagonal-pin, round-die, and four-pin die sets.

5. The inverted die lessens the possibility of bending a blank made of thin stock; and, since the cutting edges are kept free of chips, they need less regrinding.

6. Their main advantage lies in their speed of operation.

7. It is very accurate, it can eliminate shaving operations, and it is advantageous in holding close tolerances.

8. They are used to locate the workpiece in the die properly.

9. Strippers are used to move the stock from the punch after blanking or piercing operations.

10. To prevent the springs from cocking sideways and breaking.

11. The total difference between the sizes of the punch and the die.

12. 1° [illegible] 5°

13. To insure that the die does not break during the blanking operation. It also provides enough material for cap screws and dowel pins.

14. Clearance should be applied to the punch because the die cavity controls the size of the blank.

15. By the addition of shear to the punch or die.

CHAPTER 5

1. *a.* The nondestrictive inspection specialist.
   b. The metals processing specialist.

2. It is impossible to machine a part to mathematically exact dimensions because of inherent error in the machine tool, the construction of the measuring device, and in the reading of the measuring device.

3. To insure that machined parts will be accurately measured, a shop foreman can (1) send measuring tools on schedule to the PME shop for inspection and calibration; (2) refuse to tolerate any abuse in handling, maintenance, and storage of measuring tools; (3) and provide training in the use and reading of measuring devices.

4. The type of fit is determined by the purpose for which the parts are intended to serve.

5. Shrinking the hub to fit the hole results in greater resistance to slippage.

6. Honing.

7. Improperly ground tools or excessive speed and feed.

8. Machine toolmarks on die parts.

9. When the part performs the job for which it was made.

229

STOP–

1. MATCH ANSWER SHEET TO THIS EXERCISE NUMBER.

2. USE NUMBER 1 PENCIL.

53150 03 23

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 numerical sequence on answer sheet alternates across from column to column.
3. Use only medium sharp #1 black lead pencil for marking answer sheet.
4. Circle the correct answer in this test booklet. 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 with a #1 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

Chapter 1

1. (001) Which eraser is the best for erasing on sketches and drawings?

   a. Pencil and ink. c. Artgum and pink pearl.
   b. The eraser on your pencil. d. Typing eraser.

2. (003-004) The space left between extension lines and the object on a drawing should be about

   a. 1/32 inch. c. 3/32 inch.
   b. 1/16 inch. d. 1/8 inch.

3. (004) The amount of permitted variation in the dimension or surface of machine parts is called

   a. allowance. c. range.
   b. tolerance. d. limits.

4. (004) When the tolerance of a part is stated in only one direction, it is called

   a. a unilateral tolerance. c. a limit.
   b. a bilateral tolerance. d. an allowance.

5. (004) Which of the following is *not* indicated on the head of an aircraft bolt?

   a. Material the bolt is made of.
   b. Whether or not it is a special bolt.
   c. Size of the bolt.
   d. Whether or not it is a close tolerance bolt.

6. (005) Which of the following types of screws should be used on a part having countersunk holes?

   a. Fillister head screws. c. Round head screws.
   b. Flat head screws. d. Button head screws.

7. (007) Which of the following types of bolts would be best suited for use in bolting two pieces of wood?

   a. Machine bolts. c. Stove bolts.
   b. Carriage bolts. d. Aircraft bolts.

8. (005) What type of nut would most likely be used with a cotter pin?

   a. Plain aircraft nut. c. Shear nut.
   b. Checknut. d. Castle nut.

9. (005) The type of nut that should be used to lock another nut in place is a

   a. plain aircraft nut. c. shear nut.
   b. checknut. d. castle nut.

10. (006) Which of the following should be used primarily for alinement on securing of mating parts?

a. Taper pins.
b. Close tolerance bolt.
c. Straight pins.
d. Roll pins.

11. (008) Which of the following heat treatment operations is the most serious cause of residual stress in parts?

a. Hardening.
b. Carburizing.
c. Annealing.
d. Normalizing.

Chapter 2

12. (009) Which of the following is *not* a consideration in tool planning?

a. Operation required.
b. Locating points.
c. Dimensions and tolerance.
d. Surface finish.

13. (009) Jigs are more often classified according to

a. their construction.
b. their operation.
c. the type of bushing used.
d. the machine upon which they are used.

14. (010) Steel to be used for making gages should *not* contain

a. phosphorus.
b. sulphur.
c. manganese.
d. silicon.

