TECHNICAL MANUAL

MACHINING EQUIPMENT AND FUNDAMENTALS

F09603-81-C-0108

This manual supersedes TO 34-1-10 dated 26 August 2025.

Published Under Authority of the Secretary of the Air Force

13 NOVEMBER 2025

LIST OF EFFECTIVE PAGES

INSERT LATEST CHANGED PAGES. DESTROY SUPERSEDED PAGES.

NOTE The portion of the text affected by the changes is indicated by a vertical line in the outer margins of the page. Changes to illustrations are indicated by shaded or screened areas, or by miniature pointing hands.

Dates of issue for original and changed pages are:

Original . . . . 0 . . . 13 November 2025

TOTAL NUMBER OF PAGES IN THIS PUBLICATION IS 280, CONSISTING OF THE FOLLOWING:

Page No.	*Change No.
Title	0
A - B	0
C Blank	0
i - xxxiv	0
1-1 - 1-2	0
2-1 - 2-26	0
3-1 - 3-19	0
3-20 Blank	0
4-1 - 4-77	0
4-78 Blank	0
5-1 - 5-59	0
5-60 Blank	0
6-1 - 6-15	0
6-16 Blank	0
7-1 - 7-10	0
8-1 - 8-9	0
8-10 Blank	0
9-1 - 9-6	0
10-1 - 10-8	0
11-1 - 11-5	0
11-6 Blank	0

* Zero in this column indicates an original page.

LIST OF CHANGES

Content Changed	Change Type	Remarks
Content in Title Page	Modified	Updated supersedure statement.
Content in Paragraph 2	Deleted	Updated title.
Content in Paragraph 1.2.2	Modified	Updated step.
Table 1-1	Added	New table.
Content in Table 2-3	Modified	Updated table.
Content in Table 2-3	Modified	Updated table.
Content in Table 2-6	Modified	Updated table.
Content in Paragraph 4.19.23.2.5	Modified	Updated text.
Chapter 11	Added	New chapter.

TABLE OF CONTENTS

TABLE OF CONTENTS - CONTINUED

Chapter Page

TABLE OF CONTENTS - CONTINUED

TABLE OF CONTENTS - CONTINUED

TABLE OF CONTENTS - CONTINUED

TABLE OF CONTENTS - CONTINUED

TABLE OF CONTENTS - CONTINUED

Chapter Page

TABLE OF CONTENTS - CONTINUED

TABLE OF CONTENTS - CONTINUED

TABLE OF CONTENTS - CONTINUED

Chapter Page

TABLE OF CONTENTS - CONTINUED

Chapter Page

TABLE OF CONTENTS - CONTINUED

Chapter Page

TABLE OF CONTENTS - CONTINUED

TABLE OF CONTENTS - CONTINUED

Chapter Page

TABLE OF CONTENTS - CONTINUED

TABLE OF CONTENTS - CONTINUED

TABLE OF CONTENTS - CONTINUED

Chapter Page

TABLE OF CONTENTS - CONTINUED

TABLE OF CONTENTS - CONTINUED

Chapter Page

TABLE OF CONTENTS - CONTINUED

Chapter Page

TABLE OF CONTENTS - CONTINUED

Chapter Page

TABLE OF CONTENTS - CONTINUED

TABLE OF CONTENTS - CONTINUED

Chapter Page

TABLE OF CONTENTS - CONTINUED

TABLE OF CONTENTS - CONTINUED

TABLE OF CONTENTS - CONTINUED

Chapter Page

LIST OF ILLUSTRATIONS

LIST OF ILLUSTRATIONS - CONTINUED

LIST OF ILLUSTRATIONS - CONTINUED

LIST OF TABLES

LIST OF TABLES - CONTINUED

INTRODUCTION

1 PURPOSE.

This Technical Order (TO) covers machining principles and application pertinent to powered machinery, equipment and tooling installed in Aircraft Metals Technology (AMT) shops (2A7X1) in support of field level maintenance, repair, and fabrication of Aerospace Weapon System components and ground support equipment.

2 SCOPE.

This manual consist of the following chapters:

Chapter 1	Introduction
Chapter 2	Upright Drilling Machines
Chapter 3	Grinding Machines
Chapter 4	Lathes
Chapter 5	Milling Machines
Chapter 6	Metal Cutting Saws
Chapter 7	Waterjet Systems
Chapter 8	Punching Shearing Machines
Chapter 9	3D Scanners
Chapter 10	Presses and Brakes

3 SYMBOLS.

-	A dash (-) before an index number denotes a part which is not illustrated.
*	An asterisk (*) flush right of the part number denotes a marking that is to be requisitioned in accordance with DoDI 5330.03.
#	A number sign (#) flush right of a part number indicates that detail parts are listed in a separate manual.
F	The letter “F” before the figure number means “follows” and is used when an assembly or part has not been assigned an index number, and the figure and index number of the preceding part has been used.

4 ABBREVIATIONS.

All abbreviations used in this manual are shown in the list of abbreviations below. Standard abbreviations are in accordance with ASME Y14.38, Abbreviations and Acronyms for Use in Product Definition and Related Documents

°C	degree Celsius
°F	degree Fahrenheit
A	Ampere
AF	Air Force
AFI	Air Force Instruction
AFMAN	Air Force Manuals
AFTO	Air Force Technical Order
ALT	Alternate
AMT	Aircraft Metals Technology
ASM	Aircraft Structural Maintenance
CAD	Computer-Aided Design
CAM	Computer-Aided Manufacturing
CAT	Caterpillar
CBN	Cubic Boron Nitride

CFR	Code of Federal Regulation
CMM	Coordinate Measuring Machine
CNC	Computer Numerical Control
DoD	Department of Defense
DRO	Digital Readout
ESDS	Electrostatic Discharge Sensitive
ETIMS	Enhanced Technical Information Management System
FPM	Feet Per Minute
FPT	Feed Per Tooth
GD&T	Geometric Dimensioning and Tolerancing
HCI	Hardness Critical Item
HSS	High-Speed Steel
IPM	Inches Per Minute
LCD	Liquid Crystal Display
LH	Left Hand
MDI	Manual Data Input
MFTO	Metal Fabrication and Technologies Office
mm	Millimeter
m/s	Meters Per Second
NC	Numerical Control
No.	Number
NPT	National Pipe Thread
NSN	National Stock Numbers
OEM	Original Equipment Manufacturer
OSHA	Occupational Safety and Health Administration
PN	Part Number
PPE	Personal Protective Equipment
PSI	Pounds-force per Square Inch
RPM	Revolutions Per Minute
RH	Right Hand
SFPM	Surface Feet Per Minute
TCTO	Time Compliance Technical Order
TO	Technical Order
UNC	United National Coarse
UNEF	United National Extra Fine
UNF	Unified National Fine
USAF	United States Air Force
USB	Universal Serial Bus
VAC	Volt Alternating Current

5 RELATED PUBLICATIONS.

NOTE

When searching Technical Order (TO) numbers in the Enhanced Technical Information Management System (ETIMS) catalog, please use the wildcard (*) after typing in the TO number. Many TOs are not available in paper format, (i.e., digital (WA-1) or (CD-1)). This ensures TOs in all media formats will populate the search.

The following publications contain information in support of this technical manual:

List of Related Publications

Number	Title
AFI 63-140	Aircraft Structural Integrity Program
AMT 24-001	Reverse Engineering Guidance: Techniques, Tools, and the Best Practices
ASME Y14.38	Abbreviations and Acronyms for Use in Product Definition and Related Documents
DAFMAN 91-203	Air Force Occupational Safety, Fire, and Health Standards
DoDI 5330.03	Single Manager of DoD Document Services
TO 00-5-1	AF Technical Order System
TO 00-25-195	AF Technical Order System Source, Maintenance, and Recoverability Coding of Air Force Weapons, Systems, and Equipment
TO 00-25-234	General Shop Practice Requirements for the Repair, Maintenance, and Test of Electrical Equipment

6 RECORD OF APPLICABLE TIME COMPLIANCE TECHNICAL ORDERS (TCTOS).

List of Time Compliance Technical Orders

TCTO Number	TCTO Title	TCTO Date
None		

7 HCI HARDNESS CRITICAL ITEMS (HCI).

CAUTION

The HCI symbol (HCI) establishes special requirements limiting changes and substitutions and that the specific parts listed must be used to ensure hardness is not degraded.

If included, items with nuclear survivability requirements are marked with the HCI symbol (HCI). All changes to, or proposed substitutions of, HCIs must be approved by the acquiring activity.

8 ELECTROSTATIC DISCHARGE SENSITIVE (ESDS) ITEMS.

CAUTION

All ESDS parts shall be handled in accordance with the ESDS device handling procedures in TO 00-25-234.

If included, items containing ESDS parts are marked with the ESDS symbol ( ).

9 IMPROVEMENT REPORTS.

Recommended changes to this manual shall be submitted in accordance with TO 00-5-1.

SAFETY SUMMARY

10 GENERAL SAFETY INSTRUCTIONS.

This manual describes physical and/or chemical processes which may cause injury or death to personnel, or damage to equipment, if not properly followed. This safety summary includes general safety precautions and instructions that must be understood and applied during operation and maintenance to ensure personnel safety and protection of equipment. Prior to performing any specific task, the WARNINGs, CAUTIONs, and NOTEs included in that task shall be reviewed and understood.

11 WARNINGS, CAUTIONS, AND NOTES.

WARNINGs and CAUTIONs are used in this manual to highlight operating or maintenance procedures, practices, conditions, or statements which are considered essential to protection of personnel (WARNING) or equipment (CAUTION). WARNINGs and CAUTIONs immediately precede the step or procedure to which they apply. WARNINGs and CAUTIONs consist of four parts: heading (WARNING, CAUTION, or icon), a statement of the hazard, minimum precautions, and possible results if disregarded. NOTEs are used in this manual to highlight operating or maintenance procedures, practices, conditions, or statements which are not essential to protection of personnel or equipment. NOTEs may precede or follow the step or procedure, depending upon the information to be highlighted. The headings used and their definitions are as follows:

WARNING

Highlights an essential operating or maintenance procedure, practice, condition, statement, etc. Failure to comply could result in injury to, or death of, personnel or long term health hazards.

CAUTION

Highlights an essential operating or maintenance procedure, practice, condition, statement, etc. Failure to comply could result in damage to, or destruction of, equipment or loss of mission effectiveness.

NOTE

Highlights an essential operating or maintenance procedure, condition, or statement.

CHAPTER 1
INTRODUCTION

1.1 INTRODUCTION.

The information/instruction contained herein are for general use. If a conflict exists between this technical manual and the specific technical manual(s) or other approved data for a particular weapon, end item, equipment, etc., the data applicable to the specific item(s) will govern in all cases.

1.1.1 Instructions.

The use of **shall**, **will**, **should** and **may** in this Technical Order (TO) is as follows:

a. Whenever the word **shall** appears, it shall be interpreted to mean that the requirements are binding.

b. The words **will**, **should** and **may**, shall be interpreted as non-mandatory provisions.

c. The word **will** is used to express declaration of purpose.

d. The word **should** is used to express non-mandatory desired or preferred method of accomplishment.

e. The word **may** is used to express an acceptable or suggest means of accomplishment.

1.2 MACHINERY AND EQUIPMENT PROCUREMENT.

1.2.1 Centrally Procured Machinery and Equipment.

Centrally procured machinery and equipment are purchased by the support equipment and vehicles management directorate at Warner Robins Air Logistics Complex using procurement (3080) funding. The directorate calculates requirements through a defined process within the defense property accountability system. This computation process ensures that all procurement needs are accurately assessed and addressed.

1.2.2 Qualified Products List A-TEAM Drawing Number (No.) 202432237.

This Qualified Products List identifies authorized manufacturers/vendors, their designs, Commercial Item Description references, and the specific process for which the machinery is qualified. This applies to all Aircraft Metals Technology (AMT) sections at field and/or Intermediate level of maintenance. All centrally procured machinery, and equipment must adhere to the published Air Force (AF) Metal Fabrication and Technologies Office (MFTO) Qualified Products List of manufacturing equipment, A-TEAM drawing No. 202432237. This drawing can be retrieved at, https://a-team.us.af.mil/. See also Table 1-1 for a cross-referenced list of associated National Stock Numbers (NSN) for each qualified manufacturing equipment.

1.2.3 Purpose of Centrally Procured Machinery and Equipment.

The Headquarters USAF directs the use of standardized manufacturing equipment and processes whenever possible, assigning engineering authority for (AMT-2A7X1) and (Aircraft Structural Maintenance (ASM)-2A7X3) machinery and equipment to the AF Metal Fabrication and Technologies Office (AFLCMC/EZPT-MFTO) at Robins AFB, GA. The AF MFTO collaborates with industry partners, Major Commands, air logistics complexes, and other military branches to determine the specific technical requirements of machinery and equipment impacting aircraft metals technology and aircraft structural maintenance. These technical requirements are updated in commercial item descriptions and are utilized as the qualification reference in the published qualified products list.

1.2.4 AMT and ASM Equipment Standardization.

AFI 63-140_AFMCSup, requires the Metal Fabrication and Technologies Office, on behalf of AMT and ASM career fields, to:

- Provide engineering and technical support for procurement, repair, and maintainability of centrally procured manufacturing and metal fabrication equipment and serve as the engineering authority for centrally procured metals tech and structural maintenance equipment, product and item specifications, and MT and SM specific equipment requirements, options, and upgrades.

- Support and implement equipment standardization to ease training, maintenance, manufacturing, fastener replacement, machining, welding, and heat treating operations and to improve operational effectiveness and safety across the enterprise.

- Streamline and standardize metals tech Computer Numerical Control (CNC) equipment for field metals tech and structural maintenance sections.

1.2.5 To Expedite Orders of Centrally Procured Equipment. USAF installations shall request D035A priority 3 when unable to perform the mission. Requisitions shall only be for national stock numbers that are qualified to commercial item descriptions per the qualified products list. All other national stock number requisitions should be canceled and replaced with qualified national stock numbers to facilitate standardization within the USAF. For any assistance with procurement of Industrial Plant Equipment, contact the Metal Fabrication and Technologies Office at aflcmc.ezpt-mfto.metalsoffice@us.af.mil.

Table 1-1. Cross Referenced NSN List of Qualified Manufacturing Equipment

Vendor Design/Part Number.	Commercial Item Description Reference No.	NSN
CNC Machining Center		
HAAS/Mini-Mill	A-A-60025-A	3417-01-698-8845
HAAS/TM-2	A-A-60025-B	3417-01-698-8846
HAAS/VF-4	A-A-60025-C	3417-01-698-8847
HAAS/VR-8	A-A-60025-D	3417-01-698-8848
HAAS/UMC-750	A-A-60025-E	3417-01-698-8850
Manual Milling Machine		
Clausing/Bridgeport Series I	A-A-60064-3-49	3411-00-878-7743
CNC Lathe, Flatbed		
HAAS/TL-1	A-A-60023-30	3416-01-698-8843
HAAS/TL-2	A-A-60023-48	3416-01-698-9410
CNC Lathe, Slantbed		
HAAS/ST-10	A-A-60024-160	3416-01-698-9412
HAAS/ST-20	A-A-60024-225	3416-01-698-8832
HAAS/ST-30	A-A-60024-325	3416-01-698-9411
Manual Lathe		
Clausing/8044VSJ	A-A-60063-1-02	3416-01-448-3041
Clausing/CT618DT	A-A-60063-2-01	Pending
Radial Arm Drill Press, Floor Mounted		
Baileigh/RD1000M	A-A-60065-01	3413-01-725-0412
Baileigh/RD1600H	A-A-60065-02	Pending
Standard Drill Press, Floor Mounted		
Baileigh/DP-1250VS	A-A-59457B-01	Pending
Baileigh/DP-1400VS	A-A-59457B-02	Pending
Vertical Bandsaw, Metal Cutting		
DoAll/2013-V5	A-A-60058-20	3405-01-724-8509
DoAll/3613-V5	A-A-60058-36	3405-01-725-5171
Abrasive Waterjet		
OMAX/5555	A-A-60029-11	3449-01-702-7792
OMAX/55100	A-A-60029-12	3449-01-702-7793
OMAX/60X	A-A-60029-23	3449-01-702-7795
OMAX/80X	A-A-60029-24	3449-01-702-7794
Ironworker		
Piranha/P50	A-A-59422-I	Pending
Piranha/P90	A-A-59422-II	Pending
Piranha/P110	A-A-59422-III	Pending

CHAPTER 2
UPRIGHT DRILLING MACHINES

2.1 GENERAL.

Drill presses are versatile tools used to create precise holes in various materials like metal and wood. They use rotating cutting tools held in a chuck, with adjustable speeds for different materials and hole sizes. Beyond drilling, they can also be used for tapping, reaming, counterboring, countersinking, and spot facing.

2.2 SAFETY.

Safety is paramount when operating upright drilling machines, as improper use can result in severe injuries. Operators must always adhere to safety guidelines set forth by Occupational Safety and Health Administration (OSHA) standards (29 CFR 1910.) and applicable Air Force Instructions (AFI) and Air Force Manuals (AFMAN) (e.g., DAFMAN 91-203). Supervisors are responsible for maintaining manufacturer manuals and integrating them into on-the-job training and job safety training. In the absence of manufacturer guidance, a hazard analyses and local operating instructions must be developed. Operators must:

- Wear safety glasses, and may require face shields, hearing protection, and other necessary Personal Protective Equipment (PPE) to protect from flying debris, and noise.
- Ensure all appropriate guards are in place to prevent accidental contact with rotating components.
- Follow lockout/tagout procedures to ensure the machine is powered off when performing maintenance.
- Keep the work area clean and free from clutter to avoid tripping hazards.
- Be familiar with the location and operation of emergency stop buttons and other safety features to quickly shut down the machine in case of an emergency.
- Remove loose clothing, jewelry, or long hair must be secured to prevent entanglement with moving parts.

2.3 TYPES OF DRILLING MACHINES.

Drilling machines can be categorized into two primary types: upright drilling machines with vertical spindles (Figure 2-6 and Figure 2-7) and radial drilling machines with horizontal spindles. The latter are typically reserved for specialized production tasks, whereas upright drilling machines are more widely used in Aircraft Metals Technology (AMT) shops due to their versatility and suitability for general-purpose drilling operations. Two common types of upright drilling machines found in AMT shops are:

- Floor-mounted drilling machines. (See Figure 2-1.)
- Radial arm drilling machines. (See Figure 2-2.)

These versatile machines are widely used for a range of drilling tasks.

2.3.1 Floor-Mounted Upright Drilling Machines. Floor-mounted upright drilling machines (Figure 2-1) are ideal for light to medium-duty drilling. Their adjustable worktables (horizontal, vertical, tilt) allow for accurate drill positioning. The swiveling and height-adjustable head adds versatility for different workpiece setups. Feed control can be manual or powered, with power-fed models offering multiple speeds and automatic feeds. Machine capacity is typically determined by the largest workpiece diameter that can be centrally drilled.

TO-34-1-10-001

Figure 2-1. Floor Mounted Drilling Machine

TO-34-1-10-002

Figure 2-2. Radial Arm Drilling Machine

2.3.2 Radial Arm Drilling Machines. Radial arm drilling machines (Figure 2-2) are well-suited for medium to heavy-duty drilling. Their adjustable worktables (horizontal, vertical, tilt) ensure accurate drill positioning. The swiveling and height-adjustable head increases versatility for different workpiece setups. Feed can be manual or powered; power-fed models offer multiple speeds and automatic feed rates. Machine capacity is typically determined by the largest workpiece diameter that can be centrally drilled.

2.4 TWIST DRILLS.

Twist drills are the standard for drilling machines. These end-cutting tools have cutting edges at the tip and helical flutes for chip removal. While they come with either straight or tapered shanks, most drilling machines are designed for tapered shanks. (See Figure 2-3.) Straight shanks require chucks or adapters to fit. Twist drills are typically made from High-Speed Steel (HSS) or other advanced materials. The most common materials used include:

2.4.1 HSS. A steel alloy containing tungsten, molybdenum, and vanadium, known for its high hardness, wear resistance, and ability to maintain its cutting edge at high temperatures. Suitable for drilling steel, aluminum, and wood.

2.4.2 Cobalt Steel. A type of HSS with a higher percentage of cobalt (5-8 percent), offering increased wear resistance and hardness. Ideal for drilling harder materials like stainless steel and titanium.

2.4.3 Tungsten Carbide. A very hard, wear-resistant material used for drilling hard materials like steel, stainless steel, and titanium. Tungsten carbide drills are more expensive than HSS drills but offer longer tool life and improved performance.

2.4.4 Coated Drills. Some twist drills are coated with a thin layer of titanium nitride, titanium aluminum nitride, or other materials to improve wear resistance and performance. Suitable for drilling a wide range of materials, they offer improved tool life and reduced friction.

2.4.5 Carbon Steel. Less common than HSS or cobalt steel drills, carbon steel drills are still used for drilling softer materials like wood and plastic. They are less expensive than HSS or cobalt steel drills but have a shorter tool life.

2.4.6 Solid Carbide. Made from a single piece of Tungsten carbide, these drills are known for their high hardness and wear resistance. Often used for drilling hard materials, they are more expensive than HSS or cobalt steel drills.

2.4.7 Selecting the Right Material. The choice of material for a twist drill depends on the specific application, the type of material being drilled, and the desired level of performance and tool life. By considering these factors, you can select the most suitable material for your twist drill and ensure optimal drilling performance.

2.4.8 Twist Drill Sizes. Twist drills are made in about 400 standard sizes, ranging from 0.0135 inches (wire gage size Number (No.) 80) to 3-1/2 inches in diameter. Nonstandard sizes up to 6 inches in diameter are used for some special jobs. The standard sizes used in the United States are known as wire gage size number drills, letter size drills, fractional size drills, and millimeter (mm) size drills.

2.4.8.1 Wire Gage Size Twist Drills. Wire gage size twist drills range from No. 80 (0.0135 inches) to No. 1 (0.2280 inches). Refer to Table 2-1 for wire gage size twist drill diameters. Note that the larger the number, the smaller the diameter of the drill.

Table 2-1. Wire Gage Sizes for Twist Drills

Wire Gage Size (No.)	Drill Diameter (Inches)	Wire Gage Size (No.)	Drill Diameter (Inches)	Wire Gage Size (No.)	Drill Diameter (Inches)	Wire Gage Size (No.)	Drill Diameter (Inches)
1	0.2280	21	0.1590	41	0.0960	61	0.0390
2	0.2210	22	0.1570	42	0.0935	62	0.0380
3	0.2130	23	0.1540	43	0.0890	63	0.0370
4	0.2090	24	0.1520	44	0.0860	64	0.0360
5	0.2055	25	0.1495	45	0.0820	65	0.0350
6	0.2040	26	0.1470	46	0.0810	66	0.0330
7	0.2010	27	0.1440	47	0.0785	67	0.0320
8	0.1190	28	0.1405	48	0.0760	68	0.0310
9	0.1960	29	0.1360	49	0.0730	69	0.0292
10	0.1935	30	0.1285	50	0.0700	70	0.0280
11	0.1910	31	0.1200	51	0.0670	71	0.0260
12	0.1890	32	0.1160	52	0.0635	72	0.0250

Table 2-1. Wire Gage Sizes for Twist Drills - Continued

Wire Gage Size (No.)	Drill Diameter (Inches)	Wire Gage Size (No.)	Drill Diameter (Inches)	Wire Gage Size (No.)	Drill Diameter (Inches)	Wire Gage Size (No.)	Drill Diameter (Inches)
13	0.1850	33	0.1130	53	0.0595	73	0.0240
14	0.1820	34	0.1110	54	0.0550	74	0.0225
15	0.1800	35	0.1100	55	0.0520	75	0.0210
16	0.1770	36	0.1065	56	0.0465	76	0.0200
17	0.1730	37	0.1040	57	0.0430	77	0.0180
18	0.1695	38	0.1015	58	0.0420	78	0.0160
19	0.1660	39	0.0995	59	0.0410	79	0.0145
20	0.1610	40	0.0980	60	0.0400	80	0.0135

2.4.8.2 Letter Size Twist Drills. Letter size twist drills range from A (0.234 inches) to Z (0.413 inches). Refer to the table below for letter size twist drill diameters. Note that as the letters progress, the diameters become larger.

Table 2-2. Letter Sizes for Twist Drills

Letter Size	Drill Diameter (Inches)	Letter Size	Drill Diameter (Inches)
A	0.234	N	0.302
B	0.238	O	0.316
C	0.242	P	0.323
D	0.246	Q	0.332
E	0.250	R	0.339
F	0.257	S	0.348
G	0.261	T	0.358
H	0.266	U	0.368
I	0.272	V	0.377
J	0.277	W	0.386
K	0.281	X	0.397
L	0.290	Y	0.404
M	0.295	Z	0.413

2.4.8.3 Fractional Size Drills. Fractional size drills range from 1/64 to 1-3/4 inches in 1/64 inch units, from 1/32 to 2-1/4 inches in 1/32 inch units, and from 1/16 to 3-1/2 inches in 1/16 inch units.

2.4.8.4 MM Size Drills. Millimeter size drills commonly range from 3 mm (0.1181 inches) to 77 mm (3.0315 inches) in 1/2 mm units.

2.4.9 Twist Drill Parts (Nomenclature). Familiarity with the parts of the twist drill is necessary to learn to sharpen drills properly. The parts of the twist drill are illustrated in Figure 2-3 and described below.

Point

The point is the entire conical-shaped end of the drill containing the cutting edges.

Body

The body is the section between the point and the shank containing the flutes.

Shank

The shank is the portion of the drill from the body to the back end and may be straight or tapered. It is held in the chuck or spindle of the drilling machine.

Chisel Edge

The chisel edge, also called the dead center, is the sharp edge at the extreme tip end of the drill across the web. It is formed by the intersection of the conical-shaped surfaces of the point. The edge should always be centered exactly at the end of the drill's axis. The chisel edge cuts its own hole, acting as a flat drill.

Cutting Edge Lips

The sharp edges formed by grinding the flutes to a conical point, and which slope from the chisel edge to the margin of the drill are known as cutting edge lips. These lips cut like a knife when fed and rotated into the workpiece.

Heel

The heel is the conical-shaped portion of the point back of the cutting edge lips.

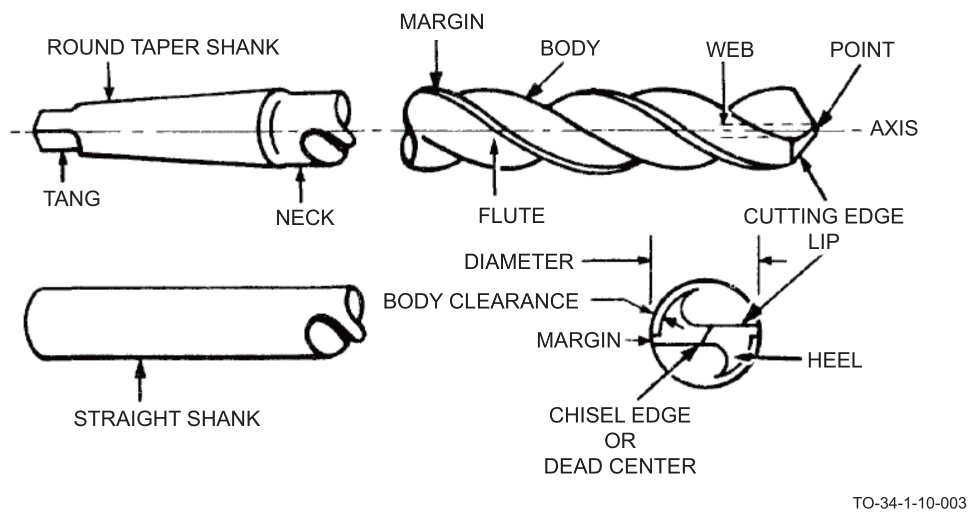


Figure 2-3. Twist Drill Nomenclature

Lip Clearance

The lip clearance, also known as relief angle, on a twist drill refers to **the small angle ground away from the cutting edge along the flank of the drill bit**. This angled surface prevents the flank of the drill bit from rubbing against the newly drilled hole, which would cause friction, heat, and premature wear.

Flutes

The helical grooves which appear twisted around the body of the drill are called flutes. They provide definite advantages to the drilling operation. They give correct rake to the lips, form channels through which chips can escape, permit cutting oil to flow to the edge of the cutting tool and cause chips to curl lightly thus occupying less space.

Margin

The margin is the narrow raised strip of the body which extends the full length of each flute. The distance from edge to edge of the margins (Figure 2-3) is the full diameter of the drill. The margin ensures the hole is the right size.

Body Clearance

Body clearance is the relieved portion behind the drill margin. This reduced diameter prevents friction by ensuring only the margin contacts the hole and allows for lubricant flow.

Tang

The narrowed end of the tapered shank drill is called the tang. The tang fits the slot in the innermost end of the drill spindle, drill chuck, or other drill holding device and aids in driving the tool. It also prevents the drill from slipping.

Web

The web of the drill is the metal section separating the flutes. It runs the length of the body between the flutes. The web gradually increases in thickness toward the shank, increasing the rigidity of the drill.

Axis

An imaginary line through the center of the drill from end to end is the axis. The drill should rotate evenly about the axis at all times.

2.5 SHARPENING DRILLS.

Technicians should be able to sharpen twist drills for the specific metal being cut. While a drill grinder simplifies sharpening, hand grinding may sometimes be necessary. Avoid overheating carbon steel drills as this causes tempering loss; dip them in water periodically.

2.5.1 Grinding Drill Lips. When grinding drill lips, ensure both lips are equal in length and form identical angles with the drill axis. (See Figure 2-4.) Grind to the lip angle specified in Table 2-3.

2.5.2 Grinding the Lip Clearance Angle. The heel of the drill should be ground below the lips at an angle between 12 to 15 degrees measured at the circumference of the drill. (See Figure 2-5.)

2.5.3 Grinding the Rake Angle. The rake angle, formed between the flute and drill axis, determines the cutting edge. (See Figure 2-5.) A 30 degree angle is most common, though it can range from 18 to 45 degrees.

Table 2-3. Lip Angles and Lip Clearances for Twist Drills

Material to be Drilled	Lip Angle (Figure 2-4) (Degree)	Lip Clearance (Figure 2-5) (Degree)
Aluminum	30	12
Aluminum alloys	45	12-15
Bakelite	30	12
Bass	59	12-15
Bronze	59	12-15
Cast iron (soft)	45	12
Copper	50	12½
Copper alloys	50	12
Fiber	30	12
Hard rubber	30	12
Plastic	59	30
Steel (hard)	75	7-12
Steel (heat-treated)	62-1/2	12
Steel (soft and medium)	59	12-15
Titanium	65-67½	8-12
Wood	30	12

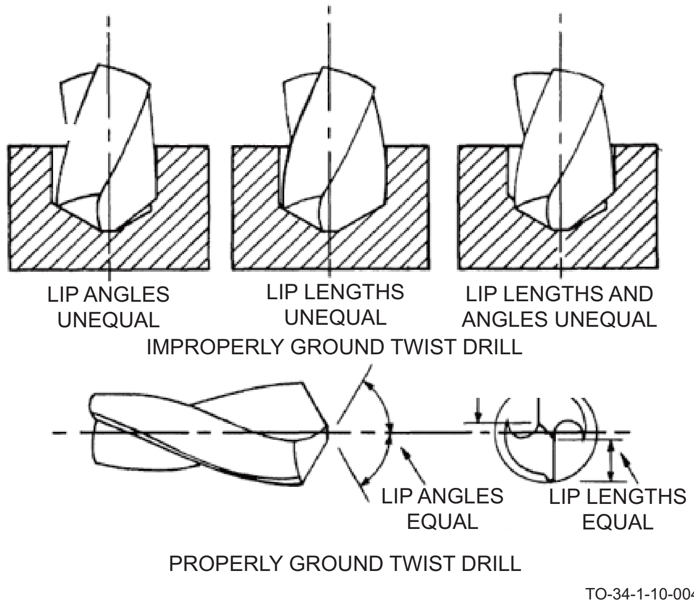


Figure 2-4. Effects of Improperly Ground Lips

2.5.4 Thinning the Web. The web of a drill is made thicker toward the shank to strengthen the tool. In smaller size drills, the difference is not noticeable, but in larger drills, when the point is ground back by repeated sharpening, the thickness of the web becomes greater, and the chisel edge of the drill becomes wider. This causes the chisel edge to scrape on the bottom of the hole and requires excessive pressure to be applied to the drill. This can be corrected by thinning the web.

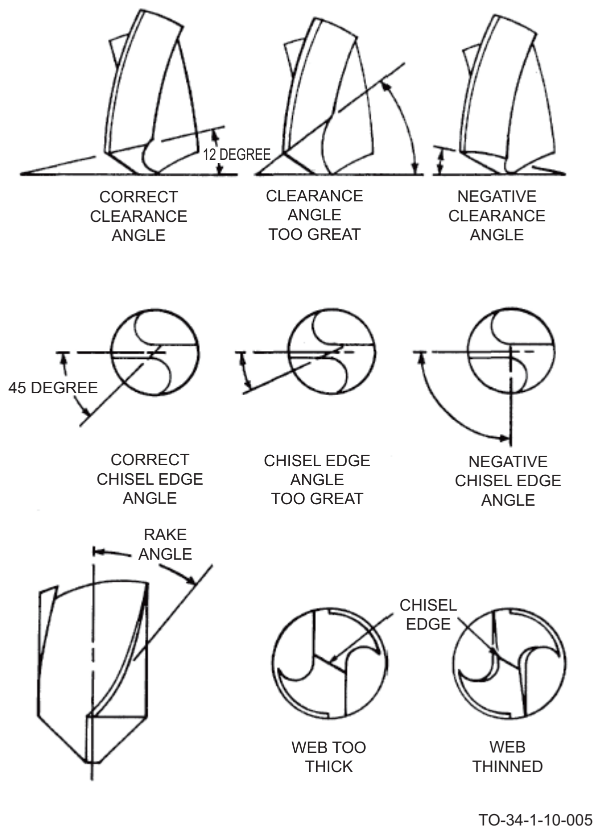


Figure 2-5. Identification of Correct Angles and Web Thickness

2.6 DRILL GRINDERS.

Drill grinders simplify the process of accurately grinding drills of all types and sizes, requiring minimal operator skill. These machines are especially valuable in high-volume applications where numerous drills need to be sharpened regularly, making them an efficient solution for maintaining a large drill inventory.

2.6.1 Bench-Type Drill Grinders. While bench-type drill grinders share a common design, they differ in drill holding mechanisms and capacity. Capacity is determined by motor horsepower and the size range of drills the machine can accommodate. The two main types are:

- Motorized: Traditional design with electric motor.
- Computerized: Offer advanced features and automation.

2.6.1.1 Motorized Drill Grinders. These electrically powered tools (Figure 2-6) provide fast, efficient drill bit sharpening for various industries. Their main function is to sharpen and maintain cutting edges, extending bit life and improving performance. Key components typically include:

2.6.1.1.1 Motor. A high-speed motor that powers the grinding wheel or stone.

2.6.1.1.2 Grinding Wheel or Stone. A rotating abrasive wheel or stone that sharpens the drill bit.

2.6.1.1.3 Drill Bit Holder. A device that securely holds the drill bit in place during the grinding process.

2.6.1.1.4 Adjustment Mechanisms. Controls that allow the operator to adjust the angle, position, and speed of the grinding wheel or stone.

2.6.1.1.5 Safety Features. Guards, shields, or other protective devices to prevent injury from flying debris or accidental contact with the grinding wheel or stone.

2.6.1.2 Computerized Drill Grinders. These machines also known as a Computer Numerical Control (CNC) drill grinder, is a machine tool designed to grind and sharpen drill bits to precise specifications using computer-controlled technology. (See Figure 2-7.) It is a type of grinder that uses a computer program to control the grinding process, allowing for high accuracy, precision, and consistency. Key features of a computerized drill grinder:

2.6.1.2.1 Computer Control. The machine is controlled by a computer program that allows for precise control over the grinding process, including the angle, speed, and feed rate of the grinding wheel.

2.6.1.2.2 Automated Grinding. The machine can automatically grind the drill bit to the desired specifications, including angle, diameter, and flute length.

2.6.1.2.3 Precision Grinding. The machine is capable of grinding drill bits to precise specifications, with tolerances as low as 0.001 inches (0.025 mm).

2.6.1.2.4 Multi-Axis Control. The machine can control multiple axes, allowing for complex grinding operations, such as grinding multiple flutes and angles.

2.6.1.2.5 Touchscreen Interface. The machine typically has a touchscreen interface that allows the operator to easily input grinding parameters, select pre-programmed grinding cycles, and monitor the grinding process.

TO-34-1-10-006

Figure 2-6. Motorized Drill Grinder

TO-34-1-10-007

Figure 2-7. CNC Drill Grinder

2.6.1.2.6 Benefits of Bench-Type Drill Grinders.

2.6.1.2.6.1 Improved Drill Bit Performance. Bench-type drill grinding machines can significantly improve drill bit performance, extending their lifespan and reducing the need for frequent replacements.

2.6.1.2.6.2 Increased Accuracy. These machines enable users to precision-grind drill bits, ensuring accurate point angles and relief angles.

2.6.1.2.6.3 Time Savings. Motorized and computerized machines can significantly reduce grinding time, increasing productivity and efficiency.

2.6.1.2.6.4 Cost Savings. By extending drill bit life and reducing the need for replacements, bench-type drill grinding machines can help reduce costs and improve overall drilling operations.

2.7 DRILL HOLDING DEVICES.

The drilling machine's revolving vertical spindle is designed to hold and drive the cutting tool, featuring a central hole with a standard Morse taper at the bottom, which varies in size depending on the drilling machine. To accommodate various drill sizes and shapes, drill holding devices such as standard drill chucks, keyless (quick-release) drill chucks, drill sockets, and drill sleeves are used to fit the spindle, enabling the use of different tools in different machines. (See Figure 2-8 through Figure 2-10.) Some drilling machines have solid spindles that can accommodate chucks with internal threads or internal tapers, providing further versatility and flexibility.

2.7.1 Drill Chucks. Drills with straight shanks are held in drill chucks, which have two or more adjustable jaws set radially. (See Figure 2-8 and Figure 2-9.) Smaller size drills are made with straight shanks due to the extra cost of providing these sizes with tapers. Chucks are made in various sizes, and a set of chucks may hold cutting tools from the smallest sizes up to 1 inch in diameter. The shank of the chuck is set into the spindle of the drilling machine, and the drill is then locked into the chuck. Some drill chucks feature a quick-release mechanism, which allows for rapid and easy changeover of drill bits, increasing productivity and reducing downtime. These quick-release chucks often have a lever or handle that can be operated by hand, eliminating the need for a chuck key and providing a convenient and ergonomic design. (See Figure 2-9.) When the lever is released, the chuck jaws open, and the drill bit is ejected from the chuck, making it easy to remove and replace the bit. Additionally, some quick-release chucks are self-ejecting, further simplifying the process and reducing the risk of injury. Other advantages of quick-release chucks include:

2.7.1.1 Improved Accuracy. Quick-release chucks ensure a secure and precise connection between the drill bit and the chuck, reducing the risk of runout and improving the overall accuracy of the drilling operation.

2.7.1.2 Increased Efficiency. The ability to quickly and easily change drill bits reduces the time spent on tool changes, allowing for more productive use of the drilling machine.

2.7.1.3 Enhanced Safety. The self-ejecting feature of some quick-release chucks reduces the risk of injury from sharp drill bits and eliminates the need for manual removal of the bit.

2.7.1.4 Versatility. Quick-release chucks can accommodate a wide range of drill bit sizes and types, making them a versatile and essential component of any drilling machine.

2.7.1.5 Durability. Quick-release chucks are designed to withstand the rigors of heavy use and are constructed from high-quality materials to ensure a long service life.

2.7.1.5.1 Morse tapers to fit drill spindles are commonly used on drill chuck shanks, providing a secure and precise connection. Overall, drill chucks, including quick-release types, provide a convenient and versatile way to hold and secure drill bits in drilling machines, making them an essential component of any drilling operation.

2.7.2 Drill Sockets and Drill Sleeves. To accommodate the varying taper sizes of cutting tools and drilling machine spindles, drill sockets and sleeves are used to mount drills in machine spindles. (See Figure 2-10.) These devices, used singly or in combination, enable smaller drills to be fitted into larger spindles by providing a taper hole that matches the drill size and a taper shank that fits the spindle. Multiple sockets or sleeves can be used to adapt drills to larger spindles, and they are available in a range of sizes and taper combinations, offering a flexible solution for drilling operations.

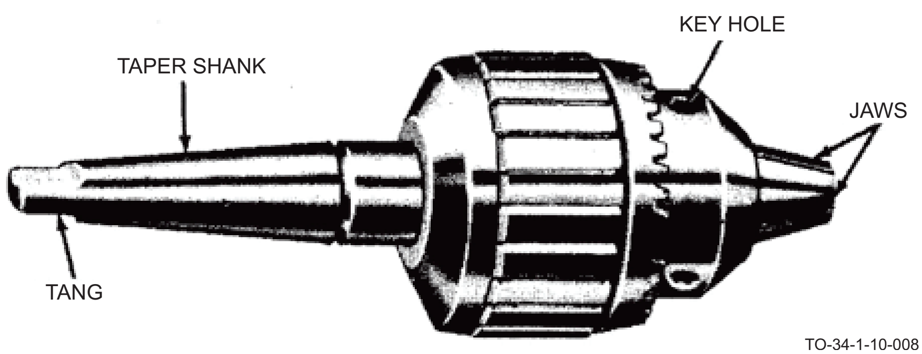


Figure 2-8. Drill Chuck

Figure 2-9. Keyless Drill Chuck

Figure 2-10. Drill Sleeve and Drill Socket

2.8 DRILL DRIFTS.

Drill Drifts (Figure 2-15) are specialized tools used to remove taper-shanked drills and holding devices from drilling machine spindles. These flat, tapered keys have a rounded edge and are designed to be forced through the slots in the drill sleeve, socket, or spindle, where they wedge the drill from its seat, allowing for safe and easy removal.

2.9 WORK-HOLDING DEVICES.

2.9.1 Machine Table Vises. A machine table vise (Figure 2-11) is a device equipped with jaws which clamp against the workpiece, holding it secure. The vise is then bolted to the drilling machine table or supported by a stop which is bolted to the worktable.

2.9.2 Parallels. To prevent the drilling tool from cutting on through the workpiece and into the table or vise, accurately machined parallel bars called parallels (Figure 2-11) are used between the workpiece and the table. The workpiece and parallels are clamped together to the table.

2.9.3 Blocks. A plain block of the required dimensions and suitable material may be used with a clamp to mount workpieces.

2.9.4 Step Blocks. Blocks with steps for workpieces of various heights are extremely useful in mounting different jobs. (See Figure 2-12.)

2.9.5 Clamps. Clamps are small, portable vises or plates which bear against the workpiece and holding devices to steady the job. Clamps are made in numerous shapes to meet the varied work holding problems. Common types of clamps include the C-Clamp, the parallel clamp, and the machine strap clamp. (See Figure 2-12.)

2.9.6 V-Blocks. V-Blocks (Figure 2-12) are used to hold round shaped workpieces. The size of the V-Block used is determined by the diameter of the work.

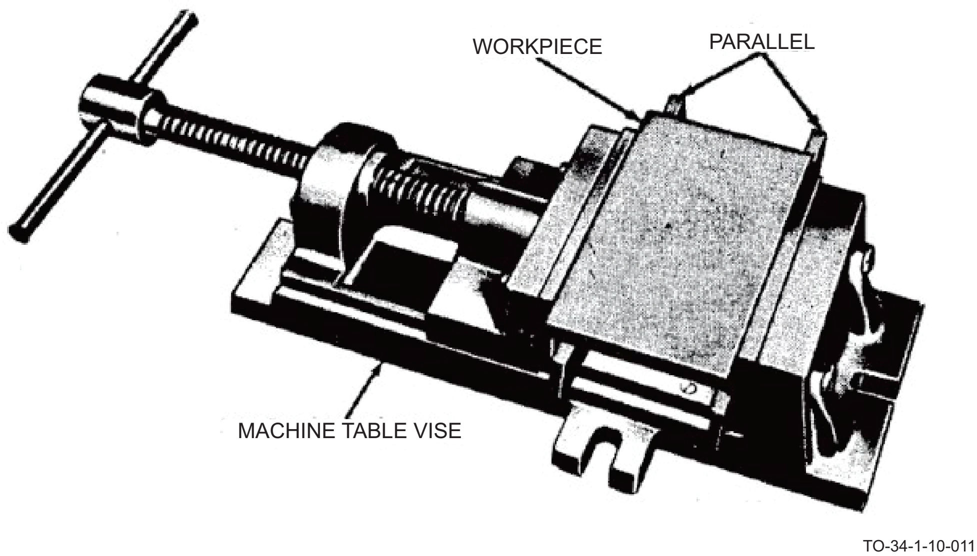


Figure 2-11. Machine Table Vise and Parallels Used to Mount Workpieces

2.9.7 Angle Plates. Angle plates (Figure 2-12) with holes and slots for clamps, or bolts to secure the workpiece and fasten the plates to worktable are used for many drilling machine operations.

2.9.8 T-Slot Bolts. T-slot bolts (Figure 2-12) are inserted in the T-slots in worktables so equipped to fasten the workpiece or work holding device to the table. The studs of the tapped T-slot bolt heads may be removed, and a number of different lengths may be used.

2.9.9 Jigs. Drill jigs are used when several workpieces have to be drilled identically. The workpiece is clamped in the jig so that the hole or holes will be drilled in the same location on each piece. The jig may guide the cutting tool through a steel bushing to locate the holes accurately.

2.9.10 Shims. Thin pieces of metal or other material, called shims are used to build up or support the holding devices when extra adjustments are needed.

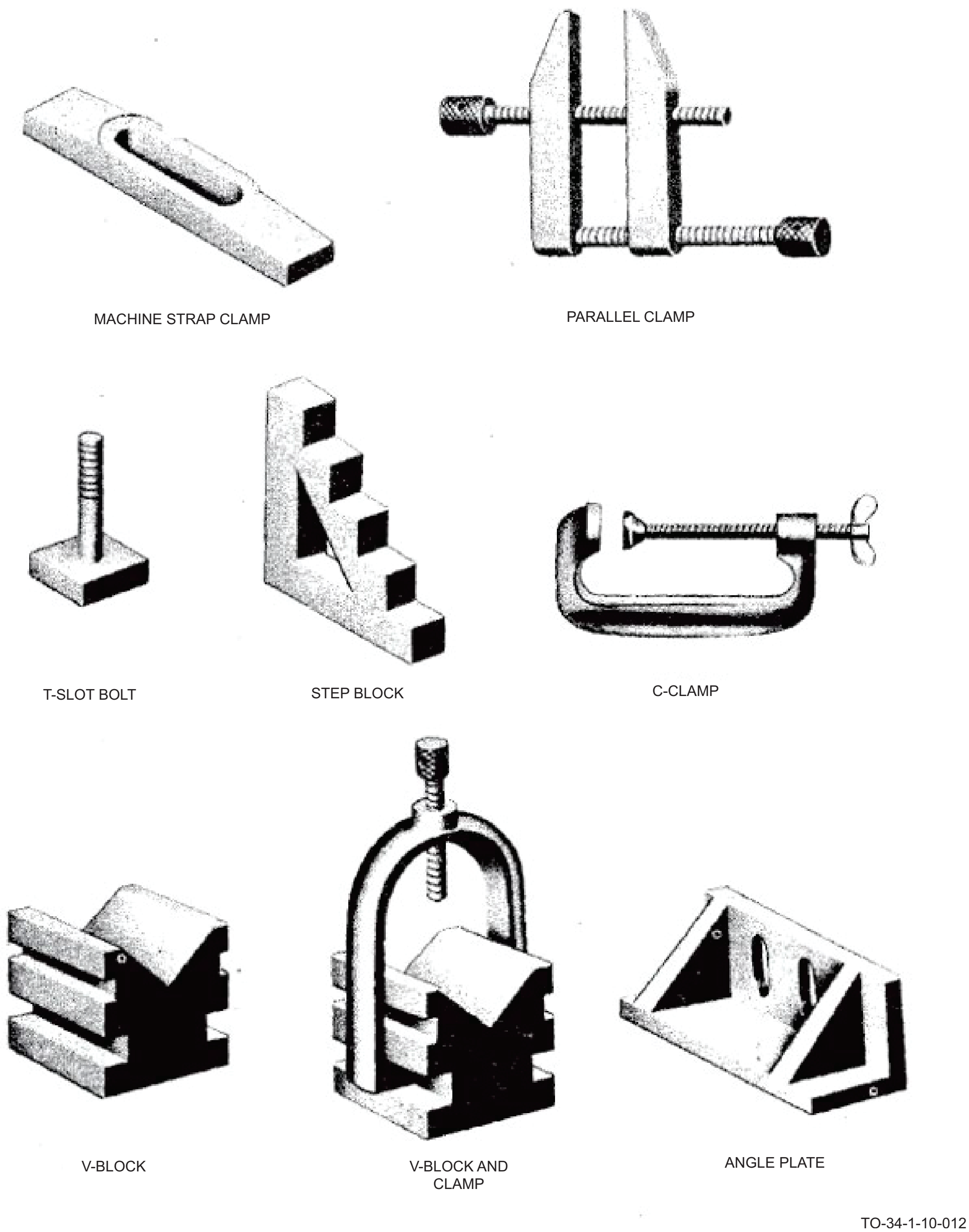


Figure 2-12. Various Types of Work Holding Devices

2.10 LAYING OUT WORK.

Laying out work for drilling operations involves identifying the centers of the holes to be drilled on the workpiece. For approximate layouts where high accuracy is not required, a simple method using a chalk pencil and steel ruler may suffice. However, for more precise jobs, it is essential to first apply layout dye to the workpiece to ensure accurate markings.

a. To mark a hole's center, scribe intersecting lines (Figure 2-13, A) and use a prick punch and hammer punch their intersection. Verify alignment with a magnifying glass if needed. Use dividers to scribe a circle (Figure 2-13, B) representing the hole's radius and create witness marks. For post-drilling checks, scribe a second, larger circle (Figure 2-13, C) for easier verification.

b. Enlarge the prick punch mark with a center punch **after** scribing to avoid divider inaccuracy. This provides a precise starting point for the drill, ensuring smooth chisel edge entry and effective cutting lip engagement.

c. For multiple-hole layouts without a jig, mark each location with intersecting lines and a circle, as in Figure 2-13. Verify hole positions relative to each other and the workpiece. For precise centering, scribe a smaller circle within the first, using the center mark as a guide. This aids in maintaining central drilling before full diameter is reached.

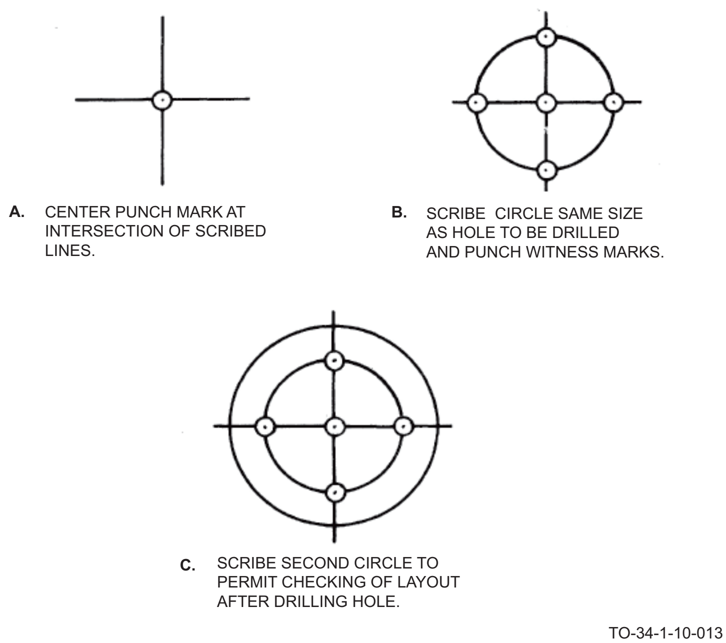


Figure 2-13. Laying Out for Accurate Drilling

NOTE

When circles become more or less obscure, re-scribing should be done from a prick punch mark rather than from a center-punch mark.

2.11 MOUNTING WORKPIECES.

Before operating a drilling machine, it is essential to ensure the workpiece is securely held in place on the table. To achieve accurate hole location, the workpiece must be firmly fastened to the table. This is typically accomplished using work holding devices (Figure 2-12), which are designed to securely position and hold the workpiece in place.

2.11.1 Securing the Workpiece.

2.11.1.1 Sensitive Drilling Machines (No Built-In Clamping). Securely clamp the workpiece in a vise (Figure 2-11) and center it under the drill. Vise weight is usually sufficient for stability.

2.11.1.2 Drilling Machines with Slotted Tables. Bolt or clamp the workpiece and holding device directly to the table for maximum security.

NOTE

Regardless of machine type, always ensure workpieces are securely held, either clamped to the table or positioned against a stop.

2.11.2 Accurate Workpiece Mounting.

a. Position clamps parallel to the table with bolts as close to the workpiece as possible. (See Figure 2-14.)

b. Place parallels near the drilling area to minimize workpiece springing and drill binding.

c. Use appropriately sized T-slot bolts, clamps, and V-blocks for a secure setup.

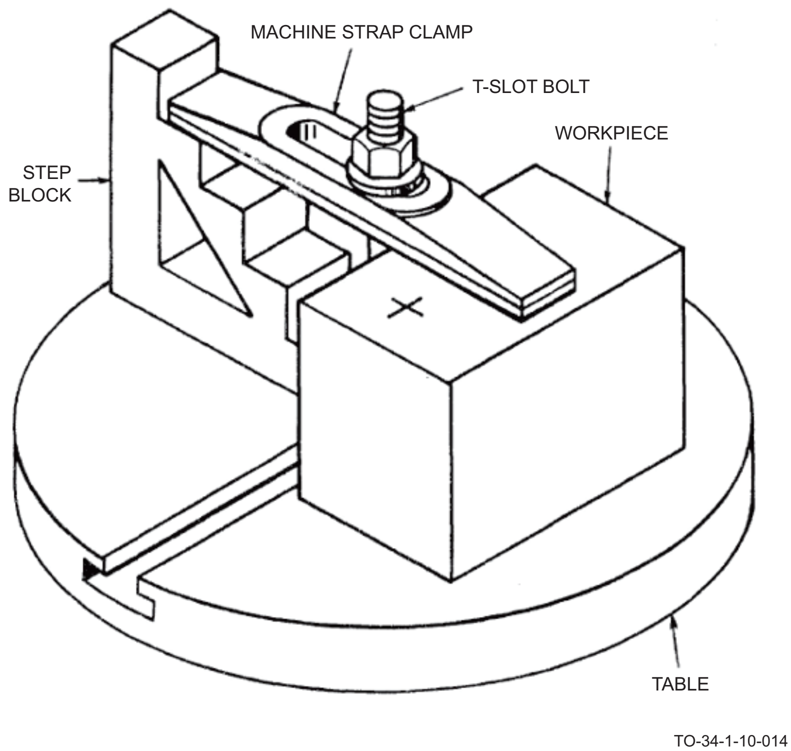


Figure 2-14. Mounting Work Using Step Block, Machine Strap Clamp, and T-Slot Bolt

2.12 GENERAL DRILLING OPERATION.

2.12.1 Selecting Drill Speed. Accurate drill speeds are essential for efficient drilling. Peripheral speed refers to the cutting speed at the drill's circumference. Understanding peripheral speed, measured in Surface Feet Per Minute (SFPM) allows you to compare cutting speeds across different machining operations.

Calculating Revolutions Per Minute (RPM):

To determine the required RPM for drilling, use this formula:

$$RPM = \frac{SFPM}{\pi D}$$

WHERE RPM = DRILL SPEED (REVOLUTIONS PER MINUTE);
SFPM = PERIPHERAL SPEED (IN. PER MINUTE);
π = 3.1416;
D = DIAMETER OF DRILL (IN.)
FOR EXAMPLE, IF A 1/16 INCH (0.0625 IN.) DRILL IS TO CUT 300 FT (3,600 IN.)
PER MINUTE TO DRILL ALUMINUM, THE DRILL SPEED IS CALCULATED AS FOLLOWS:

$$RPM = \frac{SFPM}{\pi D} = \frac{3600}{3.1416 \times 0.0625} = 18{,}336 \text{ RPM}$$

2.12.1.1 Table 2-4 provides a comprehensive guide to selecting the optimal spindle speed for drilling operations, catering to the diverse range of speeds at which drilling machines can operate. This table enables users to determine the suitable spindle speed (in RPM) for carbon steel and high-speed twist drills of various sizes, based on the specific material being drilled.

2.13 SELECTING DRILL FEED.

Drill feed rate refers to the distance the drill travels into the workpiece with each revolution, typically measured in inches per revolution. The best feed rate depends on both the drill size and the material being drilled. Generally, larger drills can handle higher feed rates, and these rates are consistent for both HSS and carbon steel drills. For smaller drills, those less than 1/4 inch, hand feeding is the recommended practice. When starting a hole, use a light feed to avoid drill damage. If you’re working with a power-fed drilling machine, start the drill manually to maintain control as excessive feed rates can cause the cutting tool to break The following feed rates are recommended:

- Drills under 1/8 inch: 0.001-0.002 inches per revolution
- Drills 1/8 to 1/4 inch: 0.003 inches per revolution
- Drills 1/4 to 1/2 inch: 0.005 inches per revolution
- Drills 1/2 to 1 inch: 0.010 inches per revolution
- Drills over 1 inch: 0.015 inches per revolution

For hard materials, use a lower feed rate than these guidelines, while softer materials like cast iron, brass, and aluminum can tolerate heavier feed rates. Refer to Table 2-4 for additional information and specific recommendations.

2.14 INSTALLING AND REMOVING DRILL.

Prior to installing a drill bit, ensure the spindle socket, drill shank, and tang are clean, smooth, and free of any burrs or nicks. A secure fit is critical, especially for large drills that require significant power. Any dirt or imperfections can prevent proper engagement, putting all the stress on the tang, which can lead to twisting or breakage. This inspection is equally crucial when using drill sockets, sleeves, and chucks.

Table 2-4. Rotational Speed and Feed for High Speed Steel [1] Twist Drills

Drill Diameter (inches)	Material and Cutting Speed Feet Per Minute (FPM)									Feed per revolution (inches)
	Aluminum	Brass	Cast Iron	Mild Steel 0.2 -0.3 carbon	Steel 0.4 -0.5 carbon	Tool Steel	3.5 Nickel Steel	Stainless Steel/ Monel	Malleable Iron	
	300	200	100	110	80	60	65	50	85	
	RPM									
1/16	18,336	12,224	6,112	6,724	4,883	3,668	3,976	3,056	5,192	0.0015
1/8	9,168	6,112	3,056	3,362	2,444	1,834	1,988	1,528	2,596	0.02-0.003

Table 2-4. Rotational Speed and Feed for High Speed Steel [1]Twist Drills - Continued

Drill Diameter (inches)	Material and Cutting Speed Feet Per Minute (FPM)									Feed per revolution (inches)
	Aluminum	Brass	Cast Iron	Mild Steel 0.2 -0.3 carbon	Steel 0.4 -0.5 carbon	Tool Steel	3.5 Nickel Steel	Stainless Steel/ Monel	Malleable Iron	
	300	200	100	110	80	60	65	50	85	
	RPM									
3/16	6,108	4,072	2,036	2,242	1,630	1,222	1,324	1,018	1,734	0.004
1/4	4,584	3,056	1,528	1,681	1,222	917	994	764	1,298	0.005
5/16	3,666	2,444	1,222	1,344	978	733	794	611	1,039	0.005
3/8	3,054	2,036	1,018	1,121	815	611	662	509	867	0.006
7/16	2,622	1,748	874	921	699	524	568	437	742	0.007
1/2	2,292	1,528	764	840	611	459	497	382	649	0.008
9/16	2,037	1,358	679	747	543	407	441	340	577	0.008
5/8	1,836	1,224	612	673	489	367	398	306	520	0.009
11/16	1,665	1,110	555	611	444	333	360	273	472	0.009
3/4	1,524	1,016	508	559	408	306	330	254	433	0.010
13/16	1,422	948	474	521	379	285	308	237	403	0.010
7/8	1,314	876	438	482	349	262	285	219	371	0.011
15/16	1,221	814	407	448	326	244	265	204	346	0.012
1	1,146	764	382	420	306	229	258	191	325	0.013
1-1/16	1,077	718	359	395	287	215	233	180	305	0.013
1-1/8	1,020	680	340	374	272	204	221	170	288	0.014
1-3/16	966	644	322	354	258	193	209	161	274	0.014
1 1/4	918	612	306	337	245	183	199	153	260	0.015
1-5/16	873	582	291	320	233	175	189	146	248	0.015
1-3/8	834	556	278	306	222	167	180	139	236	0.015
1-7/16	795	530	265	292	212	159	172	133	225	0.015
1-1/2	762	508	254	279	204	153	165	127	216	0.015

[1] Rotational speed values for carbon steel twist drills are 40 to 50 percent higher with approximately the same feed.

2.14.1 Drill Insertion and Seating.

a. Align the drill shank's tang with the spindle's keyway (Figure 2-15) and insert it. This secure connection ensures smooth rotation.

b. Seating the drill: Gently tap the drill end with a rawhide mallet **OR** lower the drill onto a wooden block using the feed handle to drive it snug.

c. For chucks, install them using the same method and then secure the drill.

2.14.2 Drill Removal.

a. Use a drill drift, inserting the rounded edge into the spindle slot. (See Figure 2-15.)

b. Tap it to release the drill.

2.15 STARTING HOLES.

Before drilling, adjust the spindle height to position the drill close to the workpiece. If using a power feed drilling machine, set the proper speed for the operation as described in Paragraph 2.12.

2.15.1 Aligning the Drill. Lower the drill to the mounted workpiece and align the drill point with the center punch mark by placing it in the center punch impression. Start the machine and feed the drill slowly by hand until the impression is slightly larger than the center punch mark or about 1/2 to 2/3 the diameter of the drill. Then, lift the drill to check if the impression is concentric with the scribed circle or circles.

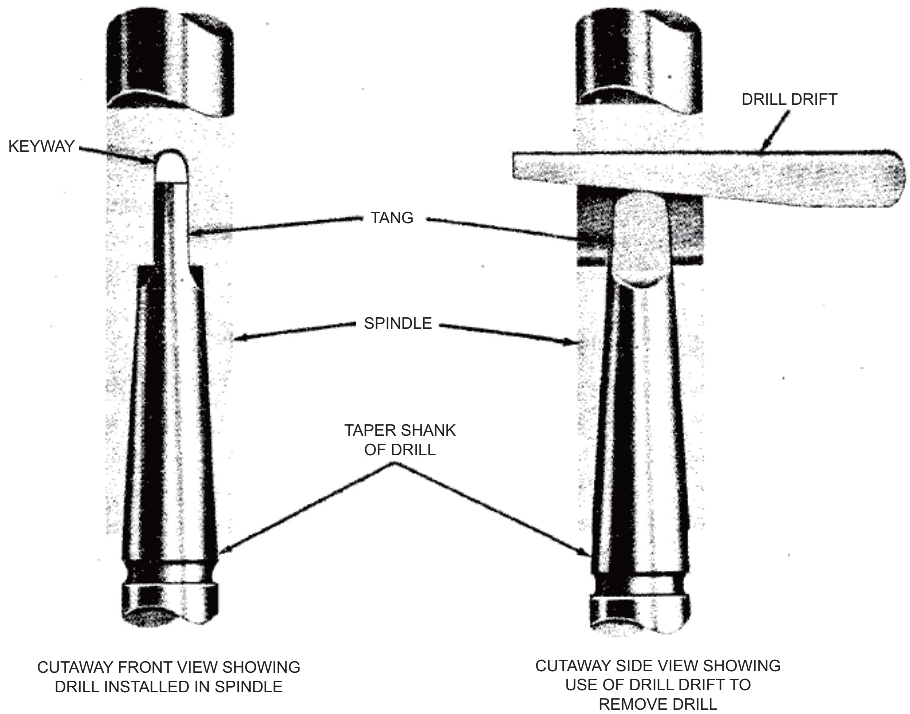


Figure 2-15. Installing and Removing Drill

2.15.2 Correcting Drill Alignment. If the drill is not centered, use a chisel to make one or more nicks or grooves on the side of the hole, toward the direction you want the drill to move. (See Figure 2-16.) The nicks will guide the drill to the correct position, as it follows the path of least resistance. Avoid moving the drill or workpiece to recenter the hole, as this can cause the drill to spring off center. Instead, make the necessary adjustments using the chisel marks.

2.15.3 Verifying Drill Alignment. After making the chisel mark, hand feed the drill back into the hole and recheck its alignment. This must be done before the drill point has enlarged the hole to full diameter, to avoid marring the workpiece surface. If the drill is properly centered, the hole will obliterate the smaller scribed circle, leaving half of each prick punch mark visible. (See Figure 2-16.)

2.15.4 Using Power Feed. Once the drill is properly centered, you can accelerate the power feed to complete the drilling operation. Remember to follow the proper procedures for power feed drilling, as described in Paragraph 2.13.

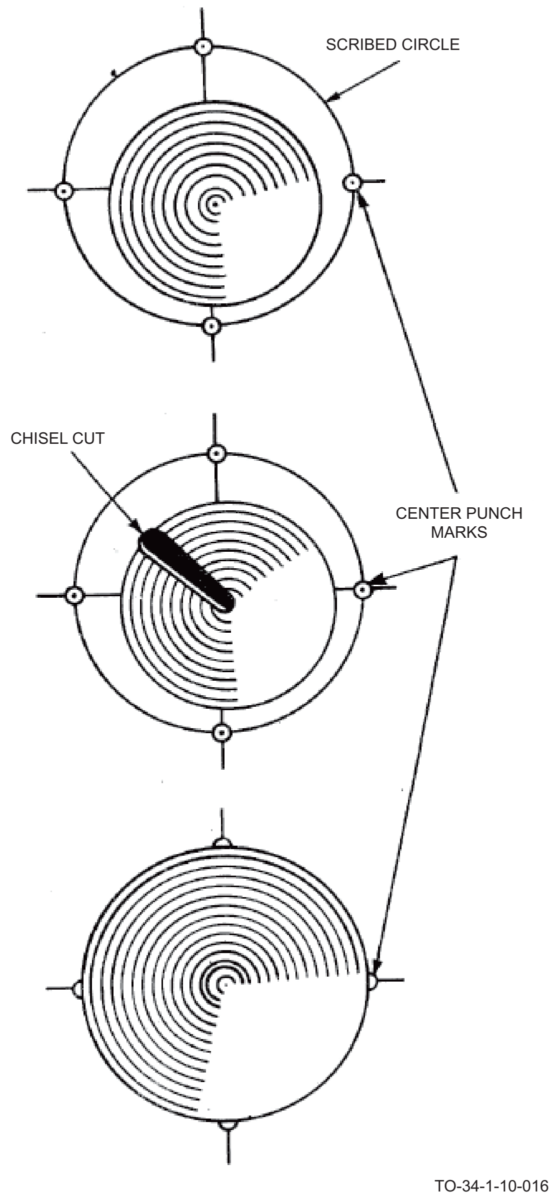


Figure 2-16. Drawing Drill Back to Correct Center

2.16 DRILLING.

After starting the hole, lubricate the drill and workpiece, and continue drilling at the optimal speed and feed rates determined in Paragraph 2.12 and Paragraph 2.13. Lubrication is crucial to maintain efficient cutting, prevent overheating, and achieve maximum cutting efficiency. The type of cutting oil or lubricant to use depends on the material being machined, as outlined in Table 2-5, which applies to both carbon steel and high-speed steel drills. Apply cutting oils generously to ensure smooth drilling operations.

Table 2-5. Cutting Oils for Drilling

Material to be Drilled	Cutting Oil
Aluminum	Soluble cutting oil; mineral-fatty blend cutting oil
Brass	Dry; soluble cutting oil; sulfurized-fatty mineral cutting oil-fatty cutting oil

Table 2-5. Cutting Oils for Drilling - Continued

Material to be Drilled	Cutting Oil
Bronze	Dry; soluble cutting oil; sulfurized-fatty mineral cutting oil-fatty cutting oil
Cast Iron	Dry
Copper	Dry; mineral-fatty blend cutting oil; soluble cutting oil
Glass	Dry
Malleable Iron	Dry; soda water, soluble cutting oil
Monel	Pure lard cutting oil; sulfurized-fatty mineral cutting oil
Slate	Dry
Alloy Steel	Mineral-fatty blend cutting oil; sulfurized-fatty mineral cutting oil
Carbon Steel	Soluble cutting oil; mineral-fatty blend cutting oil
Tool Steel	Pure lard cutting oil; sulfurized-fatty mineral cutting oil
Stainless Steel	Pure lard cutting oil; sulfurized-fatty mineral cutting oil
Wood	Dry

2.16.1 Deep Hole Drilling Precautions. When drilling holes with a depth greater than four times the drill diameter, regular chip removal is crucial to prevent accumulation and ensure efficient drilling. Remove the drill from the workpiece at regular intervals to clean out chips and debris. A slight increase in speed and decrease in feed rate can also help, allowing chips to escape more freely from the drill flutes and hole.

2.16.2 Drilling Steel. When drilling steel, check to see that no chip is lodged between the point of the drill and the bottom of the hole before restarting the machine. If a chip is present, the drill will ride on it, causing a tremendous strain on the machine and potentially leading to damage. Always start the feed by hand to ensure the drill is cutting properly before engaging the power feed.

2.16.3 Drilling Large Holes. As drill sizes increase, the size of the web and chisel edge also increase, creating a considerable strain on the machine. To eliminate this strain, drill a pilot hole first using a drill whose diameter is slightly larger than the web thickness of the large drill. This pilot hole should be drilled accurately, as the larger drill will follow the small hole. When drilling average-size holes on small drilling machines, the same procedure can be used to avoid overloading the machine.

NOTE

When drilling, choosing the correct pilot hole size and using a stepping-up method for larger holes are crucial for optimal results. A good rule of thumb for pilot holes is to select a size slightly larger than the drill bit's web, which is the narrowest central section. This prevents web rubbing and overheating while guiding the bit for a cleaner, straighter hole. For holes larger than 1/2 inch, stepping up in stages is highly recommended. This practice reduces stress on the drill bit, eases the drilling process, and improves accuracy.

2.16.4 Breaking Through. When drilling a hole entirely through a workpiece, ease up on the drill as it breaks through the bottom. The drill will have a tendency to **dig in** when breaking through, especially when drilling thin pieces. This can cause damage to the workpiece or the drill.

2.16.5 Drilling Small or Thin Workpieces. For drilling small or thin workpieces, it is sometimes convenient to set them on a block or piece of wood. The depth gage should be set to permit drilling through the workpiece but not through the wood. This helps prevent damage to the workpiece or the drill.

2.17 COUNTERSINKING.

Countersinking is the process of tapering or beveling the end of a hole using a conical cutter called a machine countersink. This operation is often used to guide and prevent burring of pins, as well as to form recesses for flat head and oval head screws. Countersinking is similar to counterboring, as described in Paragraph 2.20.

2.17.1 Machine Countersinks. Machine countersinks (Figure 2-17) used for machining recesses for screw heads typically have an included angle of 82 degrees. Some countersinks feature a pilot on the tip to guide the tool into the hole, but most common countersinks do not have this feature. Pilot-equipped countersinks are limited to single-size holes and are not practical for most small machine shop operations.

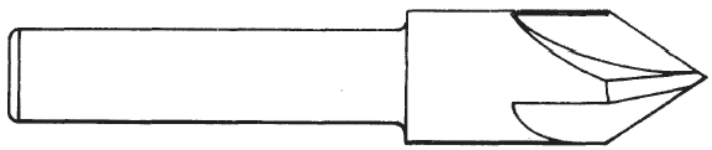
2.17.2 Countersinking Best Practices. Proper alignment of the machine countersink and the hole to be recessed is crucial. Failure to align the tool and spindle with the axis of the hole or center the tool over the hole can result in an eccentric or out-of-round recess. To ensure a smooth and accurate countersinking operation, follow these guidelines:

a. Run the machine countersink slowly and feed lightly.

b. Use a proper cutting oil to produce a smooth finish.

c. Clamp the workpiece firmly and align it properly to prevent vibration and ensure accuracy.

d. Avoid using dull tools, excessive speed, or inadequate clamping, as these can cause rough countersinking.



TO-34-1-10-017

Figure 2-17. Straight Shank Machine Countersink

2.18 TAPPING.

Drilling machines can be used for various special operations, including tapping, which involves cutting a thread in a drilled hole. There are two methods of tapping that can be performed with a drilling machine: hand tapping using the machine as a guide, and power tapping using a special tapping attachment.

2.18.1 Hand Tapping with a Drilling Machine. To perform hand tapping using a drilling machine, mount the job on the table or in a vise, and attach an adjustable tap and reamer wrench to the square shank of the tap. Install a lathe center in the drilling machine spindle and place the tap in the workpiece hole. Use the handfeed lever to apply gentle pressure to the tap, keeping it steady without forcing it. This method allows for precise control over the tapping process.

2.18.2 Selecting the Proper Twist Drill for Tapping. To ensure successful tapping, it is essential to use the correct twist drill size. Table 2-6 and Table 2-7 provide guidelines for selecting the proper twist drill size for American national screw thread pitches and national taper pipe thread pitches, respectively. Using a larger twist drill than recommended can prevent the tap from cutting threads in the hole.

2.18.3 Using a Tapping Attachment. A tapping attachment can be used to simplify the hand tapping process. (See Figure 2-18.) These attachments have a taper shank that fits into the drilling machine spindle and are equipped with a friction clutch that slips when the tap jams or encounters excessive pressure. The attachment also features a spindle reversing mechanism that operates when the spindle is raised, allowing the machine to back the tap out of the hole. To use a tapping attachment, follow these steps:

a. Mount the attachment to the drilling machine spindle.

b. Set the spindle stop to control the depth of the tap.

c. Apply a slight pressure to start the tap.

d. Allow the tap to advance and thread into the hole until the driving pins disengage.

e. Reverse the spindle to draw the tap out of the hole.

2.18.4 Best Practices for Tapping. To ensure successful tapping, follow these best practices:

a. Use plenty of cutting oil to reduce friction and prevent tap breakage.

b. Set the workpiece against one or more stops to center it and prevent movement.

c. Start slowly to ensure proper thread engagement.

d. Use a tapping attachment with a friction clutch to prevent tap breakage.

e. Set the spindle stop to control the depth of the tap and prevent over-tapping.

