

Machining

Level-III

Based on October 2023, Curriculum Version II



Module Title: Perform Advanced Milling Operations

Module Code: INDMAC3 M08 1023

Nominal Duration: 100 Hours

Prepared by: Ministry of Labor and Skill

October, 2023

Addis Ababa, Ethiopia

Page 1 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1 October, 2023
---------------	--	-------------------------------------	-----------------------------

Acknowledgment

The Ministry of Labor and skill wishes to thank and appreciation to MoLS leaders and experts, Regional Labor and skill/training Bureaus leader, experts, TVT College Deans, Instructors and industry experts who contribute their time and professional experience to the development of this Training Module.

Page 2 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
			October, 2023

Acronym

D	Diameter of the tool
FZ	Feed per tooth
f	Feed rate
CS	Cutting speed
CPT	Chip per tooth
CL	Chip load
MRR	Material removal rate
N	Spindle speed
NDT	Nondestructive testing
CMM	Coordinate measuring machine
SPC	Static process control
L.C	Leas count
QRM	Quality risk management
QC	Quality control

Contents

Acknowledgment	2
Acronym	3
Introduction to the Module	5
Unit One: Introduction to Milling.....	6
1.1. Introduction to Milling.....	7
1.2. Interpret Drawings.....	8
1.3. Operational Sequences	15
1.4. Milling Cutting Tools	21
1.5. Work Shop Safety	28
Self-Check 1.....	30
Unit Two: Set-Up Machine and Work Piece	33
2.1. Machine and Work Set Up.....	34
Self-Check 2.....	39
Operation Sheet 2.1.....	40
Operation Sheet 2.2.....	41
Lap Test 2.1.	42
Unit Three: Advanced Milling Operations	44
3.1. Calculating Cutting Parameter	45
3.2. Calculate Measurement.....	52
3.3. Milling Accessories	52
3.4. Advanced Milling Operations	68
Self-Check 3.....	95
Operation Sheet 3. 1.....	97
Operation Sheet 3.2.....	99
Operation Sheet 3.3.....	100
Operation Sheet 3.4.....	101
LAP Test 3.1	102
Unit Four : Product Quality	104
4.1. Inspect Components.....	106
4.2. Handling Deviation	116
4.3. Assure Quality	120
Self-Check 4.....	123
Reference	125

Introduction to the Module

In *machining* filed; *Perform Advanced Milling Operations* to determine job requirements, to set-up work piece, to perform milling operations, and to check/ measure work piece.

This Module covers the knowledge, skills and attitudes required to set-up and mill work-piece according to the drawing. It details the requirements for performing milling operations considered as “intermediate” such as indexing, milling splines and equally-spaced grooves, in cylindrical work –piece and rack, ratchets, converging faces, large radial slots and internal radii.

This module is designed to meet the industry requirement under the *machining* occupational standard, particularly for the unit of competency: **Perform Advanced Milling Operations.**

This module covers the units:

- Introduction to milling
- Set-up machine and work piece
- Advanced milling operation
- product quality

Learning objectives of the Module

- Understand advanced milling operation
- Set-up machine and work
- Perform Advanced milling operations
- Confirm quality

Module Learning Instructions:

For effective use these 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: Introduction to Milling

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

- Basic information of milling
- Interpret drawings
- operational sequence
- Milling Cutting tools.
- Work shop safety

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:

- Understand Basic information of milling
- Interpret drawings
- Understand operational sequence
- Identify Milling Cutting tools
- Apply Work shop safety

1.1. Introduction to Milling

Milling machines were first invented and developed by Eli Whitney to mass produce interchangeable musket parts. Although crude, these machines assisted man in maintaining accuracy and uniformity while duplicating parts that could not be manufactured with the use of a file.

It is a machine tool that will produce machined surfaces accurately by means of single or multiple tooth cutters. The work table holds attachments, fixtures or more securely in placed by means of T-bolts, clamps, etc. in addition to ordinary plain milling (or machining flat surfaces) it can also be used to cut gears of different types and in thread cutting, drilling, and boring operation. Many complicated operations such as indexing, gang milling, and straddle milling etc. can be carried out on a milling machine.

Milling machine is one of the most versatile conventional machine tools with a wide range of metal cutting capability. The number of operations which may be carried out on a given machine depends on the type of machine, type of cutter, and the accessories or attachments available for use with the machine. A large number of accessories are available for expanding the variety of operations which can be performed on these machines.

1.1.1 Methods Of Controlling Milling Machine

The method employed to control table movement in another way of classifying milling machines. Basically, there are four methods of master computer or computer on the machine control:

- **Manual:** All movement are made hand lever control
- **Semi-automatic:** movements are controlled by hand and /or by power feed
- **Fully Automatic:** A complex hydraulic feed arrangements follow two- or three-dimensional templates to guide the cutter automatically.
- **Computerized (CNC):** machining coordinates are entered into *Computer* using special programming language. Computer instructions electrically guide the cutter(s) through the required machining sequence. Table movement (feed) - can be engaged at cutting speed, however, there is a rapid traverse feed which gives fact power movement in any direction.

1.1.2 Designation of Milling Machine

All milling machines are identified by four basic factors: size, horsepower, model, and type.

The size of a milling machine is based on the longitudinal (from left to right) table travel, in inches. Vertical, cross, and longitudinal travel are all closely related as far as the overall capacity. However, for size designation, only the longitudinal travel is used. There are six sizes of knee-type milling machines, with each number representing the number of inches of travel.

Table 1.1. Size of milling

STANDARD SIZE	LONGITUDINAL TABLE TRAVEL
No. 1	22 inches
No. 2	28 inches
No. 3	34 inches
No. 4	42 inches
No. 5	50 inches
No. 6	60 inches

If the milling machine in the shop is labelled No. 2HL, it has a table travel of 28 inches; if it is labelled No. 5LD, it has a travel of 50 inches.

The horsepower designation refers to the rating of the motor which is used to power the machine. **The model** designation is determined by the manufacturer and features vary with different brands. **The type** of milling machine is designated as plain or universal, horizontal or vertical, and knee and column, or bed. In addition, machines may have other special type designations and, therefore, may not fit any standard classification.

1.2. Interpret Drawings

Drawings show the machinist what to make and identify the standards that must be followed so the various parts will fit together properly. The resulting parts will also be interchangeable with similar components on equipment already in service.

A very important skill needed for success in the machining field is the ability to interpret engineering drawings, or prints. Engineering drawings show the sizes and shapes of components and their specific features, such as holes, slots, or surfaces.

1.2.1. Orthographic Projection

An engineering drawing is a two-dimensional representation of a three-dimensional object. For that reason, each side of the object is shown by a different view. Each view represents how the object would appear when looked at from a certain perspective or position. By studying these views, the part can be visualized in three dimensions. The number of views in a blueprint is determined by the shape of the part and how complex it is.

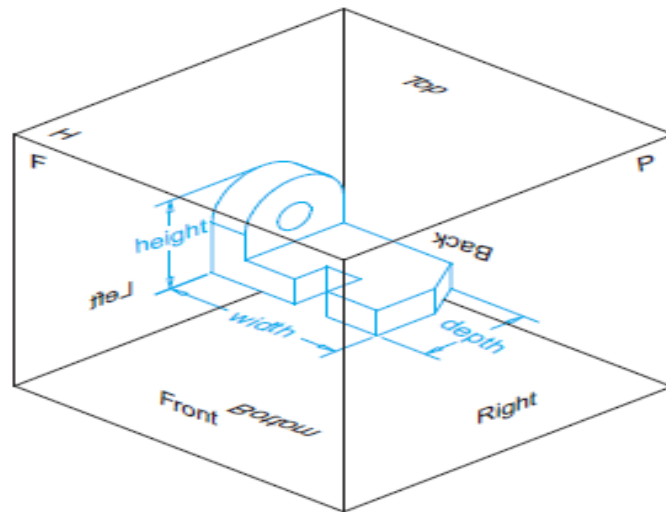


Figure 1.1 . Object placed inside glass box

1.2.2. Isometric View

Sometimes the views created through orthographic projection do not clearly show the shape of complex parts. To provide a better visualization of the part, a drawing may contain a three-dimensional view called an isometric view. Figure below shows a simple isometric view.

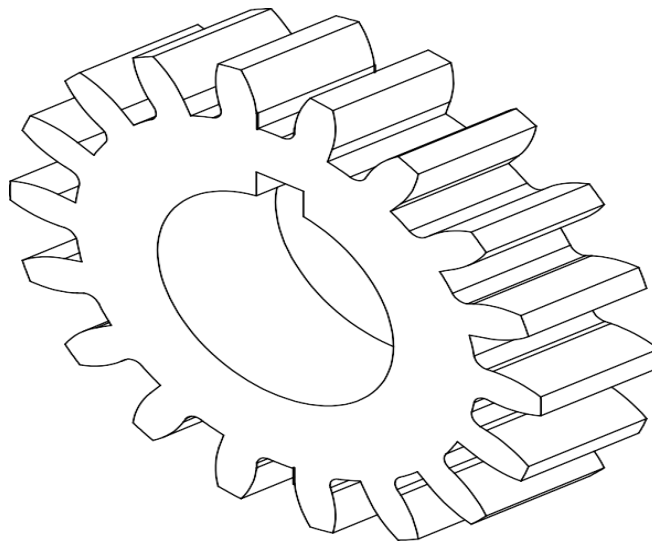


Figure 1.2. An isometric drawing

1.2.3. Oblique Drawing

Object lines in the oblique drawing also remain parallel. The oblique differs from the isometric in that one axis of the object is parallel to the plane of the drawing.

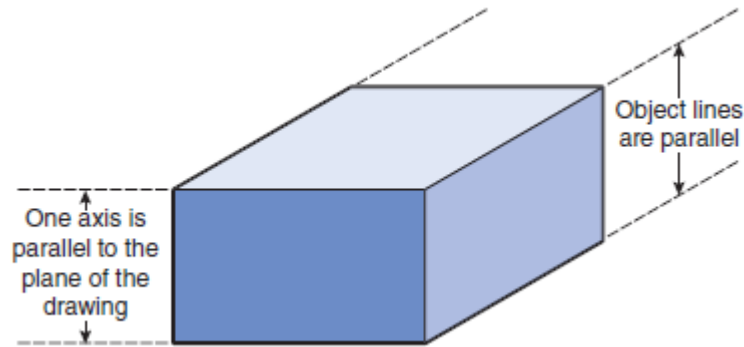


Figure 1.3. An oblique drawing

1.2.4. Types Of Line

Engineering drawings are made up of different styles of lines called line types. Each line type is used for a specific purpose. They are identified by the differences in their appearances. Line types are drawn as either thick or thin. Different types are also identified as continuous or broken, and by the size of the breaks.

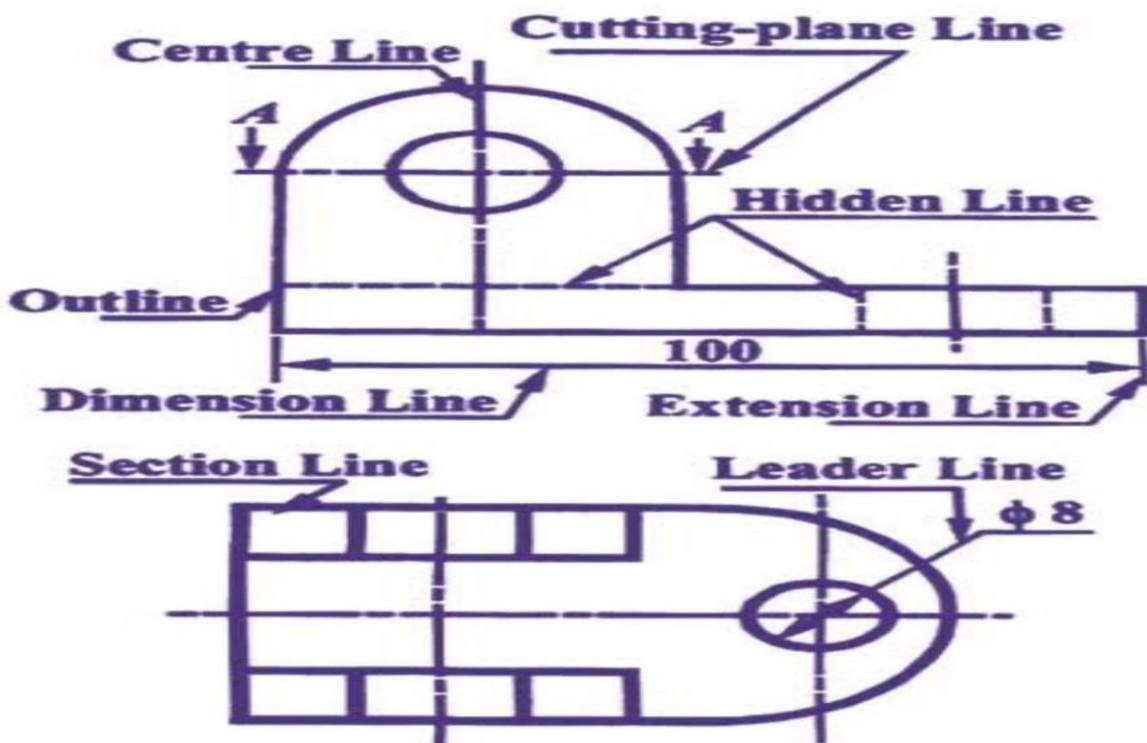


Figure 1.4. Alphabet of Lines

1.2.5. Dimensioning

Dimensioning is used on working drawings to explain to the machinist the shapes and sizes required to manufacture a part.

The type of material for the part, the number of parts required, and special notes are generally found in the title block of the drawing.

➤ Dimensioning symbols

The symbols are preferred because (1) they take less space in the drawing and (2) they are internationally recognized and therefore do not have translation issues if the part is manufactured in a country where a different language is spoken.

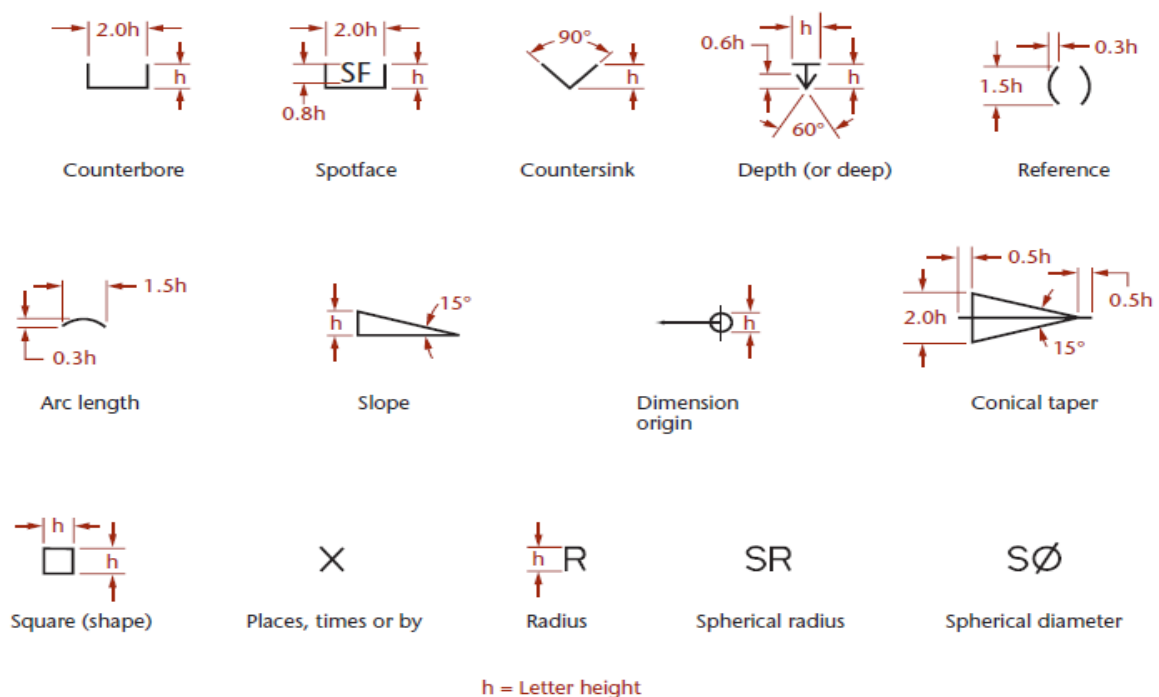


Figure 1.5. Form and Proportion of Dimensioning Symbols

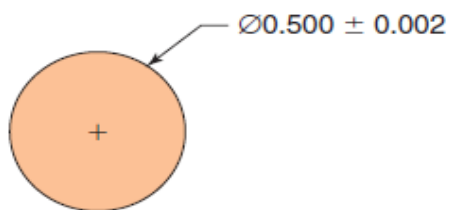
Tolerance

Recall, from the measurement section, that a tolerance is an allowable variation from a given size. A dimension shown on a print is called a basic size. A tolerance is applied to the basic size to determine the largest and smallest acceptable size for a dimension. The largest acceptable size is

often called the high limit, or upper limit. The smallest acceptable size is often called the low limit, or lower limit.

Bilateral Tolerances

A bilateral tolerance allows a dimension to vary both above and below basic size. A bilateral tolerance can take two forms. It can vary by equal amounts above and below basic size or by different amounts above and below basic size. The allowable amount above basic size is shown with a “+” symbol. The allowable amount below basic size is shown with a “-” symbol. When both amounts are equal, a “±” symbol is used.

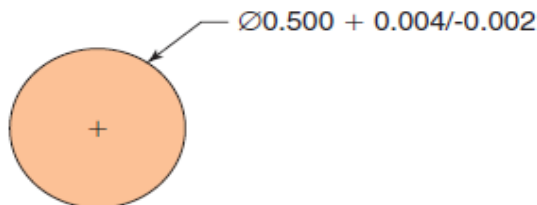


The 0.500 ± 0.002 shows an equal amount of allowable variation above and below basic size.

Upper limit (largest allowable size) = $0.500 + 0.002 = 0.502$

Lower limit (smallest allowable size) = $0.500 - 0.002 = 0.498$

Total tolerance = 0.502 (upper limit) – 0.498 (Lower limit) = 0.004



The $0.500 + 0.004/-0.002$ shows different amounts of allowable variation above and below basic size.

Upper limit (largest allowable size) = $0.500 + 0.004 = 0.504$

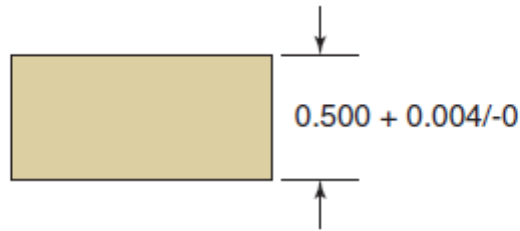
Lower limit (smallest allowable size) = $0.500 - 0.002 = 0.498$

Total tolerance = 0.504 (upper limit) – 0.498 (Lower limit) = 0.006

Figure 1.6. Bilateral tolerance

Unilateral Tolerances

A unilateral tolerance allows a dimension to vary either above or below basic size, but not both. A unilateral tolerance is also shown using a “+” or “-” symbol, but one amount is “0,” indicating no variation is allowable in that direction.



Upper limit (largest allowable size) = $0.500 + 0.004 = 0.504$
 Lower limit (smallest allowable size) = $0.500 - 0 = 0.500$
 Total tolerance = 0.504 (upper limit) - 0.500 (lower limit) = 0.004

Figure 1.7. Unilateral tolerance examples

Limit Tolerances

A limit tolerance does not use the “+,” “-,” or “±” symbol. Instead of a basic dimension being listed, the upper and lower limits are shown. They are usually separated by a bar or a slash. To determine total tolerance from a limit tolerance, simply subtract the lower limit from the upper limit.

Classes of Fit

Sometimes machining operations produce two mating parts, such as a shaft that fits inside a hub. If the print doesn't specifically call out the dimensions or tolerances, use of technical reference material may be required to determine the proper size ranges for those two mating parts. This relationship between the sizes of the two mating parts is called the class of fit. *Shaft A is a clearance fit, and shaft B is an interference fit.*

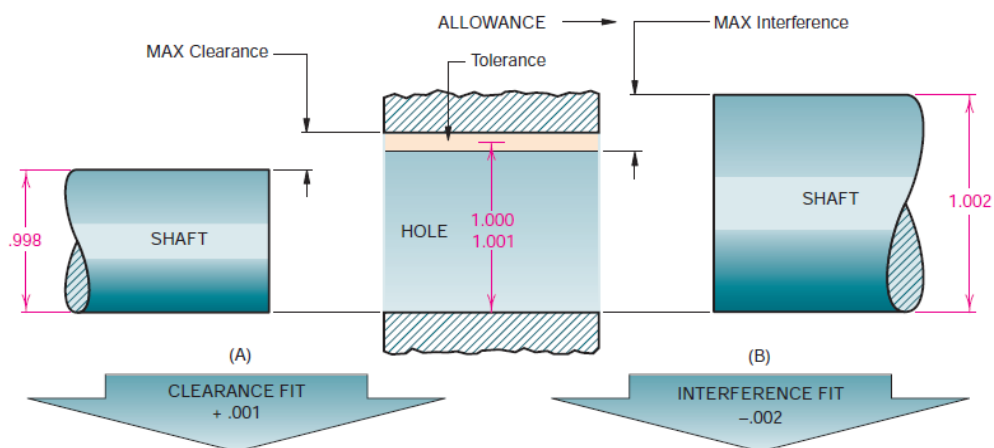


Figure 1.8. Transition fit between a shaft and a hole

When the shaft is machined to its smallest diameter (.998), there is a clearance fit with the hole. When the shaft is machined to its largest diameter (1.002), there is an interference fit with the hole.

Clearance fit occurs when two tolerance mating parts will always leave a space or clearance when assembled.

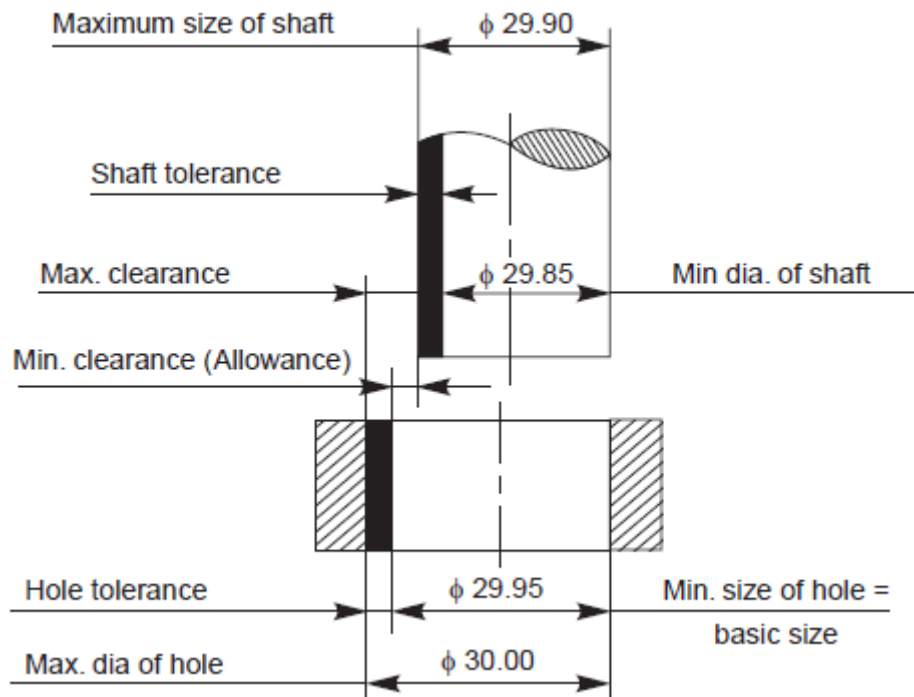


Figure 1.9. Clearance fit

Interference fit occurs when two tolerance mating parts will always interfere when assembled. Interference fit fixes or anchors one part into the other, as though the two parts were one. This means that the shaft will always be larger than the hole.

Transition fit occurs when two tolerance mating parts are sometimes an interference fit and sometimes a clearance fit when assembled.

1.2.5. Surface Finish

Surface finish refers to the roughness, waviness, lay, and flaws of a surface. Surface finish is the specified smoothness required on the finished surface of a part that is obtained by machining, grinding, honing, or lapping. Figure below shows the drawing symbol associated with surface finish.



Figure 1.10. Surface finish symbol

Surface Finish Symbol

Some of the surfaces of an object are machined to certain specifications. When this is done, a surface finish symbol is placed on the view where the surface or surfaces appear as lines, which is an edge view. The finish symbol on a machine drawing alerts the machinist that the surface must be machined to the given specification.

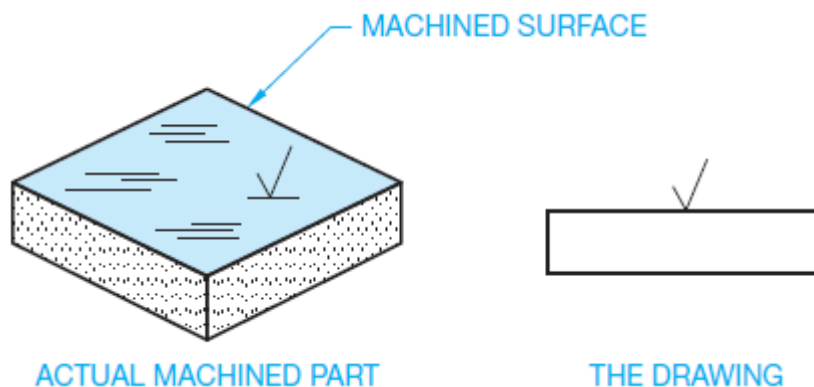


Figure 1.11. Standard surface finish symbol

1.3. Operational Sequences

It is a sequence of operation plan. Which is prepared based on the information provided on the job sheet or verbal instructions of the component to be machined has to ready for producing expected parts / components / with required specification. 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. A **work instruction** is a tool provided to help someone to do a job correctly. A Work Instruction is the most detailed description of a task. Its purpose is to explain step by step how to do a specific task or work. Work instructions are key to reducing variation, allowing manufacturers to improve quality and meet demand.

1.3.1. To Machine Squaring Stock

Several surfaces of a piece square with one another, as shown in figure below. Follow these steps:

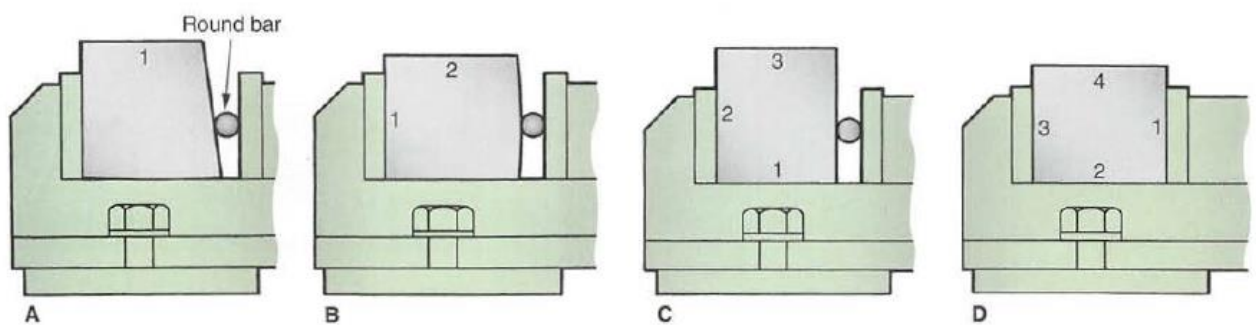


Figure 1.12. Sequence for squaring work

1. Machine the first surface.
2. Remove the burrs and place the first machined surface against the fixed vise jaw. Insert a piece of soft metal rod between the work and movable jaw if that portion of the work is rough or not square.
3. Machine the second surface.
4. Remove the burrs and reposition the work in the vise to machine the third side. This side must be machined to dimension. Take a light cut and use a micrometer to measure for size. The difference between this measurement and the required thickness is the amount of material that must be removed.
5. Repeat the above operation to machine the fourth side.

If the piece is short enough, the ends may be machined by placing it in a vertical position with the aid of a square, Figure A. Otherwise, it may be machined as shown in Figure B.

