MACHINING -LEVEL II

Based on Version 2 February 2017

Occupational Standard (OS)

Training Module –Learning Guide 28-31

Unit of Competence: **Perform Intermediate Milling Operations**

Module Title:- **Performing Intermediate Milling Operations**

TTLM Code: **IND MAC2 TTLM09 1019v1**

October 2019

Module Title :- **Performing Intermediate Milling Operations**

TTLM Code: IND MAC2 TTLM09 1019v1

This module includes the following Learning Guides

LG 28: Determine job requirements

LG Code: IND MAC2 M09 LO1-LG-28

LG 29: Set-up work piece

LG Code: IND MAC2 M09 LO2-LG-29

LG 30: Perform milling operations

LG Code: IND MAC2 M09 LO3-LG-30

LG 31: Check/ measure work-piece

LG Code: IND MAC2 M09 LO4-LG-31

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

- Interpreting drawings of produce component
- Determine sequence of operation
- Selecting Cutting tools

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, upon completion of this Learning Guide, **you will be able to –**

- Interpret drawings to produce component to specifications.
- Determine sequence of operation to produce component to specifications.
- select cutting tools according to the requirements of the operational standards

Learning Instructions:

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below.
3. Read the information written in the “Information Sheets”. Try to understand what are being discussed. Ask your trainer for assistance if you have hard time understanding them.
4. Accomplish the “Self-checks” which are placed following all information sheets.
5. Ask from your trainer the key to correction (key answers) or you can request your trainer to correct your work. (You are to get the key answer only after you finished answering the Self-checks).
6. If you earned a satisfactory evaluation proceed to “Operation sheets
7. Perform “the Learning activity performance test” which is placed following “Operation sheets” ,
8. If your performance is satisfactory proceed to the next learning guide,
9. If your performance is unsatisfactory, see your trainer for further instructions or go back to “Operation sheets”.

Information Sheet-1	Interpreting drawings of produce component

1.1. Introduction

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. No matter how skilled you are at performing machining operations, if you are unable to properly interpret these drawings, you will not be able to produce machined components independently or efficiently within required specifications.

Engineering drawings can range from simple hand-drawn sketches to complex, computer-generated prints. They are produced in a standard format that enables machinists anywhere to understand them. Printed drawing sizes range from (A4) 210 × 297up to (A0) 841 × 1189 millimeters. Drawings for very complex components may contain several sheets.

1.2. Components of Engineering Drawings

Engineering drawings are made up of several components. These elements make up a standard system of views, lines, and symbols that provide important information about required specifications for machined components.

- **Title Block**

One major component of a drawing is the title block. The title block includes information such as the part name and number, tolerances, scale, material that the part should be made from, and any required heat treatment. Engineer and draftsperson names and drawing creation date are also usually included. A history of revisions, or changes, can be shown above the title block as well in a box often called the revision block. A revision block may also be shown in the upper-right corner of a print.

- **Scale** is the size of an actual object related to its size drawn on a print.

In all cases, regardless of scale, drawing dimensions shown are actual part sizes.

- **Bill of materials**: Sometimes a print may have a bill of materials. A bill of materials can either list the raw materials used to make the machined part or list components that are assembled to produce the part specified on the print.

A sample engineering drawing with the title block, revision block, and bill of materials labeled is shown in figure below.

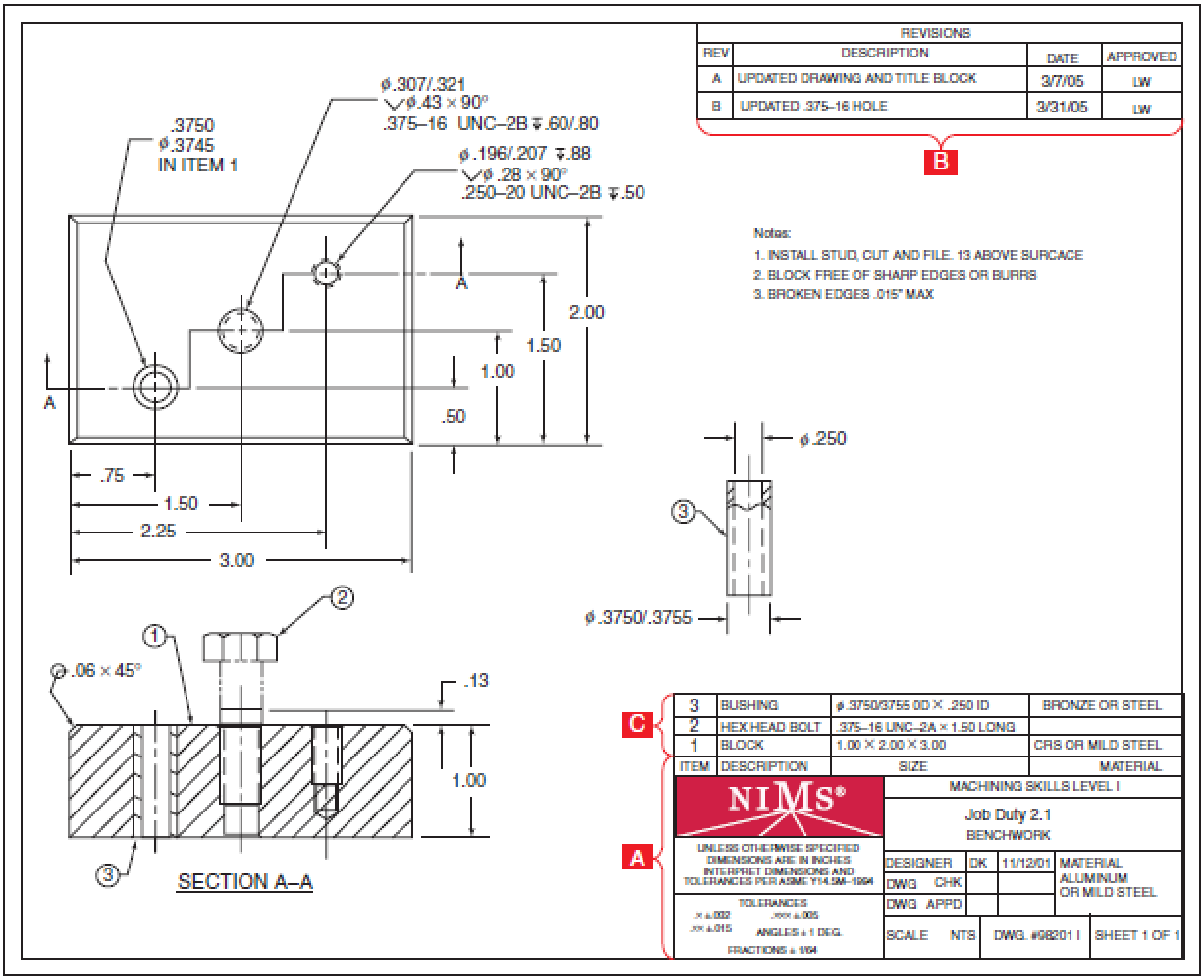


Figure 1.1. The title block (A) of an engineering drawing, or print, contains information including part name, tolerances, and scale. The revision block (B) shows a list of changes that have been made over time. The bill of materials (C) lists raw material or components used to produce the part.

1.3. 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. A simple part may only require one view to show all of the information needed to machine it. A more complex part will require more views to aid in developing a mental image of its shape. This method of representing a three-dimensional object in two dimensions using different views is called orthographic projection.

The principal views are the six mutually perpendicular views that are produced by six mutually perpendicular planes of projection. If you imagine suspending an object in a glass box with major surfaces of the object positioned so that they are parallel to the sides of the box, the six sides of the box become projection planes showing the six views as shown in figure below. The six principal views are front, top, left side, right side, bottom, and rear.

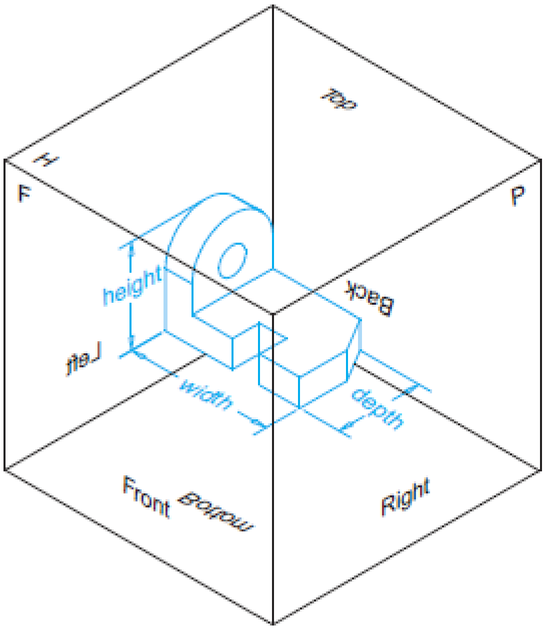


Figure 1.2. Object placed inside glass box

To understand orthographic projection, imagine an object inside a glass box with hinges where the sides meet. The top is hinged to open above the front, and the right side is hinged to open to the right of the front, as shown in Figure below.

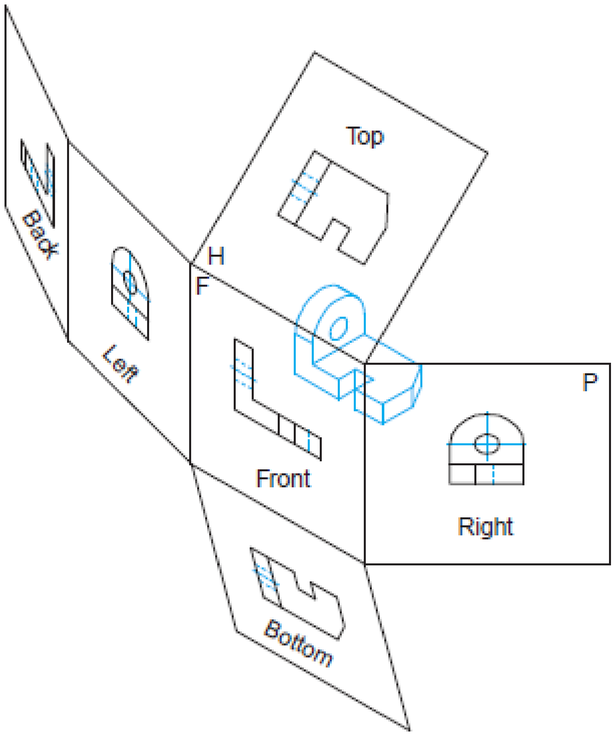


Figure 1.3. Glass box being opened

All six principal views are not normally required to completely document an object. Note in Figure below the similarities between top and bottom, front and back, and left and right. Three principal views are sufficient to fully describe most objects.

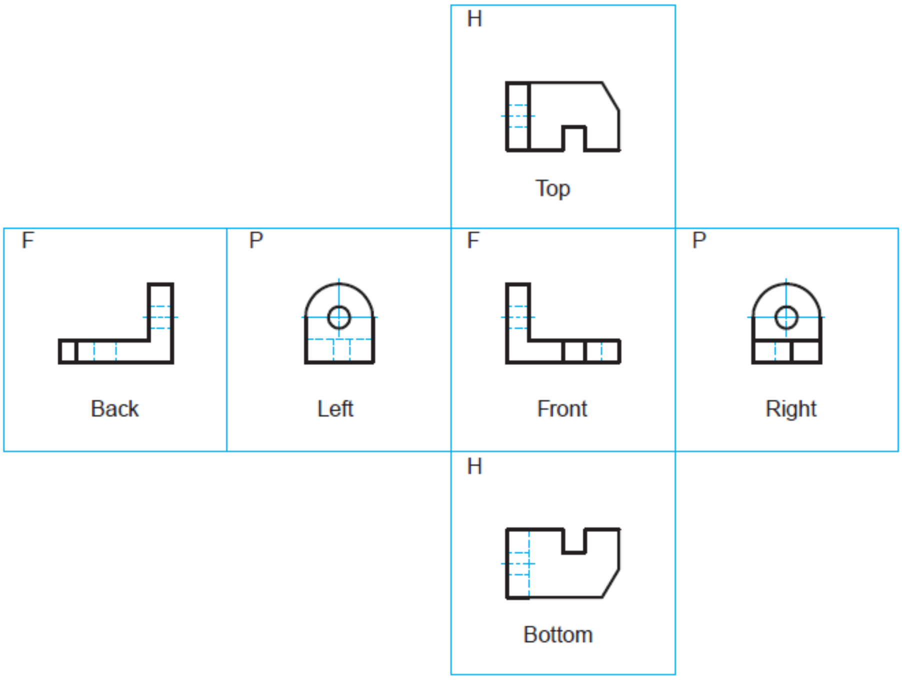


Figure 1.4. Glass box opened and laid out flat

Most commonly these views are top, front, and right, as seen in Figure below. In line with this, we normally speak of three (not six) mutually perpendicular projection planes: Horizontal (H), Frontal (F), and Profile (P). The top and bottom views are projected onto H, front and back onto F, left and right onto P.

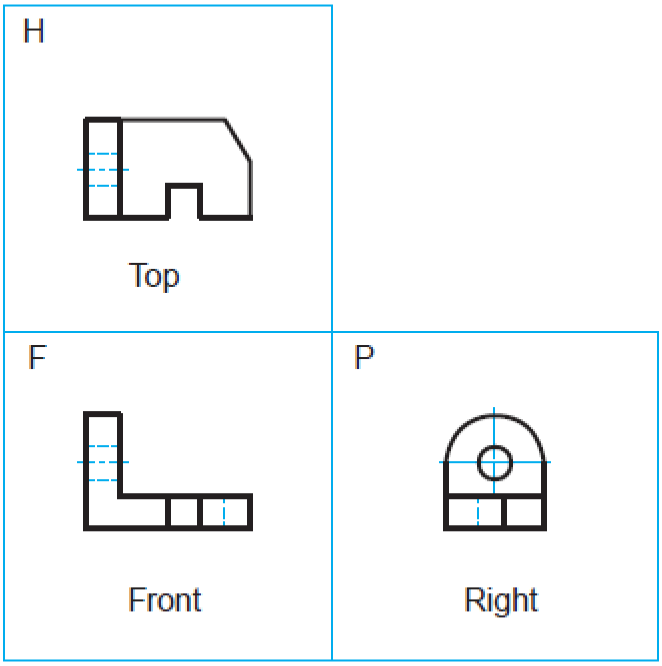


Figure 1.5. Top, front, right multiview arrangement

- **Alignment of Views**

Multiviews are always aligned according to the dictates of the glass box projection planes and their fold lines. As can be seen in Figure below, not only the extents but also the internal features should be aligned.

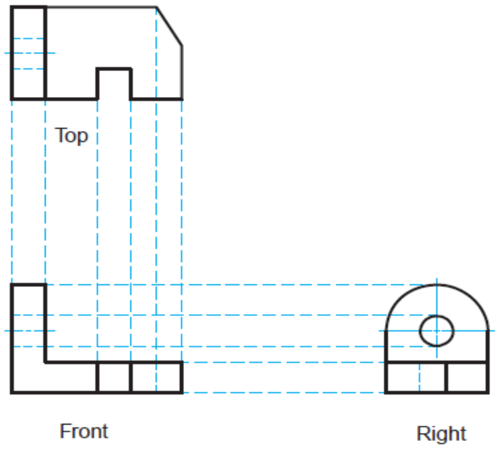


Figure 1.6. Feature alignment

- **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.