2.18.5 Power Tapping. Power tapping is a more accurate and cost-effective method of tapping than hand tapping. This method produces clean and true threads and reduces the frequency of tap breakage. With power tapping, the drilling machine provides the necessary power and control to drive the tap, eliminating the need for manual effort and reducing the risk of error.

TO-34-1-10-018

Figure 2-18. Drilling Machine Tapping Attachment

Table 2-6. American National Standard Screw Thread Pitches and Tap Drill Size

Screw Thread Size and Pitch	Outside Diameter of Screw (Inch)	Tap Drill Size (Lettered, Numbered or Fractional)	Decimal Equivalent of Tap Drill Size
National Coarse Series			
1-64	0.073	53	0.0595
2-56	0.086	50	0.0700
3-48	0.099	47	0.0785
4-40	0.112	43	0.0890
5-40	0.125	38	0.1015
6-32	0.138	36	0.1065
8-32	0.164	29	0.1360
10-24	0.190	25	0.1495
12-24	0.216	16	0.1770
1/4-20	0.250	7	0.2010

Table 2-6. American National Standard Screw Thread Pitches and Tap Drill Size - Continued

Screw Thread Size and Pitch	Outside Diameter of Screw (Inch)	Tap Drill Size (Lettered, Numbered or Fractional)	Decimal Equivalent of Tap Drill Size
5/16-18	0.3125	F	0.2570
3/8-16	0.375	5/16	0.3125
7/16-14	0.4375	U	0.3680
1/2-13	0.500	27/64	0.4219
9/16-12	0.5625	31/64	0.4843
5/8-11	0.625	17/32	0.5312
3/4-10	0.750	21/32	0.6562
7/8-9	0.875	49/64	0.7656
1-8	1.000	7/8	0.8750
National Fine Series			
0-80	0.060	3/64	0.0469
1-72	0.073	53	0.0595
2-64	0.086	50	0.0700
3-56	0.099	45	0.0820
4-48	0.112	42	0.0935
5-44	0.125	37	0.1040
6-40	0.138	33	0.1130
8-36	0.164	29	0.1360
10-32	0.190	21	0.1590
12-28	0.216	14	0.1820
1/4-28	0.250	3	0.2130
5/16-24	0.3125	I	0.2720
3/8-24	0.375	Q	0.3320
7/16-20	0.4375	25/64	0.3906
1/2-20	0.500	29/64	0.4531
9/16-18	0.5625	33/64	0.5156
5/8-18	0.625	37/64	0.5781
3/4-16	0.750	11/16	0.6875
7/8-14	0.875	13/16	0.8125
1-14	1.000	15/16	0.9375

Table 2-7. National Pipe Thread (NPT) and Tap Drill Size

NPT Size	Tap Drill Size (lettered or fractional)	Decimal Equivalent of Tap Drill Size
1/16-27	C	0.242
1/8-27	Q	0.332
1/4-18	7/16	0.438
3/8-18	9/16	0.562
1/2-14	45/64	0.703
3/4-14	29/32	0.906
1-11 1/2	1-9/64	1.141
1-1/4-11-1/2	1-31/64	1.484
1-1/2-11-1/2	1-23/32	1.719
2-11-1/2	2-3/16	1.188

2.19 REAMING.

2.19.1 Drilling Operations. Drilling operations often lack precision and may not produce a smoothly finished hole. One reason for this is that it is nearly impossible to grind drills with perfectly symmetrical cutting edges, resulting in a difference in length of several thousandths of an inch. As a consequence, drills typically cut a hole that is slightly larger than their own diameter, making it challenging to achieve exact sizing.

2.19.1.1 To achieve accurate and smooth holes, a common approach is to drill them 1/32 to 1/64 inch undersize and then ream them to the desired size. Reaming can be done using a machine, such as a lathe, reamer drive, or drilling machine, or by hand. Hand reaming often provides even greater accuracy than machine reaming and is sometimes used as a finishing step after machine reaming. Reamers come in different types, including machine reamers (Figure 2-19) with straight or taper shanks for use in drilling machines, and hand reamers (Figure 2-19) with straight shanks and square ends that fit into adjustable wrenches.

2.19.1.2 To produce a good finish, the reamer must run true. When starting the hole, exercise caution to avoid chatter, which can damage the hole's mouth. If chatter occurs, stop the machine immediately and restart it at a very slow speed. To ensure smooth entry, reamers are designed with a slight taper at the end, which should be turned into the hole at least 1-½ inches to achieve the proper size and prevent damage.

2.19.1.3 When reaming by hand, a center can be installed in the drilling machine spindle to align and steady the reamer, ensuring a stable start to the job. Using an adjustable tap and reamer wrench (Figure 2-19), slowly turn the reamer into the hole, then follow up with the center to maintain alignment. Feed the center by hand, applying gentle pressure to avoid crowding and ensure a smooth, controlled process.

2.19.1.4 To ensure effective and safe reaming, maintain sharp and smooth reamer teeth, and always start the reamer on an even surface. Additionally, never attempt to turn a reamer in reverse.

2.20 COUNTERBORING AND SPOT FACING.

Counterboring is the appreciable enlarging of the end of a hole to a certain depth. The axis of the enlarged hole coincides with the axis of the original hole. Counterbored holes are used for receiving the head of fillister head machine screws, setscrews, and to provide recessed shoulder for other applications.

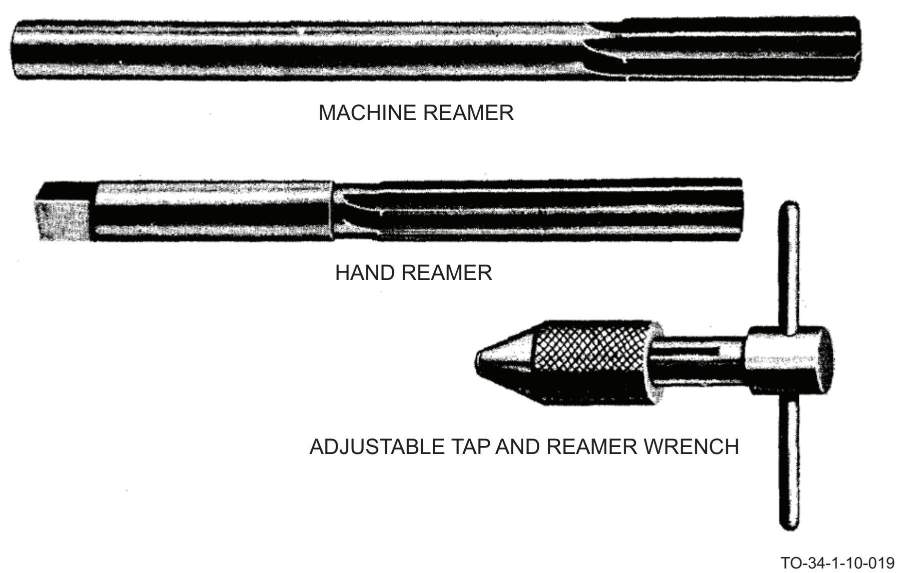


Figure 2-19. Tools Used for Reaming

2.20.1 Counterboring. Counterboring is a machining process that utilizes a counterbore tool (Figure 2-20) to enlarge existing holes to precise diameters. The counterbore features a straight shank that fits easily into a drill chuck, and a pilot at the end that guides the cutting tool into the original hole. The cutting tool itself is a flat cutting drill that matches the desired hole diameter and may have two or more cutting edges. Counterbores come in two forms: solid tools (best for high volume production) and removable blade tools (cost-effective for general-purpose and various hole size use). The latter offers a significant advantage, as a single tool with a fixed pilot size can be used with interchangeable blades to machine holes to various diameters.

TO-34-1-10-020

Figure 2-20. Taper Shank and Straight Shank Counterbores

2.20.1.1 To ensure accurate counterboring, it is crucial to align the workpiece precisely with the counterbore's pilot. Misalignment can cause the pilot to pull the tool off-center as it cuts into the workpiece. For optimal results, use a reduced feed rate (lower than that used for normal drilling operations, as described in Paragraph 2.13, and decrease the spindle speed by approximately 25 percent, as outlined in Paragraph 2.12.1. Apply adequate cutting oils to facilitate the process, except when working with cast iron, in which case cutting oils should not be used.

2.20.1.2 Spot facing is a machining operation that smooths and squares the surface around a hole, enabling a washer, nut, or bolt head to seat properly. This process is similar to counterboring and utilizes a counterbore tool. To perform spot facing, align the counterbore's pilot with the hole and use a hand-fed technique, feeding the tool only until a uniform face is achieved on the workpiece surface.

CHAPTER 3
GRINDING MACHINES

3.1 GENERAL.

Grinding is a machining process that removes material using abrasives bonded to a rotating wheel. As the abrasive particles on the wheel contact the workpiece, they function as tiny cutting tools, each removing a small chip of material. A common misconception is that grinding works through a rubbing action. It is fundamentally a cutting process, comparable to drilling, milling, and lathe turning.

3.2 PURPOSE OF THE GRINDING MACHINE.

The primary function of a grinding machine is to support and rotate the grinding wheel while often positioning and securing the workpiece in proper alignment with the wheel. This ensures precision during material removal.

3.3 APPLICATIONS OF GRINDING MACHINES.

Grinding machines are versatile tools used for both roughing and finishing operations. They are employed for machining flat, cylindrical, and conical surfaces, finishing internal bores or cylinders, forming and sharpening cutting tools, removing rough projections from castings and stampings, cleaning surfaces, and polishing or buffing components. While traditionally considered finishing tools, modern production grinding machines can perform both roughing and finishing tasks on certain types of workpieces, increasing efficiency and reducing the need for multiple machining steps.

3.4 GRINDER SAFETY.

Safety is paramount when operating grinding machines, as improper use can result in severe injuries. Operators must always adhere to safety guidelines set forth by Occupational Safety and Health Administration (OSHA) standards (29 CFR 1910.215, 29 CFR 1910.212, 29 CFR 1910.147) and applicable Air Force Instructions (AFIs) and Air Force Manuals (AFMAN) (e.g., DAFMAN 91-203). Supervisors are responsible for maintaining manufacturer manuals and integrating them into on-the-job training and job safety training outlines. In the absence of manufacturer guidance, job hazard analyses and local operating instructions must be developed. Operators must:

- Use appropriate Personal Protective Equipment (PPE), including safety glasses, face shields, hearing protection, and gloves when applicable.
- Avoid wearing loose clothing, jewelry, or gloves if specified by the manufacturer to prevent entanglement.
- Secure long hair with hair nets or caps.
- Operate machinery only if properly trained and authorized.
- Ensure guards are securely in place and operational before starting the machine.
- Follow Lockout/Tagout procedures when guards are removed for maintenance.
- Allow all moving parts to stop completely before performing maintenance or clearing debris.

3.4.1 Grinding Machines. Grinding machines must be equipped with properly adjusted wheel guards to prevent fragments from causing injury in the event of wheel breakage. Tool rests shall be adjusted to within 1/8 inch of the grinding wheel to prevent workpieces from being caught. Regular inspection and maintenance of grinding wheels, guards, and machine components are essential for safe operation. Operators must also ensure that wheels are correctly mounted and free from cracks or damage before use. Proper training and certification are mandatory for personnel operating grinding equipment.

3.5 TYPES OF GRINDING MACHINES.

Grinding machines come in a wide variety of designs, each tailored to specific applications and types of materials to be ground. The design and function of these machines can range from simple to highly complex, depending on the precision, automation, and scope of the task at hand. Understanding the types of grinding machines is essential for selecting the appropriate machine for a given operation. From the simplest to the most complex designs, grinding machines can be classified into utility grinding machines, disk and belt grinding machines, cylindrical grinding machines, and surface grinding machines. The average machinist will primarily work with floor-mounted and bench-mounted utility grinding machines, bench-type utility grinding and buffing machines, and reciprocating surface grinding machines.

3.5.1 Key Categories of Grinding Machines.

3.5.1.1 Utility Grinding Machines. These are the most commonly used grinding machines and are often found in machine and welding shops. They are used for general-purpose grinding tasks, including sharpening tools, deburring, and polishing. Utility grinding machines come in both floor-mounted (Figure 3-1) and bench-mounted versions (Figure 3-2), providing flexibility depending on the size and complexity of the work. These machines are characterized by their simple design, ease of use, and versatility for handling a variety of materials and grinding tasks. Bench-mounted models are typically used for smaller, lighter tasks, while floor-mounted machines can handle larger, more robust components. See Figure 3-3 for an illustration of the components and safety feature of a utility grinder.

TO-34-1-10-021

Figure 3-1. Floor Mounted Grinding Machine

Figure 3-2. Bench Mounted Grinding Machine

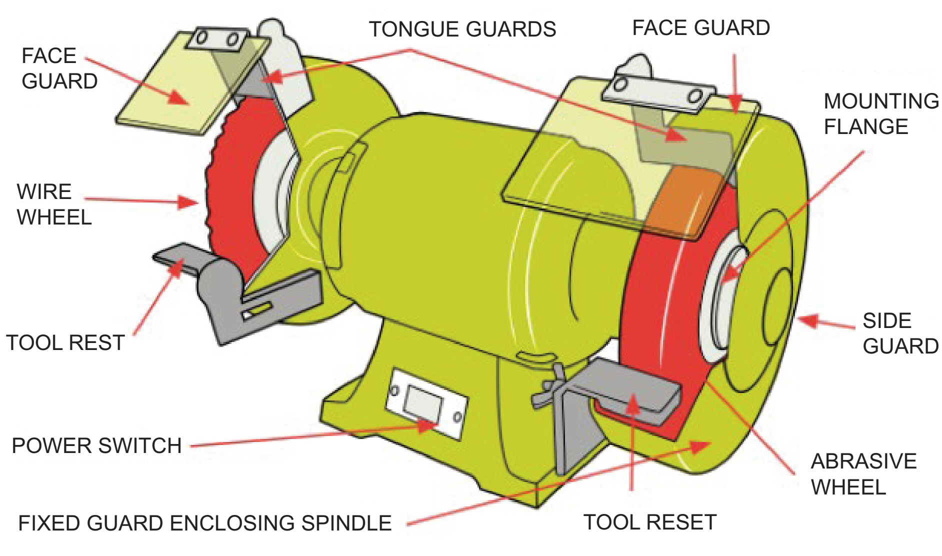


Figure 3-3. Utility Grinder Components and Safety Features

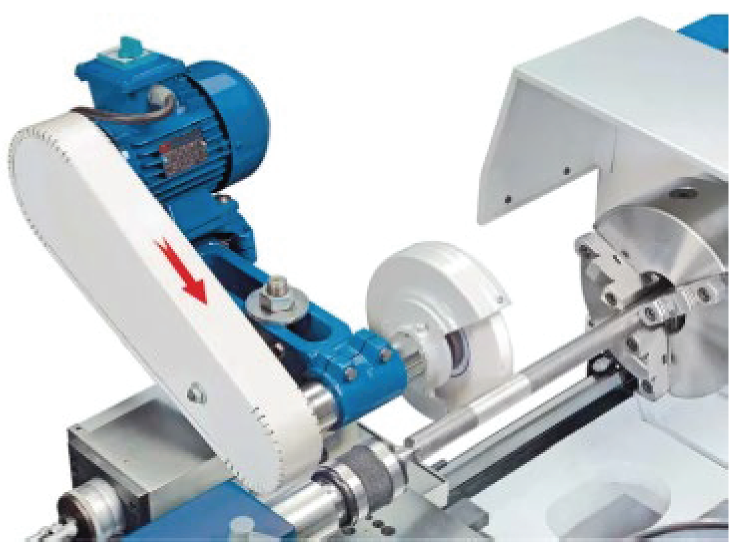
3.5.1.2 Disk and Belt Grinding Machines. These machines utilize a rotating disk or belt (Figure 3-4) as the grinding medium. They are especially effective for large-scale surface grinding, where the workpiece must be polished or finished with a high degree of smoothness. Disk grinding machines are commonly used for flat surfaces, while belt grinders can achieve similar results but are more flexible, allowing for curved or contoured surfaces. Both types are often used for sanding, smoothing, and finishing large metal sheets or other materials, and they provide excellent surface finish and material removal rates.

TO-34-1-10-024

Figure 3-4. Disk and Belt Grinding Machine

3.5.1.3 Cylindrical Grinding Machines. Cylindrical grinding machines are used to grind the outer surfaces of cylindrical workpieces. This type of machine is commonly used in the manufacturing of precision parts, such as shafts, rods, and bearings. These machines typically feature a rotating workpiece and a stationary grinding wheel that grinds the workpiece as it rotates. Cylindrical grinders can be further classified into two types: external and internal.

3.5.1.3.1 External and Internal Cylindrical Grinders. Used for grinding the outer diameter of a workpiece, while internal cylindrical grinders focus on the internal diameter. This type of machine is designed for high-precision tasks and is critical for ensuring the dimensional accuracy of cylindrical components. The lathe tool post grinding machine (Figure 3-5) is a machine tool attachment designed to mount to the tool post of engine lathes. It is used for internal and external grinding of cylindrical workpieces.



TO-34-1-10-025

Figure 3-5. Lathe Tool Post Grinder

3.5.1.4 Surface Grinding Machines. Surface grinders are designed to produce a smooth, flat surface on a workpiece. They operate by using a rotating grinding wheel to remove material from the surface and are often used to finish flat or contoured surfaces to exacting tolerances. Surface grinding machines can be further classified into horizontal (Figure 3-6) and vertical types (Figure 3-7) based on the orientation of the grinding wheel. Horizontal surface grinders are typically used for larger, flat workpieces, while vertical grinders are better suited for smaller, intricate parts. The versatility of surface grinding machines makes them suitable for a wide range of applications, including tool sharpening, mold and die production, and finishing automotive components.

TO-34-1-10-026

Figure 3-6. Horizontal Surface Grinding Machine

TO-34-1-10-027

Figure 3-7. Vertical Surface Grinding Machine

3.5.1.5 Grinding Machine Types. Each of these grinding machine types offers distinct advantages depending on the specific needs of the workpiece and the level of precision required. The average machinist will likely work with utility grinding machines for general tasks, but understanding the different types of machines and their capabilities is crucial for selecting the right tool for specialized grinding operations. Advanced grinding techniques, such as cylindrical and surface grinding, demand a higher level of skill and more precise equipment, often requiring specialized training and experience.

3.6 TOOLS AND EQUIPMENT.

3.6.1 Grinding Abrasive Wheels. The grinding abrasive wheels discussed in this section may not necessarily be compatible with all the grinding machines described in this chapter. Various wheel types and shapes can be used with tool post grinding machines, milling and grinding lathe attachments, bench-type tool and cutter grinders, and portable electric or pneumatic grinders. However, the principles behind these wheels are consistent across all types.

3.6.1.1 Grinding abrasive wheels consist of abrasive grains and a bonding substance that holds the grains in place, releasing them as they wear down. Between the grains and the bond, air spaces, or voids, allow clearance for chips removed from the workpiece by each abrasive grain. Wheels are classified based on size, shape, abrasive material, hardness grade, grain size, bond type, and wheel structure or density.

3.6.1.2 Standard types of grinding abrasive wheels come in various shapes and sizes, each designed for specific operations. Figure 3-8 illustrates common types. Other nonstandard shapes are also available, often identified by number rather than name.

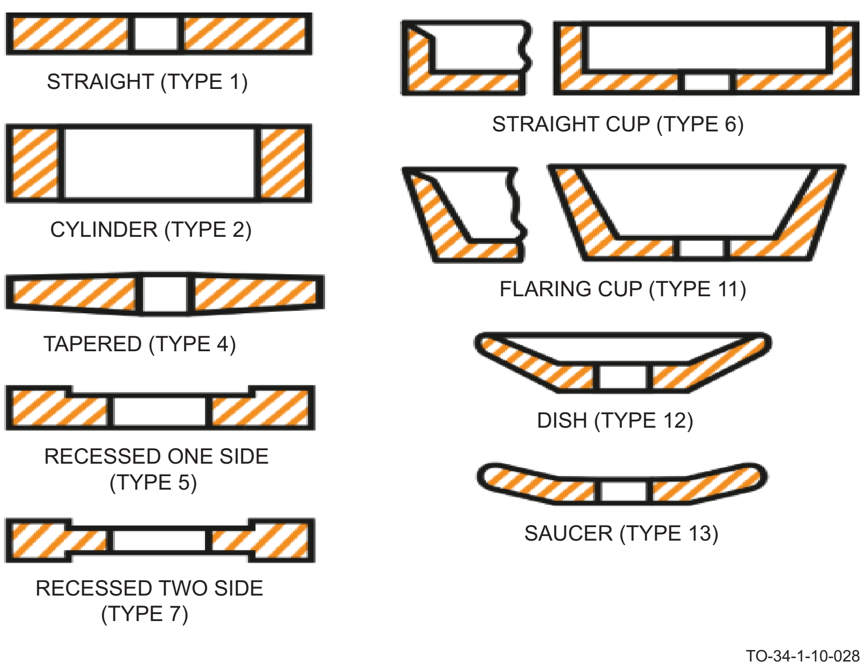


Figure 3-8. Common Grinding Wheel Shapes

3.6.1.3 Standard shapes of grinding wheel faces can have different face shapes to suit specific applications. The standard face shapes are shown in Figure 3-9, with each shape identified by a corresponding letter designation.

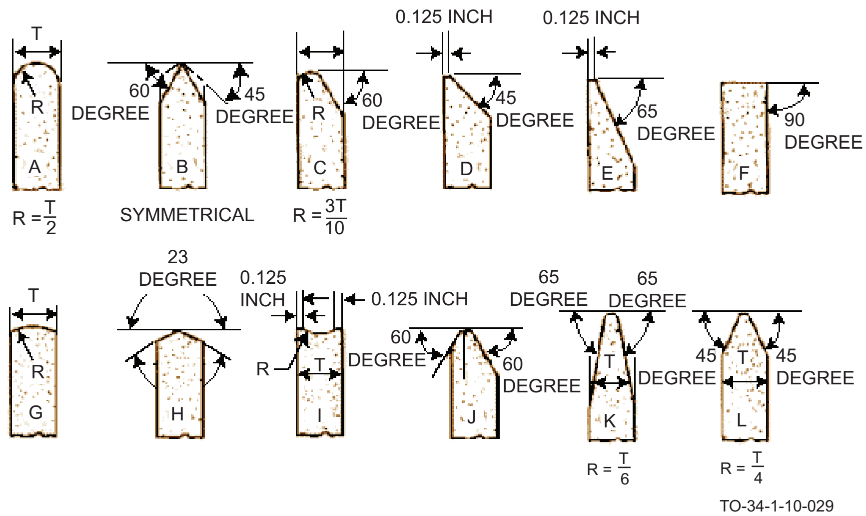


Figure 3-9. Standard Shapes of Grinding Wheel Faces

3.7 GRINDING WHEEL STORAGE.

Proper storage of abrasive grinding wheels is essential to maintain their integrity and performance over time. Grinding wheels should be stored in a clean, dry environment, away from excessive heat, moisture, or direct sunlight, as these factors can degrade the wheel's bond and abrasive materials. (See Figure 3-10.) Abrasive wheels should be stored in their original packaging or on flat surfaces to prevent distortion or damage. It is also important to avoid storing wheels in areas with high humidity, as moisture can cause rusting on metal-bonded wheels or affect the wheel's bond integrity. Wheels should be placed in a manner that avoids unnecessary stacking or pressure, as this can lead to cracks or other forms of damage that may affect their performance. To further ensure the safety and longevity of grinding wheels, they should be inspected for any visible signs of damage before use, even if they have been stored properly. Handling should be done with care, avoiding dropping or bumping the wheels, which could cause cracks or fractures. Wheels with visible cracks or defects should be discarded and not used in grinding operations. By following proper storage guidelines, abrasive wheels can retain their effectiveness and safety, leading to more consistent grinding results and reduced risk of accidents during operation.

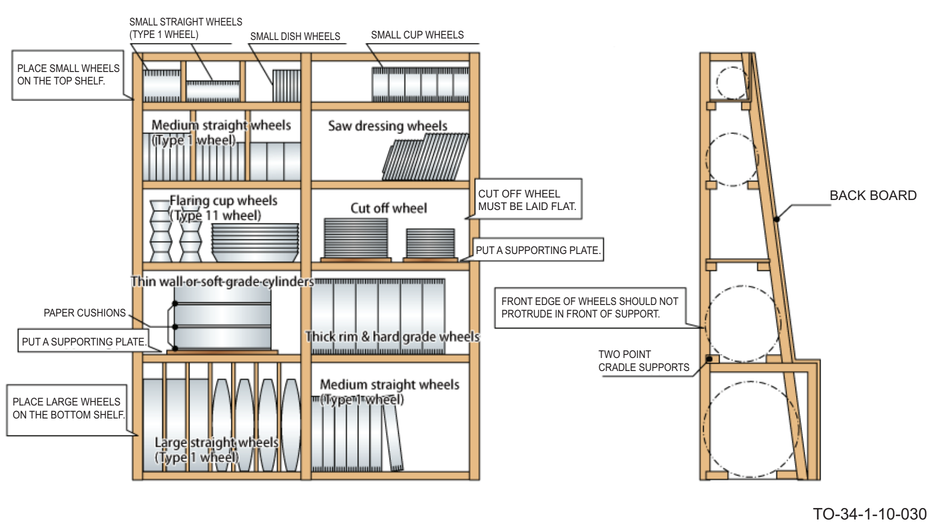


Figure 3-10. Grinding Wheel Storage

3.8 ABRASIVE MATERIALS.

3.8.1 Abrasive Grains or Materials. Abrasive grains or materials for grinding wheels use natural or manufactured abrasives. Emery and corundum are natural aluminum oxide abrasives, while manufactured abrasives can be aluminum oxide or silicon carbide, each sold under various trade names. Manufactured abrasives are more commonly used because their production process yields uniform material.

3.8.1.1 Silicon Carbide. A harder and more brittle material, suitable for grinding tough materials like cast iron, brass, stone, and rubber.

3.8.1.2 Aluminum Oxide. Softer than silicon carbide, but tougher and less prone to fracture. It is ideal for grinding high tensile strength materials such as hardened steels.

3.8.1.3 Aluminum Zirconium. A very fine, dense grain abrasive, and is the toughest abrasive derived from aluminum. These wheels are used for very rapid, aggressive stock removal and are very durable.

3.8.1.4 Seeded Gel. High performance wheels are made by a carefully controlled manufacturing process using aluminum oxide grains. The grains are powdered very small and slowly fused with a bonding agent. Seeded gel abrasive wheels last the longest of those described here, they stay sharp the longest, and they require the least maintenance.

3.8.2 Abrasive Grain Sizes. Abrasive grains are classified by the mesh size of the sieve through which they pass. For example, a grain labeled Number (No.) 40 has passed through a sieve with approximately 40 meshes per linear inch. Grinding wheels are designated as coarse, medium, or fine based on the size of the abrasive grains. See Table 3-1 for abrasive grain sizes.

Table 3-1. Abrasive Grain Sizes

Coarse	Medium	Fine	Very Fine
8	30	70	220
10	36	80	240
12	46	90	280
14	54	10	320
16	60	120	400
20	NA	150	500
24	NA	180	600

3.8.3 Bond Materials. Bond Materials hold the abrasive grains together and forms the grinding wheel. Bond types are classified into seven categories:

3.8.3.1 Vitrified Bond (V). Are made by fusing abrasive grains with a ceramic-like material (usually clay and other bonding agents). The bond is created by firing the mixture at high temperatures in a kiln. They are used in precision grinding, surface grinding, and tools like tool and cutter grinders. Their properties include:

- Strength, durability, and resistance to wear.
- Provides excellent cutting performance.
- Offers a very stable structure.
- Works well in harsh environments (e.g., with water, oils, and heat).

3.8.3.2 Silicate Bond (S). Are made using a mixture of clay and sodium silicate. This bond type is generally used for wheels that need a degree of flexibility but are not as elastic as rubber or resinoid. They are often used for grinding large parts where a more resilient bond is needed. Their properties include:

- Create thicker wheels than vitrified or resinoid bonded wheels.

- Susceptible to moisture but more adaptable in thicker forms.
- Less rigidity than vitrified bonds, but still quite durable.

3.8.3.3 Organic Bond (O). Include a wide variety of bonding agents like resins and gums, including materials like phenolic resins, rubber, and other organic substances. They are common in non-standard grinding and polishing applications where flexibility and cutting efficiency are required. Their properties include:

- Flexible and can be made thin for delicate grinding tasks.
- Ideal for high-stock removal applications due to their elastic properties.

3.8.3.4 Resinoid Bond (B). Are made using a phenolic resin that binds the abrasive grains. These wheels are generally more elastic than vitrified wheels. They are commonly used for snagging castings, cutting off operations, and cylindrical grinding. Their properties include:

- High flexibility and resistance to impact.
- Cool cutting and minimal heat generation during grinding.
- Works well for heavy stock removal.

3.8.3.5 Rubber Bond (R). Use rubber compounds to hold the abrasive grains together. They are highly flexible and resilient. They are used in fine finishing, polishing, and applications requiring a high degree of flexibility like slotting pen points or thin grinding. Their properties include:

- Provides a smooth, consistent finish.
- Works well for delicate grinding applications.
- Excellent for polishing or finishing tasks.
- High elasticity allows for thin, delicate wheels.

3.8.3.6 Shellac Bond (E). Shellac bonds are made from natural or synthetic resins, which provide a semi-porous structure. They are flexible and can hold a significant number of abrasive grains. They are used in polishing, tool sharpening, and grinding operations where cool cutting is essential. Their properties include:

- Provides a very cool cutting action.
- Typically, more porous than other bonds.
- Offers excellent wear resistance and can handle deep cuts without overheating.

3.8.3.7 Metal Bond (M). Are created by fusing abrasive grains in a metal matrix. The metal can be a variety of alloys, including copper, brass, or steel. They are used in specialized grinding like grinding hard metals, ceramics, or in the production of high-performance parts like cutting tools and saw blades. Their properties include:

- Extreme durability and can withstand high pressures and temperatures.
- Offers precision and long-lasting cutting performance.
- Used in diamond grinding wheels to help hold diamond or Cubic Boron Nitride (CBN) abrasives.

3.9 ABRASIVE GRADES OF HARDNESS.

The grade of a grinding abrasive wheel refers to the hardness of the bonding material used to hold the abrasive grains. A **soft** wheel is characterized by its ability to release the abrasive particles quickly during operation, allowing for efficient cutting. A

hard wheel is one in which the bond material resists the dislodging of the abrasive grains, providing more controlled wear and longer wheel life. The choice of wheel grade depends on the material being ground and the specific grinding application.

3.9.1 Grinding System. Grinding wheels utilize a grading system where most wheels are graded according to their hardness using a letter-based scale. Manufacturers typically utilize a letter code that ranges from **E** (very soft) to **Z** (very hard). The type of bond material used in the wheel influences its hardness range. For example:

- Vitrified and silicate bonds generally span the entire range from very soft to very hard.
- Shellac and resinoid bonds typically cover the range from very soft to hard.
- Rubber bonds are usually found in the medium to hard range.

3.9.2 Bond Type. Each bond type has distinct characteristics that influence the performance and suitability of the wheel for specific grinding applications. Table 3-2 provides the typical hardness scale for grinding wheels, using the letter grading system.

3.10 SELECTING ABRASIVE GRINDING WHEELS.

The conditions under which grinding abrasive wheels are used can vary significantly. A wheel that works effectively on one machine may be too hard or too soft for the same operation on another. When selecting grinding wheels, several key factors must be considered. (See Table 3-3.) While these guidelines are generally applicable, they should be understood as flexible, with some exceptions depending on specific circumstances.

3.10.1 Tensile Strength. The tensile strength of the material to be ground is a primary factor in selecting the appropriate abrasive. Different materials require different types of abrasives to achieve the best performance:

- Aluminum oxide abrasives are ideal for materials with high tensile strength, such as carbon steel, high-speed steel, malleable iron, and hard bronze.
- Silicon carbide abrasives are best suited for materials with lower tensile strength, including cast iron, chilled iron, brass, soft bronze, rubber, granite, and marble.

3.10.2 Selection of Grain Size. The selection of grain size for a grinding abrasive wheel depends on multiple factors that influence the material removal process and the quality of the surface finish. The key considerations include:

3.10.2.1 Material Hardness and Ductility. Softer, ductile materials require coarser grains for efficient removal. Coarse grains break away easily to prevent clogging, making them ideal for soft materials like rubber or aluminum.

3.10.2.2 Amount of Stock Removal. For large material removal, coarser grains are better suited as they reduce heat buildup and allow faster material removal. Fine grinding operations, where precision is key, require finer grains for a smoother surface.

3.10.2.3 Surface Finish Requirements. A finer grain size is needed for a fine finish. Finer grains cut more precisely, resulting in smoother surfaces, especially in precision work or polishing.

Table 3-2. Hardness Scale for Grinding Wheels

Grade	Description	Typical Bond Types
E	Very Soft	Vitrified, Silicate
F	Soft	Vitrified, Silicate
G	Medium Soft	Vitrified, Silicate, Shellac
H	Medium	Vitrified, Silicate, Shellac
I	Medium	Vitrified, Shellac
J	Medium Hard	Vitrified, Shellac, Resinoid
K	Hard	Vitrified, Resinoid
L	Hard	Vitrified, Resinoid

Table 3-2. Hardness Scale for Grinding Wheels - Continued

Grade	Description	Typical Bond Types
M	Very Hard	Vitrified, Resinoid, Rubber
N	Very Hard	Vitrified, Rubber
O	Extremely Hard	Vitrified, Rubber
P	Extremely Hard	Vitrified, Rubber
Q	Very, Very Hard	Vitrified, Rubber
R	Extremely Hard	Vitrified, Rubber
S	Extremely Hard	Vitrified, Rubber
T	Extremely Hard	Vitrified, Rubber
U	Very Hard	Vitrified
V	Extremely Hard	Vitrified
W	Very Hard	Vitrified
X	Extremely Hard	Vitrified
Y	Very, Very Hard	Vitrified
Z	Extremely Hard	Vitrified

3.10.3 Grade of Hardness. The grade of hardness in a grinding wheel determines how tightly the abrasive grains are held in place by the bond material. Several factors influence the appropriate hardness grade for different grinding applications: The harder the material, the softer the grade should be.

3.10.3.1 Material Hardness. Harder materials, such as hardened steels, require a softer wheel to allow for continuous release of sharp grains. Softer materials can be ground with harder wheels that maintain abrasive effectiveness.

3.10.3.2 Arc of Contact. A smaller arc of contact requires a harder wheel to handle the higher pressure per unit of contact, while a larger wheel can use a softer grade due to reduced pressure.

3.10.3.3 Work Speed vs. Wheel Speed. Higher work speeds require harder wheels to maintain cutting efficiency. Lower work speeds may allow for a softer wheel, as the cutting process is slower.

3.10.3.4 Machine and Bearing Conditions. Rigid, stable machines allow the use of softer wheels, while unstable machines may require harder wheels to prevent premature wear or instability.

3.10.4 Abrasive Wheel Part Number. A grinding abrasive wheel part number typically includes several key components that describe the wheel's size, material, bond type, grit, grade, and structure. For instance, a part number such as 12V9-60-J8-V can provide detailed information: the first part indicates the wheel type and shape, the second part denotes the grit size, the third specifies the wheel's hardness grade, the fourth describes the structure or density, and the last part refers to the bond type used. Understanding these components is essential for selecting the right wheel for a specific grinding application. A breakdown of a typical part number is shown in Table 3-2, which illustrates how each segment corresponds to a particular specification.

3.10.5 Structure of Grinding Wheel. Factors affecting the structure of a grinding wheel refers to the spacing between the abrasive grains. This spacing is crucial for controlling the amount of material removed, the cooling rate, and the overall grinding efficiency. Several factors influence the optimal structure for specific applications.

3.10.5.1 Material Toughness and Ductility. Softer, ductile materials require wider grain spacing to prevent clogging and heat buildup. This is important for materials like rubber and soft metals.

3.10.5.2 Surface Finish Requirements. Finer finishes require denser grain spacing for smoother surfaces. Finer grains provide better precision, especially in tool sharpening or when working with brittle materials.

3.10.5.3 Type of Grinding Operation. Surfacing operations benefit from a more open structure to allow for chip clearance, while cylindrical or tool grinding requires medium structures for a balance between material removal and precision.

3.10.5.4 Operational Load and Cooling Efficiency. Open structures improve cooling and chip evacuation in heavy grinding, while denser structures may need better cooling systems due to increased friction and heat.

3.10.6 Selection of Bond Material. The factors affecting the selection of bond material for grinding wheels are important because they directly impact the wheel's performance, durability, and suitability for specific grinding tasks. Choosing the right bond material ensures optimal results in terms of cutting efficiency, surface finish, and wheel longevity. The selection of bond material for a grinding wheel is influenced by the following factors:

3.10.6.1 Thin Cutoff Wheels and Bending Strains. Thin cutoff wheels or wheels under bending strains require flexible bonds like resinoid, shellac, or rubber to resist shocks and maintain performance.

3.10.6.2 Large-Diameter Solid Wheels. For large solid wheels, a silicate bond is ideal for strength and stability, especially in heavy grinding applications.

3.10.6.3 Speed Considerations. Vitrified bonds are best for speeds up to 6,500 Surface Feet Per Minute (SFPM), while resinoid, shellac, or rubber bonds perform better at speeds above 6,500 SFPM due to their heat resistance and flexibility.

3.10.6.4 High-Finish Applications. For high-quality finishes, resinoid, shellac, or rubber bonds are preferred as they release abrasive grains more gradually, providing smoother, finer finishes.

Table 3-3. Grinding Wheel Bond Material Selection

Category	Factors	Guidelines/Recommendations
Abrasive Material Selection	Work Material Type	Steel, Cast Iron → Aluminum Oxide Non-Ferrous Metals → Silicon Carbide Glass, Ceramics → Diamond CBN Hard Alloys → CBN, Diamond
Grain Size and Surface Finish	Application Type	Rough Grinding → Coarse (30-60) Precision Grinding → Medium (60-120) Fine Grinding → Fine (120-400)
	Desired Finish	Coarse Grains → Low Finish Medium Grains → Moderate Finish Fine Grains → High Finish
Bond Material Selection	Thin Cutoff Wheels or Bending Strains	Resinoid, Shellac, Rubber Bonds
	Large-Diameter Solid Wheels	Silicate Bonds
	Wheel Speed	Vitrified Bond → Up to 6,500 SFPM Resinoid, Shellac, Rubber → Above 6,500 SFPM
	High-Finish Requirements	Resinoid, Shellac, Rubber Bonds
Wheel Speed and Material Compatibility	Abrasive Material Type	Aluminum Oxide → Vitrified Bond (up to 6,500 SFPM), Silicon Carbide → Resin/Shellac Bond (up to 7,500 SFPM), CBN, Diamond → Resin Bond (up to 10,000 SFPM)
Grinding Wheel Specifications	Grit Size	Fine (120), Medium (60), Coarse (30)
	Grade (Hardness)	Soft (A), Medium (M), Hard (H)
	Structure (Grain Spacing)	Open (Wide), Medium, Closed (Dense)
	Bond Type	Vitrified, Resin, Rubber
Grinding Wheel Application	Surface Grinding	Vitrified or Resin Bond (Precision Finishes, Smooth Surfaces)
	Cylindrical Grinding	Resin or Vitrified Bond (External Diameter Grinding)
	Polishing	Diamond or CBN Bond (High-Precision Finishes)

3.11 MOUNTING ABRASIVE GRINDING WHEELS.

WARNING

The specified wheel size for the grinding machine must be strictly followed. The wheel's diameter and width should not exceed the machine's recommended size limits. Failure to comply, could result in injury to, or death of, personnel or long term health hazards.

Properly mounting a grinding wheel is essential for safe operation. An improperly mounted wheel can be unbalanced or insecurely fastened, potentially leading to dangerous situations, especially at high speeds. The wheel should be mounted between two flanges, which are relieved on their inner sides. This ensures the wheel is supported only at its outer edges. (See Figure 3-11.) This mounting method provides secure support with less pressure, reducing the risk of wheel breakage. For proper support, the flanges should be at least one-third, and ideally one-half, the diameter of the wheel.

3.11.1 Spindle Hole Fit. The spindle hole in the wheel should fit snugly over the spindle without being loose. A loose fit can make centering the wheel difficult. If the hole is too large:

a. Wrap paper around the spindle to achieve a better fit.

b. If the hole is significantly oversized, select a wheel with the correct size or use a suitable bushing to adapt the spindle to the wheel's hole size.

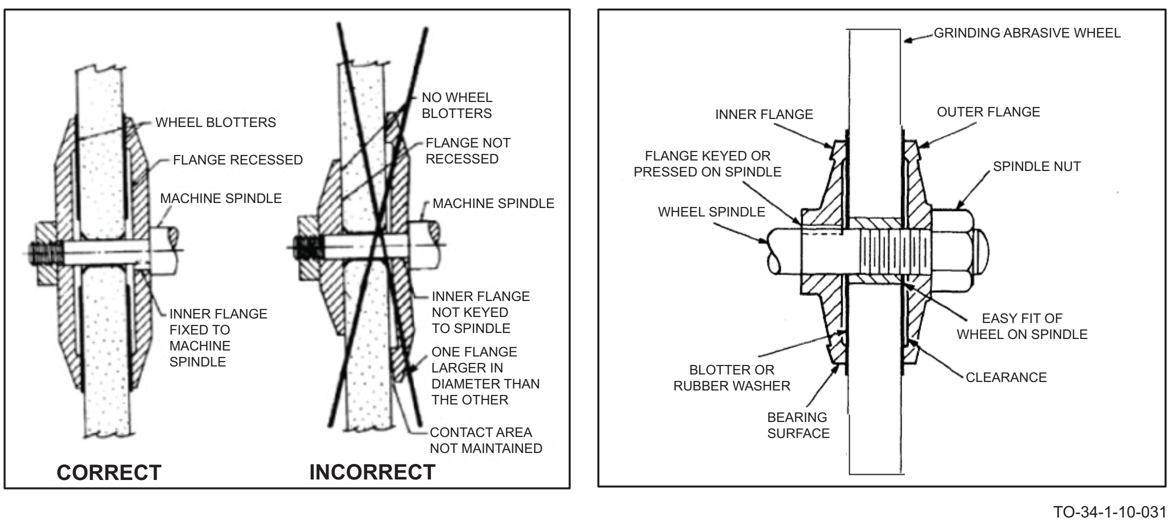


Figure 3-11. Correctly Mounted Abrasive Grinding Wheel

3.12 ABRASIVE WHEEL DRESSERS.

Grinding wheels naturally wear down during general grinding operations due to the pressure applied to the wheel's surface while cutting. Additionally, if the wrong wheel is used for a particular operation, the wheel may become clogged with metal particles, or the abrasive grains may become dull before breaking free from the bond. In either case, the wheel's performance and accuracy degrade, necessitating dressing or truing to restore its efficiency.

3.12.1 Dressing and Truing. Dressing and truing are essential maintenance procedures for grinding wheels, ensuring they continue to perform at their best. Over time, grinding wheels lose their cutting efficiency due to wear, metal buildup, or the dulling of abrasive grains. Dressing and truing restore the wheel's functionality and shape, enhancing its cutting performance and accuracy. These two processes are typically performed together but serve distinct purposes.

3.12.1.1 Dressing. This process involves cutting the surface of the grinding wheel to expose fresh abrasive grains and remove any accumulated metal particles or debris. Dressing restores the wheel's ability to cut efficiently and prevents clogging, ensuring the wheel remains effective throughout its use.

3.12.1.2 Truing. Truing restores the wheel's concentricity and corrects any irregularities in its shape. This ensures that the wheel runs smoothly and maintains the desired geometry for precise grinding operations.

3.12.2 Grinding Wheel Dressers. Grinding wheel dressers are tools used to restore the cutting surface of a grinding wheel and maintain its shape. They come in various designs, each suited to different types of grinding operations. The primary components of a grinding wheel dresser typically include an abrasive tool (usually diamond, carbide, or a metal-bonded abrasive) and a mounting mechanism to hold the tool in place while it works on the wheel. There are two main types of dressers commonly used: hand-operated dressers and powered dressers.

3.12.2.1 Hand-Operated Dressers. These are manually controlled devices that use a fixed or rotating abrasive element, such as a diamond, to dress the wheel. (See Figure 3-12.) The dresser is held against the spinning wheel, and the operator moves it across the surface to expose fresh abrasive grains. This type of dresser is commonly used for light dressing and minor repairs. Common types of hand-operated dressers include single-point diamond dressers, multi-point diamond dressers, and rotary dressers. Single-point dressers consist of a single diamond tip used to cut into the wheel's surface, while multi-point dressers feature multiple diamonds that distribute the cutting action more evenly, resulting in a more consistent finish.

3.12.2.2 Powered Dressers. These dressers are motorized and offer more precision and efficiency for heavy-duty dressing operations. Powered dressers can be equipped with rotating abrasive tools or abrasive stones that automatically dress the wheel. They are often used in high-production environments where consistent wheel performance is essential. Powered dressers allow for precise control over the dressing depth, speed, and pattern, making them ideal for truing large or complex wheels.

TO-34-1-10-032

Figure 3-12. Grinding Wheel Hand Dresser

3.12.3 Dressing the Grinding Wheel. Dressing the grinding wheel is crucial for maintaining accuracy in the production of high-quality workpieces. The purpose of dressing is to refresh the cutting surface of the wheel by exposing the sharp abrasive grains and removing excess bonding material. This ensures that the grains cut freely, without generating excessive heat during operation. While the dressing process does help to true the wheel to some extent, it is not sufficient for achieving precision grinding. For standard utility grinding machines or off-hand grinding, additional truing is generally not required.

3.12.4 Operation of a Grinding Wheel Dresser. The operation of a grinding wheel dresser involves bringing the dresser into contact with the wheel's surface while it is rotating. (See Figure 3-13.) The abrasive elements of the dresser gradually remove material from the wheel's surface, exposing fresh grains and reshaping the wheel as needed. The dressing process can be performed manually or automatically, depending on the type of dresser used and the level of precision required.

TO-34-1-10-033

Figure 3-13. Hand Dressing a Grinding Wheel

3.13 WORK LAYOUT AND MOUNTING.

There are no specific, universal rules for laying out work for grinding operations, as layout requirements are largely determined by the specific grinding machine being used. In many cases, the workpiece will be prepared or rough-machined (such as being turned on a lathe) before grinding. The primary purpose of grinding in these cases is to achieve the desired final dimensions and surface finish.

3.13.1 Planning for Grinding. When planning for grinding, it is important to account for the material that needs to be removed based on the grinding machine's capabilities. For modern, well-maintained machines, it is acceptable to leave up to 1/32 inch or more of material on larger steel parts, while for smaller machine parts, it is typically best to leave no more than 1/64 inch for grinding. The precise allowance depends on the workpiece size and the machine's performance.

3.13.2 Cylindrical Grinding Operations. For cylindrical grinding operations, such as when grinding workpieces mounted on a lathe, it is necessary to drill and countersink center holes in the workpiece to allow it to be mounted between centers. The procedure for drilling and countersinking these center holes is the same as for lathe work. After drilling the centers, they should be smoothed to remove any burrs.

3.13.3 Mounting Work. Mounting work on a surface grinder involves several critical steps to ensure accuracy and stability during grinding. First, the workpiece must be cleaned and inspected for defects such as warping or cracks. A magnetic chuck is typically used to hold the workpiece, and it should be cleaned beforehand to ensure strong magnetic holding power. Once positioned on the chuck, the workpiece must be carefully aligned with the machine's axis using a dial indicator to avoid uneven grinding. For irregular or non-ferrous materials, clamps or v-blocks may be used as alternatives to the magnetic chuck. After aligning the workpiece, the magnetic chuck is activated, or the workpiece is securely clamped into place. It is essential to check the workpiece for stability by gently tapping it to ensure it is not loose. Finally, before starting the grinding process, the machine settings, including the height and position of the grinding wheel, should be checked to confirm everything is in place. These steps are necessary to achieve precise grinding results and prevent damage to both the workpiece and the machine.

3.14 GRINDING FEEDS AND SPEEDS.

In grinding operations, selecting the correct feeds and speeds is critical to achieving the desired surface finish, dimensional accuracy, and tool life. The grinding speed, typically referred to as surface speed, is the speed at which the grinding wheel's surface contacts the workpiece. It is usually measured in Feet Per Minute (FPM) or Meters Per Second (m/s). The feed rate refers to the rate at which the workpiece is moved into the grinding wheel. It can be specified in Inches Per Minute (IPM) or millimeters per minute (mm/min). Both these parameters depend on the material being ground, the type of grinding process, and the machine's capabilities.

3.14.1 Grinding Wheel's Surface Speed and the Feed Rate. The relationship between the grinding wheel's surface speed and the feed rate is essential for optimizing grinding performance. High surface speeds are typically used for finer finishes, while slower speeds are better suited for heavier stock removal. Feed rates should be carefully adjusted based on the type of grinding wheel and the material being processed. Harder materials generally require slower feed rates, while softer materials can handle faster feeds. The choice of coolant, grinding wheel type, and machine condition also influence the optimum speed and feed rates. To determine the correct feeds and speeds, several factors need to be considered:

3.14.1.1 Grinding Wheel Specifications. The type of abrasive, grit size, and bond material in the grinding wheel influence the choice of speed and feed. For example, a fine-grit wheel requires a lower feed rate compared to a coarse-grit wheel.

3.14.1.2 Workpiece Material. Harder materials, such as hardened steels, require lower speeds and feeds to prevent excessive heat generation and to maintain the integrity of the grinding wheel. Softer materials, such as aluminum, can typically tolerate higher speeds and feeds.

3.14.1.3 Machine Capabilities. The rigidity and power of the grinding machine also play a role. Machines with higher horsepower can handle faster feeds and higher speeds, whereas less powerful machines may need slower settings to avoid overloading the motor and compromising surface finish.

3.14.1.4 Cutting Depth. The depth of cut, or how much material is removed per pass, should be set according to the material and machine. Larger depths of cut may require slower feed rates to avoid overloading the wheel.

3.14.1.5 Charts. Various grinding speed and feed charts are available, often provided by grinding wheel manufacturers, that can serve as guidelines for specific materials and applications. See Table 3-4 for general guidelines.

Table 3-4. Grinding Speeds and Feeds

Material	Grinding Wheel Type	Bond Type	Recommended Wheel Speed (Surface Speed)	Recommended Feed Rate
Steel (Mild)	Alundum, Silicon Carbide	Vetrified	3,000-6,000 FPM	0.001-0.003 inch per revolution
Steel (Hardened)	CBN, Diamond	Resin, Vetrified	2,000-4,000 FPM	0.0005-0.001 inch per revolution
Aluminum	White Aluminum Oxide	Vetrified	4,000-8,000 FPM	0.004-0.008 inch per revolution
Cast Iron	Aluminum Oxide	Vetrified	3,000-5,000 FPM	0.002-0.004 inch per revolution
Brass	Silicon Carbide	Resin	3,000-5,000 FPM	0.002-0.004 inch per revolution
Stainless Steel	Alumina, CBN	Vetrified, Resin	2,000-5,000 FPM	0.001-0.003 inch per revolution
Tool Steel	CBN, Ceramic	Vetrified	3,000-6,000 FPM	0.0005-0.002 inch per revolution
Copper	Aluminum Oxide	Vetrified	FPM	0.003-0.006 inch per revolution
Plastic (Soft)	Silicon Carbide	Resin	2,500-4,000 FPM	0.005-0.01 inch per revolution
Plastic (Hard)	Aluminum Oxide	Resin	2,000-4,500 FPM	0.003-0.008 inch per revolution

3.14.2 Rotational Speed. When determining the appropriate rotational speed for abrasive wheels, the wheel diameter plays a key role in ensuring optimal grinding conditions. Smaller diameter wheels, such as those around 1 inch, require much higher rotational speeds to maintain the recommended surface speed, ranging from 36,000 to 72,000 RPM. As the wheel diameter increases, the rotational speed decreases accordingly to preserve the desired surface speed. For example, a 6 inch wheel should rotate between 6,000 and 12,000 RPM, while a 24 inch wheel requires a rotational speed between 1,500 and 3,000 RPM. These values are crucial for selecting the correct settings for various grinding operations to ensure efficiency and safety.

To calculate the Rotational Speed (RPM) for a specific wheel diameter, use the following formula:

$$\text{RPM} = \frac{\text{SURFACE SPEED (FPM) X 12}}{\pi \text{ X WHEEL DIAMETER (INCHES)}}$$

3.14.3 Safety.

WARNING

- It is important to always use a suitable coolant for the grinding operation. Insufficient cooling can result in the generation of excessive heat, leading to thermal damage of the workpiece and the grinding wheel, and can even cause the grinding wheel to glaze or burn.
- Always wear proper PPE when performing grinding operations. This includes safety goggles, face shields, hearing protection, and gloves (as applicable). Improper PPE can result in serious injury from flying debris or noise exposure.
- Periodically inspect the grinding wheel for wear, cracks, or damage. A damaged wheel should never be used, as it can shatter during operation and pose a serious safety hazard.

CAUTION

- Always ensure that the grinding wheel is properly balanced and that the machine is running at the specified maximum allowable speed. Do not operate a grinding wheel at speeds higher than the recommended speed. Failure to comply, could result in damage to, or destruction of, equipment or loss of mission effectiveness.
- Never increase the feed rate beyond what is recommended for the material and machine. Excessive feed rates can lead to overheating of the workpiece, grinding wheel damage, and reduced tool life. Overloading the machine with too high of a feed rate can also cause the grinding wheel to become clogged with material, reducing its cutting efficiency. Failure to comply, could result in damage to, or destruction of, equipment or loss of mission effectiveness.

Safety is a critical consideration during grinding operations to prevent accidents and ensure the well-being of the operator and the integrity of the machine. Proper precautions must be taken to minimize the risks associated with grinding, including potential wheel breakage, overheating, and exposure to debris. Safety considerations are listed above.

3.15 COOLANTS.

Modern grinding processes, especially in precision or semi-precision grinding, continue to benefit from the use of coolants. However, coolant types have evolved with advancements in formulations that offer better performance in terms of heat dissipation, lubrication, and chip removal. Today, synthetic and semi-synthetic coolants are widely used due to their improved effectiveness, better environmental profiles, and less residue compared to traditional water-based coolants.

3.15.1 The Role of Coolant. The role of the coolant remains the same: to control temperature (to avoid thermal distortion), keep the wheel clean, and improve surface finish. High-performance coolants now often have enhanced anti-wear, corrosion inhibition, and cleaning capabilities, extending the life of both the wheel and the workpiece.

3.15.2 Grinding Processes. While clear water is still used in some grinding processes, especially when cooling is the primary concern (as in grinding softer materials or when minimal lubrication is needed), most industrial applications have moved to water-soluble coolants or oils, which are designed for better lubrication and chip removal.

3.15.3 Workpiece and Machine Component Protection. Many modern coolants include biodegradable additives to improve lubrication and prevent rust, ensuring that the workpiece and machine components are protected. Some coolants also offer antimicrobial properties to prevent bacterial growth in the coolant tank, which is a growing concern in industrial settings.

3.16 TOOL AND CUTTER GRINDING.

Tool and cutter grinding, particularly the sharpening of milling cutters and reamers, is a highly specialized operation typically performed on universal tool and cutter grinding machines or specialized attachments designed for this purpose. This grinding process requires a thorough understanding of the different types of cutting tools and their specific cutting edges, relief angles, and other geometric features. Proper sharpening ensures the tools maintain their cutting efficiency, accuracy, and overall lifespan

3.16.1 Sharpening Milling Cutters and Reamers . Sharpening milling cutters and reamers involves precise grinding to restore the cutting edges to their original geometry, ensuring that the tools can function optimally in machining operations. The procedure for each type of tool varies based on its design, and special care must be taken to maintain the correct angles and surface finishes. For more detailed information on grinding milling cutters, refer to the section on milling cutter grinding and for reamers, refer to the section on reamer sharpening. On the other hand, sharpening tools like mill or lathe tools typically involves an off-hand (free-hand grinding) grinding procedure.

3.16.2 Hand Grinding. Hand grinding is often used for creating or re-shaping single-point cutters, form tools, and small-diameter cutters that may not be readily available. The process typically involves using a bench grinder or small tool grinder, where the operator manually holds the tool against the grinding wheel while carefully controlling the angles and dimensions. Special care must be taken to avoid overheating the tool steel during the grinding process, as excessive heat can alter the properties of the tool material, leading to reduced hardness or brittleness.

3.16.3 Forming Cutters from Tool Steel. Forming cutters from tool steel requires precision in achieving the correct rake angle, clearance, and cutting edge geometry (Figure 3-14), which directly impacts the performance of the tool. Grinding wheels with appropriate abrasives, such as aluminum oxide or silicon carbide, are commonly used for shaping the tool steel. After grinding, tools may require additional processes such as heat treatment or honing to refine the cutting edge further. This method of grinding is typically used in tool rooms or smaller shops where custom tools are needed, and it plays a vital role in maintaining the versatility and precision of the machining process.

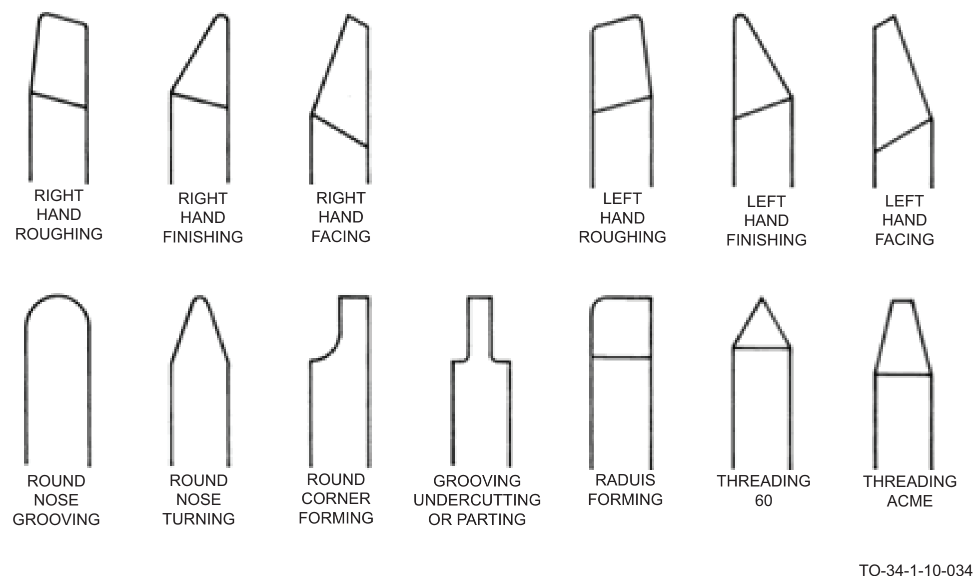


Figure 3-14. Lathe Cutting Tool Grind Examples

3.17 CONICAL AND CYLINDRICAL GRINDING.

Conical grinding is a specialized technique used in various industrial applications, where a conical shape is employed to enhance the grinding process. This grinding method typically involves a cone-shaped abrasive surface that facilitates a controlled and consistent material removal rate. The geometry of the conical shape allows for better control over the contact area between the grinding wheel and the workpiece, leading to reduced heat generation and improved surface finishes. Conical grinding is commonly used in precision manufacturing, such as in the production of gears, valves, and certain types of cylindrical parts.

3.17.1 Cylindrical Grinding. Cylindrical grinding is a widely utilized machining process that involves grinding the outer surface of a workpiece with a cylindrical shape. (See Figure 3-5.) This process is commonly used for producing smooth and precise cylindrical surfaces, typically in components such as shafts, rods, and tubes. The grinding wheel rotates around the workpiece, which is held between centers or on a chuck, ensuring uniform material removal across the surface. Cylindrical grinding provides high accuracy in terms of dimensions and surface finish, making it an essential process in manufacturing industries that require tight tolerances and high-quality finishes.

3.18 SURFACE GRINDING.

Surface grinding is a machining process used to achieve a smooth and precise flat surface on a workpiece. This process involves a rotating abrasive wheel that moves across the surface of the material, removing small amounts of material with each pass. Surface grinders are typically used for finishing flat surfaces, but they can also be used to grind profiles or contours on a workpiece. The workpiece is usually mounted on a magnetic chuck or clamped to ensure stability during grinding. Surface grinding is widely employed in industries such as aerospace, automotive, and metalworking, where high precision and smooth finishes are required. It is ideal for producing components like plates, flats, and various other parts that demand tight tolerance and high surface quality.

3.18.1 Surface Grinding Machines. Surface grinding machines can be either a manual or CNC-controlled machine (Figure 3-6 and Figure 3-7), with the grinding wheel typically made from abrasive materials such as aluminum oxide or silicon carbide, depending on the material being worked on. The process is versatile and can be used for a wide range of materials including metals, plastics, and ceramics.

CHAPTER 4
LATHES

4.1 GENERAL.

The lathe is a versatile machine tool used in various industries to shape and machine materials like metal and wood. Conventional lathes operate by rotating the workpiece against a cutter bit, allowing skilled technicians to manually create a variety of shapes and designs. These range from straight and tapered cylinders to screw threads and internal bores. The versatility of the lathe in the hands of a skilled professional makes it an essential tool for crafting intricate designs and achieving precise results. For technicians working with Computer Numerical Control (CNC) programming, the lathe takes on a new level of capability and precision. A CNC lathe automates the cutting process by using programmed instructions to guide the movement of the cutter bit. This results in greater accuracy and repeatability, especially for complex designs that would be challenging to achieve manually. CNC technology enhances the lathe's efficiency, allowing for faster production times and intricate designs. The combination of the lathe's inherent versatility and the precision of CNC technology makes it an indispensable tool in modern manufacturing.

4.2 SAFETY.

Safety is paramount when operating lathes, as improper use can result in severe injuries. Operators must always adhere to safety guidelines set forth by Occupational Safety and Health Administration (OSHA) standards (29 Code of Federal Regulations (CFR) 1910.) and applicable Air Force Instructions (AFIs) and Air Force Manuals (AFMAN) (e.g., DAFMAN 91-203). Supervisors are responsible for maintaining manufacturer manuals and integrating them into on-the-job training and job safety training. In the absence of manufacturer guidance, a hazard analysis and local operating instructions must be developed. Operators Must:

- Consider OSHA standards, relevant Air Force guidance and/or Original Equipment Manufacturer (OEM) recommendations for the specific machine and operation.
- Consult the specific manufacturer's manual for the lathe being used. Lathes vary significantly, and relying solely on general guidance can be dangerous.
- Wear safety glasses, and may require face shields, hearing protection, gloves and other necessary Personal Protective Equipment (PPE) to protect from flying debris, dust, and noise.
- Ensure all appropriate guards are in place to prevent accidental contact with rotating components.
- Follow lockout/tagout procedures to ensure the machine is powered off when performing maintenance.
- Keep the work area clean and free from clutter to avoid tripping hazards.
- Be familiar with the location and operation of emergency stop buttons and other safety features to quickly shut down the machine in case of an emergency.
- Remove or secure loose clothing and jewelry.
- Long hair must be secured to prevent entanglement with moving parts to minimize the risk of injury.

4.3 TYPES OF LATHES.

Lathes, whether manual or CNC, are fundamental machine tools in manufacturing and machining processes within various industries. They function by rotating the workpiece against a cutting tool, enabling the creation of cylindrical shapes and features. Manual lathes require skilled operators to control the cutting tool's movement, offering versatility but demanding precision and experience. Conversely, CNC lathes utilize computer programs to automate the cutting process, ensuring high accuracy and repeatability for complex designs. Understanding the different types of lathes, their capabilities, and their applications, whether manual or CNC, is essential for technicians and machinists seeking to maximize efficiency and precision in their work.

4.3.1 Engine Lathes. Engine lathes are versatile machine tools essential to most machine shops, used for a variety of turning operations. They are available in both bench-top and floor-mounted configurations and may be classified as toolroom, sliding-gap, or extension types. All engine lathes share key components: a headstock, tailstock, carriage, and bed, which provide a stable platform for machining. To handle large-diameter workpieces and heavy cuts, most include back-gearing for slow spindle speeds and high torque. They typically feature power feeds for controlled carriage movement and a lead screw with gearing for precise thread cutting. Engine lathe size is determined by swing (maximum workpiece diameter in inches), bed length (feet), and distance between centers (maximum workpiece length in inches).

4.3.1.1 Manual Lathes.

4.3.1.1.1 Engine Lathe. The most common type of manual lathe used in Aircraft Metals Technology (AMT) shops, recognized by its horizontal bed and headstock positioned on the left end. Engine lathes offer versatility in machining various shapes, from simple cylinders to complex contours. Their robust construction and wide range of spindle speeds make them suitable for diverse materials and applications. Within this category, several subtypes cater to specific needs.

4.3.1.1.1.1 Bench Lathe. (See Figure 4-1.) Compact enough for workbench mounting, these are ideal for lighter tasks, hobbyists, and educational settings.

4.3.1.1.1.2 Toolroom Lathe. (See Figure 4-2.) Prioritizing precision, these heavier-duty machines excel in crafting tools, dies, and high-precision components, boasting refined controls and a wider range of spindle speeds.

4.3.1.1.1.3 Gap Bed Lathe. (See Figure 4-3.) Featuring a removable bed section near the headstock, these accommodate larger diameter workpieces that wouldn't fit between the headstock and tailstock of a standard engine lathe.

4.3.1.1.1.4 Applications. Turning, facing, boring, threading, and taper turning on a wide range of workpieces.

4.3.1.1.2 Turret Lathe. Designed for efficiency, turret lathes feature a multi-tool turret mounted on the cross slide, replacing the tailstock of a conventional engine lathe. This turret houses multiple cutting tools, allowing for quick tool changes and execution of various machining operations in sequence without needing to reset the workpiece. This makes turret lathes ideal for high-volume production runs, particularly for parts requiring multiple machining operations.

4.3.1.1.3 Special Purpose Lathes. This category encompasses lathes engineered for specific machining tasks or industries. Examples include the wheel lathe for machining train wheels, crankshaft lathes for automotive applications, and gun drilling lathes for creating deep, precise holes. These lathes are tailored to meet the unique demands of specific machining operations or industries.

4.3.1.2 Lathe Key Components. A manual lathe's precision machining relies on the harmonious interplay of its key components. (See Figure 4-4.) Each component, from the sturdy bed to the precise tool post, plays a crucial role in ensuring accurate movement, clean cuts, and overall stability. Understanding these components is vital for safe operation, efficient maintenance, troubleshooting, and achieving optimal performance.

4.3.1.2.1 Bed. The bed forms the backbone of the lathe, providing a solid and accurately machined surface upon which all other components are mounted. Typically made of cast iron for its rigidity and vibration damping properties, the bed ensures the alignment of crucial components like the headstock and tailstock, providing a stable base for accurate machining.

4.3.1.2.2 Headstock. Located on the left end of the bed, the headstock houses the spindle, motor, and mechanisms for controlling spindle speed. The spindle, driven by the motor, rotates the workpiece at various speeds, enabling different machining operations.

4.3.1.2.3 Tailstock. Positioned opposite the headstock, the tailstock provides support for the workpiece, especially for longer pieces that might flex during machining. It can be adjusted along the bed to accommodate different workpiece lengths and can also house tools like drills and reamers, enabling operations like drilling and reaming.

4.3.1.2.4 Carriage. The carriage travels along the bed and carries the cutting tool. It consists of several interconnected parts, including the saddle, which slides along the bed ways, providing the carriage with longitudinal movement. This allows the tool to move along the length of the workpiece for various machining operations.

4.3.1.2.5 Cross Slide and Tool Post. The cross slide, mounted on the saddle, allows for precise radial movement of the cutting tool towards and away from the workpiece. This controls the depth of cut and enables facing operations. Situated on the cross slide, the tool post securely clamps and holds the cutting tool, allowing for tool height adjustment to ensure accurate positioning.

4.3.1.2.6 Feed Mechanisms. Manual lathes utilize various hand wheels and levers to control the movement of the carriage and cross slide. This enables the operator to precisely control the feed rate and depth of cut during machining.

4.3.1.2.7 Digital Readout (DRO). While not a standard feature on all manual lathes, Digital readouts are becoming increasingly common. They provide digital displays of the positions of the carriage and cross slide, improving precision and simplifying measurements compared to traditional graduated scales. This digital readout enhances the ease of use for the operator.

TO-34-1-10-035

Figure 4-1. Standard Bench-Type Engine Lathe

TO-34-1-10-036

Figure 4-2. Toolroom Lathe

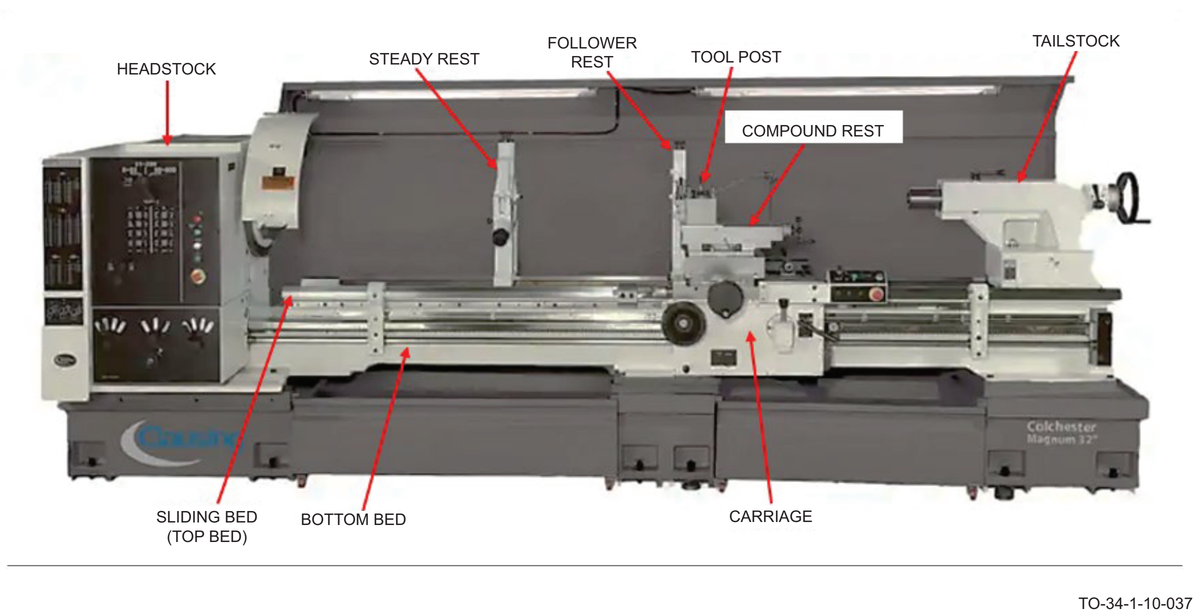


Figure 4-3. Sliding Gap-Type Engine Lathe

Figure 4-4. Manual Lathe Components

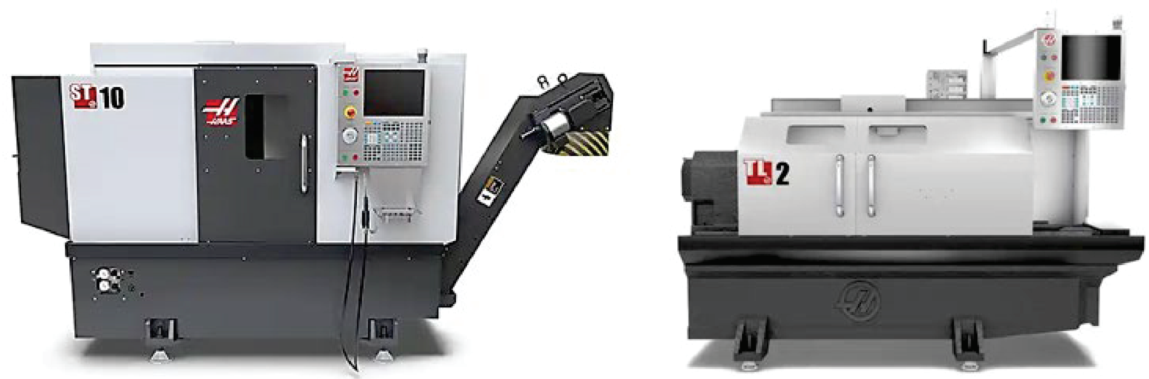
4.3.2 CNC Lathes.

4.3.2.1 2-Axis CNC Lathe. (See Figure 4-5.) The 2-axis CNC lathe, a common sight in many AMT shops, controls tool movement along two axes: X (radial) and Z (longitudinal). Ideal for cylindrical parts, these lathes efficiently execute turning, facing, grooving, and threading operations. They are widely used in industries like automotive and manufacturing for producing shafts, pins, and bushings.

4.3.2.2 3-Axis CNC Lathe. Adding a Y-axis to the traditional X and Z axes expands the capabilities of these lathes, allowing for off-center machining and the creation of more intricate geometries. The Y-axis provides movement perpendicular to both the X (radial) and Z (longitudinal) axes, essentially enabling the cutting tool to move up and down relative to the workpiece. This is crucial for features that are not concentric to the rotating axis of the workpiece. These machines are suitable for parts requiring milled features, slots, and off-center holes, making them common in industries demanding more complex designs, such as medical devices, tooling, and prototyping.

4.3.2.3 Multi-Axis CNC Lathe. For the most demanding machining tasks, multi-axis CNC lathes, equipped with four or more axes, take center stage. Often incorporating a secondary spindle and live tooling, these sophisticated machines allow for simultaneous machining operations. This capability enables the production of exceptionally complex parts in a single setup, a significant advantage in terms of efficiency and accuracy. Their ability to produce highly intricate components with tight tolerances makes them essential for industries like aerospace.

4.3.2.4 Swiss-Type CNC Lathe. Renowned for exceptional precision, Swiss-type lathes excel when machining small, intricate parts. Unlike traditional lathes where the workpiece is supported by a chuck and tailstock, Swiss-type lathes secure the workpiece in a collet, with additional support provided by a guide bushing. This setup ensures superior stability and accuracy, even for long, slender components, making them prevalent in industries like watchmaking, medical devices, and electronics, where minuscule parts with tight tolerances are paramount.