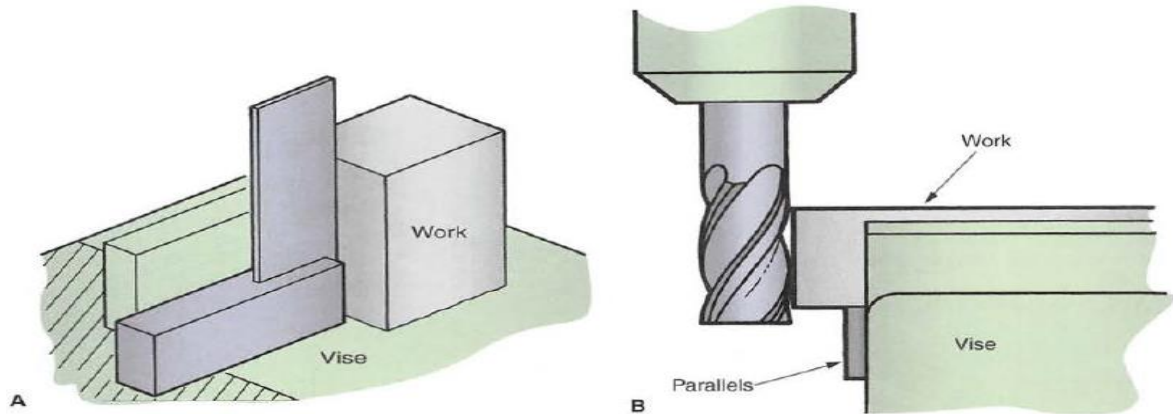


Figure 1.13.Squaring ends

1.3.2. To Machine an Angular Surface

- Lay out the angular surface.
- Clean the vise.
- Align the vise with the direction of the feed. This is of the utmost importance.
- Mount the work on parallels in the vise.

Swivel the vertical head to the required angle (Figure below).



Figure 1.14.Head swiveled to machine an angle

1. Tighten the quill clamp.
2. Start the machine and raise the table until the cutter touches the work. Carefully raise the table until the cut is of the desired depth.
3. Take a trial cut for about .50 in. (13 mm).
4. Check the angle with a protractor.
5. If the angle is correct, continue the cut.

6. Machine to the required depth, taking several cuts if necessary.

Note: It is always advisable to feed the work into the rotation of the cutter, rather than with the rotation of the cutter, which may draw the work into the cutter and cause damage to the work, the cutter, or both.

1.3.3. Alternate Method

Angles may sometimes be milled by leaving the head in a vertical position and setting the work on an angle in the vise (Figure below). This will depend on the shape and size of the work piece.



Figure 1.15. Machining an angle by adjusting the work piece

Procedure

1. Check that the vertical head is square with the table.
2. Clean the vise.
3. Lock the quill clamp.
4. Set the workpiece in the vise with the layout line parallel to the top of the vise jaws and about .250 in. (6 mm) above them.
5. Adjust the work under the cutter so that the cut will start at the narrow side of the taper and progress into the thicker metal.
6. Take successive cuts of about .125 to .150 in. (3 to 4 mm), or until the cut is about .030 in. (0.8 mm) above the layout line.
7. Check to see that the cut and the layout line are parallel.
8. Raise the table until the cutter just touches the layout line.
9. Clamp the knee at this setting.

10. Take the finishing cut.

1.3.4. Operations Sequence for a Sample Flat Part

The part shown in Figure below is used only as an example to illustrate a sequence of operations that should be followed when machining similar parts.

- These are not meant to be hard-and-fast rules, only guides.
- The part is relatively thin and has a large surface area.

Since at least .125 in. (3 mm) of work should be above the vise jaws, it would be difficult to use a round bar between the work and movable jaw for machining the large flat surfaces.

A small inaccuracy (out-of-squareness) on the narrow edge would create a greater error when the large surface was machined.

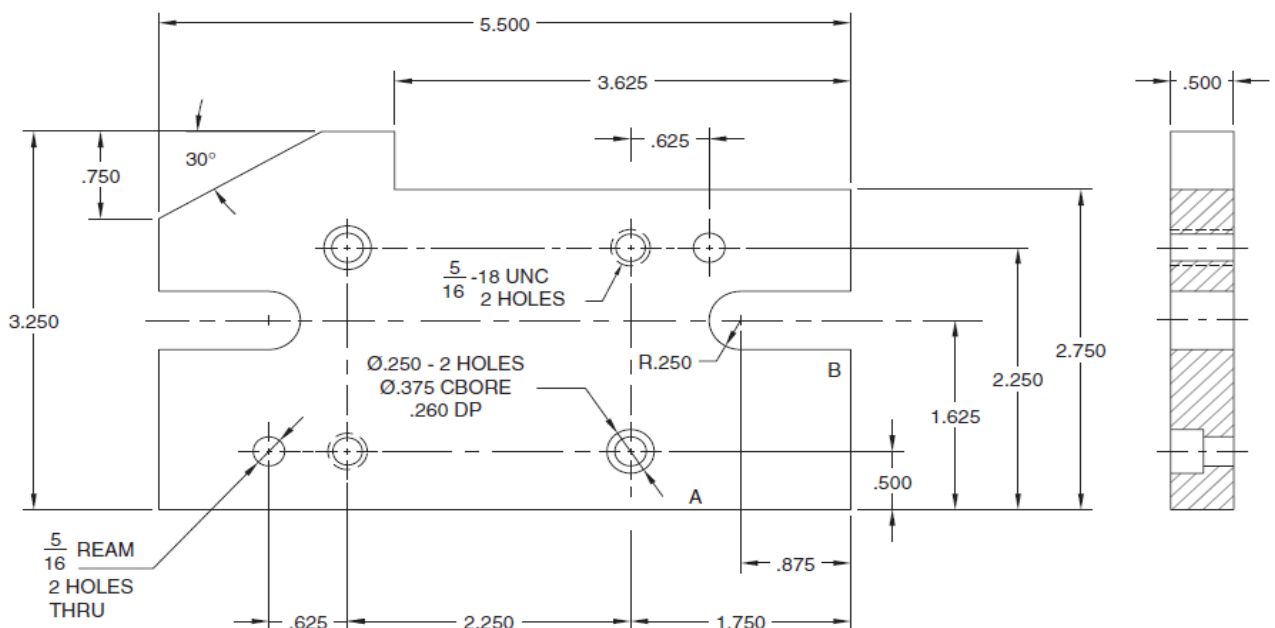


Figure 1.16.A typical flat part that must be laid out and machined

Procedure

1. Cut off a piece of steel .625 in. (16 mm) × 3.375 in. (86 mm) × 5.625 in. (143 mm) long.
2. In a milling machine, finish one of the larger surfaces (face) first.

Note: Leave .010 in. (0.25 mm) on each surface to be ground.

3. Turn the work piece over and machine the other face to .500 in. (13 mm) thick.
4. Machine one edge square with the face.

5. Machine an adjacent edge square (at 90°) with the first edge.
6. Place the longest finished edge (A) down in the machine vise and cut the opposite edge to 3.250 in. (83 mm) wide.
7. Place the narrower finished edge (B) down in the machine vise and cut the opposite edge to 5.500 in. (140 mm) long.
8. With edge A as a reference surface, lay out all the horizontal dimensions with an adjustable square, a surface gage, or a height gage (Figure below).

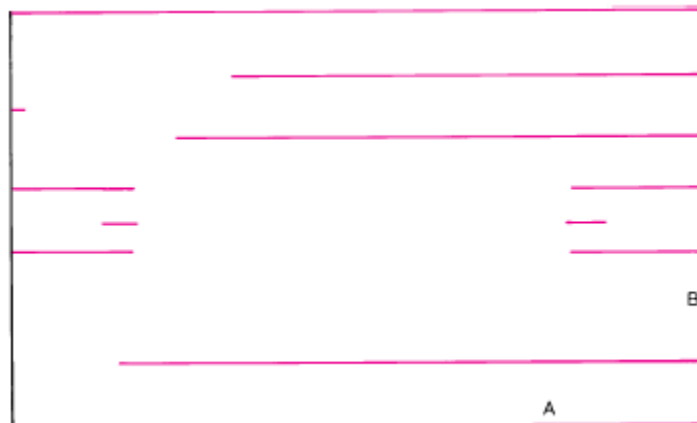


Figure 1.17.Lay out all horizontal lines using edge A as the reference surface.

9. With edge B as a reference surface, lay out all the vertical dimensions with an adjustable square, a surface gage, or a height gage (Figure below)



Figure 1.18.Lay out all vertical lines using edge B as the reference surface.

10. Use a bevel protractor to lay out the 30° angle on the upper right-hand edge.

11. With a divider set to .250 in. (6 mm), draw the arcs for the two center slots.
12. With a sharp prick punch, lightly mark all the surfaces to be cut and the centers of all hole locations.
13. Center-punch and drill 1/2-in. (13-mm) diameter holes for the two center slots.
14. On a vertical bandsaw, cut the 30° angle to within 0.30 in. (0.79 mm) of the layout line.
15. Place the workpiece in a vertical mill and machine the two .500-in. (13-mm) slots.
16. Machine the step on the top edge of the workpiece.
17. Set the work to 30° in the machine vise and finish the 30° angle.
18. 18. Prick-punch the hole locations, scribe reference circles, and then center-punch all hole centers.
19. Center-drill all hole locations.
20. Drill and counter bore the holes for the 1/4 in.–20 NC screws.
21. Tap drill the 5/16 in.-18 thread holes (F drill or 6.5 mm).
22. Drill the 1/4-in. (6-mm) ream holes to 15/64 in. (5.5 mm).
23. Countersink all holes to be tapped slightly larger than their finished size.
24. Ream the 1/4-in. (6-mm) holes to size.
25. Tap the 5/16 in.–18 UNC holes.

1.4. Milling Cutting Tools

1.4.1 Milling Cutters

The typical milling cutter is circular in shape with a number of cutting edges (teeth) located around its circumference. Milling cutters are manufactured in a large number of stock shapes, sizes, and kinds, because they can't be economically ground for a particular job as can a lathe cutter bit.

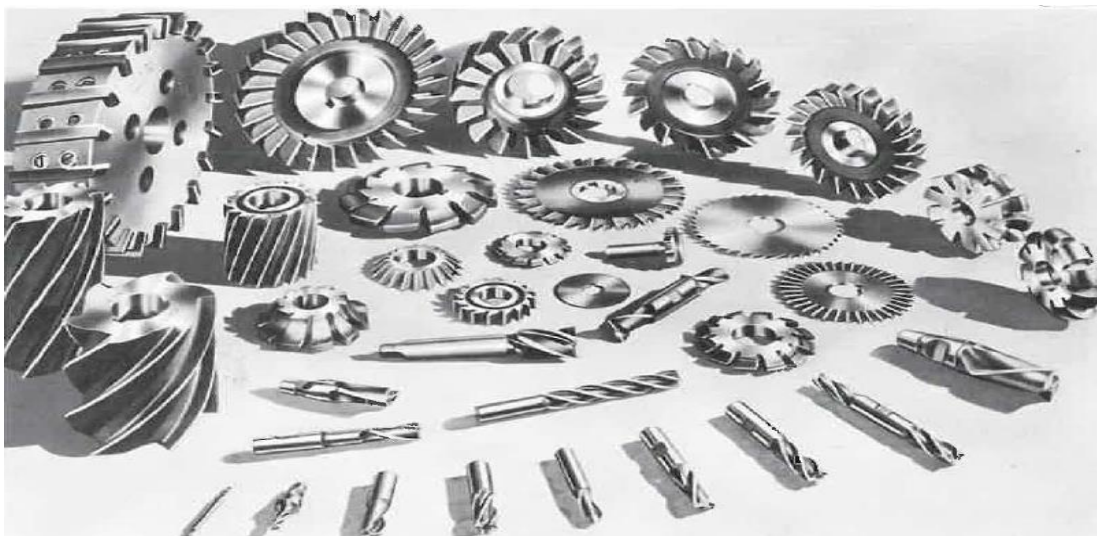


Figure 1.19.A selection of HSS (high speed steel) milling cutters

1.4.2. Types of milling cutters

There are two general types of milling cutters:

1. A **solid cutter** has the shank and body made in one piece



Figure 1.20.solid milling cutters.

2. The **inserted-tooth cutter** has teeth made of special cutting material, which are brazed or clamped in place. Worn and broken teeth can be replaced easily instead of discarding the entire cutter.



Figure 1.21.Index able carbide inserts.

1.4.3. How Milling Cutters Are Classified

Milling cutters are frequently classified by the method used to mount them on the machine:

Page 22 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
			October, 2023

- A. **Arbor cutters** have a suitable hole for mounting to an arbor.
- B. **Facing cutters** can be mounted directly to a machine's spindle nose or on a stub arbor.



Figure 1.22. Face-type milling cutter

1.4.4. Milling Cutter Material

Considering the wide range of materials that must be machined, the ideal milling cutter should have the following attributes:

1. **High abrasion resistance.** The cutting edges should not wear away rapidly due to the abrasive nature of some materials.
2. **Red hardness.** The cutting edges should not be affected by the terrific heat generated by many machining operations.
3. **Edge toughness.** The cutting edges should not readily break down due to the loads imposed on them by the cutting operation.

Since no single material can meet these requirements in all situations, cutters are made from materials that are, by necessity, a compromise.

High-speed steels (HSS) are the most versatile cutter materials. Cutters made from HSS are excellent for general purpose work and where vibration and chatter are problems. They are preferred for use on low-power machines.

HSS milling cutters can be improved by the application of surface lubricating treatments, surface hardening treatments, or coatings (such as chromium, tungsten, or tungsten carbide) to the cutting surfaces. The treated tools cost two to six times as much as conventional HSS tools, but they may last 5% to 10% longer or provide 50% to 100% higher metal removal rates with the same tool life.

Cemented tungsten carbides include a broad family of hard metals.



Figure 1.23. Inserted-tooth cutters

1.4.5. Types And Uses of Milling Cutters

Milling cutters are commonly grouped into categories based on their shape and function. Selection of the proper cutting tool is very important to efficient milling operations.

The following are the most commonly used milling cutters, with a summary of the work for which they are best suited

End mills

End milling cutters are designed for machining slots, keyways, pockets, and similar work. The cutting edges are on the circumference and end. End mills may have straight or helical flutes, and have straight or taper shanks. Straight shank end mills are available in single and double end styles.



Figure 1.24. End mills

Face milling cutters

Face milling cutters are intended for machining large flat surfaces parallel to the face of the cutter. The teeth are designed to make the roughing and finishing cuts in one operation. Because of their size and cost, most face milling cutters have inserted cutting edges.



Figure 1.25. Index able inserts

Fly-cutters

A fly-cutter is a single-point tool often consisting of a high speed or carbide tool secured in an appropriate holder. Although a fly-cutter is not truly an end mill, it is used for end milling applications. Fly-cutters are often used to take light face cuts from large surface areas. The tool bit in the fly-cutter must be properly ground to obtain the correct rake and clearance angles for the material being machined..



Figure 1.26. Single-point cutting tool

Arbor milling cutters

The more common arbor milling cutters and the work for which they are best adapted include:

Plain milling cutter: - Plain milling cutters are cylindrical, with teeth located around the circumference. Plain milling cutters less than 3/4" (20 mm) are made with straight teeth. Wider plain cutters, called slab cutters, are made with helical teeth designed to cut with a shearing action. This reduces the tendency for the cutter to chatter.

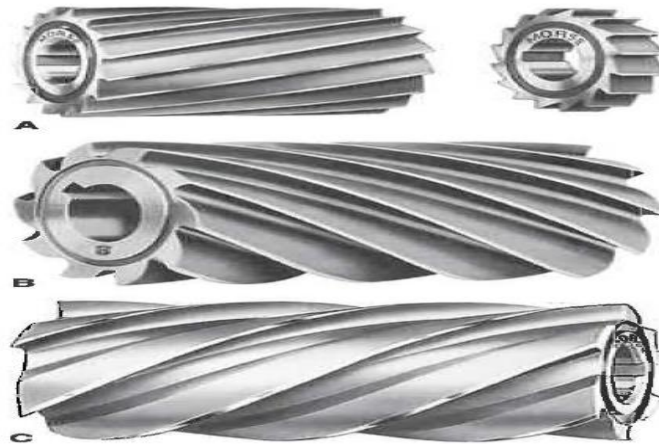


Figure 1.27. Plain milling cutters.

Side milling cutter: Cutting edges are located on the circumference and on one or both sides of side milling cutters. They are made in solid form or with inserted teeth.

Angle cutters: Angle cutters differ from other cutters in that the cutting edges are neither parallel, nor at right angles to the cutter axis.

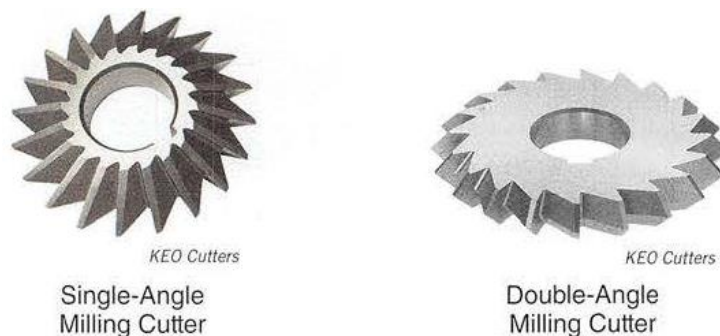


Figure 1.28. Angle Cutters.

Metal slitting saws: Metal slitting saws are thin milling cutters that resemble circular saw blades. They are employed for narrow slotting and cutoff operations. Slitting saws are available in diameters as small as 2 1/2" (60 mm) and as large as 8" (200 mm).

Formed Milling Cutters: Formed milling cutters are employed to accurately duplicate a required contour. A wide range of shapes can be machined with standard cutters available. See Figure below. Included in this cutter classification are the concave cutter, convex cutter, corner rounding cutter, and gear cutter.

1.4.6. Selection of Milling Cutters

Consider the following when choosing milling cutters:

- High-speed steel, satellite, and cemented carbide cutters have a distinct advantage of being capable of rapid production when used on a machine that can reach the proper speed.
- 45° angular cuts may either be made with a 45° single-angle milling cutter while the work piece is held in a swivel vise, or with an end milling cutter while the work piece is set at the required angle in a universal vise.
- The harder the material, the greater will be the heat that is generated in cutting. Cutters should be selected for their heat-resisting properties.
- Use a coarse-tooth milling cutter for roughing cuts and a finer-toothed milling cutter for light cuts and finishing operations.

Care Of Milling Cutter

Milling cutters are expensive and can be easily damaged if care is not taken in their use and storage. The following recommendations will help extend cutter life:

- Use sharp cutting tools. Machining with dull tools results in low-quality work and eventually damages the cutting edges to such an extent that they cannot be salvaged by grinding.
- Properly support tools and make sure the work is held rigidly.
- Use the correct cutting speed and feed for the material being machined.
- Ensure sufficient supply of cutting fluid.
- Use the correct cutter for the job.
- Store cutters in individual compartments or on wooden pegs. They should never come in contact with other cutters or tools.

1.5. Work Shop Safety

1.5.1. Safety Procedures

Safety precautions for specific operations are detailed in the text along with the description of the operation. Learn each operation thoroughly before attempting to carry it out. All safety precautions must be observed when working with the milling machine. Follow all safety instructions detailed in the workshop and make safe working practice a habit.

Instructions on how to operate milling machine are given in this TTLMas a guide to correct working. They are not intended to give the one best way of working. Be guided by your instructor in systematic, safe, and efficient working practice suitable for your workshop..

1.5.2. Safety Rules

Safety Rules and Procedures in Milling Machine Operation

Personal safety

- Dresses appropriately remove necktie, necklace, wrist, watch, rings and other jewelleryes and loose fitting. Sweater wear and apron or a properly shop fitted coat and safety glasses are a must.
- Be sure all guards are in place before attempting to operate the machine
- When cleaning the milling, do not remover chips with bare hands, an air hose should never be used to remove chips. The flying particles might injure you or nearby person.
- Do not operate machines while taking medication because of possible drawness.
- Stop the machine before trying to remove accumulated chips.

Machine safety

- Become thoroughly familiar with the milling machine before attempting to operate it. When in doubt, obtain additional instructions.
- Stop the machine before attempting to make adjustments or measurements!
- Be sure all power to the machine is turned off before opening or removing guards and covers.
- Avoid using compressed air to remove chips and cutting oil from machine.
- Keep the machine clear of tools

1.5.3. Safety Rules For Milling Machines

- Do not make contact with the revolving cutter.
- Place a wooden pad or suitable cover over the table surface to protect it from possible damage.
- Use the buddy system when moving heavy attachments.
- Do not attempt to tighten arbour nuts using machine power.
- When installing or removing milling cutters, always hold them with a rag to prevent cutting your hands.
- While setting up work, install the cutter last to avoid being cut.
- Never adjust the work piece or work mounting devices when the machine is operating.

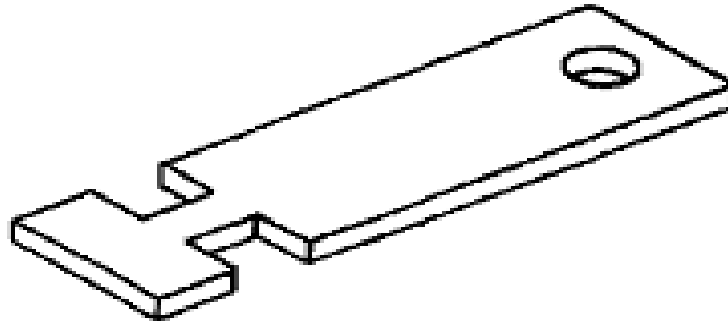


Figure 1.29. Chip rake.

Before beginning any job on the milling, you should thoroughly study and understand the blueprint or drawing from which you are to work.

Work shop safety

- Avoid horse play!
- Keep the floor around your machine clear of chips and wipe up spilled cutting fluid immediately! Place sawdust or special oil absorbing compound on slippery floors.
- Oily rugs must be placed in approved safety containers. (metal can with metal lid)

Safety when Machining Work Piece

- Do not operate any machine before understanding its mechanism.
- Always stop a machine before measuring, cleaning or making any adjustments. It is dangerous to do any type of work around moving parts of a machine.
- Never operate a machine unless all safety guards are in place

Self-Check 1

Directions: Attempt to answer all the questions listed below. Illustrations may be necessary to aid some explanations/answers. Note: Be precise and scientific in writing.

Part I: Multiple Choices

Instructions: choose the correct answer and circle the letter of your choice from the given alternatives.

1. Which one of the following is not type of Plain milling machine?
 - (a) Floor-mounted Horizontal Milling Machine
 - (b) Bench-type Horizontal Milling Machine
 - (c) Universal milling machine
 - (d) Gang milling machine

1. Drawings are used to:
 - a) Apply safety procedures
 - b) To identify machine type
 - c) Show what to make and the sizes to make it.
 - d) To operate milling.

2. Tolerances are:
 - a) The different materials that can be used.
 - b) Allowances in either oversize or undersize that a part can be made and still be acceptable.
 - c) Dimensions.
 - d) All of the above..

3. Print reading falls into two general categories. They are
 - a) Visualization and tolerance
 - b) Interpretation and dimensioning
 - c) Visualization and interpretation
 - d) Projection and orthographic

4. when tolerances are plus and minus, it is called a _____ tolerance
 - a) Bilateral tolerance
 - b) Unilateral tolerance

5. Which one of the following is not characteristics of work shop safety rule?
- A. Keep the machine clear of tools
 B. Oil rugs must be placed in approved safety containers
 C. avoid horse play
 D. wear hand gloves
6. Accidents may not take place due to _____
- A. Human causes
 B. Mechanical causes
 C. Environmental causes
 D. nature

Part II: Matching: For the following questions match from “B” to “A” column

No.	<u>“A” Column</u>	<u>“B” Column</u>
_____	Fixed bed type Milling machine	A. characterized by a vertical adjustable worktable resting on a saddle supported by a knee
_____	Column and knee type Milling machine	B. characterized by very rigid work table construction and support
_____	Vertical milling machine	C. having the cutter spindle at right angle to the top of the work table

Part III –True or false

Instructions: For the following questions say **True** or **False**.

1. Milling cutters are a **solid cutter** that has the shank and body made in one piece,
2. A vertical milling machine which is of similar construction to a horizontal milling machine except that the spindle is mounted in the vertical position
3. The machine should not be operated at speed higher than specified
4. Unguarded moving parts of machines/equipment and the sudden or uncontrolled release of their power systems can result in serious injuries
5. Loose clothing, using jewellery while operating machine which has moving parts not so much causes of accidents.

PART IV: Short answer

Instructions: Write the answer for the following questions properly. If it is necessary add some explanations or examples on the answers.

1. List five types milling cutters with their functions?
2. How will you classify milling machines?
3. What are the methods that can be employed to control table movement of milling machines?
4. What are the factors should be considered in the selection of milling cutters

Page 32 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
----------------	--	-------------------------------------	------------

Unit Two: Set-Up Machine and Work Piece

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

- Machine and work set up

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:

- Perform Machine and work set up
- Identify Work holding devises
- Set up equipment and instrument

2.1. Machine and Work Set Up

2.1.1. Introduction

The selection of material for milling depends on various factors, including the desired outcome, the specific milling operation, and the capabilities of the milling machine. Here are some important considerations when choosing materials for milling:

Work piece requirements: Consider the characteristics and requirements of the final work piece. This includes factors such as strength, hardness, durability, corrosion resistance, and thermal properties..

Machinability: Evaluate the machinability of the material, which refers to how easily and efficiently it can be machined. Factors such as cutting forces, tool wear, chip formation, and surface finish should be considered. Some materials are more difficult to machine due to their hardness, toughness, or tendency to generate excessive heat during milling.

Material properties: Different materials have unique properties that impact the milling process. For example, metals like aluminum, steel, and titanium are commonly used due to their high strength and versatility.

Tooling compatibility: Ensure that the selected material is compatible with the cutting tools available. Different materials require specific types of cutting tools, coatings, and geometries to achieve optimal milling results. Consider the tool's ability to withstand the material's hardness, abrasiveness, and other properties.

Cost considerations: Evaluate the cost of the material, as it can significantly impact the overall project budget. Some materials may be more expensive due to their scarcity, unique properties, or manufacturing processes. Balance the material cost with the desired outcome and project requirements.

Experience and expertise: Consider your familiarity and expertise in machining specific materials. Different materials require different machining strategies, speeds, feeds, and cutting parameters. It's important to note that this is a general guideline, and the specific material selection will depend on your unique project requirements.

2.1.2. Materials Used On Milling

Classifying Metals

There are many different types of metals used in the machining industry, depending on the product being produced. Because of this large variety of metal types, workers in the field may be exposed to a wide range of those metals with very different characteristics. A good starting point is to first divide metals into two major categories: ferrous and nonferrous.

1. Ferrous metals are metals that contain iron. Most ferrous metals are magnetic. One way to remember ferrous is to think about the symbol for the element iron, Fe, since Fe is the beginning of the word ferrous. Iron ore is found naturally in the earth in combination with oxygen..

2. Nonferrous metals are metals that contain no iron. Aluminum, copper, magnesium, and titanium are examples of nonferrous metals.

2.1.3. Setting up the milling

Setting up a milling machine involves a series of steps to ensure that the machine is properly configured and ready for operation. Here's a general guide on how to set up a milling machine:

Safety Precautions: Before starting any setup procedure, ensure that you follow proper safety precautions. This includes wearing appropriate personal protective equipment (PPE) such as safety glasses, hearing protection, and gloves

Machine Positioning: Place the milling machine in a suitable location in your workshop or manufacturing facility. Ensure that the machine is stable and level to prevent any vibrations or movement during operation.

Power and Utilities: Connect the milling machine to a suitable power source and ensure that it is grounded properly. Check if the coolant system and any other utilities required for the machine are available and connected.

Tooling and Work holding: Select the appropriate tooling, such as end mills or face mills, based on the machining operation and material being cut. Install the tooling securely into the spindle using the proper collet or tool holder. Set up the work holding device, such as a vise or fixture, to securely hold the work piece during machining.

Machine Alignment: Use precision measuring tools, such as dial indicators or laser alignment systems, to check and adjust the machine's alignment. This includes checking the square ness of the spindle to the table, trimming the mill head (ensuring it is perpendicular to the table), and checking for any misalignment in the X, Y, and Z axes.

Work piece Setup: Position the work piece in the work holding device, ensuring that it is securely clamped or fixture. Take measurements and alignments as necessary to ensure the work piece is properly positioned and aligned for the desired machining operations.

Tool and Work piece Clearances: Verify that there is sufficient clearance between the tool, work piece, and any machine components to prevent interference during machining. Adjust the machine's travel limits and tooling approach accordingly.

Spindle Speed and Feed Rate Selection: Determine the appropriate spindle speed and feed rate based on the material being machined, tooling, and desired cutting conditions. Refer to cutting speed charts, tooling catalogs, or machining guidelines for recommended parameters.

