Basic welding works

Level-I

Based on April 2022, Curriculum Version 1

Module Title: - Perform Bench work

Module code: IND BWW1 M04 0322

Nominal duration: 100Hour

Prepared by: Ministry of Labour and Skill

August, 2022

Addis Ababa, Ethiopia

Table of Content

Introduction to the module……………………………………………………………5

Unitone:………………………………………………………………………………..........................6

1.1. OHS policies and procedures……………………………………………………….6

1.2. Manage and maintain work area…………………………………………………8

1.3. Plan and apply work activities…………………………………………………….8

1.4. Mark dimensions on work pieces…………………………………………………9

Self check-1……………………………………………………………………………10

Unit two: Set umachines……………………………………………………………………………..25

2.1. Inspect and sharpen tools………………………………………………………26

2.2. Mount and position tools…………………………………………………………26

2.3. Calculate speed and feed…………………………………………………………..33

2.4. Set up operation…………………………………………………………………..36

Self check-2…………………………………………………………………………….37

Unit three: Perform hand tool operation…………………………………………………………….39

3.1. Clamp work pieces……………………………………………………………….40

3.2. Select and use hand tools…………………………………………………………41

3.3. Cut chip file and scrap wor pieces………………………………………………43

3.4.Cut threads……………………………………………………………………………49

3.5. Perform bench work operation……………………………………………………55

3.6. Recoginize and report hazards…………………………………………………….56

Self check-3……………………………………………………………………………….58

Operation sheet3.1………………………………………………………………………60

LAP test 3…………………………………………………………………………………..61

Unit four; Bsic drill ream and hone operation……………………………………………………….62

4.1.Preventative OHS…………………………………………………………………….62

4.2.Drill bore reamand hone holes……………………………………………………….63

4.3.Self check-4……………………………………………………………………………..75

Unit five: Perform off-handgrind cutting operations……………………………………………………79

5.1. Hone cut edges and remove rbburrs…………………………………………….79

5.2.Sharpen cutters……………………………………………………………………….81

5.3 Use appropriate cooling agent………………………………………………………82

5.4 perform cutting tool grinding……………………………………………………………86

Unit six: Assure quality of finished components……………………………………………………90

6.1. Check component for conformance…………………………………………………91

6.2. Use appropriate techniques of measuring tools and equipment……………………92

6.3. Handle deviation………………………………………………………………………92

Self check-6………………………………………………………………………………94

Acknowledgment

Ministry of Labor and Skills and Ministry of Health wish to extend thanks and appreciation to the many representatives of TVET instructors and respective industry experts who donated their time and expertise to the development of this Teaching, Training and Learning Materials (TTLM).

Acronym

OHS-occupational health and safety

HSS-high speed steel

TDS-tap drill size

PPE-personal protective equipment

RPM-revolution per minute

Introduction to the Module

The term 'bench work' refers to the production of components by hand on the bench, where as fitting deals which the assembly of mating parts, through removal of metal, to obtain the required fit. Both the bench work and fitting requires the use of number of simple hand tools and considerable manual efforts.

This module covers the units:

- Plan and prepare task
- Set-up machine
- Perform hand tool operations
- Perform basic drill, ream and hone operations
- Perform Off-hand grind cutting tools
- Assure Quality of finished component

Learning Objective of the Module

- Plan work
- Perform hand tool operation and set-up machine
- Perform drilling and grinding operation
- Apply Quality for finished component

Module Instruction

For effective use this modules trainees are expected to follow the following module instruction:

1. Read the information written in each unit
2. Accomplish the Self-checks at the end of each unit
3. Perform Operation Sheets which were provided at the end of units
4. Do the "LAP test" giver at the end of each unit and
5. Read the identified reference book for Examples and exercise

Unit one: Plan and prepare task
This unit is developed to provide you the necessary information regarding the following content coverage and topics: • OHS policies and procedures • Manage and maintain work area. • Plan and apply Work activities • Select Materials • Mark Dimensions on work piece This unit will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to: • Apply OHS policies and procedures • Identify work activities • Identify materials to specification • Clean work area

1.1.

1.1. OHS policies and procedures

A. Difference between policies and procedures

A ***policy*** is a statement about an issue in the workplace and says what the work intends to do about the issue.

A ***procedure*** sets out step-by-step instructions on how to deal with an activity in the workplace.

B. Why should you have health and safety policies and procedures?

Health and safety policies and procedures are part of a framework for effective health and safety management. A general health and safety policy states management's intention to provide a safe and healthy workplace, and states the health and safety goals of a workplace. It should also demonstrate the employer's acknowledgment of their legal duties and their intention to voluntarily comply with those duties.

Specific policies and procedures address particular issues or hazards. They are administrative measures to control workplace hazards and should be used together with other hazard control measures to eliminate or reduce the risk of workplace illness or injury.

1.2. Manage and maintain work area.

1.2.1. Manage Work Area

Working Areas Are:-

- ✓ Manufacturing and training workshops, tool rooms
- ✓ Indoor or outdoor location
- ✓ Immediate workstation space,
- ✓ Offices, class rooms and reception areas

A well-managed work area should meet such goals as:

- ✓ Confirming rules and processes are followed
- ✓ Identifying potential problems that were not anticipated during design or task analysis
- ✓ Identifying equipment deficiencies such as normal wear and tear, abuse, or misuse
- ✓ Identifying bad practice
- ✓ Identifying process requirements that are unrealistic or unattainable
- ✓ Identifying effects of changes in processes or materials
- ✓ Identifying inadequacies in hazard controls
- ✓ Providing management self-appraisal information
- ✓ Demonstrating management commitment through visible activity for health and safety.

1.2.2. Work area management

An occupational health and safety (**OHS**) **management** system encompasses more than just your health and safety program. ... Having an effective **management** system improves your ability to continuously identify hazards and **control** risks in your workplace.

How do you implement a safety management system?

a. Implementing and Operating a Health and Safety Management System

- ✓ Implement a reporting system.
- ✓ Train workers how to identify and control hazards.
- ✓ Conduct inspections.
- ✓ Collect hazard control ideas.
- ✓ Implement hazard control ideas.
- ✓ Address/anticipate workplace emergencies.
- ✓ Seek input on workplace changes.

b. What are the key elements of a safety management system?

Most successful occupational health and safety management systems contain the following key elements:

- ✓ A Way to Control and Distribute Up-to-Date Documents.
- ✓ Safety Inspection Checklists.
- ✓ Risk Assessments.
- ✓ Emergency Response Plan.
- ✓ Training Program and Documentation System

1.3. Plan and apply Work activities

Planning is the process of deciding in detail how to do something before you actually start to do it. It is the process of setting goals, developing strategies, and outlining tasks and schedules to accomplish the goals or planning is the systematic process of establishing a need and then working out the best way to meet the need, within a strategic framework that enables you to identify priorities and determines your operational principles. Planning means thinking about the future so that you can do something about it now. This doesn’t necessarily mean that everything will go according to plan. It probably won’t. But if you have planned properly, your ability to adjust, without compromising your overall purpose, will be that much greater.

b. Why do Plan?

Four reasons for planning:-

- Provides direction
- Reduces uncertainty
- Minimizes waste and redundancy
- Sets the standards for controlling

1.4. Mark Dimensions on work piece

Concepts of Marking

Marking out and measuring is a critical part of manufacturing and is usually subject to a number of quality control checks. If components are marked out and measured wrongly before being cut out, there is no chance of them fitting together when they are assembled. Always take marking out measurements from a datum as indicated in Figure 4.1. A datum edge is a flat face or straight edge from which all measurements are taken. This prevents cumulative errors being made. If you are using timber, choose the face side carefully, before marking it with a small symbol for identification purposes, as shown in Figure. Then select a face edge that is at right angles to the face side. Take all your measurements from this side and/or edge.

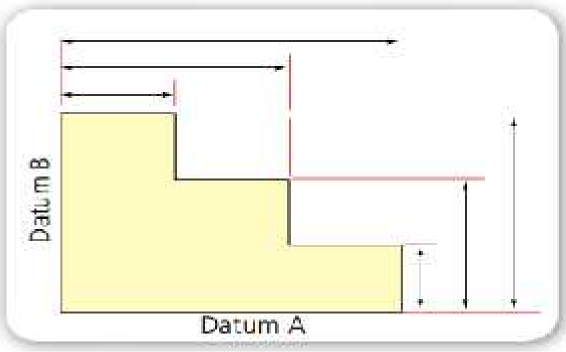


Fig.1. datum edge

These are tools used for marking out and measuring that you need to know about:

- rules
- punches
- squares
- templates
- gauges
- micrometers
- Scribers.

A. Measuring Tools

Rules:

There are two basic types of rule: steel rule and steel tape. Both start at zero and have millimeter graduations.

Steel rule

These are made up of stainless steel and are available in many sizes ranging from 1/2 ft. to 2 ft. These are marked in inches or millimeters. All the faces are machined true. The edges of steel rule should be protected from rough handling.

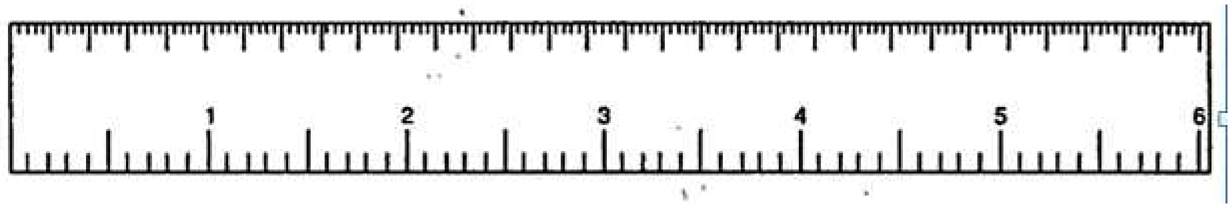


Fig.1.2. Steel rule

Micrometers:

A micrometer is a specialized instrument used to take very accurate measurements. The thimble, which rotates as the micrometer is tightened, has 50 equal divisions around its diameter, giving an accuracy of 0.01mm.A reading is taken by adding all the visible whole numbers to the nearest 0.5 mm. The reading from the thimble, which will be between 0 and 0.49 mm, is added to the main reading to get the exact measurement. Although the micrometer provides a very accurate measurement, it can be difficult to learn how to read it. A micrometer is a very useful instrument. It enables you to take measurements to within one hundredth of a millimeter (0.01mm).The metric micrometer is able to measure ranges of 25 mm (that is, for 0-25 mm, 25-50 mm, and so on).A common type is shown in Figure. The micrometer has a thread with pitch 0.5 mm.

This means that the spindle advances by 0.5 mm for each turn. However, there are 50 graduations on the thimble. So the movement advanced for each graduation of the thimble is 0.5/50 = 0.01 mm. Micrometers are one of the precision measuring tools, used to measure to one-hundredth of a mm (0.01mm). On micrometers, the pitch of a screw thread is used to determine lengths or diameters. Each complete turn of the measuring screw changes the distance between the measuring surfaces by the pitch of its thread (e.g. 0.5mm).

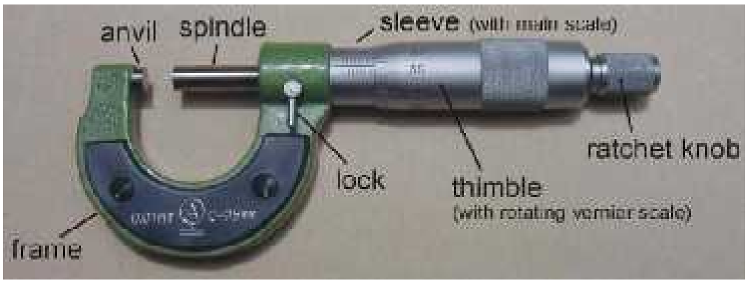


Fig1.3. outside micrometer

Types of Micrometers:

Outside Micrometer:

An outside micrometer is used for measuring outside diameter of cylindrical objects, parallel surfaces or other outside dimension. The work to be measured is placed between the anvil and the tip of the spindle.

Reading the micrometer:

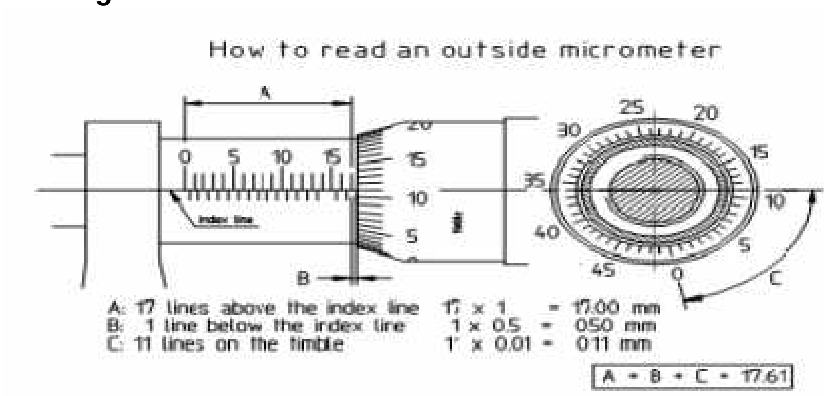


Fig.1.4. Outside micrometer reading

For example

1. Upper main scale (sleeve) 12.00 mm
2. Lower main scale (no half mm) 0.00 mm
3. Circular thimble scale 0.13 mm
4. 13 X 0.01 mm = 0.13mm
5. Total reading 12.13 mm

Inside Micrometer:

The structure of this micrometer is as similar as that of an outside micrometer. It is used for measuring internal dimensions.

Fig.1.5. inside Micrometer

Depth Micrometer:

Depth micrometers are used for measuring the depth of holes, slots, grooves, Keyway and shoulders etc. Note that the scales are graduated in reverse as compared with external or internal micrometers.

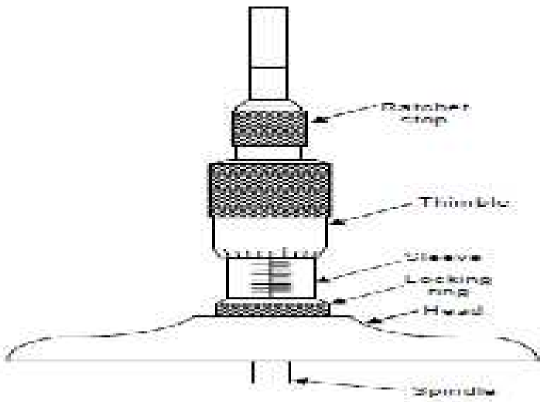


Fig.1.6. Depth micrometer

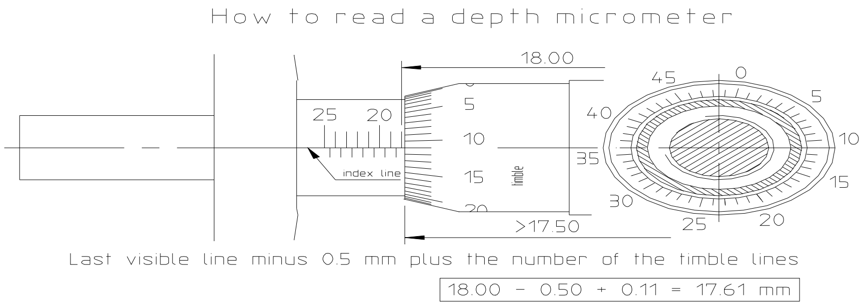


Fig.1.7. *Fig.4.8.Depth micrometer reading*

Vernier Calipers:

These are used for measuring outside as well as inside dimensions accurately. It may also be used as a depth gauge. It has two jaws. One jaw is formed at one end of its main scale and the other jaw is made part of a Vernier scale.

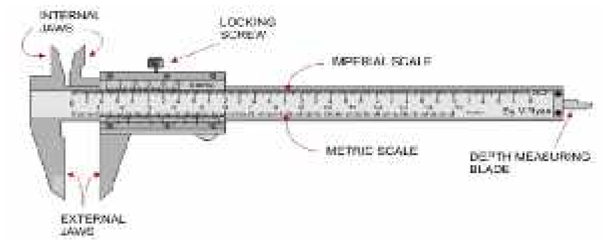


Fig.1.8. Vernier caliper

Fig.1.9.Digital caliper capable of recording inside, Outside, diagonal, hole edge and hole dimensional measurements

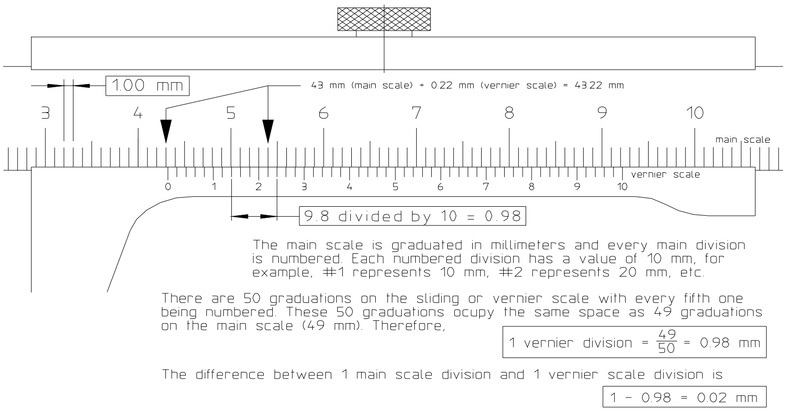


Fig.1.10.Vernier caliper reading

Calipers:

They are indirect measuring tools used to measure or transfer linear dimensions. These are used with the help of a steel Rule to check inside and outside measurements. These are made of Case hardened mild steel or hardened and tempered low carbon steel. While using, but the legs of the calipers are set against the surface of the work, whether inside or outside and the distance between the legs is measured with the help of a scale and the same can be transferred to another desired place. These are specified by the length of the leg. In the case of outside caliper, the legs are bent inwards. Calipers are used for transferring measurements.