TO-34-1-10-039

Figure 4-5. 2-Axis CNC Lathes

4.3.2.5 CNC lathes share some similarities with their manual counterparts but incorporate advanced components to achieve computer-controlled machining. (See Figure 4-6.) Understanding these key components is crucial for both operators and programmers to maximize machine capabilities and ensure safe and efficient operation.

4.3.2.5.1 Bed. The bed of a CNC lathe, often constructed from cast iron or steel, provides a rigid foundation for all other components, much like in its manual counterpart. Its robust structure and precise machining are essential for ensuring accuracy and stability during high-speed operations. By minimizing vibrations, the bed contributes significantly to the overall precision of the machine.

4.3.2.5.2 Headstock. The headstock of a CNC lathe houses the spindle, responsible for rotating the workpiece. Unlike manual lathes, the spindle motor in a CNC lathe is typically controlled by a servo motor, allowing for precise speed and positioning. This precise control enables a wider range of machining operations and contributes to the high level of accuracy achievable with CNC lathes.

4.3.2.5.3 Tailstock. The tailstock on a CNC lathe, while similar in function to its manual counterpart, offers the advantage of automated control. Often hydraulically or servo-controlled, the CNC tailstock enables automated positioning and clamping of the workpiece. This ensures rigidity and precise positioning during machining, contributing to the overall accuracy of the process.

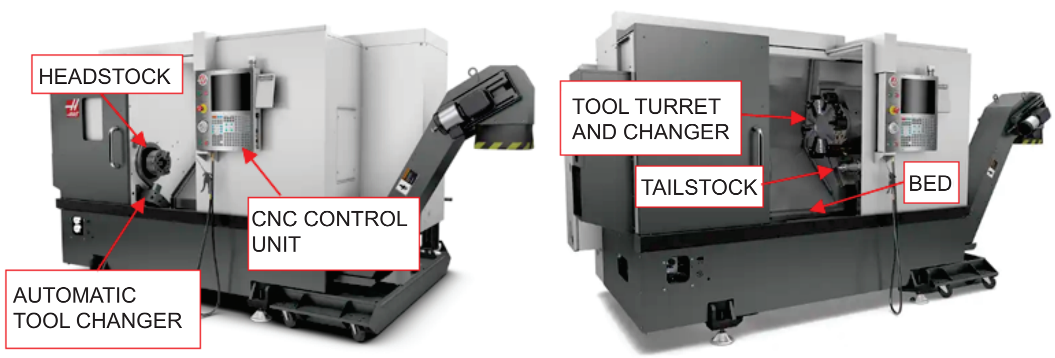
4.3.2.5.4 CNC Control Unit. The CNC control unit serves as the brain of the CNC lathe, interpreting the programmed instructions and translating them into actions. It sends signals that control the movement of the axes, spindle speed, coolant flow, and other functions, essentially orchestrating every aspect of the machine's operation. This centralized control is what allows for the automation and precision that CNC lathes are known for.

4.3.2.5.5 Servo Motors. Unlike manual lathes that rely on hand wheels and levers, CNC lathes employ servo motors on each axis (X, Z, and potentially others) to achieve precise movement. These motors receive commands from the CNC control unit and respond with high accuracy and repeatability, ensuring that the cutting tool moves to the exact positions dictated by the program. This precise and rapid movement is fundamental to the capabilities of CNC machining.

4.3.2.5.6 Tool and Turret Changer. CNC turret lathes streamline the machining process with a turret capable of holding multiple cutting tools. This turret rotates under automated control, bringing the desired tool into the cutting position precisely when needed. This eliminates the need for manual tool changes and allows for a variety of operations to be performed in a single setup, significantly increasing efficiency and reducing downtime.

4.3.2.5.7 Ball Screws and Linear Guides. Precision components like ball screws and linear guides are essential to the accuracy of a CNC lathe. These components work together to translate the rotational motion of the servo motors into smooth and precise linear movement of the cutting tool along each axis. This precise linear motion contributes significantly to the high level of accuracy and superior surface finish that can be achieved with CNC machining.

4.3.2.5.8 Automatic Tool Changer. To further enhance efficiency, many CNC lathes are equipped with an automatic tool changer. This system allows for the automatic swapping of cutting tools from a magazine, eliminating the need for manual intervention during tool changes. By automating this process, the automatic tool changer significantly reduces downtime and increases the overall productivity of the machine.



1. X- AND Z-AXIS SERVO MOTORS ARE TYPICALLY LOCATED BEHIND REAR OR SIDE PANELS.
2. THE X-AXIS BALL SCREW, DRIVING THE SADDLE ALONG THE Z-AXIS, IS POSITIONED AT THE REAR OF THE MACHINE BED. THE Z-AXIS BALL SCREW, EXTENDING FROM HEADSTOCK TO TAILSTOCK, LIES BENEATH THE SADDLE AND IS DRIVEN BY THE Z-AXIS SERVO MOTOR AT THE BED'S REAR.
3. THE X-AXIS LINEAR GUIDES ARE MOUNTED HORIZONTALLY ON THE SADDLE/TURRET BASE. Z-AXIS LINEAR GUIDES RUN PARALLEL TO THE SPINDLE ATOP THE MACHINE BED.

TO-34-1-10-040

Figure 4-6. CNC Lathe Key Components

4.4 TOOLING.

4.4.1 Lathe Cutter Bits.

4.4.1.1 Material and Technology Advancements. Lathe cutter bits efficiently remove metal through compression, shear, and heat. Sharp, well-supported cutting edges, optimized for the specific material and operation, are crucial for precision and efficiency. While traditional High-Speed Steel (HSS) bits are still common, modern materials offer significant advantages.

4.4.1.1.1 Advanced HSS. Improved hardness, toughness, and thermal resistance.

4.4.1.1.2 Powder Metallurgy Steels. Complex geometries and customized microstructures for enhanced performance.

4.4.1.1.3 Tungsten Carbide and Ceramic Coatings. Exceptional hardness and wear resistance.

4.4.1.1.4 Polycrystalline Diamond and Cubic Boron Nitride (CBN) Inserts. Ideal for machining extremely hard or abrasive materials.

4.4.1.1.5 Additively Manufactured Cutter Bits. Optimized geometries, microstructures, and features like nano-structured coatings for superior performance and longevity.

4.4.1.1.6 Smart Cutting Tools. Integrated sensors and algorithms enable real-time optimization and predictive maintenance.

4.4.1.1.7 Ceramics. Exceptionally hard and wear-resistant, surpassing even carbide in some applications. Suitable for very high-speed machining but can be brittle.

4.4.1.1.8 Carbide. Extremely hard and wear-resistant, ideal for high-speed machining of tough materials. Available as inserts (indexable or brazed) or solid carbide tools. Different carbide grades are optimized for specific material groups (e.g., steel, aluminum, cast iron). Cermets (ceramic/metal composites) are also used for high-speed finishing.

4.4.1.2 Choosing the Right Bit. Carefully consider the material being machined, the cutting operation, and desired performance. Proper maintenance, including sharpening and inspection, is essential for optimal tool life and cutting performance.

4.4.1.3 Single-Point Cutter Bits. Single-point cutter bits (See Figure 4-7.) Have specific components that affect their performance. Key terms and definitions include:

Shank

The main body of the cutter bit, which supports the cutting edge and nose.

Nose

The area that forms the cutting edge, responsible for removing material.

Face

The surface facing the workpiece, where the chip is formed and separated.

Side

The near-vertical surface that defines the bit’s profile, leading during cutting operations.

Base

The bottom surface of the shank, providing a foundation for secure mounting.

End

The near-vertical surface that forms the profile with the side, trailing during cutting operations

Heel

The section of the base that supports the face, maintaining stability and rigidity.

4.4.2 Understanding Cutter Bit Clearances and Angles. To optimize lathe performance, it is crucial to understand and set the correct angles on the cutter bit. (See Figure 4-7.) The following angles affect the cutting edge and chip removal.

4.4.2.1 Lip Angle. The angle between the side and face of the cutter bit, which varies depending on the material being machined (e.g., 61 degrees for soft steel, 85 degrees for hard cast iron).

4.4.2.1.1 Clearance Angles.

4.4.2.1.1.1 End Clearance Angle. The angle between the end of the cutter bit and a line perpendicular to the workpiece diameter at the point of contact.

4.4.2.1.1.2 Side Clearance Angle. The angle between the side of the cutter bit and the vertical.

4.4.2.1.2 Rake Angles. The angle between the face of the cutter bit and the horizontal, which can be back rake or side rake, and positive or negative.

4.4.2.1.3 Cutter Tool Holder Angle. This angle affects the end clearance and back rake angles, so it must be considered when setting these angles.

4.4.2.2 Setting Angles. Properly setting these angles is critical to prevent weakened or broken cutting edges, overheating, and reduced tool life. To optimize performance, add the cutter tool holder angle to the end clearance angle and subtract it from the back rake angle. By carefully setting these angles, you can improve lathe performance, reduce downtime, and produce high-quality workpieces with precision and accuracy.

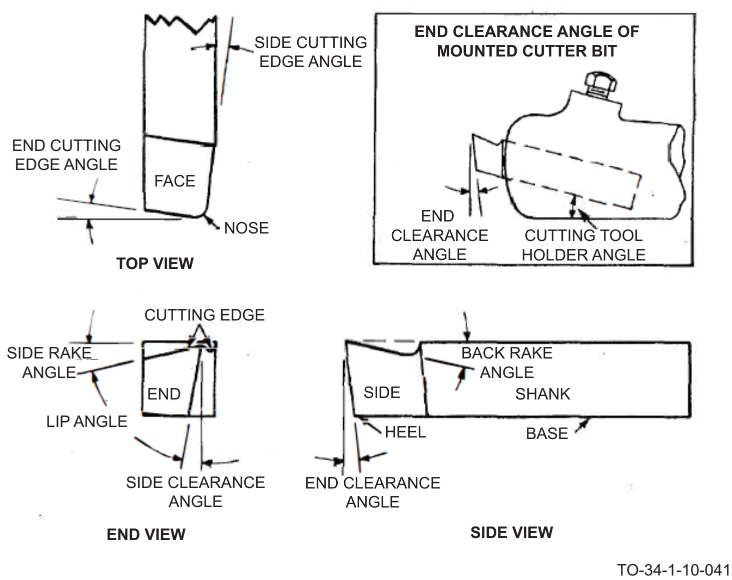


Figure 4-7. Terms Applied to Single-Point Cutter Bits

4.4.2.3 Clearance and Rake Angles. For optimal cutter bit performance, consult the table below for recommended clearance and rake angles, considering the lip angle during grinding.

Table 4-1. Clearance and Rake Angles for Lathe Cutter Bits

Material	Side Clearance Angle (Degrees)	End Clearance Angle (Degrees)	Back Rake Angle (Degrees)	Side Rake Angle (Degrees)
Aluminum	5-10	5-9	10-30	16-20
Brass	5-10	5-8	0-8	0-8
Bronze, free cutting	5-10	5-9	0-8	2-8
Bronze, tough	10-15	8-12	6-15	7-25
Cast Iron	5-10	5-9	5-8	12-14
Copper	7-14	7-12	8-16	16-25
Magnesium alloy	6-10	6-10	5-9	3-5
Monel	6-15	6-13	4-10	10-14
Plastic, cast	12	10	30	25
Plastic, laminated	8	8	30	30

Table 4-1. Clearance and Rake Angles for Lathe Cutter Bits - Continued

Material	Side Clearance Angle (Degrees)	End Clearance Angle (Degrees)	Back Rake Angle (Degrees)	Side Rake Angle (Degrees)
Steel, carbon	5-10	5-9	6-14	10-14
Steel, mild	5-12	5-9	8-17	14-20
Steel, stainless	5-12	5-10	8-17	0-10

4.4.3 Common Types of Cutter Bits. (See Figure 4-9.) Traditional HSS cutter bits often begin as blanks. (See Figure 4-8.) These blanks are then ground to the desired shape and sharpness for a specific machining application. To be used, the sharpened blanks are securely fitted into cutting tool holders, which are then fastened to the lathe's tool post. These bits are categorized by their intended function. For heavy roughing operations where a finished surface is not required, the cutter bit's nose should be ground to a small radius of approximately 1/64 inch. In contrast, for general shaping and finishing applications, a more rounded nose with a radius of 1/32 to 1/16 inch is recommended. This variation in nose radius allows for optimal performance and surface finish in different machining scenarios.

TO-34-1-10-042

Figure 4-8. Rectangle and Square HSS Lathe Cutting Blanks

4.4.3.1 Right Hand (RH) Turning Cutter Bit. (See Figure 4-9.) The RH turning cutter bit is designed to be fed from right to left, with the cutting edge located on the left side of the bit. The face of the tool slopes downward, away from the cutting edge, allowing for efficient material removal. To prevent interference with the workpiece, the left side and end of the tool are ground with sufficient clearance, enabling the cutting edge to engage the workpiece without the heel of the bit rubbing against it. This design makes the RH turning cutter bit ideal for taking light roughing cuts, as well as general-purpose machining operations, including all-around machine work.

4.4.3.2 Left Hand (LH) Turning Cutter Bit. (See Figure 4-9.) The LH turning cutter bit is a mirror image of the RH turning cutter bit, designed to facilitate cutting when fed from left to right. This complementary tool is utilized for general-purpose machining operations, particularly in situations where right-to-left turning is not feasible or practical. By providing a reversed cutting orientation, the LH turning cutter bit offers a versatile solution for all-around machine work, enabling machinists to adapt to varying workpiece configurations and operational requirements.

4.4.3.3 Round-Nose Turning Cutter Bit. (See Figure 4-9.) The round-nose turning cutter bit is a versatile tool designed for general-purpose machining operations, making it suitable for a wide range of applications. It can be used to take light roughing cuts, as well as finishing cuts, providing a high degree of flexibility in various machining tasks. Typically, the face of the round-nose turning cutter bit is ground with a righthand sloping side rake, allowing for efficient cutting when fed from right to left. However, it can also be ground without any side rake, enabling bidirectional feeding and providing machinists with the option to feed the tool from either left to right or right to left, depending on the specific requirements of the job.

4.4.3.4 RH Facing Cutter Bit. (See Figure 4-9.) The RH facing cutter bit is a specialized tool designed for facing operations on the RH side of shoulders and the right end of the workpiece. Its cutting edge is strategically located on the LH side of the bit, and the nose is precisely ground to a sharp point, allowing for the creation of a square corner with high accuracy. When using the RH facing cutter bit, it is essential to feed the tool in a direction away from the axis of the workpiece, ensuring a smooth and efficient machining process. This deliberate feed direction helps prevent interference with the workpiece and enables the production of a flat, even surface, making the RH facing cutter bit an ideal choice for various facing and squaring applications.

4.4.3.5 LH Facing Cutter Bit. (See Figure 4-9.) The LH facing cutter bit is a complementary tool to the RH facing cutter bit, designed to facilitate facing operations on the LH side of shoulders and other features. As a mirror image of its RH counterpart, the LH facing cutter bit is specifically intended for machining the LH sides of workpieces, allowing for efficient and accurate creation of flat surfaces and square corners.

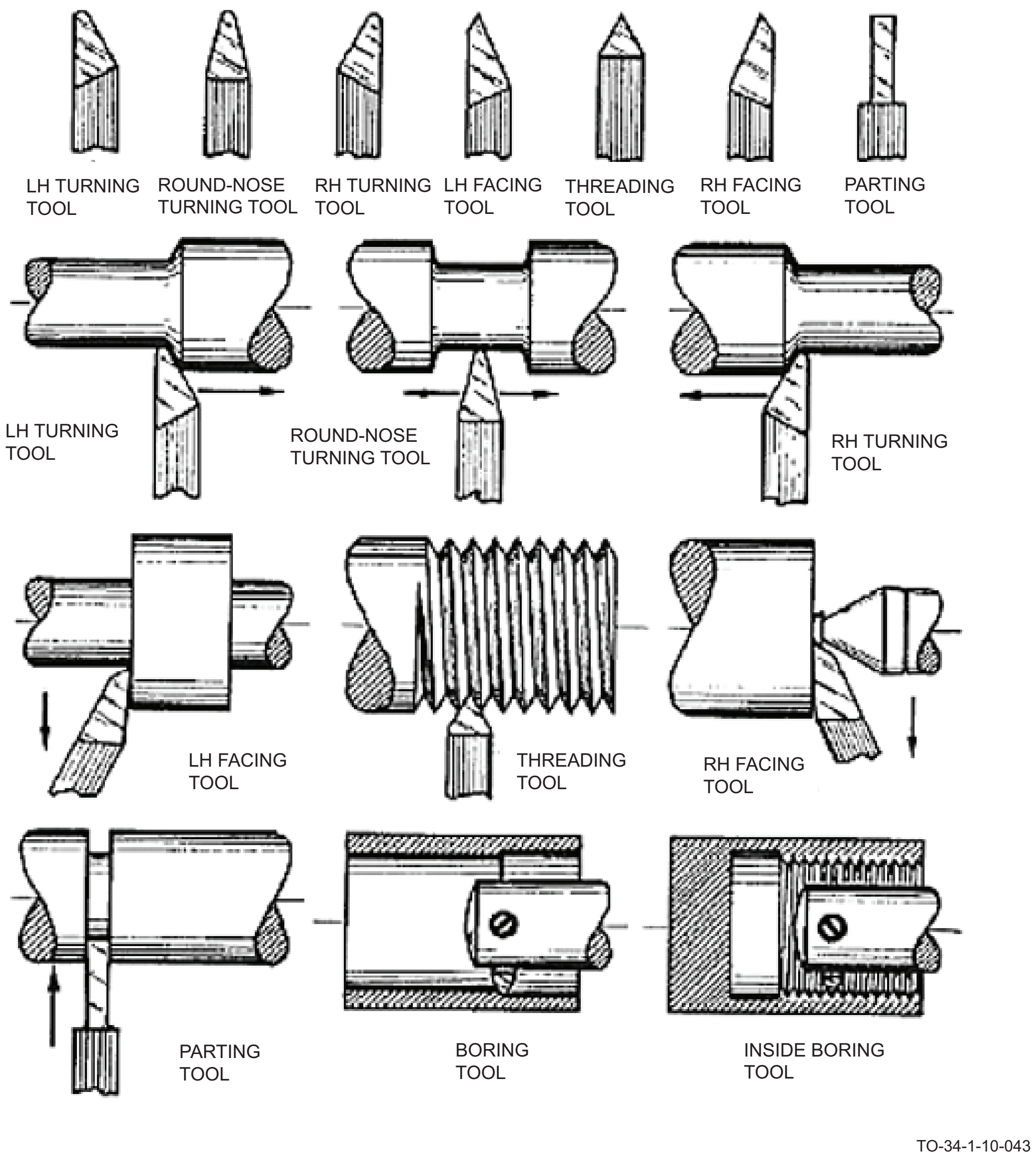


Figure 4-9. Common Types of Cutter Bits and Their Application

4.4.3.6 Parting Cutter Bit. (See Figure 4-9.) When working with a parting cutter bit, it is crucial to ensure proper grinding of both sides to achieve sufficient clearance and prevent binding. The bit should be slightly tapered, with the back being narrower than the cutting edge, to facilitate smooth and accurate machining. This specialized tool is ideal for complex features like necks, grooves, and square corners, as well as cutting-off operations, where its design enables precise material removal. To get the most out of your parting cutter bit, maintain a sharp cutting edge and regularly inspect the tool for wear or damage.

By following these best practices and leveraging the parting cutter bit's unique capabilities, you can optimize your machining operations and achieve high-quality results.

4.4.3.7 Thread Cutter Bit. (See Figure 4-9 and Figure 4-11.) The thread cutter bit is a precision tool for cutting sharp V-threads, with a 60 degree symmetrical cutting edge. For standard V-thread cutting, the bit's face is ground flat, with careful clearance grinding on both sides to prevent binding. When machining American (National) Standard screw threads, a flat is ground at the nose to accommodate the thread's flat root, with the width of this flat determined by the thread pitch. To ensure accurate thread formation, it is crucial to select the correct thread cutter bit for the specific thread pitch and profile. By using the thread cutter bit correctly, machinists can produce high-quality threads with precision and control, making it an essential tool for a variety of threading applications.

4.4.4 Special Types of Lathe Cutting Tools. Besides the common cutter bits as described in the previous paragraphs, special lathe operations and heavy production work require special types of cutting tools. Some of the more common of these tools are described below:

4.4.4.1 Tipped Carbide and Indexable Carbide Cutter Bits. (See Figure 4-10.) In high-performance machining, indexable carbide insert cutting tools and tipped cutter bits are preferred for their ability to handle high speeds and heavy cuts in tough materials. Tipped cutter bits, featuring tungsten carbide, tantalum carbide, or titanium carbide tips, are ideal for production work due to their exceptional hardness and wear resistance. These bits typically have larger shank sizes and are mounted in specialized holders, such as open side cutting tool holders, turret tool blocks, or lathe tool posts. Available in various shapes, tipped cutter bits can be used for turning, general-purpose machining, and thread cutting, offering versatility and precision. Indexable carbide insert cutting tools, in particular, provide a significant advantage by allowing for easy insert replacement, minimizing downtime and maximizing productivity.

4.4.4.2 Thread Cutting Tool Holder with Cutter. (See Figure 4-11.) For thread cutting operations, use a specialized thread cutting tool holder with a pre-formed cutter that matches the desired thread contour. This eliminates the need for complex setup or modification. To maintain the cutter, simply grind the face. No additional adjustments are required. The tool holder is designed to work with the lathe tool post, providing a secure and stable mount. This combination allows for efficient and precise thread cutting operations.

4.4.4.3 Knurling Tool. (See Figure 4-12.) The knurling tool features two rotating cylindrical cutters (knurls) mounted in a specialized toolholder. These knurls have teeth that imprint depressed patterns on the workpiece, providing a secure grip and improved functionality. To achieve different patterns and pitches, simply swap out the knurl pairs - common configurations include diamond patterns in 1/4 (14), 1/2 (21), and 1/3 (33) pitches, which produce coarse, medium, and fine textures, respectively. (See Figure 4-13.) This flexibility makes the knurling tool a valuable asset in machining applications where precise surface finish and texture control are required.

4.4.4.4 Boring Tools. When grinding boring tools, follow the same procedures for LH turning cutter bits and thread cutter bits as described in Paragraph 4.4.3.2 and Paragraph 4.4.3.7, with a focus on maintaining a sufficient end clearance angle to prevent heel interference with the bore. Mount the boring cutter bit to a boring bar, ensuring proper alignment and secure clamping. Then, install the boring bar into a suitable tool holder, which should be compatible with your lathe's tool post.

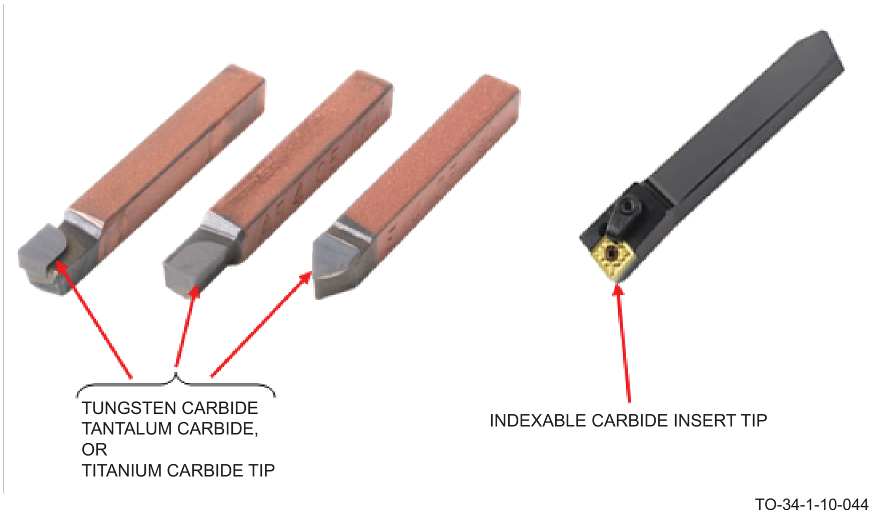


Figure 4-10. Tipped and Indexable Insert Carbide Cutting Bits

Figure 4-11. Thread Cutting Tool Holder and Cutter Bit

Figure 4-12. Knurling Tool

Figure 4-13. Diamond Knurling Tool Patterns

4.5 CUTTING TOOL HOLDERS.

Common cutter bits are generally made from standard sizes of bar stock to fit into a forged cutting tool holder which in turn is fastened to the tool post of the lathe. Special tools such as the knurling tool and the thread cutting tool holder with cutter are furnished with their own special forged tool holders and therefore may be fastened directly to the tool post of the lathe. Carbide tipped butter bits are generally unsuitable for mounting in forged tool holders and are fastened directly to lathe tool post. (See Figure 4-14.) The RH and LH offset cutting tool holders are designed to support RH and LH facing cutter bits which require that the bit be supported at an angle to the workpiece axis. The holder has a setscrew for locking the cutter bit in plate.

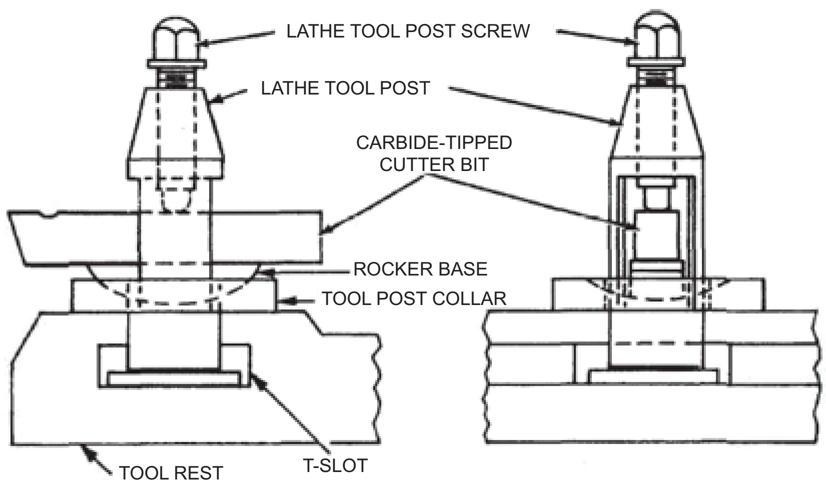


Figure 4-14. Carbide-Tipped Cutter Bit Mounted Directly to Tool Post of Lathe

4.5.1 Straight Shank Cutting Tool Holder. (See Figure 4-15.) The straight shank cutting tool holder may be used to support round-nose turning cutter bits, RH and LH turning cutter bits, and thread cutter bits. The holder is made of forged steel and contains a hardened steel set screw for locking the cutter bit in place.

4.5.2 RH and LH and LH Offset Cutting Tool Holder. (See Figure 4-15.) The RH and LH offset cutting tool holders are designed to support RH and LH facing cutter bits which require that the bit be supported at an angle to the workpiece axis. The holder has a setscrew for locking the cutter bit in place.

4.5.3 Straight Parting Cutting Tool Holder. (See Figure 4-15.) The straight parting cutting tool holder is a forged steel holder shaped to hold flat, thin sectioned parting tools which are used to separate pieces on the lathe.

4.5.4 RH and LH Offset Parting Cutting Tool Holder. (See Figure 4-15.) The RH and LH offset parting cutting tool holders are similar to the straight parting cutting tool holder but are designed to hold the parting cutter bit at an angle to the holder shank. The offset holder is generally used when the workpiece to be parted or the stationary parts of the lathe may interfere with the holder if the straight parting cutting tool holder is used. In either case, the compound rest of the lathe must be adjusted so that the parting cutter bit enters the workpiece at right angles to the workpiece axis.

4.5.5 Boring Bar Cutting Tool Holder. (See Figure 4-16.) The boring bar cutting tool holder for lathes consists of three main components: the holder, interchangeable end caps, and the boring bar. The boring bar is a threaded rod that accepts end caps with different angle slots (30, 45, and 90 degree) to hold cutter bits. Alternatively, plain boring bars can be used with slots at both ends (90 and 45 degree). The holder is made of forged steel and attaches to the lathe tool post via a screw. It features an adjustable boring bar that can be locked in place at any desired position, providing flexibility and precision for boring operations.

Figure 4-15. Different Types of Cutting Tool Holders

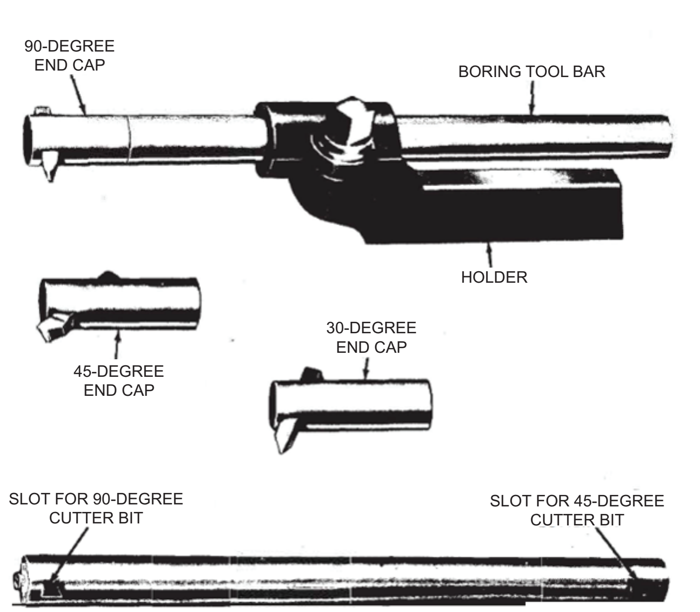


Figure 4-16. Boring Bar Cutting Tool Holder with End Caps and Cutter Bits

4.6 CHUCKS.

To secure workpieces to a lathe, machinists use various holding devices on the headstock spindle, including chucks, face-plates, and lathe centers. These chucks apply pressure to hold the workpiece in place. Common types of chucks used on lathes include independent (4-Jaw), universal scroll (3-Jaw), drill, spindle, tailstock, collet, and step chucks, each designed for specific applications.

4.6.1 4-Jaw Independent Chuck. (See Figure 4-17.) The 4-Jaw independent chuck features four individually adjustable jaws, secured to the chuck face with adjusting screws. To center a workpiece, use the scribed concentric circles for rough alignment, then fine-tune by rotating the workpiece and checking concentricity with precision gages. The jaws can be adjusted to achieve precise alignment within tight tolerances. Additionally, the jaws can be reversed to grip workpieces internally or externally. The independent chuck's versatility allows it to hold various shapes (square, round, octagonal, or irregular) in concentric or eccentric positions. Its ability to make fine adjustments makes it ideal for mounting workpieces that require extreme accuracy.

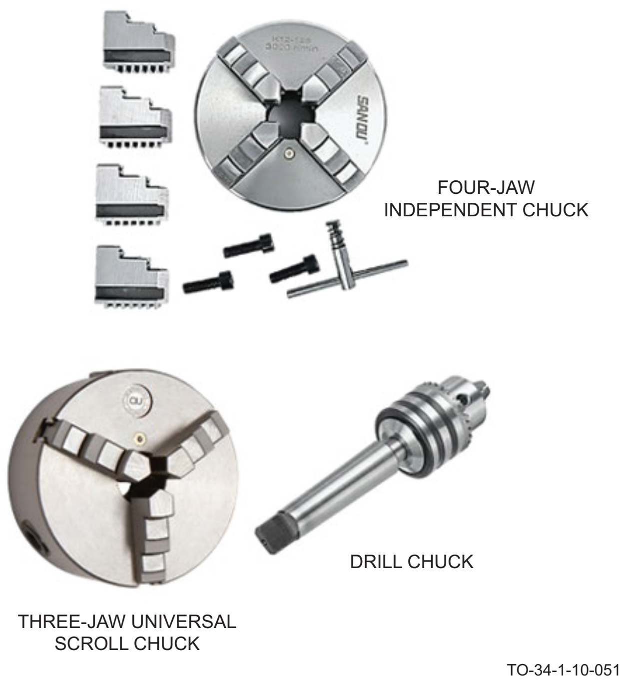


Figure 4-17. 4-Jaw Independent Chuck, 3-Jaw Universal Scroll Chuck, and Drill Chuck

4.6.2 3-Jaw Universal Chuck. (See Figure 4-17.) The universal scroll chuck, also known as a 3-Jaw chuck, features three jaws that move in unison when the adjusting pinion is rotated. This design allows for easy and automatic centering of workpieces for concentric turning, with an accuracy of ±0.003 inches. The jaws are connected to a scroll plate and move simultaneously but cannot be reversed due to their individual alignment. However, the chuck often comes with interchangeable jaw sets. The universal scroll chuck is suitable for holding and centering round or hexagonal workpieces, but is not recommended for square, octagonal, or irregular shapes due to its 3-Jaw design.

4.6.3 Drill Chuck. (See Figure 4-17.) The drill chuck is a compact, universal-type chuck that can be mounted in either the headstock or tailstock spindle. It is designed to hold straight-shank tools such as drills, reamers, and taps, as well as small diameter workpieces. The chuck features 3 or 4 hardened steel jaws that are actuated by a tapered sleeve, allowing for precise centering of tools and workpieces to within 0.002-0.003 inches when securely tightened.

4.6.4 Collet Chuck. (See Figure 4-18.) The collet chuck is the most precise method for holding small workpieces in a lathe. It consists of a spring machine collet and a collet attachment that secures and regulates the collet on the headstock spindle. The collet is a thin, accurately machined tube with a tapered exterior and three lengthwise slots, allowing it to be sprung inward to grip the workpiece. For accurate gripping, the collet must be within 0.001 inches of the workpiece diameter, so collets are typically supplied in sets with various capacities in 1/16, 1/32, or 1/4 inch increments, with a maximum capacity of 1 inch in diameter. The collet attachment includes a collet sleeve, draw bar, and handwheel or lever (Figure 4-19), which work together to pull the collet inward and tighten it around the workpiece. Collet chucks are commonly used on toolroom lathes and horizontal turret lathes, and are available in different shapes to accommodate round, square, and hexagonal workpieces of small dimensions.

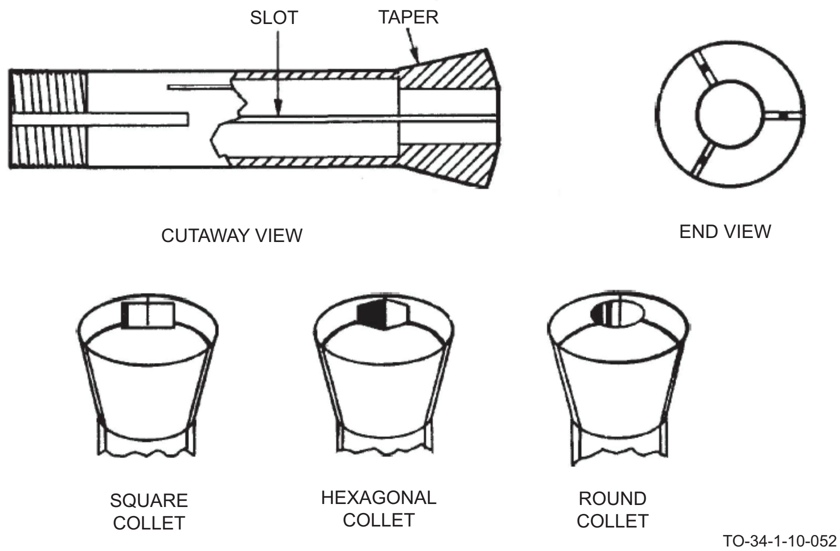


Figure 4-18. Different Shapes of Spring Collet Chucks

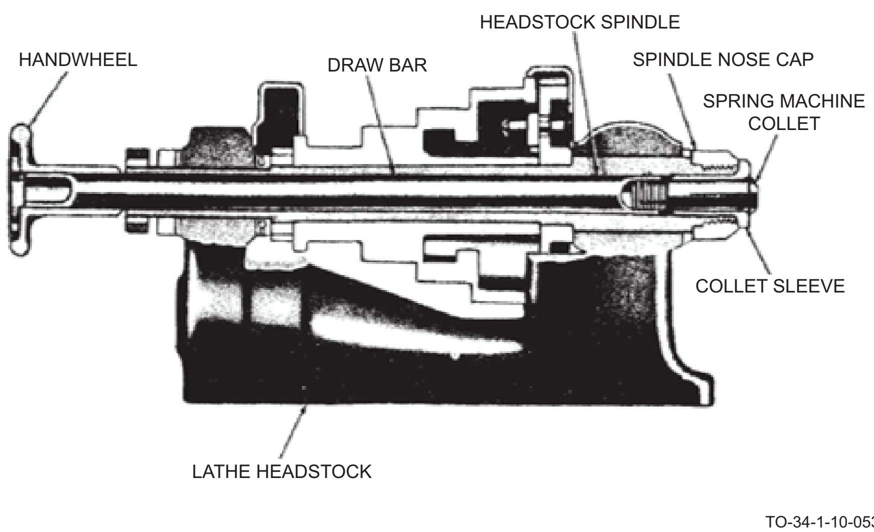


Figure 4-19. Typical Installation of Collet Chuck

4.6.5 Step Chuck. (See Figure 4-20.) The step chuck is a variation of the collet chuck designed for precise holding of workpieces larger than 1 inch in diameter. It consists of a handwheel or lever-operated collet attachment and a step chuck machine collet, which replaces the standard spring machine collet. The step chuck machine collet is split into three sections and threaded to the drawbar. As it is drawn into the collet sleeve, the collet sections are cammed against the workpiece by an internal taper, providing secure holding. Step chucks are available in 2 through 5 inch sizes, indicating the maximum workpiece diameter they can support. The machine collets are supplied blank and must be machined on the lathe to the desired step diameter, allowing for customized fitting of specific workpieces.

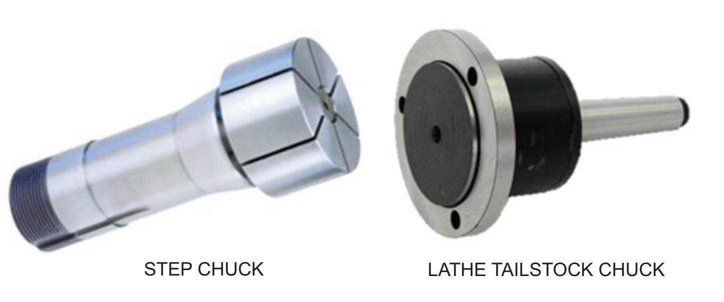


Figure 4-20. Step Chuck Machine Collet and Lathe Tailstock Chuck

4.6.6 Lathe Tailstock Chuck. (See Figure 4-20.) The lathe tailstock chuck is a device used to support workpiece ends in the tailstock when a lathe center is not feasible. It features a taper arbor that fits the tailstock spindle and three self-centering bronze jaws that can accurately grip workpieces between 1/4 and 1 inch in diameter. The bronze jaws provide a smooth bearing surface for the workpiece. To use, the jaws are adjusted to fit the workpiece diameter and then locked in place, providing secure support for the workpiece.

4.7 LATHE FACEPLATES.

A lathe faceplate (Figure 4-21) is a flat, round plate that threads onto the headstock spindle, used for holding irregularly shaped workpieces that can't be secured with chucks or mounted between centers. Workpieces are attached to the faceplate using angle plates, brackets, or direct bolting, and radial T-slots on the plate's surface provide additional mounting options. The faceplate is particularly useful for machining eccentric holes or projections, and its versatility depends on the machinist's creativity, making it a valuable tool for a wide range of applications. A driving faceplate is a small faceplate used to drive the lathe dog for workpieces mounted between centers. (See Figure 4-21.) It typically has fewer T-slots than a standard faceplate. To use, a lathe dog (Figure 4-23) is attached to the workpiece and engaged in one of the driving faceplate's slots, allowing the faceplate to drive the workpiece as it rotates.

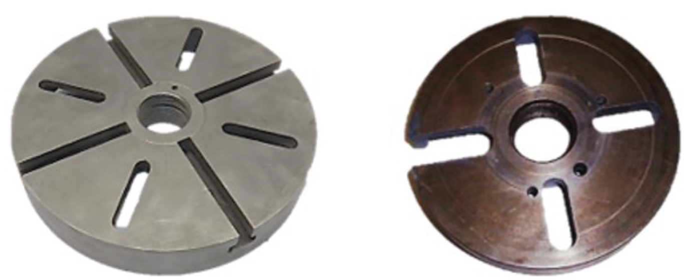


Figure 4-21. Lathe Faceplates

4.8 LATHE CENTERS.

Lathe centers are the most common method for supporting workpieces in a lathe. (See Figure 4-22.) They typically have a 60 degree tapered point and are used in pairs: one in the headstock spindle and one in the tailstock spindle. The centers have standard tapered shanks that fit directly into the tailstock and into the headstock spindle using a center sleeve. There are two main types of centers: live centers, which rotate with the workpiece, and dead centers, which remain stationary. The most common types of centers include the following:

4.8.1 Male Center (Also Known as a Plain Center). Used for general lathe turning operations, with a 60 degree cone angle. When used in the headstock, it is called a live center, and when used in the tailstock, it is called a dead center.

4.8.2 Pipe Center. Similar to a male center, but with a larger size and greater cone angle, used for holding pipe and tubing.

4.8.3 Half Male Center. A male center with a portion of the cone cut away, used as a dead center in the tailstock for facing operations.

4.8.4 Female Center. Conically bored at the tip, used to support workpieces with pointed ends.

NOTE

Dead centers require frequent lubrication to prevent overheating. For dead center lubrication, use a high-temperature, extreme pressure grease that can withstand the friction and heat generated by the stationary dead center and rotating workpiece.

4.9 LATHE DOGS.

Lathe dogs connect the headstock spindle to a workpiece mounted between centers, ensuring synchronized rotation. (See Figure 4-23.) The three main types of lathe dogs are, bent and straight tail lathe dogs that engage with the faceplate or a stud, and the bent tail clamp lathe dog. (See Figure 4-23.) For safety, use bent tail dogs with headless setscrews to minimize clothing entanglement. The bent tail clamp lathe dog is most effective for rectangular workpieces. When selecting a lathe dog, consider the workpieces shape, size, and material for optimal performance and safety.

Figure 4-22. Types of Lathe Centers

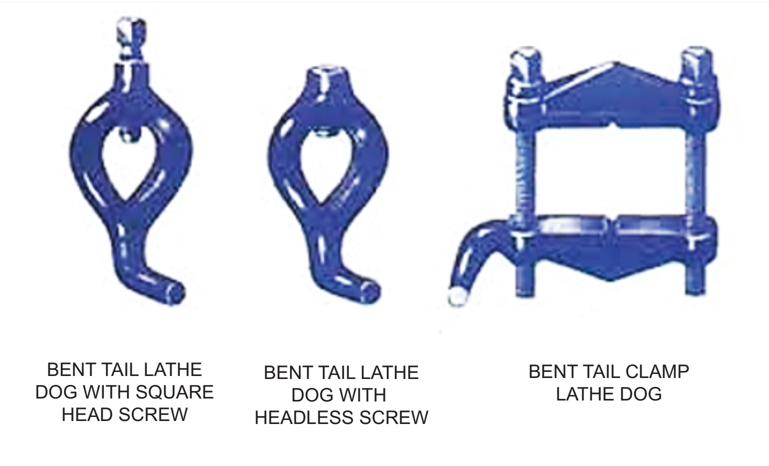


Figure 4-23. Common Types of Lathe Dogs

4.10 MANDRELS.

A workpiece with a drilled or bored axis that can't be held between centers or in a chuck is typically machined on a mandrel. A mandrel is a tapered axle that fits into the workpieces bore, supporting it between centers. Note that a mandrel is distinct from an arbor, which holds tools, not workpieces.

4.10.1 Solid Machine Mandrel. (See Figure 4-24.) A solid machine mandrel is made from hardened steel, ground to a 0.0005-0.0006 Threads Per Inch (TPI), and features precision counter-sunk centers for mounting between centers. The ends have smaller diameters, machined flats for lathe dog grip, and the size is stamped on the large end of the taper.

4.10.2 Expansion Mandrel. (See Figure 4-24.) Solid machine mandrels have a slight taper, limiting them to workpieces with specific inside diameters. Expansion mandrels offer more flexibility, accommodating a wider range of sizes by using a chuck-like mechanism that expands to grip the workpieces interior.

Figure 4-24. Types of Mandrels

4.11 RESTS.

Workpieces often need extra support, especially long, thin workpieces that tend to spring away from the cutter bit. Two common supports or rests are the steady rest and the follower rest.

4.11.1 Steady Rest. (See Figure 4-25.) The steady rest (or center rest) supports long workpieces during turning, boring, and internal threading operations. It clamps to the lathe bed and holds the workpiece within three adjustable jaws, requiring a concentric bearing surface on the workpiece. Proper alignment and lubrication are crucial, and the jaws must be locked in position. The steady rest's top section swings away for easy workpiece removal without disturbing the jaw setting.

4.11.2 Follower Rest. (See Figure 4-25.) The follower rest has one or two jaws that support the workpiece, attached to the lathe carriage to follow the cutter bit. It is applied after the cut is started and continued for a short distance, typically used for straight turning and threading of long, thin workpieces.

4.12 TOOL POST GRINDER.

The tool post grinding machine (Figure 4-26) is a lathe attachment for cylindrical grinding operations. It consists of a 1/4 or 1/2 horsepower motor, wheel spindle, and pulleys, and mounts to the compound rest via a T-slot bolt. The machine accommodates grinding wheels from 1/4 to 4 inches in diameter, with interchangeable pulleys for optimal cutting speeds. Larger wheels are attached via an arbor, while smaller wheels are fixed in a chuck. The motor is connected to a power source via a cable and plug and typically features a switch for easy start/stop control.

Figure 4-25. Steady and Follower Rests

Figure 4-26. Tool Post Grinder

4.13 LAYING OUT WORK.

To prepare a workpiece for the lathe, determine the best method for supporting it and locate centers if necessary. The method of mounting depends on the workpieces size, shape, and operation.

- Small, long workpieces are typically mounted between centers.
- Workpieces requiring extensive facing, boring, or internal threading are mounted in a chuck, with a steady rest used for long pieces.
- Irregularly shaped workpieces are fastened to a faceplate.

4.13.1 Locating Centers on Cylindrical Workpieces.

NOTE

- Prior to locating the center of the cylindrical workpiece, ensure that the surface is thoroughly cleaned and free from any debris, dirt, or residual materials. Additionally, if the workpiece has rough or sharp edges, lightly deburr or chamfer them to prevent damage to the measuring tools.
- Apply a thin, even layer of Prussian blue or layout dye to the workpiece surface to enhance visibility and reveal scribed lines, markings, or other precision features. Ensure the dye is evenly distributed and allowed to dry completely before inspecting or working with the marked surface.

4.13.2 Hermaphrodite Caliper Method. (See Figure 4-27.) Set a pair of hermaphrodite calipers to approximately one half the workpiece diameter and scribe four short arcs. The arc will enclose the center.

4.13.3 Center Head Method. (See Figure 4-27.) Hold the center head of a combination machinist's square firmly against the workpiece and scribe a line close to the blade. Give the workpiece a quarter turn and scribe a similar line. The point at which the lines intersect will be the center.

4.13.4 Divider Method. (See Figure 4-27.) With the workpiece on a flat surface, set the dividers to approximately one half the diameter of the piece and scribe four lines across each end. The center will be within the small square thus formed.

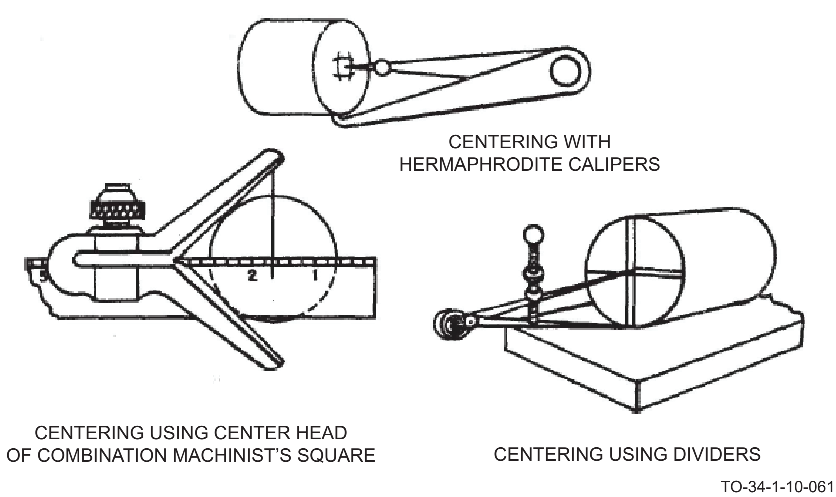


Figure 4-27. Locating Centers on Bar Stock

4.13.5 Locating Centers on Irregular Workpieces. Unlike symmetrical workpieces, irregular parts lack obvious centers, requiring a combination of common sense, good judgment, and layout experience to determine the correct location.

4.13.5.1 Examine the Workpieces Geometry. To locate centers, machinists must examine the workpieces geometry, considering features like holes, curves, and flat surfaces, as well as its intended purpose. Here are key tips to help:

4.13.5.1.1 Visual Inspection. Carefully inspect the workpiece to identify any visible features that could help determine the center, such as a hole or a depression.

4.13.5.1.2 Measurement. Take measurements of the workpiece to determine its dimensions and identify any symmetrical features that could aid in locating the center.

4.13.5.1.3 Layout Techniques. Use layout techniques such as scribing lines, creating reference points, or using precision instruments like calipers or micrometers to help locate the center.

4.13.5.1.4 Divide and Conquer. Break down the workpiece into smaller, more manageable sections, and focus on locating the center of each section individually.

4.13.5.1.5 Use of Auxiliary Tools. Utilize auxiliary tools such as centering fixtures, V-blocks, or rotary tables to help locate the center of the workpiece.

4.13.5.1.6 Experience and Intuition. Rely on experience and intuition to make an educated guess about the location of the center and then verify the guess using measurements and layout techniques.

4.13.6 Testing Centers. When the centers are accurately located, they should be tested before drilling and countersinking.

a. Carefully indent the centers lightly with a center punch.

b. Place the workpiece in the lathe between lathe centers.

c. Hold a piece of chalk lightly against the workpiece and supporting the chalk well, rotate the workpiece slowly by hand. (See Figure 4-28.) High spots can be clearly marked in this manner at either end of the piece, and the centers can be corrected as necessary.

d. If it is necessary to change the center marks after testing, a satisfactory method of doing so is to support the piece in a vise and, holding a center punch at an angle, drive the center mark in the desired direction.

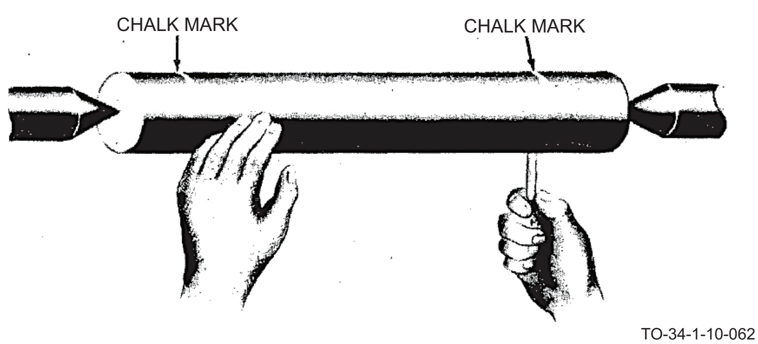


Figure 4-28. Method of Testing Location of Centers

4.13.7 Drilling and Countersinking Center Holes. Use a center drill and 60 degree countersink or, a center drill with an included angle of 60 degrees (See Figure 4-29.) Ensure the holes have a uniform 60 degree taper and clearance at the bottom for the lathe center point. Drilling and countersinking can be done on the lathe with a spindle speed of 600 Revolutions Per Minute (RPM) and light feed to prevent breaking the drill point. Figure 4-30 illustrates correctly and incorrectly drilled center holes. The holes should have a polished appearance so not to score the lathe centers.

4.13.7.1 When preparing workpieces for mounting between centers on a lathe, use a 0.043 inch drill and 1/8 inch countersink for 1/2 inch diameter stock, and a 0.157 inch drill with a 7/16 inch countersink for 2 inch diameter stock. Ensure sufficient material remains after facing for adequate lathe center bearing surface. Always consider material thickness and consult machining data for precise recommendations.

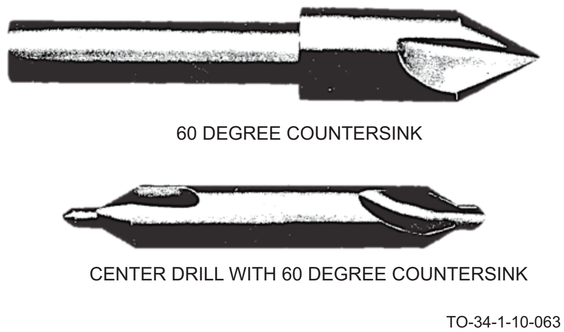


Figure 4-29. Tools for Drilling and Countersinking Center Holes

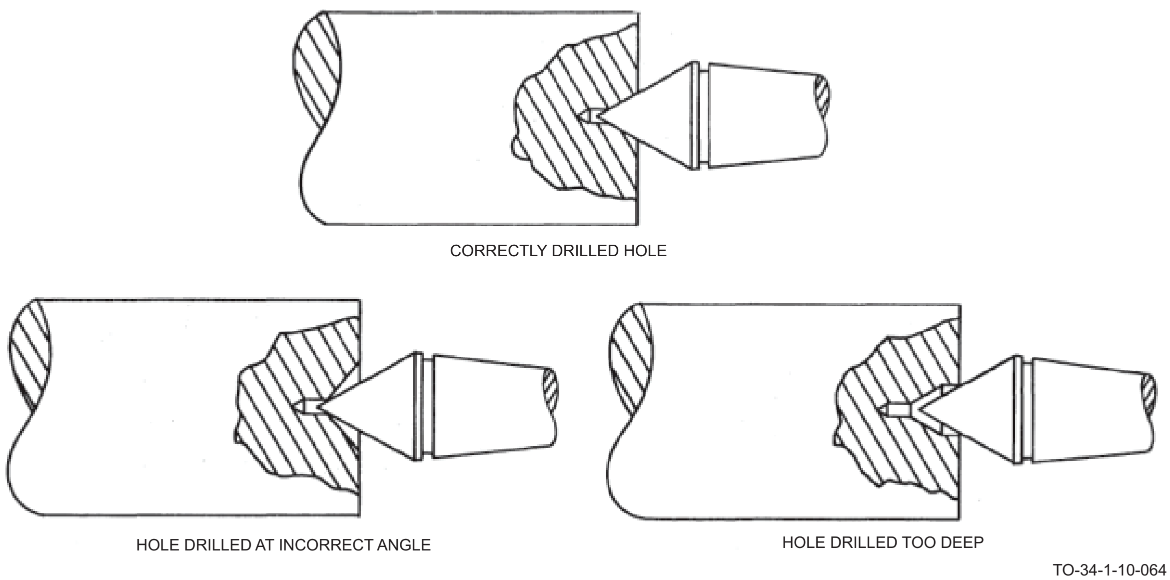


Figure 4-30. Correctly and Incorrectly Drilled Center Holes

4.14 MOUNTING WORKPIECE BETWEEN CENTERS.

4.14.1 Inserting and Removing Lathe Centers. The quality of workmanship depends as much on the condition of the lathe centers as on the proper drilling of the center holes. Before mounting lathe centers in the headstock or tailstock, thoroughly clean the centers, the center sleeve, and the tapered sockets in the headstock and tailstock spindles. Any dirt or chips on the centers or in their sockets will prevent the centers from seating properly and will cause the centers to run out of true.

a. Install the lathe center in the tailstock spindle with a light twisting motion to ensure a clean fit. Install the center sleeve into the headstock spindle and install the lathe center into the center sleeve with a light twisting motion.

NOTE

When male centers are supplied in pairs, the tailstock center is usually distinguished from the headstock center by a groove close to the tapered point. This groove indicates that the tailstock center has been hardened and tempered for use as a dead center. The un-grooved headstock center is not hardened because it will rotate with the work as a live center.

b. Remove the center from the headstock spindle by holding the pointed end with a cloth or rag in one hand and giving the center a sharp tap with a rod or a locally manufactured **knockout bar** inserted through the hollow headstock spindle.

c. Remove the center from the tailstock by turning the tailstock handwheel to draw the tailstock spindle into the tailstock. The center will contact the tailstock screw and will be bumped loose from its socket.

4.14.2 Grinding Lathe Centers. (See Figure 4-31.) Occasionally, it will be necessary to grind or redress the lathe centers when they become scored, misaligned, or worn. The grinding is accomplished in the following manner:

a. Carefully clean the spindle taper socket, lathe center, and center sleeve. Inspect these mating parts carefully and see that they are free from burrs and scoring.

b. Insert the lathe center and center sleeve solidly in the headstock spindle.

c. Set the compound rest at an angle of 30 degree to the axis of the lathe.

d. Clean and lubricate the dovetail slide upon which the compound rest moves. Adjust the tapered gib at the side of the compound rest to remove all looseness without causing the rest to bind.

e. Cover the carriage and ways with paper or cloth to protect them from particles of abrasive grit from the grinding abrasive wheel. This step is important, as the grit could become imbedded and quickly destroy the accuracy of the lathe.

f. Mount a tool post grinder in the T-slot of the compound rest. Adjust the face of the grinding abrasive wheel parallel to the conical surface of the center. See that the point of the lathe center is aligned with the center hole in the end of the arbor which retains the grinding abrasive wheel on the wheel spindle. Check the grinding machine for end play and adjust if necessary. True the grinding abrasive wheel.

g. Set the compound rest near the center of its travel and move the carriage and crossfeed to position the grinding abrasive wheel within a few thousandths of an inch from the lathe center. Lock the carriage in this position to prevent movement during the grinding operation.

h. Start lathe at moderate speed (60 to 100 Feet Per Minute (FPM)) and then start the grinding machine (lathe is typically placed in reverse gear), making sure that the lathe and grinding abrasive wheel both rotate in the same direction (the workpiece and the wheel will be moving in opposite directions at the point of contact).

i. Move the grinding abrasive wheel until it touches the rotating lathe center. Set depth of cut between 0.001 and 0.002 inch. Carefully feed the grinding abrasive wheel along the face of the lathe center and feeding toward the headstock. Make additional passes in the same direction, increasing the feed with each pass until the center is true and smooth.

j. Stop the lathe and grinding machine and use a 60 degree angle center gage (Figure 4-32) to check the center for accuracy. If the angle is not a true 60 degree, readjust the compound rest and repeat the previous two steps.

CAUTION

After the center has been grounded, always remove sharp points using an emery cloth. Failure to comply, could result in damage to, or destruction of, equipment of loss of mission effectiveness.

k. After grinding each center, polish it with a strip of fine emery cloth. Use a high spindle speed, such as 2000 RPM, for polishing. It is best practice to grind the tailstock center first, leaving the headstock center for last. This way, the freshly polished headstock center remains undisturbed in the spindle, maximizing accuracy.

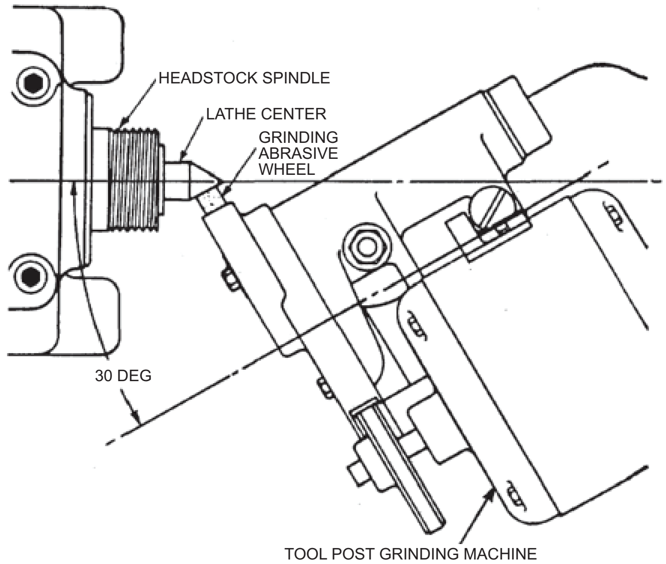


Figure 4-31. Grinding Lathe Centers with a Tool Post Grinding Machine

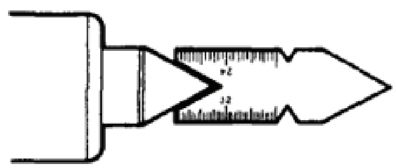


Figure 4-32. Checking a Center for Accuracy with 60 Degree Center Gage

4.14.3 Checking Alignment of Centers. To turn a shaft straight and true between centers, the centers must be in a plane parallel to the lathe's ways. This can be achieved by releasing the tailstock from the ways and using adjusting screws to move it laterally. Align the two zero lines at the rear of the tailstock (Figure 4-33) and verify by moving the tailstock close to the headstock to check the centers' relative position.

4.14.3.1 To check center alignment, mount a workpiece between centers and take light cuts at both ends without changing carriage adjustments. Measure the diameters at each end with calipers or a micrometer. If the tailstock end is larger, turn the adjusting screw on the tailstock rearward; if smaller, turn it forward. Repeat the process until both ends measure the same diameter.

4.14.3.2 Another good method of checking center alignment is by mounting a precision test bar between centers and locking the tailstock to the bed ways. Use a dial indicator on the tool post to bring both ends of the bar to a zero reading (Figure 4-33), ensuring the bar is firmly seated between centers.

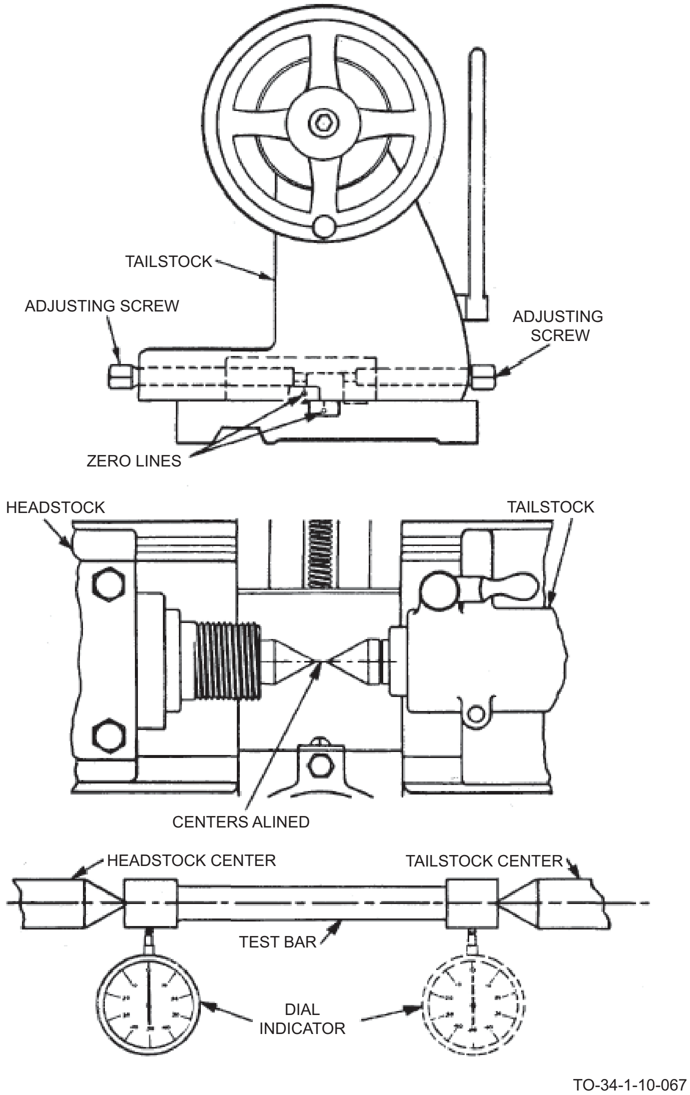


Figure 4-33. Alignment of Lathe Centers

4.14.4 Setting Up Workpieces Between Centers. After the lathe centers have been properly installed and aligned, and the workpiece correctly drilled and countersunk, mount the piece in the lathe using a driving faceplate and a lathe dog. (See Figure 4-34.)

4.14.4.1 Ensure that the external threads of the headstock spindle are clean before screwing on the driving faceplate. Screw the faceplate securely onto the spindle. Clamp the lathe dog on the workpiece so that its tail overhangs the end of the workpiece. If the workpiece is finished, place a split ring of soft material such as brass between the setscrew of the dog and workpiece. Mount the workpiece between the centers. Make sure that the lathe dog tail fits freely in the slot of the faceplate and does not bind.

4.14.4.2 Since the tailstock center is a dead center and does not revolve with the workpiece, it requires lubrication. Apply some high-temperature, extreme pressure grease to the center before the workpiece is set up. The tailstock should be adjusted so that the dead center fits firmly into the center hole of the workpiece but does not bind. The lathe should be stopped at intervals and additional lubrication applied to the dead center to prevent overheating and harm to the center and the workpiece.

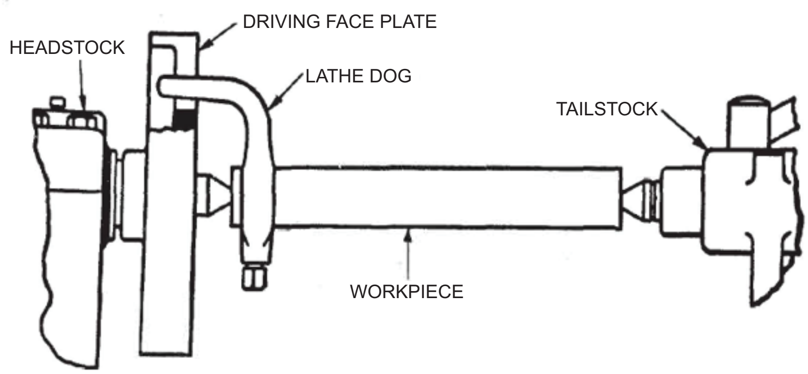


Figure 4-34. Workpiece Correctly Mounted Between Centers in Lathe

4.15 INSTALLING AND REMOVING A LATHE CHUCK ON THE HEADSTOCK.

WARNING

Prior to installing or removing chucks, faceplates, and/or drive plates, ensure that the power to the lathe is turned off. Failure to comply could result in injury to, or death of, personnel or long term health hazard.

CAUTION

As a safety measure, always place a wooden block on the bed ways beneath the chuck, during removal or installation of the chuck. The block prevents the chuck from falling and potentially causing serious injury or damage. Failure to comply could result in damage to, or destruction of, equipment or loss of mission effectiveness.

4.15.1 General. Before installing a lathe chuck, note that headstocks have three types of spindle noses: long tapered key, cam-lock, and threaded. (See Figure 4-35.) Each type requires a unique installation method for the chuck.

4.15.1.1 Long Tapered Key Spindle Nose. (See Figure 4-35.)

4.15.1.1.1 Installation.

a. Clean spindle nose and chuck; ensure both mating surface are free of dirt and debris.

b. Apply a small amount of lubricant to the tapered surface of the spindle nose.

c. Align the chuck with the spindle nose, matching the keyway on the chuck with the key on the spindle nose.

d. Carefully slide the chuck onto the spindle nose, ensuring the tapered surfaces engage smoothly.

e. Rotate the chuck back and forth slightly to ensure proper seating on the tapered surface.

f. Tighten the drawbar to secure the chuck to the spindle nose, following the recommended torque specification.

4.15.1.1.2 Removal.

a. Use a wrench or spanner to loosen the chuck by turning it counterclockwise.

b. Carefully pull the chuck off the spindle nose, taking note of the keyway and key alignment.

c. Once the chuck is removed, remove the key from the spindle nose.

d. Clean the spindle nose and chuck to prevent debris or dirt from accumulating.

e. Store the chuck in a safe and secure location, protected from damage or loss.

4.15.1.2 Cam Lock Spindle Nose. (See Figure 4-35.)

4.15.1.2.1 Installation.

a. Clean spindle nose and chuck; ensure both mating surface are free of dirt and debris.

b. Align the chuck with the spindle nose, matching the registration marks or keyways.

c. Carefully slide the chuck onto the spindle nose until it stops.

d. Rotate the cam lock lever clockwise to engage the chuck with the spindle nose.

e. Tighten the cam lock lever to the recommended torque.

f. Repeat the previous two steps if there are multiple cam lock levers around the circumference of the chuck.

g. Verify the chuck is securely locked in place and the cam lock levers are tightened.

4.15.1.2.2 Removal.

a. Use a wrench or spanner to loosen the cam lock by turning it counterclockwise.

b. Twist the chuck counterclockwise until the cam pins disengage from the spindle nose recesses.

c. Carefully pull the chuck off the spindle nose, taking note of the cam pin alignment.

d. Clean the spindle nose and chuck to prevent debris or dirt from accumulating.

e. Store the chuck in a safe and secure location, protected from damage or loss.

4.15.1.3 Threaded Spindle Nose. (See Figure 4-35.)

4.15.1.3.1 Installation.

a. Clean spindle nose and chuck; ensure both are free of dirt and debris.

b. Apply a small amount of thread lubricant to the spindle nose threads.

c. Align the chuck with the spindle nose, matching the threads.

d. Thread the chuck onto the spindle nose by hand, turning it clockwise.

e. Tighten the chuck to the recommended torque specification using a wrench or spanner.

f. Verify the chuck is securely threaded onto the spindle nose and tightened to the correct torque.

4.15.1.3.2 Removal.

a. Use a wrench or spanner to loosen the chuck by turning it counterclockwise. Be careful not to apply too much force, which can damage the spindle or chuck.

b. Once loose, hand loosen the chuck by turning it counterclockwise.

c. Carefully remove the chuck from the spindle nose, taking note of the thread alignment.

d. Clean the spindle nose and chuck to prevent debris or dirt from accumulating.

e. Store the chuck in a safe and secure location, protected from damage or loss.

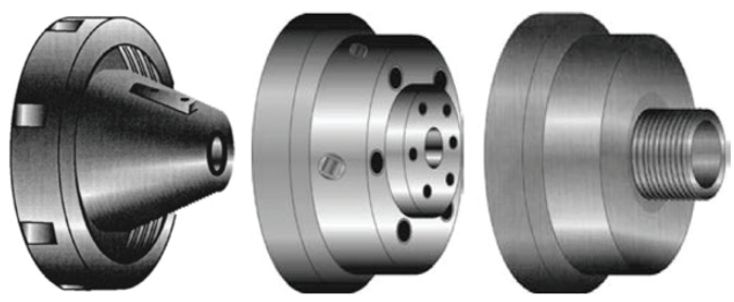


Figure 4-35. Three Types of Spindle Noses

4.16 MOUNTING WORKPIECES IN CHUCKS.

WARNING

Always keep one hand on the chuck key when removing or installing a workpiece on a chuck at all times, never leave the chuck key in the chuck unattended. If unremoved, the key can be flung out at high speed, causing severe injury to the operator or other shop personnel. Failure to comply could result in injury to, or death of, personnel or long term health hazard.

Mounting and aligning workpieces in the 4-Jaw independent chuck. The following paragraphs briefly explain the three general methods to mounting and aligning a workpiece in the 4-Jaw independent chuck.

4.16.1 Rough Centering.

a. Place the workpiece between the chuck jaws and adjust them to an approximate centered position using the concentric rings on the chuck face.

b. Start the lathe and hold a piece of chalk lightly against the revolving workpiece until a mark shows on the workpiece.

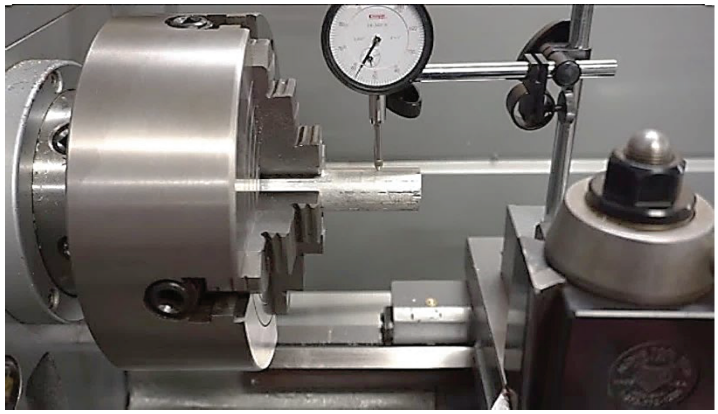
c. Readjust the chuck by loosening the jaw or jaws opposite the chalk mark and tightening the jaw or jaws on the chalk mark side.

d. Repeat the above process until the workpiece is satisfactorily aligned.

4.16.2 Centering a Workpiece with a Smooth Surface.

a. Using a dial test indicator, place the point of the indicator against the outside or inside diameter of the workpiece. (See Figure 4-36.)

b. Revolve the workpiece slowly by hand and notice any deviations on the dial. This method will indicate any inaccuracy of the centering in thousandths of an inch.



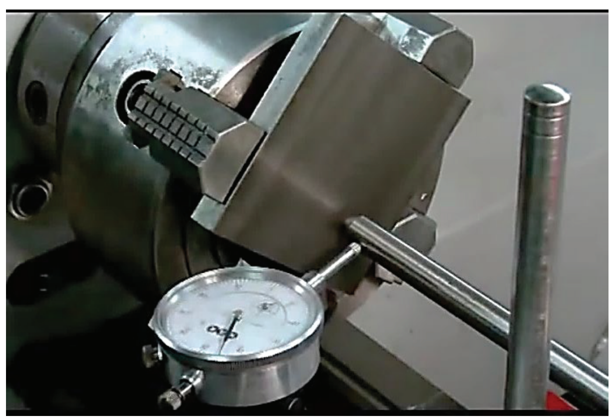
TO-34-1-10-070

Figure 4-36. Centering Work with a Dial Test Indicator

4.16.3 Centering an Irregular Shaped Workpiece. There are various methods of aligning an irregular workpiece on an independent chuck. One such method is locally manufacturing a straight, hardened steel bar, grounded with a 60 degree point on one end; the opposite end can be mounted into the drill chuck of the tailstock spindle and guided into the center punched mark on the workpiece.

a. Using a dial test indicator, place the point of the indicator against the surface of the steel bar.

b. Revolve the workpiece and adjust the adjust the jaws until the part is aligned to within 0.001 inch. (See Figure 4-37.)



TO-34-1-10-071

Figure 4-37. Centering an Irregular Shaped Workpiece

4.16.4 Mounting a Workpiece in a 3-Jaw Universal Chuck and Drill Chuck.

a. To mount a workpiece, place it in the 3-Jaw universal chuck, which is self-centering due to its synchronized jaw movement.

b. Tighten the jaws evenly with gentle to moderate pressure using the chuck key, allowing the workpiece to center itself.

c. For a drill chuck, attach it to the headstock via the Morse taper or other mounting mechanism, then insert and secure the drill bit according to the manufacturer's instructions.

d. Refer to the machine's manual for specific guidance and safety precautions.