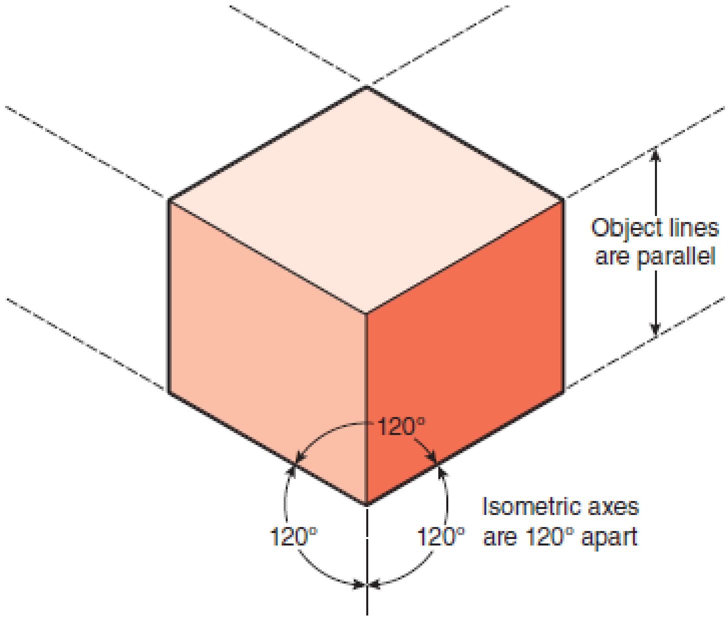


Figure 1.7. An isometric drawing

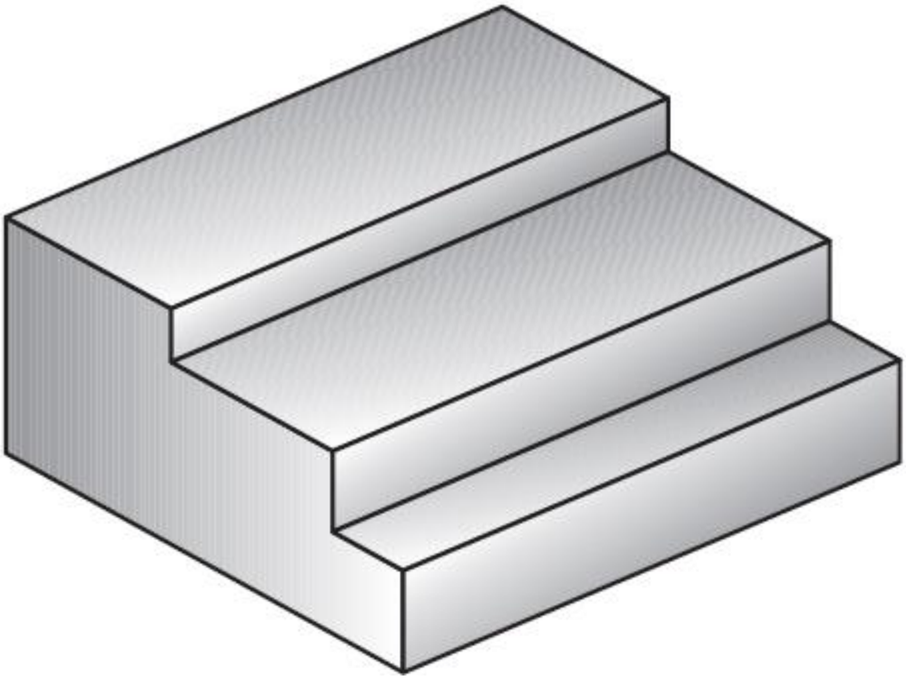


Figure 1.8. This simple isometric view shows how an object would appear in three dimensions. Isometric views are very helpful when trying to visualize complex objects

- **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.

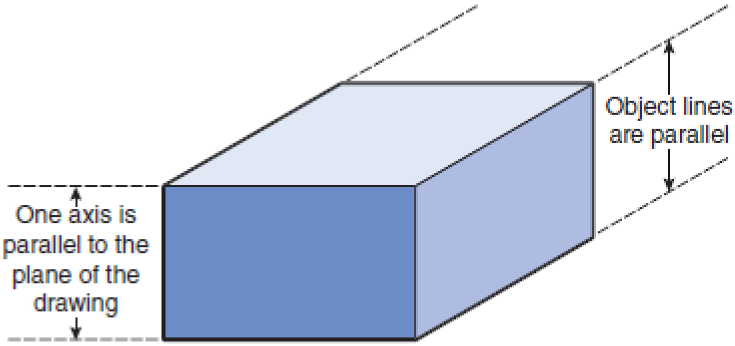


Figure 1.9. An oblique drawing

- **Exploded drawings**

The exploded drawing is a type of pictorial drawing designed to show several parts in their proper location prior to assembly. Although the exploded view is not used as the working drawing for the machinist, it has an important place in mechanical technology. Exploded views appear extensively in manuals and handbooks used for repair and assembly of machines and other mechanisms.

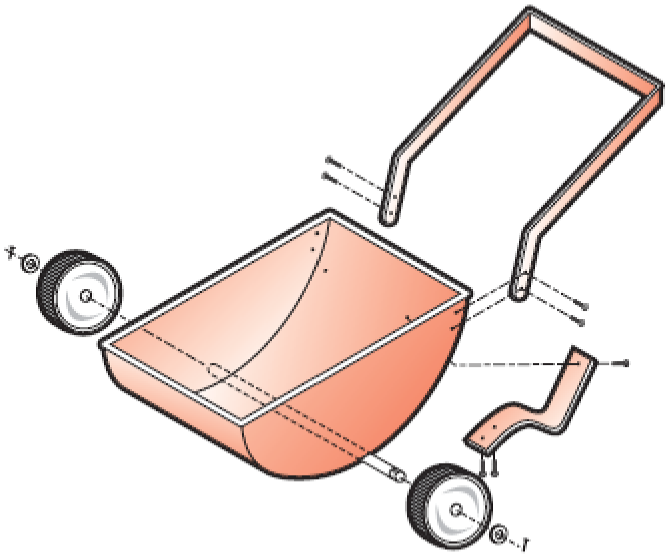


Figure 1.10. An exploded drawing

- **Line Types**

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. These different line types are used to form a drawing in the same way different letters of the alphabet are used to form words.

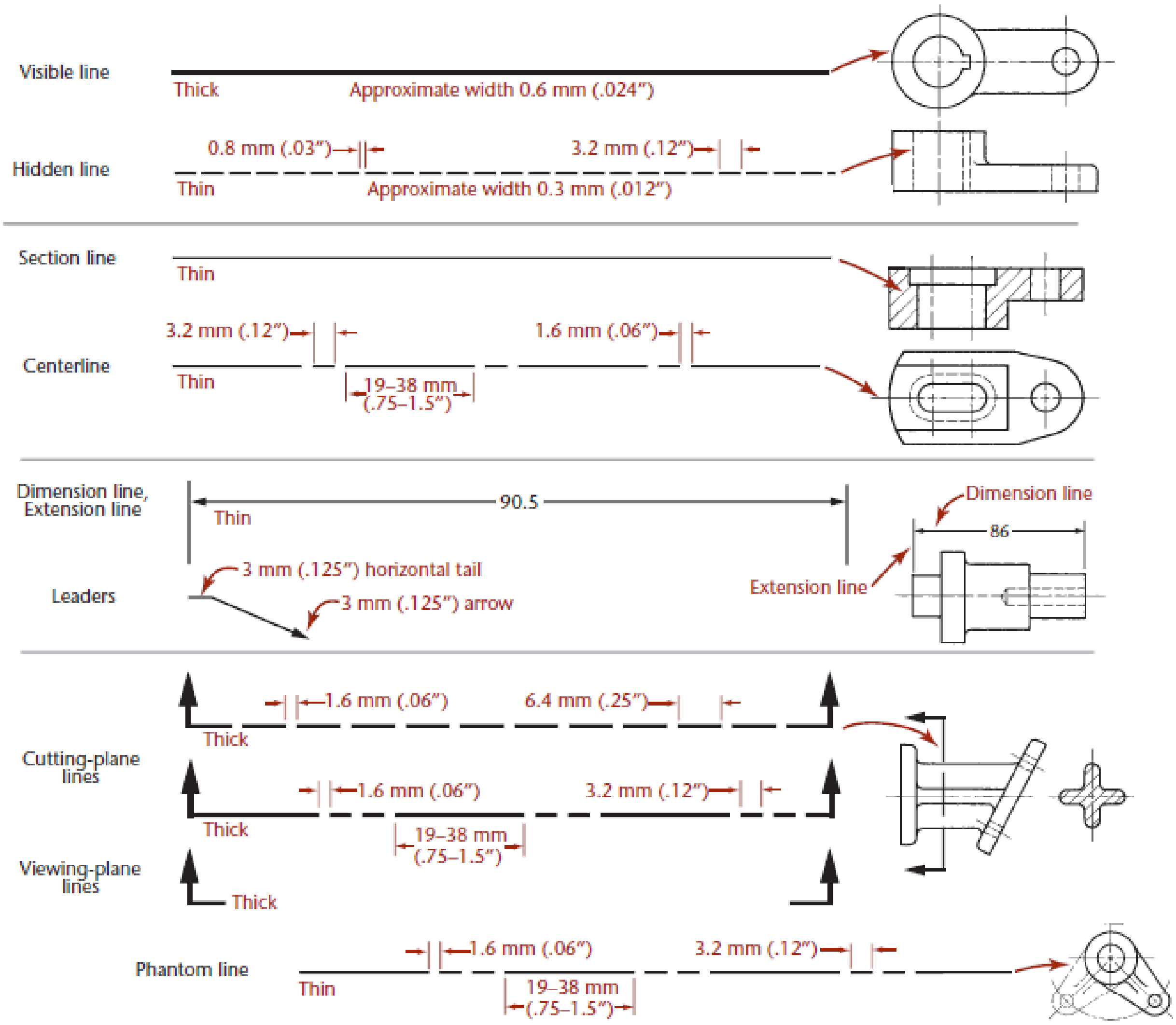


Figure 1.11. Alphabet of Lines

- **Sectioned Views**

When internal features are complex to the extent that showing them on a drawing as hidden lines would be confusing, a sectioned drawing may be employed.

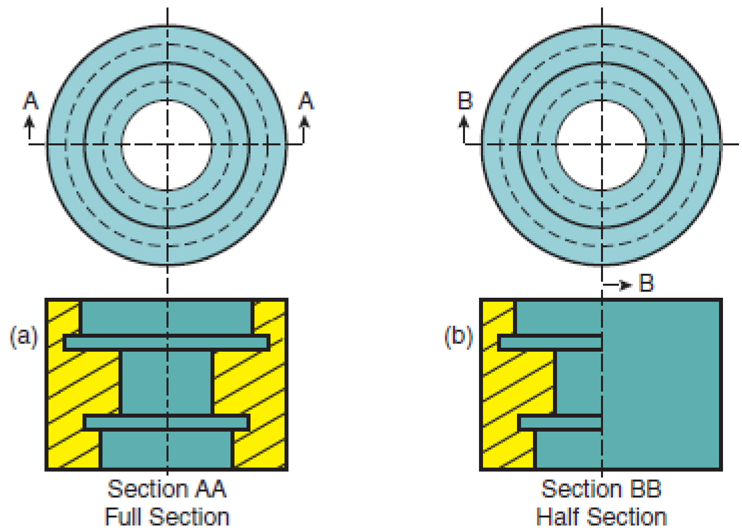


Figure 1.12**.** Drawing showing sectional views

- **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.

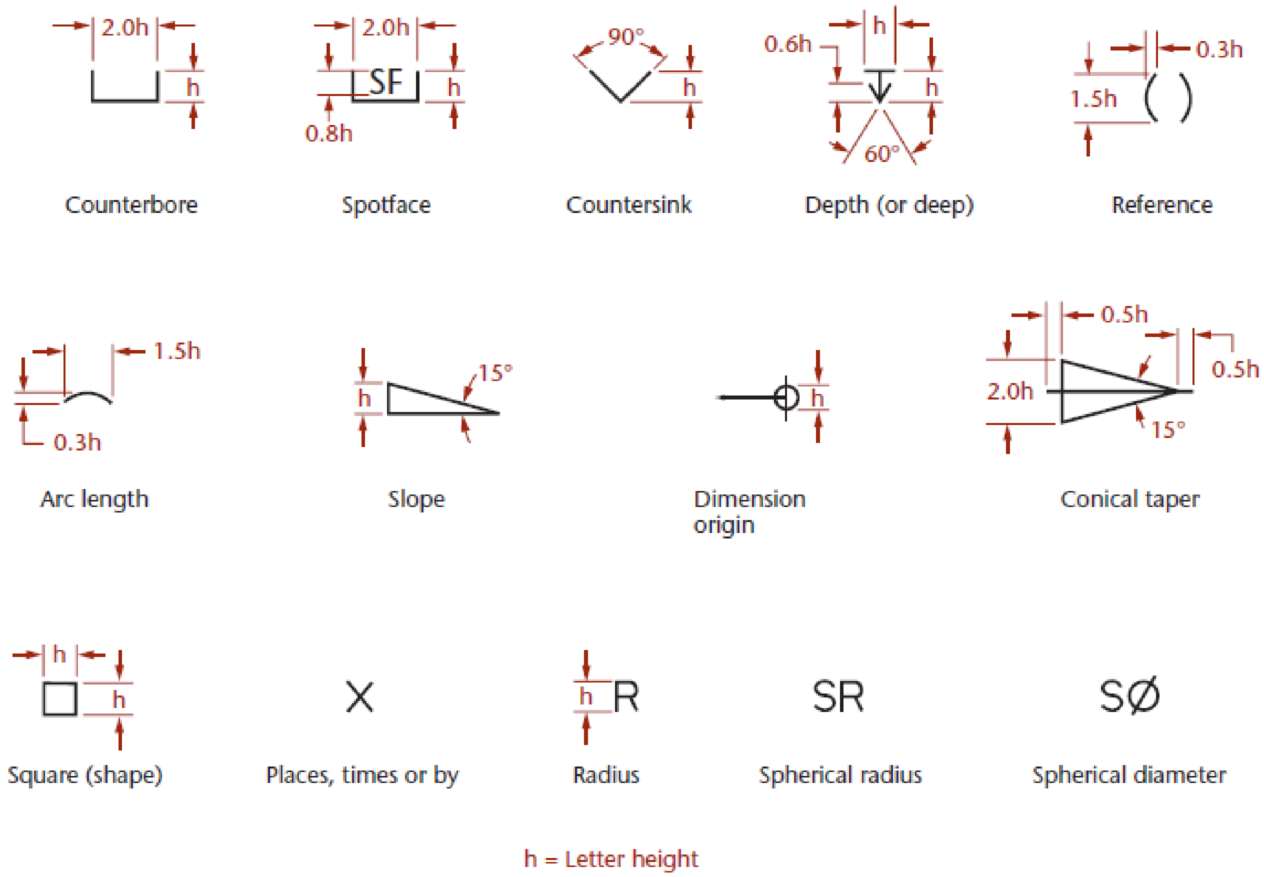


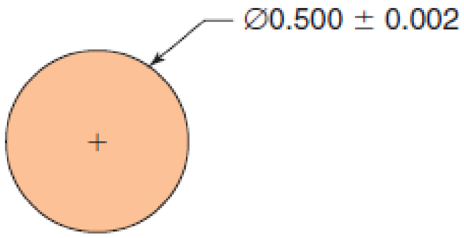
Figure 1.13. Form and Proportion of Dimensioning Symbols

- **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. The difference between the upper limit and the lower limit is the total tolerance. The amount of a tolerance can vary greatly depending on the intended use of a machined component, but the principle of interpreting any tolerance is the same.