Coolant Setup: If using a coolant system, ensure that it is properly filled and connected. Adjust the coolant flow and positioning to effectively cool the cutting tool and remove chips from the machining area.

2.1.4. Work Holding

Selection of a work-holding device and method will depend on the machining task to be done. If you are clamping directly to the mill table, be sure to follow the rules of good clamping. Use a work stop if necessary to prevent work piece slippage from cutting pressure.

Mill Vises: A mill vise is an accurate and dependable work holding tool. When milling only on the top of the work piece, it is not necessary to set up the vise square to the column or parallel to the table. However, if the work piece has already machined outside surfaces, steps, or grooves, the vise must be precisely aligned to the machine table.

Parallel-to-Table Alignment

Page 36 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
----------------	--	-------------------------------------	------------

Step 1 Fasten a dial indicator on a magnetic base and attach to the arbor or over arm. Be sure that over arm clamps are tight. Position the indicator to contact the solid jaw of the vise. Preload the indicator about half a revolution.

Step 2 Move the table by hand so that the indicator is positioned at one end of the solid jaw. Set the bezel to zero. Crank the tables so that the vise jaw moves past the indicator tips, and note the reading at the opposite end of the jaw.

Step 3 Tap the vise gently with a soft hammer so that half of the total indicated runout is canceled (back toward zero on the indicator). When tapping the vise, move it in such a direction that the solid jaw moves away from the indicator. Moving the jaw against the indicator tip can damage the delicate indicator by shocking the indicator movement. Reset the bezel to zero.

Step 4 Crank the table back and observe the indicator reading. If a zero reading is obtained, tighten the hold-down bolts securely and recheck the alignment.

Perpendicular-to-Table Alignment

A vise may be aligned at right angles to the table by the technique discussed previously. Once again, always indicate on the solid jaw and move the saddle to carry the vise jaw past the indicator. Always recheck after tightening hold-down bolts.



Figure 2.1 Aligning vise square to table travel

Squaring a Vise to the Column

A vise may be aligned by squaring the solid jaw to the column. Two paper strips may be used as feeler gages between the beam of the square and the vise jaw. This method should not be used when an accurate alignment is required.



Figure 2.2. Using a try square to align a vise on the milling machine table.

Securing the Work piece A vise will effectively secure a work piece in most cases. Whenever possible, set up the vise so that cutting pressure is applied to the solid jaw.

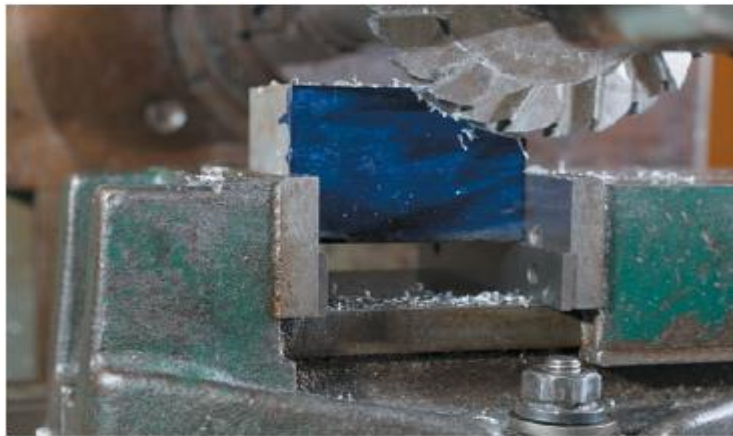


Figure 2.3. Cutting pressure against solid jaw

Avoid applying cutting pressure against the movable jaw. If the work piece is sufficiently high, it may be seated on the bottom of the vise. If not, parallels may be used to elevate the work to a point where it can be machined. In many cases it will be necessary to apply the cutting pressure parallel to the vise jaws. Remember that friction between the vise jaws and work piece holds it in place.



Figure 2.4 Work piece on parallels held in a vise

Self-Check 2.

Part I: Multiple Choices

Instructions: For each of the following question choose the best correct answer and circle the letter of your choice from the given alternatives.

1. Which one of the following instrument is used for aligning the vise parallel to the table travel.

- A. hammer
B. steel rule
C. dial indicator
D. micrometer

2. Which type of metal is containing iron?

- A. aluminum
B. copper
C. ferrous metal
D. nonferrous metal

Part II - Instructions: For the following questions say **True** or **False**.

1. The machine table and all sliding surfaces should be cleaned prior to setup or operation.
2. Any burrs found on the table or work-holding devices should be removed with a honing stone.
3. Milling vise is not an accurate and dependable work holding tool.
4. Dial indicator can be used in setup milling machine
5. It is recommended that you can change feeds or speeds while the milling machine is in operation

6. Before setting up a job, be sure that the work piece, table, the taper in the spindle, and the arbor or cutter shank are free from chips, nicks, or burrs

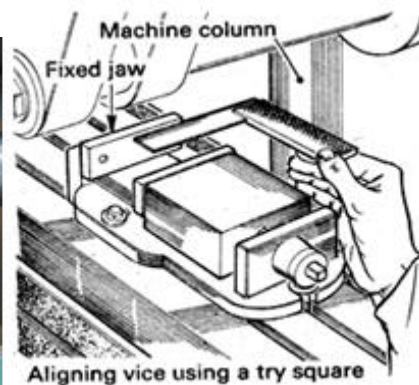
Operation Sheet 2.1.

Operation Title: Vise Parallel to the Table

Purpose: To align the vise parallel to the table travel

Required tools and equipment:

1. Dial indicator with stand
2. Try square
3. Mahelet



Milling machine, machine vise, dial indicator with magnetic stand, rag or brush

Procedure

Page 40 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
----------------	--	-------------------------------------	------------

1. Clean the surface of the table and the base of the vise.
2. Mount and fasten the vise on the table.
3. Swivel the vise until the solid jaw is approximately parallel with the table slots.
4. Mount an indicator on the arbor or cutter
5. Make sure the solid jaw is clean and free from burrs.
6. Adjust the table until the indicator registers about one-quarter of a revolution against a parallel held between the jaws of the vise.
7. Set the bezel to zero (0).
8. Move the table along for the length of the parallel and note the reading of the indicator Compare it to the zero (0) reading at the other end of the parallel.
9. Loosen the nuts on the upper or swivel part of the vise.
10. Adjust the vise to half the difference of the indicator readings by tapping it with your hand or with a soft-faced hammer in the appropriate direction.
11. Recheck the vise for alignment and adjust, if necessary, until there is no movement in the indicator as it is moved along the parallel.

Operation Sheet 2.2.

Operation Title: Milling Machine Setups

Purpose: To Setting up the Machine

Tools required:

Milling machine, masonit, a board, or a bench, wrench, brush or rag

To prolong the life of a milling machine and its accessories and to produce accurate work, the following actions should be taken when milling machine setups are made:

1. Prior to mounting any accessory or attachment, check to see that both the machine surface and the accessory are free from dirt and chips.
2. Do not place the tools, cutters, or parts on the milling machine table. Place them on a piece of mason it, a board, or a bench kept for this purpose to prevent damaging the table or machined surfaces.
3. When mounting cutters, be sure to use keys on all but slitting saws.
4. Check that the arbor spacers and bushings are clean and free from burrs.

Page 41 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
----------------	--	-------------------------------------	------------

5. When tightening the arbor nut, take care to make it only hand tight with a wrench.

When work is mounted in a vise, tighten the vise securely by hand and tap it into place with a LEAD OR SOFT-FACED HAMMER.

Lap Test 2.1.

Practical Demonstration

Name: _____

Date: _____

Time started: _____

Time finished: _____

Instruction I: Given necessary templates, tools and materials you are required to perform the following tasks within 2 hours.

Task 1: Aligning the milling machine vise parallel to the table travel

Task 2: Check the vise for alignment as it is moved along the parallel

Task 3: Hold the work piece square to the Table and a Machine Vice

Task 4: Mount the cutter in the center of arbor

Task 5: Hold the work piece on universal vise than tighten

Task 6: Aligning the milling machine vise parallel to the table travel

Task 7: Check the complete setup, as given on the drawing.

Unit Three: Advanced Milling Operations

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

- Calculating cutting parameter
- Calculate measurement
- Milling accessories.
- Advanced Milling 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:

- Calculate cutting parameter
- Calculate measurement
- Use Milling accessories.
- Perform Advanced Milling operations

3.1. Calculating Cutting Parameter

3.1.1. Calculating speeds and feeds

Although there are many different types of milling cutter, understanding chip formation is fundamental to the use of any of them. As the milling cutter rotates, the material to be cut is fed into it, and each tooth of the cutter cuts away small chip of material. Achieving the correct size of chip is of critical importance. The size of this chip depends on several variables.

Surface cutting speed (V_c): This is the speed at which each tooth cuts through the material as the tool spins. This is measured either in meters per minute in metric countries, or surface feet per minute (SFM) in America. Typical values for cutting speed are 10m/min to 60m/min for some steels, and 100m/min and 600m/min for aluminium. This should not be confused with the feed rate.

Spindle speed (S): This is the rotation speed of the tool, and is measured in revolutions per minute (rpm). Typical values are from hundreds of rpm, up to tens of thousands of rpm.

Diameter of the tool (D):

Feed per tooth (F_z): This is the distance the material is fed into the cutter as each tooth rotates. This value is the size of the deepest cut the tooth will make.

Feed rate (F): This is the speed at which the material is fed into the cutter. Typical values are from 20mm/min to 5000mm/min.

Depth of cut: This is how deep the tool is under the surface of the material being cut (not shown on the diagram). This will be the height of the chip produced. Typically, the depth of cut will be less than or equal to the diameter of the cutting tool.

The machinist needs three values: **S**, **F** and **Depth** when deciding how to cut a new material with a new tool. However, he will probably be given values of V_c and F_z from the tool manufacturer. **S** and **F** can be calculated from them.

Cutting variables are the most important factors which affect the efficiency of the milling machine.

These are:

- Cutter speed
- Feed &
- Depth of cut

Table 3.1. C.S. and feed rate for some common material

Tool Material	High Speed Steel		Carbide	
Material	Cutting Speed	Feed (f)	Cutting Speed	Feed (f)
Mild Steel	25	0.08	100	0.15
Aluminum	100	0.15	500	0.3
Hardened Steel	---	---	50	0.1

Selecting Proper Cutting Speed

The approximate values given in table 3.1 [Milling machine cutting speeds for High speed steel milling cutters (based on cutter diameter)] on the following page may be used as a guide for selecting the proper cutting speed.

Table 3.2 lists speeds for high-speed steel milling cutters. If carbon steel cutters are used, the speed should be about one-half the speed recommended in the table. If carbide-tipped cutters are used, the speed can be doubled.

If a plentiful supply of cutting oil is applied to the milling cutter and the work piece, the speeds can be increased from 50% to 100%.

Table 6.4 is providing to facilitate spindle speed computations for standard cutting speeds and standard milling cutters.

If the cutter runs too slowly or too fast it causes wastage of time or cutter wear. The rate at which the work fed into the revolving cutter is also important. If the work is fed slowly, time will be wasted and chatter may occur, which shortens the life of the cutter.

Table 3.2. Standard cutting speeds and standard milling cutter

Material	Cutting speed (sfpm) ^{1 2}			
	Plain milling cutters		End milling cutters	
	Roughing	Finishing	Roughing	Finishing
Aluminium	400 to 1,000	400 to 1,000	400 to 1,000	400 to 1,000
Brass, composition	125 to 200	90 to 200	90 to 150	90 to 150
Brass, yellow	150 to 200	100 to 250	100 to 200	100 to 200
Bronze, phosphor and manganese	30 to 80	25 to 100	30 to 80	30 to 80
Cast iron (hard)	25 to 40	10 to 30	25 to 40	20 to 45
Cast iron (soft and medium)	40 to 75	25 to 80	35 to 65	30 to 80
Molten metal	50 to 75	50 to 75	40 to 50	40 to 60
Steel, hard	25 to 50	25 to 70	25 to 50	25 to 70
Steel, soft	60 to 120	45 to 110	50 to 85	45 to 100

For carbon steel cutters, decrease values by 50 percent.

For carbide-tipped cutters, increase values by 100 percent.

Factors which should be considered in selecting the feed:

- (a) Forces are exerted against the work piece, the cutter, and their holding devices during the cutting process. The force exerted varies directly with the amount of metal removed and can be regulated by the feed and the depth of cut.
- (b) The feed and depth of cut also depend upon the type of milling cutter being used. For example, deep cuts or coarse feeds should not be attempted when using a small diameter end milling cutter, as such an attempt would spring or break the cutter. Coarse cutters with strong cutting teeth can be fed at a faster rate because the chips may be washed out more easily by the cutting oil.
- (c) Coarse feeds and deep cuts should not be used on a frail workpiece, or on a piece that is mounted in such a way that its holding device is not able to prevent springing or bending.
- (d) The degree of finish required often determines the amount of feed. Using a coarse feed, the metal is removed more rapidly but the appearance and accuracy of the surface produced may not reach the standard desired for the finished product.

Cutting speed

Cutting speed: The speed, in surface feet per minute (sf/min) or meters per minute (m /min) of which the metal may be machined efficiently.

The speed of a milling cutter is the distance in feet (meter) per minute that each tooth travels as it cuts its chips (which the metal may be machined efficiently). The number of spindle revolutions per minute necessary to give a desired peripheral speed on the size of the milling cutter.

The best speed is determined by the type of material being cut and the size and type of cutter used. The smoothness of the finish desired, the depth of cut being taken and the rigidity of the machine (power available) and work set up are other factors relating to the cutter speed.

The speed used on a milling machine depends up on a variety of factors; such as:

- The type of work material being cut
- The size and type of cutter used
 - The cutter material
 - The diameter of the cutter'
- The surface finish required
- The depth of cut being taken
- the rigidity of the machine (power available) and
- work set up

Speed Computation

Spindle Speed of milling machine

Spindle speed in revolution per minute (R.P.M.) for the cutter can be calculated from the equation:

The formula used to find the work speed in mm (for metric calculation)

$$N = \frac{CS \times 1000}{\pi d}$$

$$Rpm = \frac{cs \times 100}{D \times 3.14} = \frac{cs \times 320}{D}$$

Where, N = R.P.M. of the cutter

CS = Linear Cutting Speed of the material meter per minute (in m/min.) (see table 1)

d = Diameter of cutter in mm

Looking at the formula for the **spindle speed**, S, it can be seen that larger tools require lower spindle speeds, while small tools may be able to go at high speeds.

$$S = \frac{1000V_c}{\pi D}$$

Example:

Page 48 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
----------------	--	-------------------------------------	------------

1. Calculate the r/min required for a 75 mm diameter high speed steel milling cutter when cutting machine steel (CS 30 m/min)

$$r / \text{min} = \frac{1000V}{\pi D} = \frac{320 \times 30}{75} = 128$$

Solution:

$$\text{rpm} = \frac{30 \text{ m/min} \times 320}{75 \text{ mm}} = \frac{9600}{75} = 128$$

The formula used to find the work speed in inch

$$r / \text{min} = \frac{CS(ft)}{\text{circumference(in)}} \quad (\text{for inch calculation})$$

$$r / \text{min} = \frac{12 \times CS}{\pi D}$$

D – diameter of the cutter

CS – cutting speed

2. Calculate the speed required to revolve a 3 in diameter high-speed steel milling cutter when cutting machine steel (90 ft/min)

$$r / \text{min} = \frac{CS(ft)}{\pi D} = \frac{90}{3 \times 3.1416} \text{ or } \frac{CS \times 4}{D}$$

Since the CS is in ft and diameter is in inch, the feet must be multiplied by 12.

$$\text{There for } r / \text{min} = \frac{90 \times 12}{3 \times 3.1416} = \text{_____ } r / \text{min}$$

2. at what speed should be a 2in. diameter carbide cutter to mill a piece of cast iron (CS150 ft/mm)?

$$r / \text{min} = \frac{4 \times 150}{D} = \frac{4 \times 150}{2} = 300 r / \text{min}$$

Feed

Milling machine feed may be defined as the distance in inches or millimetres per minute that the work moves into the cutter or the rate at which the work moves in to the revolving cutter and it is measure either in inches per minute or millimetre per minute.

The feed rate used on milling machine depends up on a variety of factors, such as

- The depth and width of cut

- The design or type of cutter
- The sharpness of the cutter
- The work piece material
- The strength and the uniformity of the work piece
- The type of finish and accuracy required.

Feed Rate calculation

Feed rate (F) is defined as the rate of travel of the work piece in mm/min. But most tool suppliers recommend it as the movement per tooth of the cutter (f).

- The rate of feed is the speed at which the work piece passes the cutter.
- It determines the time required for cutting a job.

The formula for the **feed rate**, F shows that increasing S or z gives a higher feed rate. Therefore, machinists may choose a tool with the highest number of teeth that can still cope with the swarf load.

$$F = zSF_z$$

Inch calculation

$$\text{Feed (in/min)} = N \times \text{cpt} \times \text{r/min}$$

$$\text{Feed} = \text{No. of cutter teeth} \times \text{chip /tooth} \times \text{cutter r/min.}$$

Where, N- Number of teeth in the milling cutter

cpt - chip per tooth for a particular cutter and metal (find from table)

r/min - revolutions per minute of the milling cutter

Thus, $F = f \cdot u \cdot N$

Where F = table feed in mm/min

f = movement per tooth of cutter in mm (see table 1)

u = number of teeth of cutter

N = R.P.M. of the cutter

Example:

Find the feed per minute using a 3.5 in. diameter 12 teeth helical cutter to cut machine steel (CS 80)

$$\text{Solution : } r / \text{min} = \frac{4 \times \text{CS}}{D} = \frac{4 \times 80}{3.5} = 91$$

$$\begin{aligned} \text{Feed (in / min)} &= N \times \text{Cpt} \times r / \text{min} \\ &= 12 \times 0.010 \times 91 = 10.9 = 11 \end{aligned}$$

Page 50 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
----------------	--	-------------------------------------	------------

From table for helical cutter to cut machine steel = 0.010

Metric calculations

The formula used to calculate the feed in mm/min. Is:-

$$\text{Feed}(\text{rev}/\text{min}) = N \times \text{cpt} \times r/\text{min}$$

Where, N = number of teeth on the milling cutter

Cpt = chip per tooth for a particular cutter and metal (find from table)

r/min = revolution per minute of milling cutter

Example:

Calculate the feed in millimetres per minute for a 75 mm diameter, six-tooth helical milling cutter (HSS) when machining a cast iron work piece (CS 60)

Solution:

First calculate the rpm of the cutter

$$\text{Rpm} = \frac{\text{cs} \times 320}{D}$$

$$r/\text{min} = \frac{\text{CS} \times 320}{D} = \frac{60 \times 320}{75} = 256$$

$$\begin{aligned} \text{Thus, feed (mm/min)} &= N \times \text{cpt} \times \text{rpm} \\ &= 6 \times 0.18 \times 256 \\ &= \underline{276 \text{ mm/min}} \end{aligned}$$

Cpt = cut per teeth; from table 6.6, 0.25 is found for helical cutter to machine cast iron.

Thus, Feed (mm/min) = 6 x 0.25 x 256 = 384 mm/min

Table 3.3. Recommended feed per tooth (for high speed steel cutters)

Material	Face mills		Helical Mills		Slotting and side mills		End mills		Form relived Cutters	
	In	Mm	In	Mm	In	Mm	In	Mm	In	mm
Aluminium	0.022	0.55	0.018	0.45	0.013	0.33	0.011	0.28	0.007	0.18
Brass & Bronze	0.014	0.35	0.011	0.28	0.008	0.20	0.007	0.18	0.004	0.10
Machine steel	0.012	0.30	0.010	0.25	0.007	0.18	0.006	0.15	0.004	0.10
Tool steel (medium)	0.010	0.25	0.008	0.20	0.006	0.15	0.005	0.13	0.003	0.08
Stainless steel	0.006	0.15	0.005	0.13	0.004	0.10	0.003	0.08	0.002	0.05
Cast iron	0.013	0.33	0.010	0.25	0.007	0.18	0.007	0.18	0.004	0.10

Typical Feeds

- Feed for milling cutters will generally run from 0.002 to 0.250 inch per cutter revolution, depending upon the diameter of the cutter, the kind of material, the width and depth of the cut, the size of the work piece, and the condition of the machine.
- Good finishes may be obtained using a 3-inch plain milling cutter at a 40 feet per minute speed, with a feed of 0.040-inch per cutter revolution.

3.2. Calculate Measurement

3.2.1. How to calculate measurements

To calculate measurements in milling operations, there are several key parameters you need to consider. Here's a step-by-step guide on how to calculate some common measurements in milling operations:

Feed per Tooth (fz): Feed per tooth refers to the distance the cutting tool advances with each revolution. It is calculated using the following formula:

$$fz = \text{Feed rate (in mm/min)} / (\text{Number of teeth on the cutter} * \text{Spindle speed in RPM})$$

Chip Load (CL): Chip load is the thickness of the material removed with each cutting edge of the tool during one revolution. It is calculated using the following formula:

$$CL = fz * \text{Number of cutting edges on the tool}$$

Material Removal Rate (MRR): Material removal rate is the volume of material removed per unit of time. It is calculated using the following formula:

$$MRR = \text{Feed rate (in mm/min)} * \text{Axial depth of cut (in mm)} * \text{Radial depth of cut (in mm)}$$

Cutting Speed (v): Cutting speed refers to the speed at which the cutting tool moves across the workpiece. It is calculated using the following formula:

$$v = \text{Spindle speed (in RPM)} * \text{Tool diameter (in mm)} * \pi / 1000$$

Spindle Speed (N): Spindle speed is the rotational speed of the spindle, which determines the cutting speed. It is calculated using the following formula:

$$N = (\text{Cutting speed (in m/min)} * 1000) / (\text{Tool diameter (in mm)} * \pi)$$

These calculations provide you with some basic measurements and parameters used in milling operations. However, please note that there are other factors to consider, such as tool geometry, material properties, and machine capabilities, which may affect the final results.

3.3. Milling Accessories

3.3.1. Types of Milling Machines

Page 52 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
----------------	--	-------------------------------------	------------

It is difficult to classify the various categories of milling machines because their designs tend to merge with one another. For practical purposes, however, milling machines may be grouped into two large families:

- Fixed-bed milling machines
- Column-and-knee milling machines

Both groups are made with horizontal or vertical spindles. On a horizontal milling machine, the cutter is fitted onto an arbor mounted in the machine on an axis parallel with the worktable. Multiple cutters may be mounted on the spindle for some operations.

The cutter on a vertical milling machine is normally perpendicular (at a right angle) to the worktable. However, on many vertical milling machines, the spindle can be tilted to perform angular cutting operations.

Fixed-Bed Milling Machines

Fixed-bed milling machines have a very rigid worktable construction and support. The worktable moves only in a longitudinal (back and forth/X-axis) direction, and can vary in length from 3' to 30' (0.9 to 9.0 m). Vertical (up and down/Z-axis) and cross (in and out/Y-axis) movements are obtained by moving the cutter head.

Column-and-Knee Milling Machines

The column-and-knee milling machine is so named because of the parts that provide movement to the work piece. They consist of a column that supports and guides the knee in vertical (up and down/Z-axis) movement and a knee that supports the mechanism for obtaining table movements. These movements are traverse (in and out/ Y-axis) and longitudinal (back and forth/X-axis).

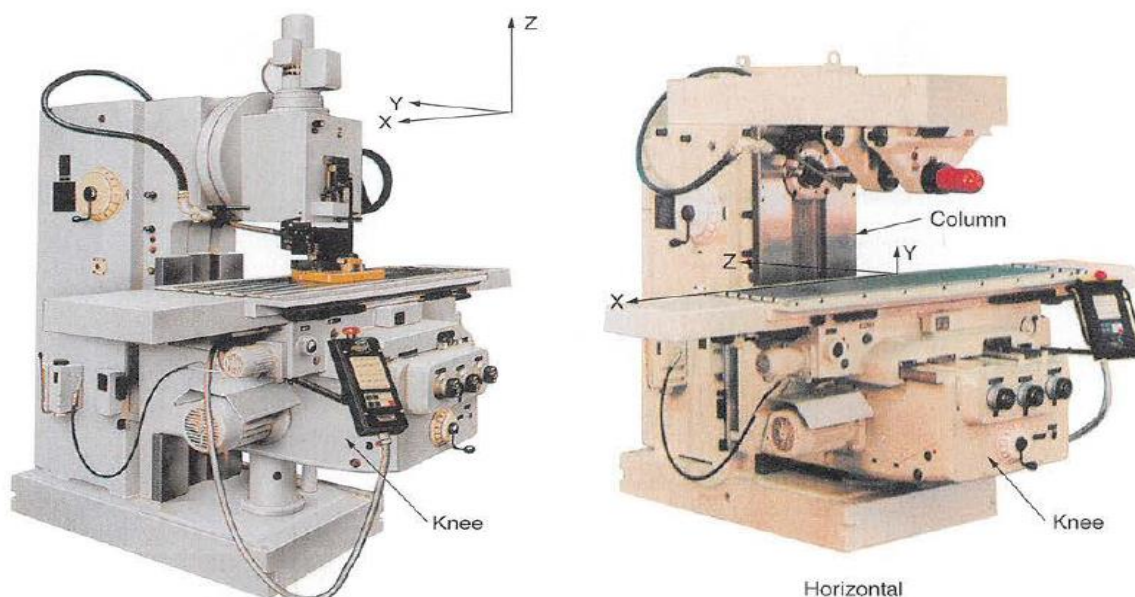


Figure 3.1. Vertical

Figure 3.2. Horizontal

These machines are commonly referred to as knee-type milling machines. The three basic categories of knee-type milling machines are plain (horizontal) milling machines, universal milling machines, and vertical milling machines.

Plain Milling Machine

On the plain milling machine, the cutter spindle projects horizontally from the column. The worktable has three movements: vertical, cross, and longitudinal (X, Y, and Z axes).

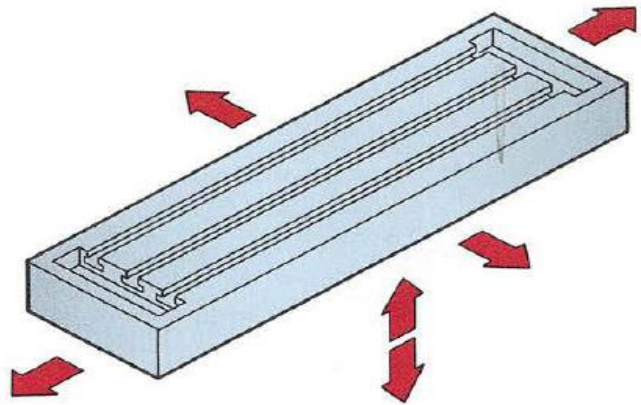


Figure 3.3. Plain mil ling horizontal machine Figure 3.4. Plain horizontal milling machine.

Universal Milling Machine

A universal milling machine is similar to the plain milling machine, but the table has a fourth axis of movement. On this type of machine, the table can be swiveled on the saddle through an angle of 45° or more. This makes it possible to produce spiral gears, spiral splines, and similar work pieces.

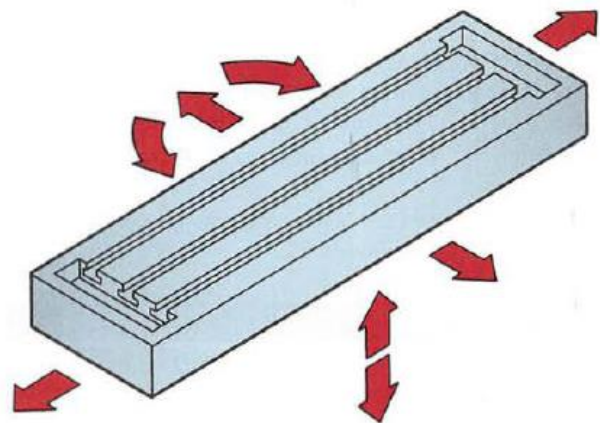


Figure 3.5. universal horizontal milling machine Figure 3.6. Table movements a universal milling machine

Vertical Milling Machine

A vertical milling machine differs from the plain and universal machines in that its cutter spindle is vertical, at a right angle to the top of the worktable. The cutter head can be raised and lowered by hand or by power feed. This type of milling machine is best suited for use with an end mill or face mill cutter.



Figure 3.7. The vertical mil ling machine

Types of vertical mills include swivel-head, sliding-head, and rotary-head mills. A swivel-head milling machine is the type often found in training programs. The spindle can be swiveled for angular cuts. On the sliding-head milling machine, the spindle head is fixed in a vertical position. The head can be moved up and down (vertically) by hand or under power.