4.17 MOUNTING WORKPIECES TO FACEPLATES.

4.17.1 Installing and Aligning the Faceplate. When screwing the faceplate on the headstock spindle, ensure that the threads are clean and lubricated prior to installation. The accuracy of the bearing surface of the faceplate is extremely important. Any unevenness of this surface should be removed by taking a facing cut. The workpiece is attached to the faceplate by bolting angle plates and brackets to secure the workpiece. Care should be exercised when clamping the piece so that neither the piece nor the faceplate will be sprung. To eliminate any spring or vibration caused by having the piece offset on the faceplate, balance weights may be used. Paper placed between the faceplate and the piece will help reduce possible slippage caused by slight unevenness on the workpiece or faceplate. Figure 4-38 illustrates a typical setup using the faceplate.

4.17.2 Checking Centering of Workpiece on Faceplate. The alignment or centering of a workpiece on faceplates should be checked by using a dial test indicator as described in Paragraph 4.16.2 and Paragraph 4.16.3.

4.17.3 Removing the Faceplate from the Lathe. Loosen the screws that hold the faceplate to the lathe spindle by turning them counterclockwise. Completely remove the screws from the faceplate. Carefully pull the faceplate away from the lathe spindle. Remove any remaining parts, such as drive pins or keys, from the faceplate or spindle.

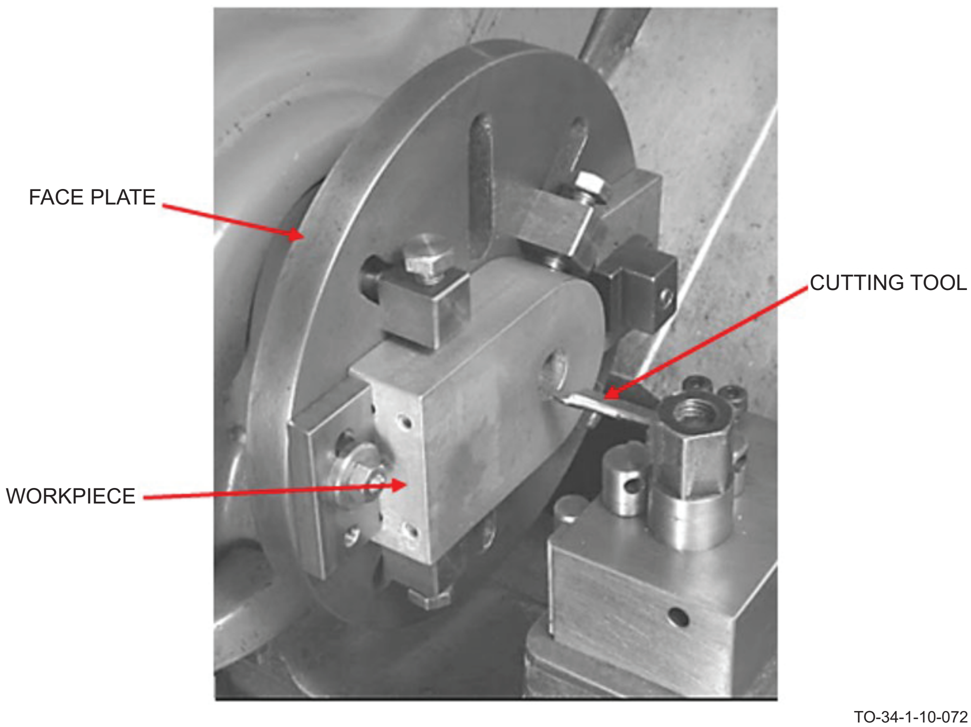


Figure 4-38. Typical Faceplate Setup

4.18 MOUNTING WORKPIECES ON MANDRELS.

a. Ensure that the mandrel selected is of the proper size for the workpiece to be mounted.

b. The bore of the workpiece and the mandrel must be free of burrs and both surfaces must be thoroughly cleaned.

c. Lubricate both the workpiece and the mandrel and press the mandrel into the workpiece.

d. Set up the mandrel between centers as though it were part of the workpiece, as described in Paragraph 4.13. Figure 4-39 shows a pulley supported in the lathe by a mandrel. Note the attachment of the lathe dog to the flat at the end of the mandrel, not to the machined taper surface.

4.19 GENERAL LATHE OPERATIONS.

WARNING

Prior to operating the lathe, ensure that safety glasses is worn, remove any loose clothing, jewelry, and long hair secured to prevent entanglement with moving parts. Failure to comply could result in injury to, or death of, personnel or long term health hazards.

4.19.1 General. To achieve good results on the lathe, technicians must understand the variables of speed, feed, cutting oil, and material properties. Each material and operation requires specific settings for speed, feed, depth of cut, and cutting oil to prevent damage to the lathe and tools, and to ensure a successful outcome. With experience, machinists can refine these parameters for individual operations to optimize results.

4.19.2 Lathe Speeds and Feeds. Optimal feeds and speeds for a lathe operation depend on several key factors, including material type, tool type, workpiece dimensions, cut type (rough or finish), cutting oil, and lathe condition.

4.19.2.1 Cutting Speed. The cutting speed of a cutter bit is defined as the number of feet of workpiece surface, measured at the circumference, that pass the cutter bit in 1 minute. The cutting speed expressed in FPM, must not be confused with the spindle speed of the lathe which is expressed in RPM. To obtain uniform cutting speed, the lathe spindle must be revolved faster for work-pieces of small diameter and slower for workpieces of large diameter.

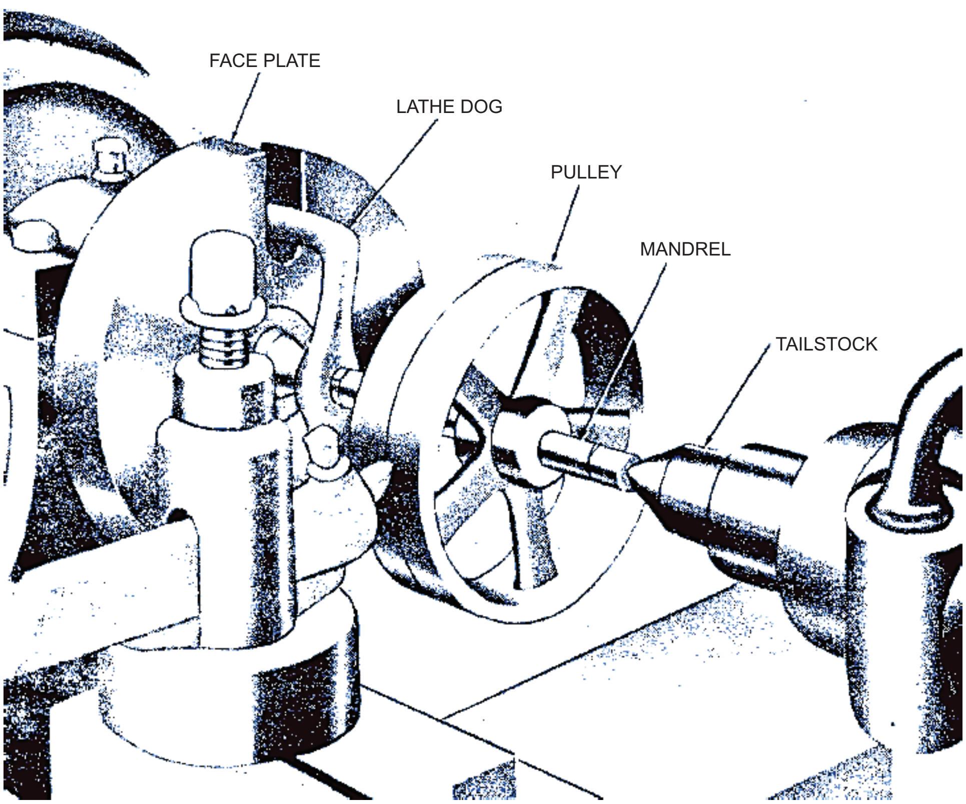


Figure 4-39. Pulley Mounted in Lathe on a Mandrel

4.19.2.1.1 Cutting speed is the rate at which the workpiece surface passes the cutter bit, measured in FPM. To maintain a constant cutting speed, the lathe's spindle speed RPM must be adjusted based on workpiece diameter; faster for smaller diameters and slower for larger diameters.

4.19.2.1.2 Cutting speed depends on material hardness, cutter bit material, feed, and depth of cut. Refer to the table below for recommended speeds for straight turning and threading. Start with these speeds and adjust as needed. Carbon-steel tools require reduced speeds due to heat sensitivity, while carbide-tipped tools can withstand higher speeds. To maintain optimal conditions, reduce cutting speed proportionally when increasing feed or depth of cut to prevent overheating and excessive tool wear.

Table 4-2. Lathe Cutting Speeds for Straight Turning and Threading

Material	Straight Turning (FPM)[1]	Threading (FPM)[1]
Aluminum	200-400	100-200
Aluminum Bronze	100-200	50-100
Brass	150-300	75-150
Cast Iron	80-150	40-80
Copper	100-200	50-100
Mild Steel	80-150	40-80
Medium Carbon Steel	60-120	30-60
Titanium	100-200	50-100
Tool Steel	50-100	25-50
Stainless Steel	50-100	25-50

[1] The speeds are based on the use of carbide insert cutter bits. The speed may be reduced 20–50 percent if high speed cutter bits are used.

4.19.2.1.3 To determine the rotational speed necessary to produce a given cutting speed, it is necessary to know the diameter of the workpiece to be cut. To calculate the spindle speed, knowing the diameter of the workpiece, use the following formula:

$$\text{RPM} = \frac{12\ \text{FPM}}{\pi D}$$

WHERE RPM = SPINDLE SPEED (IN REVOLUTIONS PER MINUTE);
FPM = CUTTING SPEED (IN FT PER MINUTE)
π = 3.1416;
D = DIAMETER (IN INCHES) OF WORKPIECE.

4.19.2.1.4 To determine spindle speed for drills and rotating cutters, use the same formula, substituting the cutter's diameter for the workpiece diameter. Alternatively, refer to Table 4-3 for a quick lookup of spindle speeds for common cutting speeds and workpiece diameters.

4.19.2.1.5 When selecting cutting speed, consider factors like cutting oil, workpiece length and diameter, and lathe condition. Excessive vibration can occur with long, small-diameter workpieces or poorly maintained lathes, resulting in a poor finish. To mitigate this, reduce the cutting speed to minimize vibration and ensure a smooth finish.

NOTE

Consult the lathe's technical manual on spindle speed setting instructions. If the desired speed is not available, choose the nearest slower speed. A machinist should be able to identify optimal speed ranges, recognizing when a speed is too slow or too fast, to ensure efficient and effective lathe operation.

Table 4-3. Lathe Rotational Spindle Speeds

Cutting Speed (FPM)	30	40	50	60	70	80	90	100	120	140	160	180	200	250
Workpiece Diameter (Inch)	Spindle speed (RPM)													
1/4	381	508	635	762	889	1,016	1,143	1,270	1,524	1,778	2,032	2,286	2,540	3,180
1/2	191	254	318	381	445	508	572	635	762	889	1,016	1,143	1,270	1,590
3/4	127	169	212	254	298	341	385	429	512	595	679	762	846	1,060
1.00	95	127	159	190	223	254	286	318	381	445	508	572	635	795
1.50	63	84	106	127	149	169	190	212	254	298	341	385	429	535
2.00	48	63	79	95	112	127	143	159	190	223	254	286	318	398
2.50	38	50	63	76	90	101	114	127	152	177	203	228	254	318
3.00	32	42	53	63	75	84	95	106	127	149	169	190	212	265
3.50	27	36	45	54	64	72	81	90	108	126	144	162	180	225
4.00	24	31	39	47	56	63	71	79	95	112	127	143	159	199
4.50	21	28	35	42	50	56	63	71	85	100	114	129	143	179
5.00	19	25	31	38	45	51	57	63	76	90	101	114	127	159
5.50	17	23	29	35	41	46	52	58	70	82	94	106	118	147
6.00	16	21	26	32	38	42	48	53	64	75	86	97	109	136
6.50	15	20	25	30	35	39	44	49	59	69	79	90	101	127
7.00	14	18	23	28	33	37	41	46	55	65	75	85	95	119
7.50	13	17	21	26	30	34	38	43	51	60	59	79	89	112
8.00	12	16	20	24	28	32	36	40	48	56	65	74	83	104
8.50	11	15	19	23	27	30	34	38	45	53	62	71	80	100
9.00	11	14	18	22	25	29	32	36	43	50	59	68	77	96
9.50	10	13	17	20	24	27	30	34	41	48	56	65	74	92
10.00	9	12	16	19	23	26	29	32	39	45	53	62	71	89
10.50	9	12	15	18	21	24	27	30	36	43	50	59	68	85
11.00	8	11	14	17	20	23	26	29	35	41	48	56	64	82
11.50	8	10	13	16	19	22	25	28	33	39	45	53	62	79
12.00	8	10	13	15	18	20	23	26	32	37	43	50	59	76

4.19.3 Feed. The **Feed** refers to the distance the cutting tool advances along the workpiece with each revolution. A heavier feed (0.010-0.020 inch per revolution) is typically used for rough cuts to quickly remove material, while a lighter feed (0.003-0.010 inch per revolution) is used for finish cuts to achieve a smooth surface finish and precise dimensions. The feed rate affects the cutting tool's load, tool life, and surface finish, and must be balanced with depth of cut and cutting speed to optimize machining performance and prevent damage to the workpiece or tool.

4.19.4 Depth of Cut. The depth of cut refers to the amount of material removed from the workpiece with each pass of the cutting tool. For rough cuts, a deeper cut (up to 0.100 inch or more) can be used to quickly remove excess material, while for finish cuts, a shallower cut (typically 0.005-0.020 inch) is used to achieve a smooth surface finish and precise dimensions. The depth of cut affects the cutting tool's load, tool life, and surface finish, and must be balanced with feed rate and cutting speed to optimize machining performance.

4.19.5 Cutting Oil. Cutting oil is a lubricant used in machining to reduce friction and heat between the tool and workpiece. It improves tool life, surface finish, and chip removal, while preventing tool wear and galling. Cutting oil is essential for high-speed machining, where heat generation is high.

4.19.5.1 Cutting Oil Use. Cutting oil is used in various machining operations, including turning, milling, and drilling. The type of cutting oil used depends on the material being machined, machining operation, and desired surface finish. Common cutting oil types include mineral oil, synthetic, and semi-synthetic.

4.19.5.2 Types of Cutting Oils. Common cutting oil types include mineral oil (general-purpose), synthetic oil (high-speed, high temperature), and semi-synthetic oil (balanced cost and performance). Specialty oils, such as vegetable-based and water-miscible, offer environmentally friendly and fire-resistant alternatives for specific applications. The table below lists cutting oils for specific lathe operations for different materials to be machined.

Table 4-4. Recommended Cutting Oils for Lathe Operations

Material	Heavy Cutting	Light Cutting	Threading
Aluminum	Water soluble coolant (5-10 percent concentration) or light cutting oil (e.g., mineral oil)	Air or light oil (e.g., mineral oil)	Water soluble coolant (5-10 percent concentration) or light cutting oil (e.g., mineral oil)
Aluminum Bronze Brass	Sulphurized oil or heavy duty cutting oil	Chlorine free cutting oil or sulphurized oil	Sulphurized oil or heavy duty cutting oil
Cast Iron	Water soluble coolant (5-10 percent concentration) or light cutting oil (e.g., mineral oil)	Air or light oil (e.g., mineral oil)	Water soluble coolant (5-10 percent concentration) or light cutting oil (e.g., mineral oil)
Copper	Sulphurized oil or heavy duty cutting oil	Chlorine free cutting oil or sulphurized oil	Sulphurized oil or heavy duty cutting oil
Mild Steel	Soluble oil (10-20 percent concentration) or heavy duty cutting oil	Chlorine free cutting oil or soluble oil (5-10 percent concentration)	Soluble oil (10-20 percent concentration) or heavy duty cutting oil
Medium Carbon Steel	Heavy duty cutting oil or sulpharized oil	Chlorine free cutting oil or sulphurized oil	Heavy duty cutting oil or sulpharized oil
Titanium	High-pressure, flood coolant specifically for titanium, or a high quality, chlorine-free cutting oil	Same as heavy cutting but at a lower rate	High-pressure, flood coolant specifically for titanium, or a high quality, chlorine-free cutting oil
Tool Steel Stainless Steel	Sulphurized oil or heavy duty cutting oil	Chlorine free cutting oil or sulphurized oil	Sulphurized oil or heavy duty cutting oil

4.19.6 Facing. The facing operation refers to the removal of material from the end face of a workpiece to create a flat surface, typically to establish a reference point or achieve a precise finish. This process is often the first step in machining and helps ensure accurate and precise results.

4.19.6.1 General Instructions for Facing a Material.

4.19.6.1.1 Mounting the Workpiece.

a. Securely mount the workpiece between the headstock and tailstock, or in a chuck.

b. Ensure the workpiece is centered.

4.19.6.1.2 Prepare and Position the Tool Bit.

a. Select a properly ground RH facing tool bit.

b. Install the tool bit in the tool holder.

c. Position the cutting edge at the same height as the workpiece centerline. (See Figure 4-40.)

d. With the lathe off, bring the tool bit close to the outer edge of the workpiece.

4.19.6.1.3 Start Facing Operation.

a. Start the lathe.

b. Engage the cross slide to take a small cut.

c. Move the tool bit from the outer edge towards the center of the workpiece. (See Figure 4-40.)

4.19.6.1.4 Continue Facing.

a. Take successive small cuts until the desired face width is achieved.

b. Maintain a consistent feed rate.

c. Use coolant if applicable.

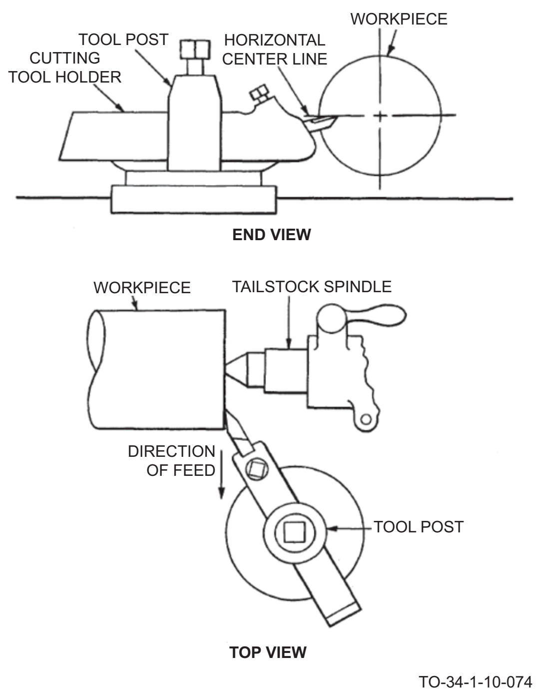


Figure 4-40. Position of Cutter Bit for Facing

4.19.7 Straight Turning. Straight turning is a fundamental machining process used to reduce the diameter of a workpiece along its length, creating a cylindrical shape. This involves feeding a cutting tool parallel to the workpieces axis of rotation, removing material as it moves along. The depth of cut determines the amount of material removed with each pass, while the feed rate controls the tool’s travel speed along the workpiece. Straight turning is essential for producing various cylindrical components with consistent diameters and smooth surface finishes.

4.19.7.1 General Instructions for Straight Turning a Material.

4.19.7.1.1 Mount the Workpiece.

a. Securely mount the workpiece between centers (for longer pieces) or in a chuck (for shorter pieces).

b. Ensure the workpiece is centered and rotates smoothly without wobble.

4.19.7.1.2 Prepare and Position the Tool Bit.

a. Select a properly ground RH turning tool bit.

b. Install the tool bit in the tool holder.

c. Align the cutting edge 5 degrees from the workpiece centerline. (See Figure 4-41.)

d. Position the tool bit slightly past the right end of the workpiece.

4.19.7.1.3 Start Turning.

a. Start the lathe.

b. Engage the carriage feed to move the tool bit along the workpiece length.

c. Take small cuts initially.

4.19.7.1.4 Adjust Cutting Parameters. Adjust the depth of cut feed rate for optimal material removal and surface finish.

4.19.7.1.5 Continue Turning.

a. Make multiple passes along the workpiece length, gradually reducing the diameter.

b. Ensure consistent diameter and a smooth surface along the entire length.

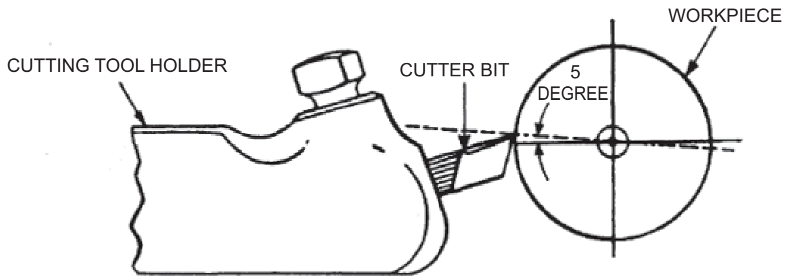


Figure 4-41. Position of Cutter Bit for Straight Turning

4.19.8 Shoulder Turning. The process of shoulder turning is similar to straight turning. The major difference between the two is that shoulder turning creates a stepped diameter **shoulder** on a workpiece. Whereas, straight turning aims to machine a uniform diameter along the entire workpieces length. Shoulder turning involves feeding a cutting tool perpendicularly into a rotating workpiece to precisely remove material, resulting in a specific diameter and surface finish for the shoulder. This operation is crucial for creating accurate seating, locating, and assembly features on components.

4.19.8.1 General Instructions for Shoulder Turning Material.

4.19.8.1.1 Workpiece and Tool Setup.

a. Securely mount the workpiece in a chuck (3-Jaw for speed, 4-jaw for precision), ensuring sufficient protrusion for the shoulder. (See Figure 4-42.)

b. Set the tool post perpendicular (90 degrees) to the lathe axis for square cuts.

c. Choose and prepare the tool bit:

- Material: Select HSS or carbide based on workpiece hardness.
- Grind: Sharpen the cutting edge. Grind perpendicular for squared shoulders, or at the desired fillet radius for rounded shoulders.
- Height: Position the tool bit's cutting edge precisely at the workpiece centerline to avoid taper.

4.19.8.1.2 Shoulder Turning Operation.

a. Clearly mark the shoulder location on the workpiece using chalk or a marker and hermaphrodite calipers.

b. Begin with a low spindle speed and gradually increase for efficient material removal.

c. Apply appropriate cutting fluid.

d. Cut:

- Initial shoulder (Facing Cut): Use a facing cut perpendicular to the workpiece axis to establish the shoulder face.
- Refine diameter (Longitudinal Cut): Approach existing shoulders longitudinally to finetune the diameter.

4.19.8.1.3 Incremental Cuts. Take small, incremental cuts to prevent chatter and ensure accuracy.

4.19.8.1.4 Measure Frequently. Use a micrometer or caliper to verify shoulder diameter against tolerances.

4.19.8.2 Specialized Shoulder Finishes.

4.19.8.2.1 Filleted Shoulder.

a. Use a form tool with the desired fillet radius.

b. Approach slowly and maintain a consistent feed rate for a smooth fillet. (See Figure 4-43.)

4.19.8.2.2 Squared Shoulder.

a. Ensure a sharp, perpendicular cutting edge.

b. Use a very slow, controlled feed rate for the final cut.

c. Take a light finishing cut for a smooth, square surface.

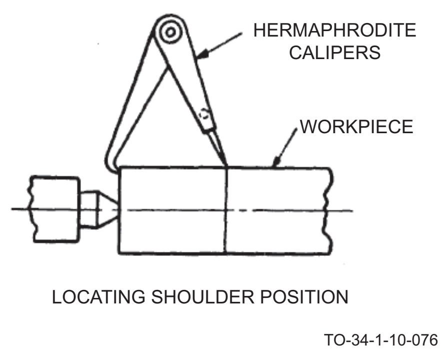


Figure 4-42. Locating Shoulder Position

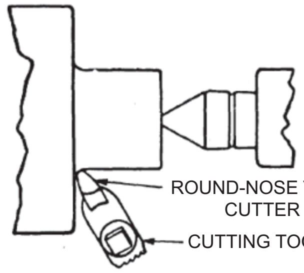


Figure 4-43. Fillet Shoulder Turning

4.19.9 Parting. The parting operation separates a workpiece on the lathe using a parting tool. (See Figure 4-44.) This tool requires careful grinding with side clearance to prevent binding. Cutting oil is crucial during parting due to increased heat generation from the tool's large cutting surface.

4.19.9.1 General Instructions for Parting a Material.

4.19.9.1.1 Workpiece Mounting and Preparation.

a. Mount the workpiece firmly in the chuck, ensuring ample clearance for the parting tool.

b. Clearly mark the desired parting line on the workpiece using a toolmaker's marker and calipers.

4.19.9.1.2 Parting Tool Selection and Positioning.

a. Select a parting tool with a width appropriate for the workpiece diameter and material.

b. Ensure the tool has adequate side clearance to prevent binding during the cut.

c. Align the tool bit perpendicular to the workpiece and at the same height as its centerline.

d. If necessary, use a steady rest to support long or slender workpieces and prevent chatter or deflection.

4.19.9.1.3 Parting Operation.

a. Begin with a slow spindle speed, especially for larger diameters or harder materials.

b. Generously apply cutting oil to the cutting zone to lubricate, cool, and aid chip removal.

c. Start Cut:

   (1) Engage the tool slowly and carefully to establish the initial groove.

   (2) Use a light feed rate to prevent tool breakage or chatter.

d. Continue feeding the tool into the workpiece at a steady rate. Avoid stopping or varying the feed, which can lead to tool breakage.

e. Regularly check the depth of the cut and adjust the tool position as needed.

4.19.9.1.4 Final Separation. As the cut nears completion, reduce the feed rate to prevent the workpiece from dropping abruptly. Be prepared to support the separated piece.

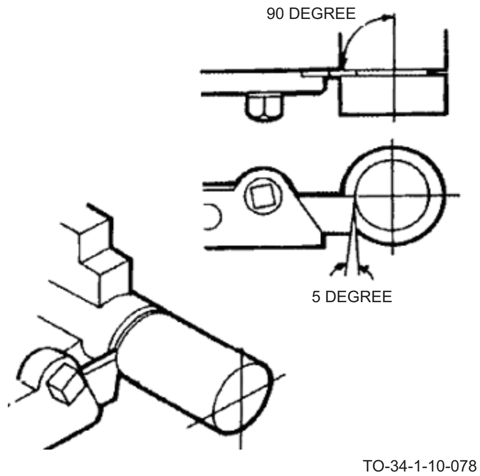


Figure 4-44. Parting Operation

4.19.10 Taper Turning. Taper turning involves creating a gradual diameter change along a cylindrical workpiece, essentially forming a cone-like shape. Taper turning essentially produce precise angles essential for fitting parts, aligning components, and ensuring functionality in various mechanical assemblies, from toolholders to valve components. This can be achieved by adjusting the compound rest, using a taper attachment for enhanced accuracy, or programming the desired angle directly into a CNC lathe. Regardless of the method, precision is paramount in taper turning, as even slight angular deviations can compromise the fit and function of machined parts.

4.19.10.1 Determining the Taper. Determining the taper on a part is crucial to ensure accurate machining and proper fit for components.

4.19.10.2 Methods to Determine Taper.

4.19.10.2.1 Direct Measurement.

4.19.10.2.1.1 Tools. Calipers or micrometers are ideal for measuring the diameters at different points along the taper.

4.19.10.2.1.2 Process.

a. Measure the larger diameter (D) at one end of the taper.

b. Measure the smaller diameter (d) at the other end of the taper.

c. Measure the Distance (L) between the points where you took the diameter measurements (this is the length of the taper).

4.19.10.2.2 Using Taper Gauges.

- Taper gauges are designed to quickly check standard tapers.
- The gauge is inserted into the tapered feature, and markings or a digital readout indicate the taper size.

4.19.10.3 Measurements. Once you have the necessary measurements, use the following formula to calculate the TPI or Taper Per Foot (TPF). The table below is provided to facilitate taper computations.

$$\text{TAPER PER INCH} = \frac{D-d}{L}$$

WHERE:

D = LARGER DIAMETER

d = SMALLER DIAMETER

L = LENGTH OF THE TAPER (IN INCHES)

$$\text{TPF (TAPER PER FOOT)} = \frac{D-d}{L} \times 12$$

THIS SIMPLY CONVERTS TPI TO TPF BY MULTIPLYING BY 12
(THE NUMBER OF INCHES IN A FOOT).

Table 4-5. Taper Sizes and Angles

	Difference in Diameters for Workpiece Lengths			
Taper (IPF)	6 inches	12 inches	18 inches	24 inches
1/16	0.03125	0.0625	0.09375	0.125
3/32	0.04688	0.09375	0.14063	0.1875
1/8	0.0625	0.125	0.1875	0.250
5/32	0.07813	0.15625	0.23438	0.3125
3/16	0.09375	0.1875	0.28125	0.375
7/32	0.10938	0.21875	0.32813	0.4375
1/4	0.125	0.250	0.375	0.500
9/32	0.14063	0.28125	0.42188	0.5625
5/16	0.15625	0.3125	0.46875	0.625
11/32	0.17188	0.34375	0.51563	0.6875
3/8	0.1875	0.375	0.5625	0.750
13/32	0.20313	0.40625	0.60938	0.8125
7/16	0.28175	0.4375	0.65625	0.875
15/32	0.23438	0.46875	0.70313	0.9375
1/2	0.250	0.500	0.750	1.00
17/32	0.26563	0.53125	0.79688	1.0625
9/16	0.28125	0.5625	0.84375	1.125
19/32	0.29688	0.59375	0.89063	1.1875
5/8	0.3125	0.625	0.9375	1.250
21/32	0.32813	0.65625	0.98438	1.3125
11/16	0.34375	0.6875	1.03125	1.375
23/32	0.35938	0.71875	1.07813	1.4375
3/4	0.375	0.750	1.125	1.500
13/16	0.40625	0.8125	1.21875	1.625
7/8	0.4375	0.875	1.3125	1.750
15/16	0.46875	0.9375	1.40625	1.875
1	0.500	1.00	1.500	2.00
1-1/8	0.5625	1.125	1.6875	2.25
1-1/4	0.625	1.250	1.875	2.500

4.19.11 Taper Turning Using the Tailstock Setover Method. (See Figure 4-45.)

4.19.11.1 Calculate Tailstock Setover

a. Determine the desired TPF or taper per inch from the part drawing or specifications.

b. Calculate the required tailstock setover using the formula:

$$TSO = \frac{TPF \times L}{24}$$

WHERE L IS THE LENGTH OF THE TAPER IN INCHES.

OR

$$TSO = \frac{TPF \times L}{2}$$

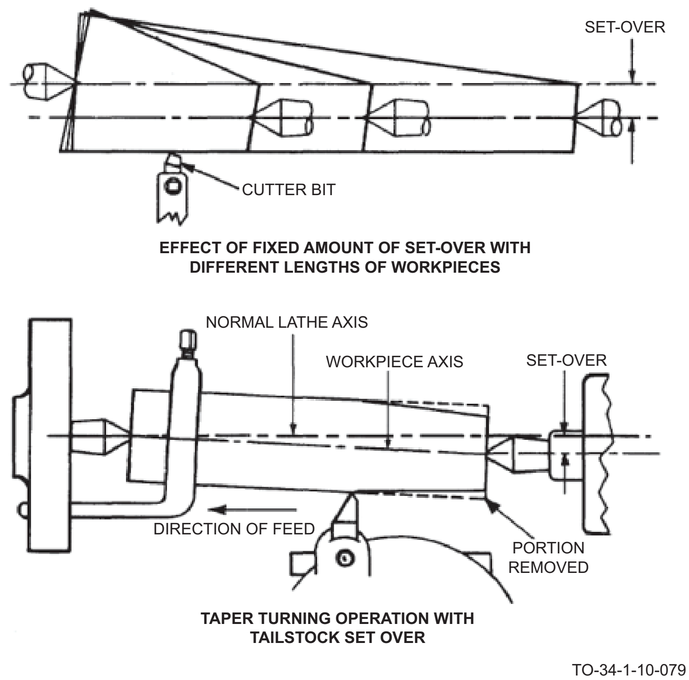


Figure 4-45. Taper Turning with Tailstock Setover

4.19.11.2 Prepare the Lathe.

a. Mount the workpiece securely between the lathe chuck and tailstock center. Ensure it runs true.

b. Install the appropriate cutting tool for taper turning.

4.19.11.3 Set the Tailstock Setover.

a. Loosen the tailstock clamping bolts that secure it to the lathe bed.

b. Using the graduated markings on the tailstock base, carefully offset the tailstock towards the operator (for external tapers) or away from the operator (for internal tapers) by the calculated tailstock setover value.

c. Tighten the tailstock clamping bolts securely.

4.19.11.4 Turn the Taper.

a. Set the lathe to the appropriate cutting speed and feed rate for the material and tool being used.

b. Start the lathe and engage the feed mechanism, moving the cutting tool along the workpiece.

c. The tailstock setover will automatically create the desired taper as the tool travels. (See Figure 4-45.)

4.19.11.5 Verify the Taper.

a. After making a test cut, stop the lathe and measure the taper using a micrometer or taper gauge.

b. If adjustments are needed, slightly loosen the tailstock clamping bolts and fine-tune the setover.

c. Re-tighten the bolts and make another test cut.

d. Repeat until the desired taper is achieved.

4.19.12 Taper Turning with Compound Rest.

4.19.12.1 Determine Taper Specifications.

a. Identify the required taper per unit of length (e.g., inches per foot) or the included angle of the taper.

b. Calculate the required taper angle using trigonometry or a taper chart if necessary.

4.19.12.2 Set the Compound Rest Angle.

a. Loosen the compound rest swivel lock.

b. For taper per unit length: Use the following formula to determine the angle (θ) to set the compound rest:

$$\tan(\theta) = \frac{\text{TAPER PER UNIT LENGTH}}{2}$$

EXAMPLE: FOR A TAPER OF 0.5 INCHES PER FOOT:

$$\tan(\theta) = \frac{0.5}{12} = 0.041667$$

$$\theta = \tan^{-1} 0.041667 = 2.38 \text{ DEGREE}$$

c. For included taper angle: Set the compound rest directly to half of the included taper angle.

d. Use the compound rest's graduated scale to set the calculated angle accurately.

e. Tighten the compound rest swivel lock.

4.19.12.3 Position the Tooling.

a. Install the appropriate cutting tool in the tool holder.

b. Position the tool holder in the tool post, ensuring the cutting edge is on centerline with the workpiece.

4.19.12.4 Set the Starting Diameter.

a. Move the tool to the end of the workpiece where the taper will begin.

b. Using the cross slide, turn the workpiece to the desired starting diameter.

4.19.12.5 Engage the Feed.

a. Disengage the cross slide feed.

b. Engage the compound rest feed.

4.19.12.6 Perform the Taper Cut.

a. Set the lathe to a suitable cutting speed and feed rate for the material and tool being used.

b. Slowly and steadily rotate the compound rest handle to feed the tool along the workpiece, creating the taper.

c. Take light cuts, especially when approaching the final diameter, to avoid chatter and achieve a smooth finish.

4.19.12.7 Measure and Adjust.

a. Periodically stop the lathe and measure the diameter of the taper using a micrometer or calipers.

b. Adjust the cross slide slightly, if necessary, to correct the taper diameter as you progress.

4.19.12.8 Complete the Taper.

a. Continue feeding the tool until the desired taper length is achieved.

b. Disengage the compound rest feed.

4.19.12.9 Inspect the Taper.

a. Carefully inspect the taper for accuracy, surface finish, and any signs of tool chatter.

b. Make any necessary adjustments and repeat the cutting process if needed.

4.19.13 Taper Turning with a Taper Attachment. (See Figure 4-46.)

4.19.13.1 Determine Taper Specifications. Identify the required taper per unit of length (e.g., inches per foot) or the included angle of the taper.

4.19.13.2 Calculate Taper Attachment Setting.

a. Using taper per unit length: Most taper attachments have graduations indicating TPF or per unit length. Set the attachment directly to the desired value.

b. Using included angle: Some attachments use degrees. If that feature exists, set it to half the included angle of the taper.

4.19.13.3 Disengage Cross Slide Feed. Ensure the cross slide feed is disengaged to prevent interference with the taper attachment's movement.

4.19.13.4 Mount and Align Taper Attachment.

a. Securely mount the taper attachment to the lathe's ways according to the manufacturer's instructions.

b. Align the taper attachment guide bar parallel to the lathe's axis. This is crucial for accurate tapers. Many attachments have adjustable guides to achieve this alignment.

4.19.13.5 Connect to Cross Slide. Connect the taper attachment to the cross slide using the provided linkage. This allows the attachment to control the cross slide movement.

4.19.13.6 Position the Tooling.

a. Install the appropriate cutting tool in the tool holder.

b. Position the tool holder in the tool post, ensuring the cutting edge is on centerline with the workpiece.

4.19.13.7 Set Starting Diameter.

a. Move the tool to the end of the workpiece where the taper will begin.

b. Using the cross slide, turn the workpiece to the desired starting diameter.

4.19.13.8 Engage the Longitudinal Feed. Engage the longitudinal feed (carriage feed) to move the tool along the workpiece.

4.19.13.9 Perform the Taper Cut.

a. Set the lathe to a suitable cutting speed and feed rate for the material and tool.

b. As the carriage moves longitudinally, the taper attachment will automatically control the cross slide, creating the taper.

c. Take light cuts, especially when approaching the final diameter, to avoid chatter and achieve a smooth finish.

4.19.13.10 Measure and Adjust.

a. Periodically stop the lathe and measure the diameter of the taper using a micrometer or calipers.

b. If adjustments are needed, adjust the taper attachment setting slightly, not the cross slide. This fine-tunes the taper angle.

4.19.13.11 Complete the Taper.

a. Continue feeding the tool until the desired taper length is achieved.

b. Disengage the longitudinal feed.

4.19.13.12 Inspect the Taper.

a. Carefully inspect the taper for accuracy, surface finish, and any signs of tool chatter.

b. If necessary, make minor adjustments to the taper attachment and repeat the cutting process.

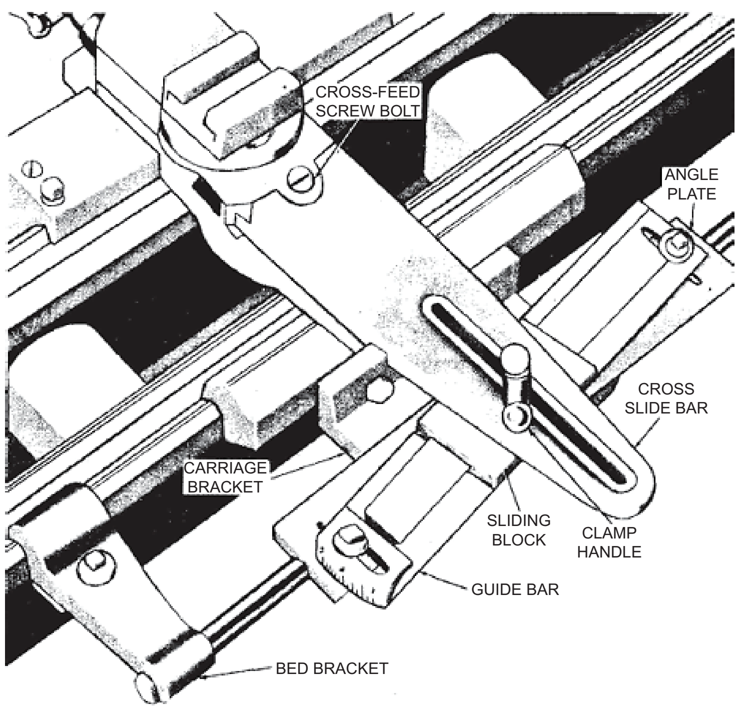


Figure 4-46. Typical Lather Taper Attachment

4.19.14 Screw Thread Cutting. A screw thread is a helical projection of uniform section on the internal or external surface of a cylinder or cone. Threads may be formed on the lathe by use of taps and dies or by cutting them with the thread cutting mechanisms of the lathe. Cutting screw threads is among the most common of lathe operations. Before attempting such operations, however, the operator should have some familiarity with the fundamental principles of threads and the types generally in use.

4.19.15 Screw Thread Terminology. (See Figure 4-47.) The common terms and definitions below are used in calculating screw threads and will be used in discussing threads and thread cutting.

4.19.15.1 External or Male Thread. A thread on the outside of a cylinder or cone.

4.19.15.2 Internal or Female Thread. A thread on the inside of a hollow cylinder or bore.

4.19.15.3 Pitch. The distance from a given point on one thread to a similar point on a thread next to it, measured parallel to the axis of the cylinder. The pitch in inches is equal to one divided by the number of threads per inch.

4.19.15.4 Lead. The distance a screw thread advances axially in one complete revolution. On a single-thread screw, the lead is equal to the pitch. On a double-thread screw the lead is equal to twice the pitch, and on a triple-thread screw, the lead is equal to three times the pitch.

4.19.15.5 Crest (also called **flat**). The top or outer surface of the thread joining the two sides.

4.19.15.6 Root. The bottom or inner surface joining the sides of two adjacent threads.

4.19.15.7 Side. The side of a thread 1s the surface which connects the crest and the root.

4.19.15.8 Thread Angle. The angle included between the sides of ad- jacent threads, measured in an axial plane. For most Y- threads, the angle is fixed at 60 degrees.

4.19.15.9 Depth. The depth of a thread is the distance between the crest and root of a thread, measured perpendicular to the axis.

4.19.15.10 Major Diameter. The major diameter is the largest diameter of a screw thread.

4.19.15.11 Minor Diameter. The minor diameter is the smallest diameter of a screw thread.

4.19.15.12 Pitch Diameter. The pitch diameter is the diameter of an imaginary cylinder, the surface of which would pass through the threads at such points to equalize the widths of the threads and spaces cut by the surface of the cylinder.

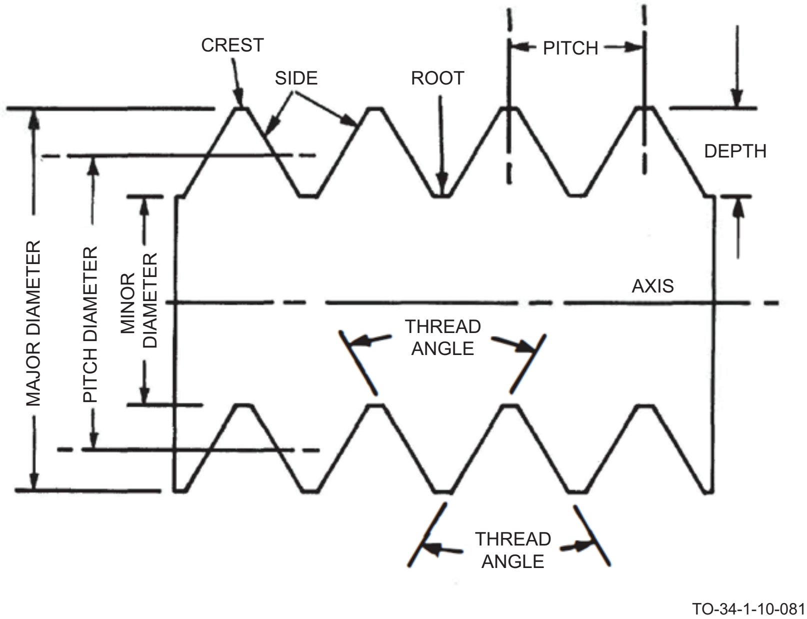


Figure 4-47. Terms Applied to Screw Threads

4.19.16 Screw Thread Forms. Understanding different thread forms is essential when working with mechanical systems, from basic assembly to complex repairs. Threads are the helical ridges that convert torque into linear force, enabling us to fasten objects together or transmit power. Here's a breakdown of several common thread forms and their unique characteristics:

4.19.16.1 Unified Screw Threads. (See Figure 4-48.)

a. Dominant standard for fastening in the United States and many other countries.
b. 60 degree thread angle with rounded root for strength and slightly flattened crest for load distribution.
c. Three main series:
   - Unified National Coarse (UNC): General purpose, good strength, tolerant of dirt and damage.
   - Unified National Fine (UNF): Tighter, more precise fit, used when vibration resistance is crucial.
   - Unified National Extra Fine (UNEF): Finest thread pitch, for very precise adjustments and thin-walled applications.
d. Identified by diameter, taper per inch, and series (e.g., 1/2 inch-13 UNC).

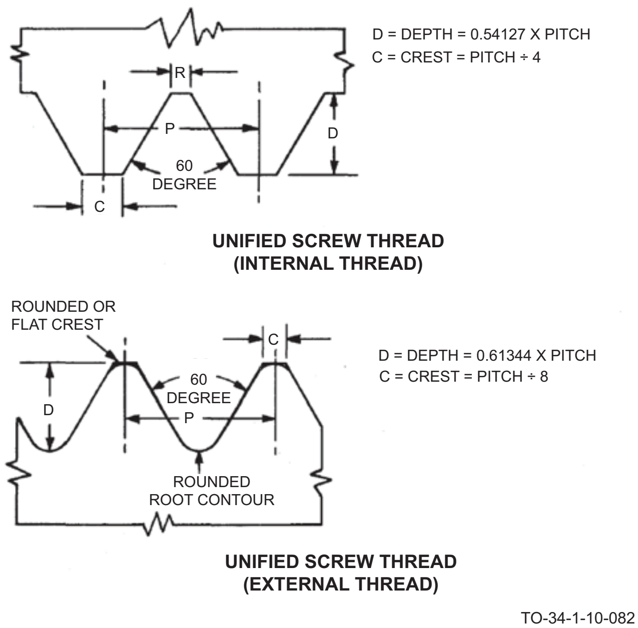


Figure 4-48. General Form Dimensions for Unified Screw Threads

4.19.16.2 Metric Threads.

- Globally recognized standard, prevalent in Europe and Asia.
- Similar to unified threads with a 60 degree thread angle but uses metric units for measurement.
- Designated by diameter and pitch in millimeters, (e.g., M10 x 1.5).

4.19.16.3 Acme Threads. (See Figure 4-49.)

- Used for power transmission, known for high load carrying capacity.
- Found in applications like lead screws, vices, and jacks.
- Trapezoidal thread form with a 29 degree thread angle.

4.19.16.4 Square Threads. (See Figure 4-49.)

- Offer even higher efficiency than Acme threads but are more difficult to manufacture and less tolerant of misalignment.
- Used in applications requiring precise linear motion, such as machine tools.
- Square-shaped thread profile.

4.19.16.5 29 Degree Worm Threads. (See Figure 4-49.)

- Specifically designed for worm gear applications.
- Provide high gear ratios and are self-locking, making them suitable for one-directional power transmission.
- Thread form similar to Acme but with a shallower angle and a lead angle specific to the gear ratio.

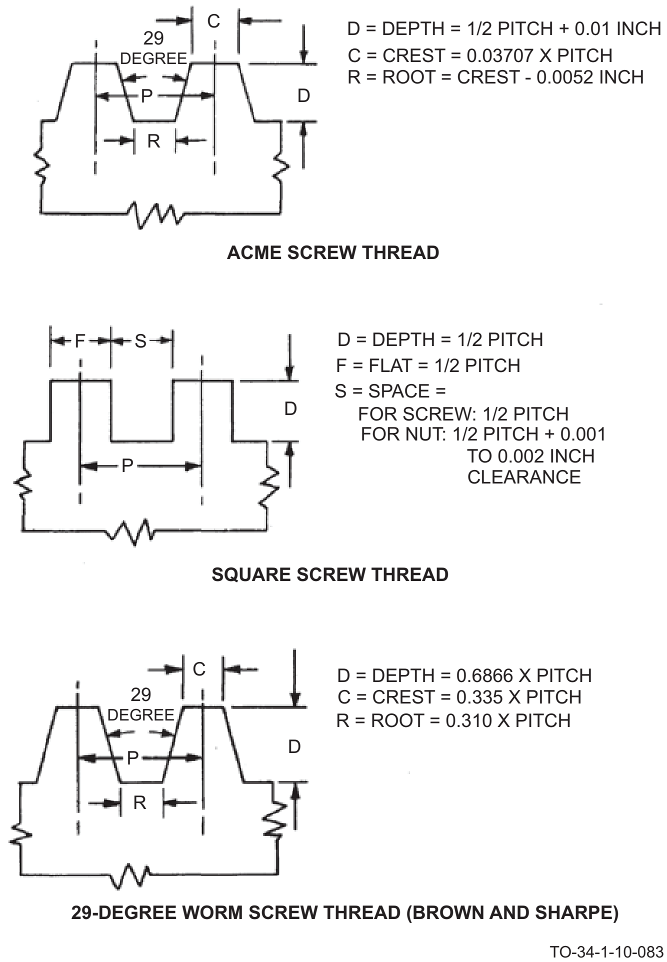


Figure 4-49. General Form Dimensions for Acme, Square, and 29 Degree Worm Screw Thread Forms

4.19.16.6 Table 4-6, Table 4-7and Table 4-8 lists the general thread pitches and dimensions for UNC, UNF Metric screw threads, respectively. These figures are basic and do not account for different classes of fit, tolerances, and allowances.

NOTE

Unlike metric screw threads that use diameter and pitch, UNC and UNF threads are designated by diameter and threads per inch. For instance, **1/4-20** represents a UNC/UNF thread with a 1/4 inch diameter and 20 threads per inch, while **M10 x 1.5** indicates a metric thread with a 10 millimeter (mm) diameter and 1.5 mm pitch. For a detailed explanation of thread designation and fit, refer to Paragraph 4.19.17.

Table 4-6. General Dimension of UNC Screw Thread

UNC Series				
Nominal Diameter (Inch)	Threads Per Inch	Major Diameter (Inch)	Pitch Diameter (Inch)	Minor Diameter (Inch)
Number (No.) 1 (0.073)	64	0.073	0.0629	0.0527
No. 2 (0.086)	56	0.0860	0.0744	0.0628
No. 3 (0.099)	48	0.0990	0.0855	0.0719
No. 4 (0.112)	40	0.1120	0.0958	0.0795
No. 5 (0.125)	40	0.1250	0.1088	0.0925

Table 4-6. General Dimension of UNC Screw Thread - Continued

UNC Series				
Nominal Diameter (Inch)	Threads Per Inch	Major Diameter (Inch)	Pitch Diameter (Inch)	Minor Diameter (Inch)
No. 6 (0.138)	32	0.1380	0.1177	0.0974
No. 8 (0.164)	32	0.1640	0.1437	0.1234
No. 10 (0.190)	24	0.1900	0.1629	0.1359
No. 12 (0.216)	24	0.2160	0.1889	0.1619
1/4	20	0.2500	0.2175	0.1850
5/16	18	0.3125	0.2764	0.2403
3/8	16	0.3750	0.3344	0.2938
7/16	14	0.4375	0.3911	0.3447
1/2	13	0.5000	0.4500	0.4001
9/16	12	0.5625	0.5084	0.4542
5/8	11	0.6250	0.5660	0.5069
3/4	10	0.7500	0.6850	0.6201
7/8	9	0.8750	0.8028	0.7307
1	8	1.0000	0.9188	0.8376
1-1/8	7	1.1250	1.0322	0.9394
1-1/4	7	1.2500	1.1572	1.0644
1-3/8	6	1.3750	1.2667	1.1585
1-1/2	6	1.5000	1.3917	1.2835
1-3/4	5	1.7500	1.6201	1.4902
2	4-1/2	2.0000	1.8557	1.7113

Table 4-7. General Dimensions of UNF Screw Thread

UNF Series				
Nominal Diameter (inch)	Threads Per Inch	Major Diameter (inch)	Pitch Diameter (inch)	Minor Diameter (inch)
No. 0 (0.060)	80	0.0600	0.0519	0.0438
No. 1 (0.073)	72	0.0730	0.064	0.0550
No. 2 (0.086)	64	0.0860	0.759	0.0657
No. 3 (0.099)	56	0.0990	0.0874	0.0758
No. 4 (0.112)	48	0.1120	0.0985	0.0849
No. 5 (0.125)	44	0.1250	0.1102	0.0955
No. 6 (0.138)	40	0.1380	0.1218	0.1055
No. 8 (0.164)	36	0.1640	0.1460	0.1279
No. 10 (0.190)	32	0.1900	0.1697	0.1494
No. 12 (0.216)	28	0.2160	0.1928	0.1696
1/4	28	0.2500	0.2268	0.2036
5/16	24	0.3125	0.2854	0.2584
3/8	24	0.3750	0.3479	0.3209
7/16	20	0.4375	0.4050	0.3725
1/2	20	0.5000	0.4675	0.4350
9/16	18	0.5625	0.5264	0.4903
5/8	18	0.6250	0.5889	0.5528
3/4	16	0.7500	0.7094	0.6688
7/8	14	0.8750	0.8286	0.7822

Table 4-7. General Dimensions of UNF Screw Thread - Continued

UNF Series				
Nominal Diameter (inch)	Threads Per Inch	Major Diameter (inch)	Pitch Diameter (inch)	Minor Diameter (inch)
1	14	1.0000	0.9536	0.9072
1-1/8	12	1.1250	1.0709	1.0167
1-1/4	12	1.2500	1.1959	1.1417
1-3/8	12	1.3750	1.3209	1.2667
1-1/2	12	1.5000	1.4459	1.3917

Table 4-8. General Dimensions of Metric Screw Thread

Metric Series						
Nominal Diameter (mm)	Pitch (mm)	Threads Per Inch. (Approximately)	Major Diameter (mm)	Pitch Diameter (mm)	Minor Diameter (mm)	Closest UNF Size
M1	0.25	100	1	0.9	0.815	No. 0
M1.2	0.25	100	1.2	1.1	1.015	No. 1
M1.6	0.35	71	1.6	1.479	1.367	No. 2
M2	0.4	63	2	1.794	1.608	No. 4
M2.5	0.45	56	2.5	2.286	2.073	No. 5
M3	0.5	50	3	2.773	2.546	No. 5
M4	0.7	36	4	3.667	3.345	No. 8
M5	0.8	32	5	4.647	4.319	No. 10
M6	1	25	6	5.563	5.185	1/4 inch
M8	1.25	20	8	7.389	6.684	5/16 inch
M10	1.5	17	10	9.266	8.653	3/8 inch
M12	1.75	14	12	11.134	10.419	7/16 inch
M16	2	13	16	14.732	13.744	5/8 inch
M20	2.5	10	20	18.376	17.052	3/4 inch
M24	3	8	24	22.051	20.303	1 inch

4.19.17 Thread Fit and Designations. Thread fit refers to the degree of looseness or tightness between mating internal (nut) and external (bolt) screw threads when they are assembled. Thread fit is crucial for achieving the desired functionality, strength, and interchangeability in threaded connections. There are three main classes of thread fit:

4.19.17.1 Class 1 Fit. This class provides the loosest fit, allowing for significant clearance between mating threads. It is often used in applications where ease of assembly and disassembly is paramount, even with slight variations in thread dimensions or the presence of dirt or debris. Examples include adjusting screws, fasteners in non-critical applications, and parts requiring frequent assembly and disassembly.

4.19.17.2 Class 2 Fit. This is the most common class of thread fit, providing a compromise between clearance and tightness. It offers a snug fit with minimal clearance, suitable for most general-purpose applications. Class 2 threads are widely used in machinery, vehicles, and structural components where a balance of strength and convenience is required.

4.19.17.3 Class 3 Fit. This class provides the tightest fit, resulting in intentional interference between mating threads. This interference creates a locking action, preventing loosening due to vibration or dynamic loads. Class 3 fits are employed in applications demanding high strength, resistance to loosening, and precise alignment, such as aerospace components, high-pressure systems, and precision instruments.

4.19.17.3.1 Thread fit is typically designated using a letter-based system, with different letters representing the tolerance class for both the internal and external threads. For instance, in the Unified Thread Standard (UTS), common designations include:

- Class 1: 1A (external) and 1B (internal)
- Class 2: 2A (external) and 2B (internal)
- Class 3: 3A (external) and 3B (internal)

4.19.17.3.2 Screw thread designation provides a comprehensive description of the thread's key characteristics, including its size, standard, and fit. The breakdown of each element is described below using **1/4-20 UNC-3A** below:

- 1/4: This represents the **nominal diameter** of the screw thread, expressed in inches. In this case, the nominal diameter is 0.25 inches or 1/4 inch.
- 20: This number indicates **threads per inch**. It signifies that there are 20 complete threads within a one inch length along the threaded portion.
- UNC: This stands for **Unified National Coarse**, specifying the thread series or standard that the thread conforms to. UNC threads are widely used for general-purpose applications.
- 3A: This alpha numeric designation indicates the **thread fit class**, specifically a Class 3A fit. Class 3 signifies an interference fit, meaning the threads are designed to engage tightly with intentional interference between the mating parts. The **A** further designates that this is the tolerance class for the external thread (typically the bolt).

4.19.18 Tool Selection and Grinding.

4.19.18.1 V-Threads (60 Degrees). Use standard thread cutting bits and cutters. (See Paragraph 4.4.4.2.)

4.19.18.2 UNC/UNF Threads.

a. Grind the cutter bit's point to match the thread root shape.

- External threads: Correct radius at the end.
- Internal threads: Flat end.

b. Grind tool to the angle specified by the thread pitch.

4.19.18.3 Acme and 29 Degrees Worm Threads. Grind the cutter bit to a 29 degree point angle.

a. Ensure sufficient side clearance to prevent rubbing (especially for steep pitches).

b. Grind the bit's end flat, matching the specific thread root width.

c. Utilize thread cutting tool gauges (Figure 4-50) for simplification.

4.19.18.4 Square Threads. Use a specialized square thread cutter bit. (See Figure 4-51.)

4.19.18.4.1 Grinding.

a. Grind side clearances within the helix angle, angling them towards the shank for additional clearance.

b. Grind the tool's end flat, equal to half the thread pitch, to create equal flats and spaces on the threaded part.

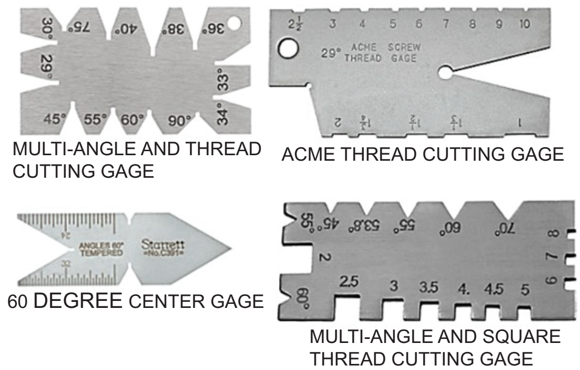


Figure 4-50. Various Thread Cutting Tool Gages

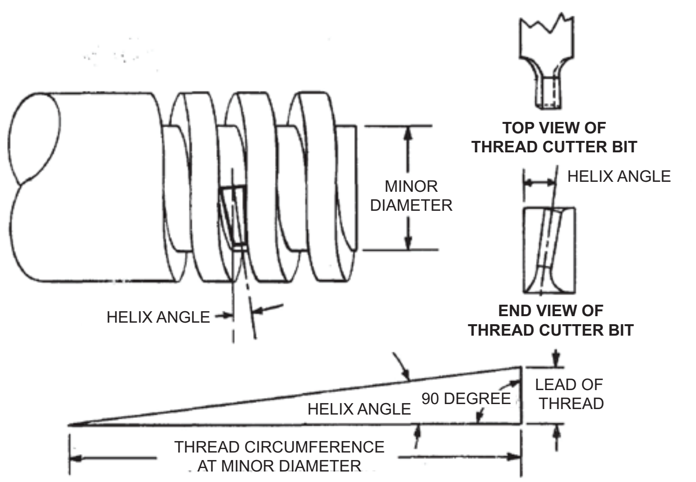


Figure 4-51. Thread Cutter Bits for Square Threads

4.19.19 General Cutter Bit Alignment.

4.19.19.1 Horizontal Alignment.

a. Ensure precise align the thread cutter bit with the workpiece axis.

b. Even slight vertical deviations will impact the cut thread angle.

4.19.19.2 Perpendicular Alignment.

a. Ensure the centerline of the bit’s ground thread angle is perpendicular to the workpiece axis.

b. Using a 60 degree center gauge (Figure 4-50):

(1) Place the gauge against the workpiece.

(2) Adjust the cutter bit on the tool post until its point fits snugly into the gauge’s 60 degree notch. (See Figure 4-52.)

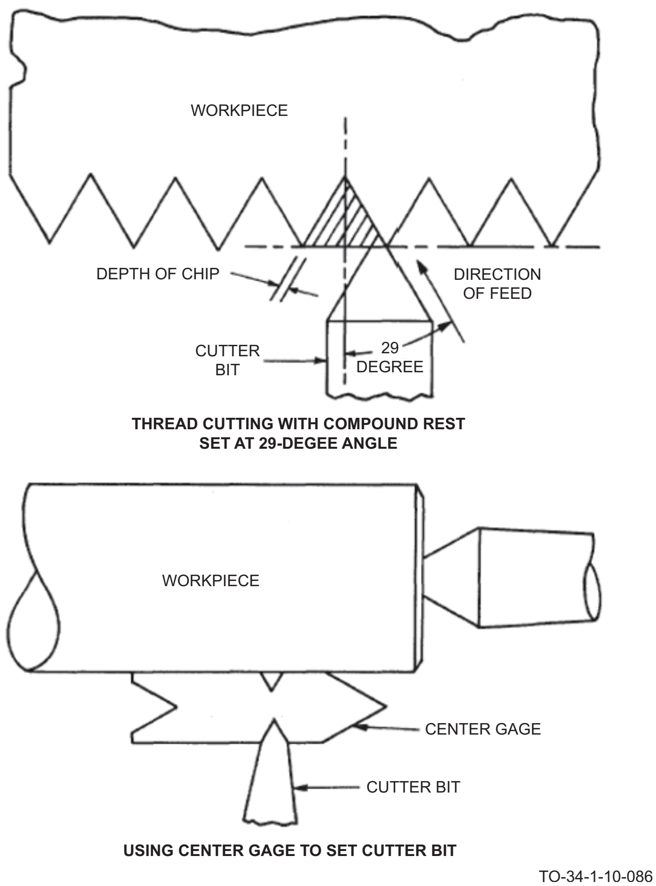


Figure 4-52. Setting Up Cutter Bit for Thread Cutting Operation

4.19.20 Engaging the Threading Mechanism.

4.19.20.1 Purpose. Engaging the lathe's threading mechanism is crucial after tool selection, grinding, and positioning to:

4.19.20.1.1 Verify Setup. Conduct a dry run without engaging the cutting tool to ensure all components are properly aligned and functioning correctly. This helps prevent potential damage to the workpiece or the lathe.

4.19.20.1.2 Precise Thread Cutting. The threading mechanism synchronizes the longitudinal movement of the carriage with the rotation of the spindle. This synchronization is essential for achieving the precise feed rate required to cut threads with the desired pitch and profile.

4.19.20.1.3 Consistent Results. Using the threading mechanism ensures consistent thread depth and lead, producing accurate and uniform threads along the entire length of the workpiece.

4.19.20.2 Gearbox and a Thread Chasing Dial. (See Figure 4-53.)

4.19.20.2.1 Gearbox.

a. Select the appropriate gear combination to achieve the desired thread pitch (consult lathe chart or manual).

b. This sets the ratio of lead screw rotation relative to the spindle, controlling tool travel per spindle revolution.

4.19.20.2.2 Thread Chasing Dial. Features marking corresponding to specific points on the lead screw.

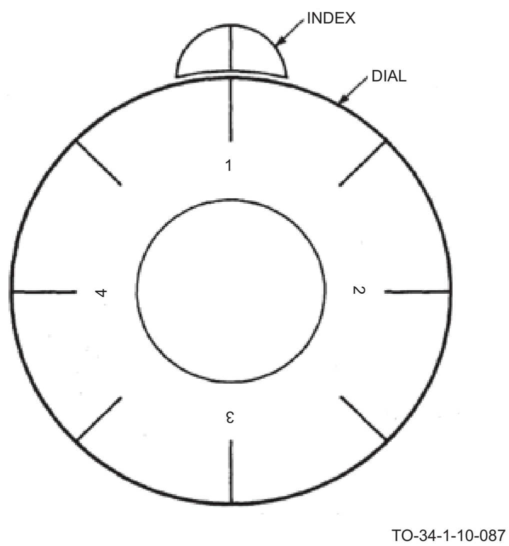


Figure 4-53. Example of a Thread Chasing Dial

4.19.20.3 Engagement Process.

a. Engage the half-nut lever (usually on the apron) to connect the carriage to the lead screw.

b. Engage the half-nut lever when the indicator on the thread chasing dial aligns with the desired mark. This ensures consistent thread starts and prevents cross-threading.

c. **Important:** Disengage the half-nut lever after each pass and re-engage at the same indicator mark for subsequent passes.

NOTE

Always double-check your gearbox settings and thread dial engagement before initiating the threading process to ensure accurate and consistent thread cutting.

4.19.21 General Thread Cutting Operation.

4.19.21.1 Workpiece Preparation.

a. Choose the correct material and size for the desired thread.

b. Face the workpiece end to ensure perpendicularity to its axis.

c. Turn the workpiece down to the desired major diameter of the thread.

d. Chamfering/Beveling.

(1) Create a chamfer or bevel at the starting edge of the workpiece.

(2) This provides a clean entry point for the cutting tool.

(3) Helps prevent chipping or damage to the tool's cutting edge.

4.19.21.2 Tooling.

4.19.21.2.1 Bit Selection. Choose a thread cutting bit matching the desired thread type (e.g., UNC, UNF, metric), pitch, and workpiece material.

4.19.21.2.2 Bit Preparation.

a. Ensure the bit is properly sharpened to the appropriate angle.

b. Verify the bit has any necessary clearances for the specific thread form.

4.19.21.2.3 Bit Mounting. Double-check that the bit is securely mounted and aligned in the tool holder. This prevents chatter and inconsistent thread forms.

4.19.21.3 Lathe Setup.

4.19.21.3.1 Engage Threading Mechanism.

a. Select the correct gear combination in the quick-change gearbox to achieve the desired thread pitch.

b. Consult your lathe's manual or the chart on the machine for appropriate gear settings.

4.19.21.3.2 Set Spindle Speed.

a. Choose a slower RPM (Table 4-2) suitable for threading based on the material.

b. A slower speed allows for better control and reduces the risk of damage to the cutting tool or workpiece.

4.19.21.4 Initial Cut.

4.19.21.4.1 Depth of Cut. Set a shallow depth of cut for the first pass to prevent overloading the tool or workpiece.

4.19.21.4.2 Engage Feed and Cut.

a. Engage the carriage feed

b. Carefully guide the tool into the rotating workpiece.

4.19.21.4.3 Disengage and Return.

a. Once the desired thread length is reached, disengage the half-nut lever to separate the carriage from the lead screw.

b. Return the carriage to its starting position.

4.19.21.5 Subsequent Passes.

4.19.21.5.1 Depth of Cut. Gradually increase the depth of cut with each pass to reach the final thread dimension.

4.19.21.5.2 Engage the Half-Nut Lever.

a. Before each pass, engage the half-nut lever only when the thread chasing dial aligns with the same mark used for the previous pass.

b. This ensures consistent thread lead and prevents cross-threading.

4.19.21.5.3 Repeat.

a. Repeat the process of engaging the half-nut at the correct mark, cutting, disengaging the half-nut, and returning the carriage to the starting position.

b. Continue until the desired thread depth is achieved.

4.19.21.6 Finishing, Chamfering, and Inspection.

4.19.21.6.1 Finishing Pass. Perform a final light finishing pass with a minimal depth of cut to create a smooth and consistent thread surface.

4.19.21.6.2 Chamfering. Cut a 45 degree chamfer at the end of the thread during the finishing pass to ease thread engagement and prevent damage.

4.19.21.6.3 Inspection.

a. Thoroughly inspect the newly cut threads:

(1) Use thread gauges or a thread micrometer.

(2) Verify the accuracy of the thread's pitch, major diameter, and overall fit.

b. Ensure the threads meet the required specifications and tolerances.

4.19.22 Taper Screw Threads. Cutting taper screw threads, commonly known as pipe threads on a lathe requires a slightly different approach than straight threads. Instead of a uniform diameter, these threads gradually increase in diameter along the length of the workpiece.

4.19.22.1 National Taper Pipe Threads (NPT). NPT are a widely used standard for creating leak-resistant threaded connections in pipes and fittings. This is achieved through their key characteristic: a tapered design where the thread diameter gradually increases along its length. Specifically, NPT threads feature a taper of 3/4 inch per foot, meaning the diameter increases by 3/4 of an inch for every 12 inches of thread length.

4.19.22.2 Primary Methods. There are two primary methods for achieving this taper on a lathe:

4.19.22.2.1 Tailstock Offset Method. This method involves precisely offsetting the tailstock center towards the operator by a calculated amount. This offset creates the desired taper angle as the workpiece rotates.

4.19.22.2.2 Taper Attachment Method. A taper attachment is a specialized accessory that mounts to the lathe's cross slide. It allows for precise and controlled angular movement of the tool post, enabling accurate taper cutting.

NOTE

When cutting a taper thread, the cutter bit should be set at right angle to the axis of the workpiece. Do not set the thread cutter bit at right angle to the taper of the thread.

4.19.23 Thread Measurement.

4.19.23.1 Determining Thread Pitch.

4.19.23.1.1 Visual Inspection. Use a machinist's scale to count the number of taper per inch. For greater accuracy, count threads over a two-inch span and divide by two.

4.19.23.1.2 Pitch Gauge. Select a gauge that matches the thread profile. The gauge provides a direct reading of the taper per inch.

4.19.23.2 Measuring Pitch Diameter.

4.19.23.2.1 Thread Micrometer Caliper. This specialized caliper directly measures the pitch diameter. Select the appropriate caliper range for the thread size. Consult a reference table to determine the target pitch diameter for comparison.

4.19.23.2.2 Three-Wire Method. (See Figure 4-54.) This highly accurate method utilizes three wires of identical diameter placed withing the thread grooves.

4.19.23.2.3 Wire Selection. Choose wires with a diameter that contacts the thread flanks at the pitch diameter. Table 4-9 provides standard wire sizes for common thread pitches Alternatively, calculate the optimal wire diameter (D*w*) using the following formula:

$$D_w = \frac{0.57735}{N}$$

WHERE D_W = DIAMETER OF WIRE (IN INCHES)

N = THREADS PER INCH.

4.19.23.2.4 Measurement. Position the wires in the thread grooves and measure the distance across them (M) using an outside micrometer (See Figure 4-54.)

4.19.23.2.5 Calculation. Calculate the pitch diameter using the following formula:

$$M = D_m + 3\ D_w - \frac{1.5155}{N}$$

WHERE M = MEASUREMENT ACROSS WIRES (IN INCHES)

D_m = MAJOR DIAMETER (IN INCHES) OF SCREW

D_w = DIAMETER (IN INCHES) OF WIRE

N = THREADS PER INCH

EXAMPLE: TO MEASURE A 0.500 INCH MAJOR DIAMETER THREAD WITH 12 TPI:

WIRE SELECTION FROM TABLE 9: D_w = 0.04811 INCHES

CALCULATION: $M = D_m + 3\ D_w - \frac{1.5155}{N}$

$= 0.500 + 3(0.04811) - \frac{1.5155}{12} = 0.51803$ INCH

THEREFORE, THE MICROMETER READING ACROSS THE WIRES IS 0.51803 INCHES.

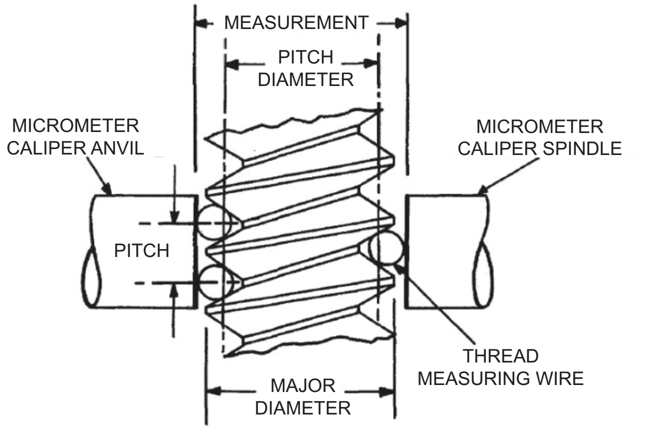


Figure 4-54. Measuring Threads by the Three-Wire Method

Table 4-9. Wire Sizes for Measuring 60 Degree Threads by the Three-Wire Method

Taper Per Inch of Part Being Measured	Wire Size (Inch)	Taper Per Inch of Part Being Measured	Wire Size (Inch)
80	0.00722	14	0.04124
72	0.00802	13	0.04441
64	0.00902	12	0.04811
56	0.01031	11-1/2	0.05020
48	0.01203	11	0.05249
44	0.01312	10	0.05773
40	0.01443	9	0.06415
36	0.01604	8	0.07217
32	0.01804	7	0.08248
28	0.02062	6	0.09623
27	0.02138	5	0.11547
24	0.02406	4-1/2	0.12830
20	0.02887	4	0.14434
18	0.03208	3-1/2	0.16496
16	0.03608	3	0.19245

4.19.24 Knurling. Knurling is a process that creates a patterned, textured surface on a workpiece. This is achieved by pressing a knurling tool with rotating knurls against the rotating workpiece, embossing the pattern.

4.19.24.1 Common Reasons for Knurling.

4.19.24.1.1 Enhanced Grip. Knurling provides a better gripping surface, especially useful for tools and handles.

4.19.24.1.2 Decorative Appeal. Knurling can add a visually appealing texture to a workpiece.

4.19.24.1.3 Diameter Enlargement. The knurling process can slightly increase the diameter of the workpiece.

4.19.24.2 General Instructions for Knurling a Material.

4.19.24.2.1 Workpiece Mounting.

a. Securely mount the workpiece using one of the following methods:

- Between centers: For longer workpieces, providing support at both ends.
- In a chuck: For shorter workpieces, clamping securely within the chuck jaws.

b. Ensure the mounting method provides adequate support to withstand the pressure exerted during the knurling process.