The three common types are:

- ✓Outside
- ✓inside and
- ✓odd-leg Hermaphrodite

You can use inside calipers as outside calipers by merely turning the legs about the hinge. and in the case of inside caliper, the legs bent outwards.

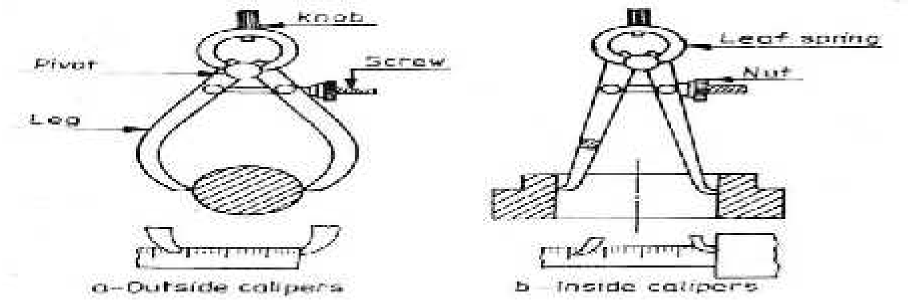


Fig. 4.12.Outside caliper and inside caliper..

Fig. 1.11 A. Firm joint outside caliper .B. Bow spring outside caliper C. Firm joint inside caliper D. Bow spring in side caliper. E. Hermaphrodite caliper

Protractor:

Engineer's Protractor

An engineer's protractor, as shown in Fig. 4.13, is a general purpose tool used for the measuring / checking angles, e.g. the angle of drill head, angle of cutting tool, and even for the marking out of angles on a component part.

Fig.1.12.Engineer's Protractor *Fig.1.13.Vernier Protractor*

Vernier Protractor:

This is a precision measuring tool that the accuracy of measurement can reach ±5 minutes of an angle through the vernier scale

Dial Indicator:

A dial indicator (dial gauge) can measure dimensions up to an accuracy of 0.01mm or even less. The principle of it is that the linear mechanical movement of the stylus is magnified and transferred to the rotation of pointer as shown in Fig. 12. It is usually used as a comparator for calibration or alignment of machine.

Fig 1.14.Dial Indicator

B. Laying out and laying out tools

Laying out

Layout is the process of making lines, circles, and other marks with a variety of hand tools to represent the features on the blueprint. These marks act as a reference for the operator during machining. They indicate features such as hole locations, dimensions, and other specific areas to be machined. The work piece is coated with a special layout dye, which helps the lines appear more clearly. Layout lines guide the operator to ensure the part fits within the acceptable tolerances. By preventing errors, layout reduces costs

Laying out tools

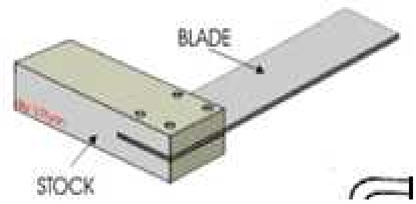
Squares

There are a number of squares:

- ✓ Try square
- ✓ Miter square
- ✓ Engineer's square.

Both the try square and engineers square are used to mark lines at 90° to an edge. A try square is used on timber and an engineer's square is used on metals. Both can be used for marking out plastics. You can also use try squares and engineer's squares to check that a cut or an edge has been made at right angles to another. Hold the stock part of the square tightly against the edge that you have just cut. If you can see light between the two edges then the cut is not square. A miter square is used for marking out 45° or 135° angles on wood and plastic. Take great care when using any form of square for marking out or checking, and ensure that it is being held firmly and tightly against the surfaces or edges of the material.

In practice, try is used for checking the squareness of many types of small works when extreme accuracy is not required .The blade of the try square is made of hardened steel and the stock of cast Iron or steel. The size of the try square is specified by the length of the blade.



Fi.1.15 Try square *fig.1.16: The uses of squares.*

- **Gauges:**

There are three basic types of gauge:

- ✓ marking gauge,
- ✓ mortise gauge
- ✓ cutting gauge.

A marking gauge is used for marking lines parallel to the face edge and side on wood. It consists of a stock that slides up and down the stem, allowing various measurements to be set. The gauge should be set using a steel rule that has a zero end. The spur (sharp point) is pushed into the wood as the gauge is pushed or pulled along the length of the timber. It is important to hold the stock tightly against the edge of the timber to ensure that you mark a parallel line.

A vernier height gauge is used for measuring height of an object or marking lines onto an object of given distance from a datum base.

A cutting gauge is used for cutting across the grain. It is used in the same way as a marking gauge, but has a blade instead of a spur. The blade cuts the fibers across the grain, making it easier and neater to cut with a saw.

A mortise gauge has two pins; one pin is fixed and the other is adjustable. It is used for marking two parallel lines where a mortise and tenon joint is to be cut. The process of marking out is exactly the same tise gauge as with the two other gauges.

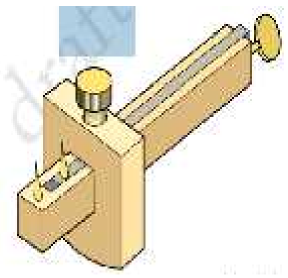


Fig.1.17 Mortise gauge

Fig.1.18.. Vernier Height Gauge

Scribers:

A scriber is used to scratch on the surface of metal and plastic lightly. If you are using a scriber on metal, it is a good idea to apply a coat of engineer's blue. This is a spirit-based liquid that is applied to a metal surface. When the scriber is dragged across the engineer's blue it leaves a clean line, which can be easily seen.

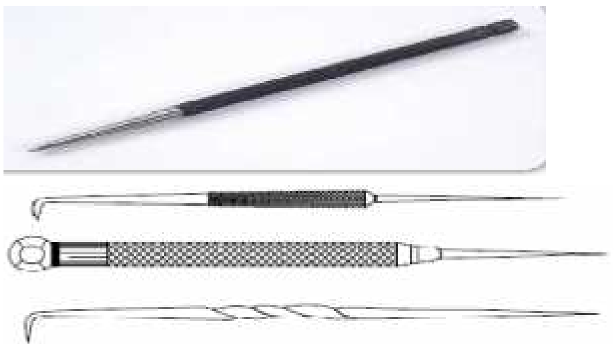


Fig1.19. Scribers

Punches:

Centre punches are used to make an indent in the surface where holes are to be drilled in metal, as shown in Figure 4.19. They provide a starting point for the drill and stop it skidding over the surface. Dot punches are used for marking the centers where dividers are to be used. They are similar to a center punch, except that the tips are ground to a 60° rather than a 90° point.

Center Punch Procedure

1. Make sure that the point of the punch is sharp before starting.
2. Hold the punch at a 45 degree angle and place the point carefully on the layout line.
3. Tilt the punch to a vertical position and strike it gently with a light hammer.
4. If the punch mark is not in the proper position, correct it as necessary.

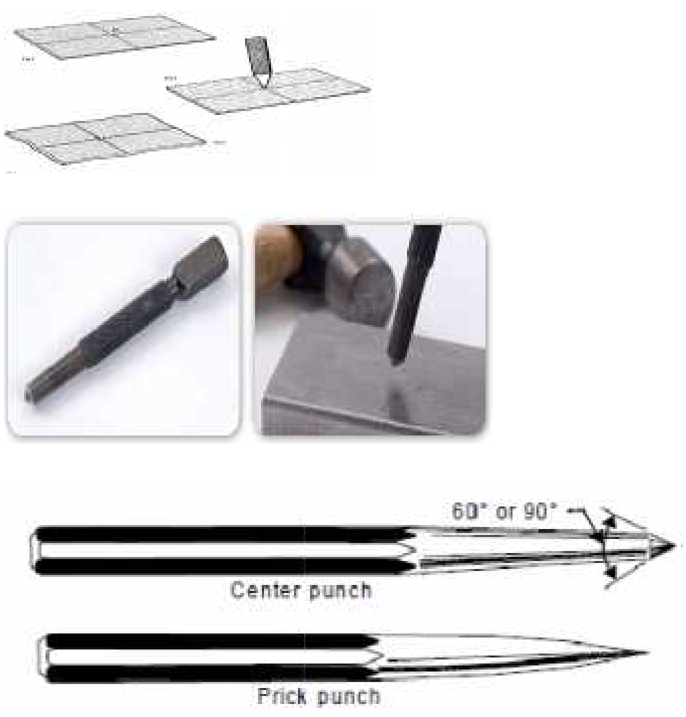
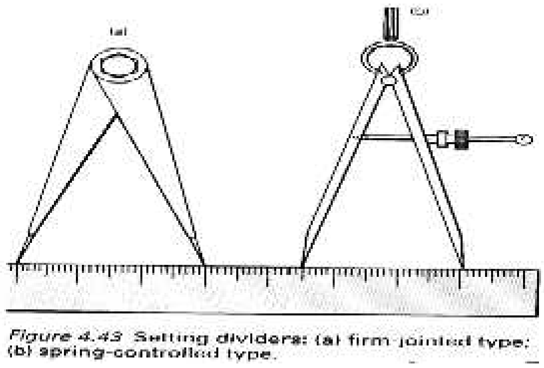


Fig.1.20. A Center punch and the punch in use.

.

Divider:

It is basically similar to the calipers except that its legs are kept straight and pointed at the measuring edge. This is used for marking circles, arcs laying out perpendicular lines, by setting
lines. It is made of case hardened mild steel or hardened and tempered low carbon steel. Its size is specified by the length of the leg**.**



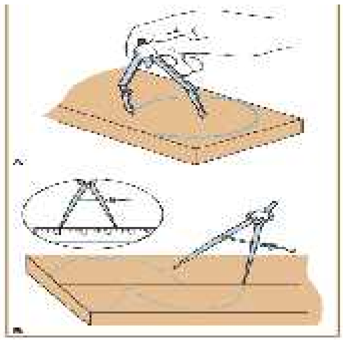
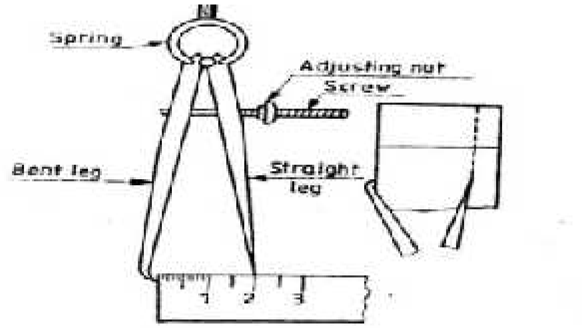


Fig.1.22.A-place and swing the compass on the center point of the circle or arc.

B-Use the Divider to Step off Measurement



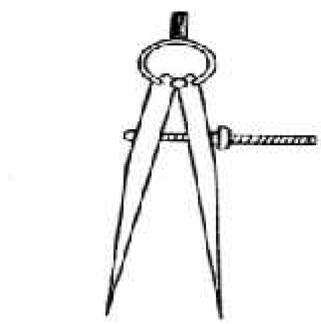
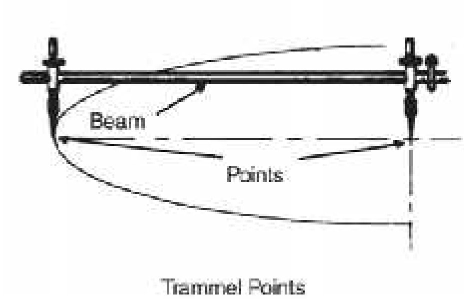


Fig.1.23.odd leg caliper and divider

Trammel Points:

Trammel points are used to mark large circles or make arcs that have large radii. They are used similarly to dividers. The beam is usually made of metal.



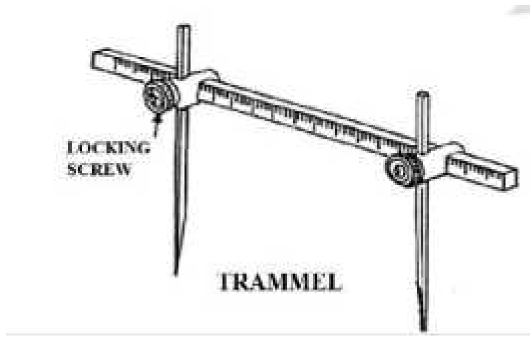


Fig.1.24. Trammel Points

Surface plate

The surface plate provides a 'plane of reference' for checking other surfaces. The plate is made from an iron casting. Its top surface is perfectly flat, and the underside is ribbed to prevent distortion of the plate (Figure 4.22). It is used generally for setting up work for marking out and testing. The surface table allows larger work to be checked for flatness.

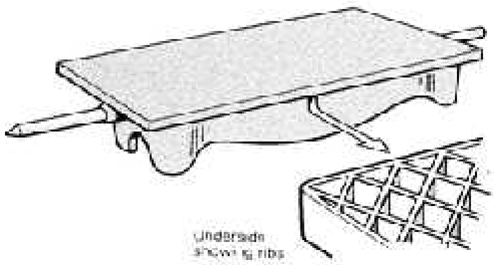


Fig.1.25.Surface plate

Surface gauge:

You use the surface gauge for marking parallel lines and finding centers. Figure 4.24 shows the procedure:

1. Set the scriber at the height you want, using the rule
2. Hold the work against an angle plate or on a vee block.
3. Move the block until the scriber touches the work.

Angle Plate:

The angle plate is made from a good-quality casting.

It consists of two faces machined at 90° to each other. You can fix work to the plate using bolts, which can pass through the slots provided. You can also use the angle plate to support work when you are using the surface gauge for marking out.

V-Blocks:

You use V-blocks to support cylindrical work when you are testing it or marking it out. The blocks, which are supplied in pairs, are made from cast iron. There are grooves along the sides, which allow the clamp to be used (Figure 4.27). If the work is long, you will need a 'matching pair

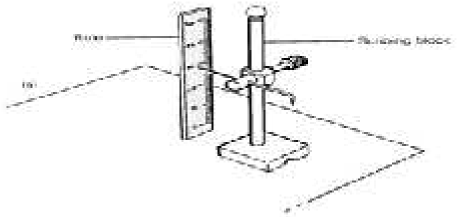


Fig.1.25. Surface gauge

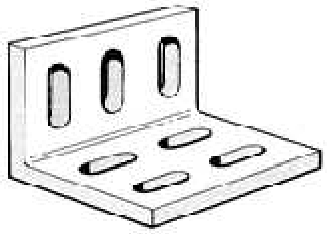


Fig.1.26. Angle plate

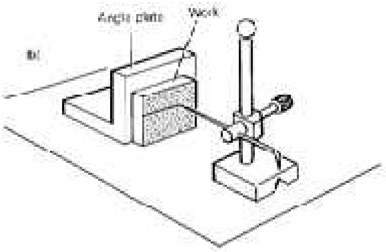


Fig1.27.Using the surface gauge
(a) Set the scriber; (b) Hold the work against angle plate and move the block against work.

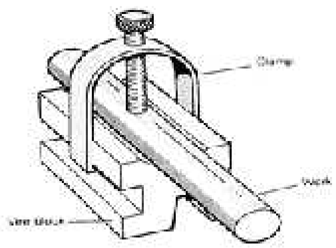


Fig1.29. Using the V-block

Combination Set:

The combination set is an important tool in the workshop, because you can use it as a center square, a try square, a protractor for marking out, measuring and testing. There are three heads (protractor, square and center), which slide onto a rule, which can be fixed at any position using the nut provided. The main parts of the combination set are used as follows.

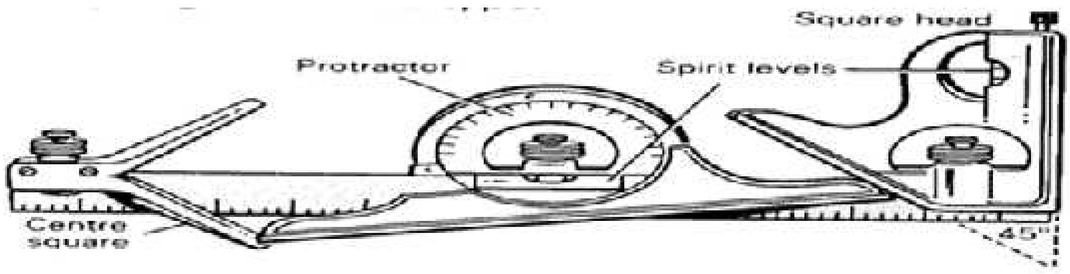


Fig.1.30.Combination set.