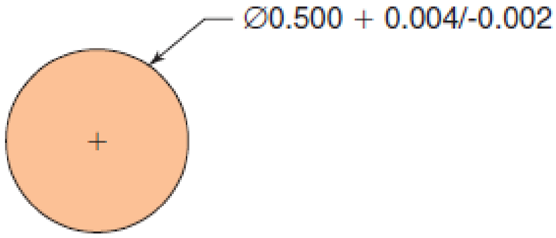
Bilateral Tolerances

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



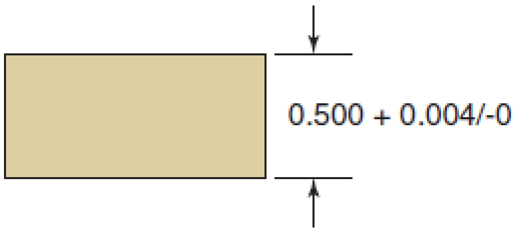
The 0.500 + 0.004/-0.002 shows different amounts of allowable variation above and below basic size.

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 1.14. Bilateral tolerance examples

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 1.15. 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.

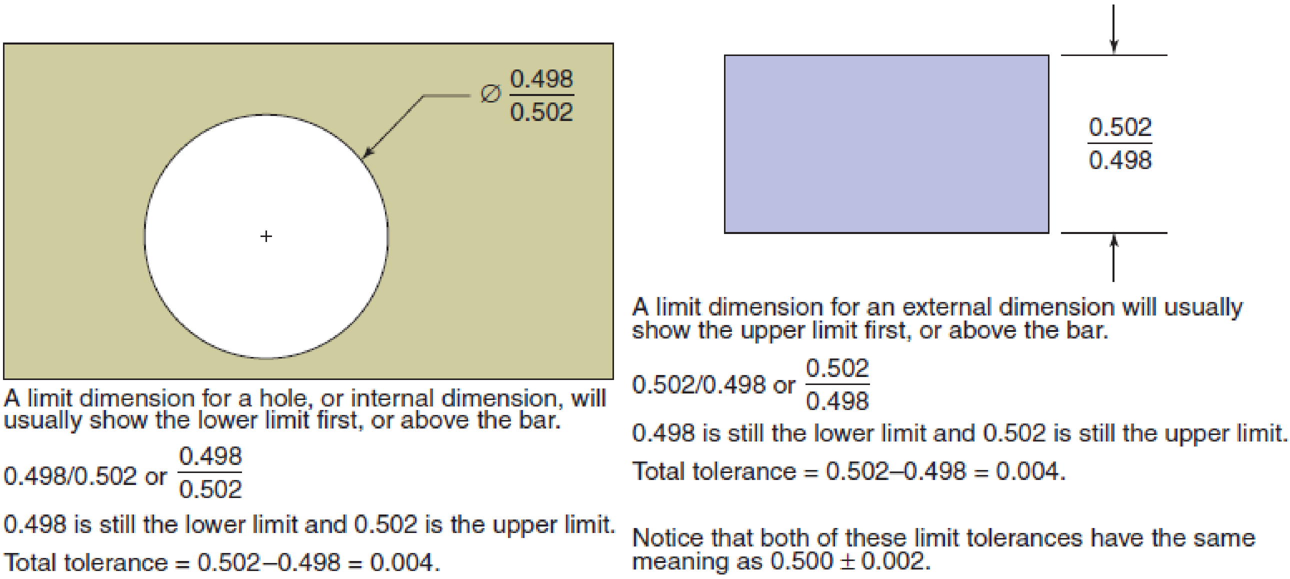
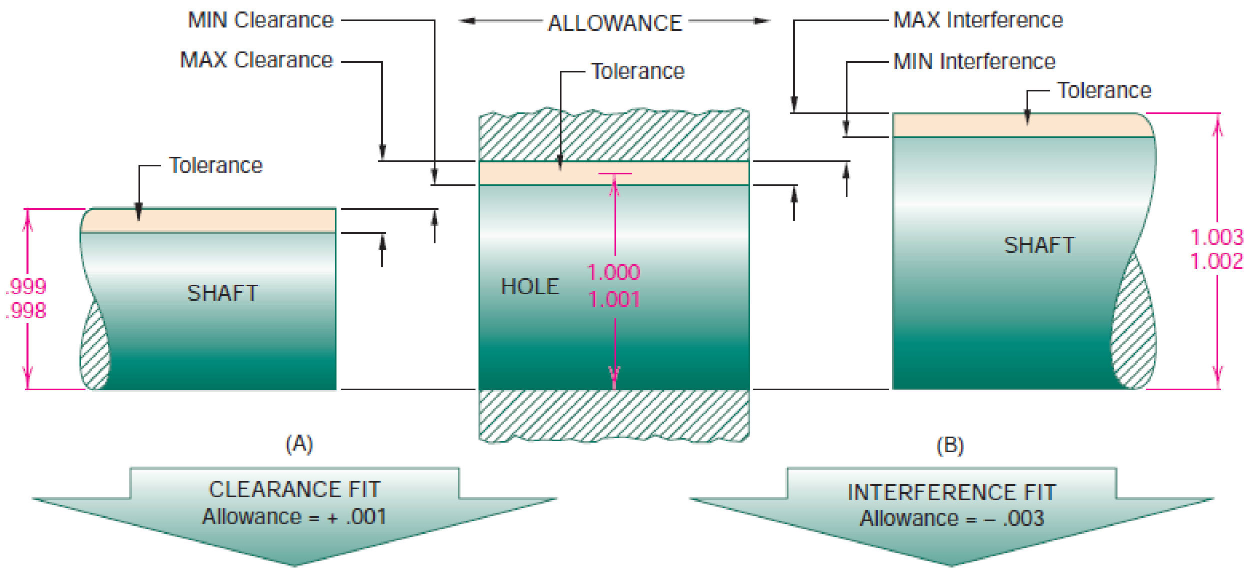


Figure 1.16. Limit tolerance examples

- **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.



Allowance always equals smallest hole minus largest shaft

Figure 1.17. Clearance and interference fits between two shafts and a hole

Shaft A is a clearance fi t, and shaft B is an interference fit.

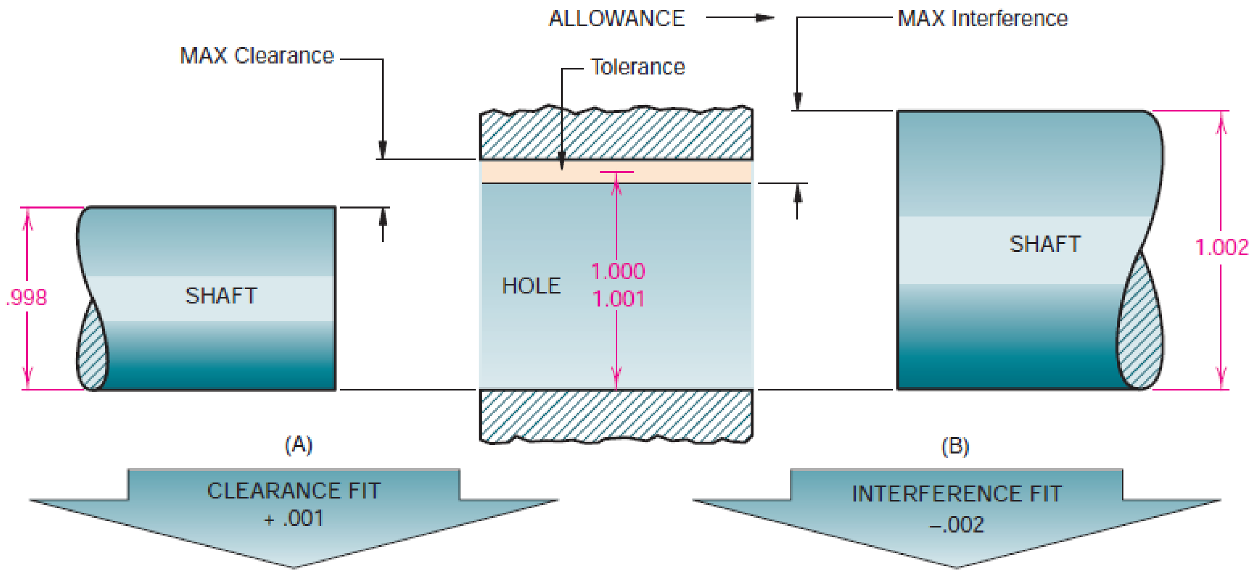


Figure 1.18. Transition fit between a shaft and a hole

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

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

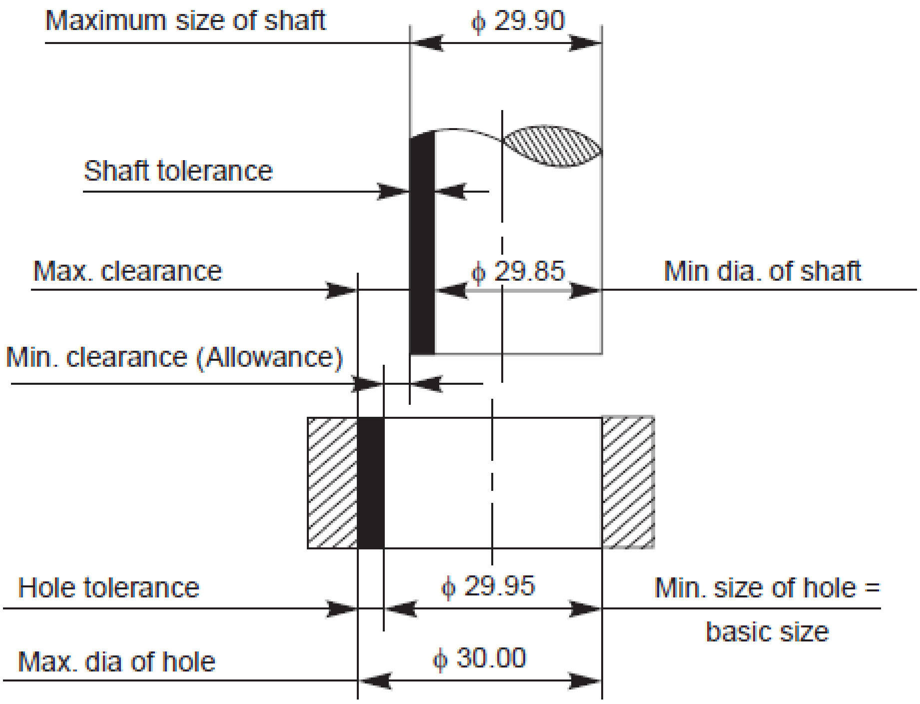


Figure 1.19. Clearance fit

Interference fit occurs when two toleranced mating parts will always interfere when assembled. An 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. In order to assemble the parts under this condition, it would be necessary to stretch the hole or shrink the shaft or to use force to press the shaft into the hole. Having an interference is a desirable situation for some design applications. For example, it can be used to fasten two parts together without the use of mechanical fasteners or adhesive.

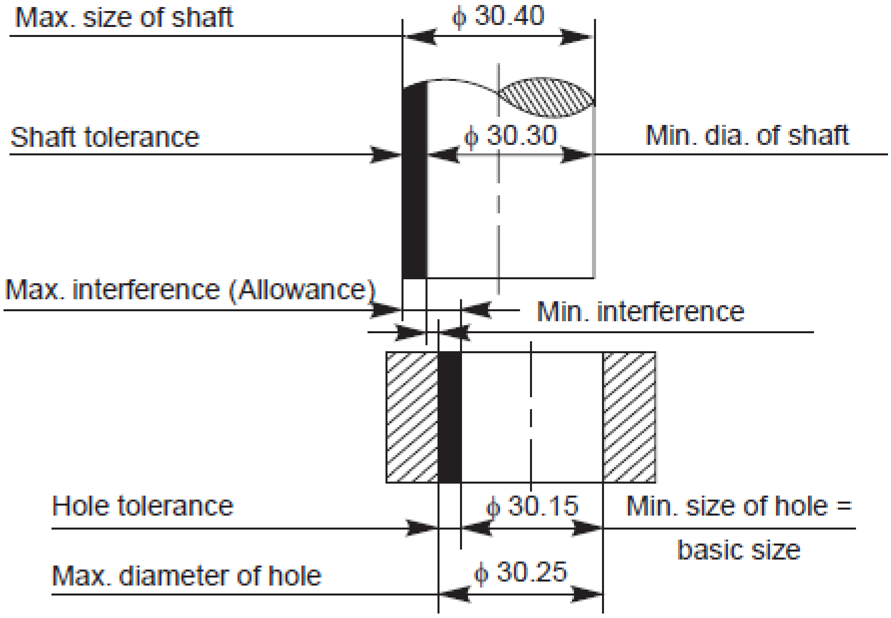


Figure 1.20. Interference fit

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

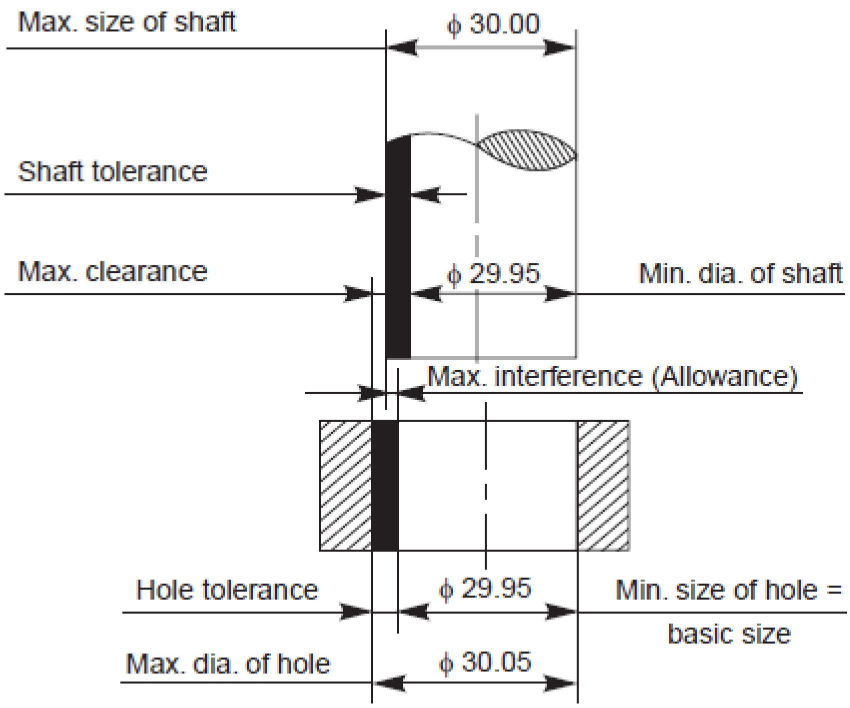


Figure 1.21. Transition fit

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 1.22. 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.