4.19.24.2.2 Knurling Tool Selection.

4.19.24.2.2.1 Pattern.

Diamond

More common pattern, offering a crisscross texture. (See Figure 4-12 and Figure 4-13.)

Straight Line

Creates a linear, textured pattern.

4.19.24.2.2.2 Pitch.

a. Select the desired coarseness of the knurl (measured in lines per inch or mm).

b. Diamond patterns typically come in a wider variety of pitches.

4.19.24.2.3 Tool Positioning.

a. Align the knurling tool so that the knurls are at equal distances above and below the workpiece centerline.

b. Ensure the working faces of the knurls are parallel to the workpiece surface.

4.19.24.2.4 Knurling Operation.

4.19.24.2.4.1 Lathe Setup.

a. Set the lathe to its lowest back gear speed.

b. Adjust the feed rate to approximately half the width of the knurl per workpiece revolution.

4.19.24.2.4.2 Initial Cut.

a. Position the knurling tool at the starting point of the desired knurled area.

b. Start the lathe and carefully press the knurls into the workpiece to a depth of about 1/64-inch using the hand crossfeed.

4.19.24.2.4.3 Pattern Check and Completion.

a. Inspect the knurl pattern for uniformity. A split diamond pattern indicates misalignment. Adjust the tool positioning if necessary.

b. Once a satisfactory pattern is achieved, knurl the entire desired area, applying cutting oil liberally throughout the process.

4.19.25 Boring. Boring on a lathe involves enlarging and finishing a pre-existing hole in a workpiece. This is achieved by rotating the workpiece while a single point cutting tool, called a boring bar, is fed linearly into the material. Boring provides greater accuracy, and a smoother finish compared to drilling and is ideal for achieving precise diameters and depths.

4.19.25.1 Mounting the Workpiece for Boring. There are several methods for mounting a workpiece for boring, depending on its size and shape.

4.19.25.2 Chuck Mounting.

a. Most common method for boring.

b. Workpiece is held securely in a 3-Jaw or 4-Jaw chuck:

c. Provides:

- Strong grip.
- Accurate centering.

4.19.25.3 Faceplate Mounting.

a. Used for larger or irregularly shaped workpieces that cannot be held in a chuck.

b. Workpiece is mounted on a faceplate using bolts and clamps.

c. Ensure:

- Secure fastening.
- Workpiece center aligns with lathe's axis of rotation.

4.19.25.4 Mounting Between Centers.

a. Used for long and slender workpieces.

b. The workpiece is supported between the headstock center and tailstock center.

c. A lathe dog:

- Is attached to the workpiece.
- Engages with a drive plate on the headstock spindle to transmit rotary motion.

4.19.25.4.1 Boring Bars.

4.19.25.4.2 Material. Typically made of HSS or carbide for durability and cutting performance.

4.19.25.4.3 Size Selection. Choose boring bars based on:

Length

Sufficient to reach the desired bore depth.

Diameter

Adequate rigidity to minimize vibrations during cutting, especially for deeper bores.

4.19.25.4.4 Boring Bar Holders.

4.19.25.4.4.1 Purpose. Securely hold the boring bar in the lathe's tool post.

4.19.25.4.4.2 Types.

Adjustable Inserts

Allow for indexable inserts to be easily replaced.

Brazed Carbide Tips

Provide a fixed cutting edge.

4.19.25.4.5 Tool Height Setting.

Importance

Crucial for perpendicular cuts and preventing taper in the bore.

Procedure

Set the cutting edge of the boring tool precisely at the same height as the lathe's centerline.

Tools: Use a tool height gauge or the tailstock center for accurate alignment.

4.19.25.4.6 Cutting Tool Geometry.

4.19.25.4.6.1 Impact. Significantly affects cutting forces, chip formation, and surface finish.

4.19.25.4.6.2 Parameters.

Rake Angle

Determines the cutting edge's angle of approach.

Clearance Angle

Prevents rubbing between the tool and the workpiece.

Nose Radius

Influences chip flow and surface finish.

4.19.25.4.6.3 Selection. Refer to machining data charts for recommended values based on the specific workpiece material.

4.19.26 Straight Boring Operation.

4.19.26.1 Setting the Speed and Feed.

a. Consult machining data charts for recommended values.

b. Select appropriate:

- Spindle speed (RPM).
- Feed rate (inch per revolution).

c. Base selection on:

- Workpiece material.
- Diameter of the bore.

4.19.26.2 Starting the Cut.

a. Position the boring bar close to the pre-existing hole. (See Figure 4-55.)

b. Start the lathe spindle.

c. Gradually feed the tool into the workpiece using the carriage handwheel.

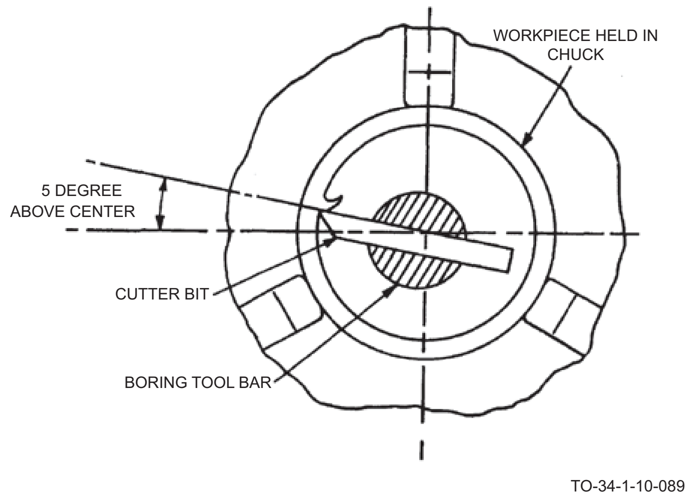


Figure 4-55. Cutter Bit Position for Straight Boring

4.19.26.3 Maintaining Constant Cutting Depth.

a. Ensure a consistent depth of cut throughout the operation.

b. Avoid interrupting the cut to prevent tool marks on the bore surface.

4.19.26.4 Lubrication and Cooling.

a. Apply cutting fluid generously to:

- Reduce heat and friction.
- Minimize tool wear.

b. Proper lubrication contributes to a smoother surface finish. (Refer to Table 4-4 and Table 4-10.)

4.19.26.5 Measuring the Bore Diameter.

a. Periodically halt the lathe.

b. Measure the bore diameter using:

- Telescopic gauge
- Micrometer

4.19.27 Taper Boring Operation.

4.19.27.1 Setting the Taper Attachment.

a. Utilize the lathe's taper attachment to achieve the desired taper angle.

b. Taper attachment function:

- Allows precise angular movement of the carriage.
- Facilitates the creation of a tapered bore.

4.19.27.2 Calculating the TPI. Determine the TPI based on the required taper angle and the length of the taper.

4.19.27.3 Setting the Taper Attachment Scale. Adjust the taper attachment scale to match the calculated TPI. This ensures the toolpath generates the correct taper angle.

4.19.27.4 Performing the Taper Boring Operation.

a. Engage the taper attachment.

b. Execute the boring operation similarly to straight boring.

c. The taper attachment automatically:

- Controls tool movement.
- Creates the desired taper.

4.19.28 Internal Threading Operation.

4.19.28.1 Selecting the Threading Tool. Choose a properly sized threading tool with the correct thread form (e.g., Unified National, Metric).

4.19.28.2 Setting the Tool Angle. Precise alignment of the threading tool with the helix angle of the thread is essential.

4.19.28.2.1 Methods.

- Tool holder with adjustable angle.
- Dedicated threading tool holder.

4.19.28.3 Setting the Compound Rest. (See Figure 4-56.) Set the compound rest angle to half the thread angle. This enables precise control over the threading tool's depth of cut.

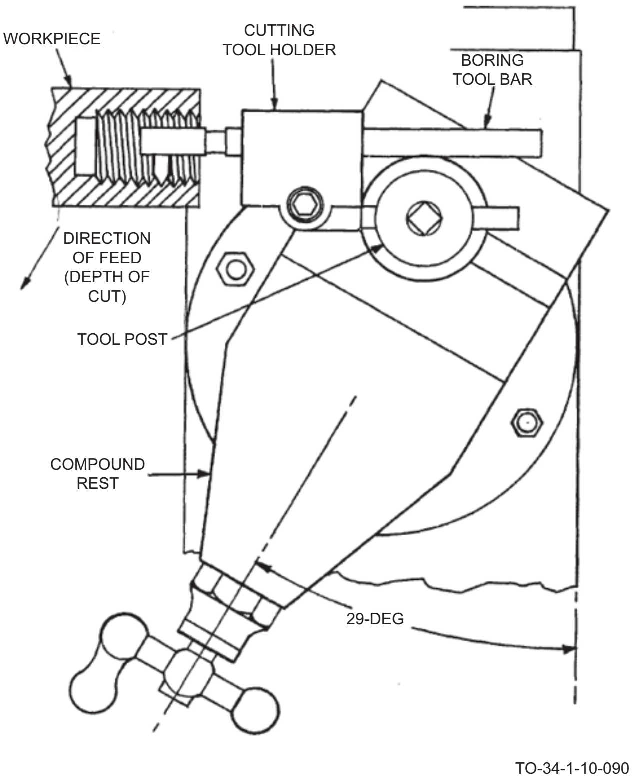


Figure 4-56. Setup Compound Rest for Internal Threading

4.19.28.4 Engaging the Thread Dial. The thread dial ensures the tool's entry and exit at the correct point for each thread pass by selecting the appropriate thread dial setting based on the thread pitch.

4.19.28.5 Performing the Threading Operation.

a. Start the lathe.

b. Slowly feed the threading tool into the pre-drill hole.

c. Maintain a light but consistent feed rate.

d. Back out the tool at the end of each pass.

e. Repeat the process, gradually increasing the depth of cut with each pass.

f. Continue until the desired thread depth is achieved.

4.19.29 Filing. Filing is a finishing operation that removes burrs, sharp edges, and tool marks, resulting in a smoother, safer, and more aesthetically pleasing surface. It helps achieve close tolerances, create specific shapes and contours, and deburr internal features.

4.19.29.1 Files. Mill files are generally preferred for lathe filing. Bastard cut files are suitable for roughing, while second cut files provide a finer finish. Round, half-round, and flat files can be used on irregularly shaped workpieces.

4.19.29.2 Lathe Speed.

a. Higher than rough turning speeds.

- Ferrous metals: 4-5 times the roughing speed.
- Non-ferrous metals: 2-3 times the roughing speed.

b. Avoid excessively slow speeds (out-of-the-round finish) and excessively high speeds (file glazing and wear).

4.19.29.3 Filing Technique.

a. Hold file at 10 degrees to the right.

b. Apply light, even pressure.

c. Slow, sliding motion from left to right.

d. Several workpiece revolutions per stroke.

e. Avoid reversing stroke direction (chatter marks).

f. Use less pressure than bench filing.

g. Clean file frequently to prevent scratching.

h. Given that filing primarily removes tool marks, leave only 0.002 to 0.005 inches for this operation.

4.19.30 Polishing.

4.19.30.1 Purpose. Achieve a smooth, reflective surface on the workpiece for aesthetics and corrosion resistance.

4.19.30.2 Process.

- Hold abrasive materials (emery cloth, cloth wheels, felt bobs) charged with progressively finer polishing compounds against the rotating workpiece.
- Controlled pressure and rotation remove minute imperfections, resulting in a mirror-like finish.

4.19.30.3 Polishing on a Lathe.

WARNING

Remove any loose clothing and jewelry prior to performing any lathe operations. Failure to comply could result in injury to, or death of, personnel or long term health hazards.

4.19.30.4 Material.

- Ferrous metals: Emery cloth is ideal.
- Non-ferrous metals: Abrasive paper is often preferred.

4.19.30.5 Speed.

- Ideal: Around 5,000 FPM.
- Use the highest speed attainable on your lathe for optimal results.

4.19.30.6 Application.

Emery cloth or abrasive cloth is typically held by hand.
For better control, especially on cylindrical surfaces:

- May wrap abrasive material around a block.
- Use an improvised clamp to hold the material.

4.19.30.7 Material Removal. Allow 0.0002 to 0.0005 inches.

4.19.30.8 Finish.

- Satin: Use oil with the abrasive.
- Brighter: Polish dry.

4.19.30.9 Finish Polish. Use crocus cloth after initial polishing for a higher luster.

4.19.31 Drilling on a Lathe. Drilling on a lathe can be accomplished securing the workpiece in the lathe chuck and rotating it at the desired speed. The twist drill is then held stationary in the tailstock, and the feed is controlled to create the hole. (See Figure 4-57.)

4.19.31.1 Supporting a Twist Drill in the Tailstock.

4.19.31.1.1 Drill Chuck.

- Most common method.
- Holds straight-shank drills and combination drill/countersinks.
- Chuck is mounted in the tailstock taper.

4.19.31.1.2 Direct Mounting. For taper-shank drills matching the tailstock spindle taper.

4.19.31.1.3 Drill Sockets/Sleeves. Adapts taper-shank drills to a different tailstock taper.

4.19.31.1.4 Lathe Dog and Tailstock Center.

- Suitable for large drills.
- Lathe dog clamped to drill shank.
- Drill tang supported by tailstock center.
- Lathe dog bears against cross slide or compound rest.

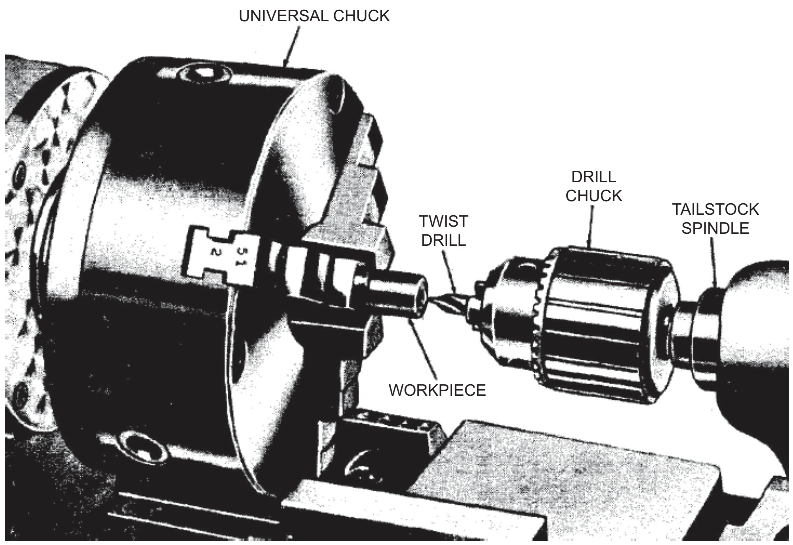


Figure 4-57. Drilling Workpiece in the Lathe

4.19.32 Drilling Preparation.

4.19.32.1 Setup.

4.19.32.1.1 Mount the Workpiece.

4.19.32.1.1.1 Chuck. Use a universal or independent chuck for secure holding.

4.19.32.1.1.2 Faceplate. Mount on a faceplate for larger or irregularly shaped workpieces.

4.19.32.1.2 Center Marking. Accurately mark and punch the hole center on the workpiece.

4.19.32.2 Center Alignment.

4.19.32.2.1 Using a Center Indicator. Verify alignment of the punch mark with the headstock axis. This ensures drilling accuracy and prevents the drill from wandering.

4.19.33 Tailstock Support. When rotating the drill in the headstock:

a. Carefully position the tailstock center and support the workpiece opposite the drill.

b. Ensure secure workpiece mounting to prevent rotation during drilling.

4.19.34 Speed and Feed Determination. Consult relevant charts for recommended drilling speeds and feeds based on:

- Drill diameter.
- Workpiece material.

4.19.34.1 Spindle Speed. Spindle speed remains consistent whether rotating the drill or the workpiece.

4.19.34.2 Drilling Procedure.

4.19.34.2.1 Feed Control.

a. Use the tailstock handwheel to control the drilling feed rate.

b. Monitor drilling depth using the graduations on the tailstock spindle.

4.19.34.2.2 Pilot Hole (For Large Drills).

(1) When using large diameter drills, start with a pilot hole.

(2) Pilot hole diameter should be slightly smaller than the larger drill bit's web thickness to:

- Guide the larger drill bit.
- Reducing drilling forces and improve accuracy.

4.19.35 Reaming Operation.

NOTE

Reamers enlarge drilled holes to precise diameters. Leave 0.004 to 0.012 inches for reaming, as they are not intended for heavy material removal. (See Figure 4-58.)

4.19.35.1 Machine Reaming.

a. Mount the workpiece in the chuck, drill the hole to within 0.012 inches of the final size.

b. Mount the reamer in the tailstock (similar to holding a drill).

4.19.35.2 Hand Reaming.

a. Mount the workpiece in the chuck and lock the headstock spindle.

b. Drill the hole to within 0.005 inches of the final size.

(1) Secure the hand reamer in a tap wrench and support it with the tailstock center.

(2) Rotate the wrench by hand while feeding with the tailstock handwheel.

c. Withdraw the reamer carefully, rotating in the same direction as reaming. Never reverse a reamer.

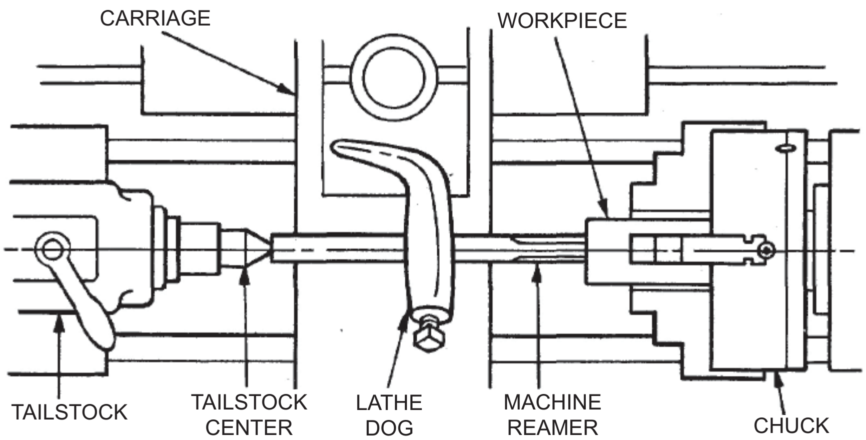


Figure 4-58. Reaming on the Lathe

4.19.36 Threading on a Lathe. (Tapping and Die Cutting).

4.19.36.1 General Considerations.

a. Consult thread standards for specifications.

b. Refer to tap drill charts to determine the correct hole size before tapping.

4.19.37 Tapping Threads.

4.19.37.1 Method 1 (Large Taps/Hard Materials).

4.19.37.1.1 Setup.

a. Lock the headstock.

b. Secure the workpiece in the chuck.

c. Position the tap in the pre-drilled hole.

d. Support the tap with the tailstock center.

4.19.37.1.2 Operation.

a. Rotate the tap with a wrench.

b. Apply pressure simultaneously using the tailstock handwheel.

4.19.37.2 Method 2 (Free-Machining Materials).

4.19.37.2.1 Setup.

a. Lock the headstock.

b. Secure the workpiece in the chuck.

c. Hold the tap in a wrench, resting the handle against the compound rest.

d. Support the tap with the tailstock center.

4.19.37.2.2 Operation.

a. Run the lathe at slow speed.

b. Apply pressure with the tailstock handwheel to cut the threads.

4.19.38 Die Cutting Threads Procedure.

4.19.38.1 Setup.

a. Mount the workpiece in the chuck.

b. Position the die on the workpiece end.

c. Rest the die stock handle on the compound rest.

4.19.38.2 Operation.

a. Run the lathe at a slow speed.

b. Apply pressure with the tailstock handwheel to cut the threads.

4.20 GENERAL MACHINE OVERVIEW - CNC.

CNC lathes are highly automated machine tools used for precision machining of cylindrical or rounded parts from various materials, including metals, plastics, and wood. These machines use CNC to execute programmed instructions, allowing for the creation of complex and repeatable shapes through operations like turning, facing, and threading. The following sections will provide detailed information on the operation, functions, and features of CNC lathes commonly found in AMT shops.

4.20.1 CNC Controller. The CNC controller is designed for ease of use, providing intuitive controls and a user-friendly interface for machinists of all levels. (See Figure 4-59.) The controller consists of a keypad, an LCD display, and a set of function keys that allow users to input commands, navigate menus, and execute machining programs. Key features include:

LCD Display and Interface

Displays active programs, tool paths, machine coordinates, and error messages.

Soft Keys

Adaptive function keys that change based on menu selections.

Handle Jog and Rapid Override

Provides manual axis movement control and speed adjustment.

Memory and Edit Mode

Allows program storage, retrieval, and modification.

Manual Data Input (MDI) Mode

Enables direct G-code entry for quick operations.

Auto Mode (Cycle Start/Feed Hold/Reset)

Controls program execution and machine cycling.

TO-34-1-10-093

Figure 4-59. Example CNC Controller

4.20.2 Powering On the CNC Lathe.

CAUTION

Before powering on the CNC lathe, check that all possible crash areas are clear (e.g., tool probe, parts catcher, tailstock, tool turret, and secondary spindle). Failure to comply, could result in damage to, or destruction of, equipment or loss of mission effectiveness.

a. Turn the main power switch (located on the side of the machine) to the ON position.

b. Press the POWER ON button on the control panel.

4.20.3 Homing the Axes.

a. Press POWER UP/RESTART. This command automatically home all axes.

b. Allow the system to boot up and display the main screen.

c. Clear any alarms by pressing RESET.

d. Alternatively, if there’s a need to home the axes manually:

(1) Press HANDLE JOG.

(2) Press X or Z followed by HOME to each axis individually.

4.20.4 Check Tool and Cutter Offsets. Access the offset table by performing the following:

a. Press OFFSET on the control panel.

b. Navigate to the TOOL OFFSETS tab to view the current offset values.

4.20.4.1 Verify Tool Length Offsets.

a. Load the tool you want to check into the turret.

b. Jog the tool close to the part or a reference surface.

c. Use a gauge or touch off the tool lightly on the part surface.

d. Record the measured distance and compare it to the existing tool offset.

(1) Press X DIAMETER MEASURE or Z FACE MEASURE.

(2) Enter the correct measured value and press ENTER.

4.20.4.2 Verify Cutter Radius Compensation, if Applicable.

a. Check the cutter compensation settings under OFFSET.

b. Compare the tool nose value in the table with the actual tool insert radius.

c. Adjust the X or Z radius compensation if needed.

4.20.4.3 Test Offsets in Graphics Mode.

a. Press MEMORY and select the program.

b. Press GRAPHICS to simulate the tool path.

c. Verify that the tool movements match expected dimensions.

4.20.4.4 Run a Trial Cut.

a. If safe, perform a test cut on the workpiece.

b. Measure the cut dimensions with a micrometer or caliper.

c. Compare the measurement with programmed dimensions.

d. Adjust offsets as necessary by entering corrections in the OFFSET table.

4.20.5 Operating the Machine Manually.

4.20.5.1 Jogging the Axes.

a. Press HANDLE JOG to activate jog mode.

b. Press X or Z-Axis key.

c. Adjust the jog speed using the 0.0001, 0.001, 0.01, or 0.1 settings.

d. Turn the jog handle to move the selected axis in the desired direction.

4.20.5.2 Spindle Control.

a. Activate spindle control by pressing MDI.

b. Start the spindle by typing the desired spindle speed and press ENTER.

c. Press FWD or REV to start the spindle in the corresponding direction.

d. Press STOP to halt spindle rotation.

4.20.5.3 Coolant Activation. Press COOLANT to toggle the coolant on or off.

4.20.6 Loading an Numerical Control (NC) Program into the Machine Controller.

4.20.6.1 Prepare the NC program.

a. Ensure the NC program is correctly formatted and saved as a .NC or .txt file

b. Store the program on a Universal Serial Bus (USB) drive, network drive, or Haas Control Memory.

4.20.6.2 Insert Storage Device.

a. If using a USB drive, insert it into the USB port on the control pendant.

b. If using a network storage, ensure the machine is connected to the network.

4.20.6.3 Access the Program List.

a. Press LIST PROGRAM to open the program manager.

b. Navigate to the USB or Network tab (use arrow keys to select the correct storage location).

4.20.6.4 Select the NC Program.

a. Use the arrow keys to highlight the desired program.

b. Press F2 to copy the program to the machine's memory.

4.20.6.5 Set the Active Program.

a. Navigate to the Memory tab in LIST PROGRAM.

b. Highlight the uploaded program and press SELECT PROGRAM to set it as active.

4.20.6.6 Verify the Program.

a. Press EDIT to review the program and ensure there are no errors.

b. Use GRAPHICS mode to simulate the program before running it.

4.20.6.7 Run the Program. Press CYCLE START to begin machining.

4.20.7 Executing a Program on the Machine Controller.

4.20.7.1 Load the Program.

a. Ensure the program is loaded into the controller's memory.

b. Press LIST PROGRAM, navigate to the Memory tab, select the desired program, and press SELECT PROGRAM.

4.20.7.2 Verify the Program.

a. Press EDIT to review the program for any errors.

b. Press GRAPHICS to simulate the program and verify tool movements without running the machine.

4.20.7.3 Set up the Machine.

a. Load and secure the workpiece in the chuck.

b. Install the required tools and set tool offsets.

c. Set the work offsets (G54, G55, etc.).

d. Ensure the correct spindle speed and feed rate are set.

4.20.7.4 Enter Memory Mode. Press MEMORY to prepare the controller for program execution.

4.20.7.5 Start the Program. Press CYCLE START to execute the loaded program.

4.21 GENERAL COOLANT SYSTEM REQUIREMENTS.

The coolant system in a CNC milling center plays a crucial role in heat dissipation, lubrication, and chip evacuation. Selecting the appropriate coolant type and maintaining proper coolant-to-water ratios ensures efficient machining and extends tool life. Refer to the table below for general information on coolant types and refractory recommendations.

Table 4-10. General Coolant Guidelines

Coolant Type	Description	Recommended Use	Maintenance Guidelines
Water-Soluble	General-purpose coolant with good cooling and lubrication	Ferrous and non-ferrous metals	Monitor concentration (6-10 percent), check for bacterial growth
Semi-Synthetic	Blend of synthetic and oil-based coolants, enhanced lubrication	High-speed machining	Maintain mix ratio, periodically clean sumps
Synthetic	Fully synthetic, excellent cooling, minimal residue	High-precision machining	Avoid contamination, check concentration regularly

4.22 GEOMETRIC AND MACHINE FUNCTIONS.

CNC lathes operate using a standardized set of G-codes (geometric functions), M-codes (machine functions), and other system commands. Understanding these codes is crucial for programming, troubleshooting, and optimizing machine operations. G-codes are responsible for controlling tool movement, positioning, and machining cycles. See the table below.

Table 4-11. Geometric Commands (G-Codes)

Code	Function	Description
G00	Rapid Positioning	Moves the tool at maximum speed to a specified location
G01	Linear Interpolation	Moves the tool in a straight line at a set feed rate
G02	Circular Interpolation (clockwise)	Moves the tool in a clockwise arc
G03	Circular Interpolation (counterclockwise)	Moves the tool in a counterclockwise arc
G04	Dwell	Pauses tool movement for a specified duration
G17	XY Plane Selection	Defines the XY plane for circular interpolation
G18	ZX Plane Selection	Defines the ZX plane

Table 4-11. Geometric Commands (G-Codes) - Continued

Code	Function	Description
G19	YZ Plane Selection	Defines the YZ plane
G20	Inch Mode	Sets the machine to interpret dimensions in inches
G21	Metric Mode	Sets the machine to interpret dimensions in millimeters
G28	Machine Home Run	Moves the tool to the machine home position
G40	Cutter Compensation Cancel	Disables tool diameter compensation
G41	Cutter Compensation Left	Offsets the tool path to the left
G42	Cutter Compensation Right	Offsets the tool path to the right
G43	Tool Length Compensation (+)	Adjusts tool height using a positive offset
G49	Tool Length Compensation Cancel	Disables tool length offsets
G54-G59	Work Offset Coordinates	Defines custom work offsets for fixture positioning
G68	Coordinate Rotation	Rotates the coordinate system for angled machining
G73	High-speed Peck Drilling	Performs rapid, efficient peck drilling
G81	Standard Drilling Cycle	Executes a single-hole drilling operation
G83	Deep Hole Peck Drilling	Uses multiple pecks to drill deep holes
G90	Absolute Positioning	All movements are based on a fixed coordinate system
G91	Incremental Positioning	All movements are relative to the tool's current position
G98	Return to Initial Plane	Sets the tool return level after drilling
G99	Return to R-Plane	Returns the tool to the clearance plane

4.22.1 Machine Commands (M-Codes). M-codes control non-movement machine functions such as spindle operation, coolant control and program execution, see the table below.

Table 4-12. M-Codes

Code	Function	Description
M00	Program Stop	Stops the machine until manually restarted
M01	Optional Stop	Stops only if the **Optional Stop** feature is active
M02	End of Program	Stops the program and resets it to the beginning
M03	Spindle On (clockwise)	Starts the spindle in a clockwise direction
M04	Spindle On (counterclockwise)	Starts the spindle in a counterclockwise direction
M05	Spindle Stop	Turns off the spindle
M06	Tool Change	Executes an automatic tool change
M07	Mist Coolant On	Activates mist coolant (if equipped)
M08	Flood Coolant On	Activates flood coolant
M09	Coolant Off	Turns off all coolant functions
M30	Program End and Reset	Ends a program and resets the machine
M98	Call Subprogram	Calls an external subroutine
M99	Subprogram Return	Returns to the main program after executing a subroutine

CHAPTER 5
MILLING MACHINES

5.1 GENERAL.

Milling is a machining process that removes material from a workpiece by feeding it against a rotating cutter. This process differs from lathe turning, where the workpiece rotates, and a stationary cutter is used. Milling machines operations range from creating flat surfaces to cutting complex shapes, such as dovetails and keyways, gears, threads, and grooves, as well as for operations such as drilling, boring, and tapping, and can be equipped with specialized attachments to enhance their functionality. These attachments can enable more complex tasks such as contouring, copying, or performing multi-axis machining. The flexibility of the milling machine allows it to be used in a variety of industries, from prototyping to largescale production. A milling machine, whether manual or Computer Numerical Control (CNC), consists of a motor-driven spindle that holds and rotates the milling cutter. The spindle provides the necessary rotational force for cutting, while the worktable positions the workpiece and feeds it past the cutter. In a manual milling machine, the operator adjusts the worktable's movement, typically along the X, Y, and Z axes. CNC milling centers are automated and controlled by a computer program, enabling more precise and complex movements along the 3rd, 4th, or 5th-axis. This automation allows CNC milling centers to perform highly detailed and intricate cuts, improving efficiency and accuracy, especially for complex geometries and high-volume production. Milling machines are highly versatile and can perform a wide range of operations.

5.2 SAFETY.

Safety is paramount when operating milling machines, as improper use can result in severe injuries. Operators must always adhere to safety guidelines set forth by Occupational Safety and Health Administration (OSHA) standards (29 Code of Federal Regulations (CFR) 1910.) and applicable Air Force Instructions (AFIs) and Air Force Manuals (AFMAN) (e.g., DAFMAN 91-203). Supervisors are responsible for maintaining manufacturer manuals and integrating them into on-the-job training and job safety training. In the absence of manufacturer guidance, a hazard analyses and local operating instructions must be developed. Operators must:

- Consider OSHA standards, relevant Air Force (AF) guidance and/or Original Equipment Manufacturer (OEM) recommendations for the specific machine and operation.
- Wear safety glasses, and may require face shields, hearing protection, gloves, and other necessary Personal Protective Equipment (PPE) to protect from flying debris, dust, and noise.
- Ensure all appropriate guards are in place to prevent accidental contact with rotating components.
- Follow lockout/tagout procedures to ensure the machine is powered off when performing maintenance.
- Keep the work area clean and free from clutter to avoid tripping hazards.
- Be familiar with the location and operation of emergency stop buttons and other safety features to quickly shut down the machine in case of an emergency.
- Remove loose clothing, jewelry, or secure long hair to prevent entanglement with moving parts to minimize the risk of serious injury.

5.3 TYPES OF MILLING MACHINES.

Milling machines, whether manual or CNC, are essential tools in manufacturing and machining processes in Aircraft Metals Technology (AMT). These machines are equipped with various components that allow precise control over the movement of the workpiece and cutting tools. In manual milling, the operator manually adjusts the machine's components to achieve the desired cut, making it a versatile but labor-intensive process. CNC milling machines, on the other hand, utilize computer numerical control to automate the cutting process. Designs are created in specialized software and translated into a series of commands that guide the machine's movements, resulting in highly accurate and repeatable cuts with minimal operator intervention. Understanding the different types of manual milling machines and their key components is crucial for safe operation and efficient machining.

5.3.1 Manual Milling Machines.

5.3.1.1 Vertical Milling Machine. (See Figure 5-1.) The most common mills found in AMT shops. The spindle axis is oriented vertically, meaning the cutting tool operates in a perpendicular direction to the surface of the workpiece. This setup is ideal for operations such as drilling, boring, and slot cutting. Vertical mills can be used for both small and large-scale tasks, depending on the machine's size and rigidity.

5.3.1.1.1 Applications. Milling flat surfaces, drilling holes, cutting slots, and creating contours.

5.3.1.2 Horizontal Milling Machine. (See Figure 5-2.) The spindle of a horizontal milling machine is mounted horizontally, which allows the cutting tool to move in a parallel direction to the surface of the workpiece. Horizontal mills are typically used for heavier, more aggressive cutting operations. These machines are often used for machining large, flat surfaces or cutting grooves in a workpiece.

5.3.1.2.1 Applications. Cutting keyways, slotting, and face milling for larger workpieces.

5.3.1.3 Universal Milling Machine. (See Figure 5-3.) A universal milling machine can perform operations in both horizontal and vertical orientations. It includes a swiveling table that can be set at various angles, making it more versatile than a standard vertical or horizontal milling machine. It is typically used for more complex milling tasks where both vertical and horizontal cuts are needed.

5.3.1.3.1 Applications. More complex geometries requiring both vertical and horizontal cutting planes.

TO-34-1-10-094

Figure 5-1. Vertical Mill

TO-34-1-10-095

Figure 5-2. Horizontal Mill

TO-34-1-10-096

Figure 5-3. Universal Mill

5.3.1.4 Milling Operations. Manual milling machines are composed of several key components that work together to perform precise machining operations. (See Figure 5-4.) Each part plays a critical role in ensuring accurate movement, proper cutting, and stability during operations. Understanding these components is essential for safe operation, efficient maintenance, and optimal performance. From the motor-driven spindle that powers the cutting tool to the worktable that positions the workpiece, every part of the machine is designed to work in harmony. Additionally, components like the knee, saddle, and column provide the necessary support and adjustability to accommodate different milling tasks. By examining the function and interaction of these parts, operators can better understand how to use the machine effectively and troubleshoot any issues that may arise during milling operations.

5.3.1.4.1 Spindle. The spindle is the central component that holds and rotates the milling cutter. It provides the necessary rotational force to drive the cutting tool during machining. Spindles in manual machines are usually powered by an electric motor. The spindle is often adjustable to accommodate different tool sizes and cutting operations. The spindle's function is to provide rotational motion to the cutter for material removal.

5.3.1.4.2 Table. The worktable is where the workpiece is mounted. It moves in multiple directions (typically along the X and Y axes), allowing the operator to position the workpiece relative to the cutter. The table's movement is manually controlled using hand wheels or cranks, which provide fine adjustments. The worktable supports the workpiece and allows for movement in different directions during machining.

5.3.1.4.3 Knee. The knee is a heavy, vertical component that supports the worktable and allows for vertical movement. It can be raised or lowered using a hand wheel to adjust the distance between the table and the cutter. The knee's adjustment is crucial for setting the correct depth of cut. The function of the knee is to provide vertical movement for the worktable.

5.3.1.4.4 Saddle. The saddle supports the knee and provides horizontal movement along the machine's axis. It can move back and forth to position the worktable along the X-Axis (left to right).

5.3.1.4.5 Column. The column is the vertical structure that houses the spindle and other key components. It is usually bolted to the base of the machine and provides structural rigidity. The column serves as a support for the spindle and other moving parts of the machine. The column functions to provide structural support and house essential machine components like the spindle.

5.3.1.4.6 Ram. The ram is an optional component on some milling machines. It extends horizontally from the column and can move in and out to position the spindle over the workpiece. In some cases, the ram can also rotate, giving additional flexibility in tool positioning. The ram provides additional flexibility for spindle positioning, particularly in horizontal milling machines.

5.3.1.4.7 Base. The base is the lower part of the machine, which provides stability and support for all the other components. It is usually made of cast iron to absorb vibrations and ensure accurate milling.

5.3.1.4.8 Quill. The quill is a hollow, tubular component in the vertical milling machine that houses the spindle. It allows the spindle to move up and down, providing vertical movement of the cutting tool. The quill is particularly useful for drilling operations, where controlled depth is required. The quill allows vertical movement of the spindle for drilling and other vertical cuts.

5.3.1.4.9 Power Feed Mechanism. The power feed mechanism is responsible for automatically moving the worktable during cutting operations. In manual mills, this mechanism is often used for feeding the workpiece in one direction (usually along the X or Y-Axis) to maintain consistent cutting speed and precision.

5.3.1.4.10 Digital Readout (DRO). The DRO is an electronic system that displays the real-time position of the worktable and spindle along the X, Y, and Z axes. It replaces traditional dials and gauges with digital displays for greater accuracy. The DRO includes linear scales attached to the machine's moving parts and a display unit showing precise position data. The DRO functions to provide precise, real-time measurements of the workpiece position, improving accuracy and efficiency in manual milling operations.

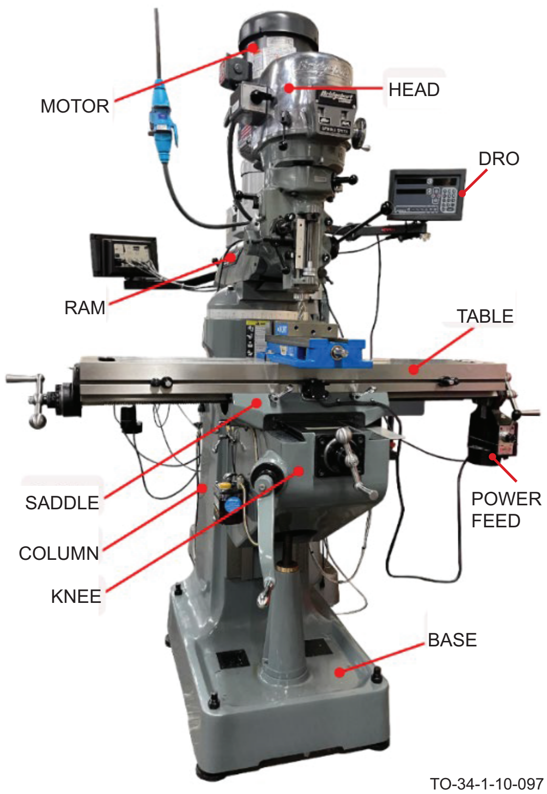


Figure 5-4. Manual Mill Components

5.3.2 CNC Milling Centers. (See Table 5-1.)

5.3.2.1 Vertical CNC Milling Center. (See Figure 5-5.) In a vertical CNC milling center, the spindle is oriented vertically, and the cutting tool operates in a perpendicular direction to the surface of the workpiece. This design is ideal for producing detailed features and complex shapes on flat surfaces.

5.3.2.1.1 Applications. Milling flat surfaces, drilling holes, creating slots, and producing detailed contours on workpieces. Used for complex, 3D parts geometries in various industries.

5.3.2.2 Horizontal CNC Milling Center. (See Figure 5-6.) A horizontal CNC milling center has the spindle mounted horizontally, allowing the cutting tool to move parallel to the workpiece surface. Horizontal DRO Power Feed milling machines are known for their ability to handle large parts and perform heavy-duty cutting operations. The horizontal orientation is well-suited for operations that require large surface areas to be machined.

5.3.2.2.1 Applications. Machining large, heavy workpieces, cutting keyways, and producing grooves or slots. Used for high-volume production of parts with complex features.

5.3.2.2.2 CNC Milling Center Categorization. CNC milling centers are categorized based on the number of axes they utilize, as detailed in the table below, with each configuration offering distinct capabilities, advantages, and limitations.

Table 5-1. Comparison of CNC Milling Center Configurations: 3-Axis, 4-Axis, 5-Axis

Feature	3-Axis CNC Mill	4-Axis CNC Mill	5-Axis CNC Mill
Axes of Movement	X, Y, Z (Linear)	X, Y, Z (Linear), A (Rotary)	X, Y, Z (Linear), A and B (Rotary)
Typical Work Holding	Vise, clamps, simple fixtures	Indexers, rotary tables, specialized fixtures	Trunnion tables, multi-axis vises, complex fixtures
Capabilities	Milling flat surfaces	All 3-axis capabilities	All 3 and 4-axis capabilities
	Drilling, boring	Machining multiple sides in one setup	Simultaneous 5-axis machining
	Cutting slots and grooves	Indexing for evenly spaced features	Machining highly complex geometries
	Simple contouring	Complex contouring and profiling	Undercuts and intricate features
Advantages	Most common and cost-effective	Increased efficiency and productivity	Highest level of precision and complexity
	Relatively simple to program and operate	Reduced setup times	Unmatched flexibility for intricate designs
	Versatile for a wide range of parts	Improved accuracy for multi-sided parts	Reduced need for multiple setups
Limitations	Limited to machining from one direction	Increased programming complexity	Most expensive CNC milling option
	Requires multiple setups for complex parts	Higher initial cost than 3-axis	Requires highly skilled programmers and operators
	Cannot create undercuts	Limited to 4-axis simultaneous movement	Longer programming and setup times
Typical Applications	Plates, brackets, simple molds	Gears, cams, turbine blades	Aerospace components, medical implants, molds with complex undercuts
	Parts with features accessible from one side	Parts with features on multiple sides	Highly complex parts requiring exceptional precision
	Prototyping and low-volume production	Medium to high-volume production	High-value, low-volume production

5.3.2.3 3-Axis CNC Milling Center. A 3-axis CNC milling center allows for movement along three axes: X (horizontal), Y (vertical), and Z (depth). The cutting tool moves in these three directions to perform basic milling operations. 3-axis mills are ideal for simpler, less complex operations and are the most commonly used type of CNC mill.

5.3.2.4 4-Axis CNC Milling Center. In a 4-axis CNC milling center, the workpiece is rotated on a 4th axis, in addition to the three linear axes of movement (X, Y, and Z). This provides the ability to machine features on multiple sides of a workpiece without having to reposition it manually, making the process faster and more efficient. Applications include milling parts with features on multiple sides or more complex geometries, such as rotary components, turbine blades, or aerospace parts.

5.3.2.5 5-Axis CNC Milling Center. A 5-axis CNC milling center provides movement along five axes: three linear axes (X, Y, and Z) and two rotational axes. This configuration allows the cutting tool to approach the workpiece from virtually any angle, making it ideal for producing highly complex and intricate parts. 5-axis machines are capable of simultaneous multi-axis machining, which is often required for advanced, high-precision parts. Applications include producing intricate, highly detailed, and complex parts that require simultaneous multi-axis movement.

TO-34-1-10-098

Figure 5-5. Vertical CNC Mill

TO-34-1-10-099

Figure 5-6. Horizontal CNC Mill

5.3.2.6 CNC Milling Center Varieties. CNC milling machines consist of a variety of components (Figure 5-7) that work together to execute precise machining operations. These components include mechanical, electrical, and software elements, each playing a crucial role in the machine's overall functionality. Understanding the key components of a CNC milling machine is essential for proper operation, maintenance, and troubleshooting.

5.3.2.6.1 Spindle. The spindle is the component that holds and rotates the cutting tool. It is one of the most critical components of a CNC milling machine because it directly impacts the machine's cutting speed, precision, and finish quality. Spindles in CNC mills are typically high-speed motors capable of reaching high Revolutions Per Minute (RPM). The spindle is responsible for driving the milling cutter during machining operations, making it crucial for operations such as drilling, boring, and slot cutting.

5.3.2.6.2 Table. The table is the flat surface on which the workpiece is secured during the milling process. It is typically made from heavy-duty materials such as cast iron or steel to ensure rigidity and stability during cutting. CNC milling machines often feature a worktable that moves in multiple directions (X, Y, and sometimes Z) to position the workpiece relative to the cutting tool. The table holds and positions the workpiece in relation to the cutting tool. In CNC milling, the workpiece is often clamped or fixtured to the table for precision cutting.

5.3.2.6.3 Axes (X, Y, Z). CNC milling machines operate along multiple axes. The most common configuration is a 3-axis system: X-axis: Horizontal movement (left to right), Y-axis: Horizontal movement (front to back), and the Z-axis: Vertical movement (up and down). These axes allow for precise control of the cutting tool's position and movement relative to the workpiece. The axes are responsible for movement of the tool or workpiece along each axis to perform various milling operations. More advanced CNC machines may include additional axis (4, 5, etc.) to provide more complex movement.

5.3.2.6.4 CNC Controller. The CNC controller is the **brain** of the CNC milling machine. It interprets the part program (often written in G-code or other machine language) and sends instructions to the machine's motors, allowing it to perform specific movements and operations. The controller manages the movement of the axes, spindle speed, and tool changes. Applications include managing the machine's operations and translating digital instructions into physical movements. It allows the operator to program and control complex machining tasks such as contouring, drilling, and tapping.

5.3.2.6.5 Servo Motors. Servo motors are electric motors that provide precise control of the machine's movements. These motors are used to drive the machine's axes (X, Y, Z) and the spindle. They are controlled by the CNC controller and offer precise position control, speed regulation, and high torque. The servo motor provides the movement required for machining operations. Servo motors ensure accurate positioning and smooth motion along the axes.

5.3.2.6.6 Linear Guides and Ball Screws. Linear guides and ball screws are mechanical components that enable smooth and precise movement of the CNC machine's table and axes. Linear guides provide low-friction movement along the rails, while ball screws convert rotary motion into linear motion with minimal backlash, ensuring precise movement and positioning. The linear guides and ball screws ensure smooth, accurate motion of the table and machine components. They help maintain the precision of the machine over long durations of use.

5.3.2.6.7 Tool Changer. A tool changer is an automated system that enables the CNC milling machine to automatically swap tools during a machining cycle. Tool changers vary in design, ranging from simple carousel-style systems to more complex robotic arm-style changers, depending on the machine's capabilities. The tool changer automatically changes tools to perform different operations without manual intervention, improving efficiency and reducing machine downtime.

5.3.2.6.8 Coolant System. The coolant system delivers coolant or lubricant to the cutting area to reduce friction and heat buildup during the milling process. Coolant is sprayed onto the cutting tool and workpiece to enhance tool life, improve the finish quality, and prevent thermal damage to the workpiece. The coolant systems cools the cutting tool, workpiece, and spindle during machining. It also helps in flushing away chips and debris from the cutting area.

5.3.2.6.9 Probing System. A probing system is used for automatic measurement and inspection of the workpiece during machining. The probe measures the dimensions of the workpiece and helps in part alignment, tool setting, and in-process quality control. The probing system automatically measures the workpiece to ensure dimensional accuracy and adjust machine parameters for precise machining. It can also be used for tool length measurement and setup verification.

5.3.2.6.10 Enclosure. The enclosure is a protective casing that surrounds the CNC milling machine, providing safety for the operator and protection for the machine's internal components. Enclosures are often made from steel and may include clear panels for visibility, along with doors for easy access. The enclosure protects the operator from flying debris and coolant, as well as preventing contamination of the machine's internal components. Enclosures also reduce noise levels during operation.

5.3.2.6.11 Clamping and Fixturing Systems. Clamping and fixturing systems are used to secure the workpiece on the table during machining. These systems can include a variety of tools such as vises, chucks, and custom fixtures designed to hold the workpiece securely in place. These systems hold the workpiece securely during machining to ensure accuracy and prevent movement that could affect the quality of the part. These systems can be adjusted and configured based on the size and shape of the workpiece.

5.3.2.6.12 Drive Mechanism. The drive mechanism consists of motors, gears, and belts that transmit power to the various moving parts of the CNC milling machine. This system enables the movement of the axes, the rotation of the spindle, and the operation of the tool changer. The drive mechanism delivers power to all necessary components, including the table, spindle, and tools, ensuring that all movements and functions of the CNC machine are powered and controlled effectively.

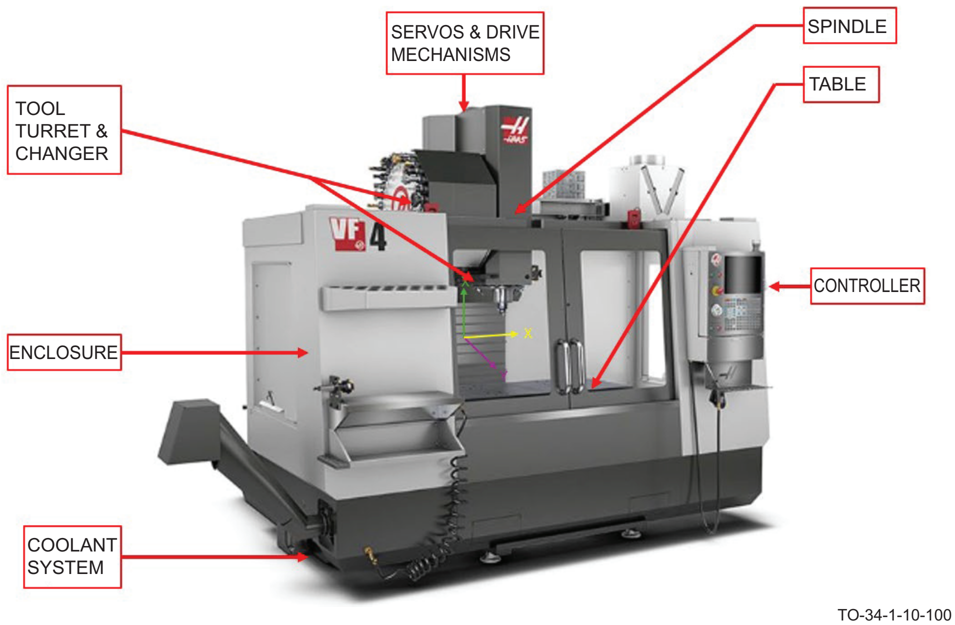


Figure 5-7. CNC Mill Components

5.4 TOOLING AND ACCESSORIES.

5.4.1 Milling Cutters. Milling cutters are classified by their geometry, application (type of milling operation), and the materials they are designed to cut. Understanding these classifications is essential for selecting the right cutter for any task, ensuring optimal performance and longer tool life.

5.4.1.1 Milling Cutter Classification. Milling cutters are generally classified based on the type of cutting action they perform during a milling operation. The two primary types of cutting action are radial and axial:

5.4.1.1.1 Radial Cutters. These cutters remove material by cutting radially from the center of the workpiece, often used in operations such as side milling, slotting, and profiling. Radial cutters cutting action is perpendicular to the axis of rotation. Radial cutters include tools like end mills, concave cutters, and radius cutters which cut in the direction of their rotation.

5.4.1.1.2 Axial Cutters. These cutters remove material along the axis of rotation, typically used in operations such as face milling and drilling. Axial cutters cutting action is parallel to the axis of rotation. Axial cutters include tools like shell mills, face mills, and fly cutter. The cutting edges are oriented perpendicular to the axis of rotation, and cutters like face mills and fly cutters operate axially.

5.4.1.2 Nomenclature of Milling Cutters. The nomenclature of milling cutters is a system of terms and definitions that describe the various parts and features of these tools. Understanding the nomenclature is essential for properly identifying, selecting, and using milling cutters for different operations. The nomenclature of milling cutters generally covers aspects such as their shape, cutting edges, mounting method, and materials used. Milling cutters are typically described using a combination of terms that relate to the following key components (Figure 5-8 and Figure 5-9):

5.4.1.2.1 Cutter Diameter. The diameter refers to the overall size of the milling cutter and is crucial in determining the size of the material being machined. Cutter diameter affects the cutting depth and the material removal rate.

5.4.1.2.2 Number of Teeth. The number of teeth on a milling cutter influences the cutting speed, finish quality, and material removal rate. More teeth generally result in a smoother finish, but fewer teeth may be more efficient for material removal due to increased chip space and reduced cutting forces, which helps prevent chip clogging and excessive heat buildup.

5.4.1.2.3 Cutting Edge. The cutting edge is the part of the milling cutter that engages with the workpiece to remove material. Different types of cutting edges are designed for specific operations (e.g., flat edges for face milling or curved edges for contouring). Single cut refers to a cutter with a single cutting edge, while multi-cut refers to cutters with multiple teeth or edges.

5.4.1.2.4 Face of the Cutter. The face of a cutter is the flat surface where cutting edges are arranged. It is the primary surface involved in material removal during face milling operations. A face mill has a cutting face designed for cutting across the surface of the material.

5.4.1.2.5 Body. The body is the main part of the milling cutter that houses the cutting edges. It is typically cylindrical or disc-shaped, and it may contain insert pockets, flutes, or grooves for chip removal. The body of an end mill is the portion that houses the cutting edges and fits into the tool holder.

5.4.1.2.6 Flutes. Flutes are the spiral or helical grooves that run along the body of the cutter. They help in removing the chips generated during the cutting process and allow for coolant flow. The number of flutes influences the cutter's performance. A 4-flute end mill has four helical grooves along the body.

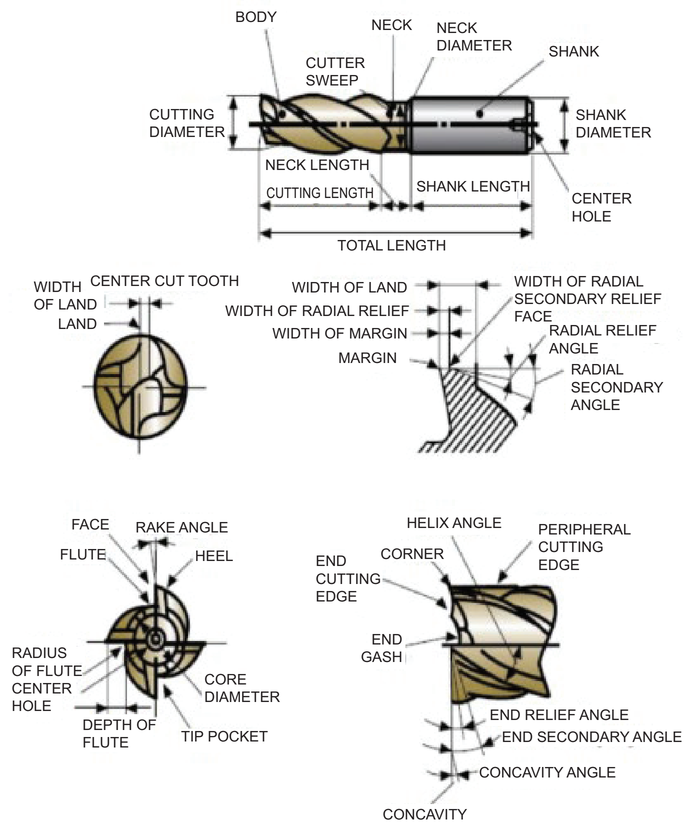
5.4.1.2.7 Shank. The shank is the portion of the cutter that fits into the tool holder or spindle of the milling machine. The shank's diameter is crucial for ensuring a secure fit in the holder, and it may be cylindrical, tapered, or threaded, depending on the machine type. A cylindrical shank is common for tool holders with collets, while a tapered shank might be used for machines with a Morse taper.

5.4.1.2.8 Hub (Insert Holder). In tools like indexable insert cutters, the hub refers to the portion of the cutter that holds the insert. The hub provides a secure seat for the inserts and facilitates their easy replacement when worn. A shell mill may have a central hub where the cutting inserts are mounted.

5.4.1.2.9 Insert. Inserts are replaceable cutting edges that are often made of hard materials like carbide or ceramic. Inserts are used in tools such as face mills and shell mills, allowing the user to change the cutting edge without replacing the entire tool. A face mill with carbide inserts refers to a milling cutter that uses carbide pieces as the cutting edges.

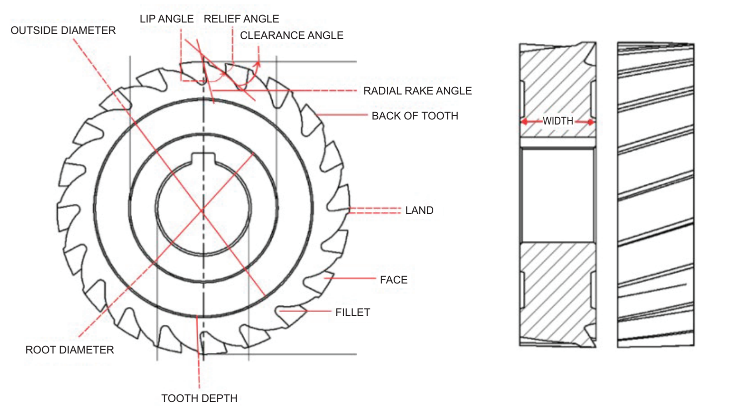
5.4.1.2.10 Lead Angle. The lead angle is the angle formed between the cutter's cutting edge and the surface of the workpiece. It is especially relevant for angular cutters, such as angle mills. A 45-degree lead angle refers to an angular cutter designed to engage with the workpiece at a 45-degree angle.

5.4.1.2.11 Clearance Angle. The clearance angle is the angle formed between the cutting edge and the surface of the workpiece that allows the cutter to clear the material without excessive rubbing. This angle is crucial for minimizing tool wear and achieving smooth cuts. The clearance angle of an end mill is typically between 5 and 15 degrees to ensure proper chip flow and reduce cutting forces.



TO-34-1-10-101

Figure 5-8. End Mill Nomenclature



TO-34-1-10-102

Figure 5-9. Plain Milling Cutter Nomenclature

5.4.1.3 Milling Cutters Shape and Purpose. In addition to these technical components, milling cutters are also classified and identified based on common terminology used to describe their shape and purpose:

5.4.1.3.1 End Mill. A cutter with cutting edges on both the end and the side, used for a wide range of operations such as slotting, profiling, and drilling.

5.4.1.3.2 Face Mill. A cutter designed with cutting edges on the face of the tool for facing operations, primarily used to machine flat surfaces.

5.4.1.3.3 Ball Nose Mill. A type of end mill with a hemispherical tip, commonly used for 3D contouring, profiling, and creating curved surfaces.

5.4.1.3.4 T-Slot Cutter. A cutter designed to create T-shaped slots, typically used in tool and die work.

5.4.1.3.5 Fly Cutter. A single-point cutter used for fine finishes on flat surfaces.

5.4.1.3.6 Indexable Cutters. Use replaceable inserts to provide multiple cutting edges, improving tool life and reducing costs. These cutters offer flexibility, precision, and efficiency, making them ideal for various milling operations such as face milling, slotting, and contouring.

Figure 5-10. Types of Milling Cutters

5.4.1.4 Selecting a Milling Cutter. When selecting a milling cutter, it is important to consider the following specific factors:

5.4.1.4.1 Coating. Some cutters are coated with materials (Table 5-2) like titanium nitride or titanium aluminum nitride to enhance wear resistance and heat tolerance. The coating may be specified as part of the cutter's nomenclature (e.g., **TiN-coated end mill**).

5.4.1.4.2 Material. The material of the cutter impacts its performance and suitability for different applications. (See Table 5-3.) Common materials for milling cutters include High-Speed Steel (HSS), carbide, cobalt alloys, and ceramics.

5.4.1.4.3 Ferrous Materials. Cutters designed for machining ferrous materials, such as steel and cast iron, are typically made from HSS or carbide, as these materials are harder and require durable cutting edges.

5.4.1.4.4 Non-Ferrous Materials. Milling cutters for non-ferrous materials, such as aluminum, brass, and copper, often have specialized coatings or are made from carbide to handle softer yet often more challenging-to-machine materials due to their tendency to stick to tools.

5.4.1.4.5 Hard Materials. For machining hard materials, such as hardened steels and titanium alloys, cutters made from carbide or ceramic materials are used. These tools are engineered to withstand high temperatures and wear.

Table 5-2. Milling Cutter Coatings

Coating	Description	Applications
Titanium Nitride	A gold-colored coating that improves hardness, reduces friction, and increases wear resistance.	Used for general-purpose milling, drilling, and turning on softer materials like aluminum, steel, and titanium alloys.
Titanium Carbonitride	A harder, wear-resistant coating that offers improved tool life and reduced friction.	Common in finishing operations on high-speed steels, stainless steels, and cast iron.
Titanium Aluminum Nitride	Offers high heat resistance and hardness, making it suitable for high-speed cutting.	Ideal for machining difficult-to-cut materials such as stainless steel, super alloys, and high temperature alloys.
Cobalt	Cobalt coatings increase toughness and heat resistance, commonly used in high-performance applications.	Suitable for high temperature cutting in tough materials like stainless steel and super alloys.
Chromium Nitride	Provides high wear resistance and corrosion resistance, as well as low friction.	Used for high-speed and high-precision machining on cast iron, stainless steel, and aluminum.
Zirconium Nitride	Provides high wear resistance and high hardness while maintaining a smooth cutting edge.	Used for finishing and high-performance machining of steel, stainless steel, and hardened steels.
Aluminum Titanium Nitride	Offers excellent high-temperature stability, wear resistance, and is highly effective in difficult cutting operations.	Ideal for heavy-duty milling, drilling, and turning of tough materials like high-carbon steels, stainless steels, and titanium alloys.
Titanium Diboride	Known for its hardness and ability to reduce friction, providing enhanced wear resistance.	Used in high-performance cutting of aluminum, non-ferrous metals, and composites.

Table 5-3. Milling Cutter Materials

Material	Description	Applications
Carbide	Commonly used for its hardness, wear resistance, and ability to withstand high cutting speeds.	Widely used for a variety of materials, from steel to non-ferrous metals.
Cermet	A composite material combining ceramic and metallic elements, used for high-precision finishing operations with excellent wear resistance.	Ideal for high-precision finishing operations.
Ceramic	Extremely hard and resistant to high temperatures, making it ideal for machining hard materials like cast iron and hardened steels.	Used for machining hard materials such as cast iron and hardened steels.
Cubic Boron Nitride (CBN)	Used for machining hardened steels and other difficult-to-machine materials due to its hardness and heat resistance.	Applied in machining hardened steels and other tough materials.

5.4.1.5 Specific Milling Operation. Another way to classify milling cutters is based on the specific milling operation (Table 5-4) they are used for. This classification allows for more direct matching of the tool to the task at hand.

Table 5-4. Milling Cutter Operation Descriptions

Milling Operation	Description	Advantages	Applications
Face Milling	Flattens workpiece surfaces.	High removal rate and smooth finish.	Cylinder heads, heat sinks.
Plain Milling	Makes the workpiece flat and creates contours.	Uniform removal; light machining.	Outer layer machining.
Side Milling	Shapes sides of the workpiece.	Forms flat profiles, and grooves.	Suspension mounts and medical implants.
Straddle Milling	Mills two parallel surfaces.	Efficient parallel slotting.	Jigs, fixtures, and gears.
Gang Milling	Uses multiple cutters for complex features.	Multiple operations simulta-neously.	Engine blocks and transmission housing.
Angle Milling	Cuts at specific angles.	Precision angular cuts.	Chamfers, T-slots.
Form Milling	Creates irregular contours.	Custom milling for intricate shapes.	Turbine blades and orthopedic impants.
End Milling	Feeds work into the end mill.	Intricate profiles, and smooth finishes.	Detailed machining tasks.
Saw Milling	Large cutter for slots and parting-off.	Effective slotting.	Slots, dividing workpieces.
Gear Milling	Specialized for gears.	High accuracy on gear teeth.	All gear types.
Thread Milling	Cuts internal and external threads.	Useful for large holes.	Engines, assembly-required products.
Computer-Aided Manufacturing (CAM) Milling	Milling for CAM compo-nents.	Precise material removal.	CAMs in mechanical systems.

5.4.1.6 Proper Care and Maintenance of Milling Machine Cutters. Proper care and maintenance of milling machine cutters are essential for ensuring their longevity, performance, and accuracy. Regular maintenance prevents premature wear, reduces downtime, and ensures that the cutters perform optimally. Below are key considerations for maintaining milling machine cutters:

5.4.1.6.1 Cleaning. After each use, milling cutters should be cleaned thoroughly to remove chips, coolant, and other debris. Any build-up of material can affect cutting performance and cause the cutter to overheat.

Procedure

Use a soft brush, and a cleaning solution if required, to remove debris from the cutter's surface and teeth. For stubborn residue, a non-abrasive cloth or brush with mild solvent may be used.

Frequency

Clean cutters after every use and before stowing, or more frequently depending on the material being machined and the level of contamination.

5.4.1.6.2 Inspection. Regular inspection (prior to use) is essential to detect any wear, cracks, or damage to the cutting edges. Catching issues early helps prevent further damage to the cutter and workpiece.

Procedure

Sharpen the cutter's edges using appropriate grinding machines or tools. Regrind the cutting tool to restore its original shape and geometry, ensuring the proper cutting angle is maintained.

Frequency

Sharpen cutters based on usage; some cutters may require sharpening after a specific number of hours of operation or when performance decreases.

5.4.1.6.3 Storage. Proper storage of milling cutters helps prevent damage and corrosion when not in use. Cutters should be kept in a clean, dry, and controlled environment (storage cabinets and dispensers) and to the maximum extent possible, in their original container.

Procedure

Store milling cutters in a designated tool storage area or container, away from moisture and contaminants. Use toolboxes, racks, or custom holders to keep cutters organized and protected from physical damage.

Frequency

Store cutters immediately after use to prevent exposure to environmental factors that can lead to corrosion or dulling.

5.4.1.6.4 Lubrication. Proper lubrication helps reduce friction between the cutting tool and workpiece, extending the cutter's lifespan and ensuring smoother operation.

Procedure

Apply appropriate lubricants, such as cutting oils or sprays, as recommended by the cutter manufacturer. Regularly check and replace lubrication in coolant systems.

Frequency

Lubricate cutters as needed, particularly during extended milling operations or when working with tougher materials.

5.4.1.6.5 Proper Handling. Mishandling milling cutters during installation, use, or storage can result in damage, misalignment, or inaccurate cutting.

Procedure

Always handle cutters with care, using proper tools (such as wrenches or holders) to avoid dropping or damaging the cutter. Wear gloves to prevent oils or contaminants from getting on the tool surfaces.

Frequency

Handle cutters carefully every time they are installed or removed from the milling machine.

5.4.2 Cutter Arbor.

CAUTION

A cutter should never be operated in reverse. Due to the clearance angle, running the cutter backward causes excessive friction and heat, which can lead to premature wear or even cutter breakage. Failure to comply, could result in damage to, or destruction of, equipment or loss of mission effectiveness.

NOTE

Always ensure the cutter does not contact hard surfaces like the vise jaws, chuck, clamping bolts, or nuts to avoid damage to both the cutter and the workpiece.

A cutter arbor is a shaft or spindle used to mount and secure milling cutters, such as saw blades, face mills, or slab mills, on a milling machine. It provides the necessary support and alignment for the cutter to operate effectively. The arbor is typically secured in the machine spindle and holds the cutting tool in place using a combination of tapers, tangs, and other clamping mechanisms. Choosing the correct arbor is essential for optimal tool performance and machine accuracy.

5.4.2.1 General. Arbors often feature specific tapered connections to ensure secure mounting in the spindle of a milling machine. (See Table 5-5.) The taper facilitates a tight, self-locking fit that ensures precision and reduces vibration during cutting operations:

5.4.2.1.1 Morse Taper. One of the most common types of tapers used for arbors. The taper is a conical shape that provides a secure fit when inserted into the machine spindle.

Applications

Suitable for smaller, lighter milling cutters and general-purpose machining. Example Sizes: MT1 through MT6.

5.4.2.1.2 Brown and Sharpe (B&S) Taper. A taper designed with a slightly steeper angle than Morse tapers. This type of taper offers a higher clamping force for secure mounting.

Applications

Frequently used for arbors in vertical and horizontal milling machines. Example Sizes: B&S No's 9, 7, 5, 3, 2, 1.

5.4.2.1.3 Boring Taper. A self-locking, high-precision taper with a slightly different design compared to traditional tapers. It includes a flange on the tool holder for more rigid connections, often incorporating a locking mechanism that improves tool holding during high-speed cutting.

Applications

Commonly used for high-speed, high-accuracy machining in CNC milling and boring operations. It is suitable for holding end mills, drills, boring tools, and other milling tools. These tapers are designed for CNC machines requiring greater precision and rigidity. Example Sizes: BT30, BT40, BT50, BT60.

5.4.2.1.4 V-Flange (Caterpillar (CAT)). A V-shaped flange on the tool holder, which engages with the spindle's clamping mechanism. This design helps to provide secure and rigid tool holding, preventing vibration and ensuring consistent tool performance during machining.

Applications

The CAT taper is commonly used to hold face mills, end mills, drills, tapping tools, and other tools for CNC milling operations. It is popular for high-speed machining and heavy-duty cutting operations. Example Sizes: CAT30, CAT40, CAT50, CAT60.

5.4.2.1.5 R8 Taper. Commonly used in smaller machines and CNC mills, the R8 taper offers a reliable, self-locking fit.

Applications

Used in tools for vertical milling machines.

Example Sizes

R8 taper is typically used with the R8 collet system.

Table 5-5. Arbor Tapers, Applications and Features

Taper Type	Taper Angle	Primary Uses	Applications	Key Features
Morse Taper	1.5 degree to 2.5 degree	General-purpose milling, drilling and turning	Commonly used in smaller machines and manual mills	Most widely used taper for small tools
			Light to moderate-duty operations	Available in multiple sizes (MT1 to MT6)
			Often found in smaller, lighter machines	Easy to install and remove

Table 5-5. Arbor Tapers, Applications and Features - Continued

Taper Type	Taper Angle	Primary Uses	Applications	Key Features
B&S Taper	1.5 to 2.5 degree	Precision machining in vertical and horizontal mills	Common in vertical and horizontal milling operations	Higher clamping force than Morse taper
			Often used for precision machining in aerospace and automotive	Used for heavier-duty machining
			Machining large components and tool changers	Found in various sizes (B&S No. 9 to BB&S No. 1)
Boring Taper	7 degree (taper angle)	High-precision, high-rigidity machining	CNC milling, drilling, and boring operations	Self-locking taper with flange for secure fit
			Suitable for heavy-duty and high-speed machining	Common in CNC applications, especially for boring tools
			Used for end mills, drills, and other high-precision tools	Common sizes: BT30, BT40, BTS0, BT60
CAT (V-Flange)	7 degree (taper angle)	High-speed CNC milling and heavy-duty cutting	CNC milling, face mills, drills, and tapping operations	V-flange design for quick tool changes
			Aerospace, automotive, and industrial applications	Quick-change system (no tools needed for tool swaps)
			Heavy-duty cutting, especially for large parts	Common sizes: CAT30, CAT40, CAT50, CAT60
R8 Taper	7.5 degree (taper angle)	Small to medium-sized CNC mills and vertical mills	Vertical milling machines, small CNC machines	Self-locking design with simple tool changes
			Ideal for general-purpose machining	Widely used in smaller milling machines
			Used for end mills, drills, and collet: systems	Very popular in hobbyist and light industrial machines

5.4.2.2 Types of Cutting Tools. Cutter arbors hold various types of cutting tools, depending on the machine's requirements and the operation being performed. The following are the most common types (Figure 5-11) of arbors in AMT shops:

5.4.2.2.1 Face Mill Arbor. A face mill is used for milling large flat surfaces and is typically held on a milling arbor with a specific taper and securing mechanism. These mills usually have multiple inserts on the periphery of the tool body for efficient material removal.

5.4.2.2.2 Saw Blades. Circular saw blades are mounted on arbors to cut through workpieces with precision. The arbor holds the saw blade securely in place with a drive tang and taper connection. These blades typically feature carbide or high-speed steel teeth for cutting through various materials like metal, wood, or plastic.

5.4.2.2.3 End Mills. Arbors are used to hold end mills for various milling operations, including slotting, profiling, and cutting shapes. End mills are often held on a special arbor that accommodates their specific taper or R8 connection.

5.4.2.2.4 Drill Chucks. Arbors with drill chucks are used to hold smaller drill bits, making them essential for drilling holes in materials during the milling process. These are typically used for drilling operations in conjunction with milling setups.

5.4.2.2.5 ER Collet Arbors. ER collet arbors are used to hold tooling that requires high precision, such as end mills, drills, and taps, ensuring secure tool clamping for milling operations. These arbors are commonly used to hold end mills, drills, and taps in CNC and manual milling setups, providing excellent concentricity and grip for precise machining tasks.

Figure 5-11. Typical CNC Mill Arbors

5.4.3 Milling Machine Vices. (See Figure 5-12.) Milling machine vices are essential tools used to securely hold workpieces during milling operations. (See Table 5-6.) They provide a stable and precise clamping system, ensuring that the workpiece remains fixed during the cutting process, reducing the risk of errors or accidents. There are several types of vices designed for different applications, each with unique features for securing various workpiece shapes and sizes.

5.4.3.1.1 Types of Milling Machine Vices.

5.4.3.1.1.1 Plain Milling Vice. The plain milling vice is the most commonly used type of vice in milling. It is designed to hold workpieces flat and securely in place. The vice has a fixed jaw and a movable jaw, with the workpiece placed between the two. The vice can be mounted directly onto the milling table. The plain milling vice is typically used for general-purpose milling operations, such as milling flat surfaces and drilling.

5.4.3.1.1.2 Universal Milling Vice. The universal milling vice is a more versatile option that allows the workpiece to be positioned at various angles. It has a swivel base that can rotate around a central axis, making it easier to perform angled cuts without re-positioning the workpiece manually. The universal milling vice is ideal for operations requiring angular cuts, such as beveling or milling at various angles.

5.4.3.1.1.3 Angle Milling Vice. The angle milling vice is designed to hold workpieces at a specific angle to the milling machine's table. This vice allows for accurate angled cuts without the need for additional fixtures. The angle milling vice is primarily used for milling parts that need to be held at a fixed angle, such as in the automotive or aerospace industries.

5.4.3.1.1.4 Clamping Plates. Clamping vices are used in combination with clamps and angle plates to hold complex or irregularly shaped workpieces. These vices are typically designed to accommodate non-standard shapes. Clamping plates are used in conjunction with clamps and angle plates for non-standard or awkwardly shaped workpieces, often in tool and die making.