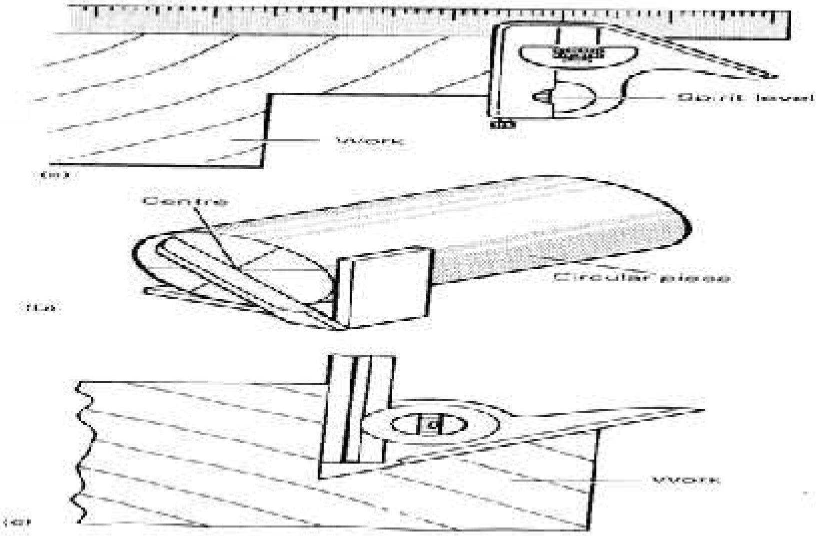


Fig.1.31.Using the combination set: (a) try square; (b) Center square; (c) protractor

Self-check-1

I. <u>Multiple choices</u>

Select one of the appropriate answers

.

1. Which one of the following is categorized under measuring tools?

A. Micro-meter

B. Centre punch

C. Surface plate

D. Scriber

2. A tool that is used for finding centers:

A. Surface plate

B. Angle plate

C. Surface gauge

D. Combination set

3. The statement that explains “Health and Safety” in terms of Occupational Health.

A. Ensuring that worker’s capabilities are matched with the respective jobs they are engaged in.

B. Ensuring that standards of wellbeing at the workplace are being maintained to prevent injuries and infections.*

C. Conducting surveillance of work environment that may have effect on the wellbeing of workers.

II. Give appropriate answers for the following questions below

1. What is Occupational Health?

2. ______is any source of potential damage, harm or adverse health effects on something or someone under certain conditions at work.

3._______ repetitive movements, improper set up of workstation,

4._______ is the chance or probability that a person will be harmed or experience an adverse health effect if exposed to a hazard.

5. How do you implement a safety management system?
6. What are the key elements of a safety management system

III. True/false

Write true for correct statement and false for wrong statement

1. Hazards do not affect the production system

2. Proper clothing can be considered as safe working

3. Workplace conditions may increase or decrease injuries

Unit Two: Set-up machine

This unit to provide you the necessary information regarding the following content coverage and topics:

- Inspect and sharpen tools.
- Mount, sett guards and position tools within machine
- Calculate speeds and feeds
- Perform setup operations

This guide will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Identify sharpen tools
- Adjust tools position
- Identify different types of accessories.
- Calculate speeds and feeds
- Perform setup operation with safety measures

2.1. Inspect and sharpen tools.

A. inspect tools

As always we should be aware of safety requirements and attempt to observe safety rules in order to eliminate serious injury to ourselves or others. Unguarded moving parts of machines/equipment and the sudden or uncontrolled release of their power systems can result in serious injuries.
Personnel working with machines must be aware of the risks involved and follow safe work practices.

Power tools should not be employed in wet environments and should never be dipped in water; they should be checked periodically for exposed wiring, damaged plugs, and loose plug pins. Nicked cords can be taped but if a cut appears to be deep, a cord should be replaced. Tools that are damaged or those that sound and feel different when used should be checked and repaired.

B. sharpen tools

The word sharpening is usually used for the final finishing of edge tools. Creating the initial shape often means that quite a lot of steel needs to be re-moved when for example, you change the point angle of a drill or you shape a broken or heavily worn drill. Once the geometry of the point is established, you maintain the sharpness by sharpening. Edge tools need to be sharp to work efficiently. The bevels of a sharp edge tool end in a uniform tip. After a period of use the tip becomes rounded and the edge is no longer sharp.

You can sharpen tools with a bench stone or, in the case of knives, with sharpening steel. This means that you work on the very tip of the bevel and the tool is sharp again. When sharpening with a steel or a bench stone, a very limited amount of steel is removed.

After several sharpening or honing, the edge angle becomes too wide and the tool must be re-shaped. Sooner or later all edge tools need to be re-shaped and this is done by grinding on a grindstone or a grinding wheel. When only a limited amount of steel is removed this operation is also called sharpening.

2.2. Mount and position tools within machine specifications

2.2.1. Mounting the cutting tools in machine tools

Proper positioning and orientation of the tool depending upon its

- type
- size and shape
- geometry

Concepts of clamping

Once work piece is located, it is necessary to press it against locating surfaces and hold it there against the force acting upon it. The tool designer refers to this action as clamping and the mechanisms used for this action are known as clamps. It is necessary that the work should be properly and securely held on for machining operations, a VISE is an effective work holding device.

2.2.2. Types of work holding devices

a. Bench vice

A bench vise is like an extra hand and is a common tool found in any shop or garage. It is attached to a workbench and its purpose is to hold material steady, allowing you to use both hands to work on the material with other tools. They are ideal for sawing, sanding, planing, drilling, screwing, soldering and more.

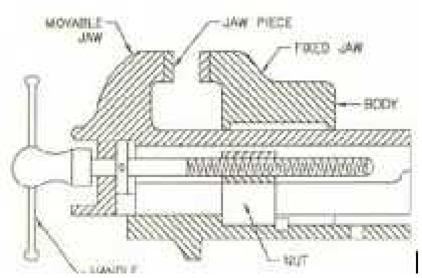
Bench Vises with Swivel Base

There are mainly three types of vises commonly used:

- Plain vise
- Swivel vise
- Tool makers universal vise

1. Removable hardened alloy steel jaw inserts.
2. Completely enclosed center screw.
3. Attractive hammered enamel finish

Fig.2.1. bench vice



b. C-Clamp

This is used to hold work against an angle plate or v-block or any other surface, when gripping is required. Its fixed jaw is shaped like English alphabet ‘C’ and the movable jaw is round in shape and directly fitted to the threaded screw at the end .The working principle of this clamp is the same as that of the bench vice.

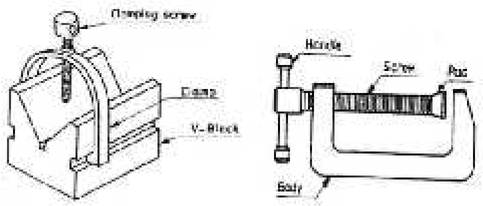


Fig. 2.2.C-clamp

c. V-block

V-block is rectangular or square block with a V-groove on one or both sides opposite to each other. The angle of the ‘V’ is usually 90^0. V-block with a clamp is used to hold cylindrical work securely, during layout of measurement, for measuring operations or for drilling for this the bar is faced longitudinally in the V-Groove and the screw of V-clamp is tightened. This grip the rod is firm with its axis parallel to the axis of the v-groove

d. Parallel Clamps

Jaws are made of hardened and tempered steel. Clamps are equipped with spring clips

Fig. 2.3.parallel clamp

Drill chucks are the most common devices used on a drill press for holding straight –shank cutting tools.

Drill sleeves are used to adapt the cutting tool shank to the machine spindle and if the taper on the cutting tool is smaller than the tapered hole in the spindle.

drill socket is used when the hole in the spindle of the drill press is too small for the taper shank of the drill.

2.2.3. Mounting a Work on milling machine

An efficient and positive method of holding work pieces to the milling machine table is essential if the machine tool is to be used to advantage. Regardless of the method used in holding, there are certain factors that should be observed in every case. The work piece must not be sprung in clamping; it must be secured to prevent it from springing or moving away from the cutter; and it must be so aligned that it may be correctly machined.

Milling machine worktables are provided with several T-slots, used either for clamping and locating the work piece itself or for mounting various holding devices and attachments. These T-slots extend the length of the table and are parallel to its line of travel. Most milling machine attachments, such as vises and index fixtures, have keys or tongues on the underside of their bases so that they may be located correctly in relation to the T-slots.

Methods of mounting work pieces

(1) Clamping a Work piece To the Table.

(2) Clasping a Work piece to the Angle Plate.

(3) Clamping Work pieces in Fixtures.

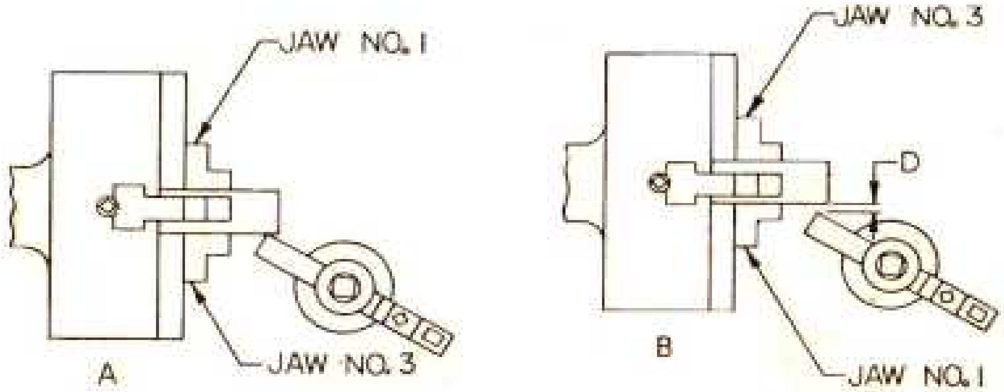
(4) Holding Work pieces Between Centers.

(5) Holding Work pieces in a Chuck.

(6) Holding Work pieces in the Vise

MOUNTING AND CENTERING OF WORKPIECE ON THE LATHE

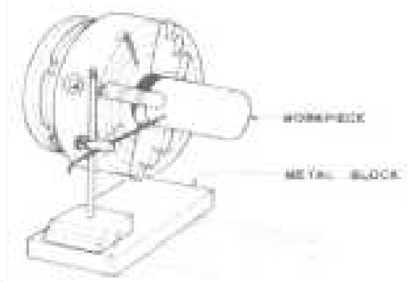
Mounting Work in a Four Jaw Independent Chuck
Visual Inspection Approach
Using High Spot – Low Spot Technique

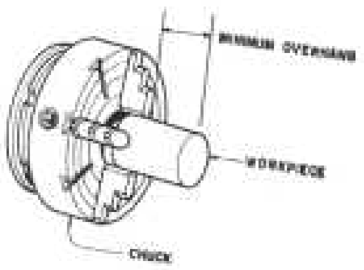


(A) High Spot is the highest point that touches the point of any centering devices.
(B) Low Spot is the gap created as it moves away from the workpiece
Illustration show the back of a loose tool holder as a means to determine equidistance

Visual Estimate Inspection Method

1. Chalk method /Surface gage /wriggler method/Alignment of Chuck's jaw



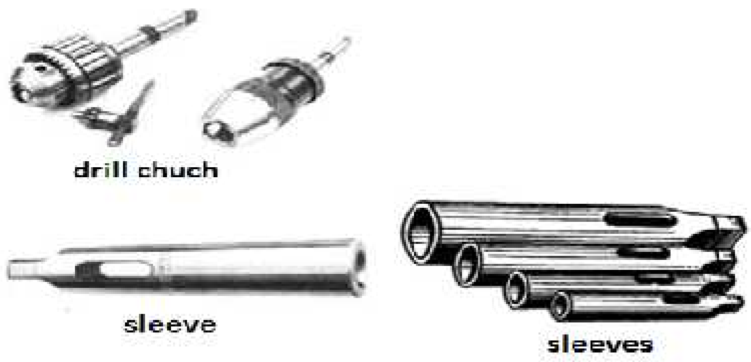


NOTE: Do visual inspection by;

1. Loosen up the jaw opposite of the high spot and screw tight the high spot's jaw to push the workpiece
2. Having plenty of light while using scratch white paper as background distinguisher between workpiece gap and the point of a centering devices
3. Repeating the procedures until an equal gap was visible in four front of the chuck's jaw. Tighten the high spot area if gap in the low spot equal 0.5mm.
4. Checking for wobble spin and remedied it by light tapping

2.2.4. Mounting of tools in Drilling machines

Small straight shank type solid HSS and carbide drill are held in a drill chuck which is fitted in the drill spindle at its taper bore.



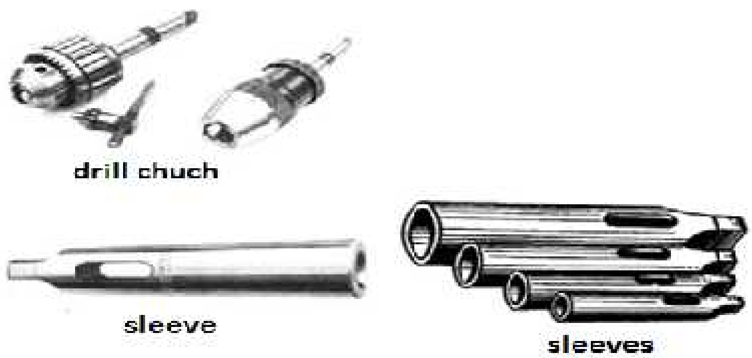


Fig. 2.4.drilling machine tools

Cutter holding devices in milling machines are:

- **Arbors** - used for mounting the milling cutter, are inserted and held in the main spindle by a draw bolter a special quick - change adapter.
- **Shell-end mill arbors** - It may fit in to the main spindle or the spindle of the vertical attachment which permit face milling to be done either horizontally or vertically.

Collet adapters - are used for mounting drills or other tapered - shank tools in the main spindle of the machine or the vertical milling attachment.

Arbor, Adapter,shell end mill arbor, sleeve

2.3 calculate speed and feed

2.3.1. Speed

A wide range of drills and drill sizes is used to cut various metals; an equally wide range of speeds is required for the drill to cut efficiently. For every job, there is the problem of choosing the drill speed which will result in the best production rates and the least amount of downtime for regrinding the drill. The recommended cutting speeds for drilling various types of materials may be found in the table shown below. The most economical drilling speed depends upon many variables such as:

- the type and hardness of the material
- the diameter and material of the drill
- the type and condition of the drill press
- the efficiency of the cutting fluid employed

To determine the correct number of r/min (revolution per minute) of a drill press spindle for a given size drill, the following should be known:

- ✓ the type of material to be drilled
- ✓ the recommended cutting speed of the material
- ✓ the type of material from which the drill is made

Table 2.1.Drill size and types of materials

Drill size	**stainless steel**	**tool steel**	**cast iron**	**machine steel**	**aluminum**
	1910	2865	3820	4775	9550
3	1275	1910	2545	3185	6365
4	955	1430	1910	2385	4775
5	765	1145	1530	1910	3820
6	635	955	1275	1590	3180
7	545	820	1090	1365	2730
8	475	715	955	1195	2390
9	425	635	850	1060	2120
10	350	520	695	870	1735
15	255	380	510	635	1275
20	190	285	380	475	955
25	150	230	305	380	765

CS/cutting speed/ = 12 CS = 18 CS = 24 CS = 30 CS = 60

- ✓ CS in m/min CS x 1000 CS x 320
- ✓ Formula r/min = --------- ⇒ ------- ⇒ -------------
- ✓ π x D in mm 3.14 x D/diameter/
- ✓ D For metric calculations, the formula is used:

$$rpm = \frac{CS(m)}{\Pi D(mm)}$$

It is necessary to convert the meters in numerator to millimeters so that both parts of the equation are in the same unit. To accomplish this, multiply the CS in meters per minute by 1000 to bring it to millimeters per minute.

$$rpm = \frac{CS \times 1000}{\Pi D}$$ This can be simplified to $$rpm = \frac{CS \times 320}{D}$$

This is done for the reason that not all machines have variable speed drives and therefore cannot be set to the exact calculated speed. Dividing 1000 by π (3.14) we arrive at the formula above. This formula is accurate enough for most drilling operations.

Example:

Calculate the rpm required to drill a 15mm hole in tool steel using a high speed steel drill.

Solution:

$$rpm = \frac{CS \times 320}{D} = \frac{18 \times 320}{15} = \frac{5760}{15} = 384$$

2.3.2. Feed

Feed is the distance that a drill advances into the work for each revolution.

In other words the feed of a drill is the distance the drill moves into the job at each revolution of the spindle. It is expressed in millimeter. The feed may also be expressed as feed per minute. The feed per minute may be defined as the axial distance moved by the drill into the work per minute.