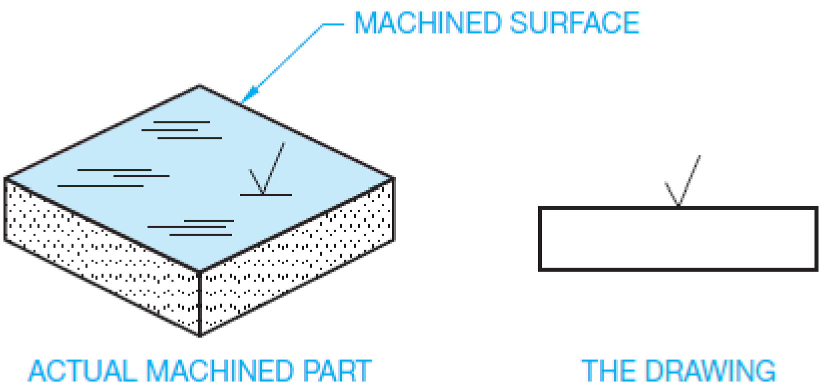


Figure 1.23. Standard surface finish symbol placed on the edge view. The symbol should always be placed horizontally when unidirectional dimensioning is used.

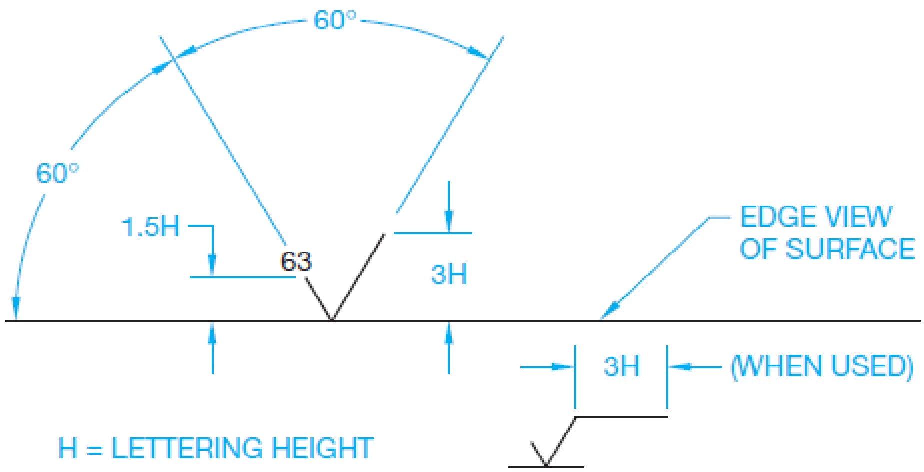


Figure 1.24. Properly drawn surface finish symbol

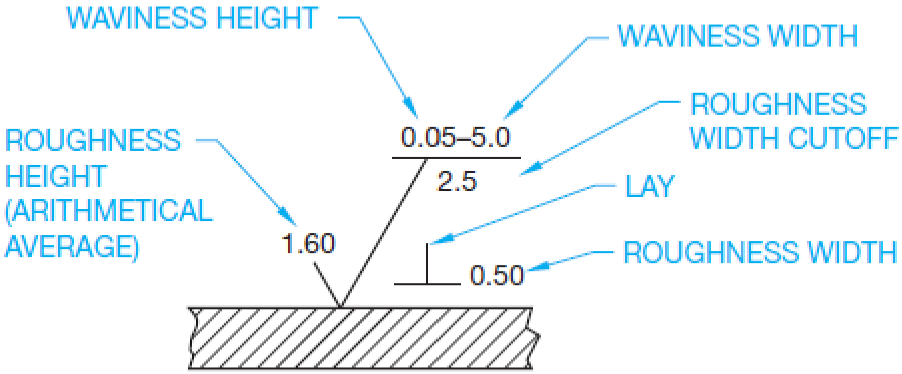


Figure 1.25. Elements of a complete surface finish symbol

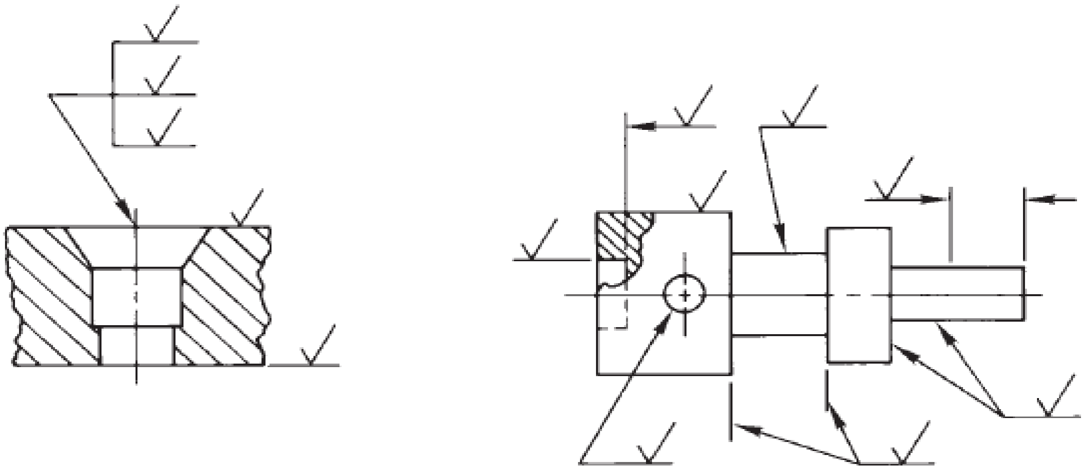


Figure 1.26. Proper placement of surface finish symbols

Self-Check -1	Written Test

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

- Drawings are used to:
  a) Show, in multiview, what an object looks like before it is made.
  b) Standardize parts.
  c) Show what to make and the sizes to make it.
  d) All of the above.
  e) None of the above.
- 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.
  e) None of the above.
- Print reading falls into two general categories. They are
  a) Visualization and tolerancing
  b) Interpretation and dimensioning
  c) Visualization and interpretation
  d) Projection and orthographic
- when tolerances are plus and minus, it is called a_________________ tolerance
  a) Bilateral tolerance b) Unilateral tolerance
- When tolerances are only plus or only minus, it is called a______________ tolerance
  a) Bilateral tolerance b) Unilateral tolerance
- Which of the following is incorrect about tolerances?
  a) Too loose tolerance results in less cost
  b) Tolerance is a compromise between accuracy and ability
  c) Too tight tolerance may result in excessive cost
  d) Fit between mating components is decided by functional requirements

Information Sheet-2	Determine sequence of operation

2.1. Operation sequence

- **To machine squaring stock**

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

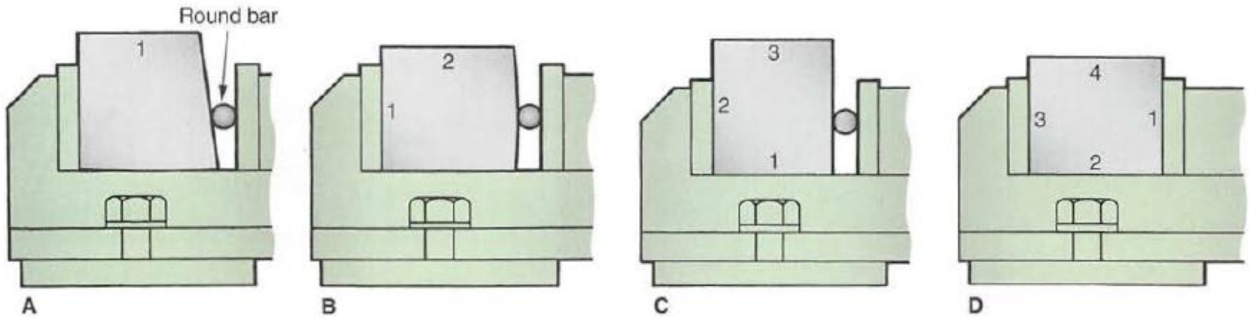


Figure 2.1. Sequence for squaring work on a milling machine. A- Square the top. B- Square one side. C- Square the bottom. D- Square the other side.

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.
6. 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.

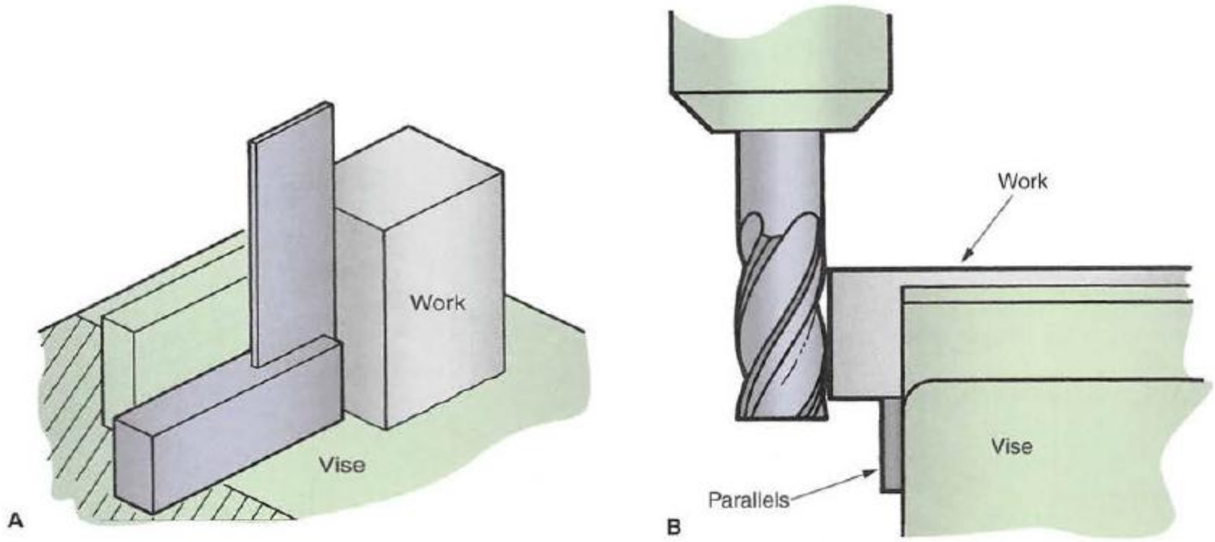


Figure 2.2. Squaring ends. A- Using a square to position short pieces for machining ends. For clarity, the movable jaw of the vice is not shown. B-Another technique for squaring the ends of work. The jaw must be checked with a dial indicator to be sure it is at a right angle to the column.

- **To machine an angular surface**
  1. Lay out the angular surface.
  2. Clean the vise.
  3. Align the vise with the direction of the feed. This is of the utmost importance.
  4. Mount the work on parallels in the vise.
  5. Swivel the vertical head to the required angle (Figure below).

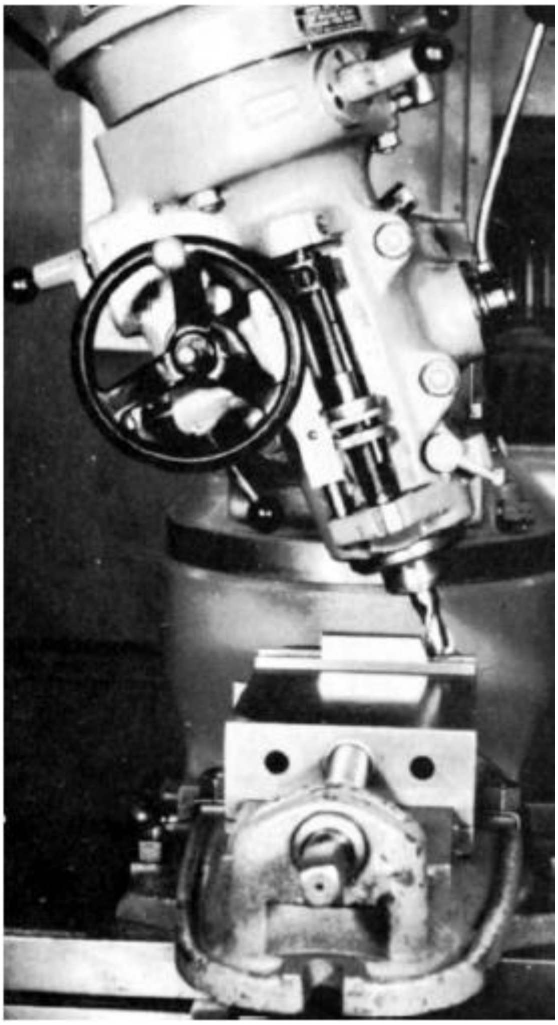


Figure 2.3. Head swiveled to machine an angle

6. Tighten the quill clamp.
7. 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.
8. Take a trial cut for about .50 in. (13 mm).
9. Check the angle with a protractor.
10. If the angle is correct, continue the cut.

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.

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

- ✓ **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 workpiece.

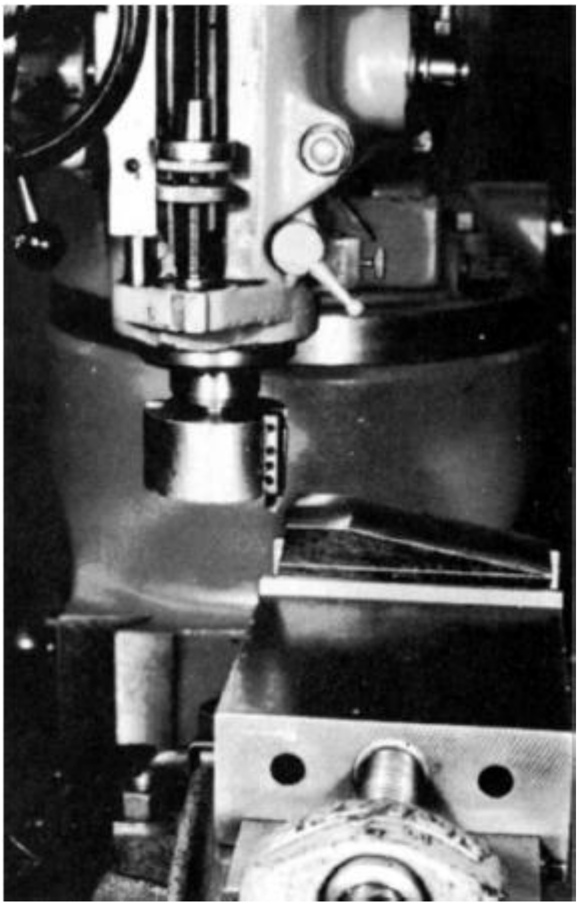


Figure 2.4. Machining an angle by adjusting the workpiece

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.

- **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.