5.4.3.1.1.5 Collet Vices. Collet vices use collets to hold round or cylindrical workpieces. The collet, when tightened, provides a firm grip on the workpiece, ensuring minimal movement during milling. Collet vices are most commonly used for holding cylindrical materials such as rods or shafts.

5.4.3.1.1.6 Hydraulic Milling Vices. Hydraulic vices use hydraulic pressure to clamp the workpiece, offering high clamping force with minimal manual effort. These vices are often used in automated CNC operations. Hydraulic milling vices are typically used in high-volume production environments where consistent and high clamping force is required.

5.4.3.1.2 Considerations for Choosing a Milling Vice.

5.4.3.1.2.1 Workpiece Size. The size of the milling vice should be chosen based on the size of the workpiece to ensure proper clamping and stability.

5.4.3.1.2.2 Material. Vices made from materials such as cast iron or steel offer durability and strength. Cast iron is often preferred for general milling tasks, while steel vices are used for heavy-duty applications.

5.4.3.1.2.3 Precision. For high-precision work, such as in aerospace or medical device manufacturing, a vice with high clamping accuracy and rigidity is essential. Vices designed for CNC machines often feature very high precision to ensure minimal deviation during machining.

5.4.3.1.2.4 Jaw Type and Pressure. Different vices feature various jaw designs (e.g., parallel, serrated) for holding specific materials. The pressure applied during clamping should also be considered to avoid damaging the workpiece while ensuring it stays in place during milling.

5.4.3.1.2.5 Jaw Width and Opening. Select a vice with jaws wide enough to accommodate the workpiece comfortably. The maximum opening should be sufficient to hold the part securely.

Table 5-6. Types of Milling Vices

Vice Type	Features	Common Applications
Plain Milling Vice	Fixed jaws, basic design	General-purpose milling, drilling, and light machining
Universal Milling Vice	Swivel base, adjustable angle	Angled milling cuts, complex shapes, bevels, and slots
Angle Milling Vice	Fixed angle design for precise angled cuts	Precision angle milling, automotive, aerospace industries
Clamping Plates	Adjustable clamps, used with angle plates	Non-standard and irregularly shaped workpieces, tool making
Collet Vice	Collets for clamping round workpieces	Cylindrical parts such as rods, shafts, or tubing
Hydraulic Milling Vice	Hydraulic clamping for high force and efficiency	High-volume CNC operations, mass production

Figure 5-12. Typical Milling Machine Vices

5.4.4 Milling Machine Attachments and Accessories. (See Figure 5-13.) The milling machine attachment and accessories significantly enhance the versatility and functionality of both manual and CNC milling operations. These tools allow for more complex machining tasks, greater accuracy, and improved efficiency in a variety of settings. They range from devices that help secure workpieces to those that automate and streamline the milling process. Some attachments extend the machine's capability to perform multi-axis machining, while others ensure precise and consistent results across different applications. The following are some of the most commonly used attachments and accessories for manual and CNC milling machines:

5.4.4.1 Rotary Table. A versatile accessory that enables the workpiece to rotate for precise angular machining. They allow for milling of complex geometries, such as circular patterns, by rotating the part while the tool remains stationary. These are essential for 4th-axis machining on both manual and CNC machines.

5.4.4.2 Indexing Head. Also known as rotary indexers, these enable the operator to rotate the workpiece in fixed increments for repetitive milling operations. Indexing heads are used for tasks like gear cutting or drilling holes in circular patterns, they help achieve precise angular positions for consistent results.

5.4.4.3 Adjustable Angle Plates. Allows the workpiece to be set at precise angles to the milling table, enabling the machining of parts with angular cuts. These plates can be adjusted to different angles for creating complex geometries and improving machining flexibility.

5.4.4.4 Vertical and Horizontal Milling Attachments. Converts a horizontal milling machine into a vertical one, offering the ability to perform a wide range of operations, including drilling and face milling. Conversely, horizontal milling attachments extend the functionality of vertical mills for more aggressive cutting and larger surface areas.

5.4.4.5 4th-Axis Head. A CNC 4th-axis attachment adds rotational capability to a CNC milling machine, enabling the workpiece to rotate on an-axis in addition to the standard X, Y, and Z axes. This attachment is particularly useful for complex parts, allowing for multi-sided machining in a single setup, and increases efficiency by consolidating machining steps into a single setup.

5.4.4.6 Trunnion Table. Specialized rotary tables used in CNC machines for 5-axis machining. These attachments allow for multi-angle operations, facilitating the machining of complex parts with multiple sides in one continuous setup.

5.4.4.7 Mill Stops. Provides consistent and repeatable positioning of the workpiece on the milling machine. They ensure that the tool stops at the same point each time, helping to maintain accuracy for processes such as drilling and tapping.

5.4.4.8 Parallels and Parallel Savers. Precision tools used to raise the workpiece off the milling table, ensuring proper clearance for the cutting tool and maintaining alignment during machining. Parallel savers are used to reduce wear on parallels, extending their lifespan and ensuring consistent results.

5.4.4.9 1-2-3 Blocks. Precision machined blocks with exact dimensions that are used for setting and aligning the workpiece in the milling machine. They ensure accurate setups and can be stacked to create different heights for various machining operations.

5.4.4.10 Types of Material Clamping. Crucial for securing the workpiece during milling operations. Common methods include vices, clamps, magnetic chucks, and vacuum fixtures. The choice of clamping method depends on the material and the type of operation being performed, ensuring the workpiece remains firmly in place while minimizing movement during machining.

5.4.4.11 Offset Boring Heads. Fine-tunes the diameter of holes with high precision. These tools are adjustable, allowing for fine adjustments to the cutting tool's offset, enabling machinists to achieve exact hole sizes for tight-tolerance applications.

5.4.4.12 Touch Off Blocks and Edge Finders. Sets the reference point for the tool, ensuring precise positioning at the beginning of the milling operation. Edge finders help locate the edges of the workpiece with high accuracy, allowing for quick and reliable setups.

5.4.4.13 Automatic Tool Changers and Pneumatic Draw Bar/Tool Changers. Commonly found in CNC machines and allow for the automatic swapping of tools, reducing setup time and improving productivity. Pneumatic drawbars are used to automatically release or secure the tool in place without the need for manual intervention.

5.4.4.14 Digital Readouts. Electronic systems that provide precise measurements of the milling machine's movements along the X, Y, and Z axes. Digital readouts enhance accuracy by allowing machinists to monitor real-time positioning and avoid errors during milling operations.

5.4.4.15 Cold Air Guns. Cools the cutting tool and workpiece during high-speed operations typically from -20 to 0 degrees Fahrenheit (°F). By blowing compressed air directly at the cutting zone, these tools help reduce heat build-up, preventing material deformation, increasing tool life during precision milling, and, in some cases, improved surface finishes.

Figure 5-13. Typical Milling Attachments and Accessories

5.5 GENERAL MILLING OPERATIONS - MANUAL.

Manual machining operations are fundamental skills in the manufacturing industry, providing a foundation for understanding more complex CNC machining processes. Manual milling machines offer operators direct control over cutting parameters and tool movement, making them indispensable for small production runs, custom parts, and precise adjustments. In this section, we will explore common manual machining operations with detailed instructions, practical examples, and visual aids to ensure clarity and precision.

5.5.1 Tramming the Milling Machine Head. This process is crucial for ensuring accuracy in machining operations. This procedure aligns the head of the mill to be perpendicular to the X and Y axes of the table, vice, or fixture. Correct tramming guarantees that cutting tools are properly aligned with the workpiece, preventing issues such as uneven surfaces or irregular cutting patterns. Inaccurate alignment can lead to poor quality results, so proper tramming is essential for maintaining precision and ensuring that milling processes produce uniform and accurate parts. Follow these general steps to tram the head:

NOTE

Tramming should be accomplished prior to the use of a milling machine to ensure strict and critical aircraft and support equipment tolerance are met.

5.5.1.1 Prepare the Machine.

a. Ensure the machine is powered off and the table is clean.

b. Place the machine in the neutral position.

5.5.1.2 Dial Indicator Sweep Method.

a. Mount a dial indicator in a collet in the spindle.

b. Lower the spindle using the quill feed handle until the indicator probe touches the table.

c. Preload the indicator by raising the table 0.005 to 0.010 inch.

d. Set the indicator to zero.

e. Sweep the dial indicator in a circular path (6 inch radius) by rotating the spindle or quill.

f. Observe the dial indicator readings as the spindle is rotated.

g. If the reading on the dial indicator stays at zero, the spindle is aligned.

h. If the reading is not zero, continue tramming the head as shown below.

5.5.1.3 Dual Indicator Spindle Square Method.

a. Mount the dual indicator spindle square in a collet in the spindle.

b. Lower the spindle using the quill feed handle until the indicator probe touches the table.

c. Preload the indicator by raising the table 0.005 to 0.010 inch.

d. Set each of the indicators to zero.

e. Slowly rotate the spindle by hand, observing the readings on both indicators.

f. If the reading on the dial indicator stays at zero, the spindle is aligned.

g. If the reading is not zero, continue tramming the head.

5.5.1.4 Tramming the X-Axis.

a. Loosen the X-axis bolts on either side of the mill saddle.

b. Re-tighten them by hand plus a 1/4 of a turn.

c. Position the indicator to the rear of the table (zeroed and preloaded at 0.005 to 0.010 inch).

d. Rotate the spindle 180 degrees so the indicator is now on the front of the table.

e. Note the direction that the dial rotates to determine the direction that the mill head needs to travel.

- A clockwise movement requires that the mill head will need to be adjusted up.
- A counterclockwise reading requires that the mill head will need to be adjusted downward.

f. Adjust the head so that half the difference between the back and front measurements is achieved.

- For example, if the rear reading is zero and the front reading is 0.020 inch, adjust the mill head so that the dial reads 0.010 inch closer to zero.

g. Re-zero the indicator in the same location as first accomplished.

h. Continue to adjust until the difference between the front and rear is no greater than 0.001 inch

NOTE

Do not fully tighten one bolt at a time as this will cause a change in the head alignment.

i. Tighten all head bolts evenly and in a clockwise motion.

j. Recheck the measurement between the front and rear of the table.

5.5.1.5 Tramming the Y-Axis:.

a. Loosen the Y-axis bolts on front of the mill head.

b. Re-tighten them by hand plus a 1/4 of a turn.

c. Position the indicator to the side of the table (zeroed and preloaded at 0.005 to 0.010 inch).

d. Rotate the spindle 180 degrees so the indicator is now on the opposite of the table.

e. Note the direction that the dial rotates to determine the direction that the mill head needs to travel.

- A clockwise movement requires that the mill head will need to be adjusted up.
- A counterclockwise reading requires that the mill head will need to be adjusted downward.

f. Adjust the head so that half the difference between the back and front measurements is achieved.

- For example, if the rear reading is zero and the front reading is 0.020 inch, adjust the mill head so that the dial reads 0.010 inch closer to zero.

g. Re-zero the indicator in the same location as first accomplished.

h. Continue to make adjustments until the difference between the front and rear is no greater than 0.001 inch

i. Tighten all head bolts evenly and in a clockwise motion.

NOTE

Do not fully tighten one bolt at a time as this will cause a change in the head alignment.

j. Recheck the measurement between the front and rear of the table.

5.5.1.6 Indicating a Vice, Fixture, or Material. Indicating a vice, fixture, or material:

a. Tighten the t-bolt or clamps snug to allow for adjustment and movement.

b. Install a dial indicator in a collet in the spindle with the probe facing away from the operator.

c. Lower the spindle and position the table's so the fixed jaw on the vice is touching the indicator.

d. Set the indicator to zero.

e. Run the indicator across the vice's face with the cross feed.

f. The indicator will stay at zero if the vice is squared.

g. If the indicator does not stay at zero, realign the vice by lightly tapping with a soft hammer until the indicator reads half of its previous value.

h. Repeat the process until the dial indicator shows zero through a complete travel from one side of the vice to the other.

i. Fasten the T-bolts securely, while not changing the orientation of the vice.

j. Recheck the alignment of the vice.

5.5.2 Direction of Cutter Rotation. The direction of cutter rotation relative to the feed direction is a fundamental consideration in manual milling that significantly impacts machining outcomes. It influences the quality of the surface finish, the wear and life of cutting tools, and the overall safety and stability of the milling process. (See Table 5-7.) The choice between climb and conventional milling depends on the specific operation, material properties, and machine rigidity, as each method offers unique advantages and challenges. Understanding these differences is essential for achieving precision and efficiency in milling applications.

- Climb Milling: Climb milling, also known as down milling, is a machining technique where the cutter rotates in the same direction as the feed. (See Figure 5-14.) This synchronized movement results in the cutter **climbing** into the workpiece material. This method produces a smoother finish, reduces tool wear, minimizes work hardening, and requires a rigid machine and setup to avoid chatter or backlash.

- Conventional Milling: The cutter rotates against the feed direction. (See Figure 5-15.) It is more forgiving on older or less rigid machines and better for removing hard surfaces or scale.

Table 5-7. Comparison of Climb Milling vs. Conventional Milling

Category	Climb Milling	Conventional Milling
Tool Rotation	Rotates in the same direction as the feed of the workpiece.	Rotates opposite to the feed direction.
Surface Finish	Provides a superior surface finish.	Typically produces a rougher surface.
Tool Wear	Less tool wear and reduced cutting forces.	Higher tool wear due to increased cutting forces.
Machine Requirements	Requires a rigid setup to prevent backlash.	More forgiving on less rigid setups.
Ideal Use	Best for finishing operations and machining non-ferrous metals.	Suitable for roughing and materials prone to hardening.
Control and Stability	May cause workpiece pulling if backlash exists.	Better control, less likely to lift the workpiece.

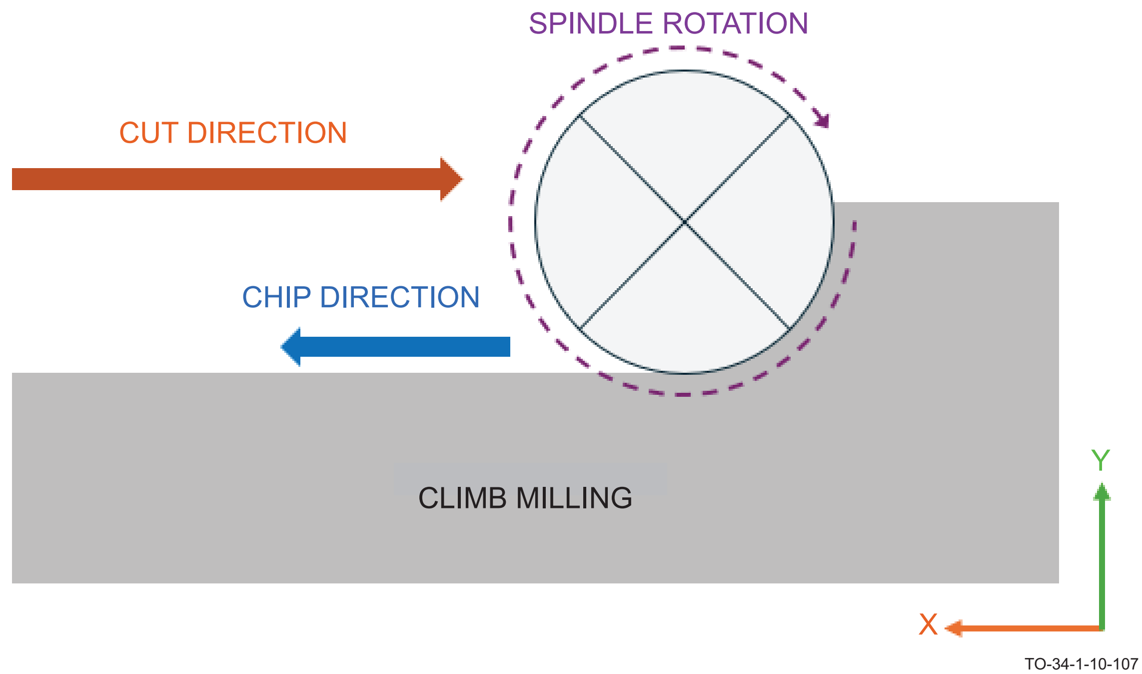


Figure 5-14. Climb Milling

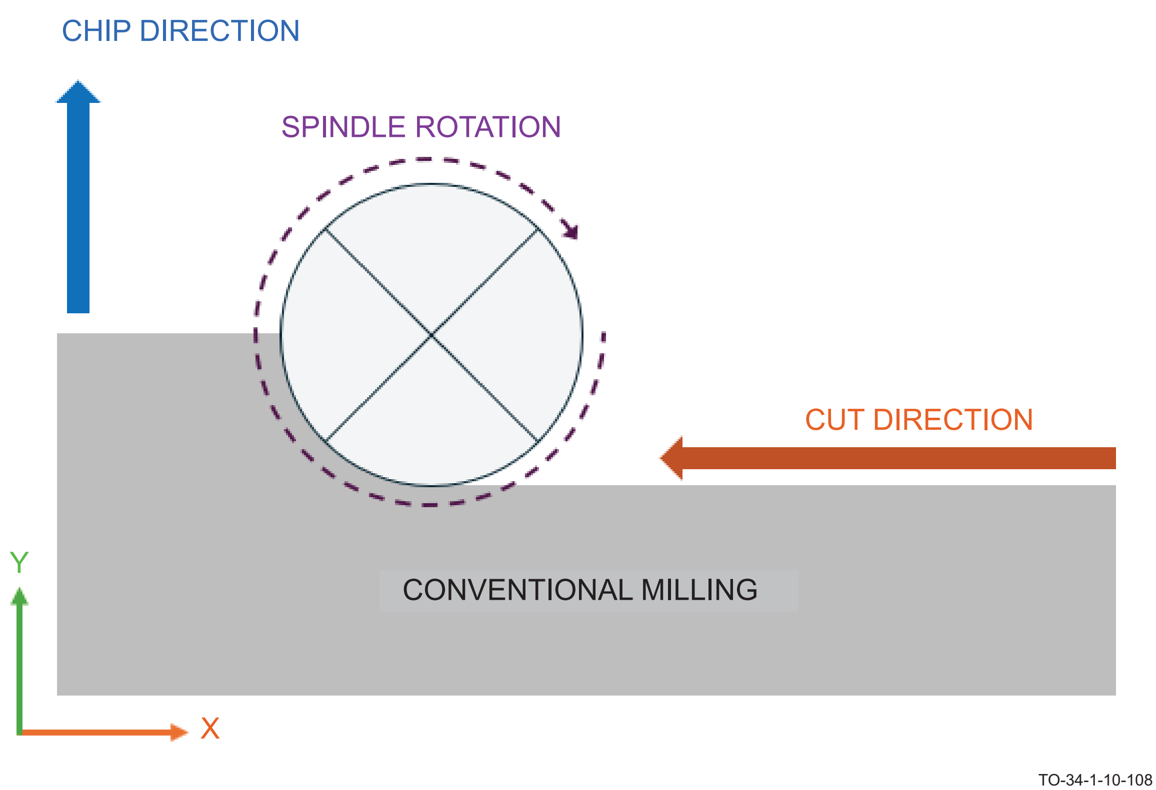


Figure 5-15. Conventional Milling

5.5.3 Speeds and Feeds. Speeds and feeds are critical parameters in machining that influence the quality, efficiency, and tool life of a cutting operation. These parameters define how quickly the cutting tool moves through the material (feed rate) and how fast it rotates (spindle speed). Speeds and feeds play a crucial role in determining the surface finish in machining. High spindle speeds can reduce tool marks and improve the surface finish, but if the RPM is too high, it can lead to overheating and premature tool wear. A slower feed rate typically results in a smoother finish because the cutter takes smaller bites from the material. Additionally, the material being machined and the type of tool used influence the optimal speed and feed settings. Properly balancing these factors ensures the best surface quality for the workpiece.

5.5.3.1 Surface Feet Per Minute (SFPM). Represents the speed at which the cutting edge of a rotating tool moves along the surface of the material being cut. It is calculated based on the distance the tool's cutting edge travels in one minute if it were in continuous contact with the workpiece, expressed in feet per minute. SFPM is essential for determining the correct spindle speed (RPM) to achieve optimal cutting conditions. The ideal SFPM depends on the material being machined and the type of cutting tool. (See Table 5-8 and Table 5-9.)

Table 5-8. SFPM by Material

Material	SFPM Range
Aluminum	500-1,000
Steel (mild)	100-200
Stainless Steel	80-150
Cast Iron	100-250
Brass	200-500

5.5.3.2 Spindle Speed (RPM). The rotational speed of the tool, depending on factors such as tool material, tool diameter, and material being machined. (See Table 5-11.)

5.5.3.3 Feed Rate Inches Per Minute (IPM). The distance the tool advances per minute, based on the number of teeth and the material's hardness.

5.5.3.4 Feed Per Tooth (FPT). Helps determine the appropriate feed rate for milling cutters based on material and cutter type. (See Table 5-10.)

5.5.3.5 Calculating Spindle Speed. The spindle speed formula calculates how fast the cutting tool should rotate based on the material and tool diameter. It is given by:

$$RPM = \frac{12\ FPM}{\pi D}$$

WHERE RPM = SPINDLE SPEED (IN RPM);

FPM = CUTTING SPEED (IN FEET PER MINUTE)

$\pi =$ 3.1416

D = DIAMETER (IN INCHES) OF CUTTER

- FOR EXAMPLE, IF YOU ARE MILLING ALUMINUM WITH A 1 INCH CUTTER USING A CARBIDE TOOL. IF THE RECOMMENDED CUTTING SPEED (SFPM) FOR ALUMINUM WITH CARBIDE IS 500 SFPM:

$$RPM = \frac{500 \times 12}{3.14 \times 1} = 1{,}909\ RPM$$

Table 5-9. Tool Selection and Cutting Speed

Material	SFPM Range
Aluminum	500-1,000
Steel (mild)	100-200

Table 5-9. Tool Selection and Cutting Speed - Continued

Material	SFPM Range
Stainless Steel	80-150
Cast Iron	100-250
Brass	200-500
Tool Material	**SFPM Range**
High-Speed Steel	100-300
Carbide	200-1,000
Cobalt Steel	100-250

5.5.3.6 Calculating Feed Rate IPM. The IPM formula calculates how fast the cutting tool should move along or be fed into the workpiece based on the material and tool diameter. It is given by:

$$\text{FEED RATE (IPM)} = \text{RPM} \times N \times F$$

WHERE RPM = SPINDLE SPEED (ROTATIONS PER MINUTE),
N = NUMBER OF TEETH OR FLUTES ON THE CUTTER,
F = THE FEED PER TOOTH (INCHES).

- TO CALCULATE THE FEED RATE AS AN EXAMPLE, IF THE TOOL HAS 4 FLUTES AND A FEED PER TOOTH OF 0.003”:

$$\text{FEED RATE (IPM)} = 1{,}909 \times 4 \times 0.003 = 22.9 \text{ IPM}$$

Table 5-10. Material Feed Per Tooth

Material	FPT (Inches)
Aluminum	0.003-0.010
Steel (mild)	0.002-0.006
Stainless Steel	0.0015-0.004

Table 5-11. Milling Cutter RPM

Cutter Diameter (Inches)	SFPM (Cutting speed)	RPM (Calculated)
1/8	15-200	45-600
3/16	15-200	60-800
1/4	15-200	90-1,200
5/16	15-200	108-1,440
3/8	15-200	135-1,800
7/16	15-200	162-2,160
1/2	15-200	180-2,400
9/16	15-200	202-2,688
5/8	15-200	225-3,000
11/16	15-200	247-3,300
3/4	15-200	270-3,600
13/16	15-200	292-3,888
7/8	15-200	315-4,200
15/16	15-200	337-4,500

Table 5-11. Milling Cutter RPM - Continued

Cutter Diameter (Inches)	SFPM (Cutting speed)	RPM (Calculated)
1	15-200	360-4,800
1-1/4	15-200	450-6,000
1-1/2	15-200	540-7,200
1- 3/4	15-200	630-8,400
2	15-200	720-9,600
2-1/4	15-200	810-10,800
2-1/2	15-200	900-12,000
2-3/4	15-200	990-13,200
3	15-200	1,080-14,400
3-1/2	15-200	1,260-16,800
4	15-200	1,440-19,200
5	15-200	1,800-24,000
6	15-200	2,160-28,800

5.5.4 Machining Cutting Fluids.

5.5.4.1 General. Machining cutting fluids are specialized substances used during machining processes to improve efficiency and ensure high-quality results. These fluids serve multiple purposes that directly impact the machining process, tool life, and the finished product. Cutting fluids are essential for achieving optimal performance in machining operations, especially when working with tough-to-machine materials or performing high-speed processes. Proper selection and use of cutting fluids can enhance productivity, improve surface finish, and extend tool life:

5.5.4.1.1 Cooling. Cutting operations generate significant heat due to friction and deformation of the material. Cooling provided by cutting fluids helps prevent tool overheating, material distortion, and damage to the workpiece.

5.5.4.1.2 Lubrication. Fluids reduce friction between the cutting tool and workpiece, minimizing wear and tear on cutting tools and enabling smoother cuts.

5.5.4.1.3 Chip Removal. Effective cutting fluids help flush away chips and debris from the cutting area, preventing re-cutting, tool clogging, and surface damage.

5.5.4.1.4 Corrosion Protection. Cutting fluids form a protective layer over machined surfaces, safeguarding against rust and oxidation during and after machining.

5.5.4.2 Cutting Fluids. Different types of cutting fluids are formulated to meet the requirements of specific materials and machining operations. (See Table 5-12.) Here is an expanded explanation of each type:

5.5.4.2.1 Soluble Oils. These are oil-water emulsions that provide a balance of cooling and lubrication. Typically mixed with water in ratios of 5-10 percent, soluble oils are versatile and widely used for machining operations on a variety of metals, including aluminum and mild steels.

Advantages

Easy to use, cost-effective, and suitable for general-purpose machining.

Disadvantages

Prone to bacterial growth and contamination, requiring regular maintenance.

5.5.4.2.2 Synthetic Fluids. These water-based fluids contain no oil, relying on chemical additives for cooling and lubrication. They offer excellent cooling properties and have long service lives, making them ideal for grinding, high-speed machining, and applications requiring precision.

Advantages

Clean, non-sticky, and effective at high temperatures.

Disadvantages

Disadvantages - Limited lubricity compared to oil-based fluids.

5.5.4.2.3 Semi-Synthetic Fluids. A hybrid of soluble oils and synthetic fluids, semi-synthetics combine the benefits of both types. They provide good cooling and lubrication, making them suitable for turning, milling, and drilling operations on ferrous and non-ferrous metals.

Advantages

Versatile and efficient for a wide range of machining tasks.

Disadvantages

Slightly higher cost than soluble oils.

5.5.4.2.4 Straight Oils. These are undiluted petroleum or mineral oils, often fortified with additives for extreme pressure performance. Straight oils provide superior lubrication and are commonly used for operations like tapping, threading, and low speed cutting.

Advantages

Excellent lubricity and chip control.

Disadvantages

Poor cooling ability and can create a smoky work environment.

5.5.4.2.5 High-Pressure Oils. These fluids contain extreme pressure additives to handle extreme machining conditions, such as high temperatures and loads. They are commonly used for materials like titanium, stainless steel, and nickel-based alloys that require high lubricity to prevent galling and tool wear.

Advantages

Outstanding performance for tough materials.

Disadvantages

Higher cost and potential health/environmental concerns.

Table 5-12. Types of Cutting Fluids

Type	Composition	Key Characteristics	Applications
Soluble Oils	Oil-water emulsions, typically 5-10 percent oil.	Good cooling and lubrication, easy to apply.	General machining of ferrous and non-ferrous metals.
Synthetic Fluids	Water-based with chemical additives.	Excellent cooling, low lubricity, long service life.	Grinding, high-speed machining, and precision operations.
Semi-Synthetic	Blend of synthetic and soluble oils.	Balanced cooling and lubrication properties.	Versatile, used in turning, milling, and drilling of various materials.

Table 5-12. Types of Cutting Fluids - Continued

Type	Composition	Key Characteristics	Applications
Straight Oils	Pure petroleum or mineral oils with additives.	High lubricity, minimal cooling.	Low-speed machining, tapping, threading, and operations requiring extreme lubrication.
High-Pressure Oils	Enhanced oils with extreme pressure additives.	Superior performance under high load and temperatures.	Difficult-to-machine materials like titanium, nickel alloys, and hardened steels.

5.5.5 Cutting Fluids and Material Compatibility.

5.5.5.1 Material Being Machined. When selecting a cutting fluid, it is crucial to consider the material being machined. Different materials have unique properties that affect how they interact with cutting fluids. (See Table 5-13.)

5.5.5.1.1 Aluminum Alloys. Aluminum is prone to galling and sticking to cutting tools, so fluids with excellent lubricity are required. Soluble oils and synthetic fluids are commonly used, but fluids with high chlorine content should be avoided to prevent staining or corrosion.

5.5.5.1.2 Stainless Steel. Stainless steels are tough materials that require fluids with extreme pressure additives to minimize work hardening and tool wear. High-pressure oils or semi-synthetic fluids are preferred.

5.5.5.1.3 Titanium and Nickel Alloys. These high-performance materials generate significant heat during machining. High-pressure oils and synthetic fluids with superior cooling and lubrication properties are essential to prevent galling and ensure precision.

5.5.5.1.4 Carbon Steels. General-purpose soluble oils or semi-synthetics are suitable for most machining operations. These fluids provide adequate cooling and lubrication for standard turning, milling, and drilling tasks.

5.5.5.1.5 Cast Iron. Cast iron generates fine, abrasive chips during machining, so cooling and chip flushing are more critical than lubrication. Synthetic fluids or soluble oils are commonly used to keep the cutting area clean and prevent buildup.

5.5.5.1.6 Plastics and Composites. Fluids must not cause swelling, cracking, or degradation of the material. Synthetic fluids or air cooling are preferred for these materials.

5.5.5.2 Effectiveness of Cutting Fluids. To maximize the effectiveness of cutting fluids, consider the following practices:

5.5.5.2.1 Mixing Ratios. Ensure proper dilution of soluble oils and semi-synthetics to avoid poor performance or damage to tools and workpieces.

5.5.5.2.2 Filtration and Maintenance. Regularly clean and filter cutting fluids to remove chips, sludge, and contaminants.

5.5.5.2.3 Application Methods. Select appropriate methods, such as flood cooling, misting, or through-tool delivery, to maximize the benefits of the cutting fluid.

5.5.5.2.4 Monitoring. Test the fluid for pH levels, concentration, and contamination to maintain its effectiveness and prolong its life.

Table 5-13. Cutting Fluid and Material Compatibility

Material	Recommended Cutting Fluids	Notes
Aluminum Alloys	Soluble oils, synthetic fluids.	Avoid fluids with high chlorine content to prevent staining or corrosion.
Stainless Steel	High-pressure oils, semi-synthetics.	Requires high lubricity to reduce work hardening and tool wear.
Carbon Steels	Soluble oils, semi-synthetics.	General-purpose fluids are sufficient for most operations.

Table 5-13. Cutting Fluid and Material Compatibility - Continued

Material	Recommended Cutting Fluids	Notes
Titanium Alloys	High-pressure oils, synthetic fluids.	Requires fluids with extreme pressure additives to handle high temperatures and prevent galling.
Cast Iron	Synthetic fluids, soluble oils.	Focus on cooling and flushing chips; avoid sticky fluids to prevent buildup on work surfaces.
Nickel Alloys	High-pressure oils, synthetic fluids.	Needs extreme pressure resistance to counteract material toughness.
Plastics/Composites	Synthetic fluids or air cooling (dry).	Minimize fluid use to avoid material swelling or degradation.

5.5.6 Securing the Workpiece on the Milling Machine. Securing the workpiece is crucial to ensure accuracy and prevent movement during machining. Various workpiece holding methods are available (Table 5-14), each designed to cater to specific types of parts, materials, and milling tasks. The choice of workpiece holding method for a milling machine depends on the shape, size, and material of the workpiece, as well as the specific milling operation being performed. By selecting the appropriate method and following the correct set-up procedures, machinists can achieve high-precision and repeatable results. Proper work holding is essential for ensuring safety, minimizing setup time, and optimizing machining accuracy.

5.5.6.1 Types of Milling Vices. For additional information on milling vices. (See Paragraph 5.4.3.1.1.)

5.5.6.1.1 Standard Milling Vices. Feature two jaws (fixed and movable) and are commonly used for clamping rectangular and square workpieces.

5.5.6.1.2 Swivel Vices. These allow the workpiece to be rotated, providing the ability to mill at different angles without removing the part from the vice.

5.5.6.1.3 Angle Vices. Used when workpieces need to be clamped at an angle to the table, such as for machining parts at an inclined position.

5.5.6.2 General Setup of the Milling Vice.

5.5.6.2.1 Prepare the Vice. Clean the jaws and ensure there is no debris or chips. Check that the vice is free of damage or wear.

5.5.6.2.2 Place the Workpiece. Position the part between the jaws. Ensure it is centered and square to the milling table for accurate cuts.

5.5.6.2.3 Tighten the Jaws. Tighten the movable jaw of the vice to clamp the workpiece firmly. Apply even pressure to avoid distorting the part.

5.5.6.2.4 Check for Alignment. Use a dial indicator or a square to confirm that the workpiece is properly aligned with the machine axis.

NOTE

When using larger or irregularly shaped parts, a vice may not provide sufficient clamping strength. Clamps and fixtures are often used to secure the part directly to the milling machine table.

5.5.6.3 Types of Clamps.

5.5.6.3.1 T-Slot Clamps. T-slot bolts and clamps are secured in the milling machine’s table slots, allowing for quick adjustments

5.5.6.3.2 Step Clamps and Blocks. Step clamps are useful for holding a workpiece at a fixed height and position, with additional blocks to prevent lifting during cutting.

5.5.6.3.3 V-Clamps and C-Clamps. These are used for holding smaller parts or delicate components where a vice may not be suitable.

5.5.6.4 General Setup Using Clamps.

5.5.6.4.1 Position the T-Slot Bolts. Insert T-nuts into the table's T-slots, then install the T-slot bolts to secure the clamps.

5.5.6.4.2 Set the Workpiece. Align the workpiece with the table and clamp it using step clamps or other necessary fixtures to prevent movement during the milling operation.

5.5.6.4.3 Tighten and Secure. Ensure that the clamps are tightened without over-stressing the part. Verify that the workpiece cannot move by applying light force in various directions.

5.5.6.4.4 Double-Check Alignment. After securing the workpiece, check its position using precision instruments such as a surface plate or dial gauge.

5.5.6.5 Rotating the Workpiece. For operations that involve rotating the workpiece (such as rotary milling or when using a dividing head), chucks are used to securely hold cylindrical or round parts.

5.5.6.6 Types of Chucks.

5.5.6.6.1 3-Jaw Chucks. Self-centering chucks commonly used for cylindrical parts, providing fast setup.

5.5.6.6.2 4-Jaw Chucks. Used when precision centering is required for irregularly shaped or non-round parts.

5.5.6.7 General Setup Using a Chuck.

5.5.6.7.1 Install the Chuck.

a. Mount the chuck on the rotary milling attachment or dividing head.

b. Ensure it is properly secured to avoid vibrations during rotation.

5.5.6.7.2 Place the Workpiece.

a. Insert the workpiece into the chuck.

b. For a 3-jaw chuck, tighten all jaws evenly to center the part automatically.

5.5.6.7.3 Tighten the Chuck.

a. For 4-jaw chucks, adjust each jaw to secure the workpiece.

b. Use a dial indicator to verify centering before tightening completely.

5.5.6.7.4 Check for Stability.

a. Rotate the chuck manually to ensure the workpiece is stable.

b. Ensure that it does not wobble during milling.

5.5.6.8 Work-Holding Methods for Milling Machines. The following summary table provides a comparison of common work-holding methods, highlighting their suitability for different workpiece types and machining requirements.

Table 5-14. Work-Holding Methods for Milling Machines

Work-Holding Method	Workpiece Type	Advantages	Disadvantages
Vices	Rectangular, Square	Quick and easy setup	Limited to specific shapes
		Good clamping force	May not be suitable for large workpieces
Standard Vice		Versatile for general use	
Swivel Vice		Allows angled machining without repositioning	
Angle Vice		Holds workpieces at specific angles	
Clamps	Larger, Irregular Shapes	Versatile and adaptable to different workpiece geometries	Can require more setup time than vices
T-Slot Clamps		Quick and adjustable	
Step Clamps/Blocks		Securely hold workpieces at a fixed height	
V-Clamps/C-Clamps	Small, Delicate Components	Suitable for delicate parts where a vice might be too bulky	
Chucks		Securely grip round stock	Specific to round workpieces
		Allow for precise centering	Can be more time-consuming to set up than vices
3-Jaw Chuck		Self-centering for fast setup with cylindrical parts	
4-Jaw Chuck	Cylindrical, Round	Provide individual jaw adjustment for precise centering	

5.5.7 Squaring the Material.

5.5.7.1 General. This is an essential process to ensure that the workpiece is accurately aligned with the machine axes before beginning any machining operations. The process involves milling one side of the workpiece to create a flat surface, then using that surface to square the remaining sides. This ensures all edges are perpendicular and aligned to the machine's reference surfaces, providing a solid foundation for further machining.

5.5.7.2 General Instructions for Squaring Material.

5.5.7.2.1 Inspect the Workpiece.

a. Exam the material for any imperfections, such as warps or irregularities.

b. Use a surface plate or digital caliper to check for any noticeable defects that could affect squaring.

5.5.7.2.2 Set Up the Vice or Clamps.

a. Secure the workpiece in a milling vice, ensuring the part is seated evenly and squarely.

b. If the workpiece is large, use clamps to secure it directly to the milling table.

c. Tighten the vice or clamps firmly to hold the part in place without distorting it.

5.5.7.2.3 Choose the Milling Tool.

a. Select an appropriate cutting tool.

b. Choose a face mill, end mill, or fly cutter based on the size of the workpiece and the required finish.

5.5.7.2.4 Mill the First Surface.

a. Using the selected tool, begin milling one side of the workpiece.

b. Take light, even cuts to avoid distortion.

c. Ensure that the first surface is perfectly flat and level with the machine table.

d. Measure flatness and use a dial indicator or surface gauge to check for any variations.

5.5.7.2.5 Flip the Workpiece.

a. After milling the first surface, release the vice or clamps and flip the workpiece over.

b. The newly machined surface must be in contact with the vice (solid jaw) or table.

5.5.7.2.6 Mill the Opposite Surface.

a. Mill the opposite side of the workpiece.

b. Ensure that it is parallel and square to the first milled surface.

c. Regularly check alignment with a dial indicator to verify the squareness of the sides.

5.5.7.2.7 Check Squareness.

a. Use a machinist square, dial indicator, calipers, or micrometer to check the squareness of all four sides of the workpiece.

b. Adjust the part as needed to correct any misalignment.

5.5.7.2.8 Final Inspection.

a. After completing the squaring process, conduct a final inspection to ensure all surfaces are flat, square, and aligned to the machine's reference planes.

b. Measure with calipers or micrometers to verify all dimensions.

5.5.8 Face Milling.

5.5.8.1 General. Face milling is a machining process used to create a flat surface on a workpiece, typically using a face mill cutter. This operation is ideal for removing material from the surface of a part to achieve a smooth, level finish and is often used to prepare surfaces for further machining or to reduce part thickness. The process ensures that the workpieces face is parallel to the milling machine's table or spindle, providing an accurate reference for additional operations.

5.5.8.2 General Instructions for Face Milling.

5.5.8.2.1 Set Up the Workpiece.

a. Secure the workpiece in the milling vice or on the milling table using clamps, ensuring it is firmly held in place.

b. The workpiece should be oriented with the surface to be machined facing upward.

c. For larger parts, consider using a rotary table or other fixture to ensure proper alignment.

5.5.8.2.2 Select the Face Mill Cutter.

a. Choose a face mill cutter that matches the size of the desired cutting parameters (e.g., cutter diameter, number of teeth, cutting material).

b. Ensure the cutter is sharp and free of damage to avoid poor finishes and unnecessary tool wear.

5.5.8.2.3 Install the Cutter.

a. Mount the face mill cutter securely into the milling machine's spindle.

b. Ensure the cutter is positioned at the correct height relative to the workpiece, and check that it is securely tightened to prevent any slippage during the cutting process.

5.5.8.2.4 Set Cutting Parameters.

a. Adjust the milling machine's speed and feed rates based on the material being machined, the cutter's specifications, and the desired finish.

b. Set the cutting depth to ensure efficient material removal without overloading the cutter.

5.5.8.2.5 Begin Milling the First Pass.

a. With the milling machine in motion, engage the face mill cutter with the workpiece.

b. Begin with a light cut to check for proper setup and alignment.

c. Gradually increase the depth of cut for subsequent passes.

d. Ensure even material removal across the entire surface.

5.5.8.2.6 Make Successive Passes.

a. Continue making passes across the surface of the workpiece.

b. Overlap each pass slightly to ensure uniform cutting.

c. Regularly measure the material with a caliper, micrometer or dial indicator to ensure that the surface remains flat and level.

d. Adjust the feed rate or cutting depth if needed.

5.5.8.2.7 Monitor Cutter Wear and Temperature.

a. Monitor the condition of the cutter for any signs of wear or overheating.

b. Pay attention to the chip formation, color, and cutter sound.

c. If necessary, pause the operation to clean or replace the cutter to maintain optimal cutting performance.

5.5.8.2.8 Final Surface Finish.

a. After completing the necessary number of passes, perform a final pass to achieve a smooth surface finish.

b. A finish pass should be shallow (0.002-0.010 inch) to provide the final clean-up and leave the surface free from tool marks or visible scratches.

5.5.8.2.9 Check for Flatness and Accuracy.

a. After completing the face milling process, inspect the workpiece for flatness using a surface gauge or dial indicator.

b. Verify the surface is parallel to the machine's table or spindle and ensure that all dimensions are within tolerance.

5.5.9 Stepping.

5.5.9.1 General. Stepping is a milling operation used to create a series of flat, stepped surfaces on a workpiece, often used for parts that require multiple height levels or a stepped profile. This process is typically achieved using a stepped cutter or by making multiple passes with an end mill at different Z-axis depths.

5.5.9.2 General Instructions for Stepping.

5.5.9.2.1 Set Up the Workpiece.

a. Secure the workpiece in a vice, fixture, or clamped to the table.

b. Ensure the part is aligned with the machine's X and Y axes.

5.5.9.2.2 Select the Cutter. Choose a cutter or end mill, depending on the desired step size and material.

5.5.9.2.3 Set Cutting Parameters.

a. Set the machine's feed rate, spindle speed, and depth of cut for the material and cutter.

b. Set the depth of each pass to correspond to the desired step height.

5.5.9.2.4 Begin Milling.

a. Engage the cutter with the workpiece, making light initial cuts to check the setup.

b. Gradually increase the cutting depth for each step, ensuring proper dimensional accuracy.

5.5.9.2.5 Inspect the Steps.

a. Measure the step heights using a micrometer or caliper to ensure they are within tolerance.

b. Adjust the machine settings if necessary.

5.5.10 Pocket Milling. Pocket milling is the process of cutting a cavity into a workpiece, typically rectangular or circular, by removing material inside a defined area. This operation is used for creating pockets, grooves, or cavities in parts like housings or base plates.

5.5.10.1 General Instructions for Pocket Milling.

5.5.10.1.1 Set Up the Workpiece.

a. Secure the workpiece in a vice, fixture, or clamped to the table.

b. Align the part with the X and Y axes to ensure correct positioning of the pocket.

5.5.10.1.2 Select the Cutter. Choose a cutter, such as an end mill or ball nose cutter, based on pocket dimensions and the required finish.

5.5.10.1.3 Set Cutting Parameters.

a. Adjust the spindle speed, feed rate, and depth of cut based on material and cutter size.

b. For deep pockets, rough out the cavity before performing the final pass.

5.5.10.1.4 Begin Milling.

a. Begin milling from the center or edge of the pocket, making incremental passes.

b. Use climb milling to reduce tool wear and improve surface finish.

5.5.10.1.5 Inspect Pocket Dimensions.

a. Check pocket depth and width using calipers or a depth gauge.

b. Adjust as needed for final accuracy.

5.5.11 Angle Milling. This process involves creating surfaces or cutting edges at a specific angle to the workpiece. This operation can be accomplished by tilting the milling head, using an angle plate, or using an angle vice. Each method offers a way to achieve precise angled cuts, depending on the setup and machine capability.

5.5.11.1 General Instructions for the Head Tilt Method.

5.5.11.1.1 Set Up the Workpiece.

a. Secure the workpiece in a vice, fixture or directly on the machine table.

b. Ensure the part is properly aligned with the milling machine's axes.

c. Check that the cutting area is clear of obstructions.

5.5.11.1.2 Tilt the Milling Head.

a. Loosen the X or Y-axis locking bolts or clamping mechanism of the mill head.

b. Use the angle scale on the milling machine to set the desired angle for the head.

c. Carefully adjust the head to the required angle.

d. Once the head is positioned correctly, tighten the locking bolts to secure it in place.

5.5.11.1.3 Set Cutting Parameters.

a. Select the appropriate cutter for the operation, considering the angle and material.

b. Adjust the spindle speed, feed rate, and cutting depth to match the cutter and material.

5.5.11.1.4 Begin Milling.

a. Engage the cutter with the workpiece at the desired angle.

b. Proceed with milling.

c. Monitor to ensure that the angle is maintained throughout.

5.5.11.1.5 Inspect the Angle and Finish.

a. Measure the angle of the cut using a protractor or angle gauge.

b. Ensure the surface finish is consistent with the operation requirements.

5.5.11.2 General Instructions for Using an Angle Plate.

5.5.11.2.1 Set Up the Workpiece.

a. Place the angle plate securely on the mills table, aligning it with the X and Y axes.

b. Clamp the workpiece to the angle plate using appropriate fixtures.

c. Ensure the workpiece is positioned at the desired angle.

5.5.11.2.2 Set Cutting Parameters.

a. Select the cutter appropriate for the desired angle and material.

b. Adjust the machine's spindle speed, feed rate, and cutting depth accordingly.

5.5.11.2.3 Begin Milling.

a. Engage the cutter with the workpiece.

b. Proceed with milling.

c. Ensure consistent feed and depths of cut.

5.5.11.2.4 Inspect the Angle and Finish.

a. Measure the cut angle with a protractor or angle gauge.

b. Ensure the surface finish is consistent with the operation requirements.

5.5.11.3 General Instructions for Using an Angle Vice.

5.5.11.3.1 Set Up the Workpiece.

a. Place the angle vice on the milling machine's table and secure it with clamps or bolts.

b. Adjust the vice to the desired angle.

5.5.11.3.2 Secure the Workpiece.

a. Place the workpiece inside the vice and secure it tightly.

b. Ensure that the part is aligned properly.

5.5.11.3.3 Set Cutting Parameters.

a. Select a cutter for the milling operation.

b. Adjust the spindle speed and feed rate according to the cutter and material type.

5.5.11.3.4 Begin Milling.

a. Engage the cutter with the workpiece.

b. Ensure the correct angle is maintained throughout the cut.

c. Make gradual passes to achieve the desired angle and surface finish.

5.5.11.3.5 Inspect the Angle and Finish.

a. Check the angle with a protractor or angle gauge to verify precision.

b. Ensure the surface finish is consistent with the operation requirements.

5.5.12 Slotting on a Milling Machine. Slotting is the process of cutting narrow, deep channels or grooves into a workpiece. This operation is commonly used for creating keyways, slots for fasteners, or other precise channels.

5.5.12.1 General Instructions for Slotting.

5.5.12.1.1 Set Up the Workpiece.

a. Secure the workpiece in the milling vice or fixture.

b. Ensure the workpiece is aligned correctly along the X and Y axes.

5.5.12.1.2 Select the Cutter. Choose a slotting cutter, saw, or end mill based on the desired slot width and material.

5.5.12.1.3 Set Cutting Parameters. Adjust the machine's speed, feed rate, and cutting depth according to the slot size and material.

5.5.12.1.4 Begin Milling the Slot.

a. Position the cutter at the starting point of the slot and begin milling in small, incremental cuts.

b. Machine multiple passes to avoid overloading the cutter, progressively widen the slot if needed.

5.5.12.1.5 Inspect Slot Dimensions.

a. Measure the slot width and depth using calipers or a depth gauge to ensure they are within tolerance.

b. Measure the surface finish to ensure that the operation has produced the required quality.

5.5.13 Drilling. Drilling is the process of creating a round hole in a workpiece using a rotating drill bit. Milling machines can be used for both vertical and horizontal drilling operations.

5.5.13.1 General Instructions for Drilling. (See Table 5-15.)

5.5.13.1.1 Set Up the Workpiece.

a. Secure the workpiece in the milling vice or fixture.

b. Layout hole patterns by marking the workpiece (layout dye, scribe, centerpunch), as required.

c. Align the part according to the required hole positions using layouts marks.

d. If more precise hole locations are required, utilize an edge finder or center finder along with the DRO, or the graduated scale on the applicable axis hand feed.

5.5.13.1.2 Center Drill the Hole.

a. Install a center drill into the milling machine's collet or drill chuck.

b. Align the spindle over the marked hole location.

c. Slowly lower the spindle and use the center drill to create a small conical indentation to prevent the drill bit from wandering.

5.5.13.1.3 Drill a Pilot Hole.

a. Replace the center drill with a drill bit of smaller diameter (pilot drill), for holes larger than 3/16 inch.

b. Adjust the spindle speed according to the pilot drill diameter and material being drilled.

c. Carefully lower the spindle and drill the pilot hole to the required depth, ensuring a steady feed rate.

d. Periodically withdraw the drill bit to clear chips and improve cutting efficiency.

e. Step drill as required:

- **Holes up to 1/2 inch**: Use increments of **1/16 to 1/8 inch**
- **Holes larger than 1/2 inch**: Use increments of **1/8 to 1/4 inch**
- **Holes larger than 1 inch**: Use increments of **1/4 to 3/8 inch**

5.5.13.1.4 Drill the Final Hole.

a. Replace the pilot drill with the drill bit corresponding to the final hole diameter.

b. Verify the spindle speed and feed rate, adjusting as necessary for the larger drill bit.

c. Slowly drill through the pilot hole to the desired depth, ensuring the drill bit does not overheat while raising the spindle periodically to clear chips.

d. Periodically withdraw the drill bit to clear chips and improve cutting efficiency.

5.5.13.1.5 Inspect the Drilled Hole.

a. Measure the hole diameter with a small hole or telescoping gauge and micrometer to ensure it meets the required specifications.

b. Check the surface finish of the hole.

5.5.14 Reaming. Reaming is a machining process used to produce highly accurate hole diameters with improved surface finish. It typically follows drilling or boring operations to achieve precise tolerances.

5.5.14.1 General Instructions for Machine Reaming. (See Table 5-15.)

5.5.14.1.1 Set Up the Workpiece.

a. Securely clamp the workpiece to the milling machine table or a vice to prevent movement during reaming.

b. Ensure the workpiece is properly aligned with the spindle axis for concentric reaming.

c. Use a drill blank, co-axial indicator, or DRO to verify the spindle alignment with the hole if required.

5.5.14.1.2 Drill or Bore a Pilot Hole.

a. Select a drill bit slightly smaller than the desired reamed diameter, typically 0.005 to 0.015 inch undersized.

b. Drill the pilot hole to the required depth, ensuring the hole is straight and free of burrs or chips.

c. Periodically withdraw the drill bit to clear chips and improve cutting efficiency.

d. For critical tolerances, consider boring the pilot hole to achieve better roundness and alignment.

5.5.14.1.3 Set Cutting Parameters.

a. Install the reamer in the spindle using a collet, drill chuck, or other appropriate tool holder.

b. Adjust the spindle speed to approximately 1/3 to 1/2 of the drilling speed for the material being machined.

c. Set a low feed rate to allow the reamer to cut smoothly without excessive force, typically 0.002 to 0.005 Inches Per Revolution.

5.5.14.1.4 Ream the Hole.

a. Align the reamer with the pilot hole to ensure concentric entry.

b. Apply cutting fluid (as required for the material) to reduce friction and improve the surface finish.

c. Lower the reamer into the pilot hole using a steady feed, avoiding interruptions or excessive force.

d. Do not reverse the spindle while the reamer is in the hole, as it can damage the tool and the workpiece.

e. Keep the spindle spinning in the same forward direction as during the cutting pass while you retract the reamer.

f. Retract slowly and steadily using a smooth, controlled motion with the quill handle to avoid jerking the reamer during withdrawal.

5.5.14.1.5 Inspect the Finished Hole.

a. Measure the hole diameter with a small hole or telescoping gauge and micrometer to ensure it meets the required specifications.

b. Check the surface finish for smoothness and the absence of tool marks.

c. Deburr the edges of the hole if necessary.

5.5.15 Boring with an Offset Boring Head. This process is a precise machining operation used to enlarge or finish pre-drilled or cast holes to a specific diameter with high accuracy and a smooth surface finish.

5.5.15.1 General Instructions for Boring Using an Offset Boring Head.

5.5.15.1.1 Set Up the Workpiece.

a. Securely clamp the workpiece to the milling machine table or a vice to prevent movement during boring.

b. Ensure the workpiece is properly aligned with the spindle axis for concentric reaming.

c. Use a drill blank, co-axial indicator, or DRO to verify the spindle alignment with the hole if required.

d. Confirm the workpiece is level, and if necessary, use shims or parallels for proper support.

5.5.15.1.2 Install the Offset Boring Head.

a. Mount the offset boring head into the spindle using an appropriate tool holder (e.g., R8, CAT, or boring taper).

b. Install the boring bar into the boring head, ensuring it is properly seated and tightened.

c. Ensure the boring bar is sharp, has the correct cutting angle for the material, and is rigidly secured in the boring head.

d. Adjust the length of the boring bar so it clears the inside of the hole and does not interfere with the workpiece during rotation.

5.5.15.1.3 Select Cutting Parameters.

a. The starting hole should be slightly smaller than the final desired diameter. It can be drilled or pre-machined to approximately **0.030 to 0.050 inch** undersize.

b. Touch off the boring bar.

- Option 1: Preferred and most accurate, utilize a dial indicator to dial in zero and/or the touch off point.
- Option 2: Adjust the boring bar head until the cutter makes light contact with the workpiece.

c. For roughing, use a depth of cut between **0.010 to 0.030 inch**.

d. For finishing, reduce the depth of cut to **0.002 to 0.010 inch**.

e. Double-check the measurement using a micrometer or caliper to ensure accuracy.

f. Adjust the spindle speed based on the material and diameter of the hole.

g. A typical starting point is 300-500 RPM for steel and 500-800 RPM for aluminum.

h. Set a slow, steady feed rate to maintain precision and surface finish.

i. Apply appropriate cutting fluid to reduce heat and improve chip evacuation.

5.5.15.1.4 Perform the Boring Operation.

a. Slowly lower the boring head into the hole using the quill or Z-axis.

b. Engage the spindle and feed the boring head into the hole, maintaining a smooth and consistent feed rate.

c. Complete the cut by boring to the required depth, taking care not to overcut.

d. Always keep the spindle spinning in the same forward direction when retracting the boring bar.

e. For deep holes, clear chips periodically by retracting the boring head slightly without stopping the spindle.

5.5.15.1.5 Inspect the Bore.

a. Measure the bore diameter with an internal micrometer or bore gauge to ensure it meets specifications.

b. Inspect the surface finish for smoothness and adjust the boring head or cutting parameters if necessary.

c. Repeat the process with fine adjustments to the boring head for additional passes if required for precision or final sizing.

5.5.16 Tapping on a Milling Machine. This process is used to create internal threads in pre-drilled holes. This operation requires precision, correct tool selection, and attention to best practices to ensure high-quality threads and avoid tool breakage or workpiece damage.

5.5.16.1 General Instructions for Tapping on a Mill. (See Table 5-15.)

5.5.16.1.1 Set Up the Workpiece.

a. Securely clamp the workpiece in a vice or fixture, ensuring it is tightly held and cannot shift during tapping. This is critical for maintaining accuracy.

b. Use proper fixturing methods, such as parallels or soft jaws, if necessary.

5.5.16.1.2 Verify Hole Size.

a. Ensure the hole to be tapped is pre-drilled to the correct tap drill size.

b. Refer to a tap drill chart for material-specific sizes. (See Table 5-16 and Table 5-17.)

c. An undersized hole will lead to excessive load on the tap and potential tap breakage.

d. Always drill a pilot hole slightly smaller than the final tap size to avoid forcing the tap into a too-small hole.

5.5.16.1.3 Choose the Correct Tap Type.

a. Select the appropriate tap based on the material and thread requirements.

b. Spiral point taps are suitable for through-holes, while spiral flute taps are better for blind holes.

5.5.16.1.4 Inspect the Tap.

a. Ensure the tap is sharp, clean, and free from damage.

b. Check the flutes for any wear or cracks that could lead to breakage during the operation.

c. Select taps with appropriate coatings or finishes for the material (e.g., TiN-coated taps for hard metals) to reduce wear and improve performance.

5.5.16.1.5 Install the Tap.

a. Mount the tap in a tapping chuck, collet, or spring-loaded tapping head.

b. Ensure the tap is centered and securely tightened in the holder to prevent wobbling or run out during operation.

5.5.16.1.6 Align the Tap.

a. Ensure the tap is aligned coaxially with the spindle.

b. Misalignment can cause crooked threads or excessive wear on the tap.

c. Use a tapping head with an automatic reversing feature for ease of operation and to prevent tap breakage from manually reversing the spindle.

5.5.16.1.7 Set Up the Milling Machine.

a. Set the spindle speed based on the material and tap size.

b. For most common materials, spindle speeds typically range from 50-300 RPM.

c. Adjust for larger taps or harder materials to reduce cutting speed and prevent excessive heat buildup.

5.5.16.1.8 Set Feed Rate.

a. Adjust the feed rate to match the thread pitch.

b. For manual tapping, ensure smooth and consistent quill feed.

c. If using a tapping head, the auto-feed will automatically adjust for proper thread engagement.

d. Use a slower feed rate when tapping harder materials and increase feed rates for softer materials to improve chip removal and prevent tap clogging.

5.5.16.1.9 Align the Tap and Position the Tool.

a. Position the milling machine's spindle directly over the hole to be tapped.

b. Use an edge finder or dial indicator to ensure precise alignment.

5.5.16.1.10 Check Depth.

a. Verify the hole depth before tapping to avoid bottoming out the tap.

b. Always check and verify hole depth with a depth gauge or micrometer before starting, as bottoming out can break the tap or distort the threads.

5.5.16.1.11 Perform the Tapping Operation.

a. Slowly feed the tap into the hole, ensuring steady pressure and avoiding forcing the tool.

b. Maintain a smooth feed, especially if manually controlling the quill.

c. If using a tapping head, allow it to engage automatically.

5.5.16.1.12 Thread Depth.

a. Tap to the required depth but avoid forcing the tap to the bottom of the hole.

b. Tap slightly shy of the bottom to prevent breaking the tap and to leave enough clearance for chips.

c. Make the tapping depth slightly shallower than required if tapping blind holes.

d. Use spiral flute taps to help evacuate chips from the bottom of the hole.

5.5.16.1.13 Apply Cutting Fluid. Use cutting fluid throughout the process, particularly for harder materials, to reduce friction, prevent heat buildup, and improve surface finish.

5.5.16.1.14 Retract the Tap.

a. If using a tapping head, ensure the spindle automatically reverses to back the tap out of the hole.

b. If manually retracting, reverse the spindle carefully to disengage the threads without damaging them.

5.5.16.1.15 Slow and Steady Retraction.

a. Do not jerk the tap or retract too quickly. Use a slow, steady feed to prevent damaging the newly formed threads.

b. Always back out slowly to allow for proper chip evacuation.

c. Avoid reversing the spindle too rapidly, as it can cause thread deformation or damage.

5.5.16.1.16 Check the Thread Quality.

a. Use a thread gauge or a fastener to check the accuracy and quality of the threads.

b. Ensure they are clean, smooth, and free of burrs or material buildup.

5.5.16.1.17 Measure Thread Depth.

a. If the hole was blind, check that the tap has not bottomed out and the threads are formed to the correct depth.

b. If threads appear incomplete or chipped, re-tap the hole with a slightly larger tap or use a hand tap to finish the threads manually.

5.5.16.1.18 Clean the Hole and Tap.

a. Clean the hole and remove any debris using compressed air or a brush.

b. Leftover chips can cause issues with thread fit or subsequent operations.

c. Remove any built-up material or chips from the tap using a soft brush to ensure it remains in good working condition for future use.

d. Regularly inspect and clean taps to extend their lifespan. Store them in a safe place to avoid damage.

Table 5-15. Drilling, Reaming and Tapping Guidelines

Operation	Tool Type	Common Sizes	Notes and Guidelines
Drilling	Twist Drill	1/16-2 inches (1.5-50 millimeters (mm))	Choose a drill size that is slightly smaller than the final reaming or tapping size.
			Use split point drills for better centering and reduced walking.
			HSS or carbide drills are used depending on the material.
Reaming	Straight Flute Reamer	1/16-2 inches (1.5-50 mm)	Ream only after drilling to correct size and ensure the hole is centered and smooth.
			Tolerance: reamers typically remove between 0.002-0.010 inch (0.05-0.25 mm) for precision sizing.
	Tapered Reamer	1/16-2 inches (1.5-50 mm)	Finish reaming typically gives a hole size with a mirror finish.
			Used for creating tapered holes, e.g., countersinking.
Tapping	Tap (Spiral Flute, Spiral Point, Hand Tap)	0-1/2 inch (M1-M12)	Spiral point tap: for through-holes (pushing chips forward.
			Spiral flute tap: for blind holes (pulling chips back out).
			Hand tap: used for smaller, manual operations.
			Tap drill size: generally, the tap drill size is based on the thread pitch, e.g., for and M10x1.5 thread, use an 8.5 mm as the tap drill size.
Tap Drill Sizes	National Pipe Thread (NPT) Thread Tap	1/16-2 inches (M1.5-M50)	For tapered thread (NPT), use specific NPT tap charts for hole sizing.
			Typically, 7/8 inch drill for NPT 1/2 inch threads, and so on.
Common Drill Sizes	Twist Drill (Metric and Imperial)	Example Metric sizes: 3, 4, 5, 6, 8 mm, etc.	
		Example Imperial sizes: 1/16, 1/8, 3/16, 1/4 inch, etc.	

Table 5-16. Standard Tap Drill Sizes

Tap Size	Thread Pitch	Tap Drill Size (Inches)
0-80	80	0.0595
1-64	64	0.0635
2-56	56	0.073
3-48	48	0.089
4-40	40	0.112
5-40	40	0.128

Table 5-16. Standard Tap Drill Sizes - Continued

Tap Size	Thread Pitch	Tap Drill Size (Inches)
6-32	32	0.138
8-32	32	0.164
10-24	24	0.190
10-32	32	0.189
1/4-20	20	0.257
1/4-28	28	0.242
5/16-18	18	0.312
5/16-24	24	0.296
3/8-16	16	0.375
3/8-24	24	0.359
1/2-13	13	0.437
1/2-20	20	0.469
5/8-11	11	0.531
5/8-18	18	0.562
3/4-10	10	0.687
3/4-16	16	0.719
7/8-9	9	0.812
7/8-14	14	0.844
1-8	8	0.921

Table 5-17. NPT Tap Drill Sizes

NPT Size	Thread Pitch	Tap Drill Size (Inches)
l/8 NPT	27	0.375
1/4 NPT	18	0.505
3/8 NPT	18	0.675
1/2 NPT	14	0.750
3/4 NPT	14	1.020
1 NPT	11	1.250
1-1/4 NPT	11	1.500
1-1/2 NPT	11	1.750
2 NPT	11	2.250
2-1/2 NPT	11	2.625
3 NPT	8	3.000
4 NPT	8	3.500
5 NPT	8	4.000
6 NPT	8	4.500

5.5.17 Indexing Head. An index head, also called a dividing head, is used to rotate the workpiece in precise increments for operations such as drilling holes at regular intervals or cutting gears. The head has a gear mechanism that allows it to rotate the part through specific angles. It is essential for operations like:

- **Creating evenly spaced features**: Drilling holes in a circular pattern, cutting gear teeth, milling slots at specific angles.

- **Machining polygons**: Milling hexagons, octagons.

5.5.17.1 Plain Indexing. In plain indexing, when you are dividing a circle into equal divisions (such as for milling polygons, gears, or other geometric shapes), you often use a worm gear with 40 teeth (standard practice). This system allows for precise divisions of a circle using a dividing head and hole plates. (See Table 5-18.) The key concept is to calculate the number of rotations and partial rotations required to achieve the desired number of equal divisions on the workpiece. (See Table 5-18.)

5.5.17.2 Turns of the Index Crank. To determine how many turns of the index crank are needed to obtain one division of any number of equal divisions on the workpiece, divide 40 by the number of equal divisions desired. This rule assumes you are using a worm wheel with 40 teeth, which is standard practice in dividing head.

$$\text{TURNS PER DIVISION} = \frac{40}{\text{NUMBER OF EQUAL DIVISIONS DESIRED}}$$

WHERE 40 = NUMBER OF TEETH IN THE WORM GEAR (STANDARD PRACTICE).
NUMBER OF EQUAL DIVISIONS DESIRED = THE NUMBER OF DIVISIONS OR EQUAL SECTIONS YOU WANT TO CREATE (E.G., 6 DIVISIONS FOR A HEXAGON, 42 TEETH FOR A GEAR).

EXAMPLE : MILLING A HEXAGON (6 DIVISIONS)
DESIRED DIVISIONS: 6 (FOR A HEXAGON).
APPLY THE FORMULA:

Table 5-18. Plain Indexing with Hole Plates

Number of Divisions	Hole Plate Option	Turns Per Division	Spaces Per Plate (Example)
6	18-hole plate	6 turns + 2/3 turn	12 spaces on an 18-hole plate
14	42-hole plate	2 turns + 2/7 turn	20 spaces on a 42-hole plate
5	20-hole plate	8 turns + 1/5 turn	4 spaces on a 20- hole plate
10	40-hole plate	4 turns + 1/4 turn	10 spaces on a 40-hole plate

5.5.18 General Instructions for Indexing on a Mill.

5.5.18.1 Determine the Number of Divisions Required.

a. Calculate the number of divisions needed for your specific operation.

b. If you need to divide a circle into 4, 6, 8, 12, 24, or more equal sections, refer to Table 5-19 for the corresponding hole plate selection.

5.5.18.2 Choose a Hole Plate.

a. Hole plates are interchangeable.

b. Select the hole plate with holes that match the number of divisions required.

c. Common hole plate increments are 1 through 6, 8, 10, 12, etc.

d. The most common plates are the 24-hole plate (which is versatile and can be used for many operations) and the 40-hole plate for finer divisions.

5.5.18.3 Mount the Hole Plate.

a. Place the chosen hole plate on the index head spindle.

b. Secure it with the proper pin and ensure that the plate is flush against the spindle and properly indexed.

5.5.18.4 Mount the Index Head on the Machine Table.

a. Secure the index head on the machine table or an indexing vise using T-slots.

b. Ensure it is locked in place tightly, and the spindle is horizontal and aligned with the machine's axis.

5.5.18.5 Align the Workpiece.

a. Place the workpiece on the index head's rotating spindle or chuck.

b. Ensure that the workpiece is securely clamped, either using a vise, clamps, or directly in the chuck.

c. Use a dial indicator to check that the workpiece is centered and aligned with the spindle axis.

5.5.18.6 Set the Initial Position.

a. Rotate the index head manually to set the workpiece to its starting position.

b. Start at the 0 or 360 degree mark, depending on the operation.

c. Tighten the index head's locking mechanism to secure the spindle in place.

5.5.18.7 Use the Direct Indexing Mechanism.

a. For simple indexing, the index head has a gear system that moves the spindle by set increments (e.g., 1, 2 degrees, etc.).

b. This is commonly used for simple divisions like dividing a circle into equal sections.

5.5.18.8 Use the Compound Indexing Mechanism for Finer Divisions.

a. For more complex divisions (such as for gear cutting), you can use a compound indexing system.

b. This uses a combination of two sets of gears: one from the index head and one from a change gear to create a finer division.

5.5.18.9 Determine the Required Angle. Calculate the angle of rotation for each division.

- For example, if you are dividing a circle into 12 equal sections, the angle for each step would be 360 degrees ÷12= 30 degrees.

5.5.18.10 Set the Index Head.

a. Using the index pin, set the index head to the required angle.

b. Most index heads have a scale or dial to show the desired angle of rotation.

c. For precision, adjust the dial to the calculated angle.

d. Use the fine adjustment knob on the index head to make small, accurate adjustments.

5.5.18.11 Rotate the Index Plate.

a. Engage the index pin into the appropriate hole of the hole plate.

b. Rotate the spindle by the required angle.

c. If more than one hole is required (e.g., for 48 divisions), rotate the index plate accordingly and engage the pin into each successive hole.

d. Repeat the process.

e. After each index step, check that the workpiece is in the correct position and ready for the next machining operation (e.g., drilling, cutting, etc.).

f. Continue indexing the part until all required operations are completed.

Table 5-19. Index Hole Plate and Number of Divisions

Number of Divisions	Hole Plate Options	Possible Divisions (Using Combinations)
1	Any hole plate	360 degrees (One complete revolution)
2	6-hole plate	180 degrees (Skip 3 holes)
3	6-hole plate	120 degrees (Skip 2 holes)
4	6-hole plate	90 degrees (Skip 1 hole)
6	6-hole plate	60 degrees (Engage all 6 holes)
2	12-hole plate	180 degrees (Skip 6 holes)
3	12-hole plate	120 degrees (Skip 4 holes)
4	12-hole plate	90 degrees (Skip 3 holes)
6	12-hole plate	60 degrees (Skip 2 holes)
8	12-hole plate	45 degrees (Skip 1 hole)
12	12-hole plate	30 degrees (Engage all 12 holes)
2	24-hole plate	180 degrees (Skip 12 holes)
3	24-hole plate	120 degrees (Skip 9 holes)
4	24-hole plate	90 degrees (Skip 6 holes)
6	24-hole plate	60 degrees (Skip 4 holes)
8	24-hole plate	45 degrees (Skip 3 holes)
12	24-hole plate	30 degrees (Skip 2 holes)
24	24-hole plate	15 degrees (Skip 1 hole)
2	48-hole plate	180 degrees (Skip 24 holes)
3	48-hole plate	120 degrees (Skip 16 holes)
4	48-hole plate	90 degrees (Skip 12 holes)
6	48-hole plate	60 degrees (Skip 8 holes)
8	48-hole plate	45 degrees (Skip 6 holes)
12	48-hole plate	30 degrees (Skip 4 holes)
16	48-hole plate	22.5 degrees (Skip 3 holes)
24	48-hole plate	15 degrees (Skip 2 holes)
48	48-hole plate	7.5 degrees (Skip 1 hole)

5.5.19 Rotary Table. A rotary table is an essential accessory for a milling machine that enables a wide range of precision machining operations. It is primarily used to machine circular and arc-shaped features, evenly space features such as bolt holes or gear teeth and perform machining tasks at precise angular intervals. The rotary table enhances the versatility of the milling machine by allowing the operator to manipulate the workpiece in rotational movements while maintaining exact alignment with the cutting tool.

5.5.19.1 The rotary table can be mounted in a horizontal or vertical orientation, depending on the required operation, and can accommodate various accessories, such as dividing plates, indexing attachments, or chucks, to expand its functionality. Mastering the use of a rotary table is critical for applications requiring high precision, such as creating complex profiles, machining curved surfaces, or cutting gears. Proper setup, accurate alignment, and adherence to best practices are essential to achieve optimal results when using a rotary table.

5.5.19.2 Key Components of a Rotary Table.

Table Surface

Circular surface where the workpiece or fixture is mounted.

Indexing Dial

Indicates the table's angle of rotation, usually in degrees.

Locking Mechanism

Secures the rotary table to the milling machine table.

T-Slots

Used to mount workpieces or fixtures securely.

Crank Handle

Used to rotate the table.

Dividing Plates (Optional)

For indexing specific divisions.

Center Hole

Often used for aligning the workpiece or mounting accessories.

5.5.20 General Instructions for using a Rotary Table on a Mill.

5.5.20.1 Preparing the Rotary Table.

a. Clean the rotary table and milling machine table.

b. Ensure both surfaces are free of debris, oil, and chips.

c. Use a lint-free cloth and degreaser for cleaning.

5.5.20.2 Inspect the Table.

a. Check that the rotary table's indexing dial moves smoothly without binding.

b. Ensure the table locks function properly.

5.5.20.3 Select and Install Necessary Accessories.

a. Install dividing plates or indexing attachments, as required.

b. Choose appropriate T-slot clamps, step blocks, and hold-down fixtures.

5.5.20.4 Orient the Table.

a. Position the rotary table flat (horizontal) or tilt it vertically if the operation requires it.

b. For vertical mounting, ensure the table's axis is perpendicular to the spindle.

5.5.20.5 Secure to the Milling Machine Table.

a. Align the rotary table with the machine's X- or Y-axis using the table's guide slots.

b. Use T-slot bolts and clamps to secure it to the milling machine table.

5.5.20.6 Verify Table Alignment.

a. Use a dial indicator mounted in the spindle to sweep the surface of the rotary table.

b. Adjust the rotary table until the run out is within 0.001 inch or as specified by the part tolerances.

c. Lock the table to maintain alignment.

5.5.20.7 Determine the Center.

a. Insert a dial indicator or a centering tool into the spindle.

b. Move the spindle until the indicator shows zero deflection at the rotary table center hole.

5.5.20.8 Mount the Workpiece.

a. Use a chuck, T-slot clamps, or a fixture to secure the workpiece to the rotary table.

b. For circular workpieces, center them in the rotary table using the center hole and an alignment pin.

5.5.20.9 Double-Check Centering. Rotate the table 360 degrees while checking the dial indicator to ensure the workpiece is centered.

5.5.20.10 Choose the Appropriate Tool.

a. Use end mills, face mills, or slot drills based on the operation (e.g., cutting arcs, drilling holes).

b. Ensure the tool is sharp and appropriate for the material being machined.

5.5.20.11 Tool Length and Clearance. Verify that the tool clears the clamps and fixtures as the rotary table moves.

5.5.20.12 Feed Rate and Spindle Speed. Adjust based on material type, cutter diameter, and the operation (refer to machinist handbooks or cutting data charts).

5.5.20.13 Cutting Depth.

a. Take light cuts (0.005-0.020) when machining arcs or intricate profiles.

b. For roughing cuts, use deeper passes, but monitor for chatter.