The rate of feed is generally governed by:

- the diameter of the drill
- the material of the work piece the condition of the machine

Table2.2.Drill size and feed per revolution.

Drill size	feed per revolution	
1 to 3		0.02 to 0.05
3 to 6		0.05 to 0.10
6 to 13		0.10 o 0.18
13 to 25		0.18 o 0.38

2.4. Perform setup operations

2.4.1. Sequence of operations

Sequencing refers to the order in which activities occur in the operations process. But there are a lot of operations are performed by different machines such as;

Operation on lathe machine Facing, Turning, Chamfering, Grooving, Forming, Knurling, Undercutting, Eccentric turning, Taper turning, Thread cutting, Drilling, Reaming, Boring, Tapping and etc. these operation are depend on one another.

Common lathe operations:

1. Facing
2. Plain turning
3. Step turning
4. Drilling
5. Boring
6. Reaming
7. Under cutting or grooving
8. Threading
9. Knurling
10. Forming
11. Taper turning

Self check-2

I. <u>True/ false</u>

Write true for correct statement and false for wrong statement

1. Types of materials may have effect on cutting speed
2. Proper work clamping results smooth operations
3. Work holding devices are the same for all machines

Directions: choose the correct answer for the following questions (**2 points each**):

1. Which one of the following is not clamping tools?
   a. Vice
   b. V- block
   c. hacksaw
   d. C- clamp
2. One of the following is not the parts of bench vice.
   a. Jaw face
   b. Thimble
   c. Fixed jaw
   d. Movable jaw
3. ______is used for clamping work piece.
   a. Bench vice
   b. C-clamp
   c. V-block
   d. All

III. <u>Matching</u>

Match group A within group B

<u>A</u>	<u>B</u>
1. Lathe operation	A. parallel clamp
2. Hand tool	B. Chisel
3. Work holding device	C. facing
4. Marking tool	D. scriber
	E. Grinding

Unit three: Perform hand tool operations

This learning guide is developed to provide you the necessary information regarding the following content coverage and topics:

.

- Clamp Work pieces
- Select and use Hand tools
- Cut, chip, file and scrap Work pieces
- Cut Threads
- Perform bench work operations
- Recognize and report hazards.

This guide will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Clamp Work pieces based on applied standards.
- Select and using Hand tools
- Cut, chip, file and scrap Work pieces
- Cut Threads
- Perform bench work operations by applying safety.
- Recognize and report hazards.

3.1. Clamp Work pieces

V-block is rectangular or square block with a V-groove on one or both sides opposite to each other. The angle of the ‘V’ is usually 900. V-block with a clamp is used to hold cylindrical work securely, during layout of measurement, for measuring operations or for drilling for this the bar is faced longitudinally in the V-Groove and the screw of V-clamp is tightened. This grip the rod is firm with its axis parallel to the axis of the v-groove

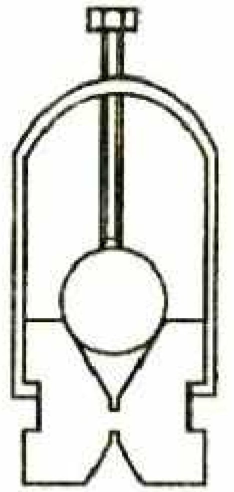


Fig.3.1. V-Block

A. Select and use Hand tools

The bench work and fitting plays an important role in engineering. Although in today's industries most of the work is done by automatic machines which produces the jobs with good accuracy but still it (job) requires some hand operations called fitting operations. The person working in the fitting shop is called fitter

Tools

Tools can be divided into two main groups: hand tools and power tools. Hand tools are operated by the physical strength of the user. Power tools require an external source of power such as electricity or compressed air to operate. Each of these groups can also be divided into sub groups

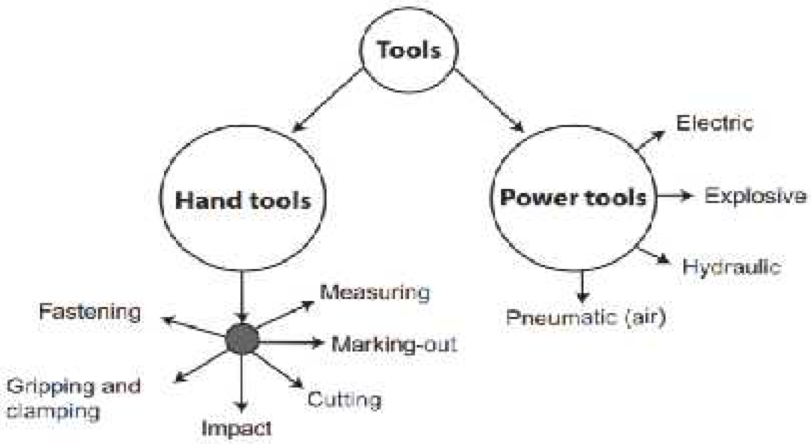


Fig.3.2. Division of tools

3.2. Select and use hand tools

3.2.1. Hand tools

Hand tools have been devised to enable trades people to carry out a job more efficiently, quickly and safely than would otherwise be possible. Some tools are quite simple, such as a screwdriver, which is almost indispensable for undoing a countersunk screw located in a recess. Others are more complicated, such as a micrometer; these are indispensable are when measuring fine tolerances.

Hand tools can be classified into several groups:

- fastening tools
- gripping and clamping tools
- impact tools
- cutting tools
- marking-out tools
- Measuring tools.

a. Fastening tools

Spanners

There are several types of spanners, each suitable for a specific job. The most commonly used spanners in a vehicle body building workshop are:

- ✓ open-ended spanners
- ✓ ring spanners
- ✓ combination spanners
- ✓ sockets and their accessories
- ✓ hook spanners
- ✓ pin spanners
- ✓ adjustable spanners
- ✓ ratchet spanners
- ✓ Flare nut spanners.

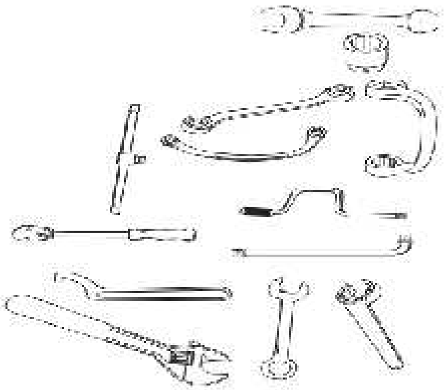


Fig.3.3. Types of spanners

Wrenches

Wrenches are tools used for holding and turning. A variety of wrenches are used in the vehicle body building industry. Adjustable pipe wrenches are sometime called still son wrenches. Typically, they are used on cylindrical objects such as pipes and rails where there are no flats on which to use a spanner. Another type of wrench is the hexagon wrench, which would typically be used for undoing hexagonal recessed drain plugs. Smaller hexagon wrenches are called Allen keys. Torque wrenches are used to tighten nuts or bolts to a specific tension and are sometimes called tension wrenches. They are used to correctly tension down the bolts holding engineering components such as power take-off units or bolts on kingpins. Never use a torque wrench to undo nuts or bolts, as this may damage or alter the accuracy of the wrench.

Fig.3.4. Torque wrench

Screwdrivers

Screwdrivers are accurate precision tools and are not designed to be used as punches or cold chisels. The most common types of screwdrivers are the standard (straight blade or flat) type and the Phillips screwdriver, and they are available in many different sizes and lengths. There are also special screwdrivers designed for hard-to-get screws, for example right-angle screwdrivers.

Fig.3.5. Flat screw driver

Keys

Keys used in the trade include drill chuck keys, lathe chuck keys and hexagon keys. Hexagon keys are also called Allen keys and include ball driver keys which can be used at an angle, unlike straight hexagon keys, which must be inserted squarely into the hexagonal recess

3.3. Cut, chip, file and scrap Work pieces

3.3.1. Concepts of Cutting metals

Sawing is the process of cutting metal stock that is impractical to use a file, a chisel or a machine with a multi-point cutting tool called a hand hack saw. A hand hack saw can also be used for cutting off a jammed bolt, pipes, tubing and rods for special or custom fittings on the job (on the field work).

Some of the most common tools used to cut metals are **hacksaws**, **band saws**, **cold chisels**, **bolt cutters, tin snips, and abrasive saws**. Large stock is sawed, while bar stock is either sawed or cut with a cold chisel. Sheet metal is usually cut with metal snips. In fabrication facilities, large amounts of metal are cut with horizontal band saws or metal shears, commonly called “ironworkers. Layout tools are used to measure and mark metal stock before cutting, shaping, and doing other types of work with cold metal.

A. Saws

Saws are used to cut material that is not needed away from material which is. Saw blades have alternate teeth bent out or 'set' in opposite directions. This is so that when they cut, they make a gap, called the kerfs. The kerfs must be wider than the saw blade so that the blade cannot get stuck. When using a saw, you should always cut to the waste side of the marked line so that you leave a small amount for finishing by either sanding or filing. Whatever you are cutting, it is important to keep as many teeth in contact with the piece being cut as possible. You should choose the correct saw for the type of material you are using. Table 2.3 on the next page shows the most common types of saws used in school workshops.

Cutting process

The saw is moved from the right to left and shows how the chip is formed. The cutting process is the result of the horizontal cutting direction and the pressure on the work piece. The angle of the teeth enables the saw to cut the material effectively. The teeth are set (bent out) that they do not get jammed in the cut.

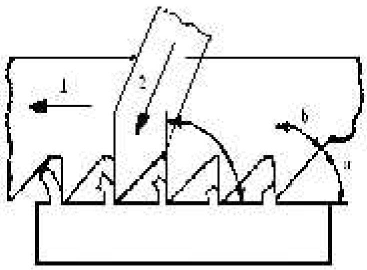


Fig.3.6. Sawing

Chipping: Chipping is the process of removing thick layers of metal by means of cold chisels. In chipping job is held firmly in vice and the metal is removed by striking the chisel on the work piece by a hammer.

B. Chisels

Types of chisels

These are sometimes referred to as cold chisels because they are used to cut cold metals. They are made of cast steel or alloy steel, with a hardened and tempered cutting edge.

The common types of chisel (Figure 3.8) include:

1. **The flat chisel**: used for general-purpose chiseling;

2. **The cross-cut chisel**: used for cutting grooves such as keyways, and for chipping;

3. **The half-round-nosed chisel**: used for cutting grooves (which are either curved or half-round);

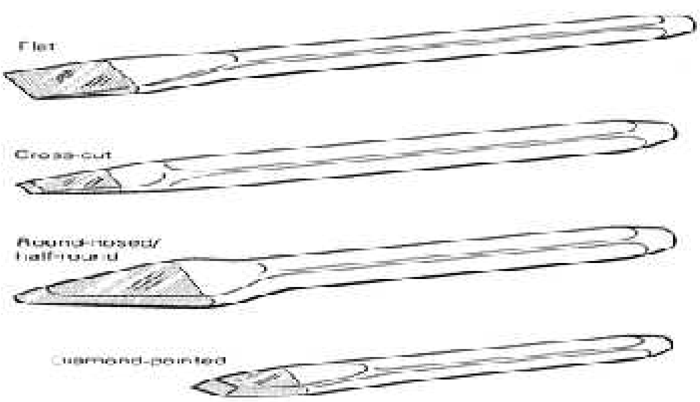


Fig.3.7. Common types of chisel

Filing is an important operation. It is carried out as an after treatment and done after chipping. It serves to remove the burr from the cut and clean the face of the cuts, and to finish the final shape of a work piece. Files can be classified, according to size, cut of teeth, sectional shape.

The Main Parts

A file is a hand cutting tool made of high-carbon steel, having a series of teeth cut on the body by parallel chisel cuts. The parts of a file are shown in fig Files are used to remove surplus metal and to produce finished surfaces.

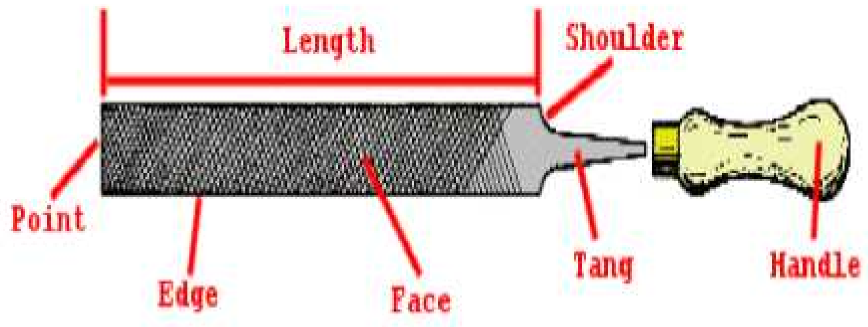


Fig.3.8.Flat file and its parts

Table3.1. grades of file cut

Cut	Typical use
Rough	Filing soft metals, plastics
Bastard	Shaping steel and fettling iron castings
Second cut	Generally used for harder metals and for good finish
Smooth cut	Draw filing and finishing hard metals
Dead smooth	Accurate filing with a high finish

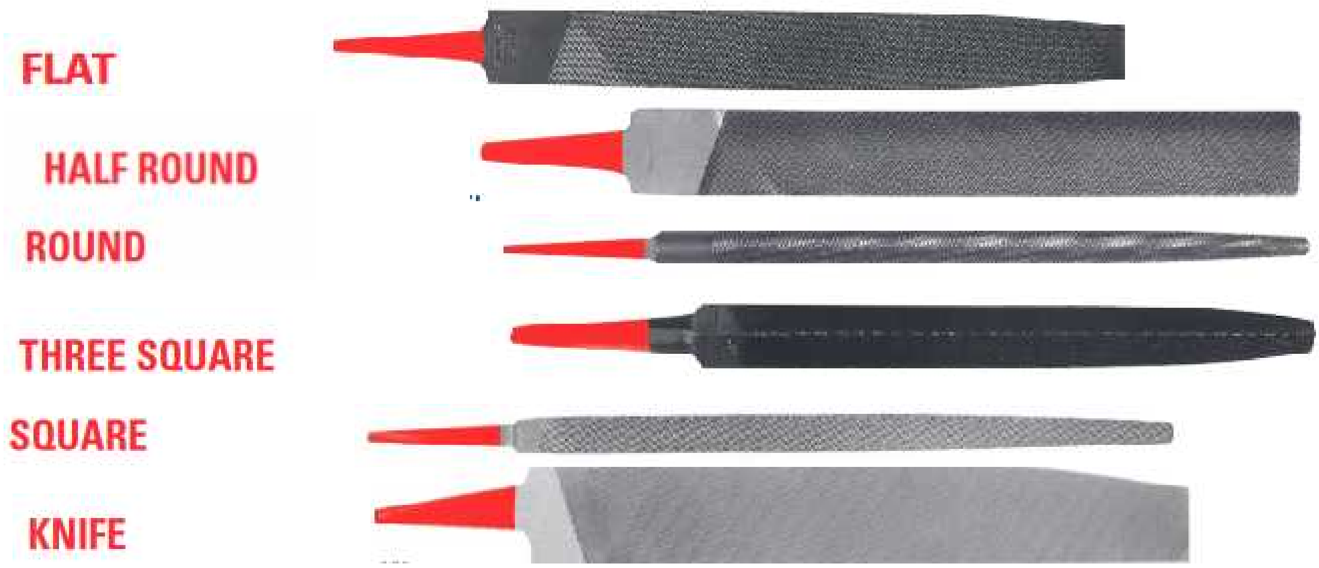


Fig.3.9. Types of file based on shapes.

Scraping

Scraping is the process of removing high spots on the surface of a piece of work. It is a difficult operation, and is not often performed in the school workshop.

Types of scrappers:

These sharp edged tools are used to remove uneven spots on the surfaces. They are of different shapes.

a. **Flat scraper**

It is used for removing metal from flat surfaces. The blade must have a slight curvature at the cutting edge. The corners are rounded to help the user, scrape at the exact spots.

b. Half round bearing scraper

This is used for scraping curved and cylindrical surface split bearings, big bush bearings etc.

c. Triangular scraper

This is used for scraping curved surfaces, holes and bores. Specification is by length. Example: 200 mm, 300 mm etc.