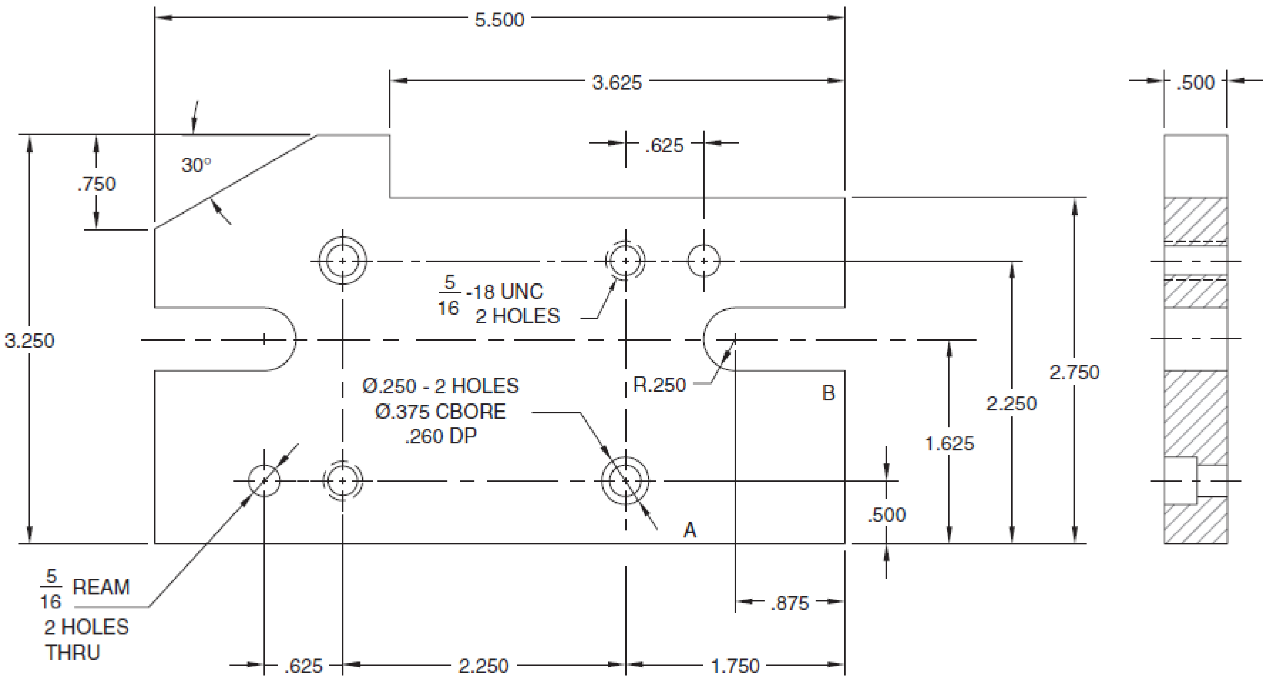


Figure 2.5. A typical flat part that must be laid out and machined

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 workpiece 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 90^0) 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).

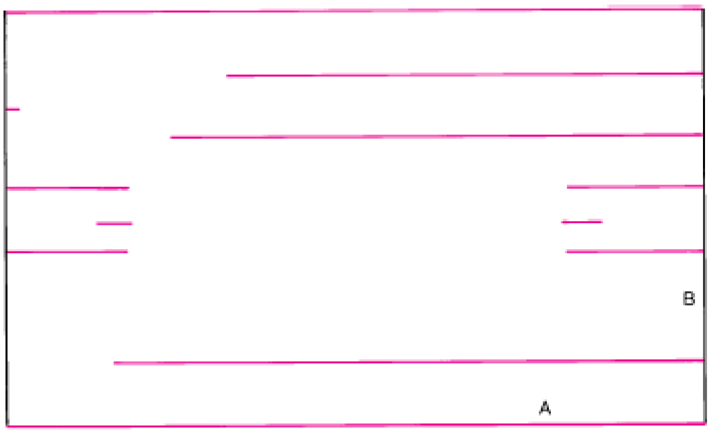


Figure 2.6. Lay out all horizontal lines using edge A as the reference surface.

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 2.7. 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.

Information Sheet-3	Selecting Cutting tools

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.

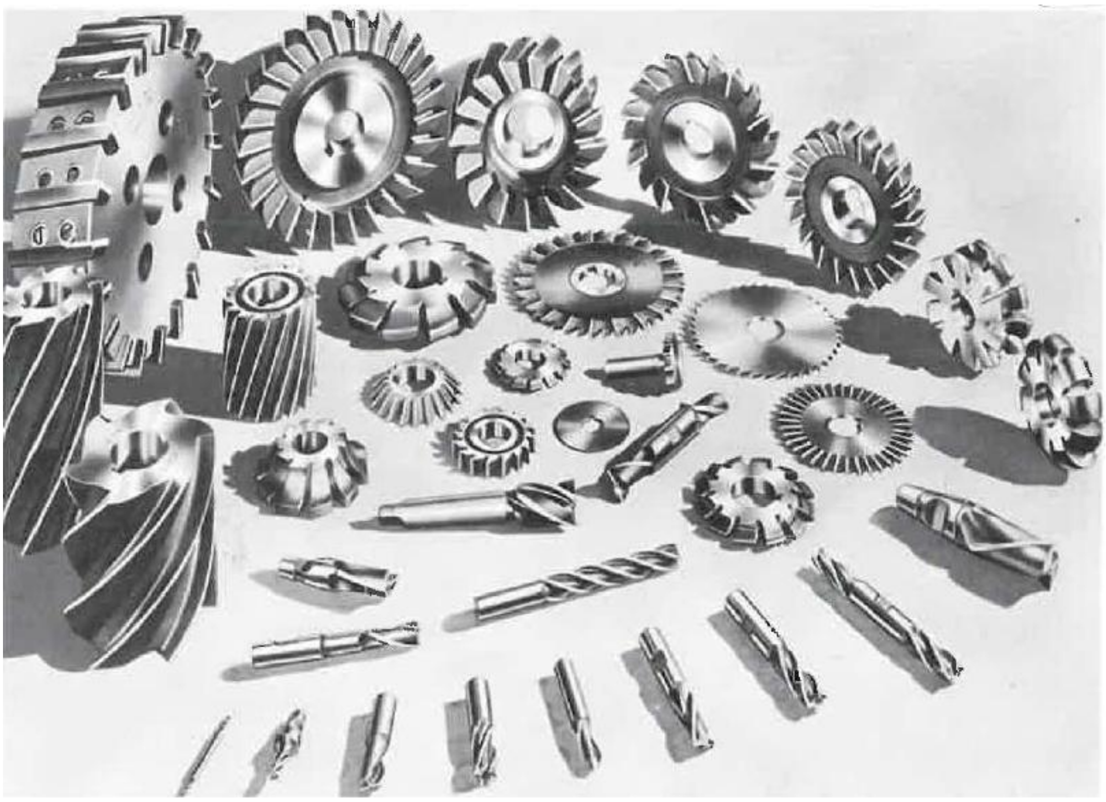


Figure 3.1. A selection of HSS (high speed steel) milling cutters

3.1. Types of milling cutters

There are two general types of milling cutters:

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

Figure 3.2. With the exception of the cutter in the back row, these are solid milling cutters.

- 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 3.3. A variety of cutters with indexable carbide inserts. When the cutting edges dull, new cutting edges are rotated into position. The inserts are available in different grades depending upon the material being machined.

3.2. How milling cutters are classified

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

- **Arbor cutters** have a suitable hole for mounting to an arbor.

Figure 3.4.An arbor-type milling cutter

- **Shank cutters** are fitted with either a straight or taper shank that is an integral part of the cutter. They are held in the machine by collets or special sleeves.

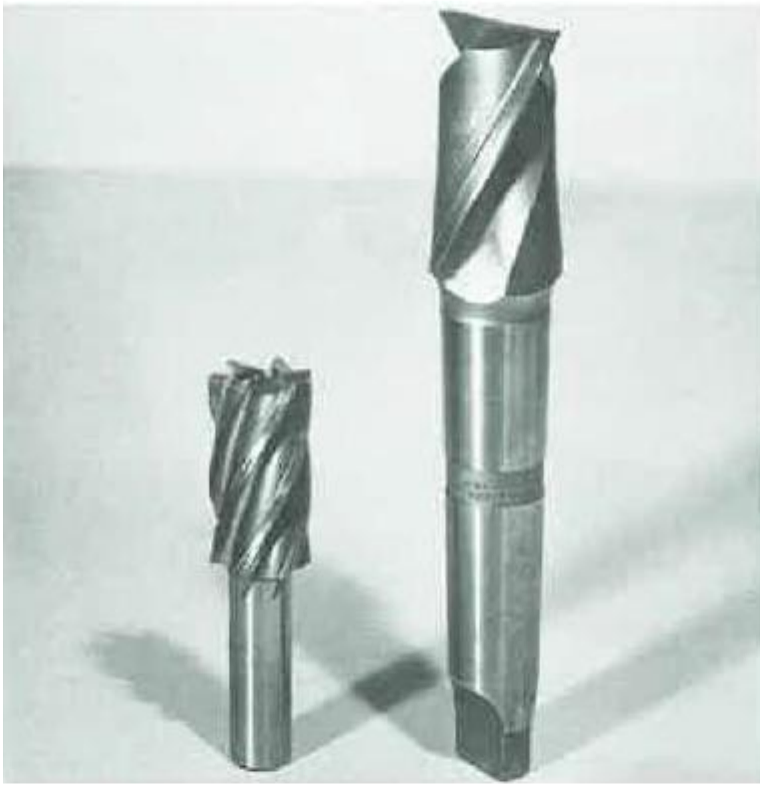


Figure 3.5. Shank type milling cutters.

- **Facing cutters** can be mounted directly to a machine's spindle nose or on a stub arbor.

Figure 3.6. This face-type milling cutter has an unusual design, making use of bearing-mounted inserts that rotate as they cut. The manufacturer claims better heat dissipation provided by the rotating inserts will increase cutter life and permit higher cutting speeds.

3.3. Milling cutter material

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

- **High abrasion resistance**. The cutting edges should not wear away rapidly due to the abrasive nature of some materials.
- **Red hardness**. The cutting edges should not be affected by the terrific heat generated by many machining operations.
- **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. They are produced by powder metallurgy techniques and have qualities that make them suitable for metal cutting tools. Cemented carbides can, in general, be operated at speeds 3 to 10 times faster than conventional HSS cutting tools. Cemented carbide cutters are excellent for long production runs and for milling materials with a scale-like surface, such as cast iron, cast steel, or bronze.

In most cases, only the cutting tips {not the entire cutter) are made of cemented carbides. They are brazed or clamped to the cutter body. Most inserted-tooth cutters use indexable inserts. Each insert has several cutting edges. When an edge becomes dull, the insert is indexed (turned) so that a new cutting edge contacts the metal.

Figure 3.7. These inserted-tooth cutters have teeth that are clamped to the cutter body. The cutter teeth (gold in this photo) can be indexed four times to present a fresh cutting edge as they wear.

3.4. 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.

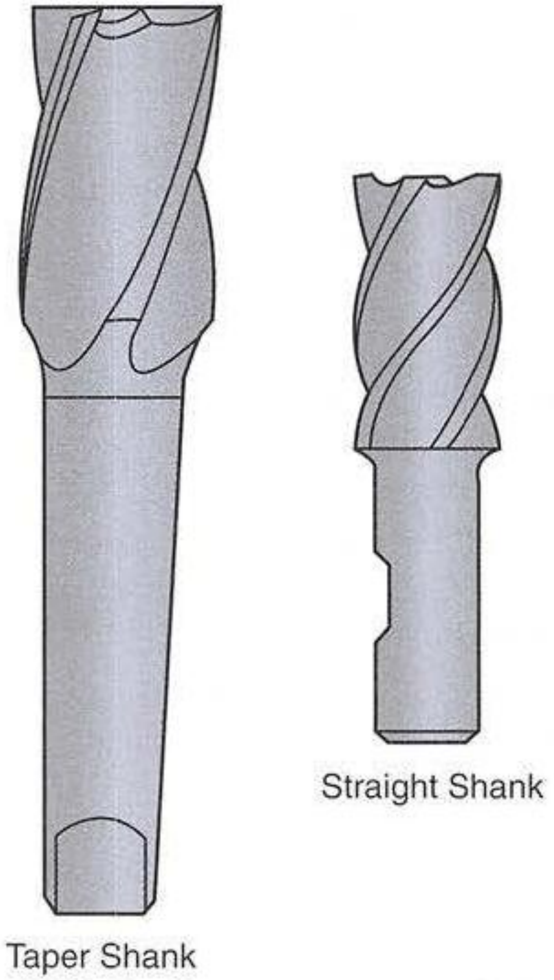


Figure 3.8. End mills with multiple indexable inserts are available with either helical or straight flutes. Two face cutters are also shown. Straight shank and taper shank end mills

Figure 3.9. Straight shank and taper shank 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 rutting edges.

Figure 3.10. An example of a face mill with indexable 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. Fly-cutters may also be used for boring operations. Care must be exercised when using a fly-cutter. When the tool is revolving, the inserted cutting tool becomes almost invisible and can injure the operator.

Figure 3.11. The fly cutter is a single-point cutting tool used for face milling.

- **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.

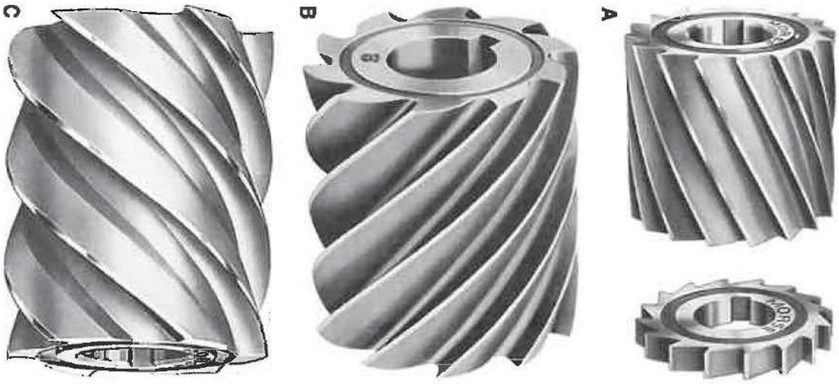


Figure 3.12. Plain milling cutters. A- light-duty cutters. B-Heavy- duty plain milling cutter. C- Helical/ plain milling culler.

- ✓ **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.

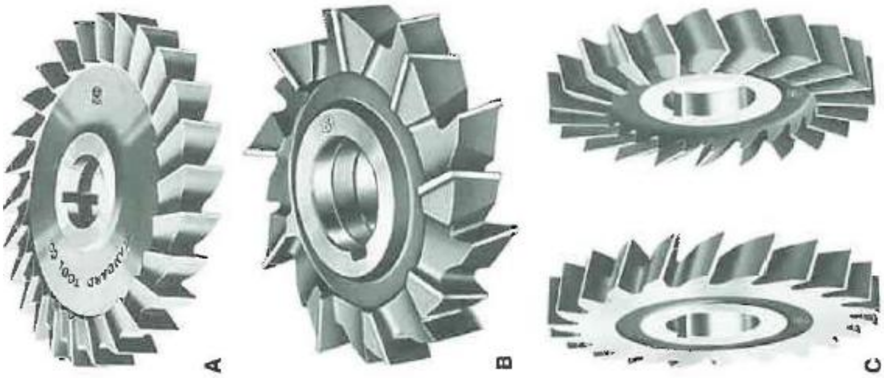


Figure 3.13. Side milling cutters. A-Plain side milling cutter. B- Staggered-tooth side milling cutter. C-Half side milling cutters.

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

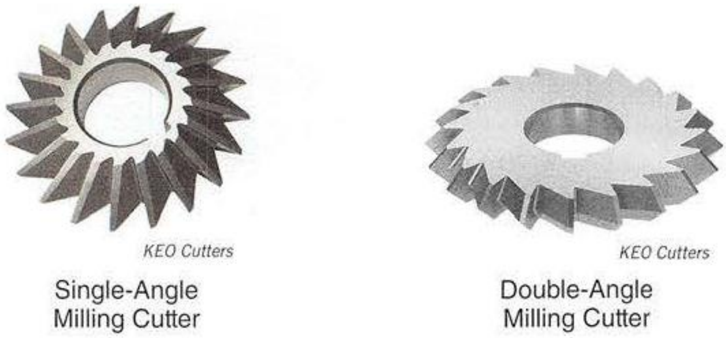


Figure 3.14. Angle cutters.