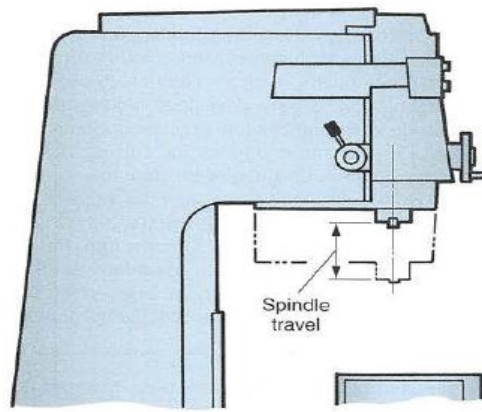


Figure 3.8. The spindle head is fixed in a vertical position on a sliding-head milling machine.

The spindle on the rotary-head milling machine can be moved vertically and in circular arcs of adjustable radii about a vertical centerline. It can be adjusted manually or under power feed.

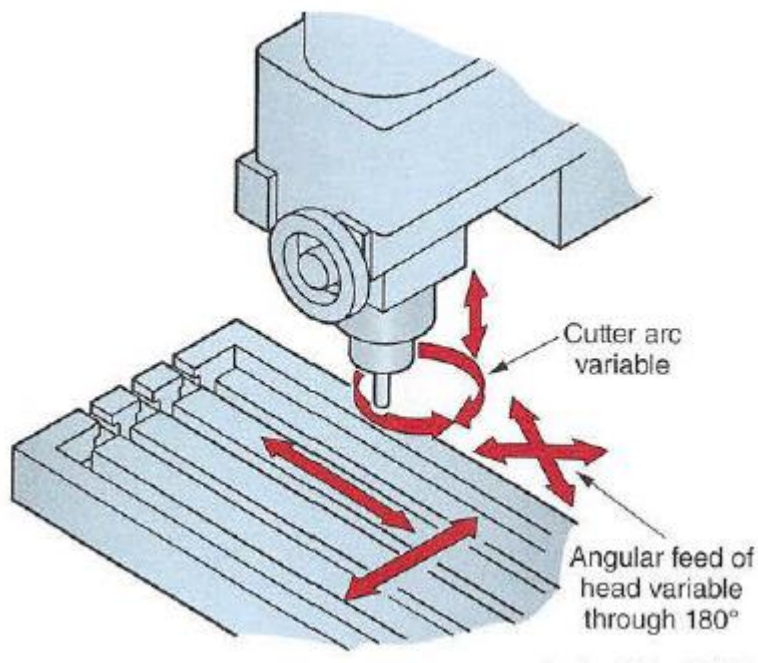


Figure 3.9. Spindle movements possible on a rotary-head vertical mil ling machine.

Milling work-holding attachments

One of the more important features of the milling machine is that it can be used with a large number of work-holding attachments. Each of these attachments increases the usefulness of the milling machine.

- A. Hold-Down Clamps:** Often the size or shape of a part makes it difficult to hold in a vise but the volume of work does not justify the expense of creating a custom fixture. Clamps are an extremely universal method for securing a workpiece to a machine table. Unfortunately, holding work with hold-down clamps provides no accurate provision for repeated locating from part to part.

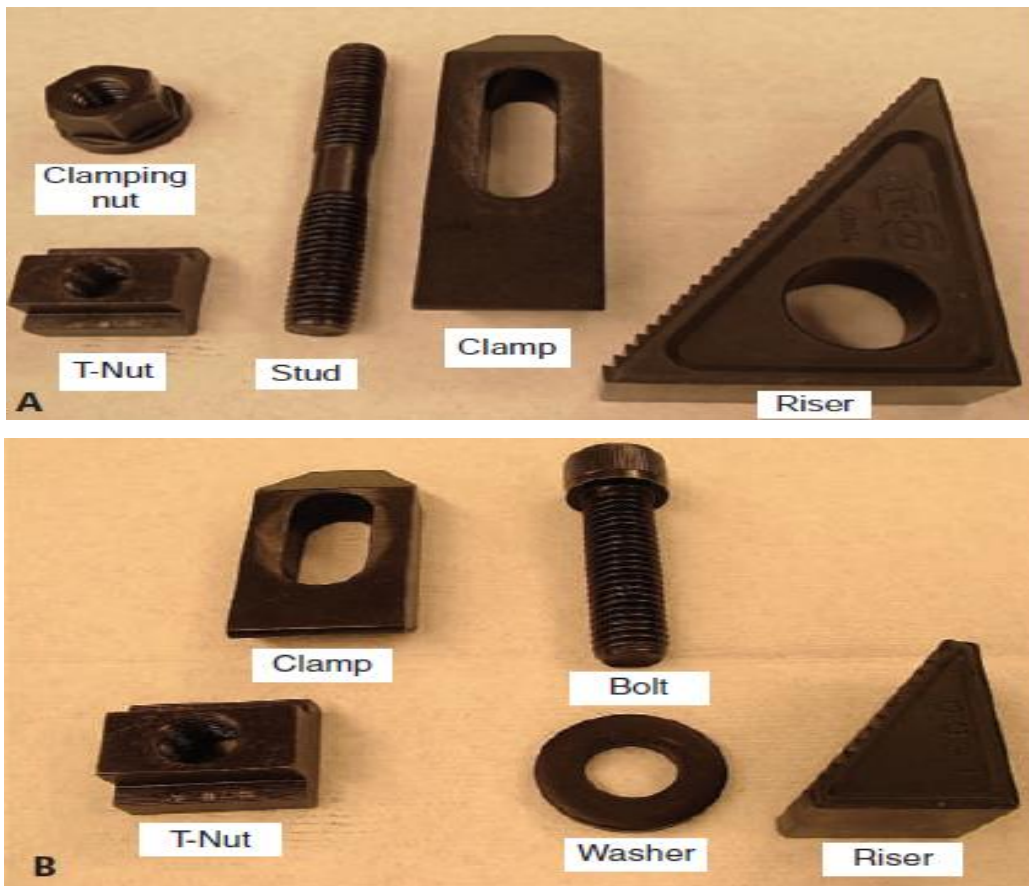


Figure 3.10. (A) A step clamp assembly usually contains a T-nut, stud, clamp, riser block, and clamping nut. (B) A bolt and washer may be used in place of the stud and nut



Figure 3.11. A typical step clamp set with T-nuts, clamping nuts, and assorted sizes of studs, clamps, and riser blocks.

Vise: The vise is probably the most widely employed device for holding work for milling. The jaws are hardened to resist wear, and they are ground for accuracy. A milling vise, like other work-holding attachments, is keyed to the table slot with lugs.

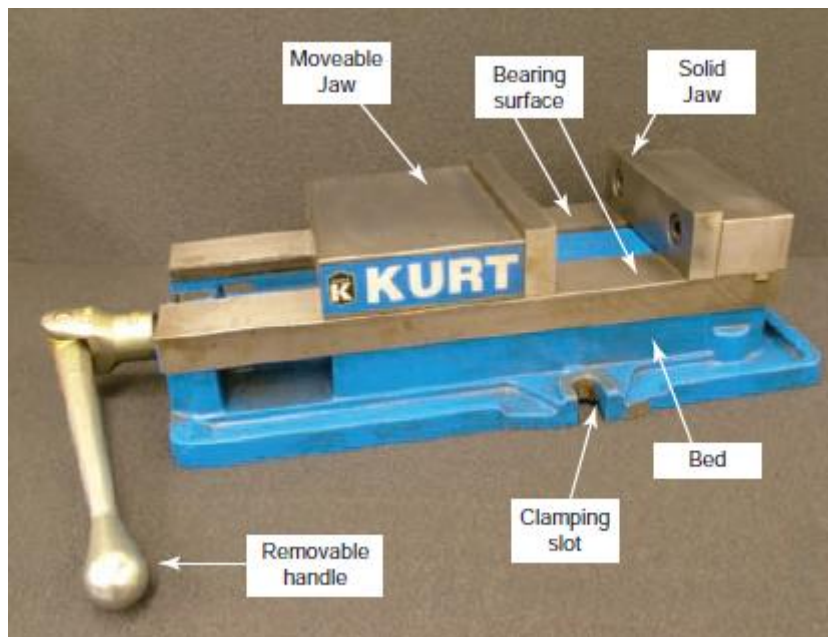


Figure 3.12. A typical milling machine vise and its parts

V-Blocks

V-Blocks hold and support round work for milling or drilling (Figure 3.12). V-Blocks come in many different sizes. On milling machines, V-Blocks are typically clamped directly on the table (Figure 3.13).



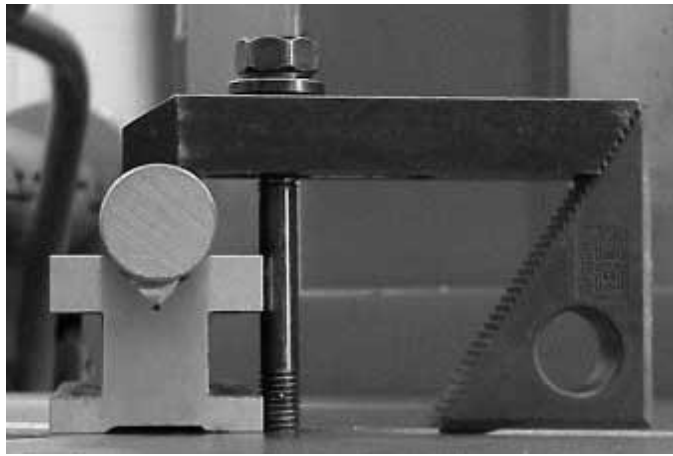


Figure3.13: AV-Block and a strap clamp being used to clamp a round part to the table.

Angel plates

An angle plate is an L shaped piece of Cast Iron or Steel that has tapped holes or slots to facilitate the clamping of the workpiece (Figure 9). Angle plates are used when parts need to have machining operations performed at a 90 degree angle to the axis of the table (Figure 3.14).



Figure 3.14: Angle Plates



Figure 3.15. Angle plate

Mounting to the table

Work that is too large or has an odd configuration is usually bolted directly to the table (Figure 3.16). This method of work holding takes the mosting equity and expertise. There are a number of accessories that can be used to help you setup the work piece.

Angling clamps incorrectly put pressure on the support, not the workpiece.

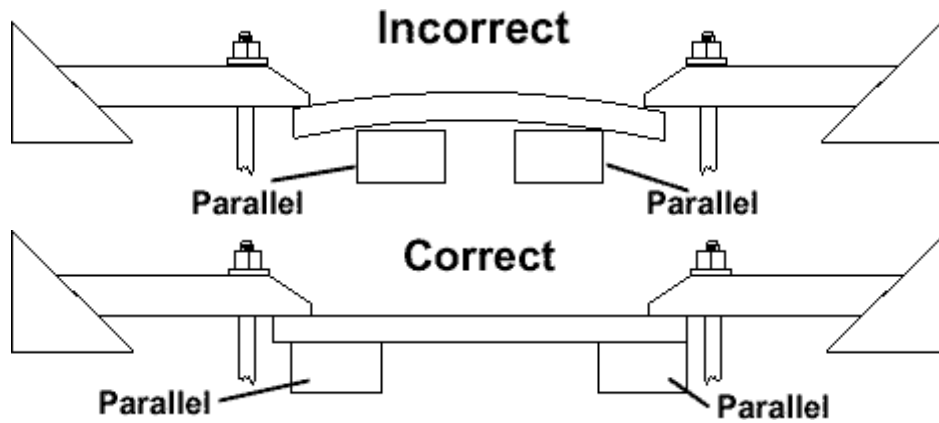


Figure 3.16. correct and incorrect holding devices

Specialty Vises: Special vises are available for angular work holding. The angle vise has a base plate that supports a hinged bed. This vise may be hinged and locked in a desired angle for milling operations. Many angle vises have a graduated support arm that can be used to visually set the vise at a desired angle.



Figure 3.17 An angle vise can be moved on a hinge and then locked to position work at nearly any desired angle

Sine vise: The sine vise is a specialty vice for milling angles of higher precision. The sine vise is similar to the angle vise, but has a special base that resembles a sine bar. This vise may be set to precision angles using gage blocks in the same manner as a sine bar is set.

Chucks/Collet Fixtures: A jaw-type chuck, similar to the type used on the lathe, or a collet fixture may be used on the milling machine to hold and locate work pieces. In vertical machines these devices may be mounted flat on their back so that parts are positioned vertically. The chucks may also be used on an indexing device either vertically or horizontally so that features may be machined around a part's periphery in angular increments or patterns such as hole circles.



Figure 3.18. A three-jaw chuck fixture for milling

Magnetic Chuck: A magnetic chuck, is ideally suited for many milling operations. The magnet eliminates the need for time-consuming hold-down clamps to mount the work to the table. The magnetic chuck can be used only with ferrous metals.

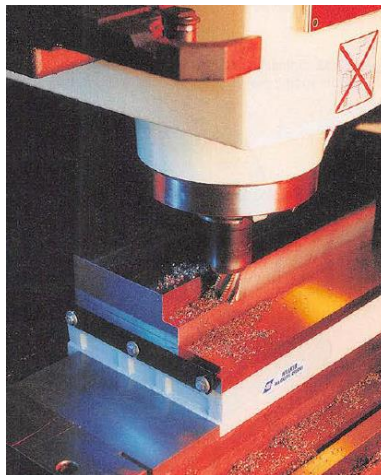


Figure 3.19. Work secured on a magnetic chuck for milling

Rotary and Index Tables: A rotary table can perform a variety of operations, including cutting segments of circles, circular slots, and cutting irregular-shaped slots. A dividing attachment can be fitted to many rotary tables in place of the hand wheel. The table is graduated in degrees around its circumference. Adjustments can be made accurately with the hand wheel to within 1/30 of a degree (2 minutes).

An index table permits the rapid positioning of work. Indexing is usually performed in 15° increments. However, a clamping device allows the table to be locked at any setting.

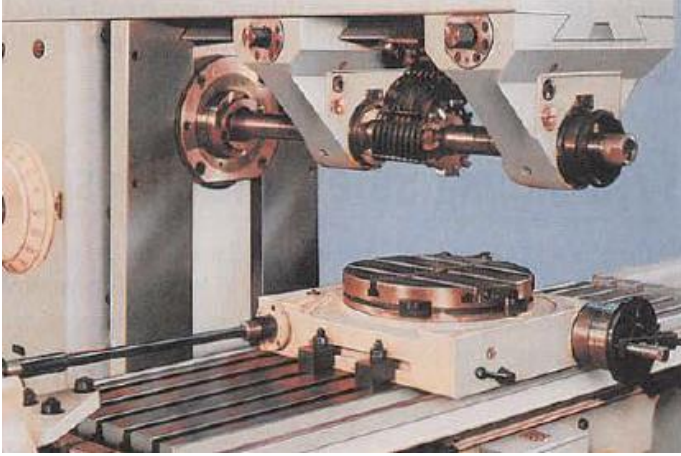


Figure 3.20 . A rotary table



Figure 3.21. Index table

Dividing Head: A dividing head divides the circumference of circular work into equally spaced units. This feature makes a dividing head indispensable when milling gear teeth, cutting splines, and spacing holes on a circle. It also makes possible the milling of squares, hexagons, and various other regular shapes. A dividing head is a precision device that has an indexing accuracy of about one minute of arc. This is the equivalent of $1/21,600$ part of a circle.



Figure 3.22. Dividing or indexing head

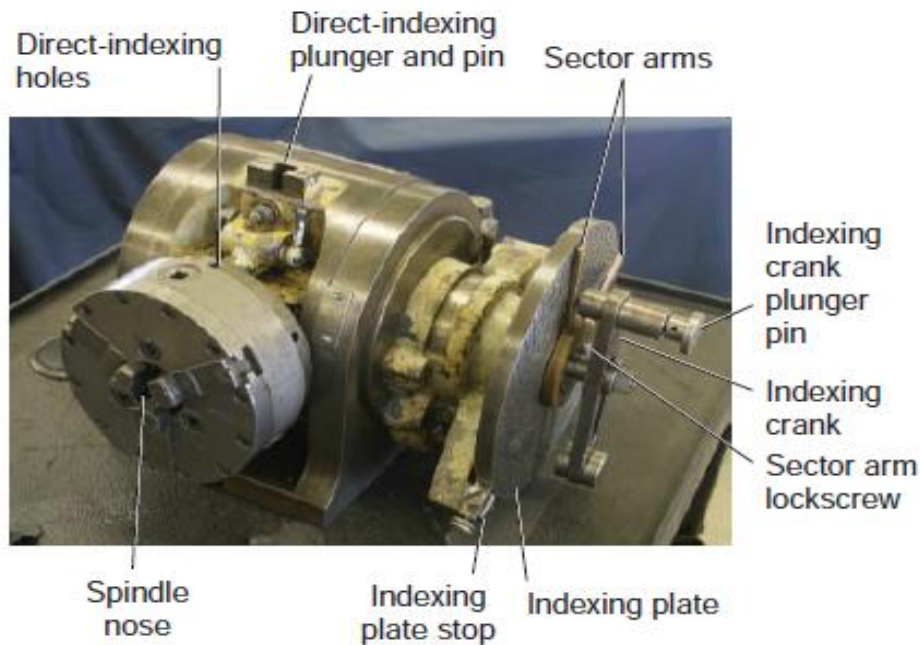


Figure 3.23. The parts of the indexing or dividing head.

Rotating the index crank causes the dividing head spindle (to which the work is mounted) to rotate. The standard ratios for the dividing head are five turns of the index crank for one complete revolution of the spindle (5:1) or 40 turns of the index crank for one revolution of the spindle (40:1).

The ratio between index crank turns and spindle revolutions, plus the index plate with its series of equally spaced hole circles, makes it possible to divide the circumference of the work into the required number of equal spaces.

Index Plate

The indexing plate is a round plate with a series of six or more circles of equally spaced holes; the index pin on the crank can be inserted in any hole in any circle. With the interchangeable plates regularly furnished with most index heads, the spacing necessary for most gears, bolt-heads, milling cutters, splines, and so forth can be obtained. The following sets of plates are standard equipment:

Brown and Sharpe type consists of 3 plates of 6 circles each drilled as follows:

Plate I -15, 16, 17, 18, 19, 20 holes

Plate 2-21, 23, 27, 29, 31, 33 holes

Plate 3-37, 39, 41, 43, 47, 49 holes

Cincinnati type consists of one plate drilled on both sides with circles divided as follows:

First side -24, 25, 28, 30, 34, 37, 38, 39,41,42,43 holes

Second side -46, 47, 49, 51, 53, 54, 57, 58, 59, 62, 66 holes

Slotting attachment

The slotting attachment converts the rotary motion of the spindle into reciprocating motion for cutting key ways, splines, templates, and irregularly shaped surfaces. The length of the stroke is controlled by an adjustable crank.

Slotting : is the process of cutting grooves or slots in the work piece. A staggered-tooth side milling cutter or an end mill can be used for this operation.

- Feed the workpiece in a direction same to the rotation of the milling cutter when milling long or deep slots.
- It is advisable to feed with the milling cutter (called “climb milling”) when cutting off stock, or when milling comparatively deep or long slots.
- When milling deep, thin slot types of work can best be milled by climb milling, if the machine is equipped with a backlash eliminator.



Fig.3.24 slotting attachment on milling

3.4. Advanced Milling Operations

3.4.1 Introduction to Milling operations

There are two main categories of milling operations:

Face milling is machining performed on a surface that is parallel to the cutter face. Large, flat surfaces are machined using this technique.

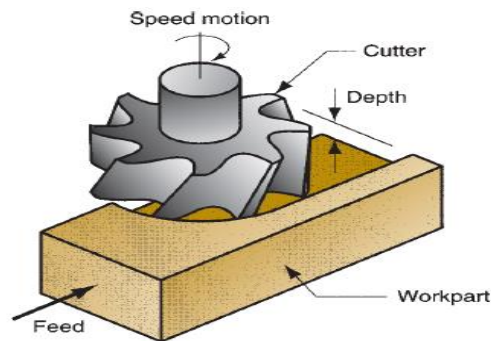


Figure 3.25 Face milling

Peripheral milling, also known as edge milling, is machining performed on a surface that is parallel with the periphery of the cutter.

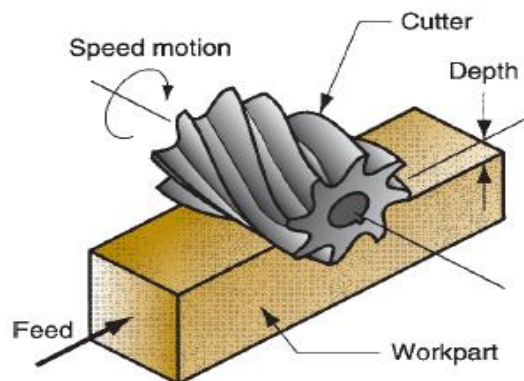


Figure 3.26 Peripheral or plain milling

3.4.2 Methods of milling

There are also two distinct methods of milling: conventional milling and climb milling.

In **conventional milling**, also known as up-milling, the work is fed into the rotation of the cutter. The chip is at minimum thickness at the start of the cut. The cut is so light that the cutter has a tendency to slide over the work until sufficient pressure builds up to cause the teeth to bite into the material. This initial sliding motion, followed by the sudden breakthrough as the tooth completes the cut, leaves the "milling marks" so familiar on many milled surfaces. The marks and ridges can be kept to a minimum by keeping the table gibs properly adjusted.

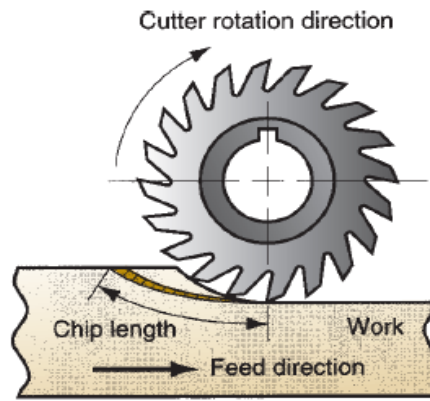


Figure 3.27. Up milling

In **climb milling or down-milling**, the work moves in the same direction as cutter rotation, Figure 17-18B. Full engagement of the cutter tooth is instantaneous. The sliding action of conventional milling is eliminated, resulting in a better finish and longer tool life. The main advantage of climb milling is the tendency of the cutter to press the work down on the worktable or holding device.

Climb milling is not recommended on light machines, nor on large older machines that are not in top condition or are not fitted with an anti backlash device to take up play. There is danger of a serious accident if there is play in the table, or if the work or work-holding device is not mounted securely.

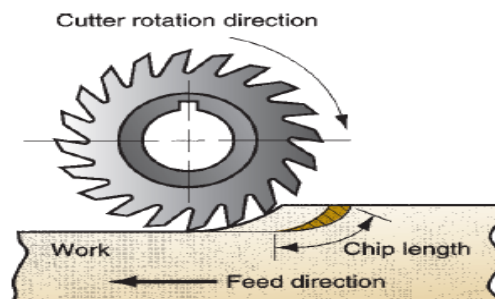


Figure 3.28. Down milling

3.4.3 Operations performed on milling machine

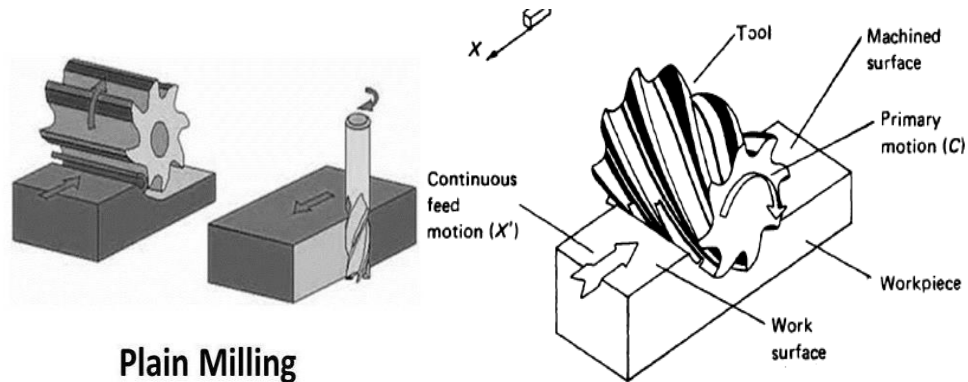
Horizontal milling: axis of cutter rotation is parallel to surface of workpiece. Includes slab milling, form milling, slotting, gang milling and slitting. Can be either up-cut or down-cut milling.

Vertical milling: axis of cutter rotation is perpendicular to surface of workpiece. Includes face milling, slotting, dovetail and woodruff milling.

Unlike a lathe, a milling cutter does not give a continuous cut, but begins with a sliding motion between the cutter and the work. Then follows a crushing movement, and then a cutting operation by which the chip is removed. Many different kinds of operations can be performed on a milling machine but a few of the more common operations will now be explained. These are:

3.4.4 Plain milling or slab milling

This operation produces flat surfaces on the workpiece. Feed and depth of cut are selected, rotating milling cutter is moved from one end of the work piece to other end to complete the one pairs of plain milling operation.



Plain Milling

Figure 3.29. Plain milling or slab milling

3.4.5 Face Milling Operation

This operation produces flat surface at the face of the work piece. This surface is perpendicular to the surface prepared in plain milling operation. This operation is performed by face milling cutter mounted on stub arbor of milling machine. Depth of cut is set according to the need and cross feed is given to the work table.

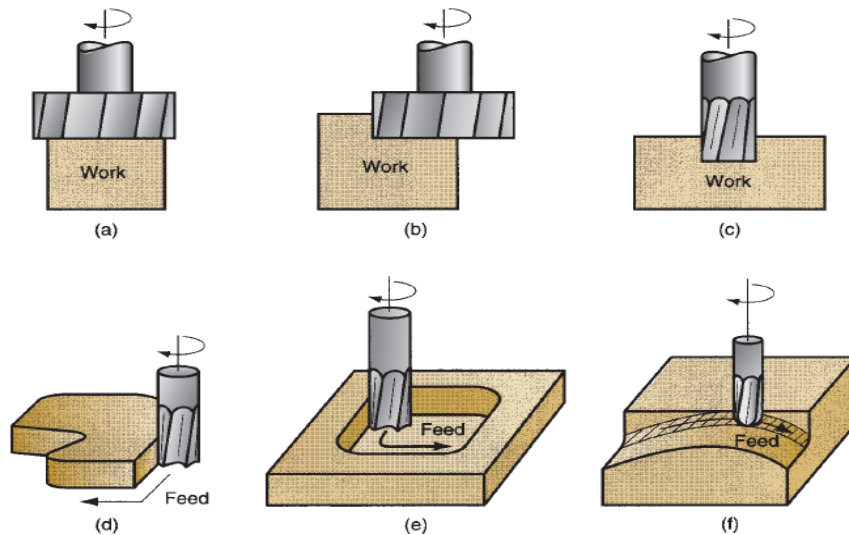


Figure 3.30. Face milling: (a) conventional face milling, (b) partial face milling, (c) end milling, (d) profile milling, (e) pocket milling, and (f) surface contouring

3.4.6. Side Milling Operation

This operation produces flat and vertical surfaces at the sides of the workpiece. In this operation depth of cut is adjusted by adjusting vertical feed screw of the work piece.

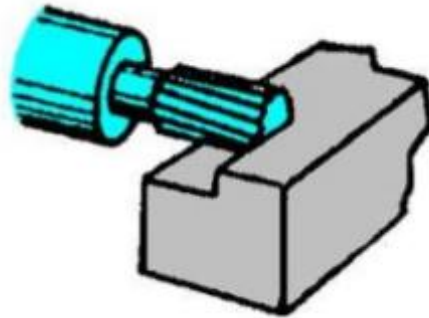


Figure 3.31 Side milling

3.4.7 Straddle Milling Operation

This is similar to the side milling operation. Two side milling cutters are mounted on the same arbor. Distance between them is so adjusted that both sides of the workpiece can be milled simultaneously. Hexagonal bolt can be produced by this operation by rotating the workpiece only two times as this operation produces two parallel faces of bolt simultaneously

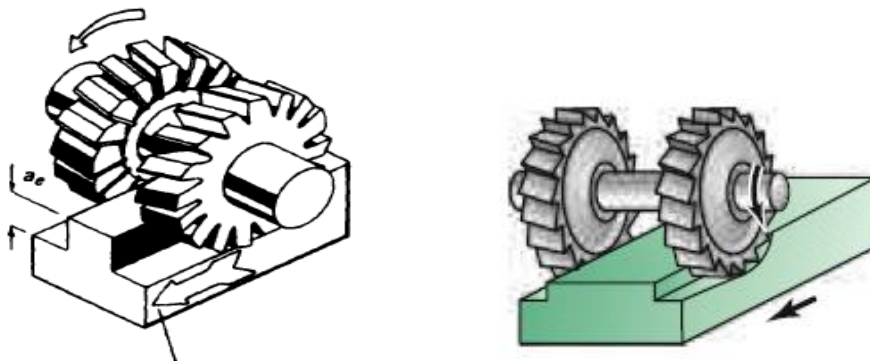


Figure 3.32. Straddle Milling

3.4.8 Angular Milling Operation

Angular milling operation is used to produce angular surface on the work piece. The produced surface makes an angle with the axis of spindle which is not right angle.