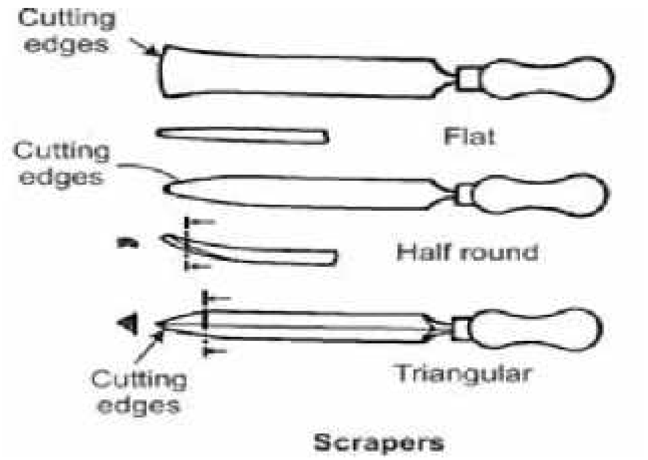


Fig.3.10. Common types of scraper

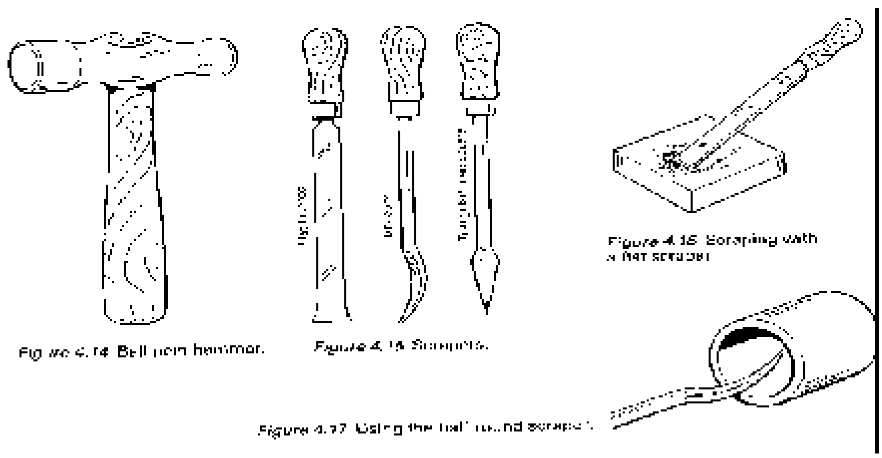


Fig.3.11. Using half round scraper

3.4. cut threas

Threads may be cut internally using a tap externally using a die. The proper selection and use of these threading tools is an important phase of machine shop work.

a. Dies

Dies are made either of high-carbon steel or of high-speed steel. Unlike taps, dies are used for cutting external (male) threads. There are three types (Figure blow). The circular split die is a circular piece with a split across one of the flutes. The split is provided to enable small adjustments to be made, using three set screws in the stock. The half die comprises two loose pieces, which are held in the stock. There is a small screw on the stock for adjustment. The die nut has a hexagonal body. This type, strictly speaking, does not cut new threads but is used to 'clean up' threads that are damaged.

b. Taps and dies

Screwing is a temporary method of fastening parts together. Methods for cutting screw threads include the use of the centre lathe. For bench work, however, taps and dies are used.

c. Taps

These are the tools used for cutting internal (female) threads. They are made of high-carbon steel or high-speed steel. The tap has a shank with a square end to take the tap wrench or holder. The shank is smaller than the threaded portion. The tap has four rows of threads, cutting edges or teeth, which suit a particular thread form. They perform the cutting action. The grooves between the cutting edges are called flutes. They allow waste material (chippings) to escape. They also allow cutting oil into the work.

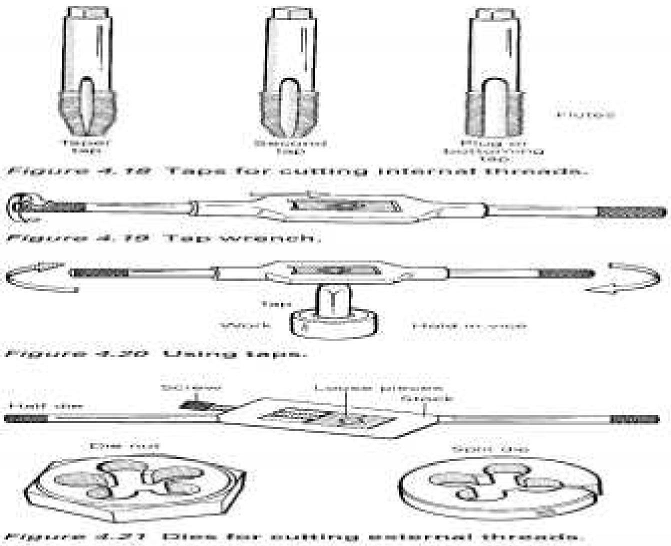


Fig.3.12. Taps and dies

The procedure for cutting external threads is as follows:

1. Square the end of the work and chamfer it (using a file, grinding machine or centre lathe) for an easy start.
2. Grip the die, held in the stock, firmly and squarely on the work.
3. Turn clockwise, about a quarter-turn, and ease back to remove chippings.
4. Apply a good supply of oil (lubricant).
5. Make adjustments of the screws after making a full cut until the depth required is achieved.

Care and maintenance is important:

1. Do not use either the stock or the die as a hammer; the threads may be broken.
2. Remove the die from the stock after every thread cutting, clean them and pack them into their boxes.
3. Use plenty of oil during cutting to reduce friction.

Tap Drill Size

Before a tap is used, the hole must be drilled to the correct tap drill size. This is the drill size that would leave the proper amount of material in the hole for a tap to cut a thread. When a chart is not available, the tap drill size for the ISO (International Standards Organization) thread can be found easily by applying this simple formula:

$$\boxed{TDS = M - P}$$

TDS = tap drill size

M = metric diameter of the tap

P = pitch of the thread in millimeters

Hand Tap

A tap is a cutting tool used to cut internal threads. Normally it's made of high-speed steel (HSS). Hand taps are usually made in sets of three, because it is better to distribute all the cutting work during the thread−process to three taps.

1 (taper) tap: 1 ring on shank

2 (plug) tap: 2 rings on shank

3 (bottoming) tap: without ring

The most common taps have two or three flutes in order to form the cutting edges, transport the chips out of the hole and give way for the lubricant. The end of the tap is square so that a tap wrench can be used to turn it into a hole.

Tapping a Hole

Before a tap is used, a hole must be **drilled** in the work piece to the correct tap drill size. The tap drill size (T.D.S.) is the size of the drill that should be used to leave the proper amount of material in the hole for a tap to cut threads. Then **countersink** both sides of the hole.

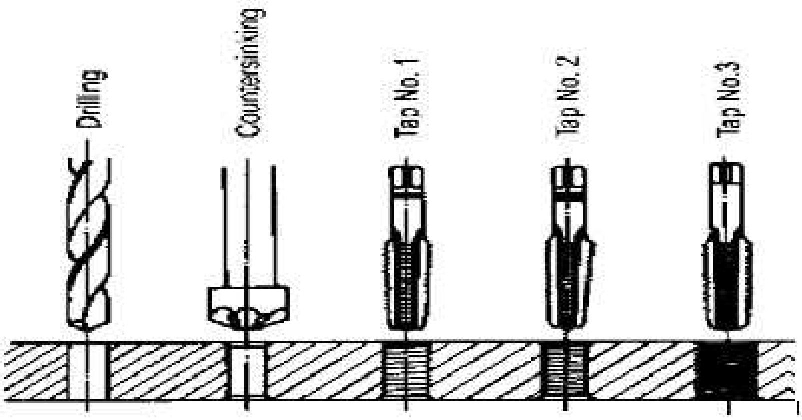


Fig.3.13. Drill, Countersink and tapping a hole

Working Steps for Hand Tapping

1. Select the correct size and type of tap for the job (blind hole or through hole).
2. Select the correct tap wrench for the size being used.
3. Use a suitable cutting fluid (No cutting fluid for brass or cast iron).
4. Place the tap in the hole as near to **vertical** as possible.
5. Apply equal down pressure on both handles, and turn the tap clockwise (for right-hand thread) for about two turns.
6. Remove the tap wrench and check the tap for squareness. Check at two positions 90 degree to each other.
7. If the tap has not entered squarely, remove it from the hole and restart it by applying slight pressure in the direction from which the tap leans. Be careful not to exert too much pressure in the straightening process, otherwise the tap may be broken.
8. Turn the tap clockwise one−half turn and then turn it backward about one−quarter of a turn to break the chip. This must be done with a steady motion to avoid breaking the tap.

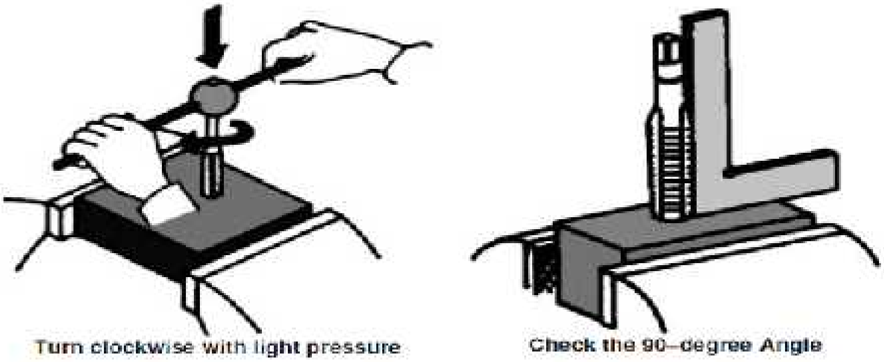


Fig.3.14.Tapping operation

Threading Dies

A threading die is used to cut external threads on round work pieces. The most common threading dies are the adjustable and solid types. The round adjustable die is split on one side and can be adjusted to cut slightly over or undersized threads. It is mounted in a die stock, which has two handles for turning the dies onto the work. The solid die, cannot be adjusted and generally used for recutting damaged or oversized threads. Solid dies are turned onto the thread with a **special** diestock, or adjustable wrench.

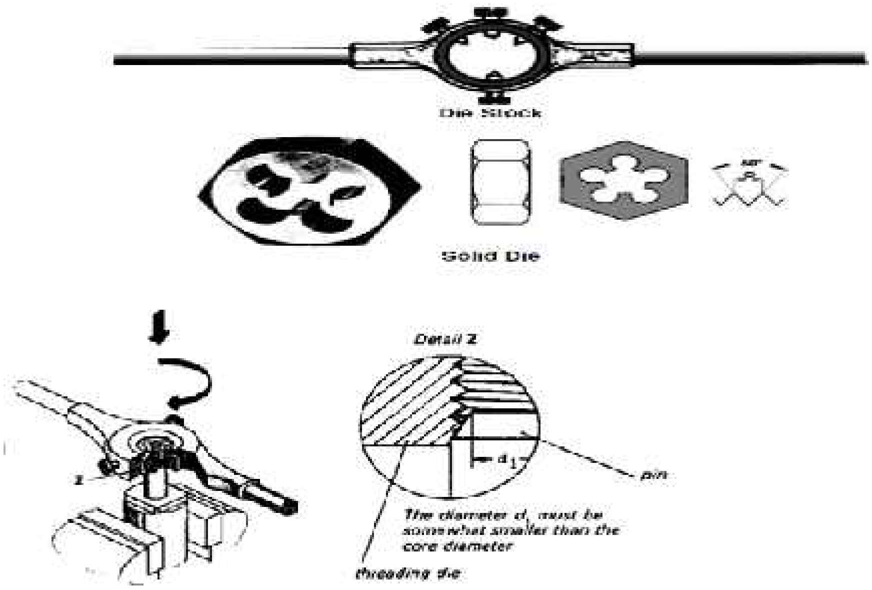


Fig.3.15. Die and its operation

Thread with a Hand Die Working Steps

The threading process requires the machinist to work carefully to produce usable parts and avoid damage. The following describes the procedure to be used.

1. Chamfer the end of the work piece with a file or on the grinder. Consider that a 3/4” thread requires a bolt with an outside diameter of 3/4”.
2. Fasten the work piece securely in a vise. Hold small diameter work short to prevent it from bending.
3. Select the proper die and die stock.
4. Lubricate the tapered end of the die with a suitable cutting lubricant.
5. Place the tapered end of the die squarely on the work piece.
6. Apply down pressure on both die stock handles and turn clockwise several turns.
7. Check the die to see if it has started squarely with the work.

8. If it is not square, remove the die from the work piece and restart it squarely, applying slight pressure while the die is being turned.
9. Turn the die forward one turn, and then reverse it approximately one half of a turn to break the chip.
10. Apply cutting fluid frequently during the threading process.

Metric Threads

These threads are identified by the letter „M“, the nominal diameter, and the pitch. For example, a metric thread with an outside diameter of 5mm and a pitch of 0.8mm would be identified as follows: M5x0.80

Table3.3. Pitch of metric thread

nom. dia.	pitch	nom. dia.	pitch
M3	0.50	M9	1.50
M4	0.70	M10	1.75
M5	0.80	M11	2.00
M6	1.00	M12	2.50
M8	1.25	M13	3.00

3.5 Perform bench work operations

The term bench work refers to the production of components by hand on the bench, where as fitting deals which the assembly of mating parts, through removal of metal, to obtain the required fit. Both the bench work and fitting requires the use of number of simple hand tools and considerable manual efforts.

Bench work and fitting have important roles to play to complete the production of our articles by hand on the bench. Fitting if the assembling to gather of parts and remove metal to secure the necessary fits and may or may not be carry out at the bench .The working of metals is made possible by the manipulation of tools and machines. Hand tools are the basic tools normally used in the school and college workshop to realize designed artifacts. The first part of this chapter covers bench work tools such as files, saws and chisels; the second part deals with tools for marking out, measurement and inspection.

The common types of operations of bench work are:

- Marking out
- Sawing
- Chipping
- Filing
- Scraping
- Grinding
- Drilling
- Reaming
- Tapping
- dieing

3.6. Recognize and report hazards

The safety in Workshops has been written not only to provide appropriate safety procedures but also to assist trained workshop personnel with the provision of a reference document outlining the general principles of safe working practices relevant to the mechanical engineering aspects. It relates to specific areas where definite safety measures are required for workshop operations.

General safety rules are established for three good reasons:-

- ✓ To protect you and your colleagues from badly harm.
- ✓ To minimize damage to facilities, machinery and tools with which you work.
- ✓ To enable you experience a positive safety attitude not only in the work shop but also in your entire professional career.

Safety is not only the responsibility of a single fellow. It is the responsibility of every one.

a.. **Classification of Safety**

1. Personal safety:

- Wear approved safety glasses or goggles at all times.
- Wear approved foot wears at all times.
- Remove all rings, Watches, or bracelets.
- Long hair must be protected by a hair net or an approved protective shop cap.
- Avoid horse play at all times, since an accidental slip or fall can cause a serious cut or body injury.
- Never handle sharp tools or cutters by hand.
- Use proper lifting techniques whenever lifting tools or machines.
- Do not carry sharp tools on pockets.
- Remove all rings, Watches, or bracelets.
- Don’t wear loose clothes.
- Do not use a file without handle.

2. Bench work and fitting shop safety:

- Keep hands and tools wiped clean and free of dirt, oil and grease.
- Always keep the work shop clean.
- Do no keep working tools at the edge of the table.
- Keep the floor free of oil and grease
- While sawing, keep the blade straight; otherwise it will break
- Clean the vice after use.
- Keep the floor around a machine or bench free of tools or stock.
- Sweep up the metal chips on the floor frequently.

3. **Machine and tool safety**

- Do no keep working tools at the edge of the table.
- Never place tools or materials on machine tools.
- Always keep the machine clean.
- Always stop a machine before attempting to clean it.
- Do not use vice as an anvil.

b. Cause of accidents

1. improper dressing
2. poor house keeping
3. insensible behavior
4. incorrect procedure

c. Preventing accidents

1. proper dressing
2. good house keeping
3. sensible behavior
4. correct procedure

Self check-3

I. Multiple choice

Directions: Choose the best answer for the following questions.

1. Which one of the following is not a bench work and fitting shop safety?
   a. Do not carry sharp tools in your pockets   c. keep your-self from car accident
   b. Clean the vice after use   d. do not use a file without handle
2. What is the reason that general safety rules are established for?
   a. To protect you and your colleagues from badly harm
   b. To decrease accidents in the work shop
   c. To minimize damage of facilities, machinery and tools   d. All
3. Which one of the following is the cause of accident?
   a. Improper dressing   c. Sensible behavior
   b. Good housekeeping   d. correct procedure
4. Which one of the following is used for cutting external threads?
   a. Tap   b. drill bit   c. Die   d. saw

5 Which one of the following is a personal safety?
   a. Remove all rings, watches, bracelets   c. Wear approved safety glasses or goggles
   b. Wear approved foot wears at all times   d. All

II. True/false

Write true if the statement is correct and false for statement

1. External threads are produced by taps
2. Thread making taps and dies have similar functions
3. Chipping is the process of removing metals by file
4. Vice can be used as anvil

III. Matching

Match group A within group B

A	**B**
1. Operation on bench work	A. taps
2. Male threads	B. external thread
3. Chipping	C. sawing
4. Thread cutting tool	D. chisel
5. To protect hands	E. gloves
	F. file
	G. apron

IV. Give answers for the given questions below

1. Write the difference between filing and file
2. Write some of operations can be performed on bench work

Operation sheet 3.1: thread cutting

Operation title: cutting external thread

Purpose: To produce external thread

Instruction: Using the figure below and given equipments cut external thread on the metal. You have given 30 minute for the task.