15. (011) Which of the following gages should be used to check the diameter of a shaft?

a. Form gage.
b. Receiving gage.
c. Snap gage.
d. Plug gage.

16. (011) Which of the following tool materials would be used to fabricate a die having an intricate shape?

a. Plain carbon steel.
b. High-speed steel.
c. Nonferrous tool materials.
d. Nondeforming tool steel.

17. (011) Which of the following tool materials is prone to be erratic in response to heat treatment?

a. Plain carbon steel.
b. High-speed tool steel.
c. Nondeforming tool steel.
d. Nonferrous tool materials.

18. (012) The best way to keep thermal expansion to a minimum when machining dissimilar [illegible] use

a. slow R.P.M. and slow feed.
b. slow R.P.M. and fast feed.
c. sharp tools and an ample volume of coolant.
d. fast R.P.M. and slow feed.

19. (012–013) Which of the following methods of layout would afford a very accurate layout [illegible] used to drill four holes in a rectangular workpiece?

a. Toolmaker's buttons.
b. Vernier height gage.
c. Toolmaker's knee.
d. Disc.

Chapter 3

20. (014) The primary difference between a jig and a fixture is that the jig

a. is fixed to the table.
b. locates the workpiece in relation to the cutting tool.
c. guides the cutting tool.
d. has gages for positioning the cutting tool.

21. (014) What type of jig should be used for limited production and should be used more for accurate layout than as a labor- or time-saving device?

a. Plate type jig.
b. Channel jig.
c. Angle plate jig.
d. Template jig.

22. (015) Which of the following types of jigs would be best suited for drilling setscrew holes in pulleys?

a. Channel jig.
b. Angle plate jig.
c. Indexing jig.
d. Closed or box jig.

23. (015) Which type of jig would be used for drilling holes at angles to each other?

a. Channel jig.
b. Indexing or rotary jig.
c. Closed or box jig.
d. Angle plate jig.

24. (015) Combination drill jigs are made possible by the use of

a. slip renewable bushings.
b. a combination of boring and drilling jigs.
c. two or more types of jigs.
d. a combination of closed and indexing jigs.

25. (015) Which of the following types of drill bushings should be used in a simple type of jig for limited production of parts?

a. Fixed-renewable.
b. Slip-renewable.
c. Press-fit.
d. Screw.

26. (016) Which type of bushing should be used for clamping the workpiece as well as for guiding the tool?

    a. Press-fit.
    b. Slip-renewable.
    c. Fixed-renewable.
    d. Screw.

27. (016) The class of a fixture is determined by the type of

    a. machine on which it is used.
    b. operation.
    c. cutter to be used.
    d. work used in it.

28. (017) Fixtures so designed as to permit removal of a completed part while another part is being machined are called

    a. auxiliary use jaws.
    b. angle plate fixtures.
    c. duplex fixtures.
    d. plate fixtures.

29. (017–018) Which of the following body constructions is the most versatile and is often preferred?

    a. Builtup type.
    b. Welded type.
    c. Cast type.
    d. One-piece type.

30. (018) In how many directions must a free body be restricted in order to prevent any motion?

    a. Three directions.
    b. Four directions.
    c. Five directions.
    d. Six directions.

31. (018–019) What governs the type and number of locators required to position a part?

    a. Type of operation to be performed.
    b. Surface condition of the part.
    c. The number of operations to be performed.
    d. All parts require the same number and type of locators.

32. (019) When a part is found to have an untrue surface after being machined in a fixture, the most probable cause could be

    a. too many rest buttons.
    b. not enough rest buttons.
    c. improperly clamped.
    d. uneven rest buttons.

33. (019–020) The type of locator that is often used to hold work between centers on a lathe is a

    a. cylindrical locator.
    b. conical locator.
    c. V-locator.
    d. stop-pen locator.

Chapter 4

34. (021) The press operation that cuts out a part with a punch and die is

a. bending.
b. forming.
c. piercing.
d. blanking.

35. (022) Which of the following press operations should be used in the production of cups, shells, and boxes?

a. Drawing.
b. Forming.
c. Binding.
d. Crimping.

36. (022) The press operation that consists of pinching or squeezing the sides of a shell in order to hold an object is

a. piercing.
b. coining.
c. crimping.
d. drawing.