- ✓ **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).

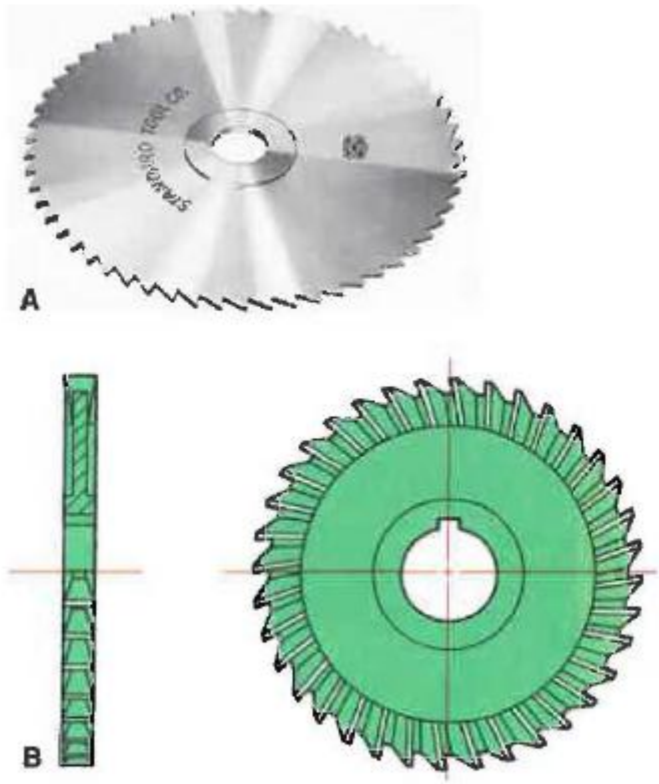


Figure 3.15. Slitting saws. A-Plain metal slitting saw is used for slotting and cutoff operations. B-The construction of the side chip clearance slitting saw allows it to be used for deep slotting applications.

- ✓ **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.

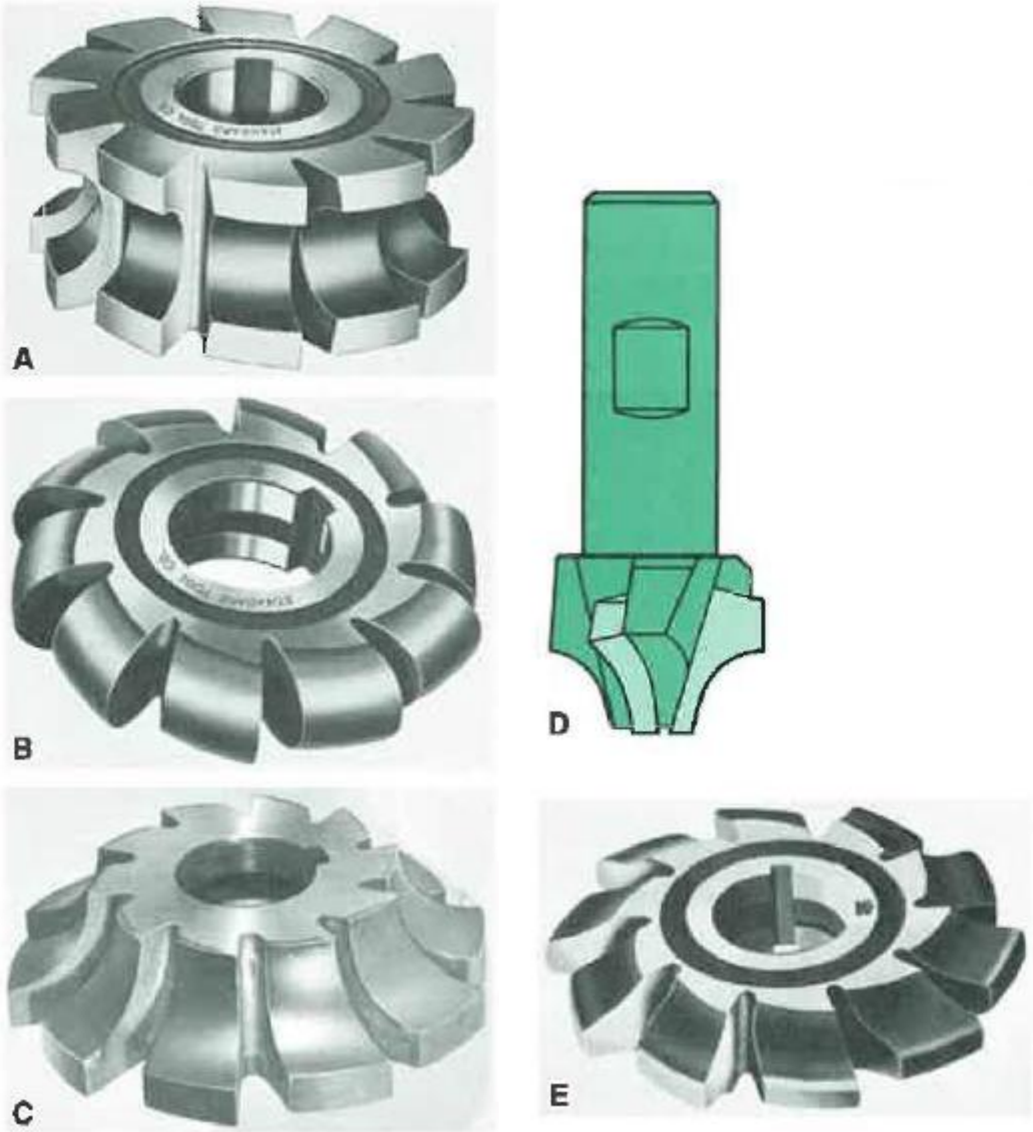


Figure 3.14. Formed milling cutters. A- Concave milling cutter will produce an accurate shape along cut. B-Convex milling cutter will produce a curved slot. C-Comer rounding milling cutter is available as left and right-hand cut. D-Corner-rounding end mill with replaceable tungsten carbide cutting edges. E-Gear cutter

- **Miscellaneous milling cutters**

Included in this category are cutters that do not fit into any of the previously mentioned groups.

- The **T-slot milling cutter** has cutting edges for milling the bottoms of T-slots after cutting with an end mill or side cutter.

Figure 3.15. AT-slot milling cutter

- A **Woodruff key seat cutter** is used to mill the semicircular keyseat for a Woodruff key

Figure 3.16. A Woodruff keyseat cutter

- A **dovetail cutter** mills doveta.il-type ways and it is used in much the same manner as the T-slot cutter.

Figure 3.17. A dovetail cutter

3.5. Selection of Milling Cutters

Consider the following when choosing milling cutters:

- High-speed steel, stellite, 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 workpiece is held in a swivel vise, or with an end milling cutter while the workpiece 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.
- When milling stock to length, the choice of using a pair of side milling cutters to straddle the work-piece, a single-side milling cutter, or an end milling cutter will depend upon the number of pieces to be cut.
- Some operations can be done with more than one type of cutter such as in milling the square end on a shaft or reamer shank. In this case, one or two side milling cutters, a fly cutter, or an end milling cutter may be used. However, for the majority of operations, cutters are specially designed and named for the operation they are to accomplish.
- The milling cutter should be small enough in diameter so that the pressure of the cut will not cause the work-piece to be sprung or displaced while being milled.

3.6. Care of Milling Cutters

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.
- Clean cutters before storing t hem. If they are to be
- Stored for any length of time, it is best to give them a light protective coating of oil.
- Never hammer a cutter onto an arbor. Examine the arbor for nicks or burrs if the cutter does not slip onto it easily. Do not forget to key the cutter to the arbor.
- Place a wooden board under an end mill when removing it from a vertical milling machine. This will prevent cutter damage if it is dropped accidentally. Protect your hand with a heavy cloth or gloves.

Self-Check - 3	Written Test

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

Match each term with the correct sentence below.

A

1. Can be fed into work like a drill.
2. Recommended for conventional milling when plunge cutting (going into work like a drill) is not required
3. Mounts on a stub arbor
4. Intended for machining large flat surfaces parallel to the cutter face
5. A facing mill with a single- point cutting tool
6. Cylindrical cutter with teeth located around the circumference.
7. Cutter with helical teeth designed to cut with a shearing action.
8. Has cutting teeth on the circumference and on one or both sides.
9. Has alternate right-hand and left-hand helical teeth.
10. Thin milling cutter designed for machining narrow slots and for cutoff operations.

B

A. Side milling cutter
B. Slab cutter
C. Fly cutter
D. Two-flute end mill
E. Plain milling cutter
F. Multiflute end mill
G. Face milling cutter
H. Metal slitting saw
I. Shell end mill
J. Staggered-tooth side cutter

Operation Sheet 1	Selecting Cutting tools

To select and mount a milling cutter

Tools required:

Horizontal milling machine, arbor, wrench, collar and arbor nut

Procedure

1. Select appropriate cutting tool considering the material to machined
2. Remove the arbor nut and collar, and place them on a piece of masonite.
3. Clean all spacing collar surfaces of cuttings and burrs.
4. Set the machine to the slowest spindle speed.
5. Check the direction of the arbor rotation by starting and stopping the machine spindle.
6. Slide the spacing collars on the arbor to the position desired for the cutter
7. Fit a key into the arbor keyway at the position where the cutter is to be located.
8. Hold the cutter with a cloth and mount it on the arbor. Make sure that the cutter teeth point in the direction of the arbor rotation.
9. Slide the arbor support in place and be sure that it is on a bearing bushing on the arbor.
10. Put on additional spacers, leaving room for the arbor nut. TIGHTEN THE NUT BY HAND.
11. Lock the arbor support in position.
12. Tighten the arbor nut firmly with a wrench, using only hand pressure.
13. Lubricate the bearing collar in the arbor support.
14. Make sure that the arbor and arbor support will clear the work

LAP Test	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. Select appropriate milling cutter and mount a milling machine properly

Instruction Sheet	Learning Guide #-29

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

- Setup Work piece
- Performing setup operations
- Ensuring Setup safe operation

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, upon completion of this Learning Guide, **you will be able to –**

- Setup work piece to required level of accuracy using appropriate instruments/equipment and in accordance with standard procedures.
- Perform setup operations applying safety procedures and using personal protective devices based on OHS.
- Ensure setup to be safe with compliance to work operation.

Learning Instructions:

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below.
3. Read the information written in the "Information Sheets". Try to understand what are being discussed. Ask your trainer for assistance if you have hard time understanding them.
4. Accomplish the "Self-checks" which are placed following all information sheets.
5. Ask from your trainer the key to correction (key answers) or you can request your trainer to correct your work. (You are to get the key answer only after you finished answering the Self-checks).
6. If you earned a satisfactory evaluation proceed to "Operation sheets
7. Perform "the Learning activity performance test" which is placed following "Operation sheets" ,
8. If your performance is satisfactory proceed to the next learning guide,
9. If your performance is unsatisfactory, see your trainer for further instructions or go back to "Operation sheets".

Information Sheet-1	Set-up work piece

1.1. Classifying Metals

- **Introduction**

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.

- ✓ **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. The ore is heated to high temperatures and carbon is added to remove the oxygen and produce iron in metal form.
- ✓ **Nonferrous metals** are metals that contain no iron. Aluminum, copper, magnesium, and titanium are examples of nonferrous metals.

1.2. Setting up the mill

The machine table and all sliding surfaces should be cleaned prior to setup or operation. Any nicks or burrs found on the table or work-holding devices should be removed with a honing stone.

- **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 workpiece slippage from cutting pressure.

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

Many mill vises are equipped with keys that fit snugly into the table T-slots. Once the keys are engaged, the vise is accurately aligned. Before mounting a vise or any other mill fixture, inspect the bottom for small chips, nicks, and burrs. Use a honing stone to remove these and make sure that the tooling and table are clean. Set the vise gently on the table and position it according to the job to be done. Install the hold-down bolts and tighten just enough to permit the vise to be moved by gentle tapping with a lead or other soft hammer. If the mill vise does not have alignment keys, the following procedures may be used to align it parallel or perpendicular to the machine table.

Parallel-to-Table Alignment

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

Figure 1.1. Aligning vise parallel to table

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 table so that the vise jaw moves past the indicator tip, 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 1.2. 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 1.3. Using a square to align a vise on the milling machine table.

Angular Alignment A protractor may be used to set a vise at an angle other than 90 degrees to the table. The accuracy of the angle is dependent on the type of tool used and the technique. Errors can be introduced from the angle setting on the protractor and the relative alignment of the vise jaw along the protractor blade. Considerable care must be exercised in making a setup by this technique.

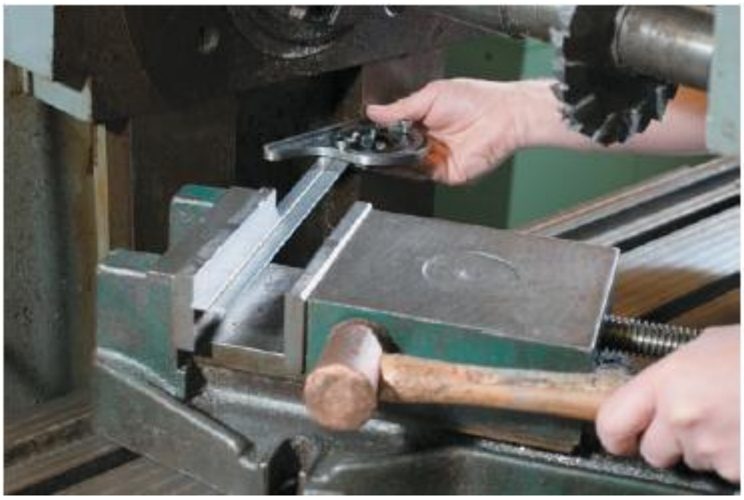


Figure 1.4. Using a protractor to align a vise on the milling machine table.

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

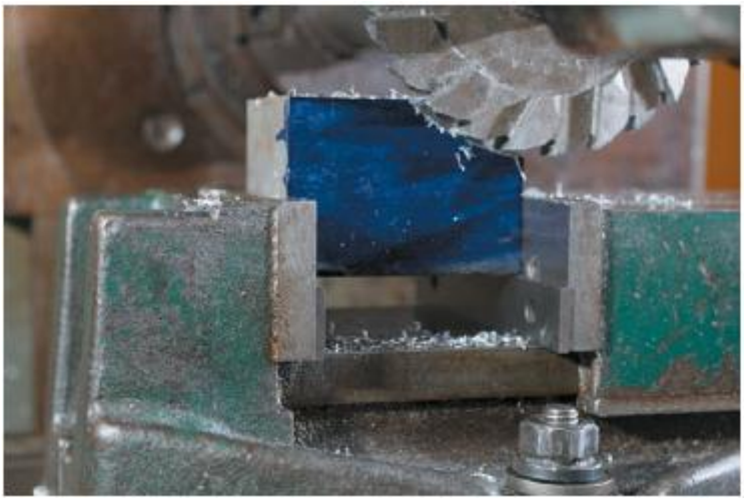


Figure 1.5. Cutting pressure against solid jaw

Avoid applying cutting pressure against the movable jaw. If the workpiece 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 workpiece holds it in place.