Quality Criteria: the pitch is measured within the standard

Precautions: follow all safety rules

Tools and requirement:

1. Dies
2. Die holder
3. Coolant
4. Work holding devices
5. Personal protective equipment

Steps in doing the task

1. Prepare the metal to the required dimension
2. Hold the work properly in the vice
3. Select the appropriate die
4. Continue cutting process
5. Add the necessary coolant
6. Complete the work
7. Check the pitch

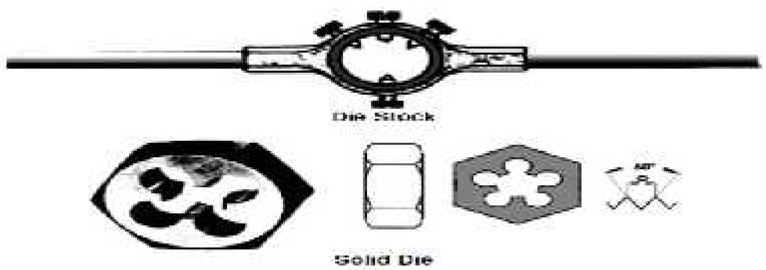


Figure: tools for thread cutting

Lap Test-3

Task-1: Perform measurement using measuring tools

Task-2: mark the object using marking tools

Task-3: adjust work holding devices

Task-4: check to the standard

Unit four: basic drill, ream and hone operations

This learning guide is developed to provide you the necessary information regarding the following content coverage and topics:

- preventative OHS
- Drill, bore, ream and hone holes

This guide will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Avoid hazards in drilling ,reaming and honing
- Produce holes using drill machine
- Identify types of operation on drill
- Identify types of drilling machine

4.1. Preventative OHS

A drill is a tool fitted with a cutting tool attachment or driving tool attachment, usually a drill bit or driver bit, used for drilling holes in various materials or fastening various materials together with the use of fasteners. The attachment is gripped by a chuck at one end of the drill and rotated while pressed against the target material. The tip, and sometimes edges, of the cutting tool does the work of cutting into the target material.

Further, the misuse of, or interference with, equipment provided by an employer for health and safety purposes is a *criminal offence*. It is up to all workers to develop a sense of *safety awareness* by following the example set by their instructors.

Regrettably not all older workers observe the safety regulations as closely as they should. Take care that you choose for your ‘role model’

The basic requirement for safe working is to operational procedures

Operational procedures:

1. Dress grinding wheel with dressing tool.
2. Hold drill bit against face of wheel at 59^0 angle on cutting lip.
3. Carry drill bit up the wheel face by dropping end and rotating very slightly in a clockwise direction.
4. Make slow deliberate strokes, the full width of the cutting lip.
5. Do not lower cutting lip below the horizontal position as this will round the cutting edge.
6. When one lip is ground, rotate the drill one half turn and grind the other lip.
7. Use tool gauge to check equal lengths of lips, 590 angle cutting lip and 12-150 lip clearance.
8. Test bit by boring hole in mild steel plate.
9. Stop while drilling, turn drill press in reverse direction to release drill bit from hole.
10. Make grinding corrections on drill bit as indicated by hole.
11. Submit drill bit and metal for evaluation.

4.2. Drill, bore, ream and hone holes

Drilling is a process of producing round holes in a solid material or enlarging existing holes with the use of multi tooth cutting tools called drills or drill bits. In other words it is the process of making holes of cylindrical shape on metals and other materials using drill bits and drilling machines.Drilling machines are driven either manual or by electrical power.

A. Classification of drilling machines

Drilling machines are classified into hand and breast drill, portable electrical drill, bench drill, pillar dills and others.

1. **The hand and breast drill:** are driven by hand and are commonly used where electricity is unavailable and are used for light work
2. **Portable electrical drill:** are most suitable to work which cannot be done with bench drill.
3. **Bench drill:** is one of the most common used machines in the work shop. This machine has the following parts. The base, the column, the head, the spindle, the pulleys, the motor, the belt, the safety swatch, the feed handle, the depth gauge, the head locking handle, the gear lever, the collar, the chuck and the main switch.
4. **Pillar dills:** is similar in design to the bench drill. But it is floor mounted and usually much large.

Various cutting tools are available for drilling, but the most common is the *twist drill*.

B. operations on drilling machines are:

1. Countersinking – is the operation of producing a tapered or cone shaped enlargement to the end of the hole.

2. Reaming is the operation of sizing and producing a smooth round hole from a previously drilled or bored hole with the use of a cutting tool having several cutting edges.

3. Boring is the operation of enlarging and truing a hole by means of a single-point cutting tool which is usually held in a boring bar.

4. Spot-facing is the operation of smoothing and squaring the surface around a hole to provide a seat for the head of a cap screw or a nut. For the spot facing operation, the work being machined should be securely clamped and the machine set approximately ¼ of the drilling speed.

Spot facing is a process of machining a flat surface around the mouth of a hole in order to provide a flat seat for the head of a bolt or a nut.

5. Tapping is the operation of cutting internal threads in a hole with a cutting tool called a tap. Special machine or gun taps are used with a tapping attachment when this operation is performed by power in a machine.

6. Counter boring is the operation of enlarging the top of a previously drilled hole to a given depth to provide a square shoulder for the head of a bolt or a cap screw. Counter boring is used to form a flat, recessed seating for a cheese head bolt or cap screw.

Sensitive Drill Presses

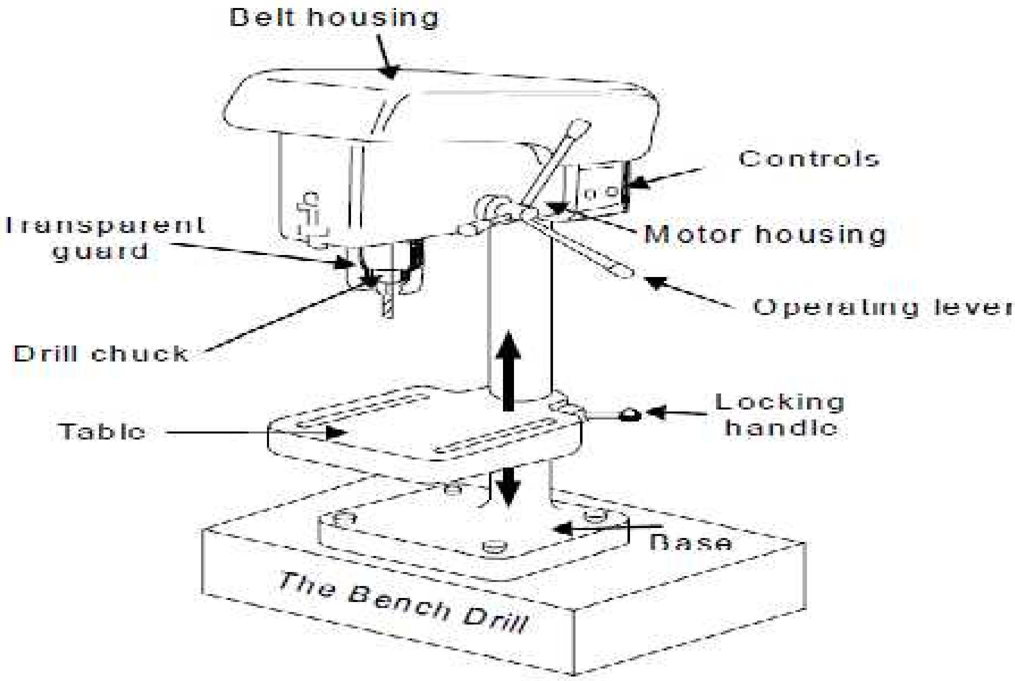


Fig4.1. Parts of drill Presses.

C. **Drill Bits**

Twist drills are end-cutting tools used to produce holes in most types of material. On standard drills, two helical grooves, or flutes, are cut lengthwise around the body of the drill. They provide cutting edges and space for the cuttings to escape in the drilling process. Since drills are one of the most efficient tools, it is necessary to know the main parts, how to sharpen the cutting edges, and the correct speeds and feeds for drilling various metals in order to use them most efficiently and prolong their life.

D. Parts of Twist drills

Shank

Most twist drills used in machine shop work today are made of high-speed steel. High-speed drills have replaced carbon-steel drills since they can be operated at double the cutting speed and the cutting edge lasts longer. A drill may be divided into three main parts: the shank, the body and the point.

- Generally drills up to 13mm in diameter have straight drill shanks, while those over this diameter usually have tapered shanks. Straight-shank drills (fig. 08/02) are held in a drill chuck; tapered-shank drills (fig. 08/01) fit into the internal taper of the drill press spindle.

- A tang (fig. 08/01) is provided on the end of tapered-shank drills to prevent the drill from slipping while it is cutting and to allow the drill to be removed from the spindle or socket without the shank being damaged by using a drill drift.

Body

The body is the portion of the drill between the shank and the point. It consists of a number of parts important to the efficiency of the cutting action.

The flutes are two or more helical grooves cut around the body the body of the drill. They form the cutting edges, admit cutting fluid, and allow the chips to escape from the hole.

- The margin is the narrow, raised section on the body of the drill. It is immediately next to the flutes and extends along the entire length of the flutes. Its purpose is to provide a full size to the drill body and cutting edges.
- The lip clearance is the undercut portion of the body between the margins and the flutes. It is made smaller to reduce friction between the drill and the hole during the drilling operation.
- The web is the thin partition in the center of the drill which extends the full length of the flutes. This part forms the chisel edge at the cutting end of the drill. The web gradually increases in thickness toward the shank to give the drill strength.
- Point The point of a twist drill consists of the chisel edge, the lips, the lip clearance angle and the heel.
- The chisel edge (web) is the chisel-shaped portion of the drill point.
- The lip (cutting edge) a formed by the intersection of the flutes. The lips must be equal length and have the same angle so that the drill will run true and will not cut a hole larger than the size of the drill.
- The lip clearance angle is the relief which is ground on the point of the drill extending from the cutting lops back to the heel. The average lip clearance is from 8° to 12°, depending upon the hardness or softness of the material to be drilled.

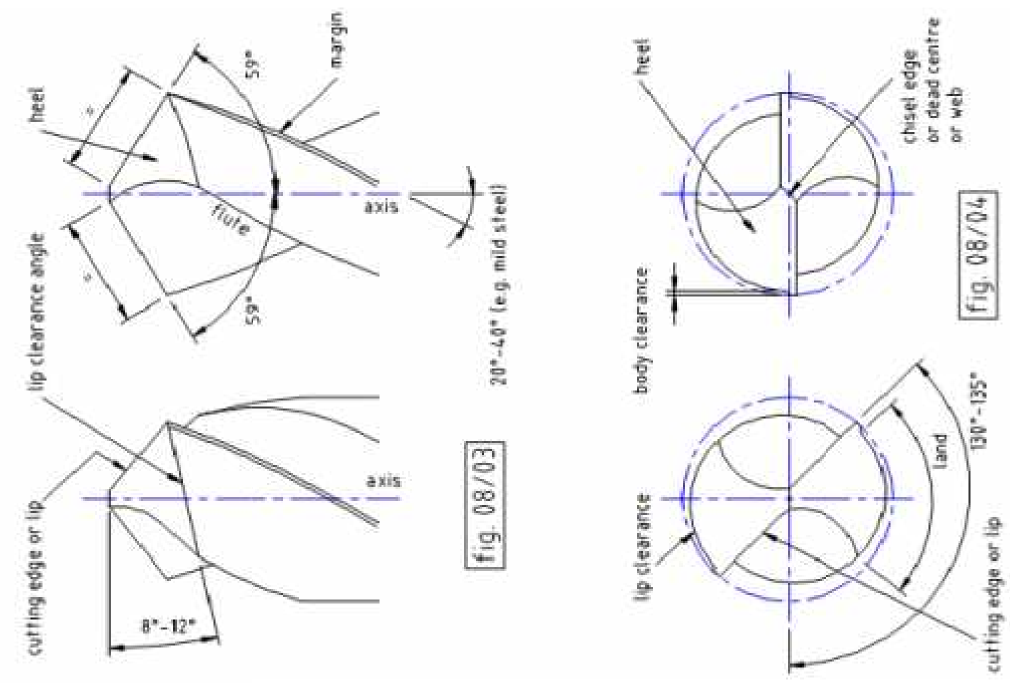


Fig.4.2. drill bit cutting edge arrangement

E. Speed

A wide range of drills and drill sizes is used to cut various metals; an equally wide range of speeds is required for the drill to cut efficiently. For every job, there is the problem of choosing the drill speed which will result in the best production rates and the least amount of downtime for regrinding the drill. The recommended cutting speeds for drilling various types of materials may be found in the table shown below. The most economical drilling speed depends upon many variables such as:

- the type and hardness of the material
- the diameter and material of the drill
- the type and condition of the drill press
- the efficiency of the cutting fluid employed

To determine the correct number of r/min (revolution per minute) of a drill press spindle for a given size drill, the following should be known:

- ✓ the type of material to be drilled
- ✓ the recommended cutting speed of the material
- ✓ the type of material from which the drill is made

Table 4.1.Drill size and types of materials.

Drill size	stainless steel	tool steel	cast iron	machine steel	aluminum
2	1910	2865	3820	4775	9550
3	1275	1910	2545	3185	6365
4	955	1430	1910	2385	4775
5	765	1145	1530	1910	3820
6	635	955	1275	1590	3180
7	545	820	1090	1365	2730
8	475	715	955	1195	2390
9	425	635	850	1060	2120
10	350	520	695	870	1735
15	255	380	510	635	1275
20	190	285	380	475	955
25	150	230	305	380	765
	CS = 12	CS = 18	CS = 24	CS = 30	CS = 60

- ✓ CS in m/min   CS x 1000   CS x 320
- ✓ Formula r/min = --------- ⇒ ------- ⇒ -------------
- ✓ π x D in mm   3.14 x D
- ✓ D For metric calculations, the formula is used:

$$rpm = \frac{CS(m)}{\Pi D(mm)}$$

It is necessary to convert the meters in numerator to millimeters so that both parts of the equation are in the same unit. To accomplish this, multiply the CS in meters per minute by 1000 to bring it to millimeters per minute.

$$rpm = \frac{CS \times 1000}{\Pi D}$$ This can be simplified to $$rpm = \frac{CS \times 320}{D}$$

This is done for the reason that not all machines have variable speed drives and therefore cannot be set to the exact calculated speed. Dividing 1000 by π (3.14) we arrive at the formula above. This formula is accurate enough for most drilling operations.

Example:

Calculate the rpm required to drill a 15mm hole in tool steel using a high speed steel drill.

Solution:

$$rpm = \frac{CS \times 320}{D} = \frac{18 \times 320}{15} = \frac{5760}{15} = 384$$

F. **Feed**

Feed is the distance that a drill advances into the work for each revolution.

In other words the feed of a drill is the distance the drill moves into the job at each revolution of the spindle. It is expressed in millimeter. The feed may also be expressed as feed per minute. The feed per minute may be defined as the axial distance moved by the drill into the work per minute.

The rate of feed is generally governed by:

- the diameter of the drill
- the material of the work piece
- the condition of the machine

Table4.2.Drill size and feed per revolution.

Drill size	feed per revolution	
1 to 3		0.02 to 0.05
3 to 6		0.05 to 0.10
6 to 13		0.11o 0.18
13 to 25		0.19o 0.38

1. Boring holes

In machining, **boring** is the process of enlarging a hole that has already been drilled (or cast) by means of a single-point cutting tool (or of a **boring** head containing several such tools), such as in **boring** a gun barrel or an engine cylinder

The enlargement of holes is achieved via boring operations. The hole diameter is either enlarged with a single insert attached to a long boring bar, or with a boring head which has a diameter equal to the diameter of the hole to be enlarged. Long boring bars statically and dynamically deform under the cutting forces during boring operations. Excessive static defections may violate the dimensional tolerance of the hole, and vibrations may lead to poor surface, short tool life and chipping of the tool. Predictions of the force, torque and power are required in order to identify suitable machine tool and fixture set up for a boring operation. A comprehensive engineering model, which allows prediction of cutting forces, torque, power, dimensional surface finish and vibration free cutting conditions, is required in order to plan boring operations in the production floor.Boring is performed on the inside diameter of an existing hole.

Fig4.3. Boring operation.

1. Boring Tools

Metal boring tools are used in metal work. In the art of, metal working applications include drilling holes during car manufacture, precision cutting and roof installation. Some artists, searching for more ways to express themselves, make excellent sculptures and other works using different metals. They produce high-quality using metal-boring machines, including hand drills, drill presses and lathes. Almost all of these tools require external power sources.

2. Reaming holes

Reaming is similar to drilling operation. It is basically used for finishing of holes and enlarging of small holes. Reaming is carried out by the reamer, which has large number of flutes. Each component in a product must be made to exact standards in order for that product to function properly. Since it is impossible to produce holes which are round, smooth and accurate to size by drilling, the reaming operation is very important. Reamers are used to enlarge, and finish a hole previously formed by drilling or boring. Speed, feed, and reaming allowances are the three main factors which will affect the accuracy and finish of the hole and the life of the reamer.