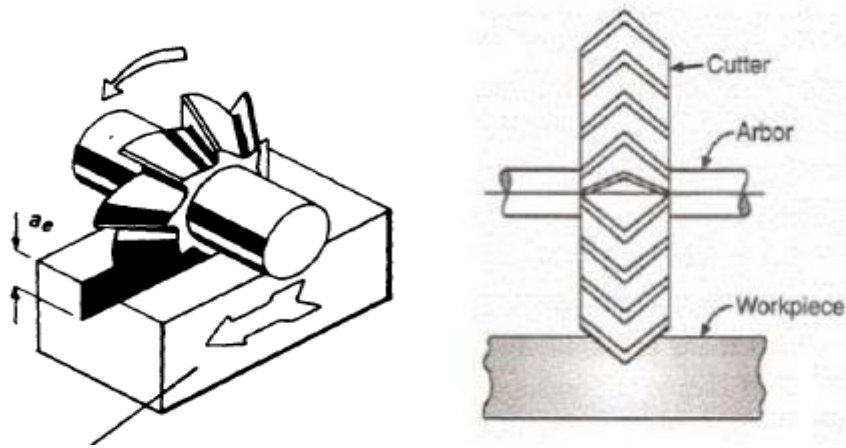


Figure 3.33. Angular milling

3.4.9 Gang Milling Operation

As the name indicates, this operation produces several surfaces of a work piece simultaneously using a gang of milling cutters. During this operation, the work piece mounted on the table is fed against the revolving milling cutters.

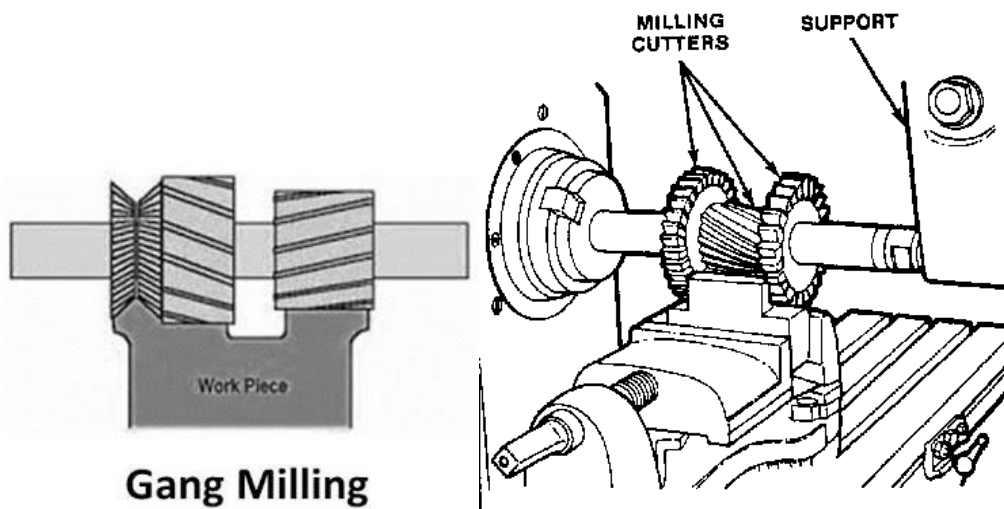


Figure 3.34. Gang milling

3.4.10 Form Milling Operation

This operation produces irregular contours on the work surface. These irregular contours may be convex, concave, or of any other shape. This operation is done comparatively at very low cutter speed than plain milling operation.

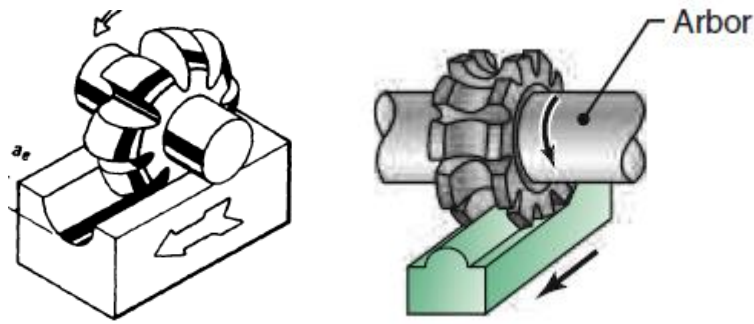


Figure 3.35. Form milling

3.4.11 Profile Milling Operation

In this operation a template of complex shape or master die is used. A tracer and milling cutter are synchronized together with respect to their movements. Tracer reads the template or master die and milling cutter generates the same shape on the workpiece. Profile milling is an operation used to generate shape of a template or die.

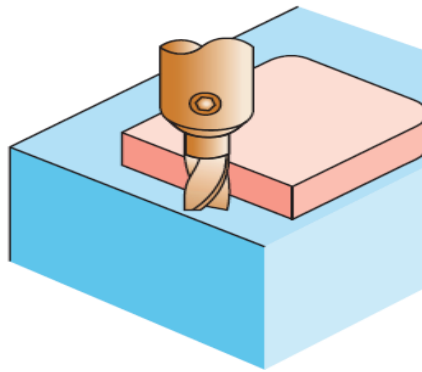


Figure 3.36. Profile milling

3.4.12 End Milling Operation

End milling operation produces flat vertical surfaces, flat horizontal surfaces and other flat surfaces making an angle from table surface using milling cutter named as end mill. This operation is preferably carried out on vertical milling machine.

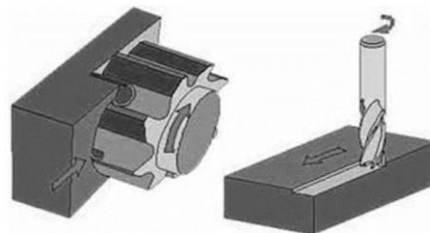


Figure 3.37. End milling

3.4.13. Saw Milling Operation

Saw milling operation produces narrow slots or grooves into the work piece using saw milling cutter. This operation is also used to cut the work piece into two equal or unequal pieces which cut is also known as “parting off”. In case of parting off operation cutter and work piece are set in a manner so that the cutter is directly placed over one of the “T” slot of the worktable as illustrated in figure below.

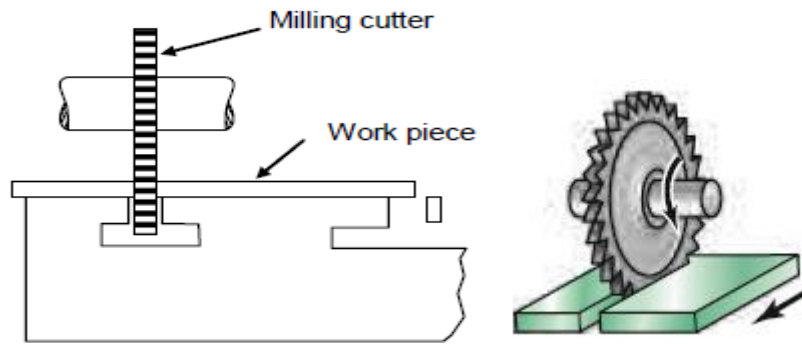


Figure 3.38. Saw milling

3.4.14. Gear Cutting Operation

The operation of gear cutting is cutting of equally spaced, identical gear teeth on a gear blank by handling it on a universal dividing head and then indexing it. The cutter used for this operation is cylindrical type or end mill type. The cutter selection also depends upon tooth profile and their spacing. Gear cutting operation is illustrated in figure below.

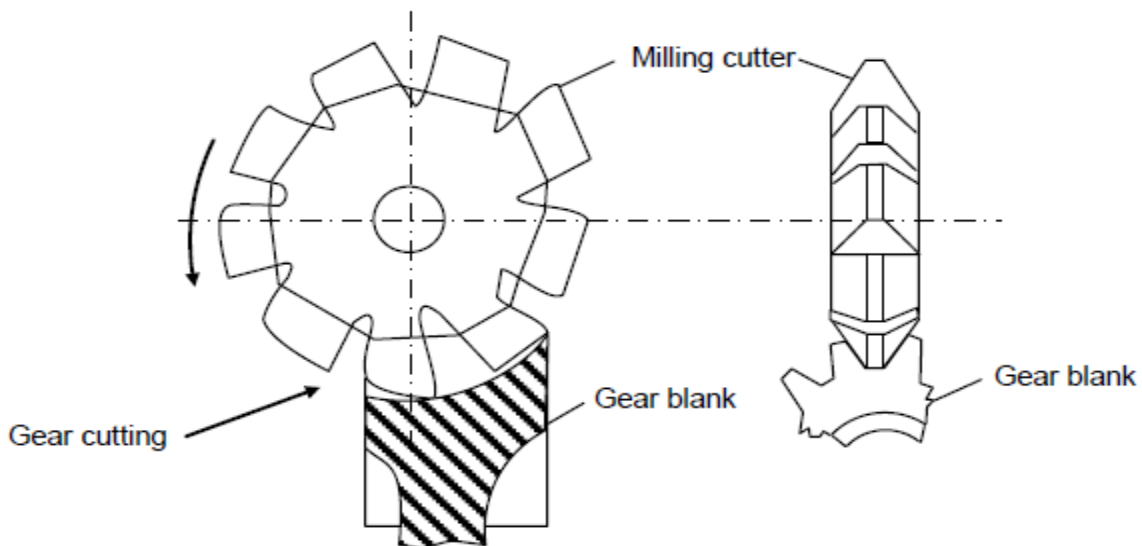


Figure 3.39. Gear cutting

3.4.16 Helical Milling Operation

Helical milling produces helical flutes or grooves on the periphery of a cylindrical or conical work piece. This is performed by swiveling the table to the required helix angle, then rotating and feeding the work piece against revolving cutting edges of milling cutter. Helical gears and drills and reamers are made by this operation.



Figure 3.40. Helical milling

3.4.18 Thread Milling Operation

The operation thread milling produces threads using thread milling centers. This operation needs three simultaneous movements revolving movement of cutter, simultaneous longitudinal movement of cutter, feed movement to the workpiece through table. For each thread, the revolving cutter is fed longitudinal by a distance equal to pitch of the thread. Depth of cut is normally adjusted equal to the full depth of threads.

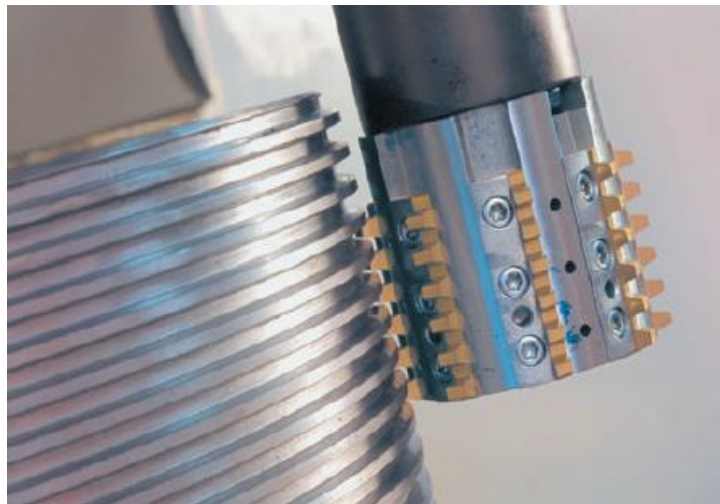


Figure 3.41. Typical thread milling cutters.

3.4.19. Ratchet milling operation

A ratchet is a mechanical device that allows continuous linear or rotary motion in only one direction while preventing motion in the opposite direction. Ratchets are widely used in machinery and tools. A ratchet consists of a round gear or a linear rack with teeth, and a pivoting, spring-loaded finger called a pawl that engages the teeth. The teeth are uniform but asymmetrical, with each tooth having a moderate slope on one edge and a much steeper slope on the other edge.

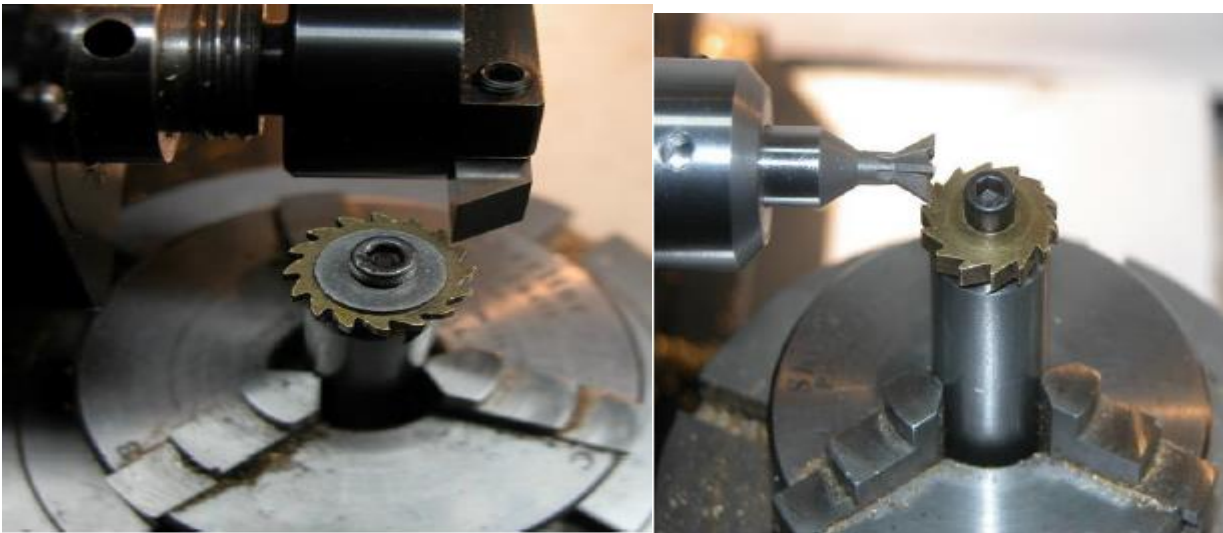


Figure 3.42. Ratchet milling

3.4.20. Indexing head operations

There are two major styles of indexing devices, direct indexing and simple indexing. Some indexing heads have the ability to perform both simple and direct indexing.

Direct Indexing Head Operations

To perform direct indexing, the proper direct indexing plate must be selected and mounted to the dividing head. These plates have either notches around the circumference or equally spaced holes in a circle near the outer edge. Common numbers of notches (or holes) are 24, 30, and 36. The direct indexing plate is chosen based on how many divisions must be machined on the work piece. These divisions are determined by number of features to be machined. These features can be holes in a bolt circle, or equally spaced flats or slots to be machined around a circumference. The number of notches in the plate must be equally divisible by the number of divisions needed.

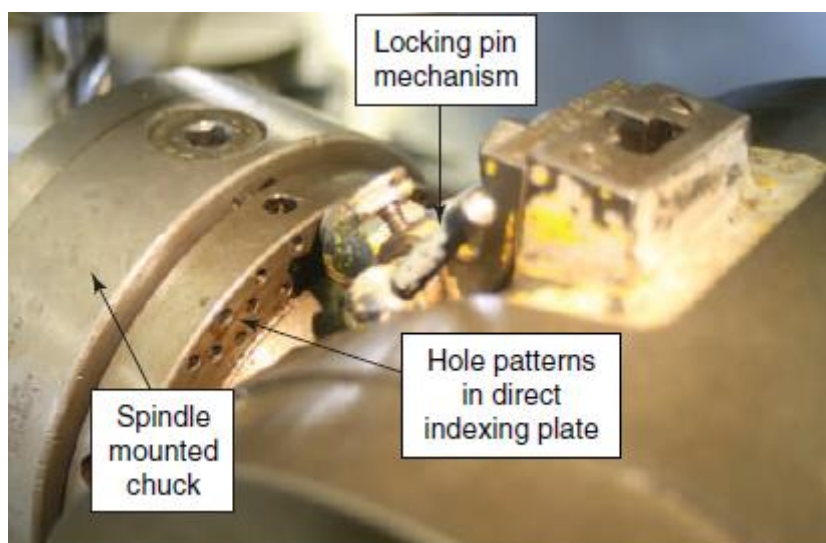


Figure 3.43 The proper indexing plate and plunger positions must be selected for direct indexing.

EXAMPLE: For a part needing six equal divisions, such as a part that needs hexagonal flats for a wrench: Since 24 is equally divisible by 6, a direct indexing plate with 24 notches can be used. When positioning for each division, the spindle must be rotated 4 notches because $24 \div 6 = 4$.

Simple Indexing

Simple indexing requires the crank handle to be rotated to orient the indexing head spindle. The pin on the end of the crank handle then also holds the work in position for machining. Simple indexing plates with evenly spaced holes in a circular pattern are used to measure the distance the crank handle is rotated. Each circular pattern has a different amount of holes.



Figure 3.44. A simple indexing plate.

By selecting the correct plate and rotating the crank the correct amount, many equally spaced divisions can be achieved. Many more possible numbers of divisions may be achieved with this method than with the direct indexing method, because the hole patterns allow precise measurement of full and partial turns of the crank. A spring-loaded locking pin on the end of the crank is used with the indexing plate to align and hold the crank handle in the desired hole.

Since 40 turns of the crank result in 1 turn of the spindle, the amount of crank turn(s) is found by dividing 40 by the number of desired divisions. Again, these divisions can be holes of a bolt circle, or other features machined around a circumference such as flats or slots. Then a suitable indexing plate is selected to accurately rotate the required number of full and/or partial turns

Since 40 turns of the crank result in 1 turn of the spindle, the amount of crank turn(s) is found by dividing 40 by the number of desired divisions:

$$\frac{40}{D} = T$$

T = number of turns of the crank handle

D = number of desired divisions

Remember that if the spacing between features is given as an angular dimension, first divide 360 by that angular dimension to obtain the D value.

EXAMPLE: Drill a hole every 72° in a circular pattern on a work piece: First, determine D using the given 72° :

$$D = \frac{360}{72} = 5$$

Then use the D value of 5 to calculate turns:

$$T = \frac{40}{5} = 8$$

Eight full turns of the crank are needed.

Any indexing plate could be selected because no partial turns are needed. After mounting the plate and workpiece, place the pin on the crank in the starting hole of any hole circle pattern. The crank would be rotated 8 full turns and the pin placed in the same hole each time.

The math will not always work out to be convenient, whole numbers (complete turns). Sometimes only a partial turn must be made with the crank.

Example: Machine 50 slots around the circumference of a workpiece:

$$T = \frac{40}{D} = \frac{40}{50} = \frac{4}{5}$$

In this case, a partial turn of the crank will be needed because the answer is a fraction. The crank needs to be rotated $\frac{4}{5}$ of a turn for each division on the workpiece.

An index plate with a number of holes divisible by the denominator must be selected. Plate #1 from the standard set is a good choice because it contains two hole circles that are divisible by five: 15 and 20. Either can be used. For this example, suppose the 15-hole circle is chosen. The $\frac{4}{5}$ then needs to be converted to a fraction with a denominator of 15:

$\frac{4}{5} \times \frac{3}{3} = \frac{12}{15}$. The $\frac{12}{15}$ means that the crank will be rotated 12 holes in the 15-hole circle, resulting $\frac{4}{5}$ in of a turn for each division on the workpiece.

Gear Cutting

Gear teeth are cut on the milling machine using formed milling cutters called involutes gear cutters. These cutters are manufactured in many pitch sizes and shapes for different numbers of teeth per gear. If involutes gear cutters are not available and teeth must be restored on gears that cannot be replaced. A lathe cutter bit ground to the shape of the gear tooth spaces may be mounted in a fly cutter for the operation. The gear is milled in the following manner:

NOTE: This method of gear cutting is not as accurate as using an involutes gear cutter and should be used only for emergency cutting of teeth which have been built up by welding.

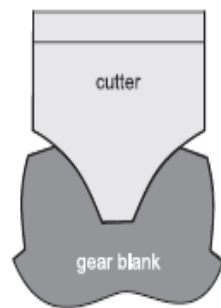
Fasten the indexing fixture to the milling machine table. Use a mandrel to mount the gear between the index head and footstock centers. Adjust the indexing fixture on the milling machine table or adjust the position of the cutter to make the gear axis perpendicular to the milling machine spindle axis.

Align the tooth space to be cut with the fly cutter arbor and cutter bit by turning the index crank on the index head.

Proceed to mill the tooth in the same manner as milling a keyway.

Gear forming

In gear form cutting, the cutting edge of the cutting tool has a shape identical with the shape of the space between the gear teeth.

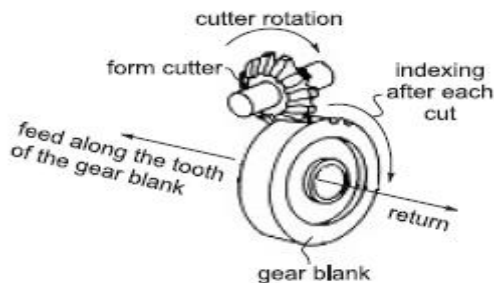


The principle of gear forming.

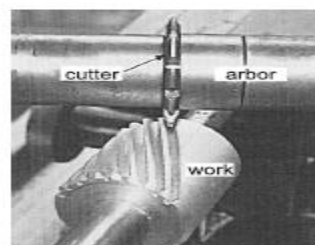
Figure 3.45. principle of forming

A. Form milling

In *form milling*, the cutter called a *form cutter* travels axially along the length of the gear tooth at the appropriate depth to produce the gear tooth. After each tooth is cut, the cutter is withdrawn, the gear blank is rotated (*indexed*), and the cutter proceeds to cut another tooth. The process continues until all teeth are cut.



Setup of form milling.



Form milling of a helical gear.

Figure 3.46 form milling of helical

Types of Gear

Gears are toothed wheels used for transmission of power from one shaft to another when the center distance between the shaft axes is small. Each gear is provided with certain projections rightly called teeth and with intermediate depressions called tooth spaces. The teeth of one gear enter the spaces of the other and thus, when one gear rotates, the other also gets, rotated, thus transmitting power from one gear shaft to the other.



Spur Gear



Helical Gear



Bevel Gear



Worm and Worm Wheel

Figure 3.47. varies types of milling operations

Spur gears

Spur gears have straight teeth cut parallel to the rotational axis. Thus, they are used for transmission of power between parallel shafts. Spur gears are most commonly used they are simple in design and easy to manufacture. In spur gears the contact occurs across the entire tooth length at one instant and it changes from one tooth to the next rather abruptly.



Figure 3.48 spur gears have straight teeth cut parallel to the rotation axis

Spur gears Nomenclature

Pinion and rack gear: when two gears come in contact the smaller gear is called pinion and the bigger one is called gear.

Table 3.3. **Spur gears Formula**

Term	Symbol	Definition	Formula
Addendum	a	Radial distance from pitch circle to the top of tooth.	$a = 1/p$
Circular pitch	p_c	A distance measured along pitch circle from a point on one tooth to corresponding point on the adjacent tooth: including one tooth and one space.	$p_c = \pi D/N$ $p_c = \pi/P$
Clearance	c	Distance between top of a tooth and bottom of mating space: equal to the dedendum minus the addendum.	$c = b - a$ $= 0.175/P$
Dedendum	b	Radial distance from pitch circle to the bottom of tooth space.	b $= 1.157/P$
Diametral pitch	P	A ratio equal to number of teeth on the gear per inch of pitch diameter.	$P = N/D$
Number of tooth	N_G or N_{pG}	Number of tooth on the gear or pinion	$N = P \times D$
Outside diameter	D_o	The diameter of addendum circle; equal to pitch diameter plus twice the addendum.	D_o $= D + 2a$
Pitch circle		An imaginary circle of gear that corresponds to circumference of the friction gear from which the spur gear is derived.	
Pitch diameter	D_G or D_{pG}	Diameter of pith circle of gear and pinion	$D = N/P$
Pressure angle	ϕ	Angle that determines direction of pressure between contacting teeth and designates shape of involute teeth.	
Root diameter	D_R	Diameter of the root circle; equal to pitch diameter minus twice the dedendum.	D_R $= D - 2b$

Whole depth	h_t	Total height of the tooth; equal to the addendum plus the dedendum.	$h_t = a + b$ $= 2.157/P_c$
Working depth	h_k	Distance a tooth projects into mating space; equal to twice the addendum.	$h = 2a$ $= 2/P$

Addendum circle: - is the circle passing through the crest of the gear teeth. The diameter of this circle is given by the equation: $D_o = D + 2a$ where D is pitch diameter a addendum

Dedendum circle: - is the circle passing through the root of the gear teeth this is also known as root circle. The diameter of this circle is given by: $D_R = D - 2b$ where b dedendum

Center distance (C): - The distance between the parallel axes of spur gears. Also, it is the distance between the centers of the pitch circles.

$C = \frac{D_G + D_P}{2}$ Where, D_G & D_P are pitch diameter of gear and pinion.

Module (m): - is defined as the ratio of the pitch circle diameter to the number of teeth on the gear.

$m = \frac{D}{N}$ m is expressed in mm

Tooth face: - is the face of the side of the tooth above the pitch circle.

Tooth flank: - is the face of the side of the tooth below the pitch circle.