The more contact area there is, the better the holding power. When cutting pressure is applied parallel to the vise jaws, there is always a possibility that the part will be pushed from the vise. Therefore, if you must use parallels to elevate the workpiece, raise it only high enough to accomplish the required machining, because this reduces contact area, and the vise may not be able to hold the workpiece safely and securely.

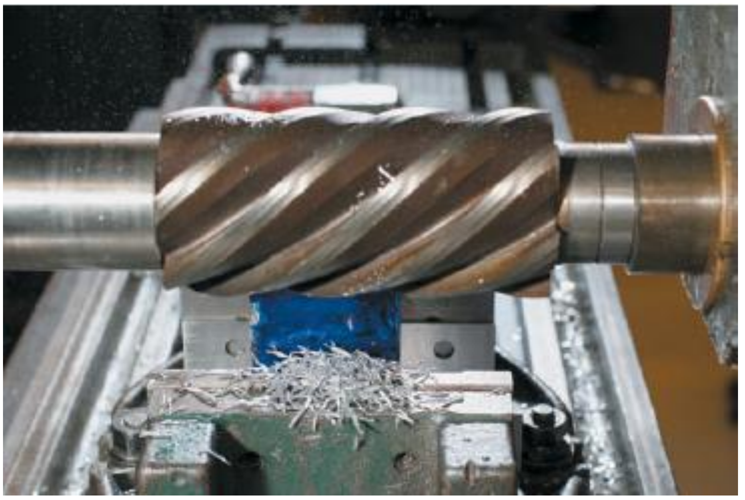


Figure 1.6. Workpiece on parallels held in a vise

Self-Check -1	Written Test

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

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. Mill vise is not an accurate and dependable workholding tool.
4. Dial indicator can be used in setup milling machine

Information Sheet-2	Performing setup operations

The success of any milling operation depends on, before setting up a job, be sure that to a great extent, upon judgment in setting up the job, workpiece, the table, the taper in the spindle, selecting the proper milling cutter, and holding the cutter by the best means under the circumstances. Some fundamental practices have been proved by experience to be necessary for and the arbor or cutter shank are all clean and good results on all jobs. Some of these practices are mentioned below:

- ✓ Before setting up a job, be sure that the workpiece, table, the taper in the spindle, and the arbor or cutter shank are free from chips, nicks, or burrs.
- ✓ Do not select a milling cutter of larger diameter than is necessary.
- ✓ Check the machine to see if it is in good running order and properly lubricated, and that it moves freely, but not too freely in all directions.
- ✓ Consider direction of rotation. Many cutters can be reversed on the arbor, so be sure you know whether the spindle is to rotate clockwise or counterclockwise.
- ✓ Feed the workpiece in a direction opposite the rotation of the milling cutter (conventional milling).
- ✓ Do not change feeds or speeds while the milling machine is in operation.
- ✓ When using clamps to secure a workpiece, be sure that they are tight and that the piece is held so it will not spring or vibrate under cut.
- ✓ Use a recommended cutting oil liberally.
- ✓ Use good judgment and common sense in planning every job, and profit from previous mistakes.
- ✓ Set up every job as close to the milling machine spindle as circumstances will permit.

Self-Check -2	Written Test

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

Say true or false

1. Select a milling cutter of larger diameter than is necessary
2. It is recommended that you can change feeds or speeds while the milling machine is in operation
3. Before setting up a job, be sure that the workpiece, table, the taper in the spindle, and the arbor or cutter shank are free from chips, nicks, or burrs

Information Sheet - 3	Ensuring Setup safe operation

Pre-operational safety checks

Be for operating milling machine check the following to ensure safety operation:

- Ensure no slip/trip hazards are present in workspaces and walkways.
- Locate and ensure you are familiar with the operation of the ON/OFF starter and E-Stop (if fitted).
- Ensure there is no equipment on top of the machine.
- Check that machine guards are in position.
- Ensure cutter is in good condition and securely mounted.
- Check coolant delivery system to allow for sufficient flow of coolant.
- Faulty equipment must not be used. Immediately report suspect machinery.
- Identify the locations for all controls and the machine E-stop
- Ensure that work piece is securely supported and clamped, and that the machine table is free from any other non-secured items
- Disengage any machine feeds
- Ensure that the cutting tool is securely tightened and spindle wrench/key is removed
- Ensure that the spindle brake is disengaged
- Select the proper speed range and motor rotation for the required operation

Self-Check -3	Written Test

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

Say true or false

1. It has no difference if machine guards are in position or not.
2. Be for operating milling machine check no slip/trip hazards are present in workspaces and walkways.
3. Make sure that work piece is not securely supported and clamped, and that the machine table is free from any other non-secured items

Operation Sheet 1	Set-up work piece

Milling Machine Setups
Tools required:
Milling machine, masonite, 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 masonite, 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.
5. When tightening the arbor nut, take care to make it only hand tight with a wrench.
6. 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.

Operation Sheet 2	Set-up work piece

To align the vise parallel to the table travel
Tools required:
Milling machine, machine vise, dial indicator with magnetic stand, rag or brush
Procedure

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.

.

LAP Test	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. Setups Milling Machine

Task 2. Align the vise parallel to the table travel

Instruction Sheet	Learning Guide #-30

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

- Calculating speeds and feeds
- Using milling machine accessories
- Performing milling operations produce component including gears
- Performing milling operations

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, upon completion of this Learning Guide, **you will be able to –**

- Calculate speeds and feeds using appropriate mathematical techniques and reference material based on standards.
- Make milling machine accessories used appropriate to the requirements of the operational standards.
- Perform milling operations to produce component inclusive gears to specifications.
- Perform milling operations applying knowledge on safety procedures and using personal protective devices based on OHS

Learning Instructions:

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below.
3. Read the information written in the "Information Sheets". Try to understand what are being discussed. Ask your trainer for assistance if you have hard time understanding them.
4. Accomplish the "Self-checks" which are placed following all information sheets.
5. Ask from your trainer the key to correction (key answers) or you can request your trainer to correct your work. (You are to get the key answer only after you finished answering the Self-checks).
6. If you earned a satisfactory evaluation proceed to "Operation sheets
7. Perform "the Learning activity performance test" which is placed following "Operation sheets" ,
8. If your performance is satisfactory proceed to the next learning guide,
9. If your performance is unsatisfactory, see your trainer for further instructions or go back to "Operation sheets".

Information Sheet-1	Performing milling operations Safety

1.1. Safety procedures

Milling machines, like all machine tools, should be cleaned after each work session. A brush should be used to remove accumulated chips. Chips are razor-sharp. Never use your hand to remove them. Never remove chips with compressed air. The flying chips may injure you or a nearby person. Also, if cutting oil was used in the machining operation, the compressed air will create a highly flammable oily mist. If ignited by an open flame, the mist can explode. Finish by wiping down the machine with a soft cloth.

The following procedures are suggested for the safe operation of a milling machine.

- Become thoroughly familiar with the milling machine before attempting to operate it. When in doubt, obtain additional instructions.
- Never attempt to operate a milling machine while your senses are impaired by medication or other substances.
- Wear appropriate clothing and approved safety glasses.
- Stop the machine before making adjustments or measurements, or trying to remove accumulated chips.
- Be sure all power to the machine is turned off before opening or removing guards and covers.
- Use a piece of heavy cloth or gloves for protection when handling milling cutters. Avoid using your bare hands.
- Get help to move any heavy machine attachment, such as a vise, dividing head, rotary table, or large work.
- Be sure the work-holding device is mounted solidly to the table and the work is held firmly. Spring or vibration in the work can cause thin cutters to jam and shatter.
- Before engaging a rotating tool against the workpiece, take the time to verify that the spindle is rotating in the correct direction.
- Never reach over or near a rotating cutter.
- Avoid talking with anyone while operating a machine tool. Do not allow anyone to turn on the machine for you.
- Never "fool around" when operating a milling machine. Keep your mind on the job and be ready for any emergency.
- Keep the floor around the machine clear of chips, and wipe up spilled cutting fluid immediately. Place sawdust or oil-absorbing compound on slippery floors. Place all oily rags in an approved metal container that can be closed tightly.
- Be thoroughly familiar with the location of the machine's emergency stop switch, lever, or button.
- Treat any small cuts and skin punctures as potential infections. Clean them thoroughly. Apply antiseptic and cover the injury with a bandage. Report any injury, no matter how minor, to your instructor or supervisor.
- Launder work clothes frequently. Greasy clothing is a fire hazard.

1.2. Using personal protective devices.

PPE, or personal protective equipment, refers to safety equipment that is to be worn to protect a person from potential dangers.

- **Eye Protection:** Eye protection is the most common type of PPE used in the machining industry. Safety glasses with side shields must always be worn when entering a shop environment, even if just visiting or observing. Always wear safety glasses that meet the standard for protection set.

Figure 1.1. Safety glasses and goggles come in many different styles.

Choosing appropriate clothing to wear when working in a machining environment is important.

- **Clothing**: Wear long pants to protect legs from sharp or hot metal shavings. Shorts, skirts, or dresses are not recommended. Short-sleeved, close-fitting shirts are best. If wearing long sleeves, roll sleeves up past the elbows. Shirts or sweatshirts with hoods or drawstrings should be removed. If a shop coat or apron is worn, it should fit snugly and any ties or strings secured so they do not hang loosely.

- **Footwear**: Hard, flat-soled work shoes should always be worn because metal shavings, or chips, often fall onto the shop floor and can possibly cut through the soles of other shoes and cause serious cuts. Slip-resistant soles should also be worn because oils and other fluids may spill onto the floor and create slippery conditions. Leather work shoes are preferred, and sometimes safety-toe shoes or even those with metatarsal shields are required.

- **Gloves:** There are a few situations in machining where the use of gloves is acceptable. Canvas or leather gloves can be worn to protect hands from cuts when handling raw materials, saw blades, or large cutting tools.
- **Hard Hats:** Some machining environments deal with very large materials and parts. When working in an area where items are stored overhead or moved overhead by hoists or cranes, hard hat usage should be implemented.

Self-Check -1	Written Test

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

1. The first item of procedure to operate the milling machine is
   a) Secure work piece.
   b) Set cutting speed.
   c) Turn on power.
   d) Obtain teacher permission

2. Make measurements or set-ups only when
   a) You have permission.
   b) You have read and understand safety rules.
   c) The machine is at a dead stop.
   d) All of the above.
3. Remove spindle wrench immediately after securing bit in collet.
   a) True
   b) False
4. Milling machine cutters are sharp and must be handled carefully.
   a) True
   b) False
5. When machine is at a dead stop, chips should be removed with
   a) Hand.
   b) Cloth
   c) Brush
   d) None of the above.
6. Never clean chips away from cutter while machine is running.
   a) True
   b) False

Information Sheet-2	Calculating speeds and feeds

The time required to complete a milling operation and the quality of the finish is almost completely governed by the cutting speed and feed rate of the cutter.

Milling cutting speed refers to the distance, measured in feet or meters that a point (tooth) on the cutter's circumference moves in one minute. It is expressed in feet per minute (fpm) or meters per minute (mpm). Milling cutting speed depends on the revolutions per minute (rpm) of the cutter.

$$\text{Cutting Speed}(S) = \frac{\pi\ DN}{1000}\text{meter / minute}$$

Where: D = the diameter of the milling cutter in mm

N = Spindle speed in rpm

The cutting speed depends upon the material to be machined, the cutter material, depth of cut, feed, type of operation and the coolant used.

Example: Calculate the cutting speed to perform milling with a cutter of diameter 60mm and spindle speed of 250rpm.

Solution:

Given: Diameter of cutter (D) = 60 mm

Spindle speed (N) = 250 rpm

$$\text{Cutting Speed}(S) = \frac{\pi\ DN}{1000}\text{meter / minute}$$

$$\text{Cutting Speed}(S) = \frac{\pi\ \times 60 \times 250}{1000} = 47.12\text{meter / minute}$$

Milling feed is the rate at which work moves into the cutter. It is given in feed per tooth per revolution (ftr). Proper feed rate is probably the most difficult setting for a machinist to determine. In view of the many variables (width of cut, depth of cut, machine condition, cutter sharpness), feed should be as coarse as possible, consistent with the desired finish.

F = ftr x T x rpm

Where ftr = feed per tooth per revolution

T = cutter number of teeth

rpm = revolutions per minute

Self-Check -2	Written Test

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

1. Calculate the feed, in millimeters per minute, of an 80 mm diameter milling cutter with 24 teeth operating at a cutting speed of 35 meters per minute and cutting at a feed of 0.02 mm per tooth.
2. ____________is the rate at which the work moves into the cutter.
3. ____________refers to the distance, measured in feet or meters that a point (tooth) on the cutter's circumference moves in one minute.

Information Sheet-3	Using milling machine accessories

3.1. Types of Milling Machines

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 workpiece. 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).

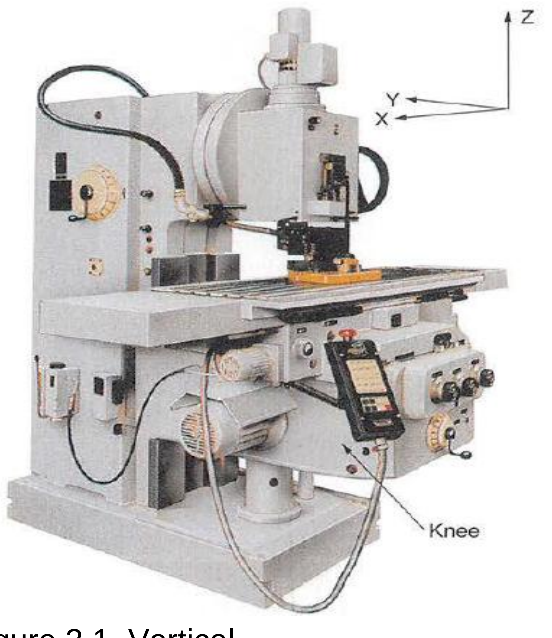


Figure 3.1. Vertical

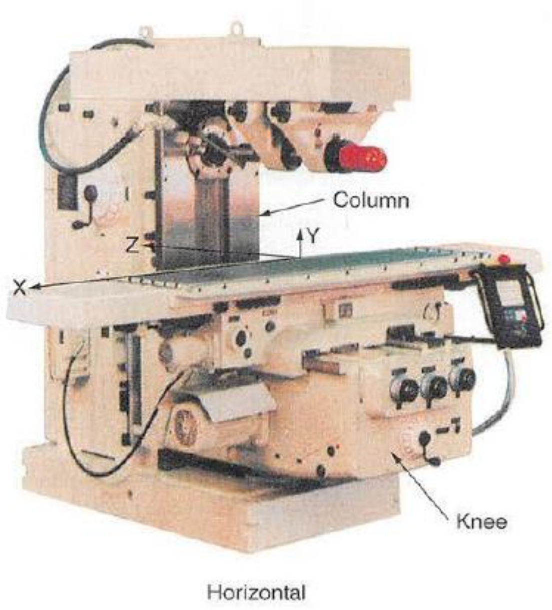


Figure 3.2. Horizontal

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).