2.1. Reamers

A reamer is a rotary cutting tool with several straight or helical cutting edges along its body. It is used to accurately size and smooth a hole which has been previously drilled or bored. Some reamers are operated by hand (hand reamers), while others may be used under power in any type of machine tool (machine reamers).

2.2. Parts of the Reamer

Reamers generally consist of three main parts: shank, body and angle of chamfer.

The ***shank***, which may be straight or tapered, is used to drive the reamer. The shank of machine reamers may be straight or tapered, while hand reamers have a square end on the end to accommodate a tap wrench.

The ***body*** of a reamer contains several straight or helical grooves or *flutes*, and *lands* (the portion between the flutes). A *margin* (the top of each tooth) runs from the angle of chamfer to the end of the flute. The *body clearance angle* is the relief or clearance behind the margin which reduces the friction while the reamer is cutting. The *rake angle* is the angle formed by the face of the tooth when a line is drawn from a point on the front marginal edge through the center of the reamer. If there is no angle on the face of the tooth, the reamer is said to have radial land.

The ***angle of chamfer*** is the part of the reamer which actually does the cutting. It is ground on the end of each tooth and there is clearance behind each chamfered cutting edge. On rose reamers, the angle of chamfer is ground ion the end only and the cutting action occurs at this point. On fluted reamers, each tooth is relieved and most of the cutting is done by the reamer teeth.

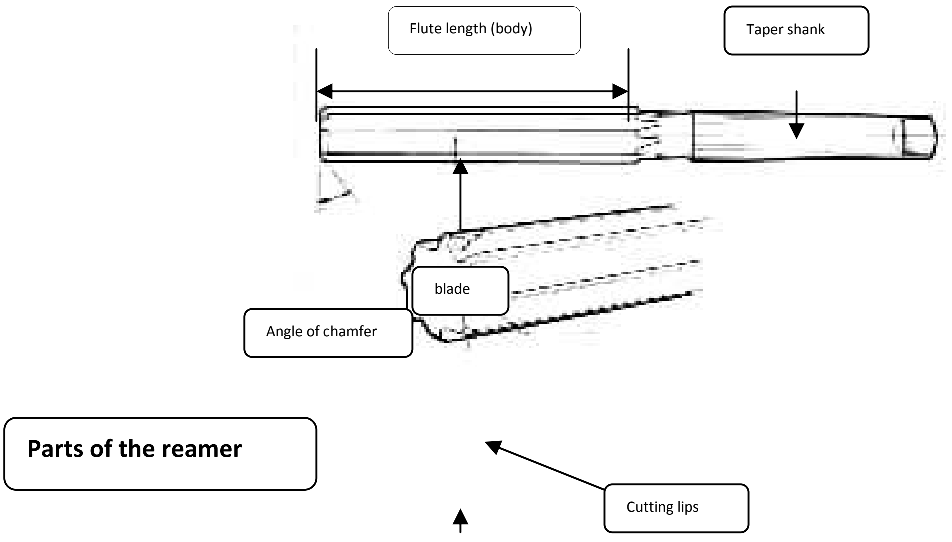


Fig4.4.Reamer and its parts.

2.3. Types of Reamers

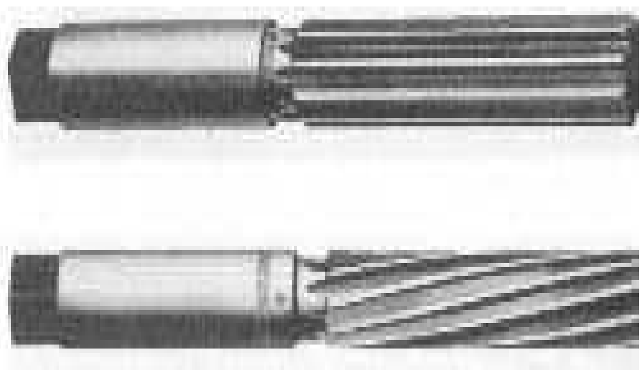
Reamers are available in a variety of design and sizes; however, they fall into two general classifications: ***machine*** and ***hand*** reamers.

Hand Reamers

Hand reamers are finishing tools used when a hole must be finished to a high degree of accuracy and finish. Holes to be hand reamed should be bored to within 0.07 to 0.12mm of the finish size. Never attempt to ream more than 0.12mm with a hand reamer.

A square on the shank end allows a wrench to be used for turning the reamer into the hole. The teeth on the end of the reamer are tapered slightly for a distance equal to the reamer diameter so that it can enter the hole to be reamed.

A hand reamer should never be used under mechanical power and should never be turned backwards. When using a hand reamer, keep it true and straight with the hole. The dead center in a lathe or a stub center in a drill press will help keep the reamer aligned during the hand reamer operation.



Straight and helical fluted hand reamers

Fig4.5. Hand reamers

Reaming cannot correct a badly positioned hole; it can only smooth it.

Points to remember in hand reaming include the following:-

1. Drill the hole to be reamed with care.
2. Hold the work securely in the vice.
3. Use a good supply of cutting lubricant to help remove chips and reduce friction to obtain a smooth finish.

3. Honing holes

Honing is the abrading process done mostly for finishing round holed Produced by drilling, reaming or boringly means of bonded abrasive stones Called 'hones'. Honing is a machining process and is used to remove metal up to 0.25 mm. The surface roughness value can be maintained between

0.025and 0.4 microns. So honing is used to correct some out of roundness, tapers, tool marks and axial distortion.

Honing is a finishing process performed by a honing tool, which contains a set of three to a dozen and more bonded abrasive sticks. The sticks are equally spaced about the periphery of the honing tool. They are held against the work surface with controlled light pressure, usually exercised by small springs. The honing tool is given a complex rotational and oscillatory axial motion, which combine to produce a crosshatched lay pattern of very low surface roughness:

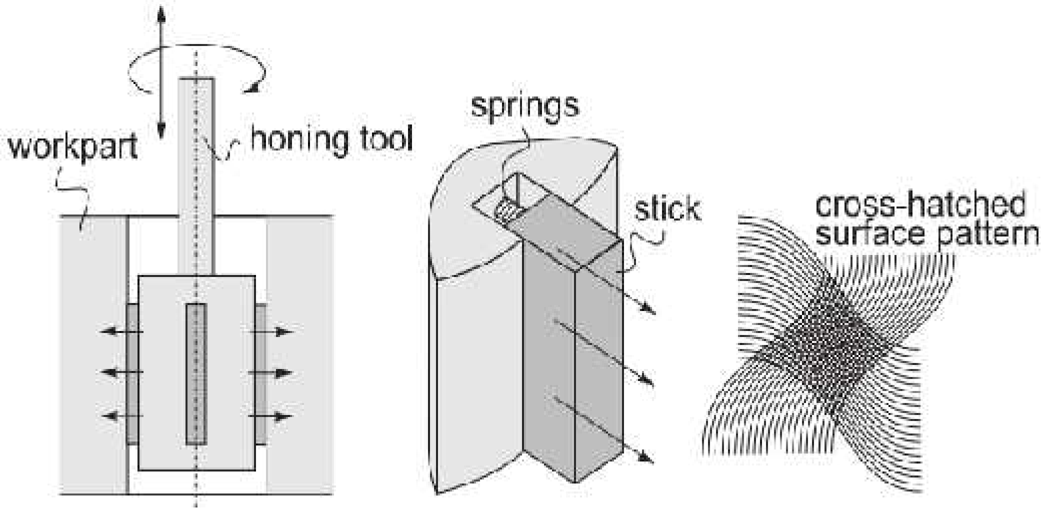


Fig4.6. Honing operation

Schematics of honing process showing the honing tool, how the abrasive sticks are pressed against the work surface by springs, and the resulting surface pattern.

In addition to the surface finish of about 0.1 µm, honing produces a characteristic crosshatched surface that tends to retain lubrication during operation of the component, thus contributing to its function and service life. A cutting fluid must be used in honing to cool and lubricate the tool and to help remove the chips. A common application of honing is to finish the holes. Typical examples include bores of internal combustion engines, bearings, hydraulic cylinders, and gun barrels.

4. **Lapping**

In lapping, instead of a bonded abrasive tool, oil-based fluid suspension of very small free abrasive grains (aluminum oxide and silicon carbide, with typical grit sizes between 300 and 600) called a lapping compound is applied between the work piece and the lapping tool. The lapping tool is called a lap, which is made of soft materials like copper, lead or wood. The lap has the reverse of the desired shape of the work part. To accomplish the process, the lap is pressed against the work and moved back and forth over the surface in a figure-eight or other motion pattern, subjecting all portions of the surface to the same action. Lapping is sometimes performed by hand, but lapping machines accomplish the process with greater consistency and efficiency.

The cutting mechanism in lapping is that the abrasives become embedded in the lap surface, and the cutting action is very similar to grinding, but a concurrent cutting action of the free abrasive particles in the fluid cannot be excluded. Lapping is used lo produce optical lenses, metallic bearing surfaces, gages, and other parts requiring very good finishes and extreme accuracy.

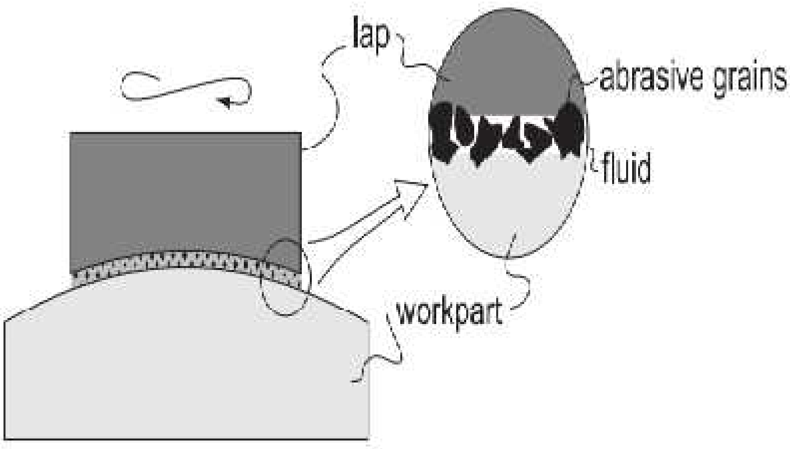


Fig4.7.7. Schematics of lapping process showing the lap and the cutting action of suspended abrasive particles.

Self check-4

I. <u>Multiple choice</u>

Select one of the appropriate answers and give on the space provided

1. _____is the process of producing round hole in a solid material or enlarging existing holes.
   a. Reaming b. Drilling c. Honing d. Filling
2. ____Which one of the following is a finishing process of drilled holes?
   a. Honing b. filling c. Sawing d. Drilling
3. ____is the process of enlarging a hole that has already been drilled.
   a. Drilling b. Honing c. Boring d. None
4. ____one of the following is the parts of bench drill
   a the base b the column c the head d the spindle e all
5. _____which one of the following is different from the others?
   a. shank b. Body c. tip d. lever

II. <u>True/ false</u>

Write true if the statement is correct and false if it is wrong

1. A drill can produce a hole without using power

2. Drill size can be related with the thickness of metals

3. It is possible to produce a hole safely without holding the work in a vice

III. Matching

Match column A within column B

A	**B**
1. Enlarging a hole	A. drill bit
2. Hole producing tool	B. reamer
3. Kind of drilling machine	C. hand drill
4. for cooling purpose	D. boring
5. Tool for reaming a hole	E. lubricants
	F. shank
	G. flute

Operation sheet 4.1: produce a hole

Operation title: producing a hole

Purpose: To cut a hole

Instruction: Using the given equipments and tools produce a hole of 30mm diameter. You have given 30 minute for the task.

Quality Criteria: the pitch is measured within the standard

Precautions: follow all safety rules

Tools and requirement:

1. Drilling machine
2. Drill bit
3. Coolant
4. Work holding devices
5. Drill chuck
6. Personal protective equipment

Steps in doing the task

1. Prepare the metal to the required dimension
2. Hold the work properly in the vice
3. Select the appropriate drill bit
4. Continue cutting process
5. Add the necessary coolant
6. Complete the work
7. Check the size of the hole

Lap Test-4

Task-1: Perform measurement using measuring tools

Task-2: mark the object using marking tools

Task3: identify and inspect the machine for safety

Task-: adjust work holding devices

Task5: select and prepare drill bit

Unit five: **Perform Off-hand grind cutting tools**
This learning guide is developed to provide you the necessary information regarding the following content coverage and topics: • Hone cut edges and remove burrs. • Sharpen Cutter. • Grind cutters using appropriate cooling agents. • Perform Cutting tool grinding safely This guide will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to: • Remove burrs • Sharpen cutters • Use appropriate cooling agents. • Perform cutting tool grinding

5.1. Hone cut edges and removes burrs

Off – hand grinding is the term used in engineering to describe the process where the work is held by hand material is removed using an abrasive wheel.

This type of grinding is carried out in the workshop for such work as:

a. Removing excess materials

b. Smoothening surfaces

c. Preparing plates for welding

d. Sharpening cutting tools (drills, chisels, punches, shaper and lathe tools)

Off – hand grinding must be performed with great regard of safety. The principle of operation requires an exposed portion of the abrasive wheel to be in close proximity to the operator.

Hazard may be created by having relatively heavy abrasive wheels rotating a high speed. The wheels on all types of machines must be heavily guarded.

The guard exposes enough of the wheel surface to enable the operator to perform the work require

Parts and Functions (features of machines)

a. Work Rests
b. Wheel Guards
c. Wheel Speed
d. Wheel Rotation

5.2. Importance of honing:

Honing is an abrasive machining process that produces a precision surface on a metal work piece by scrubbing an abrasive stone against it along a controlled path. Honing is primarily used to improve the geometric form of a surface, but may also improve the surface texture.

Typical applications are the finishing of cylinders for internal combustion engines, air bearing spindles and gears. There are many types of hones, but all consist of one or more abrasive stones that are held under pressure against the surface they are working on.

In terms of sharpening knives, honing steel does not actually hone knives, but simply realigns the metal along the edge.

Other similar processes are lapping and super finishing.

Honing uses a special tool, called a *honing stone* or a *hone*, to achieve a precision surface. The hone is composed of abrasive grains that are bound together with an adhesive. Generally, honing grains are irregularly shaped and about 10 to 50 micrometers in diameter (300 to 1,500 mesh grit). Smaller grain sizes produce a smoother surface on the work piece.

A honing stone is similar to a grinding wheel in many ways, but honing stones are usually more friable so that they conform to the shape of the work piece as they wear in. To counteract their friability, honing stones may be treated with wax or sulfur to improve life; wax is usually preferred for environmental reasons.

Any abrasive material may be used to create a honing stone, but the most commonly used are corundum, silicon carbide, cubic boron nitride, or diamond. The choice of abrasive material is usually driven by the characteristics of the work piece material. In most cases, corundum or silicon carbide are acceptable, but extremely hard work piece materials must be honed using super abrasives.

The hone is usually turned in the bore while being moved in and out. Special cutting fluids are used to give a smooth cutting action and to remove the material that has been abraded. Modern advances in abrasives have made it possible to remove much larger amount of material than was previously possible. This has displaced grinding in many applications where "through machining" is possible. External hones perform the same function on shafts.

5.2. Sharpen cutters

5.2.1. Concepts of Sharpening Cutters

The word sharpening is usually used for the final finishing of edge tools. Like all edge tools, a drill bit needs to have the right shape before you can start to sharpen it. Creating the initial shape often means that quite a lot of steel needs to be re-moved when for example, you change the point angle of a drill or you shape a broken or heavily worn drill. Once the geometry of the point is established, you maintain the sharpness by sharpening. Edge tools need to be sharp to work efficiently. You can sharpen tools with a bench stone or, in the case of knives, with a sharpening steel.

This means that you work on the very tip of the bevel and the tool is sharp again.

When sharpening with steel or a bench stone, a very limited amount of steel is removed.

After several sharpening or honing, the edge angle becomes too wide and the tool must

be re-shaped. Sooner or later all edge tools need to be re-shaped and this is done by grinding on a grindstone or a grinding wheel. When only a limited amount of steel is removed this operation is also called sharpening.

Grinding means that so much steel is removed from the tool that the edge is restored to the original angle or altered on purpose to a new angle. The shape of the tool can also be changed according to your requirements.

For some tools it is very important to keep them sharp at all times. Common tools, such as scribers, center punch, chisels, drill bits, tool bits for lathe machine needs to be sharpened every time you feel that they do not cut well.