Face width (B): - is the width of the tooth face measured along the gear axis. It can be taken as

$B = 2p_c$ to $4p_c$ or $2\pi m$ to $4\pi m$ Where p_c is circular pitch

Spur gear calculation

Given: module = 4mm; number of teeth = 32; shaft diameter = 30mm

Pitch diameter $D = mN = 4 \times 32 = 128$

Addendum circle diameter $D_a = D + 2a = D + 2m = 128 + 2 \times 4 = 136$

Dedendum circle diameter $D_b = D - 2b = 128 - 2m \left(1 + \frac{\pi}{20}\right) = 128 - 2 \times 4 \left(1 + \frac{\pi}{20}\right) = 118.75$

Face width $B = 3 \times p_c = 3 \times \frac{\pi D}{N} = 3 \times \frac{\pi \times 128}{32} = 37.7$ mm

Hub diameter $D_h = 1.5D_s$ to $2D_s$ where D_s is the shaft diameter

Assume $D_h = 1.75D_s = 1.75 \times 30 = 52.5\text{mm}$

Hub length $L = B$ to $1.4B$

Assume $L = 1.2B = 1.2 \times 37.7 = 45.24\text{mm}$

Web thickness $T_w = p_c = \frac{\pi D}{N} = \frac{\pi \times 128}{32} = 12.6\text{mm}$

Rim thickness $T_r = a + b = m + \left(1 + \frac{\pi}{20}\right)m = 4 + \left(1 + \frac{\pi}{20}\right) \times 4 = 8.65\text{mm}$

Spur gearing

Given gear $m = 5$

pinion $m = 5$

$N_G = 40$

$N_P = 16$

$D_{sg} = 40$

$D_{sp} = 40$

Pitch diameter of pinion $D_P = mN_P = 5 \times 16 = 80\text{mm}$

Pitch diameter of gear $D_G = mN_G = 5 \times 40 = 200\text{mm}$

Addendum circle diameter of pinion $D_{aP} = D + 2a = D_P + 2m = 80 + 2 \times 5 = 90\text{mm}$

Addendum circle diameter of gear $D_{aG} = D + 2a = D_G + 2m = 200 + 2 \times 5 = 210\text{mm}$

Denendum circle dia. of pinion $D_{bP} = D_P - 2b = 80 - 2m \left(1 + \frac{\pi}{20}\right) = 80 - 2 \times 5 \left(1 + \frac{\pi}{20}\right) = 68.43 \cong 68\text{mm}$

Denendum circle dia. of gear $D_{bG} = D_G - 2b = 200 - 2m \left(1 + \frac{\pi}{20}\right) = 200 - 2 \times 5 \left(1 + \frac{\pi}{20}\right) = 188.43 \cong 188\text{mm}$

Face width $B = 3 \times p_c = \frac{3\pi D_P}{N_P} = \frac{3\pi D_G}{N_G} = \frac{3 \times \pi \times 80}{16} = \frac{3 \times \pi \times 200}{40} = 47.1 \cong 47\text{mm}$

Hub diameter of pinion $D_{hP} = 1.5D_s = 1.5 \times 40 = 60\text{mm}$

Hub diameter of pinion $D_{hG} = 1.75D_s = 1.75 \times 40 = 70\text{mm}$

Hub length $L_P = 1.1B = 1.1 \times 47 = 51.7 \cong 52\text{mm}$

Hub length $L_G = 1.2B = 1.2 \times 47 = 56.4 \cong 56\text{mm}$

Web thickness $T_{wG} = p_c = \frac{\pi D_G}{N_G} = \frac{\pi \times 200}{40} = 15.7 \cong 16\text{mm}$

Page 83 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
----------------	--	-------------------------------------	------------

$$\text{Rim thickness } T_r = a + b = m + \left(1 + \frac{\pi}{20}\right)m = 5 + \left(1 + \frac{\pi}{20}\right) \times 5 = 10.75 \cong 11\text{mm}$$

Key way width= 12mm depth = 4mm

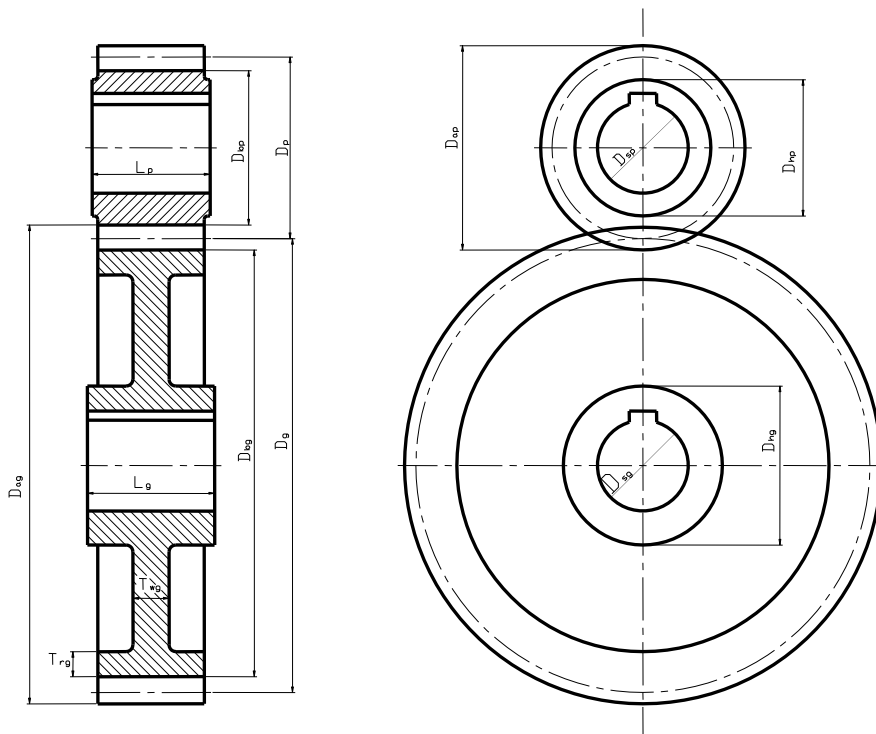


Figure 3.49 Spur gears nomenclature

Rack-and-pinion

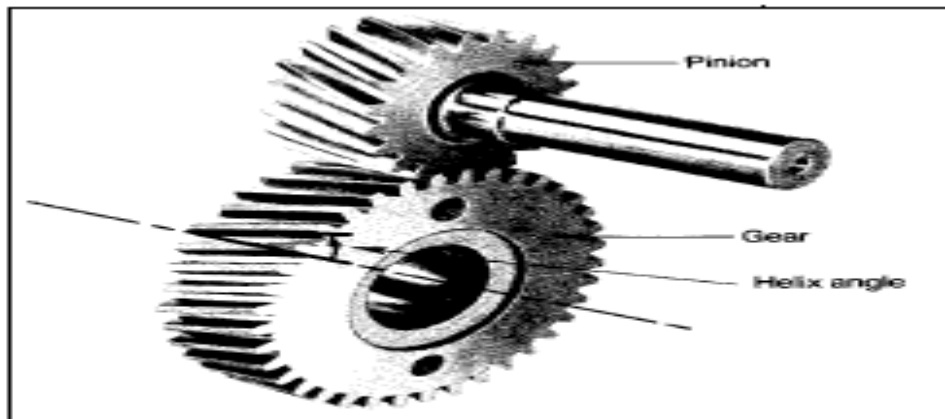
Gears a straight bar with teeth cut straight across it, is called a rack. Basically, this rack is considered to be a spur gear unrolled and laid out flat. Thus, the rack-and pinion is a special case of spur gearing. The rack-and-pinion is useful in converting rotary motion to linear and vice versa. Rotation of the pinion produces linear travel of the rack. Conversely, movement of the rack causes the pinion to rotate. The rack-and-pinion is used extensively in machine tools, lift trucks, power shovels, and other heavy machinery where rotary motion of the pinion drives the straight-line action of a reciprocating part.



Rack-and-pinion gearing produces linear travel from rotational input. Shown here is spur gearing. Helical gearing is also available, but is not as common because the helical teeth create thrust, which produces a force acting across the face of the rack. Worm rack is also available, the axis of the worm (pinion) being parallel to, rather than perpendicular to, the rack.

Helical gears

Helical gears are used to connect parallel shafts. Helical gearing differs from spur in that helical teeth are cut across the gear face at an angle rather than straight. Thus, the contact line of the meshing teeth progresses across the face from the tip at one end to the root of the other, reducing the noise and vibration characteristic of spur gears. Also, several teeth are in contact at any one time, producing a more gradual loading of the teeth that reduces wear substantially.



Helical gears have teeth cut across the face at an angle for gradual loading.

Figure 3.51 helical gears have teeth cut

The following are the new terms added in helical gears

Normal pitch P_n this is measured normal to tooth faces

$P_n = P_c \cos \phi$ where P_c is the circular pitch in the diametral plane and ϕ is the helix angle

Then, pitch circle diameter $D = \frac{NP_c}{\pi} = \frac{NP_n}{\pi \cos \phi}$ where N is the number of tooth

The following analysis shows how to calculate all the dimensions of a helical gear from the given data in order to draw its views.

Given $\phi = 20^\circ$

Normal circular pitch $P_n = 10\text{mm}$

Gear shaft diameter $D_s = 30\text{mm}$

$m = 5\text{mm}$

Thus, pitch circle diameter $D = \frac{NP_n}{\pi \cos \phi} = \frac{32 \times 10}{\pi \cos 20^\circ} = 108.4 \cong 108 \text{mm}$

Addendum circle diameter $D_a = D + 2a = D + 2m = 108.4 + 2 \times 5 = 118.4 \cong 118 \text{mm}$

Dedendum circle diameter $D_b = D - 2b = D - 2 \left(1 + \frac{\pi}{20}\right) m = 108.4 - 2 \left(1 + \frac{\pi}{20}\right) \times 5 = 96.84 \text{mm}$

Face width $B = 3P_c = 3 \times \frac{P_n}{\cos \phi} = 3 \times \frac{10}{\cos 20^\circ} = 32 \text{mm}$

Hub diameter $D_h = 1.5D_s \text{ to } 2D_s$ assume $D_h = 1.75D_s = 1.75 \times 30 = 52.5 \text{mm}$

Hub length $L = B \text{ to } 1.4B$ assume $L = 1.2B = 1.2 \times 32 = 38.4 \text{mm}$

Web thickness $T_w = P_c = \frac{P_n}{\cos \phi} = \frac{10}{\cos 20^\circ} = 10.6 \text{mm}$

Rim thickness $T_r = a + b = m + \left(1 + \frac{\pi}{20}\right) m = 5 + \left(1 + \frac{\pi}{20}\right) \times 5 = 10.785 \cong 11 \text{mm}$

➡ *The drawing of helical gear is the same as that of spur gear when it is drawing in sectional views.*

Bevel gears

Unlike spur and helical gears with teeth cut from a cylindrical blank, bevel gears have teeth cut on an angular or conical surface. Bevel gears are used when input and output shaft centerlines intersect. Teeth are usually cut at an angle so that the shaft axes intersect at 90 deg, but any other angle may be used. A special class of bevels called miter gears has gears of the same size with their shafts at right angles.

There are two basic classes of bevels: straight-tooth and spirals.

Straight-tooth bevels: these gears, also known as plain bevels, have teeth cut straight across the face of the gear. They are subject to much of the same operating conditions as spur gears in that straight-tooth bevels are efficient but somewhat noisy. They produce thrust loads in a direction that tends to separate the gears.

Spiral-bevels: Curved teeth provide an action somewhat like that of a helical gear. This produces smoother, quieter operation than straight-tooth bevels. Thrust loading depends on the direction of rotation and whether the spiral angle at which the teeth are cut is positive or negative

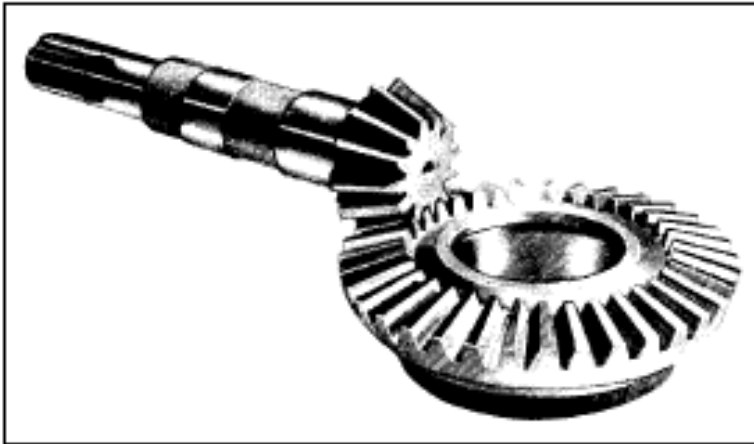
Table 3.4. **Spiral bevels formula**

Term	Symbol	Definition	Formula
------	--------	------------	---------

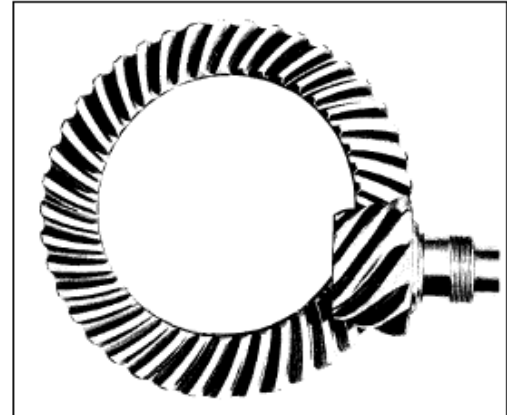
Addendum	a	Distance from pitch cone to top of tooth measured at larger end	$a = 1/P$
Addendum angle	α	Angle subtended by addendum; same for gear and pinion.	$\tan \alpha = a/A$
Back angle		Usually equal to the pitch angle	
Backing	Y	Distance from base of pitch cone to rear of hub	
Crown backing	z	More practical than backing for shop use; dimension Z given on drawing other than Y .	$Z = Y + a \sin r$
Crown height	x	Distance, parallel to gear axis, from cone apex to crown of the gear.	$X = \frac{1}{2} D_o / \tan r_o$
Dedendum	b	Distance from pitch cone to bottom of tooth; measured large end of tooth.	$b = 1.188/P$
Dedendum angle	δ	Angle subtended by dedendum; same for gear and pinion	$\tan \delta = b/A$
Face angle	r_o	Angle between top of the teeth and the gear axis.	$r_o = r + \alpha$
Face width	F	Should not exceed $\frac{1}{3} A$ or $10/P$, whichever is smaller.	
Mounting distance	M	A dimension used primarily for inspection and assembly purpose.	$M = Y + \frac{1}{2} D_o / \tan r$
Outer cone distance	A	Slant height of pitch cone; same for gear and pinion.	$A = D/2 \sin r$
Outer diameter	D_o	Diameter of outside or crown circle of the gear.	$D_o = D + 2a \sin r$
Pitch diameter	D_G or D_P	Diameter of base of pitch cone of gear or pinion.	$D_G = N_G / P$ $D_P = N_P / P$
Root angle	Γ_R	Angle between the root of the teeth and the gear axis	$\Gamma_R = \Gamma - \delta$

Pitch angle is the angle between the pitch line and the axis of the gear.

Pitch line is a straight line joining the vertex and the point on the outer edge of the tooth at pitch circle radius.



Straight-tooth bevel gears are efficient but somewhat noisy.



Hypoid gears resemble spiral bevels, but the shaft axes do not intersect. Therefore, both shafts can be supported at both ends.



Spiral bevel-gears have curved teeth for smoother operation.

Figure 3.52. Spiral gears have curved teeth

Bevel gearing

Pinion

gear

$m=5\text{mm}$

$m=5\text{mm}$

$N_p=18$

$N_g = 54$

$D_s = 30\text{mm}$

$D_s = 40\text{mm}$

Keyway width x thickness = 10 x 4 Keyway width x thickness = 12 x 4

Shaft angle 90°

$$\left(\begin{array}{l} \text{Pitch angle of pinion or} \\ \text{Back angle of pinion} \end{array} \right) \alpha_p = \tan^{-1} \frac{\sin \theta_s}{\frac{N_G}{N_P} + \cos \theta} = \tan^{-1} \frac{N_P}{N_G} = \tan^{-1} \frac{18}{54} = \tan^{-1} \frac{1}{3} = 18.43^\circ$$

$$\cong 18^\circ$$

$$\left(\begin{array}{l} \text{Pitch angle of Gear or} \\ \text{Back angle of gear} \end{array} \right) \alpha_G = \tan^{-1} \frac{\sin \theta_s}{\frac{N_P}{N_G} + \cos \theta} = \tan^{-1} \frac{N_G}{N_P} = \tan^{-1} \frac{54}{18} = \tan^{-1} 3 = 71.57^\circ$$

$$\cong 72^\circ$$

Pitch diameter of pinion $D_p = mN_p = 5 \times 18 = 90\text{mm}$

Pitch diameter of Gear $D_G = mN_G = 5 \times 54 = 270\text{mm}$

Outer cone distance of pinion $A = \frac{D_p}{2 \sin \alpha_p} = \frac{90}{2 \sin 18.43} = 142\text{mm}$

Outer cone distance of Gear $A = \frac{D_G}{2 \sin \alpha_G} = \frac{270}{2 \sin 71.57} = 142\text{mm}$

Face width $F = \frac{R_C}{3} = \frac{142}{3} = 47.33 \cong 47\text{mm}$

Addendum $a = m = 5\text{mm}$

Dedendum $b = \left(1 + \frac{\pi}{20}\right) m = \left(1 + \frac{\pi}{20}\right) \times 5 = 5.785 \cong 6\text{mm}$

Hub diameter $D_{hP} = 1.5 \times D_s = 1.5 \times 30 = 45\text{mm}$

Hub diameter $D_{hG} = 2 \times D_s = 2 \times 40 = 80\text{mm}$

Hub length $L_P = 1.2 \times B = 1.2 \times 47 \cong 56.4 \cong 56\text{mm}$

Hub length $L_G = 1.4 \times B = 1.4 \times 47 \cong 65.8 \cong 66\text{mm}$

Web thickness of gear $T_{wG} = P_C = \frac{\pi D_G}{N_G} = \frac{\pi \times 270}{54} = 15\text{mm}$

Rim thickness of gear $T_{rG} = a + b = 10.785 \cong 11\text{mm}$

Rim thickness of pinion $T_{rP} =$ assume 5mm

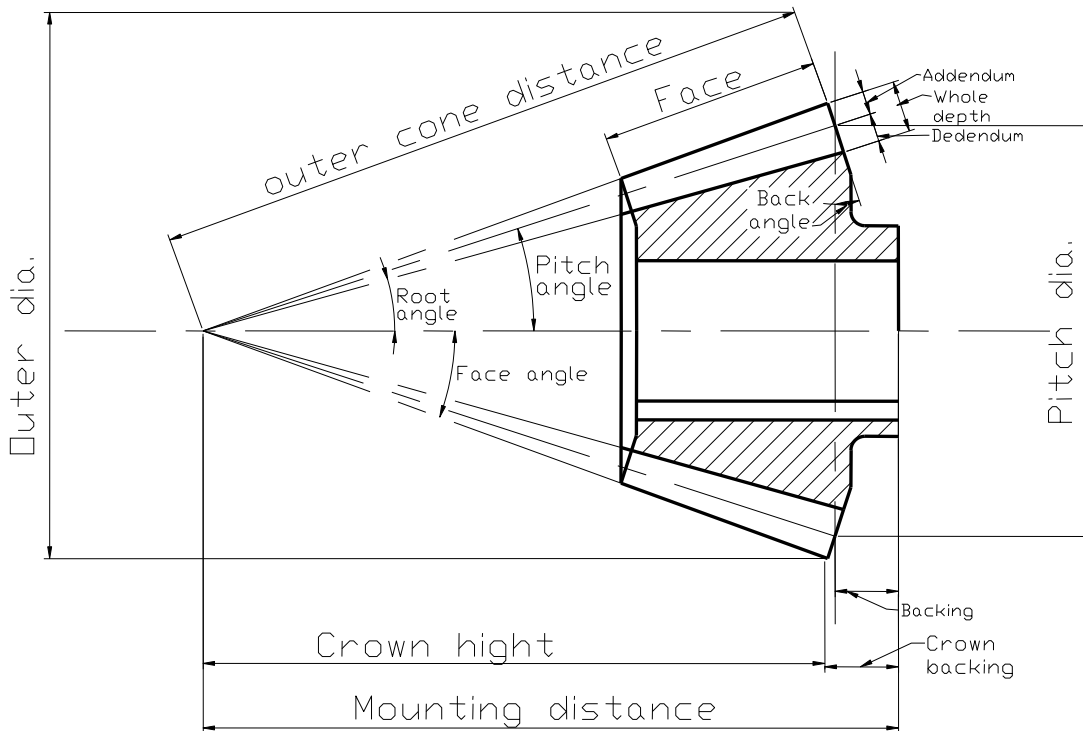
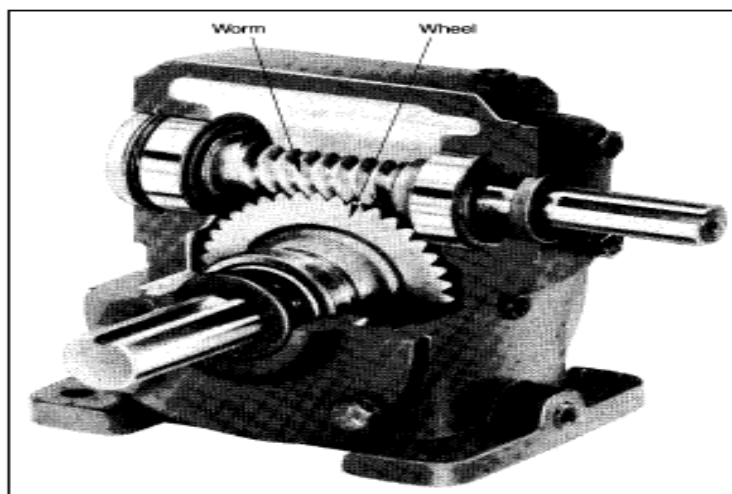


Figure 3.53. Bevel gear nomenclature

Worm and worm gear

Worm gear sets, consist of a screw-like worm (comparable to a pinion) that meshes with a larger gear, usually called a wheel. The worm acts as a screw, several revolutions of which pull the wheel through a single revolution. In this way, a wide range of speed ratios up to 60:1 and higher can be obtained from a single reduction. Most worms are cylindrical in shape with a uniform pitch diameter. However, a double-enveloping worm has a variable pitch diameter that is narrowest in the middle and greatest at the ends. This configuration allows the worm to engage more teeth on the wheel, thereby increasing load capacity. In worm-gear sets, the worm is most often the driving member.



Worm gearing has perpendicular, nonintersecting shafts in which the worm acts as a screw. Several revolutions of the worm pull the wheel through one revolution.

Figure 3.54 Worm gearing function

Proportions

A worm and worm gear have a circular pitch of 15.748mm and the gear has 32 teeth of $14\frac{1}{2}^\circ$ involute form. The worm is double threaded make an assembly drawing. Calculate dimensions accurately and use AGMA proportions.

Shaft diameter of worm 29mm

Shaft diameter of Gear 41mm

Worm worm gear

$P = 15.748\text{mm}$ $P = 15.748\text{mm}$

$z = 2$ $N_G = 32$

$D_{sw} = 59\text{mm}$ $D_{sG} = 41\text{mm}$

For the worm

Pitch diameter $D_w = 2.4P + 27.94 = 2.4 \times 15.748 + 27.94 = 65.74 \cong 66\text{mm}$

Whole depth $h_t = 0.686p = 0.686 \times 15.748 = 10.8 \cong 11\text{mm}$

Outside diameter $D_o = D_w + 0.636p = 66 + 0.636 \times 15.748 \cong 76\text{mm}$

Face length $B = p \left(4.5 + \frac{N_G}{50} \right) = 15.748 \left(4.5 + \frac{32}{50} \right) \cong 81\text{mm}$

Hub diameter $D_{hw} = 1.75D_{sw} = 1.75 \times 29\text{mm} = 50.75 \cong 51\text{mm}$

Keyway width X thickness = 8 x 3.5mm

For the worm wheel

Pitch diameter $D_G = P \left(\frac{N_G}{\pi} \right) = 15.748 \left(\frac{32}{\pi} \right) = 160.4 \cong 160\text{mm}$

Throat diameter $D_t = D_G + 0.636P = 160 + 0.636 \times 15.748 = 160 + 10 = 170\text{mm}$

Outside diameter $D_o = D_t + 0.4775P = 170 + 0.4775 \times 15.748 = 177.9 \cong 178\text{mm}$

Face radius $R_f = \frac{1}{2}D_w - 0.318P = \frac{1}{2} \times 66 - 0.318 \times 15.748 = 27.992 \cong 28\text{mm}$

Rim radius $R_r = \frac{1}{2}D_w + P = \frac{1}{2} \times 66 + 15.748 = 48.748 \cong 49\text{mm}$

Face width $B = 2.38P + 6.35 = 2.38 \times 15.748 + 6.35 = 43.83 \cong 44\text{mm}$

Center distance $C = \frac{1}{2}(D_G + D_w) = \frac{1}{2}(160 + 66) = 113\text{mm}$

Hub diameter $D_{hg} = 2 \times D_{Gs} = 2 \times 41 = 82\text{mm}$

Hub length $L_G = 1.4 \times B = 1.4 \times 44 = 61.6 \cong 62\text{mm}$

Web thickness $T_w = P = 15.748 \cong 16\text{mm}$

Keyway width x thickness = 12 x 4mm

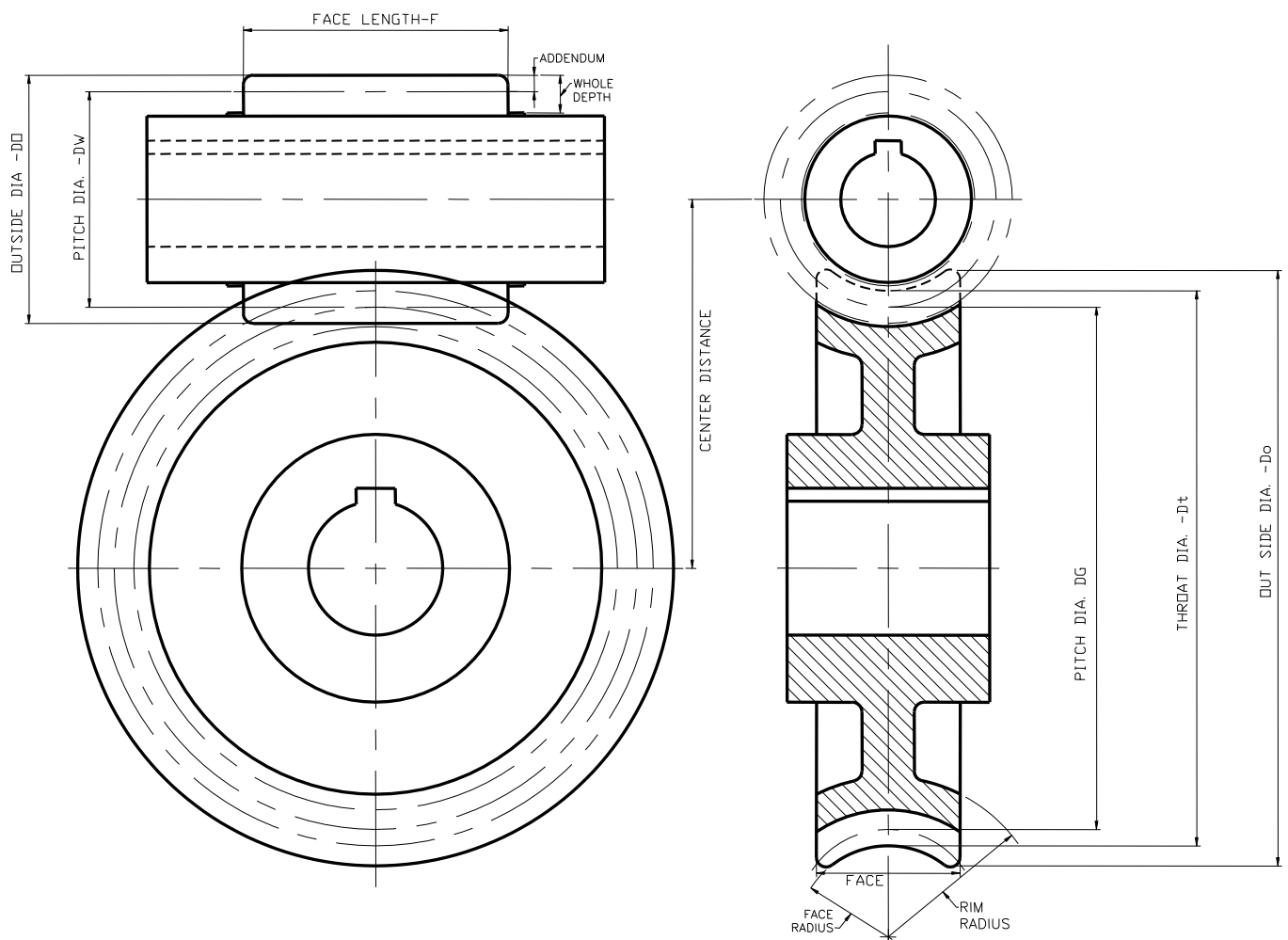


Figure 3.55. Worm gear nomenclature

Splines

Splines are often used instead of keys to transmit power from a shaft to a hub or from a hub to a shaft. Splines are in effect a series of parallel keys formed integrally with the shaft mating with corresponding grooves in the hub or fitting. They are particularly useful where the hub must slide axially on the shaft,

either under load or freely. Typical applications for splines are found in geared transmissions, machine tool drives and in automatic mechanisms.

Milling Splines

Spline shafts can be milled on the milling machine in a manner similar to the cutting of keyways.

The shaft to be splined is set up between centers in the indexing fixture.

Two side milling cutters are mounted to an arbor with a spacer and shims inserted between them. The spacer and shims are chosen to make space between the inner teeth of the cutters equal to the width of the spline to be cut.

The arbor and cutters are mounted to the milling machine spindle and the milling machine is adjusted so that the cutters are centered over the shaft.