37. (022) What press operation should be used in the manufacture of military medals?

a. Coining.
b. Drawing.
c. Crimping.
d. Blanking.

38. (022) Which of the following forming operations can *not* be completed in one operation in the cold working of metal?

a. Drawing.
b. Coining.
c. Crimping.
d. Deep drawing.

39. (023) In relation to the grain of the metal being bent, bends should *not* be laid out at less than

a. 15°.
b. 20°.
c. 30°.
d. 45°.

40. (023) What type of die set is usually used to mount a coining die?

a. Back-pin.
b. Diagonal-pin.
c. Round-die.
d. Four-pin.

41. (023–024) What type of standard die set should you select when maximum rigidity and accuracy of alinement are required?

a. Four-pin set.
b. Diagonal-pin set.
c. Back-pin set.
d. Center-pin set.

42. (024) In arranging the sequence of operations for a progressive die, which type of operation should be performed last?

a. Blanking.
b. Forming.
c. Piercing.
d. Trimming.

43. (024) In arranging the sequence of operations for a progressive die, which of the following types of operation should be performed first?

a. Blanking.
b. Forming.
c. Piercing.
d. Trimming.

44. (026--027) A disadvantage of the stop-pin method of locating stock is that it

a. requires considerable skill.
b. will not be as accurate as mechanical stops.
c. is not as fast as mechanical stops.
d. is more dangerous than mechanical stops.

45. (027) The width of the groove for channel strippers should be at least 0.005 inch larger than the width of the stock, and the height of the groove should be

a. 1½ times the thickness of the stock.
b. 2 times the thickness of the stock.
c. 2½ times the thickness of the stock.
d. 3 times the thickness of the stock.

46. (028) What type of pressure pad for dies would be satisfactory for short runs when deflection is *not* too great and there is only a slight possibility of oil splash?

a. Spring-actuated.
b. Rubber-cushion.
c. Combination stripper- and pressure-pad.
d. Combination spring- and cushion-actuated.

47. (028) Die clearance should be stated as being on one side *only* in working with part of a

a. symmetrical form.
b. nonsymmetrical form.
c. rectangular shape.
d. square shape.

48. (028) Why does heavier material require greater clearance between the punch and die than thin material?

a. To prevent ragged or burred edges.
b. To allow the stock to be stripped more easily.
c. To lessen the possibility of breaking the punch or die.
d. Greater clearance will increase the amount of blanking pressure.

49. (028) Using the formula stock thickness multiplied by 5 percent, what would the clearance be for a die to blank 0.125-inch brass?

a. 0.005-inch.
b. 0.006-inch.
c. 0.007-inch.
d. 0.008-inch.

50. (028) What amount of angular relief is ordinarily given a blanking die?

a. 1/8° to 1°.
b. ¼° to 1°.
c. 1/8° to 2°.
d. ¼° to 2°.

51. (029) What should be the width of the margin around the die opening?

a. 1 1/8 inches.
b. 1¼ inches.
c. 1½ inches
d. 2½ inches.

52. (029) By placing suitable shear on the punch or die, the amount of blanking pressure may be reduced by as much as

a. 30 percent.
b. 40 percent.
c. 50 percent.
d. 75 percent.

Chapter 5

53. (031–032) Which of the following factors involving errors in measurement are you unable to correct?

a. Instrument error.
b. Error in use of tools.
c. Reading error.
d. Inherent error of indication.

54. (032) Which of the following is *not* one of the general groups of fits according to the American Standard Fits System?

a. Wringing.
b. Locational.
c. Force fits.
d. Running and sliding.

55. (032) Which of the following fits is the tightest that can be used on a cast-iron external member?

a. FN–1 light drive fit.
b. FN–2 medium drive fit.
c. FN–3 heavy drive fit.
d. FN–4 shrink fit.

56. (032) What type of force fit would be suitable for thin sections, long fits, and for assembling a part in a cast-iron external member?

a. FN–1 light drive fit.
b. FN–2 medium drive fit.
c. FN–3 heavy drive fit.
d. FN–4 shrink fit.

57. (032) How many times greater is the holding power of a shrink fit than that of a force fit?

a. 1 time greater.
b. 2 times greater.
c. 3 times greater.
d. 4 times greater.

58. (033) Failure of a die to complete a blanking operation can usually be traced to

a. incorrect material.
b. improper heat treatment.
c. incorrect machine alinement.
d. incorrect die clearance.