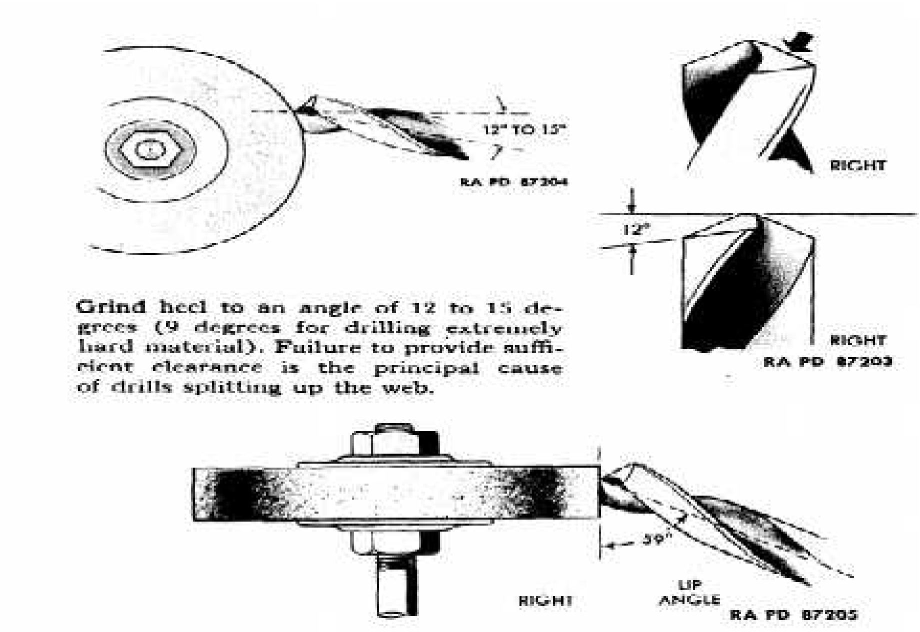


Fig.5.1. tool grinding

5.3. Use appropriate cooling agents.

5.3.1. Concepts of cooling agents

Cutting fluids are used in metal machining for a variety of reasons such as improving tool life, reducing work piece thermal deformation, improving surface finish and flushing away chips from the cutting zone.

Cutting fluids consist of those liquids and gases that are applied to the tool and the material being machined to facilitate the cutting operation. Vast quantities are used annually to accomplish a number of objectives. (Boston, 1952)

1) To prevent the tool from overheating, i.e. so that no temperature is reached where the tool's hardness and resistance to abrasion are reduced, thus decreasing the tool life.
2) To keep the work cool, preventing machining those results in inaccurate final dimensions.
3) To reduce power consumption, wear on the tool , and the generation of heat, by affecting the cutting process. This investigation wishes to establish a relationship between the surface

chemistry of the lubricants involved and how they can accomplish reducing the contact length on the rake face of the tool where most of the heat during cutting is produced.

4) To provide a good surface finish on the work.

5) To aid in providing a satisfactory chip formation (related to contact length)

6) To wash away the chips/clear the swarf from the cutting area.

7) To prevent corrosion of the work, the tool and the machine.

The desirable properties of cutting fluids in general are (Boston, 1952)

1) High thermal conductivity for cooling
2) Good lubricating qualities
3) High flash point, should not entail a fire hazard
4) Must not produce a gummy or solid precipitate at ordinary working temperatures
5) Be stable against oxidation.
6) Must not promote corrosion or discoloration of the work material.
7) Must afford some corrosion protection to newly formed surfaces.
8) The components of the lubricant must not become rancid easily
9) No unpleasant odour must develop from continued use
10) Must not cause skin irritation or contamination
11) A viscosity that will permit free flow from the work and dripping from the chips.

1. Types of cutting fluids

Cutting fluids may be divided into four main categories (FVTC, 2000):

a. Straight or neat cutting oils

b. water miscible or water-based fluids

c. gases

d. paste or solid lubricants

a. **Straight Cutting Oils**

Straight cutting oils are not mixed with water. Cutting oils are generally mixtures of mineral oil and animal, vegetable or marine oils to improve the wetting and lubricating properties. Sulphur, chlorine, and phosphorous compounds are sometimes added to improve the lubrication qualities of the fluid for extreme pressure applications. There are two main types of straight oils: active and inactive.

b. Water miscible or water-based fluids

The water-based fluids act mainly as coolants and the neat cutting oils act mainly as lubricants. There are many variants of both types. Fatty acids are often incorporated in the neat oils. Until recently both the emulsions or soluble oils as they are also called and the neat oils, contained chlorine and sulphur additives that improved lubrication under extremely difficult conditions. Chlorine affects the skin detrimentally and its degradation products are often carcinogenic and sulphur is environmentally unacceptable. Consequently other lubrication improvers under difficult conditions are searched for. Ester technology is used successfully for softer materials where high rates of metal working are needed, and where heat generation is not a major problem.

These can operate at higher temperatures as they have better resistance to thermal degradation than mineral oils. They are biodegradable and do not cause dermatitis and are therefore more environmentally acceptable. In many cases phosphor and sulphur do however still form part of the cutting fluid.

For the water miscible fluids water quality has a large effect on the coolant. Hard water (high mineral content) can cause stains and corrosion of machines and work pieces.

Water can be de-ionized to remove the impurities and minerals. Water is the best fluid for cooling. It has the best ability to carry heat away. Water, however, is a very poor lubricant and causes corrosion.

Oil is excellent for lubrication but very poor for cooling, and it is also flammable. It is clear that, from a lubrication point of view water and oil have strengths but also some weaknesses. If water and oil are combined and an attempt is made to minimize the weaknesses the best properties of both may be balanced to obtain desirable end properties for the cutting fluid. Water-soluble fluids have been developed which have good lubrication, cooling ability, low-flammability and corrosion resistance. These fluids are usually mixed on site. It is crucial that the mixing directions and concentrations are followed very closely to get the maximum benefit from the coolant.

c. Emulsions

An emulsion is a dispersion of oil droplets in water. Soluble oils are mineral oils that contain emulsifiers. Emulsifiers are soaps or soap-like agents that allow the oil to mix with water and stay in suspension. Emulsions (soluble oils) when mixed with water produce a milky white product. Lean concentrations (more water, less oil) provide better cooling but less lubrication. Rich concentrations (less water, more oil) have better lubrication qualities but poorer cooling properties.

There are different types of soluble cutting fluids available including extreme pressure soluble oils. These are used for extreme machining conditions like broaching and gear hobbling for example.

d. Chemical Fluids

Chemical coolants are also miscible cutting fluids. Chemical cutting fluids are pre-concentrated emulsions that contain very little oil. Chemical fluids mix very easily with water to form an emulsion. The chemical components in the fluid are used to enhance the lubrication, bacterial control, and rust and corrosion characteristics. There are several types of chemical coolants available including coolants for extreme cutting conditions.

Inactive chemical cutting fluids are usually clear fluids with high corrosion inhibition, high cooling, and low lubrication qualities. Active chemical fluids include wetting agents.

They have excellent rust inhibition and moderate lubrication and cooling properties.

Sulphur-, chlorine- and phosphorous- containing compounds are sometimes added to improve the extreme pressure characteristics. These are usually in an organic form, i.e. the sulphur, chlorine or phosphorus IS grafted onto a hydro-carbon backbone.

c. Gases and vapors

Cutting oils and water miscible types of cutting fluids are the most widely used.

Compressed air, inert gases like carbon dioxide, Freon, and Nitrogen are sometimes used. A vortex tube may be used to apply gaseous lubricants or coolants. Using this tube, it is possible to apply the gases at a very low temperature and under medium pressure thereby facilitating a higher gas density and cooling and lubrication capability. Cutting using sub-zero cold gas is known as cryogenic cutting. The gas stream also helps to blow away chips from the cutting area.

d. Paste and Solid Lubricants

Waxes, pastes, soaps, graphite and molybdenum disulphide are examples falling into this category. These are generally applied directly to the work piece or tool or in some cases impregnated directly into the tool, for example the grinding wheel of a grinder. One example of a paste lubricant is lard. Many experienced journeymen recommend lard for tapping.

5.4. Perform cutting tool grinding

Grinding is a metal cutting operation like any other process of machining removing metal in comparatively smaller volume. The cutting tool used is an abrasive wheel having many numbers of cutting edges. The machine on which grinding the operation is performed is called a grinding machine. Grinding is done to obtain very high dimensional accuracy and better appearance. The accuracy of grinding process is 0.000025mm. The amount of material removed from the work is very less.

Grinding machines are made in a variety of types and sizes, depending upon the class of work for which they are to be used.

Procedures to Perform cutting tool grinding operation:

Step1- Adjust safety glass shields on the grinder to permit clear vision of the part to be ground and still protect the operator from flying particles.

Step 2- Dress grinding wheel with dressing tool.

Step 3- Hold drill bit against face of wheel at 59^0 angle on cutting lip.

Step 4- Carry drill bit up the wheel face by dropping end and rotating very slightly in a clockwise direction.

Step5- Make slow deliberate strokes, the full width of the cutting lip.

Step 6- Do not lower cutting lip below the horizontal position as this will round the cutting edge.

Step 7- When one lip is ground, rotate the drill one half turn and grind the other lip.

Step 8- Use tool gauge to check equal lengths of lips, 590 angle cutting lip and $12\text{-}15^0$ lip clearance.

Step 9- Test bit by boring hole in mild steel plate.

Step 10- Stop while drilling, turn drill press in reverse direction to release drill bit from hole.

Step 11- Make grinding corrections on drill bit as indicated by hole.

Step 12- Submit drill bit and metal for evaluation.

Self-check-5.1

I .<u>true/false</u>

Write true if the statement is correct and false if it is wrong

1. Grinding is the process of producing holes

2. Grinding can be used for sharpening cutting tools

3. Wheel of a grinder is used to hold the work piece

II. <u>Matching</u>

Match column A within column B

<u>A</u>	**<u>B</u>**
1. Used to remove burrs	A. chisel
2. Types of cutting fluids	B. liquid oil
3. Part of grinder	C. grinder
	D. tool rest

III. **<u>Give short answers for the given questions below</u>**

1. What is sharpening?

2. Write the differences between portable grinder and portable drill

3. write some of the uses of grinding machine

Operation sheet 5.1: sharpen a drill bit

Operation title: sharpening drill bit

Purpose: To sharpen drill bit

Instruction: Using the given equipments and tools sharpen drill bit. You have given 30 minute for the task.

Quality Criteria: the sharp edge of drill bit

Precautions: follow all safety rules

Tools and requirement:

1. Grinding machine
2. Cutting fluid
3. Personal protective equipment

Steps in doing the task

1. Identify edge damaged drill bit
2. Check that the machine is in good condition
3. Dress the wheel properly
4. Continue sharpening process
5. Add the necessary coolant
6. Complete the work
7. Check the edge for sharpness

Lap Test-5

Task-1: identify the drill bit damaged

Task-2: adjust the machine and its wheel

Task3: select the appropriate cutting fluid

Task4: hold the work piece firmly on tool rest

Task5: observe its sharpness

Unit six: **Assure Quality of finished component**

This learning guide is developed to provide you the necessary information regarding the following content coverage and topics:

- Check Component for conformance.
- Use appropriate techniques, *measuring tools* and equipment.
- Handle deviations

This guide will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Identify component conformance
- Apply appropriate techniques
- Assign deviations

6.1. Check Component for conformance

Inspection or checking of components or products with required specifications is very minutely related with quality control. It is generally an accepted fact that no two things can ever be exactly same. It also holds true with manufactured parts. Therefore certain variations or deviations in dimensions and other product specifications are accepted. However, only few produced articles or parts may be rejected if the deviations go beyond the specified quality standards. Therefore it becomes essential to detect errors so that the manufacturing of faulty product does not go uncorrected. The philosophy of inspection is only preventive and not remedial. In other words the inspection of products is measuring or checking its quality in terms geometrical tolerances of other specified feature of needed design. Generally, there are three basic areas of inspection namely receiving inspection, in-process inspection and final inspection. In the receiving inspection, inspections are performed on all incoming materials and purchased parts. In the in-process inspection the products are inspected as they are in processed in stages from starting station to finished station. In the final inspection, all finished products or parts are inspected finally prior to delivering them to the customer.

The main motive of manufacturing is to process engineering materials and produce desired and useful components or products to the specified shape, size and finish. The specifications for the shapes, sizes and finishes on the products are furnished by the manufacturing operations through specified process plan using part drawings or manufacturing drawings. These specifications basically termed as called quality characteristics. The quality of manufactured product is depend always upon the process capability of controlling manufacturing functions which may lead to a certain amount of variation as a result of chance and some cause. Also some chance or cause is inherent in any particular scheme of production and inspection. The reasons for variation outside this stable system should be discovered and corrected to minimize wastage and finally to improve quality.

6.2. Use appropriate techniques, *measuring tools* and equipment

Full inspection of the finished product by usual inspection devices is time consuming and costly. Also in a continuous production process 100% inspection may not be found practicable.

Certain statistical techniques have been formulated to evaluate materials, processes and products by observing capabilities and trends in variations so that continual analysis predictions may be made to control the desired quality level of the part.

It is essential to meet consistently all of the customer's expectations all the time and every time, to keep them satisfied and loyal.

The techniques are

1. Control for measurable quality characteristics.
2. Control for fraction defective.
3. Control for number of defects per unit.

Quality Control techniques are commonly used in quality control in industries to maintain a continuous evaluation of the manufacturing process. A process is said to be in control if the observed values are influenced only by chance causes fall within the limits and out of control when assignable causes seem to be operating in the system and the observed value fall outside the limit

6.3. Handle deviations

It is difficult to manufacture any product or component to its exact size. Tolerance on the parts is therefore the amount of variation in size tolerated to cover reasonable imperfections in workmanship and it varies with different grades of work. Tolerance on a dimension can also be specified as the difference between the maximum limit of size and the minimum limit

Zero Line and Deviation

In graphical representation of limits, straight line to which the deviations are of zero deviation and the represented of zero deviation are referred. The zero line is also known as line of zero deviation is generally drawn as horizontal line and the positive deviations are shown above this line and the negative deviations below it.

The algebraic difference between the actual or maximum size and the corresponding the basic size is called deviation. The deviations from the basic dimensions at the boundaries of the tolerance zone are called upper and lower deviations as depicted in

Upper Deviation

It is the algebraic difference between the two maximum limit of any size of the part and the corresponding basic size.

Lower Deviation

It is the algebraic difference between the minimum limit of any size of the part and the corresponding basic size.

Mean Deviation

It is the arithmetical mean between the upper and lower deviations of any size of the part.

Fundamental Deviation

It is the one of the deviations, which is conventionally chosen to define the position of tolerance zone in relation to the zero line. The deviation of the tolerance band on shaft or hole away from the basic size is called the fundamental deviation.

The zero line shown is the line of zero deviation and represents the basic size. A zero line is a straight line to which the deviations are referred. For conventions, the zero line is drawn horizontally. it. honing, super finishing, belt grinding, polishing, sanding, tumbling, organic finishes, burring, electroplating, buffing, metal spraying, painting, inorganic coating, anodizing, galvanizing, plastic coating, metallic coating and anodizing. The characteristics of good finish or roughness of surfaces can be understood by following components, which are described as under broaching and grinding. on the surface.

Self-check 6.1

I. **multiple choice**

Select one of the appropriate answers and give on the space provided

1________which one of the following is true about quality?

A. all products have the same quality

B. the quality of products cannot be varied

C. Two products may not contains the same quality

D. none

2______quality of a product can be related with:

A. production system B. number of products

C. number of workers D. all

3. ______Quality of products produced on bench work can be affected by:

A. error from measurement B. error from cutting

C. error from holding work piece D. all

4. _____which one of the following processes used to produce smoother surface?

A. chiseling B. sawing C. scraping D. filing

II. **True/ false**

Write true for correct statement and false for wrong statement

1. In any production system quantity should be taken as a criteria than quality
2. Producing a product beyond the given tolerance does not have any problems
3. The sharpness of a drill bit related to the smoothness of hole
4. The arrangement of the teeth of hack saw can reduce the quality of its cutting

III**. Matching**

Match column A within column B

A	B
1. Dimension	A. quantity
2. Number of products	B. size
3. Defect	C. crack on the object
4. Finished surface	D. on all incoming materials
5. Receiving inspection	E. smooth
	F. on all processing materials

Reference

1. Introduction to Basic Manufacturing process and work shop Technology

2. Welding Science and Technology

3. Welding theory and Technology

4. Welding Technology 1

5. Manufacturing Engineering

6. Introduction to basic Manufacturing

Participants of this Module (training material) preparation

No	Name	Qualification (Level)	Field of Study	Organization/ Institution	Mobile number	E-mail
1	Adugna Mosisa Ejeta	A(MSC)	Manufacturing Technology	Nekemte Poly Tec.	0911921931	
2	Ermiyas Ambaye Neway	A(MSC)	Manufacturing	Nefas SILK	0911169904	Ambayeermi@
3	Tadele Worku Bidika	A(MSC)	Manufacturing Technology	Asosa Poly Tec.	0913970611	Tadeworku09@gmail.com
4	Wolela Seid	A(MSC)	Manufacturing Technology	Hawasa Poly Tec.	0912134346	Wolelaseid11@gmail.com