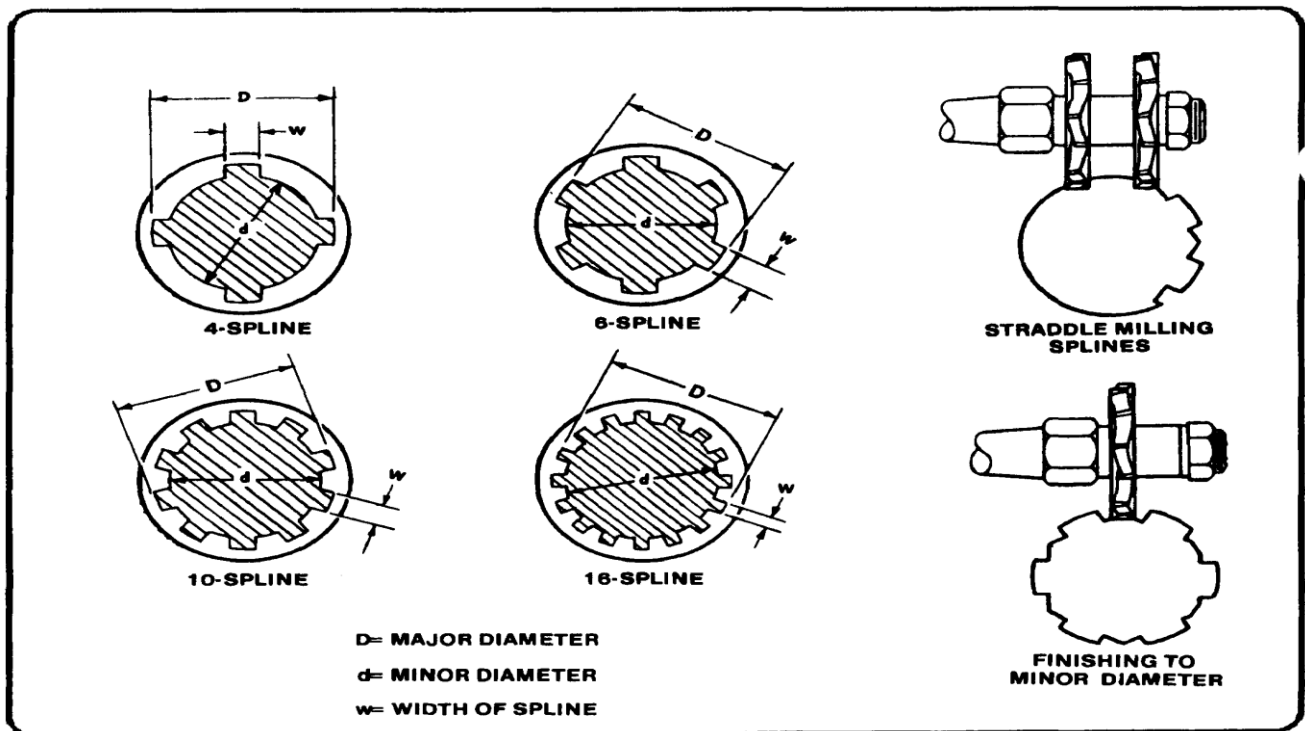


Figure 3.56. Milling splines on shafts

Performing gear hobbling

Performing gear hobbling is a machining process used to produce gears with precise tooth profiles. It involves using a gear hobbing machine, specifically a hobbing machine, which is designed specifically for this purpose. Here's an overview of the steps involved in performing gear hobbling:

Setup: The first step is to set up the gear hobbing machine for the specific gear being produced. This involves mounting the work piece, known as the gear blank, securely on the machine's spindle. The hob, which is a specialized cutting tool, is also mounted on the machine.

Tool Selection: The hob is selected based on the desired tooth profile and specifications of the gear. The hob has cutting edges that correspond to the shape of the gear teeth that need to be produced.

Gear Hobbing Process: Once the machine is set up, the gear hobbing process begins. The spindle rotates the gear blank, while the hob is fed into the blank's surface. The hob's cutting edges gradually remove material from the gear blank, forming the gear teeth.

Cutting Parameters: The cutting parameters, such as hob speed, feed rate, and depth of cut, are set based on the material of the gear blank and the desired tooth profile. These parameters ensure optimal cutting performance and dimensional accuracy of the gear.

Indexing: The hobbing machine is equipped with indexing mechanisms that control the rotation of the gear blank and the hob. The indexing ensures that each tooth on the gear blank is accurately cut by the hob, resulting in a uniform tooth profile.

Finishing: After the initial hobbing process, the gear teeth may undergo additional finishing operations to improve their surface finish and accuracy. This can include processes such as gear shaving, gear grinding, or gear honing, depending on the desired specifications.

Gear hobbing is a widely used method for producing gears in various industries, including automotive, aerospace, and machinery..

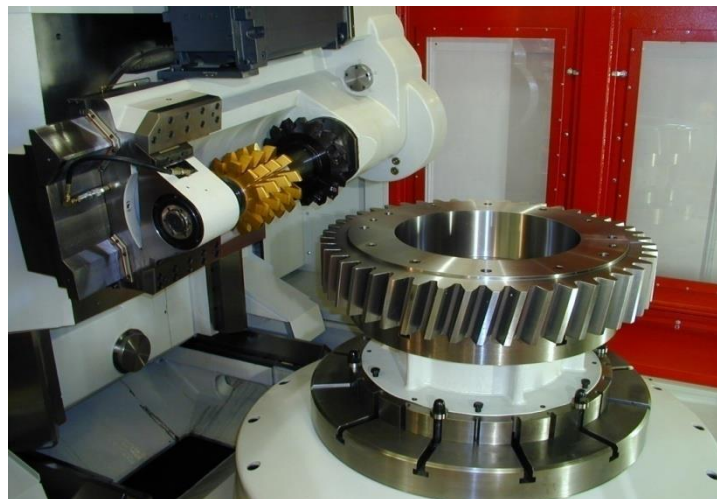


Figure 3.57. Gear Hobbing picture

Self-Check 3.

Instruction: Answer all the questions listed below. Use the Answer sheet provided in the next page:

Part I: Multiple Choices

1. Gang milling means:
 - a) Several cutters being used at same time to machine a job.
 - b) Two or more cutters straddling the job.
 - c) Several side cutters being used at same time to machine a job.
 - d) Single point cutter only used
2. _____ is used to divide circumference of work-piece into equally spaced divisions.
A. Indexing head B. vice C. dividing head D. A and C
3. Which of the following is false about dividing head
 - A. Is one of the more important attachments for milling machine
 - B. Is used when milling gear teeth, squares, hexagons, and octagons
 - C. Is used to rotate work-piece at predetermined ratio to table feed rate.
 - D. it is used only for gear cutting
4. A 16 gear teeth is to be milled on milling machine using dividing head. If 16 plate hole circle is to be used , the indexing would be,
 - A. Two full turn and eight holes in sixteen hole circle
 - B. Two full turn and two holes in sixteen hole circle
 - C. Two full turn and sixteen holes in sixteen hole circle
 - D. Two full turn and one holes in sixteen hole circle
5. _____ is an operation in which two milling cutters are used to mill both sides at the same time.
 - A. Straddle milling Slitters C. plain milling D. end milling
6. Which one of the following us a milling attachment
 - A. Indexing head B. rotary table C. spindle D. spur gear E. A and B

7. Which type of machining not possible to done by milling machine?

- a) cutting keyways
- b) slots and grooves
- c) gears
- d) turning

Part II: Matching

“A”

“B”

- | | |
|---------------------|--|
| 1. Addendum | A. Radial distance from pitch circle to the top of tooth. |
| 2. Dedendum | B. The diameter of addendum circle |
| 3. Outside diameter | C. Diameter of pith circle of gear and pinion |
| 4. Pitch diameter | D. Radial distance from pitch circle to the bottom of tooth space. |
| 5. Root diameter | E. Diameter of the dedendum circle |

Operation Sheet 3. 1

Performing Milling Operations Produce Component Including Gears

Operation Title: Cutting spur gear on milling machine (gear cutting operation)

Purpose: To cut a spur gear teeth on a given circular blank by gear cutting processes.

Tools required:

Horizontal milling machine, Spur gear cutter, vernier caliper, gear tooth vernier, indexing attachment.

Sequence of Operations:

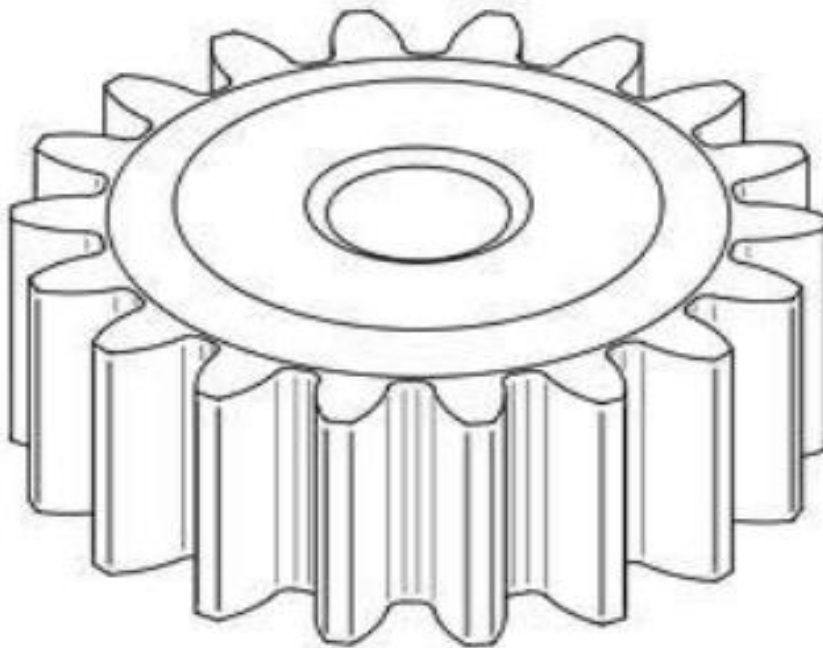
1. Gear blank obtained by turning casted circular rod to desired dimensions with help of lathe.
2. A hole of 15mm is drilled exactly at the face center of the turning bar to the desired depth.
3. The blank of 25 mm width and 81mm external diameter and 15mm is parting off from the bar.
4. Select suitable cutter for spur gearing of module 2mm.
5. The gear blank is fixed in between dead center and dividing head center of the milling machine table.
6. Select suitable index plate and fix properly in the dividing head.
7. Raise the milling machine table by elevating screw, it will approach the cutter and check the alignment of the gear blank and the cutter and check the table moment limits for both transverse and cross feed.
8. Select the cutting speed of m/mm shifting, shifting speed lever and feed lever of the milling machine in milling head.
9. Switch on the machine, check for direction of cutter with respect to work piece.
10. Feed the gear blank against the rotating cutter by manual feeding of table by giving depth of cut in twice or thrice.
11. After completion of each tooth of gear, gear blank is moved away from the cutter.

Page 97 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
----------------	--	-------------------------------------	------------

12. Form dividing head suitable method of indexing is adopted to get the equal number of divisions or made on gear blank according to indexing calculations.
13. Now move the sector arm of the dividing head to the next point and rotate the crank of dividing head to the desired number of rotation and move the crank pin on desired number of holes of the index plate to exact number of equal division on the gear blank.
14. Again the gear blank is fed on the rotating cutter and second teeth are completed by following step number 9.
15. Supply suitable cutting fluids at the time of machining and wear safety devices while machining is going on.

Result:

The desired spur gear is obtained by gear cutting operation and dimensional accuracy



Operation Sheet 3.2.

Performing Milling Operations Produce Component Including Gears

Operation Title: producing components including gears

LIST OF OPERATIONS

1. Outer diameter turning.
2. Indexing
3. Gear cutting
4. Milling Key way

Procedure

Spur gear cutting

1. Copied the drawing.
2. Turned the gear blank to the required diameter and chamfered its two edges.
3. Fixed the gear cutter on the milling machine arbor.
4. Performed the Setting of Dividing head and Milling machine table.
5. The gear blank is mounted on a mandrel which is supported between the chuck of the dividing head and center of the tail stock. At a time one tooth space is cut by the milling cutter, and a dividing head is used to index the job to the next required tooth space. The cutter is chosen according to the module and number of teeth of the gear to be cut. Before the gear can be cut, it is necessary to check the cutter centered accurately relative to the gear holding mandrel. One way is to adjust the machine table vertically and horizontally until one corner of the cutter just touches the tail stock center. After centering of table, table make downward direction. The table is then fed vertically so that the blank just touches the cutter. The vertical dial is then set to zero. This is required to give the depth of cut on the job. After proper depth of cut, started gear cutting.

Page 99 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
----------------	--	-------------------------------------	------------

After one tooth space is cut, the blank is indexed through $1/z$ revolution by means of the dividing head, and the process is repeated until all the teeth are cut.

Gear cutting done with proper depth of cut.

Operation Sheet 3.3.

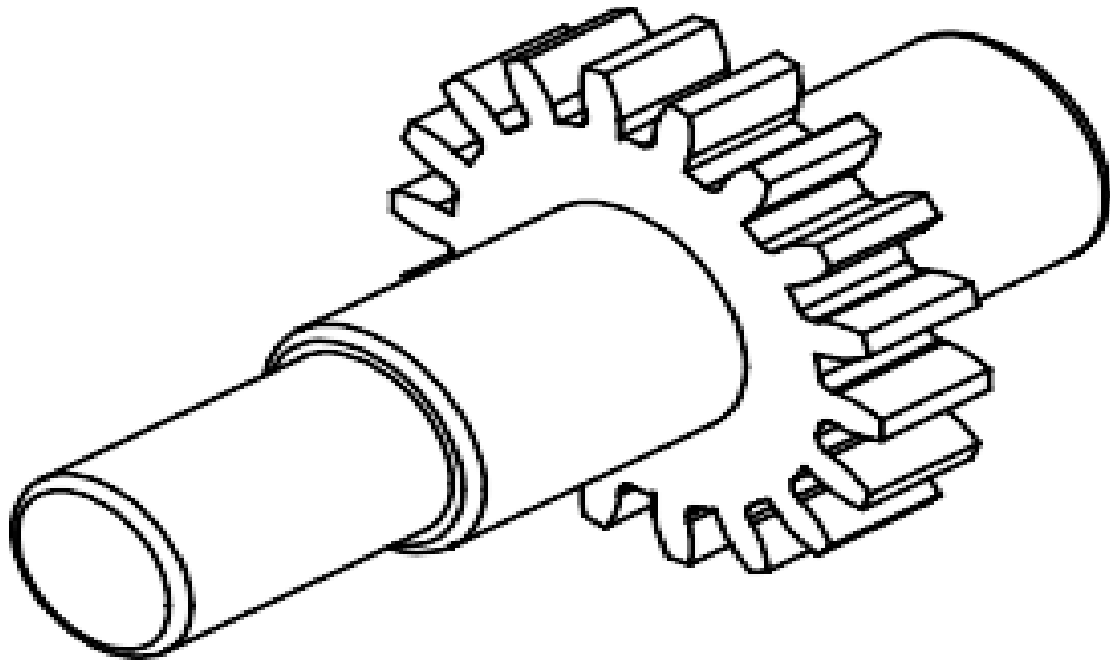
Performing spur gear with shaft

Operation Title: Spur gear with shaft

1. Copied the drawing.
2. Fix the work piece on machine indexing head chuck
3. mount the milling module cutter on the arbor.
4. Calculate all given dimension from drawing
5. Performed gear cutting operation milling with proper depth of cut.
6. Check the dimensions.

RESULT:

The spur gear is formed on the work piece. Key way is formed on the work piece. Studied the method of Indexing.



Operation Sheet 3.4.

Performing slotting operation

Operation Title: Slotting on milling machine

Purpose: To perform the slotting on milling machine

Tools and equipment

Milling machine, key slot milling cutter, vernier caliper

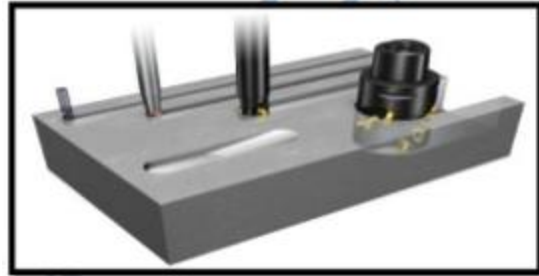
Material

Aluminum or m/steel block/shaft

Procedure

1. Hold the work piece in the table of the milling machine using the vice

2. Place parallels under the work piece to raise the surface to be milled above the level of the vise jaws.
3. Raise the table to the required position
4. Adjust the cutting speed and feed
5. Start the machine after selecting the proper speed and feed
6. Mark the dimensions on the work piece perform the slotting operation



LAP Test 3.1

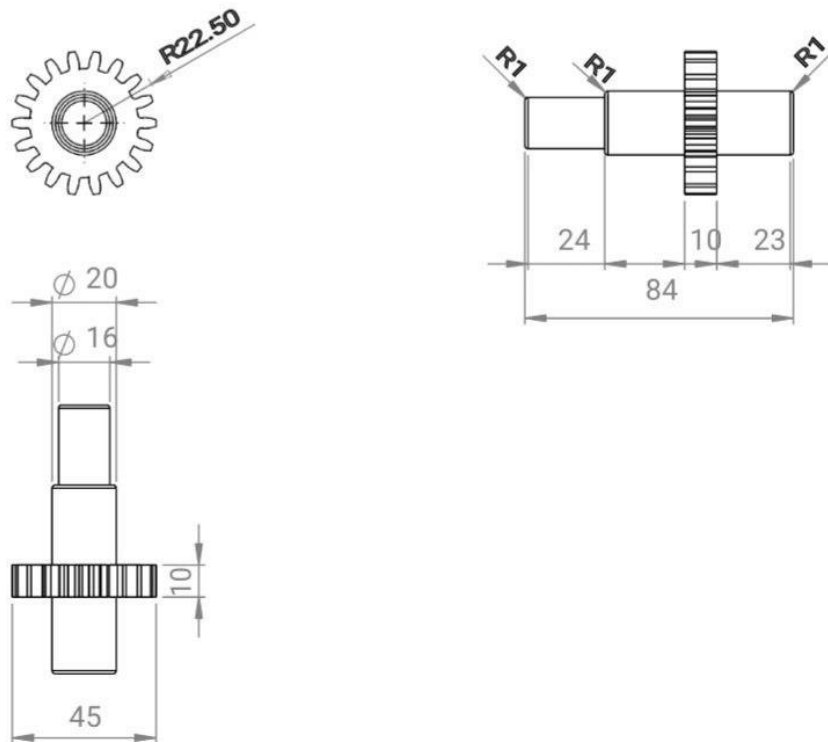
Practical Demonstration

Name: _____ Date: _____

Time started: _____ Time finished: _____

Instructions: Given necessary templates, tools and materials you are required to perform the following tasks within --- hour.

Task 1. Cut spur gear on milling machine



Task 2. Cut key way on milling machine

Task 3. Calculating speeds and feeds

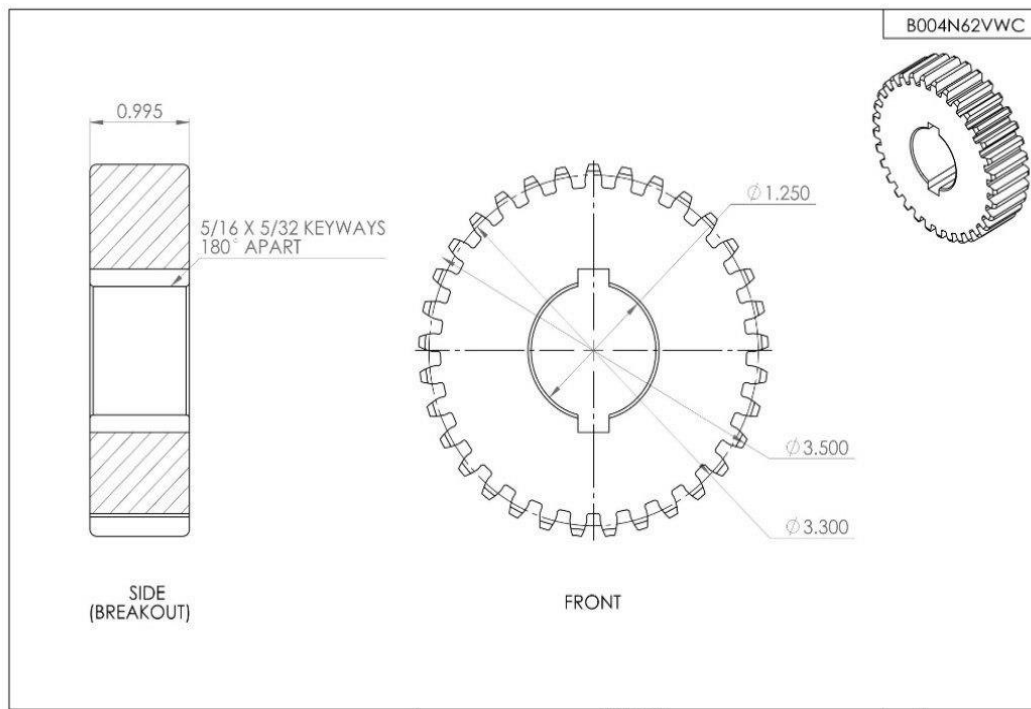
- A. Calculate the r/min required for a 75-mm diameter highspeed steel milling cutter when cutting machine steel (CS 30 m/min).
- B. Calculate the feed in millimeters per minute for a 75-mm diameter, six-tooth helical carbide milling cutter when machining a cast-iron workpiece (CS 60).

Task 4. Perform the key way on milling machine

Task5. Perform the slotting on milling machine

Task 6. Mill T-slots

Task7.perform helical gear operation on milling according to the give drawing



Unit Four : Product Quality

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

- Inspect components
- handling deviation
- Assure Quality.

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:

- Inspect components
- Handle deviation
- Assure Quality

4.1. Inspect Components

4.1.1. Introduction Checking /measuring work-piece

Inspecting components refers to the process of examining and evaluating manufactured parts or components to ensure they meet the required specifications, quality standards, and functional requirements. Component inspection is a critical step in quality control and is performed to identify any defects, deviations, or non-conformities that may affect the performance or reliability of the part.

Component inspection can involve various methods and techniques depending on the type of component and the specific requirements. Here are some common methods used in component inspection:

Visual Inspection: This is a basic inspection method that involves visually examining the components for any visible defects, such as cracks, surface irregularities, scratches, or discoloration.

Dimensional Inspection: This method assesses the dimensional accuracy of the components by using measuring tools, such as calipers, micrometers, gauges or coordinates measuring machines (CMM). It ensures that the components meet the specified tolerances and dimensional requirements.

Non-Destructive Testing (NDT): NDT techniques are used to inspect components without causing damage to them. Examples of NDT methods include ultrasonic testing, magnetic particle inspection, dye penetrant testing, radiographic testing, and eddy current testing. These techniques are employed to detect internal and surface defects, cracks, voids, or material inconsistencies.

Functional Testing: Functional testing involves verifying that the components perform their intended function correctly. It may include applying mechanical, electrical, or environmental conditions to the components to assess their performance and reliability.

Material Analysis: Component inspection can also involve analyzing the material composition and properties to ensure they meet the required specifications. This may involve using techniques such as spectroscopy, hardness testing, or material strength testing.

4.1.2. Reading Metric Rules

Metric rules are graduated in millimeter and one-half (0.5) millimeter divisions. Quick-reading numbers every 5 or 10 millimeters simplify measurement.

Calipers

Page 106 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
-----------------	--	--	------------

Calipers used in semi-precision measurement have two legs that make contact with part surfaces to obtain measurements. One type of caliper that is very similar to the rule is the slide caliper. It can be used to measure external or internal dimensions. Readings are made where a reference line on the moveable jaw aligns with a graduation on the scale.



Figure 4.1 Calipers

Venire caliper and digital calipers

Venire caliper

A vernier caliper is similar to a semi-precision slide caliper, but its vernier scale allows it to be used for measurements as small as 0.001" or 0.02 mm. A vernier caliper has a solid jaw and a moveable jaw that are brought in contact with part surfaces to measure external dimensions. Internal dimensions are measured by placing the “nibs” between two surfaces.

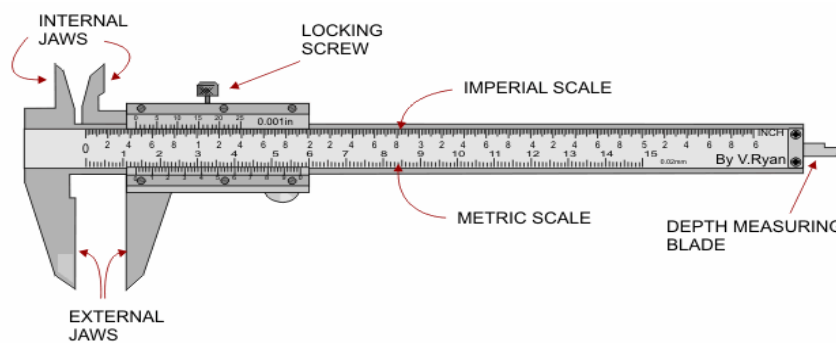


Figure 4.2. Parts of the Vernier caliper

Main Scale

Main scale is graduated in cm and mm.

Vernier Scale

- It slides on the main scale.
- On Vernier scale 0.9cm is divided into 10 equal parts.

Jaws

- Two inside jaws (Upper)
- Two outside jaws (Lower)

Least Count (Accuracy)

Least count (L.C) is the smallest reading we can measure with the instrument.

- $L.C = 1 \text{ main scale division} - 1 \text{ vernier scale division}$

$$L.C = 1\text{mm} - 0.9\text{mm}$$

$$L.C = 0.1\text{mm} = 0.01\text{cm}$$

- Least Count = Value of the smallest division on MS/ Total number of division on VS

$$L.C = 1\text{mm} / 50 = 0.02\text{mm}$$

Digital calipers

Most digital calipers measure in both English and metric units. To use, bring the jaws together and power the caliper on. This sets “0.” If the caliper is already powered on, press a button marked “zero” or “0” to set the zero point. A “zero” location can be set at any position in the caliper’s travel. When a measurement is taken, the entire measurement is displayed on the digital readout. To switch between inch and millimeter display, simply press an “inch/mm” or similar conversion button.

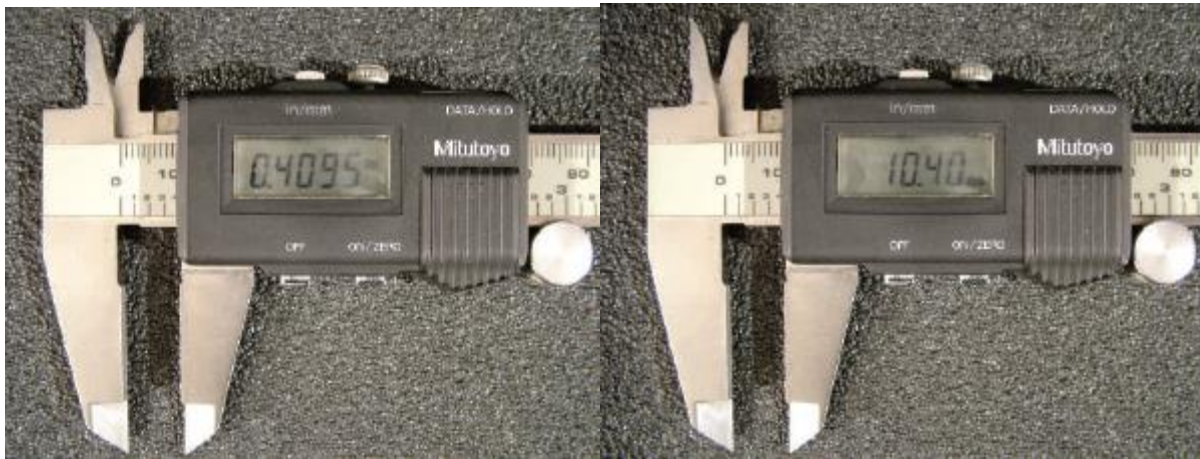


Figure 4.3. Digital calipers

Micrometer (internal, external and depth micrometer)

Working Principle Of The Micrometer:

- The micrometer works on the principle of screw and nut
- The longitudinal movement of the spindle during one rotation is equal to the pitch of the screw
- The movement of the spindle to the distance of the pitch or its fractions can be accurately measured on the barrel and thimble

Ranges Of Out Side Micrometer:

- Outside micrometer are available in ranges of 0 to 25mm, 25 to 50mm, 50 to 75mm, 75 to 100mm, 100 to 125mm, 125 to 150mm

Page 108 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
-----------------	--	-------------------------------------	------------

- For all range of micrometer, the graduations marked on the barrel is only 0-25 mm

Reading of outside micrometer

- Read the whole mm. graduation on the barrel above datum line
- Read half mm. graduation on the barrel below datum line
- Read the value of the visible line on the left of the thimble edge.

Main reading = 13.0mm

Half mm. reading= 0.5mm

Thimble reading= $13 \times 0.01 = 0.13\text{mm}$

Total reading= $13.0 + 0.5 + 0.13$
 = 13.63mm

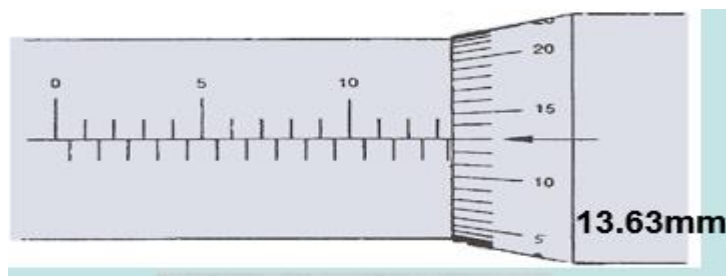


Figure 4.4. Micrometer reading example

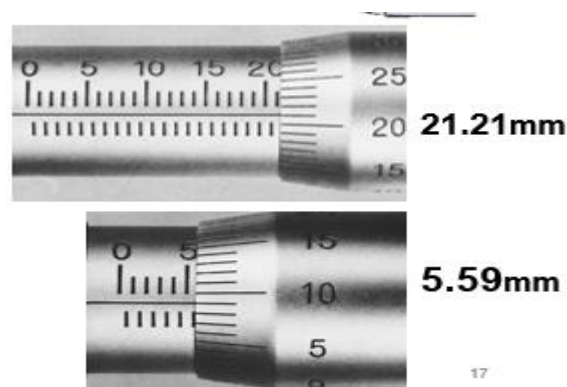


Figure 4.5. Micrometer reading exercises

Depth micrometers

Depth micrometers are special micrometers used to measure

- The depth of holes
- The depth of grooves and recesses
- The heights of shoulders or projections

Depth micrometers constructional features:

- The depth micrometer consists of a stock on which a graduated sleeve is fitted.
- The other end of the sleeve is threaded with a 0.5 mm pitch 'v' thread.
- A thimble which is internally threaded to the same pitch and from, mates with the threaded sleeve and slides over it.
- The other the thimble has an external step machined end of and threaded to accommodate a thimble cap.