5.5.20.14 Lock the Rotary Table During Machining. For precision, lock the table during each pass unless continuous rotation is required.

5.5.20.15 Set Starting Point. Align the workpiece and spindle to the starting point of the arc.

5.5.20.16 Calculate Angular Movements. Determine the arc’s start and stop angles from the drawing.

5.5.20.17 Machining the Arc.

a. Rotate the table to the starting angle.

b. Lock the rotary table, make a cut, and unlock it to rotate incrementally until the final angle is reached.

5.5.20.18 Calculate the Division. Use the following formula:

$$\text{DEGREES PER DIVISION} = \frac{360}{\text{NUMBER OF DIVISIONS}}$$

5.5.20.19 Set the Table.

a. Rotate the crank handle to the desired degree using the indexing dial.

b. Machine the feature.

c. Perform the cut, then rotate to the next position.

5.5.20.20 Set Hole Positions.

a. Calculate angular positions for each hole.

b. Rotate the table to the correct angle and lock it.

5.5.20.21 Drill Each Hole. Use peck drilling to avoid excessive heat buildup and ensure chip clearance.

5.5.20.22 Deburr the Workpiece. Remove any sharp edges or burrs caused by machining.

5.5.20.23 Clean the Rotary Table. Remove chips, oil, and residue from the rotary table.

5.6 GENERAL MACHINE OVERVIEW CNC.

5.6.1 CNC Milling Centers. CNC milling centers are highly automated machine tools used for precision machining of various materials, including metals, plastics, and composites. These machines use CNC to execute programmed instructions, allowing for complex and repeatable machining operations. The following sections provide detailed information on the operation, functions, and features of CNC milling centers commonly found in AMT shops.

5.6.2 The CNC Controller. The CNC controller (Figure 5-16) is designed for ease of use, providing intuitive controls and a user friendly interface for machinists of all levels. The controller consists of a keypad, an Liquid Crystal Display (LCD) display, and a set of function keys that allow users to input commands, navigate menus, and execute machining programs. Key features include:

LCD Display and Interface

Displays active programs, tool paths, machine coordinates, and error messages.

Soft Keys

Adaptive function keys that change based on menu selections.

Handle Jog and Rapid Override

Provides manual axis movement control and speed adjustment.

Memory and Edit Mode

Allows program storage, retrieval, and modification.

Manual Data Input (MDI) Mode

Enables direct G-code entry for quick operations.

Auto Mode (CYCLE START/FEED HOLD/RESET)

Controls program execution and machine cycling.

TO-34-1-10-109

Figure 5-16. Example CNC controller

Before Operation

Before operating a CNC mill, ensure the following pre-start checks.

5.6.2.1 Power on the Machine.

a. Turn the main power switch (located on the side of the machine) to the ON position.

b. Press the POWER ON button on the control panel.

c. Allow the system to boot up and display the main screen.

5.6.2.2 Machine Zero Return (Homing).

a. Press RESET to clear alarms or warnings.

b. Press POWER UP/RESTART to home all axes.

c. Wait until the machine moves to its reference position.

5.6.2.3 Check Tool and Work Offsets.

a. Press OFFSET and navigate to the TOOL OFFSET and WORK OFFSET pages.

b. Verify that all necessary offsets are correctly set.

5.6.2.4 Operate the Machine Manually. To operate the machine manually, refer to these general procedures.

5.6.2.4.1 Jogging the Axes.

a. Press HANDLE JOG to enable manual movement.

b. Use the jog handle to move the axes in small increments.

c. Adjust the jog speed using the 0.0001, 0.001, 0.01, 0.1 settings.

5.6.2.4.2 Spindle Control.

a. Press MDI and input S1000 M03 (for 1,000 RPM clockwise spindle rotation).

b. Press CYCLE START to execute the command.

c. Use M05 to stop the spindle when necessary.

d. Press COOLANT to activate the flood or mist coolant system.

5.6.2.5 Load an NC Program into the Controller. To load an NC program into the controller, follow these general procedures.

5.6.2.5.1 Insert the Universal Serial Bus (USB) Drive or Connect via Ethernet.

a. Insert a USB drive with the NC program into the front USB port.

b. Navigate to LIST PROGRAM and select the USB tab.

5.6.2.5.2 Load the NC Program.

a. Use the arrow keys to highlight the desired file.

b. Press F2 to copy the file to the machine's memory.

c. Navigate to the MEMORY tab and select the program.

5.6.2.5.3 Verify the Program.

a. Use GRAPHICS mode to simulate the program.

b. Review the tool paths and ensure no unexpected movements.

5.6.2.6 Execute a Program. To execute a program on the machine controller (cycle start), follow these general procedures.

5.6.2.6.1 Load the Program into Memory. Ensure the correct NC program is selected in MEMORY mode.

5.6.2.6.2 Set the Machine to Auto Mode.

a. Press MEM on the control panel.

b. Verify the tool and work offsets are correctly set.

5.6.2.6.3 Select 5 Percent Rapid Override.

a. Before starting the cycle, set the Rapid Override to 5 percent to ensure safe machine movements during initial execution.

b. This reduces rapid traverse speeds, allowing for better monitoring of the first run.

5.6.2.6.4 Start the Cycle.

a. Press CYCLE START to begin program execution.

b. Monitor the operation and be ready to press FEED HOLD or RESET if needed.

5.6.3 4th-Axis Head. The 4th-axis head, also known as a rotary table, expands a CNC milling center's capabilities by allowing rotational machining along the A-axis. This enables features such as complex contouring, indexing, and multi-sided machining without manual repositioning. Proper installation, alignment, and configuration are essential for precision and repeatability.

5.6.3.1 General Installation of the 4th-Axis Head.

5.6.3.1.1 Prepare the Machine and Work Area.

a. Power off the CNC machine and clean the machine table to remove debris.

b. Gather all necessary tools, including T-slot nuts, bolts, and precision indicators.

5.6.3.1.2 Positioning the 4th-Axis Head.

a. Carefully place the rotary table onto the machine bed using a hoist if needed.

b. Align the rotary table's mounting holes with the T-slots on the machine table.

5.6.3.1.3 Securing the 4th-Axis Head.

a. Insert T-slot nuts into the table and loosely fasten studs and nuts, or bolts.

b. Adjust the rotary table's position based on job requirements (centered or offset).

c. Fully tighten the mounting bolts in a crisscross pattern to ensure even clamping force.

5.6.3.1.4 Rough Alignment.

a. Place a dial indicator on the spindle and sweep across the rotary table surface.

b. Adjust the table until it is parallel with the X-axis within 0.001 inch tolerance.

5.6.3.1.5 Fine Alignment.

a. Indicate along a known precision surface of the rotary table (e.g., faceplate or chuck).

b. Adjust the mounting bolts incrementally until runout is minimized.

c. Lock down the bolts and recheck alignment to ensure stability.

5.6.3.1.6 Connect the Rotary Table to the Controller.

a. Plug the 4th-axis motor cable into the designated rotary port on the control cabinet.

b. Ensure the rotary axis is enabled in the parameter's menu (settings > rotary axis).

5.6.3.1.7 Set Work Offsets and Fixture Offsets.

a. Use G54-G59 work offsets to define the rotary table's zero position.

b. Set G68 rotational offsets if necessary for angular machining.

5.6.3.1.8 Verify Axis Movement.

a. Manually jog the A-axis to confirm proper movement and zero return.

b. Test rotation using MDI Mode with simple commands (e.g., G00 A90).

5.6.3.1.9 Verify Post-Processor Settings.

a. Ensure the CAM software is configured for 4-axis machining.

b. Use the appropriate 4th axis post-processor.

c. Verify toolpath simulations before executing programs.

5.6.4 5th Axis Trunnion Table. The 5th axis trunnion table allows simultaneous movement along both the A and B axes, enabling full 5-axis machining. Proper installation and calibration are critical to achieving accurate multiaxis machining.

5.6.4.1 General Installation of the 5th Axis Table.

5.6.4.1.1 Prepare the Machine and Work Area.

a. Power off the CNC machine and clean the machine table thoroughly.

b. Gather mounting hardware, precision indicators, and required tools.

5.6.4.1.2 Positioning the Trunnion Table.

a. Use a hoist if necessary to safely place the trunnion table on the machine bed.

b. Align the mounting holes of the table with the T-slot positions on the machine bed.

5.6.4.1.3 Securing the Trunnion Table.

a. Insert T-slot nuts and bolts, then lightly tighten them.

b. Adjust the table's position based on setup needs.

c. Securely tighten all bolts in a balanced pattern to prevent misalignment.

5.6.4.1.4 Rough Alignment.

a. Use a dial indicator to sweep the surface of the trunnion table.

b. Adjust for parallelism with the X and Y axes.

5.6.4.1.5 Fine Alignment.

a. Indicate along the rotational axes (A and B) and adjust until within tight tolerances.

b. Check perpendicularity using a test bar or gauge block.

c. Secure and recheck alignment to ensure accuracy.

5.6.4.1.6 Connect the Trunnion Table to the Controller.

a. Attach the A and B-axis motor cables to their respective rotary ports.

b. Enable 5-axis control in the parameter's menu.

5.6.4.1.7 Set Work and Fixture Offsets.

a. Establish G54-G59 work offsets for primary positioning.

b. Use G68 for rotation adjustments and fixture calibration.

c. Confirm all offsets are correctly stored and tested.

5.6.4.1.8 Verify Axis Movement.

a. Jog the A and B axes to check functionality and zero return.

b. Run test rotations via MDI mode (G00 A90 B45).

5.6.4.1.9 Post-Processor Settings for 5th Axis.

a. Use the appropriate 5-axis post-processor in CAM software.

b. Define tool orientation for accurate multi axis toolpaths.

c. Simulate toolpaths thoroughly before machining.

5.7 GENERAL COOLANT SYSTEM REQUIREMENTS.

The coolant system in a CNC milling center plays a crucial role in heat dissipation, lubrication, and chip evacuation. Selecting the appropriate coolant type and maintaining proper coolant-to-water ratios ensures efficient machining and extends tool life. See the table below for general information on coolant types and refractory recommendations.

Table 5-20. General Coolant Guidelines

Coolant Type	Description	Recommended Use	Maintenance Guidelines
Water-Soluble	General-purpose coolant with good cooling and lubrication	Ferrous and non-ferrous metals	Monitor concentration (6-10 percent), check for bacterial growth
Semi-Synthetic	Blend of synthetic and oil-based coolants, enhanced lubrication	High-speed machining	Maintain mix ratio, periodically clean sumps
Synthetic	Fully synthetic, excellent cooling, minimal residue	High-precision machining	Avoid contamination, check concentration regularly

5.8 GEOMETRIC AND MACHINE FUNCTIONS.

CNC milling centers operate using a standardized set of G-codes (geometric functions), M-codes (machine functions), and other system commands. Understanding these codes is crucial for programming, troubleshooting, and optimizing machine operations. G-codes are responsible for controlling tool movement, positioning, and machining cycles. (See Table 5-21.)

Table 5-21. Geometric Commands (G-Codes)

Code	Function	Description
G00	Rapid Positioning	Moves the tool at maximum speed to a specified location
G01	Linear Interpolation	Moves the tool in a straight line at a set feed rate
G02	Circular Interpolation (clockwise)	Moves the tool in a clockwise arc
G03	Circular Interpolation (counterclockwise)	Moves the tool in a counterclockwise arc
G04	Dwell	Pauses tool movement for a specified duration
G17	XY Plane Selection	Defines the XY plane for circular interpolation
G18	ZX Plane Selection	Defines the ZX plane
G19	YZ Plane Selection	Defines the YZ plane
G20	Inch Mode	Sets the machine to interpret dimensions in inches
G21	Metric Mode	Sets the machine to interpret dimensions in mm
G28	Machine Home Return	Moves the tool to the machine home position

Table 5-21. Geometric Commands (G-Codes) - Continued

Code	Function	Description
G40	Cutter Compensation Cancel	Disables tool diameter compensation
G41	Cutter Compensation Left	Offsets the tool path to the left
G42	Cutter Compensation Right	Offsets the tool path to the right
G43	Tool Length Compensation (+)	Adjusts tool height using, a positive offset
G49	Tool Length Compensation Cancel	Disables tool length offsets
G54-G59	Work Offset Coordinates	Defines custom work offsets for fixture positioning
G68	Coordinate Rotation	Rotates the coordinate system for angled machining
G73	High-speed Peck Drilling	Performs rapid, efficient peck drilling
G81	Standard Drilling Cycle	Executes a single-hole drilling operation
G83	Deep Hole Peck Drilling	Uses multiple pecks to drill deep holes
G90	Absolute Positioning	All movements are based on a fixed coordinate system
G91	Incremental Positioning	All movements are relative to the tool's current position
G98	Return to Initial Plane	Sets the tool return level after drilling
G99	Return to R-Plane	Returns the tool to the clearance plane

Table 5-22. Machine Commands (M-Codes)

Code	Function	Description
M00	Program Stop	Stops the machine until manually restarted
M01	Optional Stop	Stops only if the **Optional Stop** feature is active
M02	End of Program	Stops the program and resets it to the beginning
M03	Spindle On (clockwise)	Starts the spindle in a clockwise direction
M04	Spindle On (counterclockwise)	Starts the spindle in a counterclockwise direction
M05	Spindle Stop	Turns off the spindle
M06	Tool Change	Executes an automatic tool change
M07	Mist Coolant On	Activates mist coolant (if equipped)
M08	Flood Coolant On	Activates flood coolant
M09	Coolant Off	Turns off all coolant functions
M30	Program End and Reset	Ends a program and resets the machine
M98	Call Subprogram	Calls an external subroutine
M99	Subprogram Return	Returns to the main program after executing a subroutine

CHAPTER 6
METAL CUTTING SAWS

6.1 GENERAL.

Metal cutting industrial saws are essential machines in Aircraft Metals Technology (AMT) and Aircraft Structural Maintenance (ASM) shops, providing precision and efficiency in cutting various metals. These machines are designed to cut through a wide range of materials, including steel, aluminum, and other alloys, and they are critical in manufacturing, construction, and metalworking industries. Industrial saws operate using various cutting methods, such as friction cutting, abrasive cutting, and precision blade cutting, each suited for different applications. The efficiency of these machines depends on factors such as blade material, tooth geometry, cutting speed, and feed rate. Proper selection of these parameters ensures optimal performance, extended tool life, and reduced material waste. The cutting process in industrial sawing involves forces that generate heat and produce metal chips or debris. To manage this, many sawing machines incorporate coolant systems to regulate temperature, reduce friction, and improve blade longevity. Modern advancements in sawing technology have led to the development of automated and Computer Numerical Control (CNC)-controlled saws, enhancing accuracy and repeatability. These advanced systems allow for programmable cutting operations, reducing manual labor and increasing production efficiency. Understanding the fundamentals of metal cutting sawing machines, including their operational principles, material capabilities, and efficiency factors, is crucial for selecting the right machine for specific applications. Proper training and adherence to operational guidelines further enhance safety and performance in industrial sawing processes.

6.2 SAFETY.

Safety is paramount when operating saws, as improper use can result in severe injuries. Operators must always adhere to safety guidelines set forth by Occupational Safety and Health Administration (OSHA) standards (29 Code of Federal Regulations (CFR) 1910.) and applicable Air Force Instructions (AFIs) and Air Force Manuals (AFMAN) (e.g., DAFMAN 91-203). Supervisors are responsible for maintaining manufacturer manuals and integrating them into on-the-job training and job safety training. In the absence of manufacturer guidance, a hazard analysis and local operating instructions must be developed. Operators must:

- Wear safety glasses, face shields (if necessary), hearing protection, cut-resistant gloves (when applicable).
- Ensure all machine guards, barriers, and other safety devices are used to prevent accidental contact with moving blades.
- Follow lockout/tagout procedures before performing maintenance, blade changes, or troubleshooting to prevent accidental startup.
- Check saw blades for wear, cracks, or missing teeth. Any damaged blades must be replaced immediately to prevent breakage and ensure precise cuts.
- Be familiar with emergency stop buttons and safety features to understand how to quickly shut down the machine in case of an emergency.
- Remove loose clothing or jewelry, and long hair must be secured to prevent entanglement with moving parts.
- Keep the work area free from metal shavings, excess lubricant, and debris to reduce trip hazards and minimize fire risks.

6.3 TYPES OF METAL CUTTING SAWING MACHINES.

6.3.1 Industrial Sawing Machines. Industrial sawing machines come in various designs, each suited for specific applications based on material type, precision requirements, and production volume. The following are the primary types of metal cutting saws used in machining and fabrication shops.

6.3.2 Band Saws. Band saws are among the most versatile and commonly used industrial sawing machines. They employ a continuous looped blade with evenly spaced teeth to cut a variety of metals. Band saws come in two primary configurations:

6.3.2.1 Horizontal Band Saws. (See Figure 6-1.) Used primarily for cutting large workpieces, horizontal band saws provide consistent and precise cuts with minimal material waste. They are ideal for structural steel, tubing, and bar stock.

6.3.2.2 Vertical Band Saws. (See Figure 6-2.) These machines allow for intricate and curved cuts, making them suitable for complex fabrication tasks. Operators manually guide the workpiece through the stationary blade, offering greater flexibility in cutting shapes.

TO-34-1-10-110

Figure 6-1. Horizontal Band Saw

TO-34-1-10-111

Figure 6-2. Vertical Band Saw

6.3.3 Circular Saws. Circular saws use a rotating toothed blade or abrasive cutting wheel/blade to perform high-speed, precision cuts. These saws are suitable for high-production environments due to their fast-cutting speeds and clean-cut edges.

6.3.3.1 Cold Saws. (See Figure 6-3.) These use a toothed blade and operate at lower speeds with coolant, minimizing heat buildup and producing burr-free cuts. Cold saws are commonly used for cutting tubing, pipes, and solid bars with high accuracy.

6.3.3.2 Abrasive Chop Saws. (See Figure 6-4.) Designed for cutting hard metals, abrasive chop saws use a high-speed spinning disc that grinds through material rather than slicing it. These saws generate significant heat and are often used for rough cuts.

TO-34-1-10-112

Figure 6-3. Cold Saw

TO-34-1-10-113

Figure 6-4. Abrasive Chop Saw

6.4 SAWING MACHINE COMPONENTS.

Understanding the key components of different sawing machines is essential for ensuring proper operation, maintenance, and troubleshooting. The following sections provide a detailed breakdown of the primary components found in various types of industrial metal cutting saws, including horizontal band saws, vertical band saws, cold saws, and abrasive chop saws. By familiarizing yourself with these components, operators can optimize machine performance, troubleshoot issues efficiently, and extend the life of the equipment.

6.4.1 Horizontal Band Saw Components.

6.4.1.1 Frame. The robust frame supports all other components and absorbs vibrations during cutting. Constructed from high-strength materials, the frame ensures stability and precision throughout the cutting process.

6.4.1.2 Blade Drive Motor. This motor powers the blade, providing the necessary torque and speed for efficient cutting. Depending on the saw type, it can be a variable-speed motor for flexibility in cutting different materials.

6.4.1.3 Blade Wheels. These wheels guide the band saw blade in a continuous loop. Made of durable materials like cast iron or steel, they ensure smooth blade movement and stability.

6.4.1.4 Blade Tension Mechanism. This system maintains the appropriate tension on the band saw blade, ensuring proper cutting alignment and preventing blade wear.

6.4.1.5 Cutting Head. The cutting head holds the saw blade and moves it through the material. On automatic machines, the head is usually adjustable for various cutting angles and depths.

6.4.1.6 Hydraulic System. Often used for controlling the cutting head's feed rate, this system ensures smooth and consistent cutting. It also controls the downfeed pressure to ensure that the blade cuts through material efficiently.

6.4.1.7 Coolant System. Coolant systems maintain the temperature of the blade during operation, reducing heat buildup that can lead to blade wear. They also flush away debris and improve cutting accuracy.

6.4.1.8 Material Feed System. A conveyor or vice holds the workpiece in place and feeds it through the cutting area. These systems can be manual or automated, depending on the saw's design.

6.4.2 Vertical Band Saw Components.

6.4.2.1 Frame and Base. The frame holds the entire assembly together and supports the motor, guides, and other key components. A stable, solid base ensures precision cuts, especially for taller workpieces.

6.4.2.2 Motor. The motor drives the blade's rotation. Vertical band saws typically use smaller, less powerful motors than horizontal saws due to the different cutting dynamics.

6.4.2.3 Blade Guides. These components stabilize the blade during cutting and are critical for maintaining accurate cuts. Adjustable blade guides allow for different blade sizes and materials.

6.4.2.4 Blade Tension System. Similar to horizontal models, vertical saws utilize a blade tensioning system to ensure proper alignment and consistent cutting performance.

6.4.2.5 Worktable. A large, flat work surface supports the material as it is fed into the saw. Some tables can be tilted for angled cuts, enhancing the versatility of the saw.

6.4.2.6 Coolant Delivery System. In vertical band saws, coolant is applied directly to the cutting area to reduce heat, prolong blade life, and improve cutting quality.

6.4.2.7 Blade Shear. The blade shear is a mechanism used to cut the band saw blade to the desired length and to ensure a clean, precise edge. It helps maintain blade integrity by reducing the risk of unnecessary stress or damage to the blade during setup.

6.4.2.8 Blade Grinder. The blade grinder is used to resharpen the band saw blade. Regular grinding helps maintain the blade's cutting efficiency, ensuring smooth, accurate cuts. It removes built-up material from the teeth and restores the proper cutting profile.

6.4.2.9 Blade Welder. The blade welder is used to join the ends of a band saw blade, creating a continuous loop. It typically uses high heat and pressure to fuse the metal, making it essential for maintaining the saw blade's structural integrity and preventing downtime when the blade is damaged.

6.4.3 Cold Saw Components.

6.4.3.1 Blade and Arbor. Cold saws use circular blades made from High-Speed Steel (HSS) or carbide-tipped materials. The arbor holds the blade in place and allows it to rotate at high speeds.

6.4.3.2 Motor. A powerful motor drives the blade, delivering consistent cutting force. Cold saws often have variable-speed motors to adjust for different materials and cutting conditions.

6.4.3.3 Saw Arm. The saw arm holds the blade and moves it toward the material. This movement is typically guided by a hydraulic or pneumatic system for controlled, smooth operation.

6.4.3.4 Clamping System. The workpiece is held in place with a clamping system that can be adjusted to accommodate various sizes and shapes. The clamping system is designed to hold the material securely during cutting.

6.4.3.5 Coolant System. Cold saws rely heavily on coolant systems to keep both the blade and the workpiece cool. This minimizes heat buildup and maintains cutting accuracy.

6.4.3.6 Chip Removal System. Cold saws also incorporate a system for removing chips and debris from the cutting area to maintain clear visibility and prevent obstruction during operation.

6.4.4 Abrasive Chop Saw Components.

6.4.4.1 Motor. The motor in an abrasive chop saw drives a rotating abrasive disc or wheel. The motor is typically designed to handle the high demands of cutting hard materials like steel or metal.

6.4.4.2 Abrasive Wheel. These are the cutting tools in abrasive chop saws, typically made from aluminum oxide or other tough materials. They work by grinding away material through friction.

6.4.4.3 Work Clamp. A clamp or vise holds the material in place during cutting. The clamp is adjustable to accommodate different workpiece sizes and shapes.

6.4.4.4 Safety Guard. A crucial safety feature, the guard prevents sparks and debris from flying out during cutting. It also protects the operator from the rotating abrasive wheel.

6.4.4.5 Blade Support. The blade support system helps stabilize the abrasive wheel and ensures consistent pressure during the cut, preventing wobbling and ensuring smooth operation.

6.4.4.6 Coolant or Dust Collection System. While abrasive saws don’t use liquid coolant, many include dust collection systems to reduce airborne debris and improve the working environment.

6.5 TOOLS AND ATTACHMENTS.

The versatility of industrial sawing machines is often enhanced by specialized tools and attachments. These additional components allow for more complex cutting operations, improved precision, and better material handling. The following outlines the most common tools and attachments for horizontal saws, vertical saws, cold saws, and abrasive chop saws.

6.5.1 Tools and Attachments for Horizontal Band Saws.

6.5.1.1 Roller Tables. Used to support long or heavy workpieces, roller tables help feed the material into the cutting area with minimal effort. These can be positioned before and after the saw for easy material movement.

6.5.1.2 Automatic Feed Systems. These systems control the speed at which the material is fed into the saw. Often programmable, they allow for consistent cutting of multiple pieces without manual adjustments.

6.5.1.3 Cutting Angle Adjusters. These attachments allow the operator to adjust the angle of the cutting head, enabling more precise cuts for angled workpieces.

6.5.1.4 Material Vices. Vices hold the workpiece securely during cutting. Some models feature pneumatic or hydraulic systems to provide a firm grip and reduce vibrations during operation.

6.5.1.5 Blade Brush. The blade brush is used to clean the saw blade, removing debris and cutting residue to maintain cutting efficiency and reduce wear on the blade.

6.5.1.6 Chip Removal Systems. These systems collect chips and debris from the cutting area to prevent clogging and maintain a clear view of the cut. They often use blowers or vacuum systems for removal.

6.5.2 Tools and Attachments for Vertical Band Saws.

6.5.2.1 Table Extensions. These provide additional support for larger workpieces and can be easily adjusted or removed to accommodate different sizes of material.

6.5.2.2 Miter Gauges. Used to make precise angle cuts, miter gauges can be adjusted to a variety of angles and help ensure uniformity in cuts, especially for cross-cutting operations.

6.5.2.3 Tilting Table Attachments. Some vertical band saws have attachments that allow the worktable to be tilted for angled cuts, adding flexibility to the machine.

6.5.2.4 Blade Cleaning Brushes. Similar to horizontal saws, vertical saws often use cleaning brushes to remove sawdust and debris from the blade, preventing material buildup that could affect cutting performance.

6.5.2.5 Workpiece Support Arms. These adjustable arms provide extra support for larger or heavier workpieces, reducing the likelihood of bending or movement during cutting.

6.5.2.6 Sanding or Polishing Attachments. Some vertical saws have optional attachments that allow for post-cut finishing work, such as sanding or polishing the cut edge.

6.5.3 Tools and Attachments for Cold Saws.

6.5.3.1 Chip Guards. Cold saws produce metal chips during cutting, and chip guards are used to contain and direct the chips into a collection bin, preventing them from interfering with the cutting process.

6.5.3.2 Variable-Speed Drives. Some cold saws are equipped with variable-speed drives, allowing operators to adjust the cutting speed based on the material being cut and the desired finish.

6.5.3.3 Cooling Units. To maintain optimal operating conditions, some cold saws are equipped with external cooling units that regulate coolant temperature, ensuring it doesn’t overheat during high-volume cutting.

6.5.3.4 Clamping Jaws. These specialized jaws grip the material securely during cutting, offering additional stability for more precise cuts, especially in high-tolerance operations.

6.5.3.5 Material Feeders. These attachments automatically advance the material through the saw, allowing for more efficient and continuous cutting, often used in mass production environments.

6.5.4 Tools and Attachments for Abrasive Chop Saws.

6.5.4.1 Blade Guards. A critical safety feature, blade guards protect the operator from sparks and debris generated during cutting. Some models feature hinged or adjustable guards for easy access.

6.5.4.2 Dust Collection Systems. Abrasive chop saws can create a significant amount of dust and debris. Attachments like dust bags or external collection units help capture this material, keeping the work environment cleaner and safer.

6.5.4.3 Quick-Release Clamps. These clamps allow the operator to quickly secure and release the workpiece, improving efficiency when changing materials or resetting the saw for new cuts.

6.5.4.4 Splash Guards. Often used in conjunction with coolant systems, splash guards help contain the coolant and prevent it from splattering during operation, ensuring cleaner work surfaces.

6.5.4.5 Abrasive Wheels. These wheels are commonly used for cutting metals and are made of a tough abrasive material like aluminum oxide. The abrasive chop saw uses the friction of the wheel to grind through the workpiece. Abrasive wheels come in various grades, allowing operators to choose the right one depending on the material hardness and desired finish.

6.5.4.6 Carbide-Tipped Blades. Carbide blades are used for cutting harder materials such as steel and metal alloys. These blades are designed to withstand high temperatures and wear, providing longer tool life and better cutting performance than standard abrasive wheels, especially in heavy-duty cutting tasks.

6.5.4.7 Adjustable Cutting Jigs. For repetitive cuts, adjustable jigs allow the operator to set the exact cutting angle and workpiece position. This attachment increases the efficiency of production runs and improves cut accuracy.

6.6 SAW BLADES.

Saw blades are crucial components in the performance of both vertical and horizontal saws. Understanding the specific characteristics (Figure 6-5) of saw blades, such as their types, materials, pitch, kerf, tooth set, and width, ensures that the right blade is selected for each job. The following sections provide a detailed overview of each of these important blade features.

6.6.1 Types of Saw Blades.

6.6.1.1 Band Saw Blades. Used in both vertical and horizontal saws, these blades are made of continuous steel and are available in various sizes and configurations. They are primarily used for cutting metal, wood, plastics, and composites.

6.6.1.1.1 Standard Teeth. The most common configuration, designed for general-purpose cutting.

6.6.1.1.2 Skip Teeth. These blades have wider teeth spacing, making them ideal for cutting thick, dense materials without clogging.

6.6.1.1.3 Hook Teeth. Designed for fast, aggressive cutting, hook teeth are used when high-speed, efficient cuts are required.

6.6.1.1.4 Variable Pitch. These blades feature teeth with a varying pitch to reduce noise, vibration, and harmonics during cutting, improving cutting efficiency and finish.

6.6.1.2 Cold Saw Blades. These circular saw blades are typically made of HSS or carbide-tipped materials, designed for cutting metal with minimal heat generation. Cold saw blades are ideal for clean, precise cuts, particularly in high-tolerance applications.

6.6.1.3 Abrasive Saw Blades. These blades are used in abrasive chop saws, typically made from a composite of abrasive particles and a bonding agent. They are primarily used for cutting hard materials such as steel, stainless steel, or cast iron.

6.6.2 Blade Materials and Application. (See Table 6-1.)

6.6.2.1 Carbon Steel. Commonly used for general-purpose blades, carbon steel offers good flexibility and durability at an affordable price. However, it may wear more quickly when cutting harder materials.

6.6.2.2 Bi-Metal. Made by welding a high-speed steel tooth to a flexible steel back, bi-metal blades are durable and resistant to heat, making them suitable for cutting a wide range of materials. These blades offer a good balance between strength and flexibility.

6.6.2.3 HSS. Known for its ability to retain sharpness at high temperatures, HSS blades are used in applications where extended tool life and cutting efficiency are needed, especially for harder materials.

6.6.2.4 Carbide-Tipped. Carbide-tipped blades feature tips made of tungsten carbide, providing superior wear resistance and cutting power. These blades are ideal for cutting tough materials, such as stainless steel or high-alloy metals.

6.6.2.5 Tungsten Carbide. These blades are extremely hard and durable, ideal for cutting abrasive or high-strength materials, offering extended life and reduced wear.

Table 6-1. Band Saw Blade Selection

Blade Material	Material to be Cut	Pitch	Kerf	Tooth Set	Width
High Carbon Steel	Wood, Plastics, Non-Ferrous Metals	4-6 teeth per inch	Narrow	Raker, Wave	1/4 to 3/4
Bi-Metal (HSS Teeth)	Carbon Steels, Stainless Steel, Alloys	3-4 teeth per inch for thicker materials, 6-10 teeth per inch for thinner	Standard (slight)	Variable, Positive	1/2 to 1-1/4
Cobalt Steel	High-Strength Steels, Titanium, Alloy Steels	2-3 teeth per inch for thick materials, 6-8 teeth per inch for thin	Standard (moder-ate)	Variable, Raker	1/4 to 1 inch
Carbide-Tipped	Hard Metals, Tool Steels, Cast Iron	2-4 teeth per inch for tough cuts	Wide (for toughness)	Positive, Raker	3/4 to 1-1/2
Tungsten Carbide	Harder Materials, High-alloy Steel, Stainless Steel	1-3 teeth per inch for heavy cutting, 4-6 teeth per inch for light cuts	Wide (aggressive)	Variable, Heavy Duty	1/2 to 1-1/2

6.6.3 Pitch of a Band Saw Blade. The pitch of a band saw blade refers to the distance between the teeth. The correct pitch ensures efficient cutting performance and affects the blade’s cutting speed, material feed rate, and cut quality. At least two or three teeth must be in contact with the workpiece at all times to prevent snagging and tooth breakage. When cutting sheet metal or tubing, select a blade with a sufficient pitch to maintain this contact, regardless of the material type.

6.6.3.1 Fine Pitch. A fine pitch (typically 14-24 TPI) is used for cutting thin materials or for precision work where a smooth, clean cut is essential. Fine pitch blades reduce the risk of vibration and provide better finish quality.

6.6.3.2 Medium Pitch. Medium-pitch blades (around 8-14 TPI) are versatile and suitable for a wide range of materials, offering a good balance of cutting speed and finish quality.

6.6.3.3 Coarse Pitch. Coarse pitch blades (around 4-8 TPI) are best for cutting thicker materials or those with a high hardness level. Coarse teeth remove material quickly and are ideal for fast, rough cuts.

6.6.3.4 Variable Pitch. Blades with variable pitch have teeth with varying spacing, which reduces vibrations, improves cutting efficiency, and produces a smoother finish. They are particularly useful for cutting materials with varying hardness or thickness.

6.6.4 Kerf of a Band Saw Blade. The kerf of a band saw blade is the width of the cut made by the blade. A smaller kerf results in less material waste, while a larger kerf provides greater strength and stability.

6.6.4.1 Narrow Kerf. Narrow kerf blades are designed to minimize material waste and provide faster cutting speeds. They are ideal for cutting thin materials or for applications where material conservation is critical.

6.6.4.2 Wide Kerf. Wide kerf blades offer greater rigidity and strength, making them ideal for cutting thicker or harder materials. While they create more waste, they provide stability and accuracy during heavy-duty cutting.

6.6.5 The Tooth Set of a Bland Saw Blade. The tooth set of a bland saw blade refers to the way the teeth are configured along the blade. The set affects the quality of the cut, material handling, and the lifespan of the blade.

6.6.5.1 Raker Set. In this arrangement, teeth alternate in left and right directions, creating a wider cut. This pattern is common for general-purpose cutting and is particularly effective for thicker materials.

6.6.5.2 Wave Set. Teeth are set in a wavy pattern, which provides smoother cuts with less friction. This set is used for cutting wood or non-ferrous materials and minimizes heat buildup.

6.6.5.3 Straight Set. The teeth are set in a straight line, providing a narrower kerf and is often used for thin material or when precision cuts are necessary.

6.6.5.4 Double-Set. A more complex tooth set where teeth are set in alternating left and right directions, and then some teeth are set in a double pattern. This configuration is useful for high-speed cutting, especially for tough materials.

6.6.6 The Width of the Band Saw Blade. The width of the band saw blade influences its ability to make straight cuts and its overall stability.

6.6.6.1 Narrow Blades. Narrow blades are more flexible and can be used for tight radius cutting. However, they are more prone to flexing and may wear faster when cutting harder materials.

6.6.6.2 Wide Blades. Wider blades provide greater stability and are less likely to bend during cutting, making them suitable for straight cuts, especially in heavy-duty applications. They are ideal for cutting thicker materials.

6.6.7 Metal Cutting Circular Saw Blades. Metal cutting circular saw blades are classified using several key characteristics that determine their suitability for specific materials and cutting conditions. Unlike bandsaw blades, which are commonly classified by pitch teeth per inch (TPI), circular saw blades are typically identified by overall tooth count and geometry, blade size, material composition, and application-specific factors. These attributes directly affect the blade's cutting speed, surface finish, and service life. The primary classification criteria are as follows:

6.6.7.1 Tooth Count (Total Teeth). Indicates the number of cutting teeth on the blade. Higher tooth counts generally produce smoother cuts in thinner materials, while lower tooth counts are preferred for thicker stock or faster cutting.

6.6.7.2 Blade Diameter. Specifies the overall size of the blade, typically ranging from 5-3/8 to 14 inches for handheld and stationary machines. Larger diameters allow for deeper cuts and more teeth.

6.6.7.3 Tooth Geometry. Refers to the shape and grind of each tooth. Common geometries include:

Triple Chip Grind

Ideal for non-ferrous metals and hard materials.

Alternate Top Bevel

Provides a cleaner finish, typically used in multi-purpose blades.

Flat Top

Strong and durable, often used in rough cutting operations.

6.6.7.4 Kerf Width. The thickness of the cut made by the blade. Thin kerf blades reduce material loss and require less cutting power, beneficial for light-duty metal cutting.

6.6.7.5 Blade Material and Tip Composition.

Carbide-Tipped Blades

Common for ferrous and non-ferrous metals.

Cermet-Tipped Blades

Offer longer life and cleaner cuts, often used in cold saws.

Abrasive Composite Blades

Used for high-speed saws and less precise applications.

6.6.7.6 Revolution Per Minute (RPM) Rating (Maximum Operating Speed). Determines the blade's safe operating speed. Many metal cutting blades are rated for use on low-RPM machines (e.g., cold saws), while others are designed for high-RPM tools (e.g., abrasive chop saws).

6.6.8 Abrasive Cutting Wheels. Abrasive cutting wheels, commonly used on chop saws, stationary cutoff saws, and hand-held cut-off tools, are classified based on their construction, abrasive material, bond type, size, and application-specific features. These wheels are designed to grind through material rather than shear it like toothed blades. Proper classification ensures compatibility with the material being cut and safe operation at the appropriate RPM. The primary classification criteria include:

6.6.8.1 Wheel Diameter and Thickness. Denotes the physical size of the wheel. Common diameters range from 4 to 16 inches, with thicknesses typically between 1/16 and 1/4 inch. Thinner wheels allow for faster cuts with less material loss, while thicker wheels provide greater durability.

6.6.8.2 Abrasive Material Type. Indicates the type of grit used in the wheel. Common types include:

Aluminum Oxide

Suitable for ferrous metals such as mild steel and carbon steel.

Silicon Carbide

Used for non-ferrous metals, cast iron, and softer materials.

Zirconia Alumina

Offers long life and aggressive cutting for tough metals.

Ceramic Alumina

High-performance abrasive used for stainless steel and hardened alloys.

6.6.8.3 Bond Type. Refers to the method used to hold abrasive grains together:

Resinoid Bond (Type R)

The most common for cut-off wheels; provides flexibility, impact resistance, and high-speed capability.

Rubber or Shellac Bonds

Rare in cutoff wheels, more common in precision grinding.

6.6.8.4 Reinforcement (Fiberglass Mesh). Many abrasive wheels are reinforced with one or more layers of fiberglass mesh to increase strength and resist breakage under stress.

6.6.8.5 Maximum RPM Rating. Specifies the highest safe rotational speed for the wheel. This must match or exceed the rated speed of the machine. Overspeeding can result in wheel failure.

6.6.8.6 Wheel Type and Shape (American National Standards Institute (ANSI) Type Codes). Identifies the geometry of the wheel. Common types include:

Type 1

Flat, straight-sided wheel for straight cuts.

Type 27

Depressed center wheel used for flush cutting and grinding.

Type 41

Reinforced version of Type 1.

Type 42

Reinforced depressed center version of Type 27.

6.6.8.7 Grit Size and Hardness Grade (Optional Markings). May include details such as:

Grit Size

Coarse (24–36) for fast stock removal; fine (60+) for smoother finishes.

Grade

Indicates the hardness of the bonding matrix (soft, medium, or hard).

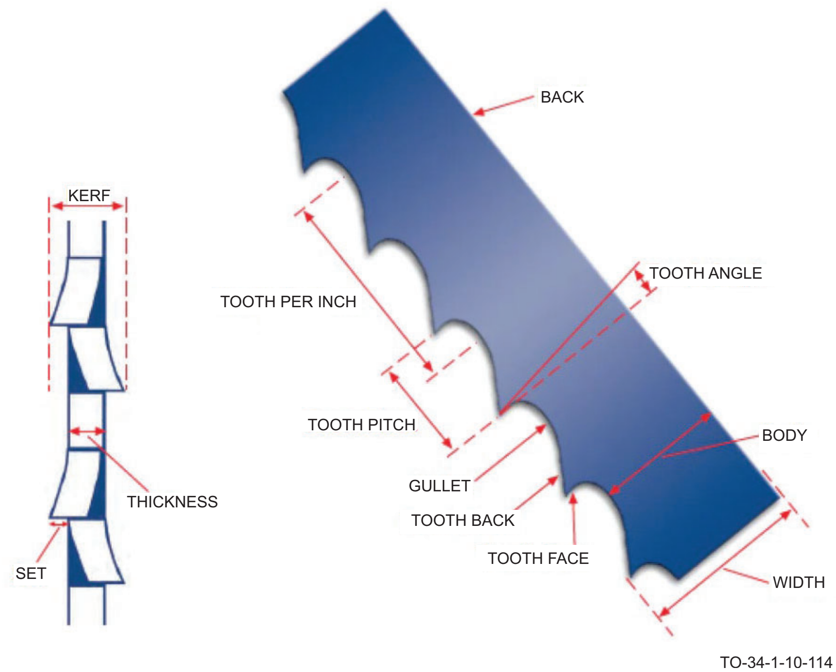


Figure 6-5. Bandsaw Blade Features

6.7 SAWING SPEEDS.

Sawing speed, measured in Feet Per Minute (FPM), is a critical factor in optimizing cutting performance, extending blade life, and ensuring efficient material removal. The correct speed depends on several variables, including the material being cut, the type of blade, and the presence of coolant or lubrication.

6.7.1 Factors Influencing Sawing Speed.

6.7.1.1 Material Hardness and Composition. Softer materials such as aluminum and brass can be cut at much higher speeds, often exceeding 3,000 FPM, whereas harder alloys like tool steel and Inconel require significantly lower speeds to prevent excessive blade wear.

6.7.1.2 Blade Material and Tooth Design. Carbon steel blades are generally used at lower speeds due to their limited heat resistance, while bi-metal and carbide-tipped blades can operate at much higher speeds due to their superior wear resistance.

6.7.1.3 Coolant and Lubrication. The application of cutting fluids helps dissipate heat, allowing for higher sawing speeds while reducing blade wear and preventing work-hardening of certain alloys like stainless steel.

6.7.1.3.1 Proper selection and regulation of sawing speeds (Table 6-2) play a key role in maximizing efficiency, reducing operational costs, and improving overall cut quality. Understanding these parameters ensures that band saw blades perform optimally while minimizing downtime due to premature wear or blade failure.

6.7.2 Speed Adjustments and Considerations.

6.7.2.1 Higher Speeds for Efficiency. When cutting non-ferrous metals and soft materials, a higher FPM results in cleaner cuts and faster processing times.

6.7.2.2 Slower Speeds for Harder Materials. Cutting hard metals at excessive speeds generates heat and accelerates tool wear. Using a lower FPM allows for controlled cutting and longer blade life.

6.7.2.3 Variable Speed Controls. Modern saws with adjustable speed settings provide flexibility to optimize FPM for different materials and blade types. Operators should always consult manufacturer recommendations for their specific saw and blade combination.

Table 6-2. Sawing Speeds by Materials and Blade Type

Material	Carbon Steel Blade (FPM)	Bi-Metal Blade (FPM)	Carbide-Tipped Blade (FPM)
Aluminum and Soft Metals	1,000-3,000	1,500-3,500	2,500-4,000
Mild Steel (Low Carbon)	100-300	200-400	500-800
Stainless Steel	50-150	100-250	250-600
Tool Steel	40-120	80-200	200-500
Titanium and Alloys	30-80	60-150	150-400
Cast Iron (Ductile)	50-150	100-250	250-600
Cast Iron (Gray)	40-100	80-200	200-500
Inconel and Super Alloys	20-60	40-120	100-300
Hardened Steels (HRC >45)	10-40	20-80	100-250

6.8 SAWING OPERATIONS.

Proper workpiece layout is essential for efficient material utilization, minimizing waste, and ensuring accuracy in the sawing process. Factors to consider include grain direction (for composite and sheet metal materials), part orientation, and optimal nesting of parts for production efficiency.

6.8.1 Contours.

Band Saw Contour Cutting

Used to cut curved or irregular shapes from stock material. Requires precise layout marking and blade selection appropriate for the material and contour complexity.

Template Usage

In aerospace and sheet metal applications, templates or CNC-programmed cutting paths ensure accuracy and repeatability.

Cut Sequencing

Avoid sharp directional changes that can cause blade deflection. Consider relief cuts for intricate patterns.

6.8.2 Starting Cuts in Holes.

Drilled Pilot Holes

Used when starting an internal cut. The hole must be large enough for the blade width to pass through without binding.

Blade Threading

For bandsaws with a blade welder, the blade can be cut, threaded through a drilled hole, and rewelded in place before resuming cutting.

Plunge Cutting

Certain vertical bandsaws allow controlled plunge cutting by angling the workpiece and lowering it onto the moving blade.

6.8.3 Clamping and Fixturing.

Fixed vs. Adjustable Clamps

Use fixed clamps for repeatability in production settings and adjustable clamps for versatile setups.

Soft Jaws and Fixtures

Essential for holding thin-walled aerospace materials to prevent deformation.

Magnetic and Vacuum Fixtures

Useful for non-ferrous sheet metal work where mechanical clamping is impractical.

Dampening Methods

Vibration control is critical when sawing thin or exotic materials to prevent workpiece movement and excessive tool wear.

6.9 GENERAL BLADE CHANGING INSTRUCTIONS.

6.9.1 Horizontal Saw Blade Removal.

a. Ensure the machine is powered off and properly locked out/tagged out.

b. Use the blade tension release mechanism to remove strain from the blade.

c. Remove side guards to access the blade pulleys and guides.

WARNING

Use gloves when handling saw blades to avoid injury from sharp edges. Failure to comply could, result in injury to, or death of, personnel or long term health hazards.

d. Carefully remove old blade.

6.9.2 Horizontal Saw Blade Installation.

a. Ensure correct blade size and tooth configuration for the material.

b. Position the blade around the drive and idler wheels.

c. Adjust tracking: Rotate the wheels manually to ensure the blade aligns properly.

d. Tension the blade: Follow manufacturer’s recommended tension settings.

e. Verify guide alignment: Ensure proper engagement with blade guides and bearings.

f. Reinstall guards and perform test run: Check blade tracking under power before cutting.

6.9.3 Vertical Saw Blade Removal.

a. Power down and lockout/tagout.

b. Release blade tension.

c. Open access doors: Provides entry to the upper and lower wheels.

WARNING

Use gloves when handling saw blades to avoid injury from sharp edges. Failure to comply could, result in injury to, or death of, personnel or long term health hazards.

d. Remove old blade carefully.

6.9.4 Vertical Saw Blade Installation.

a. Ensure blade teeth face the correct cutting direction.

b. Position the blade around the wheels and through guide blocks.

c. Adjust blade tracking using tracking knob while manually turning the wheels.

d. Set proper blade tension.

e. Check guide block position to avoid lateral blade movement.

f. Close access doors and perform test run before making cuts.

6.10 GENERAL BLADE WELDING INSTRUCTIONS.

6.10.1 Utilizing Sawing Machine Welder.

a. Cut blade ends square using a shear or grinder.

b. Clean ends to remove debris and oxidation.

c. Align ends in the welding clamp, ensuring proper contact.

d. Set welding parameters (voltage and pressure) based on blade thickness.

e. Initiate weld cycle and allow the machine to complete the fusion.

f. Anneal the weld by applying controlled heating to reduce brittleness.

g. Grind weld joint smooth to ensure proper passage through blade guides.

h. Reinstall blade and verify operation.

6.10.2 Utilizing Tungsten Inert Gas Welding.

a. Square and clean blade ends.

b. Use a jig or clamp for precise alignment.

c. Use a fine tungsten electrode (1/16 or 3/32 inch).

d. Set amperage appropriate for blade material (typically 20-40A for carbon steel, higher for bimetal).

e. Use a low-heat, high-control technique to avoid overheating the blade.

f. Perform a fusion tack weld to hold alignment.

g. Complete the full weld, ensuring smooth penetration and minimal warping.

h. Anneal the weld zone to reduce brittleness. (See additional instructions and procedures in Paragraph 6.10.3.)

i. Grind and finish weld for a smooth surface.

j. Reinstall and test blade to ensure performance stability.

6.10.3 Post-Weld Torch Annealing.

a. Place the welded bandsaw blade on a firebrick or other heat-resistant surface.

b. Ensure it is flat and supported to prevent warping during heating.

c. Use a neutral flame (not carburizing or oxidizing) to evenly heat the welded section and the surrounding area (about 1/2 to 1 inch on each side).

d. Move the torch in a circular or side-to-side motion to distribute heat uniformly.

e. The ideal annealing temperature for most high-carbon and bi-metal bandsaw blades is between 600-900 degrees Fahrenheit (°F) (316-482 degrees Celsius (°C)).

f. If using a temperature-indicating crayon, apply it near the weld. The crayon will change color when the correct temperature is reached.

g. The steel should take on a dull red color in dim lighting.

h. Maintain the heat for 30-60 seconds, depending on blade thickness.

i. Avoid overheating, as it can weaken the blade.

j. Gradually remove the torch and allow the blade to cool slowly in still air.

k. Avoid quenching, as rapid cooling will harden the steel and make it brittle.

l. After cooling, test the flexibility of the blade.

m. If the weld area is still too hard, repeat the annealing process with slightly lower heat.

n. Check for any signs of cracks, warping, or excessive softening.

o. Lightly grind or polish the weld area if necessary to remove oxidation.

CHAPTER 7
WATERJET SYSTEMS

7.1 GENERAL.

Abrasive waterjet systems are high-performance cutting machines widely used in aerospace manufacturing and maintenance, repair, and overhaul environments and have become more common in Aircraft Metals Technology (AMT) shops. These systems utilize a high-pressure stream of water mixed with abrasive particles to cut through various materials with precision. Unlike traditional cutting methods, waterjets do not generate heat-affected zones, minimizing material distortion and preserving material properties. Their versatility enables the cutting of metals, composites, and other materials commonly found in aerospace applications. Waterjet cutting has advantages where excessive heat or stresses from other types of processes could compromise integrity and properties of the material. The ability to cut complex geometries with high accuracy, to reduce secondary finishing operations, and reduce material waste makes waterjet cutting operations beneficial to increasing efficiencies and effectiveness in AMT shops and fabrication flights. The environmental benefits of waterjet cutting, such as the reduction of hazardous waste and the elimination of airborne dust, contribute to a safer and more sustainable industrial environment. Waterjet cutting is not suitable for all aircraft part production. When authorized, it must be performed in strict accordance with process control specifications due to its impact on mechanical and physical properties, as outlined in the Air Force Life Cycle Management Center (AFLCMC), Flight Systems Engineering, Flight Systems Bulletin: EZ-SB-12-001, Revision B, 21 February 2024. For further details in current usage allowables, contact the Metal Fabrication and Technologies Office : robins.metals.to@us.af.mil.

7.2 SAFETY.

Safety is paramount when operating abrasive waterjets, as improper use can result in severe injuries. Operators must always adhere to safety guidelines set forth by Occupational Safety and Health Administration (OSHA) standards (29 Code of Federal Regulations (CFR) 1910.) and applicable Air Force Instructions (AFIs) and Air Force Manuals (AFMAN) (e.g., DAFMAN 91-203). Supervisors are responsible for maintaining manufacturer manuals and integrating them into on-the-job training and job safety training. In the absence of manufacturer guidance, a hazard analysis and local operating instructions must be developed. Operators must:

- Always wear safety glasses, and may require face shields, hearing protection, and other necessary Personal Protective Equipment (PPE) to protect from flying debris, dust, high-pressure lines, water quality (bacterial growth) and noise.
- Ensure all appropriate guards are in place to prevent accidental contact with rotating components and high-pressure water jets.
- Always follow lockout/tagout procedures to ensure the machine is powered off when performing maintenance.
- Keep the work area clean, organized and free from clutter to avoid slip and/or tripping hazards.
- Be familiar with the location and operation of emergency stop buttons and other safety features to quickly shut down the machine in case of an emergency.
- Remove or secure loose clothing, jewelry, or long hair to prevent entanglement with moving parts.

7.3 TYPES OF WATERJET SYSTEMS.

Waterjet systems are versatile tools used for cutting various materials with high precision. The choice of waterjet system largely depends on the specific cutting requirements, such as the material type, thickness, and the level of precision needed. By understanding the different types of waterjet systems available, technicians can select the most appropriate equipment for each specific application, ensuring efficiency and accuracy in their cutting operations. This knowledge is essential for maximizing the potential of waterjet technology and for ensuring the desired results are achieved.

7.3.1 Pure Waterjet Systems. (See Figure 7-1.) Utilizes only high-pressure water and are suitable for cutting soft materials such as rubber, foam, and textiles. These systems are commonly used on materials that require precision cutting of non-metallic materials without introducing contamination.

7.3.2 Abrasive Waterjet Systems. (See Figure 7-2.) Introduces an abrasive material, such as garnet (Table 7-1), into the water stream, allowing for the precise cutting of hard materials, including metals, composites, and ceramics. The addition of abrasives enables these systems to cut through thick and dense materials with minimal edge taper and high accuracy.

7.3.3 3-Axis Waterjet Systems. (See Figure 7-3.) Provides cutting capabilities along the X, Y, and Z axes, making them ideal for flat sheet material cutting. These systems are widely used for precise two-dimensional cuts with minimal setup time.

7.3.4 5-Axis Waterjet Systems. (See Figure 7-4.) Enables complex cuts, including beveling, contouring, and 3D profiling, which are particularly useful in applications requiring precise geometries and multi-directional cutting capabilities. These systems allow for intricate cutting of components with varied thicknesses and angles, reducing the need for secondary machining.

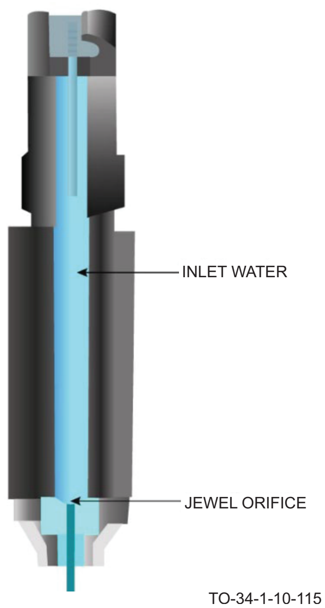


Figure 7-1. Pure Water Head

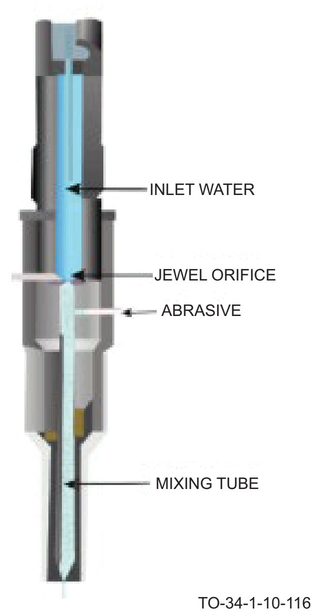


Figure 7-2. Abrasive Water Head

TO-34-1-10-117

Figure 7-3. 3-Axis Waterjet

TO-34-1-10-118

Figure 7-4. 5-Axis Water Jet

Table 7-1. General Garnet Characteristics and Applications

Garnet Type	Mesh Size Range	Grain Characteristics	Cutting Aggressiveness	Cutting Speed	Cut Surface Quality	Typical Use Case
River Garnet	100-150 mesh	Smoother, smaller, rounded grains	Low	Slower	High (smoother cut)	Finishing cuts, precision parts, thin materials
River Garnet	80-100 mesh	Moderately fine, rounded grains	Medium	Moderate	Medium to High	General-purpose cutting with balance of speed/quality
Rock Garnet	60-80 mesh	Coarse, angular, fractured grains	High	Faster	Medium (rougher edges)	Rough cuts, thick material, prep for secondary ops
Rock Garnet	30-60 mesh	Very coarse, highly angular	Very High	Fastest	Low (rough cut)	Rapid material removal, minimal quality needed

7.4 WATERJET SYSTEM COMPONENTS.

7.4.1 Waterjet System. Each component of the waterjet system (Figure 7-5) plays a crucial role in its operation, and proper maintenance ensures consistent performance and longevity. The primary components include the high-pressure pump, which generates the necessary pressure to propel the water or abrasive mixture; the cutting head, which directs the jet onto the material to be cut; and the nozzle, which shapes the waterjet stream to ensure precision. The high-pressure pump, in particular, is responsible for creating the intense pressure required for cutting, while the cutting head and nozzle work together to focus and control the stream of water or abrasive.

7.4.2 Main Components. In addition to the main components, other critical elements of the system include the abrasive delivery system, which transports the abrasive material in abrasive waterjet systems, and the filtration system, which removes impurities from the water to prevent clogs and maintain system integrity. The system's control panel and software are also vital for programming and monitoring the cutting process, ensuring that the correct parameters are followed for each job.

7.4.3 Plumbing, Hoses, and Valves. The plumbing, hoses, and valves direct the flow of water and abrasives throughout the system, ensuring that all components are supplied with the necessary materials to perform effectively. Regular inspection and maintenance of each of these components are essential to prevent wear and tear, maintain system efficiency, and extend the lifespan of the equipment. Proper attention to these elements helps minimize downtime, reduce repair costs, and maintain high-quality cutting performance.

7.4.3.1 High Pressure Pump. Generates high-pressure water, typically between 30,000 and 90,000 Pounds-force per Square Inch. Regular inspections of seals, filters, and pressure levels are required. Pumps can be direct-drive (Figure 7-6) or intensifier-based (Figure 7-7), with intensifier pumps providing higher efficiency in industrial applications.

7.4.3.1.1 Intensifier Pumps. Intensifier pumps are generally preferred for demanding aerospace applications. Their higher efficiency at high pressures, rapid pressure adjustment capabilities, and longer seal life outweigh the higher initial cost. The ability to precisely control pressure and reduce wear on components is crucial for maintaining tight tolerances and achieving high-quality cuts on expensive aerospace materials. However, for less demanding applications or where budget constraints are significant, crankshaft pumps might be a viable alternative.

7.4.3.2 Cutting Head and Nozzle. Direct the high-pressure water and mix it with abrasives for precise material removal. Nozzle wear should be monitored for optimal cut quality. The orifice, typically made of diamond or sapphire, must be checked regularly for wear to maintain consistent pressure and flow.

7.4.3.3 Abrasive Hopper and Feeder. Regulate the flow of abrasives into the waterjet stream. Operators must clear blockages to maintain consistent performance. Proper abrasive metering ensures uniform cutting and reduces wastage of material.

7.4.3.4 Catcher Tank. Absorbs the kinetic energy of the waterjet after it has passed through the material, preventing excessive splash back and collecting spent abrasives. Regular cleaning of the tank prevents clogging and improves system efficiency.

7.4.3.5 Machine Controller. A CNC unit precisely guides the cutting head along the programmed path. Routine calibration is necessary to maintain precision. Modern waterjet systems may incorporate laser or camera-guided systems for enhanced accuracy.

7.4.3.6 Filtration and Recycling System. Manages water usage by filtering out used abrasives and debris, extending pump and nozzle life while promoting sustainability. Proper filtration reduces maintenance costs and ensures a longer operational life for the system.

Table 7-2. Waterjet System General Troubleshooting

Component	Problem	Symptoms	Causes	Troubleshooting
High-Pressure Pump	Cavitation	• Loud Noises (gravel-like) • Pressure fluctuations • Reduced cutting power	• Insufficient water supply • Air leaks in inlet lines • Worn pump components	• Check inlet pressure • Inspect lines for leaks • Inspect/replace worn parts
	Leaking Seals	• Water dripping from pump • Low pressure • Inconsistent cutting	• Worn seals • Improper installation • Excessive back pressure	• Inspect and replace worn seals • Ensure proper installation • Check for blockages downstream

Table 7-2. Waterjet System General Troubleshooting - Continued

Component	Problem	Symptoms	Causes	Troubleshooting
Cutting Head and Nozzle	Nozzle Clogging	• Reduced or uneven cutting stream • Pressure fluctuations • Visible debris in the nozzle	• Poor water quality • Abrasive buildup • Foreign object obstruction	• Clean or replace the nozzle • Improve water filtration • Check abrasive quality
	Premature Orifice Wear	• Increased orifice size • Reduced cutting precision • Faster cutting speed with reduced power	• Incorrect abrasive type • Improper abrasive flow rate • Excessive pressure	• Use recommended abrasive type and size • Adjust abrasive flow rate • Check cutting parameters
Abrasive Delivery System	Inconsistent Abrasive Flow	• Uneven cutting • Inconsistent surface finish • Abrasive buildup in lines	• Moisture-contaminated abrasive • Clogged line. or valves • Worn metering components	• Use dry, high-quality abrasive • Check and clean lines/valves • Inspect and replace worn parts
	Abrasive Hopper Issues	• Insufficient abrasive delivery • Pump starving for abrasive	• Empty hopper • Bridging of abrasive • Malfunctioning feed mechanism	• Refill hopper • Break up any abrasive clumps • Inspect and repair feed system
Plumbing and Fittings	High-Pressure Leaks	• Water spraying • Hissing sounds • Pressure drops	• Loose fitting, • Damaged hoses • Worn seals	• Inspect all fitting, and tighten • Replace damaged hoses • Replace worn seals
	Water Hammer	• Loud banging noises in pipes • Vibration • Potential damage to components	• Sudden valve closures • Trapped air in lines • Improper pipe support	• Install slow-closing valves • Bleed air from lines • Ensure proper pipe support
Machine Controller	Inaccurate Cutting Path	• Parts do not meet dimensional tolerances • Inconsistent cuts	• Lost calibration • Damaged encoder • Software glitches	• Recalibrate the machine • Inspect and replace encoder • Restart software or reload program
	Communi-cation Errors	• Machine unresponsive • Error messages on the controller	• Loose connections • Faulty cables • Software conflicts	• Check and secure all connections • Replace damaged cables • Update or reinstall software

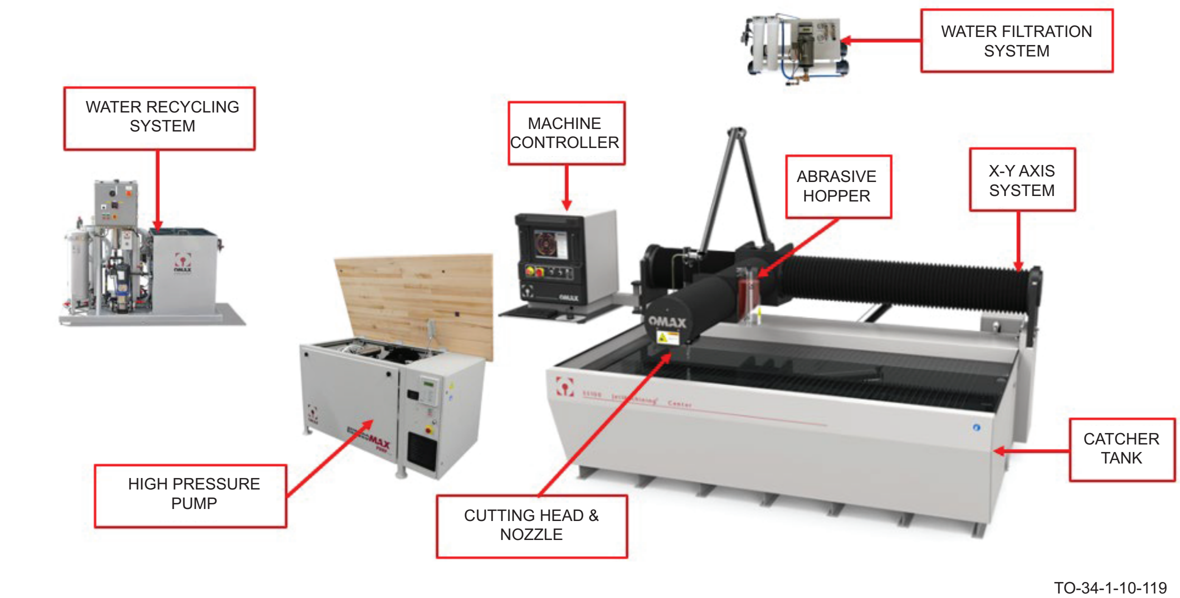


Figure 7-5. Waterjet System Components

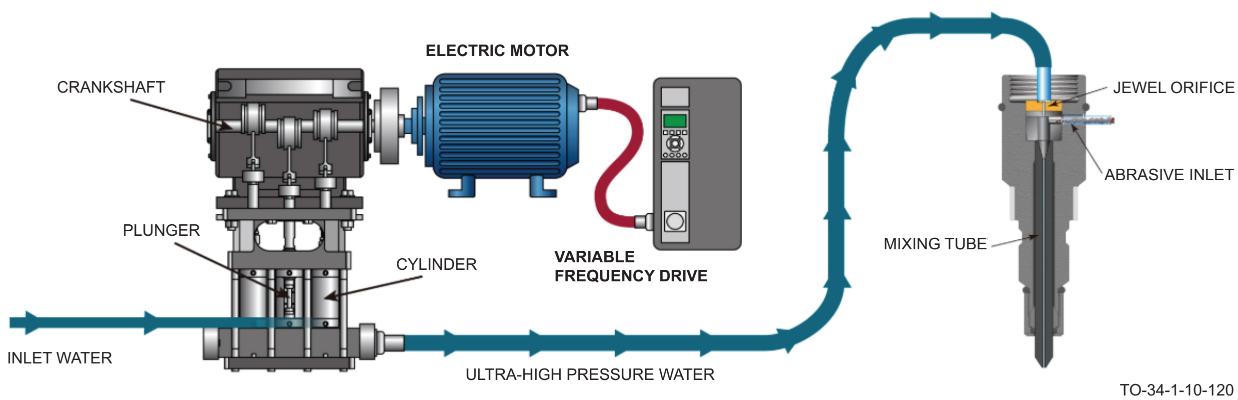
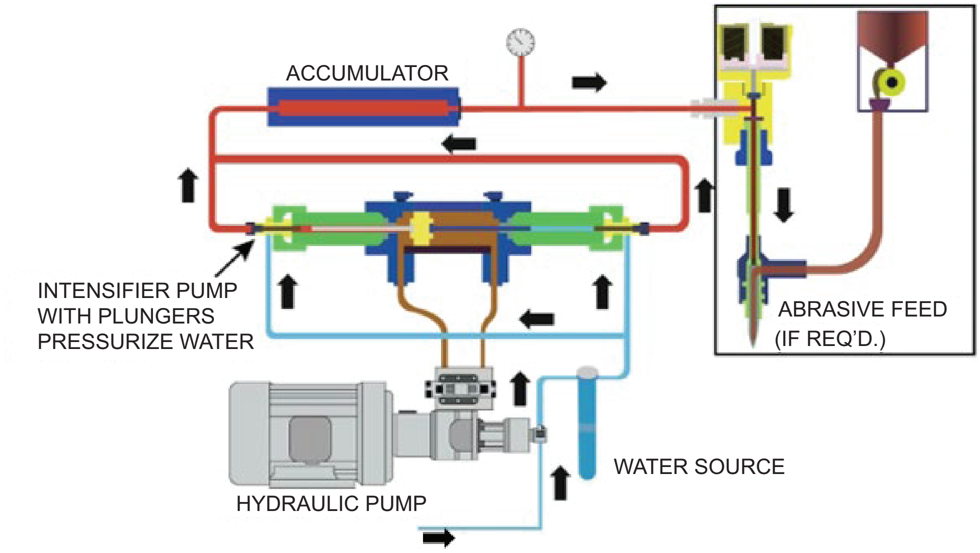


Figure 7-6. Direct Drive Pump



TO-34-1-10-121

Figure 7-7. Intensifier Pump

7.4.4 Cutting Head. The cutting head is one of the most critical components in a waterjet system, as it determines cutting accuracy, efficiency, and overall performance. The orifice and mixing tube work together to create the high-velocity stream that cuts through materials. Use the following general selection criteria to determine which head is best suited for the cutting operation.

7.4.4.1 Material Type. Orifices are commonly made from sapphire, ruby, or diamond. Diamond orifices last significantly longer but are more expensive.

7.4.4.2 Orifice Size. The orifice size impacts the cutting speed and precision. Smaller orifices create finer cuts but require higher pressure.

7.4.4.3 Mixing Tube Size. The diameter of the mixing tube should correspond with the orifice size to ensure optimal flow and mixing.

7.4.5 Orifice and Mixing Tube. The orifice and mixing tube are under extreme conditions due to the continuous high-pressure water stream and abrasive particles passing through them. Operations over time will lead to these components' erosion or wear, affecting the precision and efficiency of the cutting process. Understanding wear patterns and implementing a proactive maintenance schedule can help minimize downtime and ensure consistent performance. Some wear indicators to consider are:

7.4.5.1 Decreased Cutting Efficiency. If the cut takes longer than usual or the material isn't being cleanly removed, the orifice or mixing tube may be worn.

7.4.5.2 Inconsistent or Wider Kerf. A worn mixing tube leads to an uneven or wider cut than expected.

7.4.5.3 Increased Abrasive Consumption. Excessive abrasive use without improved cutting performance can signal wear.

7.4.5.4 Fluctuating Water Pressure. Inconsistent water jet pressure may indicate internal erosion of the orifice or mixing tube.

7.4.5.5 Visible Wear or Chipping. Physical damage or deformities in the orifice or mixing tube are clear signs that replacement is necessary.

7.5 TOOLS AND EQUIPMENT.

To effectively operate and maintain a waterjet system, technicians require specialized tools and equipment that support the functionality, safety, and lifecycle of the system. These tools range from basic hand tools for routine maintenance tasks to more complex diagnostic and alignment tools used to ensure the system operates at peak efficiency. Equipment may include water filtration systems, abrasive delivery systems, and cutting heads that are essential for controlling the water pressure, flow rate, and abrasive material delivery. Technicians must also be familiar with consumables such as nozzles, orifices, and mixing tubes, which wear down over time and need to be replaced regularly. Monitoring and control systems are critical for tracking operational parameters like pressure, temperature, and water flow, which help optimize performance and prevent damage. Proper knowledge of these tools and equipment is vital for ensuring that the waterjet system runs smoothly, reduces downtime, and produces high-quality cuts with minimal maintenance costs.

7.5.1 Material Handling Equipment. Lifting devices, carts, and work holding fixtures support material loading and positioning, reducing strain on operators and improving workflow efficiency.

7.5.2 Abrasive Storage and Dispensing System. Maintains a steady supply of cutting abrasives to prevent machine downtime. Proper storage prevents moisture contamination, which can lead to inconsistent abrasive flow.

7.5.3 Water Filtration Units. Maintain water quality by removing particulates and dissolved minerals that could affect pump and nozzle performance. Routine filter replacements are necessary to sustain peak performance.

7.5.4 Precision Measuring Instruments. Calipers, micrometers, and Coordinate Measuring Machines (CMMs) allow operators to verify cut quality and tolerances, ensuring parts meet required specifications.

7.5.5 Maintenance Toolkit. Includes wrenches, replacement nozzles, high-pressure seals, pressure gauges, and hoses for routine upkeep and repairs. Specialized tools for high-pressure fittings and seals are essential for minimizing downtime.

7.6 GENERAL WATERJET OPERATIONS.

Operating a waterjet system involves several key steps to ensure efficient and precise cutting. The material's thickness, hardness, and type are important factors to consider, as they will influence the cutting parameters such as water pressure, cutting speed, and abrasive flow rate. Once the material type is determined, the cutting parameters must be programmed into the system through the control panel or software interface.

7.6.1 System Components. The technician must ensure that the system components are properly calibrated, and that the waterjet nozzle is aligned correctly. This helps ensure the accuracy and precision of the cut. After initiating the cutting process, the system uses high-pressure water (and abrasives if necessary) to cut through the material according to the programmed specifications. Throughout this process, it is crucial to monitor the system for any potential issues such as pressure drops, nozzle wear, or irregular cutting patterns.

7.6.2 Proper Post-Cutting Procedures. After the cutting is completed, the technician must verify the quality of the cut and check the material for any defects or imperfections. Proper post-cutting procedures, such as cleaning the cutting area and inspecting the system for wear or damage, are essential to maintain the system's performance and ensure readiness for future operations. Regular review of cutting outcomes and machine conditions allows for continuous improvements in the cutting process and helps identify any adjustments needed to enhance efficiency. Follow these general steps to operate a waterjet system:

7.6.2.1 Prepare the Work Area.

a. Clear the work area of any obstructions.

b. Ensure the waterjet system is clean and free of debris.

c. Verify that the machine is properly connected to power and water sources.

7.6.2.2 Inspect the Waterjet System Components.

a. Check the high-pressure pump, cutting head, and nozzle for any damage or wear.

b. Inspect the abrasive delivery system (for abrasive waterjets).

c. Verify that hoses, valves, and other plumbing are secure and free of leaks.

7.6.2.3 Prepare the Material.

a. Select the appropriate material for cutting and ensure it is clean and free of contaminants.

b. Measure and mark the material to align with the cutting path, if required.

c. Secure the material firmly to the cutting bed to prevent movement during the cutting process.

7.6.2.4 Set Cutting Parameters. (See Table 7-3.)

a. Input the correct cutting parameters into the control panel or software such as water pressure, cutting speed, and abrasive flow rate (for abrasive waterjets).

b. Adjust settings based on the material type, thickness, and desired cut quality.

7.6.2.5 Align the Cutting Head and Nozzle.

a. Position the cutting head over the material, ensuring proper alignment with the cutting path.

b. Check the nozzle to ensure it is properly attached and clean.

7.6.2.6 Calibrate the System.

a. Perform a calibration check to ensure the waterjet system is correctly aligned.

b. Confirm that the cutting head and nozzle are positioned properly.

7.6.2.7 Start the Waterjet System.

a. Activate the pump to initiate water flow and pressurize the system.

b. Start the cutting process by initiating the program from the control panel or software.

7.6.2.8 Monitor the Cutting Process.

a. Continuously monitor the system during operation to ensure proper pressure, water flow, and cutting precision.

b. Watch for any signs of wear or issues such as nozzle clogging, pressure drops, or irregular cutting patterns.

7.6.2.9 Complete the Cut.

a. Allow the waterjet to complete the cut according to the programmed specifications.

b. Once the cut is finished, stop the system and power down the machine.

7.6.2.10 Inspect the Cut.

a. Check the material for any defects, burrs, or imperfections in the cut.

b. Verify that the cut quality meets the required standards.

7.6.2.11 Clean the System.

a. Turn off the waterjet system and disconnect it from power.

b. Clean the cutting bed, nozzle, and cutting head to remove any debris or water.

c. Replace any worn components, such as the nozzle or mixing tubes, if necessary.