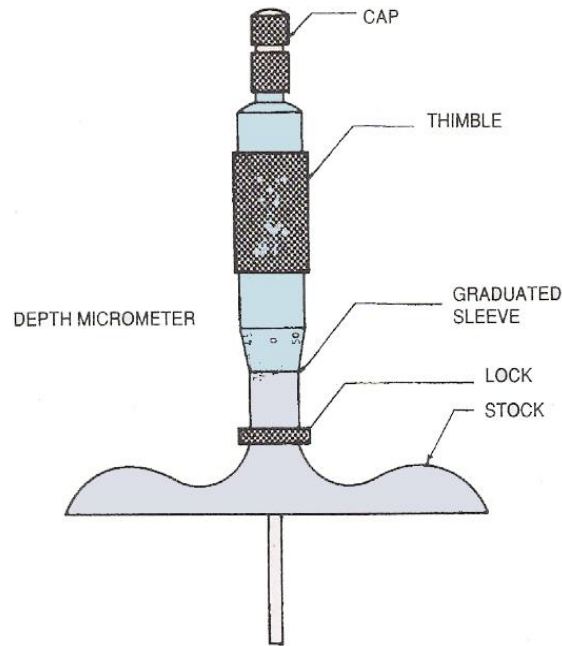


Figure 4.6. Depth micrometers

Vernier Height Gage

A vernier height gage is like a caliper mounted on a solid base for use on a surface plate. It measures vertical dimensions from the reference zero created by the surface plate's horizontal plane and has 0.001" or 0.02-mm graduations like the vernier caliper. Small vernier height gages might have a measuring range of 0–12" (0–300 mm), while large inch-based models can reach measurements up to 72" and metric models up to 900 mm. The base takes the place of the solid jaw and a removable point takes the place of the moveable jaw.

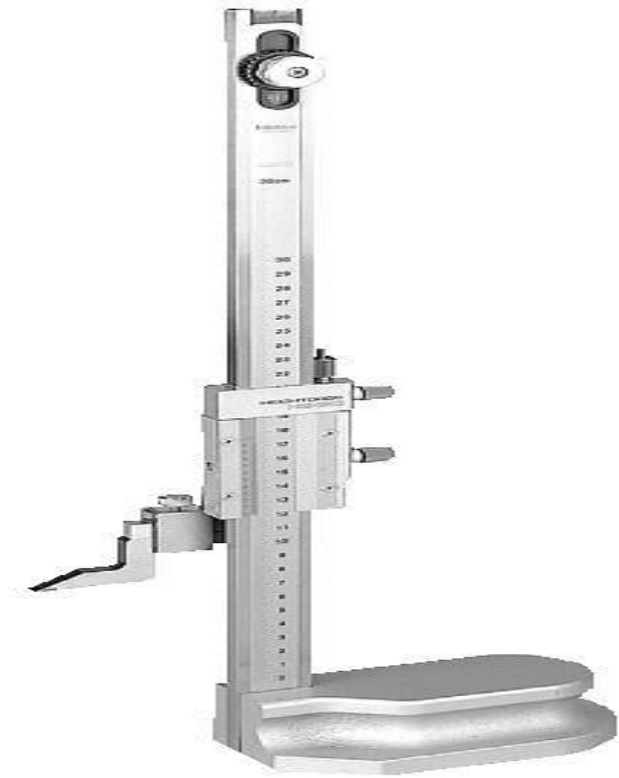


Figure 4.7. vernier height gage

Dial and Digital Bore Gages

When measuring internal diameters to close tolerances, a dial or digital bore gage can be used. Figure A and B shows some examples of bore gages, and Figure C shows a bore gage in use. A “zero” is normally set with a ring gage equal to the desired hole size. Then the gage is placed in the hole to be checked and variation from that size is read on the dial or digital readout.



Figure 4.8. (A) Digital bore gage (B) dial bore gage (C) A dial bore gage being used to check a hole

Surface Finish Measurement

Surface finish refers to the texture of the surface of a machined part. Roughness and waviness are the two factors generally considered when discussing surface finish. Roughness is the peaks and valleys created by the cutting action of a machining process. Waviness is the variation of those peaks and valleys over a larger distance. Figure below shows the difference between roughness and waviness.

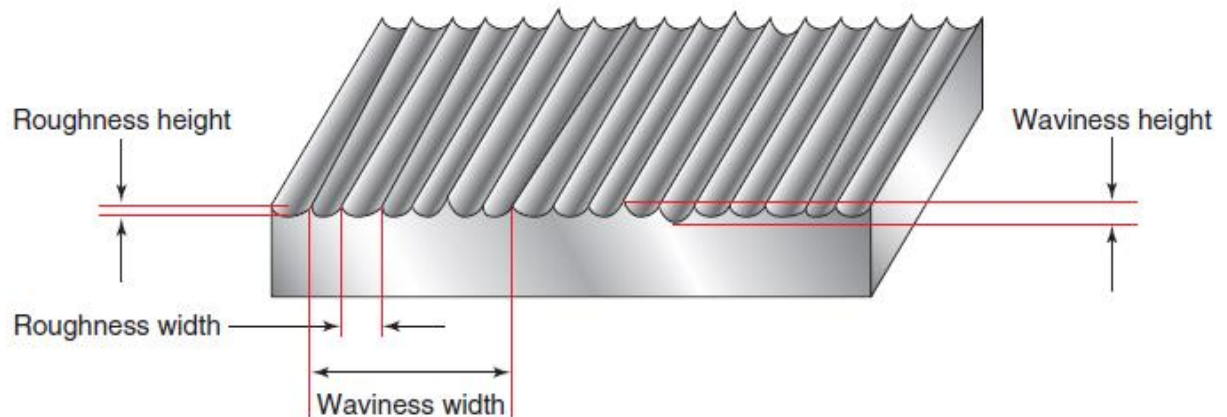


Figure 4.9. Surface characteristics

There are several systems for measuring surface finish, but the most widely used system measures roughness and is called Ra, or arithmetical average. It is a measure of the average height of a surface above a line that is midway between the highest peak and lowest valley of a surface within a given waviness length.

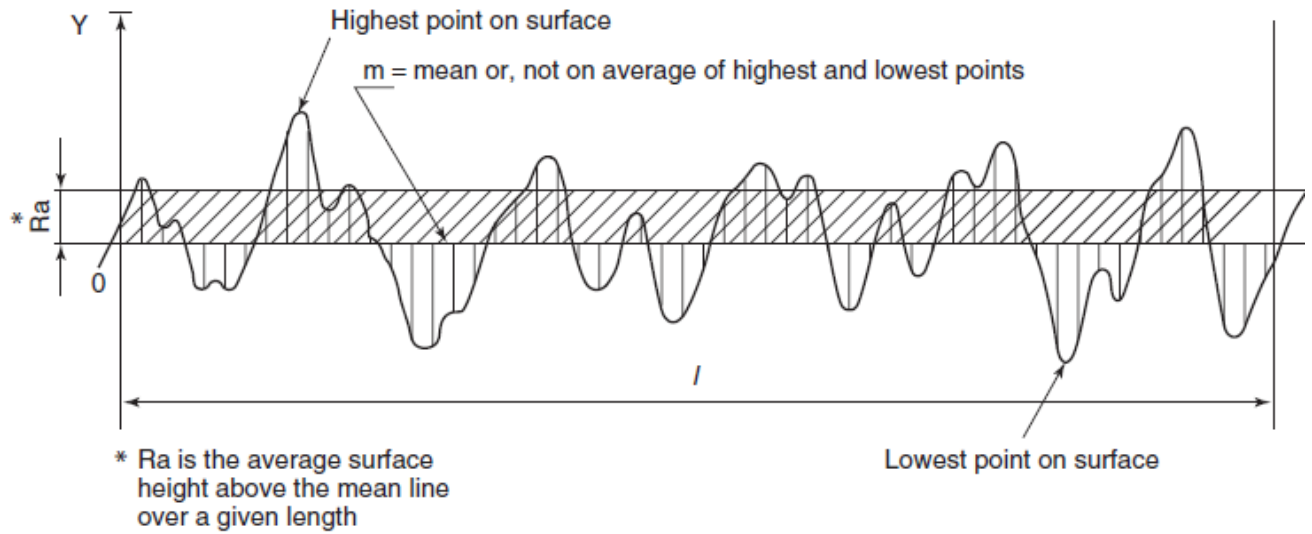


Figure 4.10. The basic concept of Ra surface finish measurement

Surface Roughness Comparator

Ra surface roughness is normally inspected by one of two different methods. A surface roughness comparator gage shows examples of different levels of Ra. Machined surfaces are visually compared to these samples to determine if they are within acceptable limits. This method does not give an actual measurement in micro inches or micrometers, but it is simple, highly portable, cost effective, and adequate for many applications.



Figure 4.11. Surface roughness comparator

Radius Gauges

Radius and Fillet Gages Radius and fillet gages are used to check outside corner and insider corner (fillet) radii. A set is shown in Figure A. Figure B shows how radius and fillet gages can be used.

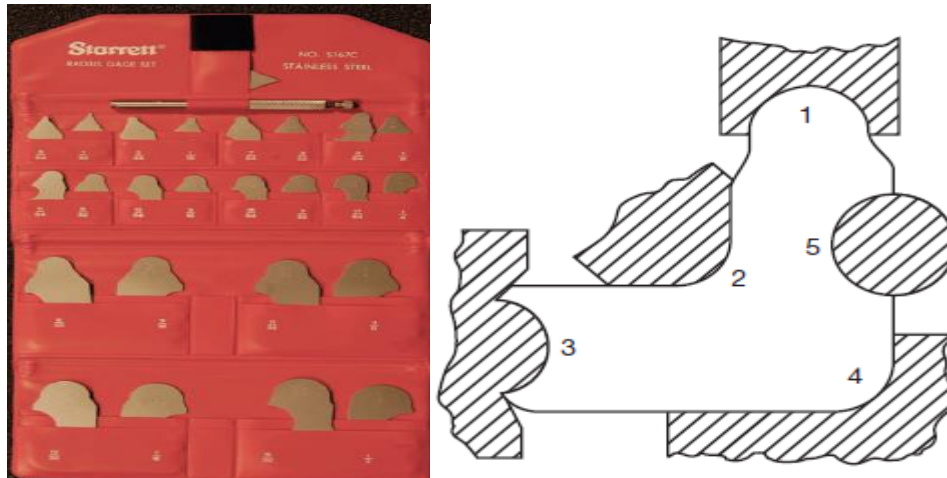


Figure 4.12. A. A radius and fillet gage set

B. Uses of radius and fillet gages

Gage Blocks

Gage blocks are extremely accurately sized blocks with very smooth surfaces that can be used for part inspection or to check the accuracy of other precision measuring tools. They are normally purchased in sets with a certain number of blocks of various sizes and are available in rectangular and square versions.

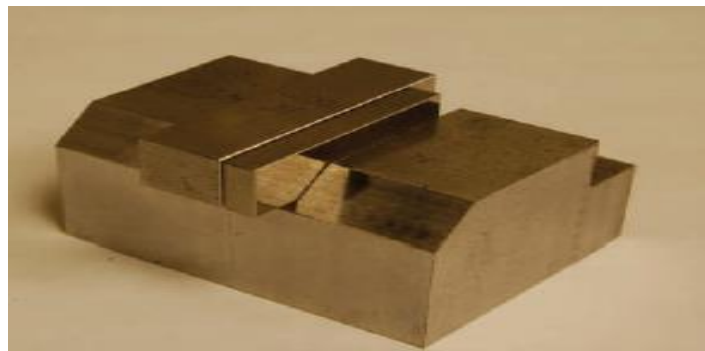


Figure 4.13. A typical rectangular gage block set Figure 4.14. Measuring a slot with gage blocks

Gear tooth caliper

The vernier gear tooth caliper combines two vernier scales with 0.001" or 0.02-mm graduations. It is used to measure the thickness of gear teeth at a certain depth. One movement is used to set the depth, similar to the vernier depth gage. Then the second movement is used to measure gear tooth width, similar to the vernier caliper.

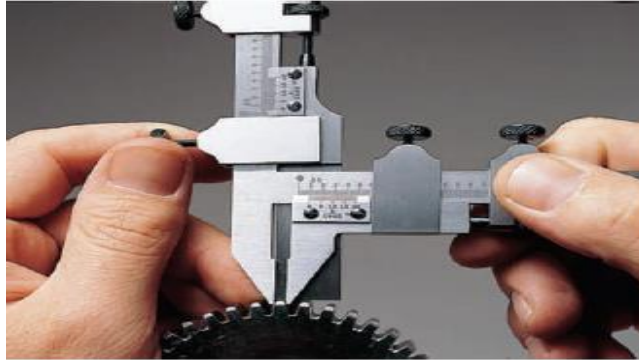


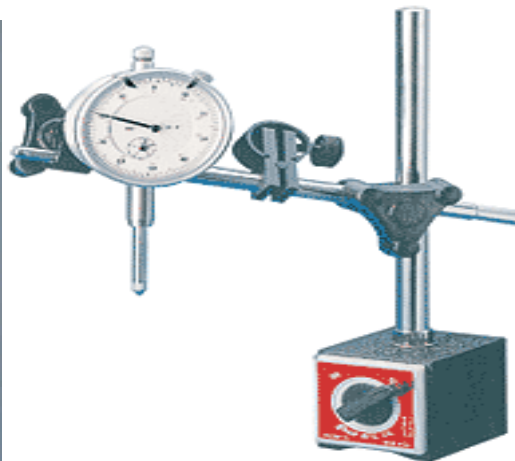
Figure 4.15. Vernier gear tooth caliper

a. Dial indicator

Dial indicators are used to center and align work on machine tools, check for eccentricity, and inspect manufactured parts. They are designed with shockproof movements and have jeweled bearings (similar to fine watches). Dial indicators must be mounted to rigid holding devices.



Figure 4.16. A. Dial indicators



B. Dial indicators with magnetic stand

Dial Indicators Discrimination of dial indicators typically ranges from .00005 to .001 in. In metric dial indicators, the discrimination typically ranges from .002 to .01 mm. Indicator ranges, or the total reading capacity of the instrument, may commonly range from .003 to 2.000 in., or .2 to 50 mm for metric instruments.



Figure 4.17. Dial indicators application

4.2. Handling Deviation

Quality Risk Management was mainly designed to be used prospectively when manufacturing operations are defined and validated. Therefore, potential deviations are identified and avoided by implementing risk control measures and preventive actions. QRM is based on the identification of product attributes and operational parameters which are critical to manufacturing operations in order to identify in advance their associated risks. This guidance document describes how this information may be used as criteria for the categorization and treatment of events, and eventually, deviations.

Under this approach, a sequence of steps may be identified when handling events and possible deviations:

1. Event Detection
2. Decision Making Process / Deviation Categorization

Page 116 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
-----------------	--	-------------------------------------	------------

3. Deviation Treatment
4. Root cause investigation
5. CAPA

4.2.1. Concept of deviations

The differences of measurement from the given tolerances and the differences of measurement between each value in from working drawing's dimensions.

Actual deviation.

This difference between a particular size and the basic. On the clearance fit diagram below the:-
 Lower the deviation on the hole = the minimum diameter hole – basic size. The upper deviation on the shaft = basic size – maximum Diameter shaft.

4.2.2. Errors in Measurement System

An error may be defined as the difference between the measured value and the actual value. For example, if the two operators use the same device or instrument for finding the errors in measurement, it is not necessary that they may get similar results. There may be a difference between both measurements. The differences that occurs between both the measurements. Systematic Error / Random Error

Examples:

- A worn out instrument: For example, a plastic tape measure becomes slightly stretched over the years, resulting in measurements that are slightly too high,
- An incorrectly calibrated or tarred instrument, like a scale that doesn't read zero when nothing is on it,
- A person consistently takes an incorrect measurement.

4.2.3. Organization procedures and standard for handling deviations

Policies and procedures are designed to influence and determine all major decisions and actions, and all activities take place within the boundaries set by them. Procedures are the specific methods employed to express policies in action in day-to-day operations of the organization.

Page 117 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
-----------------	--	-------------------------------------	------------

Causes of deviation

- Using the wrong tool
- In accurate taking care of operation
- Lack of operator skill
- Selecting wrong material

Corrective and Preventive Action

Corrective and Preventive Action (CAPA) focuses on the investigation of deviations. It does so in an attempt to either prevent their recurrence or their occurrence in the first place. To ensure the effectiveness of any corrective and preventive actions, organizations should continue monitoring them after the completion of the RCA and overall investigation. When it comes to CAPA, regulatory authorities expect organizations to ensure:

- The identified CAPA addresses the root cause;
- The solution can be implemented;
- There is clear understanding of the overall impact of the CAPA;
- Timelines and responsibilities (for implementation) have been reviewed and agreed to;
- There is a plan; and
- There is a monitoring phase. If an organization makes it through the investigation and determines the root cause, that forms just part of the equation. In the case of an inappropriate CAPA, further problems may ensue. Thus, the appropriate CAPA should be applied and monitored to ensure its effectiveness.

4.2.4. 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

Page 118 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
-----------------	--	-------------------------------------	------------

the boundaries of the tolerance zone are called upper and lower deviations as depicted in Figure 4.18.

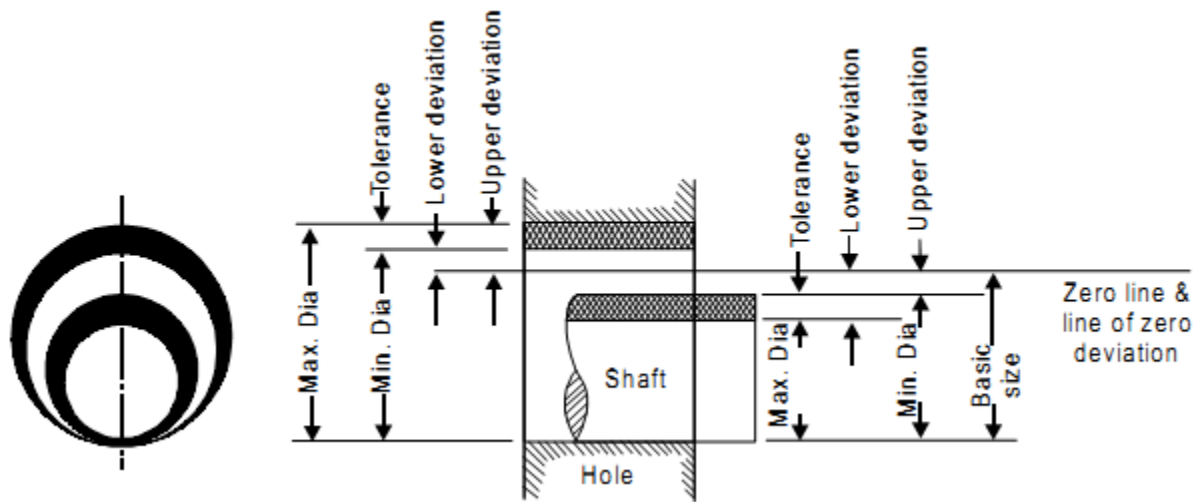


Fig. 4.18 Upper and lower deviations for both shaft and hole

4.2.5. Upper Deviation

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

4.2.6. Lower Deviation

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

4.2.7. Mean Deviation

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

4.2.8. 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. It is shown in Figure 4.20. 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. The positive deviations are shown above and the negative deviations are shown below it.

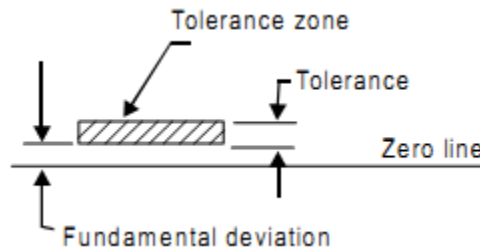


Figure. 4.20 Fundamental deviation

4.3. Assure Quality

4.3.1. Conformance in the manufacturing

Conformance at the manufacturing level is a subset of the overall quality definition. Manufacturing quality means conformance to requirements. Requirements include drawings, procedures, specifications, workmanship details, plus fitness of the product for the intended use. A simple definition of manufacturing conformance says:

Conformance is obtained when manufacturing follows an unbroken sequence of operations from beginning to end for successfully meeting the requirements. All things produced must be in agreement and harmony with the requirements. Economics of conformance require making products right the first time and every time. Product conformance always involves cost and scheduled deliveries.

4.3.2. Quality Issues

- Machinability of the material to be processed is an important issue with regards to: surface roughness, surface integrity, tool life, cutting forces and power requirements. Machinability is expressed in terms of a 'machinability index' for the material.
- Rigidity of milling cutter, work piece and milling machine important in preventing deflections during machining.
- Selection of appropriate cutting tool, coolant/lubricant, depth of cut, feed rate and cutting speed with respect to material to be machined is important.
- Coolant also helps flush swarf from cutting area.

Page 120 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
-----------------	--	-------------------------------------	------------

- Regular inspection of cutting tool condition and material specification is important for minimum variability.
- Surface detail good.
- Surface roughness values in the range 0.2–25 μm Ra are obtainable.

Milling surface finish

The average surface finishes that can be expected on free-machining materials range from 60 to 150 min. Conditions exist, however, that can produce wide variations on either side of these ranges. For example, some inserts are designed with wiper flats (short parallel surface behind the tool tip).

Table 4.1.Milling surface finish

milling Operation	Tolerance Capability—Typical		Surface Roughness AA—Typical	
	mm	in	μm	μm
Milling			0.4	16
Peripheral	± 0.025	± 0.001		
Face	± 0.025	± 0.001		
End	± 0.05	± 0.002		

Typical tolerances and surface roughness values (arithmetic average) achievable in machining operations.

Quality Control

When you purchase parts or have them manufactured by another company, you must have a way to ensure that the parts are manufactured precisely enough.

Before paying for parts, most companies have a process to quality certify (QC) the parts against the drawing or model.

Larger batches of parts may use statistical methods in which a relevant sample of the parts are inspected instead of all the parts. Some companies require certification from the part vendor rather than inspecting parts themselves.

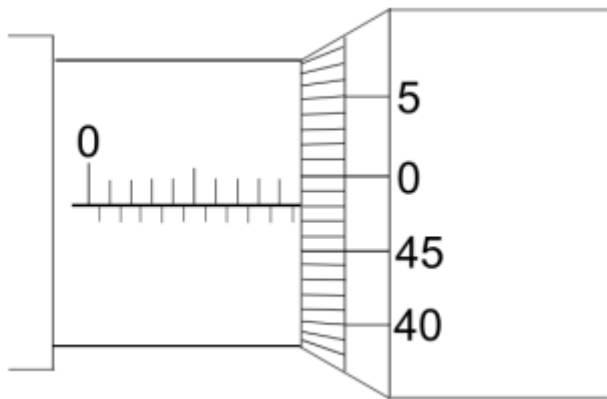
A tolerance must be specified for each dimension so that it can be determined how accurately the part must be manufactured to be acceptable. The tolerances that you specify are based on the part's function and fit.

Page 122 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
-----------------	--	-------------------------------------	------------

Self-Check 4.

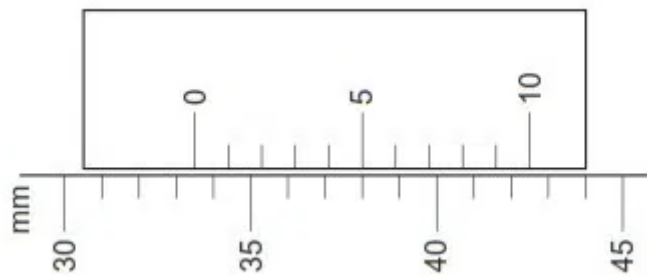
Part I: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. A tolerance is which one of the following:
 - a) Clearance between a shaft and a mating hole,
 - b) Measurement error,
 - c) Total permissible variation from a specified dimension,
 - d) Variation in manufacturing?
2. An outside micrometer would be appropriate for measuring which of the following (two correct answers):
 - a) Hole depth,
 - b) Hole diameter,
 - c) Part length,
 - d) Shaft diameter, and
 - e) Surface roughness?
3. What is the reading as shown in the figure below?



- a) 9.98 cm
- b) 9.48 cm
- c) 9.98 mm
- d) 9.48 mm

4. The dimensions of a small book is measured as 32.2 mm, 54.2 mm, and 13.7 mm. What measuring tool could have been used to obtain these readings?
- Vernier Caliper
 - Measuring Tape
 - Micrometer Screw Gauge
 - Meter Rule
5. The diagram shows part of a vernier scale.



What is the correct reading?

- 30.5 mm
- 33.5 mm
- 42.5 mm
- 38.0 mm

Part II: Answer all the questions listed below. Use the Answer sheet provided in the next page:

Instruction: Say true if the statements true or say false if the statements is false

- Measurements are always perfect.
- A machinist must understand the basic principles of measurement error and assume the proper responsibility in the selection, calibration, and application of measuring instruments.

Page 124 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
-----------------	--	-------------------------------------	------------

3. Tolerance is the total amount a specific dimension is permitted to vary.
4. Part designers are aware of measurement error.

Part III: Matching - Instructions: For the following questions match from “B” to “A” column

No.	<u>“A” Column</u>	<u>“B” Column</u>
_____ 1.	Deviation	A. The algebraic difference between the actual or maximum size and the corresponding the basic size
_____ 2.	Upper deviations	B. the algebraic difference between the minimum limit of any size of the part and the corresponding basic size
_____ 3.	Lower deviations	C. the algebraic difference between the two maximum limit of any size of the part and the corresponding basic size

Reference

Frederick E. Giesecke, Shawna Lockhart, Marla Goodman, Cindy M. Johnson (2016). *Technical Drawing with Engineering Graphics* (15th edition) Prentice Hall

Page 125 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
-----------------	--	-------------------------------------	------------

J.T. Black, Ronald A. Kohser(2011)DeGarmo's*Materials and Processes in Manufacturing* (11th edition)
Wiley

John R. Walker (2000). *Machining Fundamentals From Basic to Advanced Techniques* (7th edition) Goodheart-Willcox Company

John R. Walker, Bob Dixon. (2018). *Machining Fundamentals* (10th edition) Goodheart-Willcox

K. G. Swift, J. D. Booker (2013) Manufacturing Process Selection Handbook Butterworth-Heinemann

Kok, Chi-Wah_ Schmid, Steven R. Kalpakjian, Serope. (2010). *Manufacturing processes for engineering materials* (6th edition) Pearson-Prentice Hall

Page 126 of 128	Ministry of Labor and Skills Author/Copyright	Perform Advanced Milling Operations	Version -1
-----------------	--	-------------------------------------	------------

Developers Profile

No.	Name	Qualification (Level)	Field of Study	Organization/ Institution	Mobile number	E-mail
1	MesayTeshome	BSC	Manufacturing Technology	HPTC	905456888	Meayy2018@gmail.com
2	Tsegaye Mulat	MSc.	Manufacturing Technology	A.A tegbaried poly technic college	910411511	ttshgm97@gmail .com
3	Mesret woldemariam	MSc.	Manufacturing Technology	wolkite poly technic college	912117416	bmasi2007@gmail.com
4	Boka Terefe	MSc.	Manufacturing Technology	Ambo poly technic college	917038870	bokaterefe@gmail.com
5	Abebe hailye	MSc.	Manufacturing Technology	mejer General mulugeta bule poly technic college	913521369	aabebe52@gmail.com
6	Mohammed Bedri	MSc.	Manufacturing Technology	Robe poly technic college	913897297	
7	Mohammed Abdela	MSc.	Manufacturing Technology	Ethio italy poly technic college	973794977	mehamed abdela 49@gmail.com