7.6.2.12 Check for Wear and Perform Maintenance.

a. Inspect the system for any signs of wear or damage to components.

b. Perform any necessary maintenance, such as cleaning the filtration system, replacing worn parts, or checking for leaks.

Table 7-3. Waterjet Cutting Parameters

Material Type	Nozzle Size (Inches)	Abrasive Type	Abrasive Grain Size (Mesh)	Cutting Speed Inches per Minute	Maximum Kilo Pounds-force per Square Inch	Abrasive Flow Rate (Pounds per Minute)	Stand-off Distance (Inches)	Cut Quality	Comments
Aluminum	0.03	Garnet	80-120	3-6	55	0.5-1	0.1-0.2	Smooth	Ideal for thin sheets, fast cutting speeds
Stainless Steel	0.04	Garnet	80-120	1-3	60	1-2	0.2-0.3	Smooth to Fine	Requires higher pressure for thicker materials
Mild Steel	0.04	Garnet	80-120	4-7	55	1-2	0.15-0.25	Smooth	Works well for general cutting
Titanium	0.03	Garnet	80-120	2-5	60	0.7-1.5	0.2-0.3	Fine	Requires precise control for cutting thicker titanium
Glass	0.02	Abrasive Free	N/A	1-2	40	N/A	0.05-0.1	Smooth	Requires special techniques for fragile material
Ceramics	0.02	Abrasive Free	N/A	0.5-2	40	N/A	0.05-0.1	Fine	Can be brittle, must avoid excessive pressure
Plastic (Acrylic)	0.03	Garnet	60-100	4-6	45	0.5-1	0.1-0.2	Smooth	Cuts easily with lower pressure
Stone (Granite)	0.04	Garnet	120-200	0.5-1	60	2-3	0.25-0.5	Rough to Smooth	Slow cutting due to hardness of material
Copper	0.04	Garnet	80-120	2-4	55	1-2	0.15-0.25	Smooth	Requires higher pressure to cut efficiently
Composites	0.03	Garnet	120-180	3-5	55	0.5-1	0.1-0.2	Smooth	Abrasive type may vary depending on composite type

CHAPTER 8
PUNCHING AND SHEARING MACHINES

8.1 GENERAL.

Punching and shearing machines are essential tools in metal fabrication shops, offering precision and efficiency in cutting, notching, and hole punching operations occurring in Aircraft Metals Technology (AMT) and Aircraft Structural Maintenance (ASM). These machines are widely used in aircraft machining, welding, and structural repair shops due to their ability to handle a variety of metal types and thicknesses. Designed for heavy-duty applications, they streamline the fabrication process, reducing manual labor while improving accuracy and repeatability. These machines are used for cutting structural steel, aluminum alloys, or stainless-steel components. They provide high-speed operation with minimal material waste. Understanding the proper operation, safety considerations, and maintenance requirements of these machines is essential for ensuring workplace safety, machine longevity, and consistent performance in demanding environments.

8.2 SAFETY.

Safety is paramount when operating lathes, as improper use can result in severe injuries. Operators must always adhere to safety guidelines set forth by Occupational Safety and Health Administration (OSHA) standards (29 Code of Federal Regulations (CFR) 1910) and applicable Air Force Instructions (AFIs) and Air Force Manuals (AFMAN) (e.g., DAFMAN 91-203). Supervisors are responsible for maintaining manufacturer manuals and integrating them into on-the-job training and job safety training. In the absence of manufacturer guidance, a hazard analysis and local operating instructions must be developed. Operators must:

- Consider OSHA standards, relevant Air Force guidance and/or Original Equipment Manufacturer (OEM) recommendations for the specific machine and operation.
- Consult the specific manufacturer’s manual for the punching and shearing machines being used. Punches and shears vary significantly, and relying solely on general guidance can be dangerous.
- Wear safety glasses, face shields, hearing protection, and other necessary Personal Protective Equipment (PPE) to protect from flying debris, dust, and noise.
- Ensure all appropriate guards are in place to prevent accidental contact with hazardous components.
- Follow lockout/tagout procedures to ensure the machine is powered off when performing maintenance.
- Keep the work area clean and free from clutter to avoid tripping hazards.
- Be familiar with the location and operation of emergency stop buttons and other safety features to quickly shut down the machine in case of an emergency.
- Remove loose clothing and jewelry.
- Long hair must be secured to prevent entanglement with moving parts.

8.3 TYPES OF PUNCHING AND SHEARING MACHINES.

Punching and shearing machines come in several configurations, each designed for specific applications and material processing needs. Some machines operate through hydraulic or electric power, while others are operated mechanically to execute jobs such as cutting and punching. Versatile options like ironworker machines integrate multiple fabrication functions, allowing for increased efficiency in metal processing applications. Lighter-duty manual models are also available for small workshops and maintenance tasks where automation is not required. In fabrication flights, various types of punching and shearing machines are used to perform specific tasks depending on the material, thickness, and the complexity of the cut. These machines offer different advantages and capabilities, making them suitable for a wide range of applications, from simple shearing operations to complex hole punching and notching. Selecting the right type of universal punching and shearing machine depends on factors such as material thickness, production volume, precision requirements, and available shop space. Proper machine selection ensures optimal performance and longevity while meeting specific operational needs. Additionally, considering factors like time to set-up, maintenance requirements, and adaptability to future production demands will further enhance efficiency and cost-effectiveness in fabrication settings. The common types of punching and shearing machines in fabrication flights include (Figure 8-1):

8.3.1 Ironworkers. Versatile machines that combine punching, shearing, and notching. They are ideal for heavy-duty tasks, offering power and efficiency for cutting thick materials like steel and titanium in Air Force (AF) fabrication shops.

8.3.2 Rotary Turret Punches. High-speed hole-punching machines designed for repetitive tasks. They provide precision and efficiency when creating multiple holes in sheet metal, commonly used in aircraft panel fabrication and structural components.

8.3.3 Hydraulic Shears. Powered by hydraulics, these shears are capable of cutting through thick and high-strength materials, making them ideal for trimming and straight cuts on heavy metals, such as steel and aluminum.

8.3.4 Foot Shears. Manual machines operated by foot pedals, perfect for light-duty, small-scale shearing tasks like trimming thin sheet metal or making quick cuts for minor repairs.

8.3.5 Scroll Shears. Specialized for cutting curved or irregular shapes in metal, these shears are used for custom-shaped parts or components with complex, non-linear cuts required in aerospace applications.

8.3.6 Throatless Shears. Highly versatile shears with an open throat design, allowing for cutting a wide range of materials and irregularly shaped pieces, ideal for precise and intricate cuts in medium-sized materials.

Figure 8-1. Types of Punches and Shears

8.4 USES AND APPLICATIONS OF PUNCHING AND SHEARING MACHINES.

Punching and shearing machines are essential tools in aircraft repair and fabrication shops, where precision, efficiency, and repeatability are critical to maintaining structural integrity and airworthiness. These machines allow technicians to quickly and accurately cut, punch, and shape metal components used in aircraft and support equipment repairs, eliminating the need for slower, more labor-intensive manual methods. Their ability to handle a range of aerospace materials - including aluminum, stainless steel, and titanium - makes them indispensable for fabricating airframe and support equipment components, brackets, doublers, and custom repair fittings.

8.4.1 Punching Machines. Punching machines, including ironworkers, rotary turret punches (e.g., Rotex), and hydraulic punch presses, are used to create precision holes and cutouts in sheet metal, plate, and structural components. These machines are particularly beneficial in aircraft repair because they produce clean, burr-free holes without distortion, preserving the strength and integrity of the material.

8.4.1.1 Airframe Structural Repairs. Punching is used to create precise rivet and fastener holes in replacement skins, stringers, and formers. Unlike drilling, punching ensures a consistent hole diameter without work-hardening the material, which is critical for fatigue resistance in high-stress areas.

8.4.1.2 Bracket and Doubler Fabrication. Brackets and doublers, commonly used to reinforce weakened or damaged airframe sections, require precise hole placement for proper load distribution. Punching machines allow for rapid hole placement that aligns perfectly with existing aircraft structures.

8.4.1.3 Bulkhead and Rib Modifications. When modifying or fabricating bulkheads, ribs, and other internal structural components, rotary punches are used to create lightening holes, access holes, or conduit openings without compromising material integrity.

8.4.1.4 Electrical and Hydraulic System Cutouts. Rotary and hydraulic punches are used to create access holes in sheet metal enclosures, avionics panels, and hydraulic control boxes, ensuring precise fits for wiring, conduit, and connectors without the risk of deforming thin-gauge materials.

8.4.1.5 Steel and Aluminum Structural Fabrication. Used to create bolt holes and attachment points in beams, channels, and plate steel for building frameworks, supports, and equipment mounts.

8.4.1.6 Welding Fixture Preparation. Punching is used to create precise holes for alignment pins, bolts, and fasteners in weld fixtures, ensuring accurate part positioning.

8.4.1.7 Support Equipment Repair. Hydraulic punch presses are used to create mounting holes in thick steel plates for brackets, reinforcement members, and machine components.

8.4.2 Shearing Machines. Shearing machines, including hydraulic shears, foot shears, scroll shears, and throatless shears, are used to cut sheet metal and plate with high accuracy, producing straight, clean edges without excessive burr formation. In aircraft repair, shearing is often preferred over abrasive cutting methods because it minimizes heat-affected zones, preserving the mechanical properties of aerospace alloys.

8.4.2.1 Aircraft Skin Repairs. Hydraulic shears are used to cut replacement aluminum skin panels to exact dimensions before forming and riveting. A precise, clean cut is critical to ensure proper fitment and minimize gaps that could compromise aerodynamics and structural integrity.

8.4.2.2 Fabrication of Structural Components. Shearing is used to size raw stock for the production of longerons, spars, and stiffeners, ensuring straight edges that aid in proper alignment and load distribution in aircraft structures.

8.4.2.3 Removal of Damaged Sections. When removing corroded or damaged sections of an aircraft's frame or skin, shears allow technicians to make precise, controlled cuts that preserve the surrounding material and reduce the need for additional finishing.

8.4.2.4 Customizing Sheet Metal for Forming Operations. Before being shaped into leading-edge fairings, cowling panels, or fuselage skins, sheet metal must be cut to size with minimal waste. Throatless shears allow for intricate, curved cuts that help maximize material usage while maintaining smooth edges.

8.4.2.5 Sheet Metal Preparation for Welding. Ensures straight, burr-free cuts in steel, aluminum, and stainless-steel sheets before welding, reducing post-weld grinding and cleanup.

8.4.2.6 Cutting Steel Plate for Structural Fabrication. Hydraulic shears efficiently cut thick steel and aluminum plates used in construction, heavy equipment, and industrial machinery.

8.4.2.7 Fabrication of Panels and Enclosures. Used in the production of control panels, machine covers, and electrical enclosures where clean, precise edges are required.

8.4.3 Fundamental. Punching and shearing machines are fundamental to the efficiency and accuracy of aircraft maintenance, ensuring that repairs and new fabrications meet the stringent requirements of aviation standards. These machines:

8.4.3.1 Enhance Precision. Aircraft structures rely on tight tolerances for hole placement and edge quality. Punching and shearing provide repeatable, high-accuracy cuts essential for maintaining structural integrity.

8.4.3.2 Reduce Material Waste. Compared to manual cutting and drilling, these machines optimize material usage, reducing scrap and ensuring cost-effective repairs.

8.4.3.3 Increase Productivity. Faster than manual methods, punching and shearing allow technicians to complete repairs more efficiently, minimizing aircraft downtime.

8.4.3.4 Prevent Material Damage. Unlike grinding or torch cutting, which can introduce heat stress and distortion, these cold-cutting methods preserve the mechanical properties of aerospace alloys, ensuring the longevity and reliability of repaired components.

8.5 TOOLS AND EQUIPMENT.

Proper tools and equipment are essential for effective punching and shearing operations. These components ensure precision, efficiency, and safety while extending the lifespan of machines and maintaining high-quality output. The following sections provide an overview of the essential tools and equipment used in punching and shearing operations.

8.5.1 Punching Operations. Punching machines require a variety of specialized tools to ensure accurate hole placement, clean cuts, and minimal material distortion. The selection of punch and die sets by material thickness and tonnage required (Table 8-1), along with proper alignment and lubrication, plays a crucial role in maintaining efficiency and prolonging tool life. Below are the key tools and equipment used in punching operations:

8.5.1.1 Punch and Die Sets. The primary tooling components used in punching operations. These sets come in various shapes and sizes, including round, square, slotted, and custom profiles. Proper punch-to-die clearance is critical to achieving clean cuts and minimizing burr formation.

8.5.1.2 Turret Assemblies (for Rotary Turret Punches). Rotating mechanisms that house multiple punch and die sets, allowing for quick tool selection without manual changes.

8.5.1.3 Strippers. Hold the material in place and prevent distortion when the punch retracts. They also help extend punch life by reducing lateral forces.

8.5.1.4 Backgauges and Stops. Used to position and align the material for consistent hole placement. These can be manually adjustable or pre-set for repetitive work.

8.5.1.5 Punch Lubricants. Reduce friction and heat buildup, extending the life of the punch and die while improving cut quality.

8.5.1.6 Workpiece Clamps. Hold sheet metal or plate securely during punching to prevent movement and misalignment.

8.5.1.7 Deburring Tools. Used after punching to remove sharp edges or burrs that may result from the cutting process.

Table 8-1. Tonnage Required for Punching Holes in Mild Steel

Material Thickness	Punch Size													
	1/8	3/16	1/4	5/16	3/8	7/16	1/2	9/16	5/8	11/16	3/4	13/16	7/8	15/16	1
3/32	1	1	2	2	3	3	4	4	5	5	5	6	7	7	8
1/8	1	2	3	3	4	4	5	6	6	7	8	8	9	9	10
3/16	3	4	5	6	7	8	9	9	10	11	12	13	14	15	16
1/4	5	6	8	9	10	11	13	14	15	16	18	19	20	22	23
5/16	8	9	11	13	14	16	17	19	20	22	23	25	28	30	33
3/8		11	13	15	17	19	21	23	24	26	28	30	33	35	38
7/16			15	18	20	22	24	26	28	30	33	35	38	40	42
1/2			20	23	25	28	30	33	35	38	40	42	45	47	50
9/16				26	28	30	34	36	40	42	45	47	50	53	56
5/8				31	34	38	41	44	47	50	53	56	60	63	65
11/16					38	41	44	48	51	55	57	61	65	70	75
3/4					45	49	53	56	60	65	70	75	80		
13/16						53	57	61	65						
7/8						61	66	70							
15/16						71	75								
1							80								

8.5.2 Shearing Operations. Shearing operations require precise cutting tools and support equipment to ensure clean, straight cuts with minimal deformation. The quality of the cut is highly dependent on factors such as blade sharpness, material hold-down mechanisms, and proper clearance adjustments. Below are the key tools and equipment used in shearing operations:

8.5.2.1 Shear Blades. The cutting edges of the shearing machine. Available in various materials, such as high-carbon steel and carbide, depending on the material being cut. Regular sharpening is required to maintain a clean cut.

8.5.2.2 Blade Clearance Adjusters. Devices that set the gap between the upper and lower shear blades. Proper clearance depends on the thickness and type of material being cut.

8.5.2.3 Hold-Down Clamps. Mechanisms that secure the workpiece during shearing to prevent shifting and ensure a clean, accurate cut.

8.5.2.4 Backgauges and Front Gauges. Adjustable positioning systems that help ensure accurate and repeatable cuts. These are commonly found on hydraulic shears and foot shears.

8.5.2.5 Material Supports and Roller Tables. Used to support large sheets and plates before and after cutting, reducing operator strain and improving cut accuracy.

8.5.2.6 Squaring Arms and Angle Gauges. Assist in aligning the material at precise angles for accurate shearing.

8.5.2.7 Lubricants and Coolants. Applied to the blades and workpiece to reduce heat buildup, minimize wear, and improve cutting performance.

8.5.2.8 Safety Guards and Barriers. Essential for protecting operators from moving parts and preventing accidental contact with blades.

8.5.3 Optimal Performance. To maintain optimal performance, both punching and shearing machines require regular maintenance and inspection. Specialized tools are used to sharpen, align, and monitor critical components, ensuring consistent operation and preventing premature tool wear. Below are the essential maintenance and inspection tools.

Blade Sharpening Equipment

Used for reconditioning shear blades to maintain a precise cutting edge.

Punch and Die Grinding Tools

Ensure smooth, sharp edges on punches and dies for clean hole punching.

Dial Indicators and Micrometers

Used to check blade clearance, punch alignment, and material thickness for precision work.

Hydraulic Fluid and Pressure Gauges

Monitor hydraulic systems in ironworkers and hydraulic shears to ensure consistent force application.

8.6 GENERAL PUNCHING OPERATIONS.

Punching is a fundamental metalworking process used to create holes in sheet metal, plate, and other materials by forcing a punch through the workpiece into a die. This process is widely used in fabrication, manufacturing, and repair applications due to its efficiency, repeatability, and ability to produce precise hole patterns. Punching is typically performed using machines such as ironworkers and hand-operated rotary turret punches, each designed to handle specific material types and thicknesses.

8.6.1 Successful Punching Operations. Successful punching operations require careful setup, proper tool selection, and precise execution to ensure clean, accurately placed holes while minimizing burr formation, material distortion, and tool wear. Selecting the correct punch and die combination, setting appropriate clearances, and aligning the workpiece correctly are critical factors in achieving high-quality results. Additionally, routine maintenance of punching tools and adherence to safety protocols are essential for consistent operation and prolonged equipment life. The following step-by-step instructions provide a detailed guide for performing general punching operations using ironworkers and hand-operated rotary turret punches:

NOTE

Before beginning a punching operation, ensure that all necessary tools, equipment, and safety measures are in place.

8.6.1.1 Pre-Operation Preparation.

a. Check for any visible damage, worn components, or hydraulic leaks (if applicable).

b. Ensure the machine is clean and properly lubricated.

c. Confirm the material type, thickness, hole size, and layout.

d. Select the appropriate punch and die set for the required hole size and shape.

e. Ensure that backgauges, material clamps, and safety equipment are available.

8.6.1.2 Machine Setup and Tool Selection.

a. Before installing tools, ensure that the machine is powered off and locked out, if required.

b. Choose a punch and die set with the appropriate diameter and shape.

c. Ensure that the die clearance matches the material thickness.

d. Securely mount the punch in the ram and the die in the lower holder.

e. Tighten fasteners and verify alignment.

f. Lower the ram slightly to confirm that the punch and die are properly aligned without interference.

g. Set backgauges or material stops to ensure consistent hole placement for multiple workpieces.

h. Cycle the machine without material to verify smooth operation and correct punch movement.

8.6.1.3 Positioning the Workpiece.

a. Position the sheet metal or plate on the machine's work surface, ensuring it rests flat.

b. Use pre-marked hole locations or align the material against backgauges or guides.

c. Engage material clamps or hold-down devices (if available) to prevent shifting during punching.

8.6.1.4 Operate the Ironworker or Punch Press.

a. Check that the punch is centered over the desired hole location.

b. Activate the hydraulic or mechanical system to drive the punch through the material.

c. Let the punch travel completely through the workpiece and retract fully.

d. Check for proper hole size, burrs, and distortion. If necessary, adjust punch clearance or alignment.

8.6.1.5 Operate the Hand Turret Punch.

a. Ensure the proper punch and die set is aligned for the operation.

b. Align the material using the machine's guide marks or stops.

c. Pull the lever or engage the foot pedal to drive the punch through the workpiece.

d. Fully retract the punch before repositioning the material for the next hole.

8.6.1.6 Post-Punching Inspection and Cleanup.

a. Ensure that holes are properly sized, positioned correctly, and free of excessive burrs or deformation.

b. Use a deburring tool or file to remove any rough edges.

c. Measure hole diameter and center-to-center spacing with calipers or a gauge.

d. Remove metal shavings and scrap material from the machine and worktable.

e. Wipe down the machine, inspect tooling for wear, and apply lubrication as needed.

8.7 GENERAL SHEARING OPERATIONS.

Successful shearing operations require careful setup, proper blade selection, and precise execution to ensure clean, accurately cut edges while minimizing burr formation, material distortion, and blade wear. Selecting the correct blade gap, securing the workpiece properly, and applying the correct cutting force are critical factors in achieving high-quality results. Additionally, routine maintenance of shearing tools and adherence to safety protocols are essential for consistent operation and prolonged equipment life. The following step-by-step instructions provide a detailed guide for performing general shearing operations using hydraulic shears, foot shears, scroll shears, and throatless shears.

NOTE

Before beginning a shearing operation, ensure that all necessary tools, equipment, and safety measures are in place.

8.7.1 Pre-Operation Preparation.

a. Check for any visible damage, worn components, or hydraulic leaks (if applicable).

b. Ensure the machine is clean and properly lubricated.

c. Confirm the material type, thickness, and cut layout.

d. Select the appropriate blade type and ensure it is sharp and properly installed.

e. Ensure that backgauges, material clamps, and safety equipment are available.

8.7.2 Machine Setup and Blade Adjustment.

a. Before adjusting or installing blades, ensure that the machine is powered off and locked out, if required.

b. Verify the blade gap is appropriate for the material thickness.

c. Securely mount the upper and lower blades, ensuring proper alignment and clearance.

d. Tighten all fasteners and confirm that the blade movement is smooth.

e. Set backgauges or material stops to ensure consistent cut dimensions for multiple workpieces.

f. Cycle the machine without material to verify smooth operation and correct blade movement.

8.7.3 Positioning the Workpiece.

a. Position the sheet metal or plate on the machine's work surface, ensuring it rests flat.

b. Align the cut line with the shear blade using guide marks or bachgauges.

c. Engage material clamps or hold-down devices (if available) to prevent shifting during cutting.

8.7.4 Operate the Shearing Machine.

a. Check that the blade is properly aligned with the cut line.

b. Activate the hydraulic or mechanical system to drive the blade through the material.

c. Allow the blade to travel completely through the workpiece and retract fully.

d. Inspect the cut edge for smoothness, burrs, and distortion. Adjust blade clearance if necessary.

8.7.5 Operate the Foot Shear, Scroll Shear, or Throatless Shear.

a. Ensure the proper blade is installed and aligned for the operation.

b. Align the material using guide marks or stops.

c. Apply steady pressure on the foot pedal or hand lever to drive the blade through the workpiece.

d. Allow the blade to fully retract before repositioning the material for the next cut.

8.7.6 Post-Shearing Inspection and Cleanup.

a. Ensure that cut edges are smooth, properly sized, and free of excessive burrs or deformation.

b. Use a deburring tool or file to remove any rough edges.

c. Measure cut dimensions with calipers or a gauge to verify accuracy.

d. Remove metal shavings and scrap material from the machine and worktable.

e. Wipe down the machine, inspect tooling for wear, and apply lubrication as needed.

CHAPTER 9
3D SCANNERS AND COORDINATE MEASURING MACHINES (CMM)

9.1 GENERAL.

Precision measurement is a critical aspect of manufacturing and repair operations in fabrication flights, ensuring that components meet exact specifications and tolerances. Two of the most advanced tools used for dimensional inspection and quality assurance are 3D scanners and CMMs. These technologies provide highly accurate measurements of complex geometries, allowing technicians to verify part dimensions, analyze deviations, and ensure compliance with design specifications. 3D scanners utilize optical, laser, or structured light technology to capture detailed surface data from an object, creating a high-resolution digital representation. These systems are particularly useful for reverse engineering, rapid prototyping, and quality assurance applications where complex shapes and freeform surfaces must be measured quickly and accurately. Unlike traditional measuring tools, 3D scanners provide noncontact measurement, reducing the risk of surface damage and allowing for the inspection of delicate or flexible materials. CMMs employ a probe-based system to capture precise measurements of a workpiece by physically contacting its surface at specific points. Available in manual and Computer Numerical Control (CNC)- controlled configurations, CMMs are widely used in aerospace, automotive, and precision machining industries to verify critical dimensions and geometric tolerances. Their ability to perform highly accurate inspections with repeatable results makes them indispensable for quality assurance and metrology applications.

9.2 CARE AND USE OF 3D SCANNERS AND CMMS.

While 3D scanners and CMMs are generally safe to operate, they require careful handling and proper maintenance to ensure accuracy and longevity. Operators should keep the work area clean and free of obstructions to prevent tripping hazards and accidental equipment damage. For laser-based 3D scanners, users should follow manufacturer guidelines on eye protection to avoid potential hazards. CNC-controlled CMMs have automated probe movements, posing a pinch or collision risk, so operators must stay clear of moving parts. Proper fixturing of the workpiece is essential to prevent shifting, which can lead to inaccurate measurements or probe damage. Handheld scanners require a steady hand to avoid errors, while reflective or transparent surfaces may require special settings or treatments for accurate scanning.

9.2.1 Care and Use Guidelines.

Handle with Care

Avoid dropping or mishandling scanners and probes to prevent misalignment or damage.

Regular Calibration

Perform scheduled calibration to maintain measurement accuracy.

Keep Lenses and Probes Clean

Dust, dirt, or smudges can affect accuracy; clean surfaces with appropriate materials.

Store Properly

When not in use, keep scanners and CMM probes in protective cases or designated areas.

Secure Workpieces

Ensure parts are properly fixtured to prevent movement during measurement.

Monitor Environmental Conditions

Extreme temperatures, vibrations, and humidity can affect precision; use equipment in a controlled environment when possible.

9.3 TYPES OF 3D SCANNERS AND CMMS.

3D scanners and CMMs come in various types, each suited for specific applications in precision measurement and quality control. 3D scanners use different technologies to capture surface data, offering non-contact measurement solutions ideal for complex geometries and delicate materials. CMMs, in contrast, use probes to physically contact a workpiece, providing high-accuracy dimensional measurements. Choosing the right system depends on factors such as accuracy requirements, part complexity, and inspection speed.

9.3.1 Types of 3D Scanners.

9.3.1.1 Laser Scanners. Use laser beams to capture surface data by detecting reflected light. They are ideal for scanning reflective or dark surfaces and are commonly used in reverse engineering and quality inspection.

9.3.1.2 Structured Light Scanners. Structured Light Scanners: Project a pattern of light onto an object, measuring distortions to create a 3D model. These scanners provide high-resolution results and are often used in metrology and medical applications.

9.3.1.3 Photogrammetry-Based Scanners. Utilize high-resolution images from multiple angles to reconstruct a 3D model. They are suitable for large-scale measurements, such as aerospace components.

9.3.1.4 Scanning Arms. (See Figure 9-1.) A portable CMM with an articulated arm and integrated laser scanner, offering high-precision contact and non-contact measurement. Ideal for large-part inspection, reverse engineering, and in-field metrology, it provides flexibility with wireless operation and real-time data capture.

9.3.1.5 Handheld Scanners. (See Figure 9-2.) Portable devices that allow flexible scanning in various environments. They are widely used for field measurements, reverse engineering, and large object scanning.

TO-34-1-10-123

Figure 9-1. Scanning Arm

TO-34-1-10-124

Figure 9-2. Handheld 3D Scanner

9.3.2 Types of CMMs.

9.3.2.1 Bridge CMMs. The most common type, featuring a fixed gantry with a moving probe. They offer high precision and are used in machining, aerospace, and automotive industries.

9.3.2.2 Gantry CMMs. Similar to bridge CMMs but designed for measuring large, heavy components such as aircraft structures and industrial machinery.

9.3.2.3 Cantilever CMMs. Feature a single-arm structure, providing easy access to small and medium-sized parts. They are commonly used in production lines for rapid measurements.

9.3.2.4 Portable CMMs (Articulating Arm CMMs). Feature a flexible, jointed arm that allows manual probing of various angles. These are useful for inspecting large or immovable parts.

9.3.2.5 Optical and Laser CMMs. Use non-contact measurement methods, such as laser or structured light, for high-speed inspection of complex or delicate surfaces.

9.4 TOOLS, EQUIPMENT, AND SOFTWARE.

9.4.1 3D Scanners and CMMs. Both 3D scanners and CMMs require an array of specialized tools to facilitate accurate measurement, setup, and maintenance. These tools aid in fixture placement, calibration, and part alignment, ensuring reliable data collection. There are various tools, supporting equipment used in conjunction with 3D scanners and CMMs, as well as the essential software required for data processing, analysis, and integration into digital manufacturing workflows. By utilizing the appropriate tools, ancillary equipment, and software, manufacturers can maximize the accuracy, efficiency, and reliability of 3D scanning and CMM measurement processes. The following sections provide information into individual components, their applications, and best practices. Common tools and equipment items are as follows:

9.4.1.1 Calibration and Verification Tools. These include certified calibration spheres, gauge blocks, and precision artifacts that are essential for maintaining measurement accuracy. Calibration tools must conform to specific industry standards.

9.4.1.2 Target Markers and Reference Dots. Commonly used in optical 3D scanning, these adhesive or magnetic markers provide reference points for aligning multiple scans and improving accuracy in large-volume scanning applications.

9.4.1.3 Probe and Stylus Kits. CMM machines utilize various probe types, including ruby-tipped styli, star configurations, and disc probes, to measure internal and external geometries accurately.

9.4.1.4 Fixture Clamping Systems. Modular fixturing kits with adjustable clamps, magnetic bases, and tension rods help secure workpieces in repeatable positions during scanning or probing operations.

9.4.1.5 Surface Treatment Tools. For optical 3D scanners, a mattifying spray or developer powder is used to reduce reflectivity and enhance scan accuracy when measuring glossy or transparent surfaces.

9.4.1.6 Alignment and Positioning Tools. Optical scales, laser trackers, and precision levels help ensure proper setup of both CMMs and 3D scanners, reducing errors caused by misalignment.

9.4.2 Optimize Measurement Efficiency and Accuracy. To optimize measurement efficiency and accuracy, ancillary equipment is integrated into the scanning or probing environment. These supporting systems enhance stability, environmental control, and automation.

9.4.2.1 Granite Surface Plates. Provide a stable and flat reference surface for both contact and non-contact measurements, essential for ensuring dimensional accuracy.

9.4.2.2 Vibration Isolation Tables. Minimize the impact of external vibrations that could affect CMM measurements, particularly in high-precision environments.

9.4.2.3 Temperature and Humidity Control Systems. Environmental stability is critical for maintaining measurement accuracy, especially for highly sensitive optical and laser-based scanners.

9.4.2.4 Automated Turntables and Motion Stages. Used with 3D scanners to automate part rotation, enabling seamless multi-angle scans and reducing manual repositioning errors.

9.4.2.5 Portable Articulating Arms. Extend the measurement capabilities of CMMs by enabling flexible, on-site inspection of large components, reducing the need for disassembly.

9.4.2.6 Robotic Integration for Automation. In high-volume manufacturing, CMMs and 3D scanners can be integrated with robotic arms to automate repetitive inspection tasks and improve throughout.

9.4.3 Software. Software plays a critical role in processing raw measurement data, aligning scans, performing analysis, and generating inspection reports. Modern solutions integrate with Computer-Aided Design and Computer-Aided Manufacturing systems and support various industry standards.

9.4.3.1 Point Cloud Processing Software. Converts raw scan data into usable 3D models, including noise reduction, mesh generation, and surface reconstruction. Examples include PolyWorks, Geomagic, and GOM Inspect.

9.4.3.2 Metrology and Inspection Software. Used for dimensional analysis, tolerance verification, and Geometric Dimensioning and Tolerancing (GD&T) comparisons. Examples include PC-DMIS, Calypso, and Verisurf.

9.4.3.3 Reverse Engineering Software. Enables conversion of scan data into parametric or solid models compatible with computer-aided design (CAD) software. Common tools include Siemens NX, SolidWorks Scan to 3D, and Autodesk ReCap.

9.4.3.4 Automated Measurement Software. Integrated with robotic systems to enable real-time feedback, adaptive inspection, and automated quality control processes. Examples include Metrology X4 and Creaform VXinspect.

9.4.3.5 Simulation and Digital Twin Software. Advanced metrology solutions utilize digital twins for predictive maintenance, process optimization, and virtual prototyping.

9.5 OPERATIONAL FACTORS.

9.5.1 Accuracy of 3D Scanning and CMM Measurements. The accuracy of 3D scanning and CMM measurements is highly dependent on the surrounding environment and operational conditions. Variations in temperature, humidity, vibration, and lighting can introduce measurement errors, particularly for optical and laser-based scanning systems.

9.5.1.1 Temperature Control. Since materials expand and contract with temperature fluctuations, maintaining a controlled environment within ±1 degree Celsius (°C) is critical for precision metrology. Many high-end CMMs are housed in climate-controlled rooms to prevent thermal distortion.

9.5.1.2 Vibration Mitigation. External vibrations from nearby machinery, foot traffic, or structural movement can disrupt both scanning and probing accuracy. Installing vibration isolation tables or mounting CMMs on dedicated foundations minimizes these effects.

9.5.1.3 Lighting Conditions. Optical 3D scanners rely on structured light or laser projection, making ambient lighting an important factor. Reducing direct sunlight, using diffused Light-Emitting Diode lighting, or working in enclosed booths enhances scan reliability.

9.5.1.4 Air Quality. Dust and airborne contaminants can affect both laser and optical scanning performance, particularly in high-precision applications. High-Efficiency Particulate Air-filtered enclosures may be necessary for sensitive measurement tasks.

9.5.2 Materials and Surface Finishes. Different materials and surface finishes present unique challenges in both 3D scanning and CMM probing. Understanding these factors ensures proper selection of measurement techniques and preprocessing treatments.

9.5.2.1 Reflective and Transparent Surfaces. Optical and laser-based scanners struggle with high-gloss, chrome-plated, polished, or transparent surfaces. Applying an anti-reflective spray or using structured light with polarization filters can improve scan quality.

9.5.2.2 Soft and Deformable Materials. Flexible materials, such as rubber or thin sheet metal, can distort under contact probes. Non-contact 3D scanning methods, like structured light or blue laser scanning, are preferred for these applications.

9.5.2.2.1 Complex Geometries and Deep Cavities. While traditional CMM probes excel in measuring deep features, line-of-sight limitations affect optical scanners. Using specialized probe extensions or hybrid scanning-CMM systems allows for more comprehensive measurement of intricate parts.

9.5.2.2.2 Material Conductivity and Probing Sensitivity. Some touch-trigger probes rely on electrical conductivity for detection. When measuring non-conductive materials such as composites or ceramics, alternative probing techniques like optical or ultrasonic methods may be necessary.

9.5.2.3 When choosing between a handheld 3D scanner and a CMM, several factors must be considered (Table 9-1), including accuracy, speed, flexibility, and application requirements. Handheld 3D scanners offer portability and fast data acquisition, making them ideal for scanning large or complex objects, reverse engineering, and applications where mobility is essential. However, they generally provide lower accuracy compared to CMMs and can be affected by environmental factors such as lighting and vibration.

9.5.2.4 CMMs are highly precise, offering micrometer-level accuracy, making them the preferred choice for quality control, GD&T, and precision machining verification. While CMMs require a controlled environment, stable fixturing, and trained operators, they excel at measuring critical geometric features such as tight tolerances, holes, and sharp edges.

9.5.2.5 Handheld 3D scanners are best suited for freeform and organic surfaces, whereas CMMs are ideal for rigid parts with strict dimensional requirements. Cost considerations also play a role, with handheld scanners typically having a lower initial investment but potentially requiring additional software for advanced analysis. Ultimately, the choice depends on the specific measurement needs, part geometry, and the level of accuracy required for the application.

Table 9-1. Handheld Scanners vs. CMM

Consideration	Handheld 3D Scanner	CMM
Measurement Accuracy	Typically lower accuracy (±0.02 - ±0.1 millimeter (mm) but suitable for general reverse engineering and inspection.	High accuracy (0.001 - ±0.01 mm), ideal for precision metrology and GD&T verification.
Speed and Efficiency	Fast data acquisition; capable of scanning large surfaces quickly.	Slower, point-by-point probing; requires more time for complex geometries.
Ease of Use	Portable and user-friendly; requires minimal setup and training.	Requires trained operators and careful setup for repeatable accuracy.

Table 9-1. Handheld Scanners vs. CMM - Continued

Consideration	Handheld 3D Scanner	CMM
Surface Compatibility	Can capture reflective and transparent surfaces with pretreatment (e.g., scanning spray).	Works on all materials but may require special probes for soft or delicate materials.
Part Size and Flexibility	Ideal for large or complex parts, especially those difficult to position on a fixed machine.	Best for smaller, rigid parts that can be securely fixtured on the measurement table.
Environmental Sensitivity	Sensitive to external lighting, vibration, and operator movement.	Requires a stable, controlled environment for optimal performance.
Non-Contact vs. Contact	Non-contact measurement with laser or structured light technology.	Contact-based measurement using a stylus probe (unless equipped with an optical scanner).
Geometric Feature Measurement	Best suited for organic, freeform, and complex surfaces; struggles with deep holes and tight tolerances.	Excellent for precise geometric features, including small holes, edges, and sharp corners.
Mobility and Portability	Highly portable; can be used in field or in production environments.	Stationary; requires dedicated space in a metrology lab or shop floor.
Fixture Requirements	Typically no fixturing required; scanner moves around the part.	Requires secure fixturing for repeatability and accuracy.
Automation and Integration	Can integrate with robotic systems for automated scanning but typically used manually.	Can be automated with CNC-controlled probe paths for consistent measurements.
Software Compatibility	Works with point cloud processing, reverse engineering, and CAD comparison software.	Integrated with metrology software for GD&T, CAD comparison, and statistical process control.
Cost Consideration	Lower upfront cost; minimal infrastructure required.	Higher investment cost, including maintenance and calibration expenses.
Best Applications	Reverse engineering, rapid prototyping, wear analysis, and large-scale inspections.	Precision machining verification, quality control, first-article inspection, and aerospace applications.

9.6 GENERAL SCANNING OPERATIONS – HANDHELD SCANNER.

Handheld 3D scanners are portable, high-precision metrology tools designed for fast and accurate data acquisition. They operate using specialized software and require proper setup, calibration, and part preparation to ensure optimal performance. The scanning process involves connecting the device, configuring scanning parameters, and utilizing positioning targets for spatial alignment. With user-friendly interfaces and smart control features, handheld scanners simplify complex scanning tasks, making them essential for applications such as reverse engineering, quality control, and precision measurement. For detailed instructions on using the hand-held scanner and reverse engineering techniques, government personnel can access Metal Fabrication and Technologies Office_AMT 24-001, **Reverse Engineering Guidance: Techniques, Tools, and Best Practices** This document is available at MFTO_AMT 24-001.

CHAPTER 10
PRESSES AND BRAKES

10.1 GENERAL.

Fabrication shop equipment plays a crucial role in shaping, forming, and assembling metal components for various applications in Aircraft Metals Technology (AMT) and Aircraft Structural Maintenance (ASM) shops. These machines provide the necessary force and precision to bend, shear, press, and form metal sheets and structural components efficiently. Understanding the function and operation of each type of equipment is essential for maintaining accuracy, productivity, and workplace safety. This chapter provides a general overview of key fabrication shop equipment, including sheet metal brakes, box and pan brakes, hydraulic presses, arbor presses, and hydroforming presses. Each machine will be described by its primary function, typical applications, and general operating principles. Additionally, tooling and dies associated with these machines will be briefly discussed, along with basic forming and press techniques to ensure proper usage and optimal results. While this chapter does not delve into advanced metal forming methods, it will outline the fundamental principles necessary for effective fabrication. Tables will be included where beneficial to summarize important specifications, material compatibilities, and operational guidelines.

10.2 SAFETY.

Safety is paramount when operating brakes and presses, as improper use can result in severe injuries. Operators must always adhere to safety guidelines set forth by Occupational Safety and Health Administration (OSHA) standards (29 Code of Federal Regulations (CFR) 1910) and applicable Air Force Instructions (AFIs) and Air Force Manuals (AFMAN) (e.g., DAFMAN 91-203). Supervisors are responsible for maintaining manufacturer manuals and integrating them into on-the-job training and job safety training. In the absence of manufacturer guidance, a hazard analysis and local operating instructions must be developed. Operators must:

- Consider OSHA standards, relevant Air Force (AF) guidance and/or Original Equipment Manufacturer (OEM) recommendations for the specific machine and operation.
- Consult the specific manufacturer’s manual for the brakes and presses being used. Brakes and presses vary significantly, and relying solely on general guidance can be dangerous.
- Wear safety glasses, face shields (if necessary), hearing protection, cut-resistant gloves (when applicable).
- Ensure all machine guards, barriers, and other safety devices are used to prevent accidental contact with moving components and pinch-points.
- Check machine components for wear, cracks, or any damage.
- Be familiar with emergency stop buttons and safety features to understand how to quickly shut down the machine in case of an emergency.
- Remove or secure loose clothing and jewelry.
- Long hair must be secured to prevent entanglement with moving parts to minimize the risk of injury.
- Keep the work area free from metal scrap, excess lubricant, and debris reduces trip hazards and minimizes fire risks.

10.3 TYPES OF BRAKES.

Brakes are essential tools in metal fabrication, used to bend and form sheet metal with precision. Several types of brakes are commonly found in fabrication shops (Figure 10-1), each designed for specific applications and material handling requirements. (See Table 10-1.) Each type of brake has unique capabilities and limitations, and selecting the appropriate brake depends on factors such as material thickness, bend complexity, and production volume. The following sections will explore the types of operations, applications, and tooling considerations for each type of brake in more detail. The most commonly used brakes include:

10.3.1 Sheet Metal Brakes. Also known as cornice brakes, these are used for making simple, straight bends in sheet metal. They feature a clamping mechanism to hold the material while the operator lifts a leaf to create the bend.

10.3.2 Box and Pan Brakes. Similar to sheet metal brakes but equipped with removable fingers, allowing the user to bend metal into complex shapes such as boxes and enclosures. The adjustable tooling provides greater flexibility for intricate forming tasks.

10.3.3 Hydraulic Press Brakes. These powered machines use hydraulic pressure to apply controlled force to bend metal. They offer higher precision, greater bending capacity, and the ability to form thicker materials compared to manual brakes.

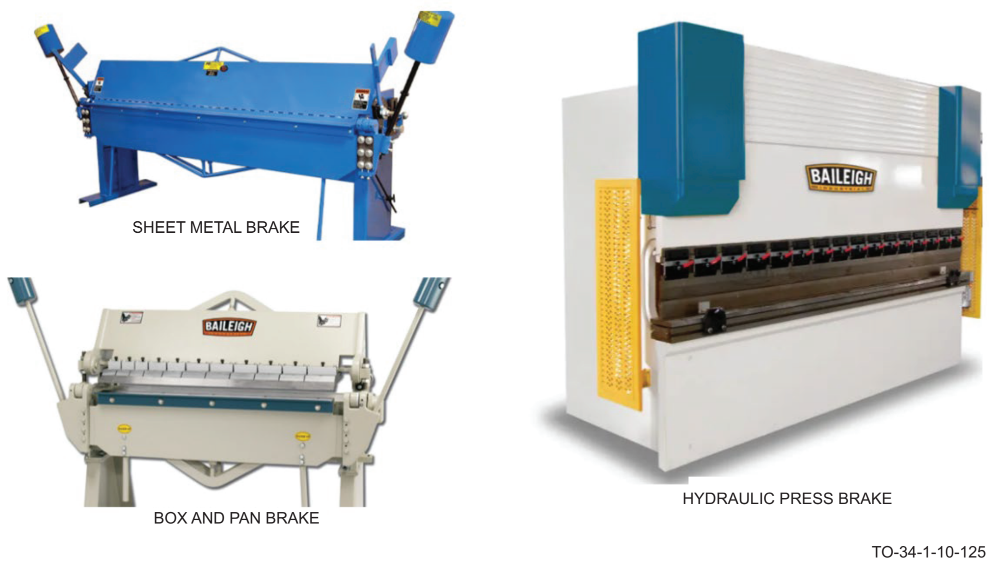


Figure 10-1. Types of Brakes

10.3.4 Operation of Metal Brakes. The operation of metal brakes varies depending on the type of brake and the complexity of the bend required. However, the general process for forming metal using brakes follows similar fundamental steps. By following these steps, operators can achieve consistent and precise bends while minimizing defects. Proper training and adherence to machine specifications ensure safe and effective brake operations.

10.3.4.1 Material Preparation. The metal sheet must be measured, marked, and cut to the required dimensions before bending to ensure precise measurements prevent errors and ensure uniform bends.

10.3.4.2 Machine Setup. The brake must be properly adjusted for the specific material thickness and desired bend angle. For hydraulic press brakes, settings such as pressure and back gauge position must be configured.

10.3.4.3 Clamping the Material. The sheet metal is positioned against the bending edge, and the clamping mechanism secures it in place. Proper alignment is crucial for achieving an accurate bend.

10.3.4.4 Bending Process. The operator engages the bending mechanism, whether manually lifting the leaf in a sheet metal brake or activating the hydraulic system in a press brake. The force applied gradually bends the metal to the required angle.

10.3.4.5 Verification and Adjustment. After bending, the part is checked for accuracy using angle measurement tools or templates. If necessary, adjustments can be made by reapplying pressure or modifying settings on the machine.

10.3.4.6 Finishing. Any sharp edges, deformations, or irregularities are corrected as needed. The part may undergo further forming, welding, or assembly processes depending on the application.

10.3.4.7 Brakes. Brakes are widely used in various industries and fabrication tasks, offering versatility in metal forming applications. The selection of the appropriate brake type depends on factors such as material thickness, production volume, and required precision. Proper understanding of brake applications ensures efficient and accurate metal forming across various industries. The primary applications of brakes include:

10.3.4.7.1 Aircraft Skin Panels. Bending aluminum and titanium sheets for fuselage, wing, and empennage skins.

10.3.4.7.2 Stringers and Longerons. Forming stiffening components that reinforce the aircraft structure.

10.3.4.7.3 Bulkheads and Frames. Shaping internal structural components for fuselage support.

10.3.4.7.4 Leading and Trailing Edge Components. Forming wing and control surface edges.

10.3.4.7.5 Ducting and Fairings. Bending sheet metal for aerodynamic covers and internal air ducts.

10.3.4.7.6 Patch Repairs on Fuselage and Wings. Fabricating formed aluminum patches for structural repairs.

10.3.4.7.7 Flanged Doublers and Stiffeners. Reinforcing cracked or weakened structural components.

10.3.4.7.8 Replacement of Dented or Corroded Panels. Forming new panels to match original contours.

10.3.4.7.9 Hinge Brackets and Mounting Plates. Fabricating small precision brackets for structural attachment points.

10.3.4.8 Tooling. Tooling for sheet metal brakes, box and pan brakes, and hydraulic press brakes is essential for achieving accurate and consistent bends in metal fabrication. The selection of the right tooling depends on the material, thickness, and desired bend specifications. (See Table 10-1.) Proper maintenance and inspection of the tooling ensure optimal performance and longevity.

10.3.4.8.1 Sheet Metal Brakes. Typically use straight or V-shaped dies, chosen based on the required bend angle and radius.

10.3.4.8.2 Box and Pan Brakes. Feature segmented dies that allow for multiple bends, especially for complex shapes such as boxes or pans.

10.3.4.8.3 Hydraulic Press Brakes. Utilize various die types, including air bending dies, bottoming dies, and coining dies, each designed for specific bending methods and material thicknesses.

10.3.4.8.4 Material Compatibility. Tooling must be selected according to the material type, thickness, and bend radius to ensure precision.

10.3.4.8.5 Tool Maintenance. Regular inspection and maintenance of dies and punches are essential for maintaining tool performance and ensuring consistent, high-quality bends.

Table 10-1. Brake Types, Operations, Applications, and Tooling Requirements

Brake Type	Operations	Applications	Tooling Requirements
Sheet Metal Brakes	Bending, folding, and forming flat sheets	General sheet metal fabrication, heating, ventilation, and air conditioning, enclosures	Straight or V-shaped dies, punch for specific bend angles and radii
Box and Pan Brakes	Bending complex shapes, multi-angle bends	Fabrication of boxes, pans, and enclosures	Segmented dies, adjustable finger bars for multiple bend angles
Hydraulic Press Brakes	Bending, air bending, coining, and bottoming	Large-scale metal forming, automotive parts, panels	Air bending dies, bottoming dies, coining dies, specialized tooling for different thicknesses and bend radii
Manual Press Brakes	Light to moderate bending	Small-scale fabrication, prototypes, and custom work	Basic punch and die sets, typically straight or simple V dies

10.4 TYPES OF PRESSES.

Presses are essential machines in fabrication shops, used for shaping, forming, and assembling metal components by applying force through various mechanisms. Different types of presses (Figure 10-2) are designed for specific applications, material thicknesses, and production requirements. Each type of press offers unique advantages based on the required force, precision, and application. The following sections will explore the types of operations, tooling considerations, and best practices for achieving optimal results. The most commonly used types include:

10.4.1 Hydraulic Presses. Utilize hydraulic fluid pressure to generate force, providing high precision and the ability to apply consistent force over a large area. They are ideal for deep drawing, stamping, and forming thick materials.

10.4.2 Arbor Presses. Hand-operated presses used for small-scale pressing, assembly, and light forming tasks. These presses are commonly used for staking, riveting, and installing bearings.

10.4.3 Hydroforming Presses. Specialized presses that use high-pressure hydraulic fluid to shape ductile materials into complex shapes. They are widely used in aerospace and automotive applications for producing lightweight, high-strength components.

10.4.4 Press Operations. Press operations involve the use of various machines and tools to perform shaping, forming, and cutting operations on materials such as metals and composites. The process generally involves applying a high amount of force to a workpiece to achieve the desired shape or cut. Presses can be categorized into mechanical, hydraulic, and pneumatic types, each suited for different applications based on force requirements, material properties, and precision.

Shaping - Using presses to form material into specific shapes or profiles.
Cutting - Involves shearing or stamping material to specific sizes or forms.
Punching - Punches a hole or shape into the material using a punch and die set.
Forming - Material is bent or formed into shapes such as flanges, curves, or angles.
Blanking - Cutting material into desired shapes or sizes for further processing.

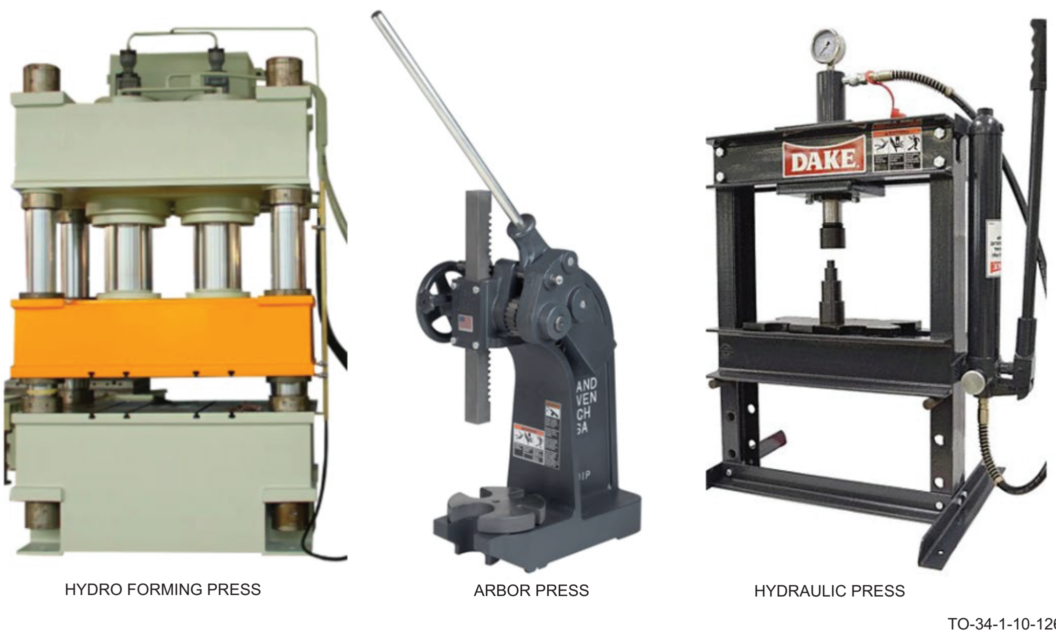


Figure 10-2. Types of Presses

10.4.4.1 Appropriate Tooling. Selecting the appropriate tooling is critical to the success of press operations. Tooling must be chosen based on the material type, thickness, and the specific operation to be performed. Properly designed tooling ensures precision, improves operational efficiency, and extends the life of the press machine. Tool maintenance is also vital to ensure continued performance and avoid defects in the final product.

10.4.4.1.1 Material Compatibility. Tooling should match the material being worked with, such as selecting hardened steel tools for harder metals.

10.4.4.1.2 Die Design. Die design plays a significant role in the quality of the formed or cut part. Proper alignment, clearance, and geometry are essential.

10.4.4.1.3 Tooling Maintenance. Regular inspection and maintenance are necessary to avoid wear, cracking, or misalignment, which can affect quality and operational efficiency.

10.4.4.1.4 Tooling Materials. Tooling materials must withstand the stresses of the press operation, with hardened steel, carbide, and other durable metals being common choices.

10.4.4.1.5 Clearance and Tolerances. Correct die clearance and tolerance levels should be carefully considered to ensure clean cuts and proper material flow.

10.4.4.2 Best Practices. Following best practices during press operations can significantly improve efficiency, safety, and product quality. Proper setup, safety measures, and process monitoring are key elements to ensure optimal performance and longevity of both tooling and machines. Adopting best practices also minimizes downtime and defects.

10.4.4.3 Proper Setup. Always ensure the press machine is properly aligned, calibrated, and adjusted according to the specifications of the tooling and material being used.

10.4.4.4 Safety Protocols. Adhere to safety guidelines, such as using proper guarding, wearing protective gear, and ensuring that emergency stops are functional.

10.4.4.5 Quality Control. Regularly inspect parts during and after press operations for dimensional accuracy and surface quality.

10.4.4.6 Preventative Maintenance. Schedule regular maintenance and inspections for both machines and tooling to detect wear and prevent breakdowns.

10.4.4.7 Material Handling. Ensure that materials are properly fed into the press to avoid misalignment or jams, which could cause damage to the press or tooling.

Table 10-2. Press Operations, Considerations, and Best Practices

Application	Detail	Description
Press Operations	Shaping	Forming material into specific shapes or profiles.
	Cutting	Shearing or stamping material to specified sizes or forms.
	Punching	Creating holes or shapes in the material using a punch and die.
	Forming	Bending material into shapes like flanges, curves, or angles.
	Blanking	Cutting material into specific shapes or sizes for further processing.

Table 10-2. Press Operations, Considerations, and Best Practices - Continued

Application	Detail	Description
Tooling Considerations	Material Compatibility	Tooling must match the material being worked with (e.g., hardened steel tools for hard metals).
	Die Design	Proper alignment, clearance, and geometry are critical for quality parts.
	Tooling Maintenance	Regular inspections and maintenance to prevent wear, misalignment, or damage.
	Tooling Materials	Use durable materials like hardened steel, carbide, and others for tooling.
	Clearance and Tolerances	Correct die clearance and tolerances are necessary to ensure clean cuts and proper material flow.
Best Practices	Proper Setup	Ensure press machine is aligned, calibrated, and adjusted to specifications.
	Safety Protocols	Use proper guarding, wear protective gear, and ensure functional emergency stops.
	Quality Control	Inspect parts regularly during and after operations for dimensional accuracy and surface quality.
	Preventative Maintenance	Schedule regular inspections for press machines and tooling to detect wear and avoid breakdowns.
	Material Handling	Properly feed material into the press to avoid misalignment, jams, or tool damage.

10.5 BRAKE OPERATIONS.

Operations on sheet metal brakes, box and pan brakes, and hydraulic press brakes are essential for various AMT and ASM tasks, enabling precise and efficient shaping, bending, and forming of materials used in aircraft maintenance and repair operations. Each brake type offers unique capabilities, making them suitable for different tasks in the fabrication of components for aircraft structures, panels, and other metal parts.

10.5.1 Commonly Found Brakes in ASM. Sheet metal brakes, box and pan brakes, and hydraulic press brakes are essential tools for fabrication and repair operations, each serving different purposes depending on the complexity of the part and the material being worked with. Understanding the capabilities and applications of each brake ensures that the appropriate brake is selected for the required operation, leading to efficient and accurate fabrication of aircraft components. Below is an expanded breakdown of these operations, their capabilities, and applications.

10.5.1.1 Sheet Metal Brakes.

10.5.1.1.1 Primary Operation. Sheet metal brakes are designed for the bending of flat sheet metal into specific angles, typically between 0 and 135 degrees. This operation is performed by pressing the metal sheet between a punch and die to create the desired bend.

10.5.1.1.2 Capabilities. These brakes are versatile and can handle a range of metal thicknesses and lengths. They are primarily used for lighter, thinner materials, such as aluminum, mild steel, and galvanized sheet metal.

10.5.1.1.3 Applications. Sheet metal brakes are commonly used for fabricating parts such as panels, brackets, and other structural components. They are ideal for producing uniform bends in aircraft components, such as fuselage panels, control surfaces, or other sheet metal assemblies.

10.5.1.1.4 Operation Details. The operator positions the sheet metal in the brake, and the machine's ram moves downward, applying pressure on the metal until it conforms to the angle of the die. The punch or die configuration can be adjusted to produce different bend radii and angles as needed for specific applications.

10.5.1.2 Box and Pan Brakes.

10.5.1.2.1 Primary Operation. Box and pan brakes allow for more complex, multi-bend operations, especially for parts that require multiple bends in different directions. These brakes feature segmented dies (also known as fingers) that are adjustable, allowing for varied bend angles.

10.5.1.2.2 Capabilities. These machines are designed to create shapes such as boxes, pans, and other components that require multiple bends or intricate angles in the same piece of material.

10.5.1.2.3 Applications. Box and pan brakes are used for creating more intricate parts, such as custom-sized panels, brackets, and other components requiring multiple bends. They are particularly useful when a part has more than one bend or requires bends at different angles, such as in the fabrication of toolboxes, trays, or complex structural parts for aircraft.

10.5.1.2.4 Operation Details. The adjustable finger bars allow for precise control over each bend. The operator positions the finger to hold the metal in place and uses the brake to create each bend. This setup is beneficial for making complex parts that would require more than one bend in different directions, such as with component enclosures or housings in aircraft assemblies.

10.5.1.3 Hydraulic Press Brakes.

10.5.1.3.1 Primary Operation. Hydraulic press brakes use hydraulic force to bend metal, providing more power and precision than mechanical brakes. This allows the machine to handle thicker and harder materials and ensures greater accuracy in the bending process.

10.5.1.3.2 Capabilities. Hydraulic press brakes are capable of bending a wide range of material thicknesses, from thin sheets to thick plates, including various metals used in aircraft components. These machines offer higher precision and are ideal for forming parts that require high levels of accuracy.

10.5.1.3.3 Applications. Hydraulic press brakes are commonly used for bending thicker metals or when precise bends are required in structural components or more complex parts, such as bulkheads, ribs, or other primary structural elements.

10.5.1.3.4 Operation Details. The hydraulic system applies force through a ram that moves downward to bend the material. Precision is controlled by adjusting the stroke length of the ram and the tooling setup. Hydraulic press brakes can be equipped with various dies, including air bending, bottoming, and coining dies, each suitable for different bending methods and material properties. Programmable controls are often used to ensure repeatable, accurate results in high-precision tasks.

10.6 PRESS OPERATIONS.

Presses, including arbor presses, hydraulic presses, and hydroforming presses, are integral in AMT and ASM operations for shaping, forming, and cutting materials. Each press type offers distinct features and capabilities suited to specific applications in the fabrication and maintenance of aircraft components.

10.6.1 Commonly Found Presses in AMT and ASM. Arbor presses, hydraulic presses, and hydroforming presses – all offer unique capabilities suited to specific applications in fabrication operations. Arbor presses are best for smaller, lighter tasks, hydraulic presses provide powerful force for heavier operations, and hydroforming presses enable the creation of complex, high-strength parts with precision. Understanding the appropriate press for a given task ensures that the correct method is applied for efficient, accurate, and high-quality results in aircraft component fabrication and repair. Below is an expanded breakdown of the operations, capabilities, and applications of each press type.

10.6.1.1 Arbor Presses.

10.6.1.1.1 Primary Operation. Arbor presses are manually operated presses that utilize a mechanical lever system to apply force for light pressing, punching, and bending operations. These presses typically operate with a vertical ram driven by a hand lever or wheel.

10.6.1.1.2 Capabilities. Arbor presses are best suited for small-scale operations involving light to medium-duty tasks. They can handle smaller parts or low-volume operations, such as pressing bearings, bushings, or installing rivets.

10.6.1.1.3 Applications. In aircraft maintenance, repair, and overhaul, arbor presses are used for assembling parts like bearings, bushings, or inserting pins in smaller components. They are also useful for tasks such as riveting or setting small fasteners where a lighter, manual force is required.

10.6.1.1.4 Operation Details. The operator positions the workpiece under the ram, and by manually applying pressure using the lever, force is exerted to perform the necessary operation. Arbor presses provide precise control for light-duty tasks and are ideal for small components where minimal force is needed.

10.6.1.2 Hydraulic Presses.

10.6.1.2.1 Primary Operation. Hydraulic presses use hydraulic fluid to generate force, allowing for the bending, forming, and punching of materials. The press applies force to the material through a ram, driven by hydraulic cylinders, offering significantly more power than mechanical presses.

10.6.1.2.2 Capabilities. Hydraulic presses are capable of applying substantial force, making them ideal for medium to heavy-duty operations. These presses can handle a wide range of materials, including metals used in aircraft structures, and are suited for tasks such as bending, forming, or cutting thicker materials.

10.6.1.2.3 Applications. In aircraft maintenance, repair, and overhaul, hydraulic presses are commonly used for forming large, heavy parts, such as bulkheads, panels, or brackets. They are also employed for tasks like straightening bent parts, de-painting or de-coating, or creating complex shapes that require high-force application.

10.6.1.2.4 Operation Details. The hydraulic press operates by using a pump to generate pressure in the hydraulic fluid, which drives the ram. The ram then applies force to the material to bend, shape, or cut it. Hydraulic presses often come with adjustable controls to regulate pressure, stroke length, and speed, ensuring precision when forming or shaping materials used in aircraft components.

10.6.1.3 Hydroforming Presses.

10.6.1.3.1 Primary Operation. Hydroforming presses use high-pressure hydraulic fluid to shape malleable materials, typically metal, into precise shapes. The material is placed in a die, and pressurized fluid forces the material to conform to the die's shape, allowing for the creation of complex, high-strength parts.

10.6.1.3.2 Capabilities. Hydroforming is well-suited for creating complex shapes with minimal material waste. It provides a high level of precision while maintaining the integrity of the material, making it ideal for parts that require both strength and intricate geometries.

10.6.1.3.3 Applications. Hydroforming presses are used for creating lightweight, high-strength components, such as structural aircraft parts, fuselage sections, or other parts that require complex bends and shapes. This method is particularly useful for forming thin-walled, high-strength parts with tight tolerances.

10.6.1.3.4 Operation Details. The process begins by placing the material in a die and sealing it within the press. High-pressure hydraulic fluid is then introduced into the die, forcing the material to conform to the die's shape. This method allows for the production of parts with high precision and reduced material wastage, making it ideal for creating lightweight yet strong aircraft components.

CHAPTER 11
COMPUTER NUMERICAL CONTROL (CNC) MACHINERY FACILITIES GUIDANCE

11.1 GENERAL.

This chapter provides guidance for establishing suitable facilities for deploying CNC machinery within United States Air Force (USAF) Aircraft Metals Technology (AMT) shops. The primary function of these facilities is to support the repair and manufacture of weapons systems and aerospace equipment components. Ensuring proper facilities for CNC machinery is critical for maintaining operational readiness, extending equipment lifespan, and promoting a safe and efficient working environment. Prioritizing the selection and preparation of a suitable facility is paramount to maximizing machinery capabilities and ensuring safety and productivity.

These facilities will accommodate a variety of CNC machine types, each categorized by function and capacity. The CNC machinery covered by this guidance includes:

- **Type 1: CNC Lathes** (Class A: Flatbed, Class B: Slantbed) - Primarily used for turning operations, creating cylindrical parts and features.
- **Type 2: CNC Milling Centers** (Class A: Small, Class B: Medium, Class C: Large) - Used for a wide range of milling operations, creating complex shapes and features through material removal.
- **Type 3: Abrasive Waterjet Cutting Systems** (Class A: Medium, Class B: Large) - Used for cutting various materials with high-pressure water and abrasive, ideal for intricate shapes and heat-sensitive materials.

Associated Commercial Item Descriptions (CIDs) have been published in ASSIST for the machinery noted above. The facility housing CNC machinery must meet the minimum requirements outlined in the manufacturer's owner manual concerning installation and operational environments, in addition to the requirements outlined in this chapter. Compliance with AFMAN91-203, installation occupational safety office guidance, and consultation with Civil Engineering are essential. The facility should also comply with current industry standards, best practices for machine maintenance, and safety requirements.

CNC machines are designed and intended for indoor use only. The facility must maintain operating temperature and humidity parameters specified in Table 11-1 through Table 11-3, whether the machine is active or idling. CNC machinery requires well-prepared facilities to ensure proper operation, precision performance, and long-term reliability. Regardless of type, all CNC equipment shares several core facility needs: stable foundations, reliable electrical service, clean and dry compressed air, controlled environmental conditions, and safe operator access. Inadequate preparation can result in machine instability, premature component wear, reduced accuracy, and unsafe working conditions.

The facility must provide a level and structurally sound floor to prevent vibration and misalignment. Electrical service must be dedicated, stable, and sized appropriately for the machine's load. Compressed air systems must be clean, regulated, and delivered at sufficient pressure and flow rates to support pneumatic functions. Environmental controls are equally important; temperature and humidity stability preserve tolerances, and adequate ventilation ensures safe operation, particularly when cutting fluids, mist, or abrasive media are in use. Additionally, facilities must allow safe workflow clearance around each machine for part handling, maintenance, and integration of ancillary equipment.

11.2 SAFETY.

Always adhere to manufacturer guidelines and DAFMAN 91-203. CNC machines should be installed in an area segregated from welding, cutting, grinding, open flames, spark-producing operations, and strong acids or bases. Consult your installation occupational safety office for specific local requirements and safety training.

11.3 GENERAL FACILITY LAYOUT.

The facility layout must provide sufficient space for a workbench, toolbox, accessory cabinets, and unimpeded access, complying with applicable OSHA, NFPA, DAFMAN 91-203 guidelines, and local requirements. Additional square footage may be necessary to ensure a safe, suitable, and effective work area. Specific square footage requirements are based on individual machinery Type and Class, as detailed in Table 11-1 through Table 11-3. AMT shops, requiring additional machinery, will necessitate further consideration for adequate floor space. Refer to DAFMAN 91-203 or contact your installation occupational safety office and Civil Engineering for further guidance.

Specific Requirements:

- **Flooring:** The floor in the CNC machine area should facilitate easy cleanup of metal chips, shavings, and cutting fluids using common methods such as brooms, dustpans, mops, and cleaning towels. Follow local Bioenvironmental and Environmental Safety and Health requirements.
- **Foundation:** Each machine shall be placed on its own individual, solid, sound, and stable steel bar reinforced concrete slab, a minimum of 12 inches thick from the slab's edge. The concrete must have a minimum compressive strength of 3,500 Pound-force per Square Inch (PSI) at 28 days. A standard 6-inch steel bar reinforced concrete slab floor is typically suitable. Contact your installation Civil Engineering for specific guidance. The floor should be free from noticeable vibrations; avoid placing machines near sources of vibration (e.g., heavy punches, presses).
- **Office Space:** Office space for computer-aided design and computer-aided manufacturing is required to support the repair and/or manufacture of weapons system and aerospace equipment components using CNC machinery. This office space should be located as near as possible to the CNC machinery for ease of data transfer and should have adequate lighting for detailed work and inspection, designed to minimize glare and shadows.
- **Utilities:**
  - **Compressed Air:** Each machine requires a dedicated compressed air supply meeting the minimum requirements outlined in Table 11-1 through Table 11-3.
  - **Water:** Machines require access to a water supply for coolants, initial fill, and periodic topping off, as well as a drain for periodic maintenance. Contact your local Bioenvironmental Flight for guidance when draining machinery.
  - **Electrical Service:** CNC machines require a dedicated electrical service. Refer to Table 11-1 through Table 11-3 for the electrical service requirements for each machine Type and Class.

11.4 MACHINE SPECIFIC ENVIRONMENTAL REQUIREMENTS.

In addition to the general facility layout requirements, adherence to specific environmental conditions is crucial for proper CNC machine operation. Operating temperature and humidity must be maintained within the parameters specified in Table 11-1 through Table 11-3, regardless of whether the machine is in use or idle. To ensure optimal performance and prevent localized temperature fluctuations, avoid positioning the machine near heat sources (e.g., radiators), cooling sources (e.g., air conditioners, drafts), or direct sunlight. The HVAC system must be capable of maintaining temperature within the ranges detailed in Table 11-1 through Table 11-3. Furthermore, adherence to altitude restrictions and maximum equipment weight/floor loading limits, as defined in Table 11-1 through Table 11-3, are mandatory.

Table 11-1. Type 1 - CNC Machinery Utility Requirements

Requirement	Class A:	Class B:	Details
Workspace Area, Minimum	112 square feet	180 square feet	
Ceiling Height, Minimum	97 inches	114 inches	
Temperature	Minimum 41 °F, Maximum 122 °F		

Table 11-1. Type 1 - CNC Machinery Utility Requirements - Continued

Requirement	Class A:	Class B:	Details
Humidity	20-90 percent relative humidity		
Altitude	Sea Level - 6,000 feet		
Maximum Equipment Weight and Floor Loading	4,200 pounds distributed on four pads	15,000 pounds distributed on four pads	
Electrical Service	220 Volt Alternating Current 25 Ampere (A) Three Phase, or 440 VAC 13A Three Phase, or 220 VAC 40A Single Phase	ST-10 and ST-20 or equivalent 220 VAC 40A Three Phase; ST-30 or equivalent 440 VAC 20A Three Phase, or 220 VAC 70A Three Phase, or 440 VAC 35A Three Phase	Machines should be on a dedicated circuit. Three phase power is recommended.
Compressed Air Service	80 PSI minimum; 2 horsepower compressor, 20-gallon tank minimum, per machine; 4 Standard Cubic Feet per Minute (SCFM) at 100 PSI		

Table 11-2. Type 2 - CNC Machinery Utility Requirements

Requirement	Class A:	Class B:	Class C:	Details
Workspace Area, Minimum	112 square feet	152 square feet	238 square feet	
Ceiling Height, Minimum	97 inches	140 inches	171 inches	
Temperature	Minimum 41 °F, Maximum 122 °F			
Humidity	20-90 percent relative humidity			
Altitude	Sea Level - 6,000 feet			
Maximum Equipment Weight and Floor Loading	4,200 pounds	15,000 pounds	25,000 pounds	Maximum equipment operating weight distributed on four pads.
Electrical Service	220 VAC 25A Three Phase, or 440 VAC 13A Three Phase, or 220 VAC 40A Single Phase	220 VAC 70A Three Phase	220 VAC 70A Three Phase or 440 VAC 35A Single Phase	Machines should be on a dedicated circuit. Three phase power is recommended.
Compressed Air Service	80 PSI minimum; 2 horsepower compressor, 20-gallon tank minimum, per machine; 4 SCFM at 100 PSI			

Table 11-3. Type 3 - CNC Machinery Utility Requirements

Requirement	Class A:	Class B:
Workspace Area, Minimum	140 square feet	160 square feet
Ceiling Height, Minimum	136 inches	
Temperature	Minimum 50 °F, Maximum 90 °F	
Humidity	≤ 95 percent	
Maximum Equipment Weight and Floor Loading	13,000 pounds. operating weight, distributed on five skids.	22,000 pounds. operating weight, distributed on seven skids.
Electrical Service	Machine should be on a dedicated circuit. 380-480 VAC ±10 percent 50 - 60 Hertz Three Phase	
Compressed Air Service	Minimum 75 PSI, Maximum 120 PSI ≥ 2.0 SCFM (additional machine options may require ≥ 16.0 SCFM) Air quality must be clean, dry, and oil free.	

11.5 FACILITY MAINTENANCE.

Ongoing facility maintenance is essential for sustaining the accuracy, safety, and reliability of CNC machinery. While machine-specific preventative maintenance addresses lubrication, calibration, and component wear, facility-level upkeep ensures the supporting infrastructure-power supply, compressed air, water handling, foundations, and environmental controls-remains within required operating standards. Poorly maintained facilities can lead to misalignment, voltage fluctuations, coolant contamination, or unsafe conditions that directly affect machine performance. To mitigate these risks, a structured preventative maintenance program must be implemented for the facility itself. This program should encompass the following tasks:

- Daily: Verify air pressure, monitor shop temperature and humidity, and confirm electrical readiness.
- Weekly: Drain condensate from air systems, check coolant and water quality, and ensure ventilation and lighting remain effective.
- Monthly: Inspect electrical connections, verify machine leveling, and clean drains or chip collection areas.
- Annual: Conduct vibration testing of foundations, review power quality, and ensure compliance with relevant standards (e.g., Air Force (AF) standards).

Figure 11-1 outlines facility maintenance actions in detail and provides supervisors and facility managers with a reference framework for scheduling and documentation. Consistent application of these practices helps preserve machining accuracy, extend equipment service life, and maintain compliance with safety and environmental regulations.

General Facility Maintenance (CNC Support Systems)	
Daily	
☐	Verify shop temperature and humidity are within acceptable limits.
☐	Check compressed air supply pressure and dryness.
☐	Inspect electrical panels for tripped breakers or unusual heat.
☐	Confirm work areas are free of obstructions and debris.
Weekly	
☐	Drain condensate from compressed air dryers/filters.
☐	Check coolant/water supply quality and refill as needed.
☐	Inspect lighting levels around workstations.
☐	Verify that exhaust/ventilation systems are functioning.
Monthly	
☐	Inspect electrical connections for wear, corrosion, or overheating.
☐	Level-check major machines using precision tools.
☐	Clean facility drains and chip/waste collection areas.
☐	Test emergency shutoffs and safety interlocks.
Annual	
☐	Calibrate facility power supply monitoring equipment.
☐	Conduct vibration and settlement inspection on foundations.
☐	Replace worn electrical panels, conduit, or outlets as required.
☐	Review facility safety compliance (OSHA, local codes, environmental).

TO-34-1-10-127

Figure 11-1. General Facility Maintenance List