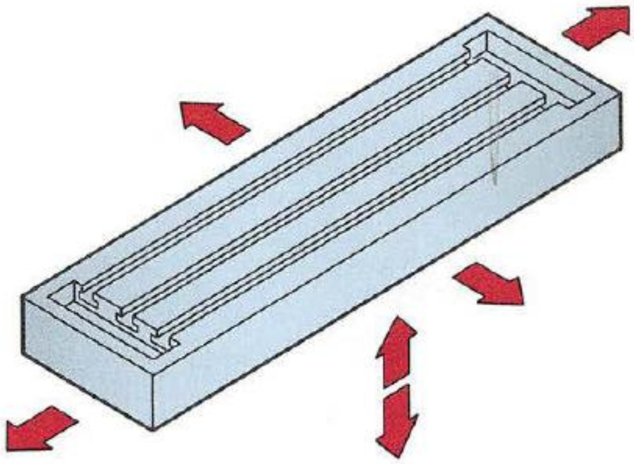


Figure3. 3. Plain mil ling horizontal machine   Figure 3.4.Table movements of a plain horizontal milling machine.

- ✓ **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 workpieces.

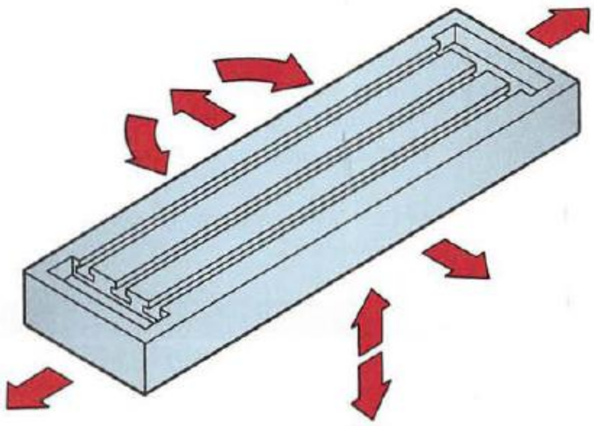



Figure 3.5. On a universal horizontal milling

machine, the table can be swiveled 45° or more. milling machine

Figure 3.6. Table movements possible on a universal

.

- ✓ **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.

Figure 3.8. A typical swivel-head milling machine

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.

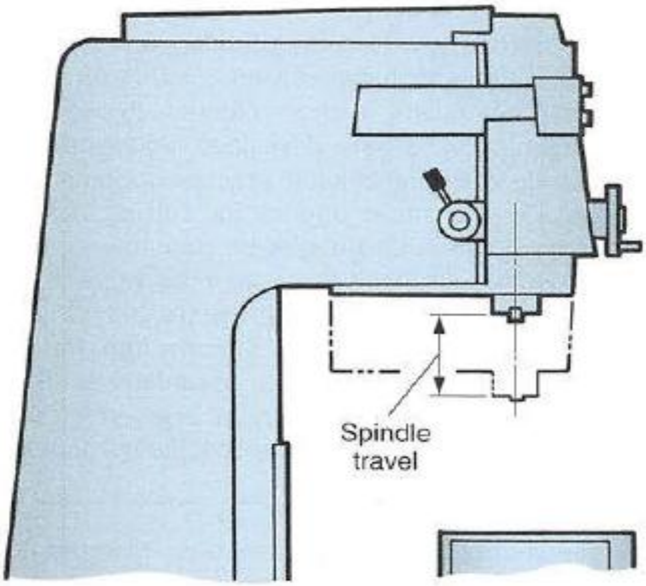


Figure 3.9. The spindle head is fixed in a vertical position on a sliding-head milling machine. The entire head is moved to make cutting adjustments.

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.

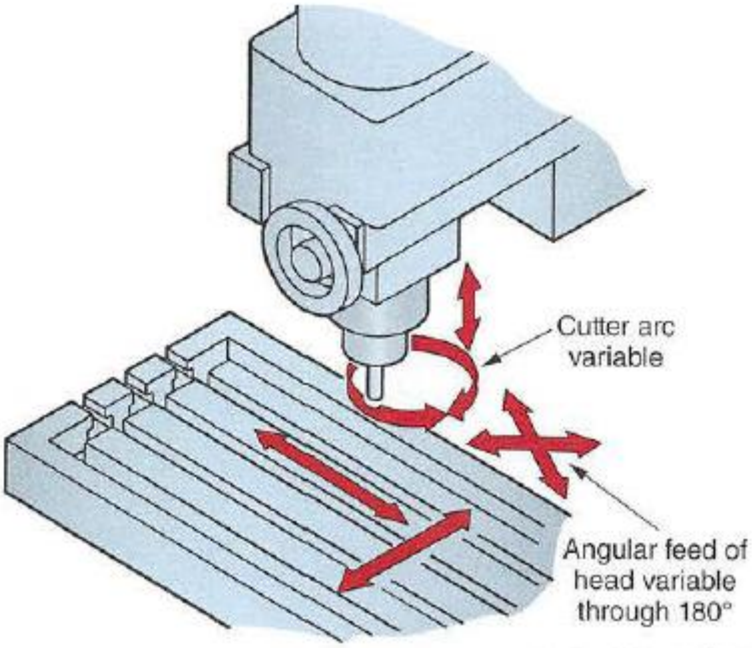


Figure 3.10. Spindle movements possible on a rotary-head vertical mil ling machine. Mil ling machines with CNC capabilities are replacing this type of machine.

3.2. 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.

- **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. Therefore, every time a workpiece is clamped it must be properly aligned and located. A trick is to use two dowel pins that fit snugly into the table T-slots as a “backstop.” A straight edge of the workpiece can be slid against these two pins and the part will be quickly aligned parallel to the machine table.

Clamps are available in many different variations. One type of clamping system is the step block clamp style, which allows a stud or bolt to be anchored to the machine table T-slot and a strap to be drawn down onto a part’s surface to secure it to the table. Step clamps are generally purchased in a set that consists of the following: T-nuts, studs of various lengths, clamps of various lengths, riser blocks of various heights, and necessary nuts and washers.

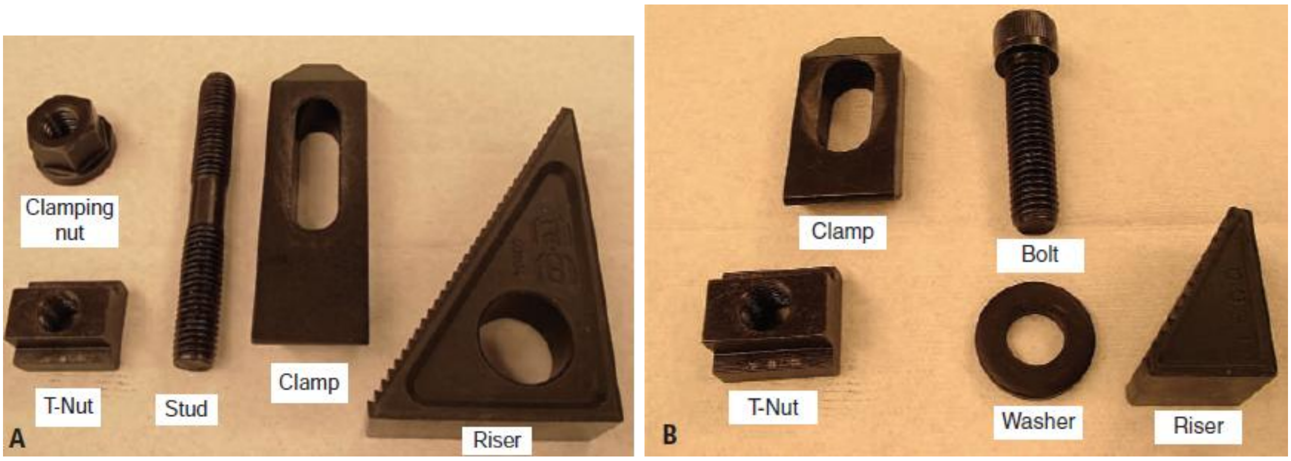


Figure 3.11. (A) A step clamp assembly usually contains a T-nut, stud, clamp, riser block, and clamping nut. (B) A bolt and washer may be used in place of the stud and nut

Figure 3.12. 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.

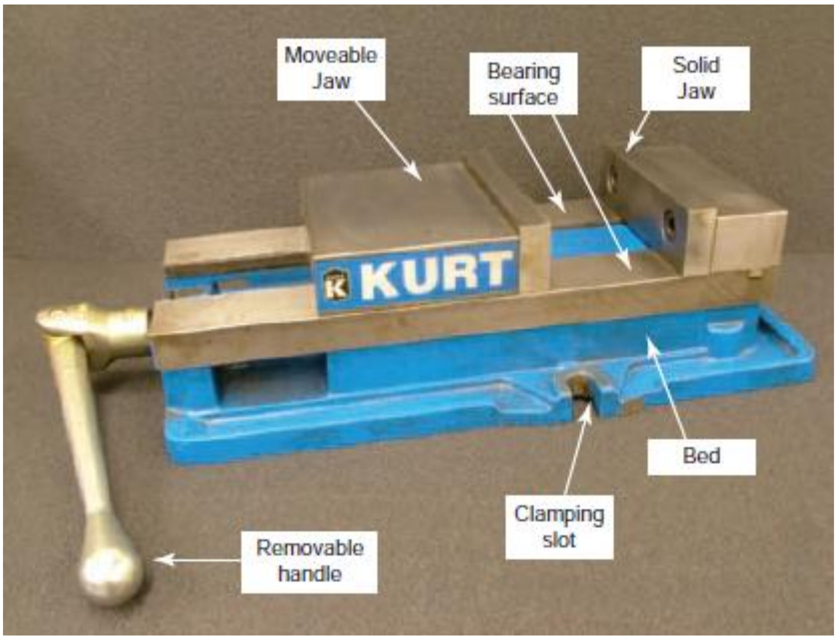


Figure 3.13. A typical milling machine vise and its parts

- **Specialty Vises**: Special vises are available for angular workholding. 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 3.14. 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.

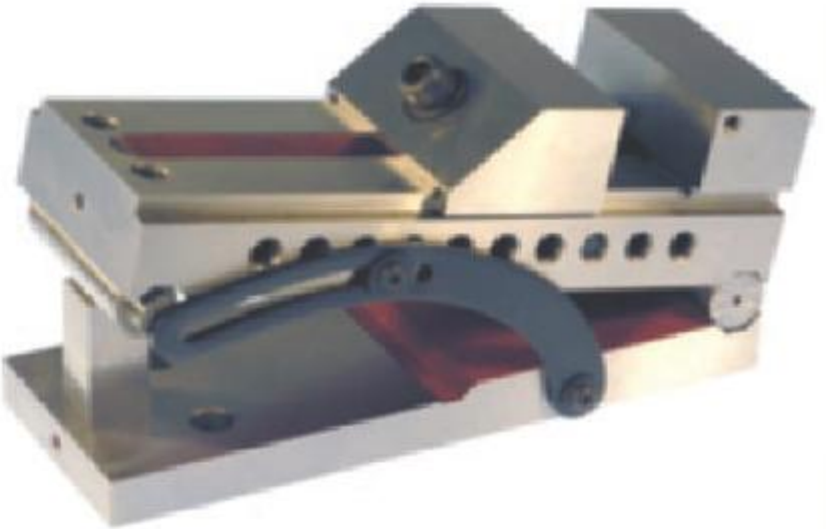


Figure 3.15. The sine vise can be set using gage blocks for very precise angular positioning

- **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 workpieces. 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 3.16. A three-jaw chuck fixture for milling

Figure 3.17. Square- and hex-shaped collet blocks

- **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.

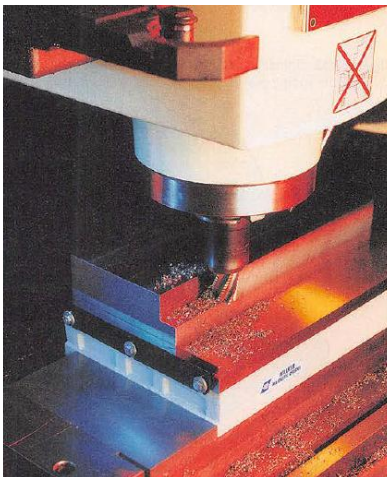


Figure 3.18. Work secured on a magnetic chuck for milling

- **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 handwheel. The table is graduated in degrees around its circumference. Adjustments can be made accurately with the handwheel 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.

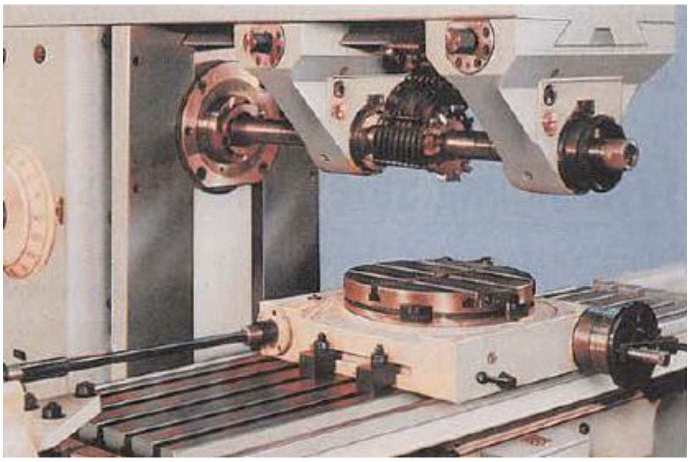


Figure 3.19. A rotary table

Figure 3.20. 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. The dividing head consists of two parts: the dividing unit and the footstock. Work may be mounted between centers, in a chuck, or in a collet. An index plate, identified by circles of holes on its face, and the index crank, which revolves on the index plate, are fundamental in the dividing operation.

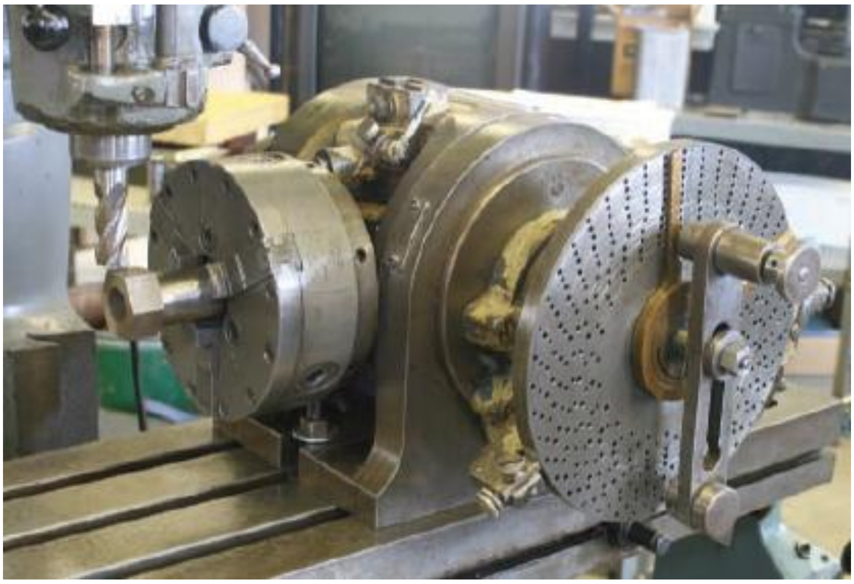


Figure 3.21. Dividing or indexing head

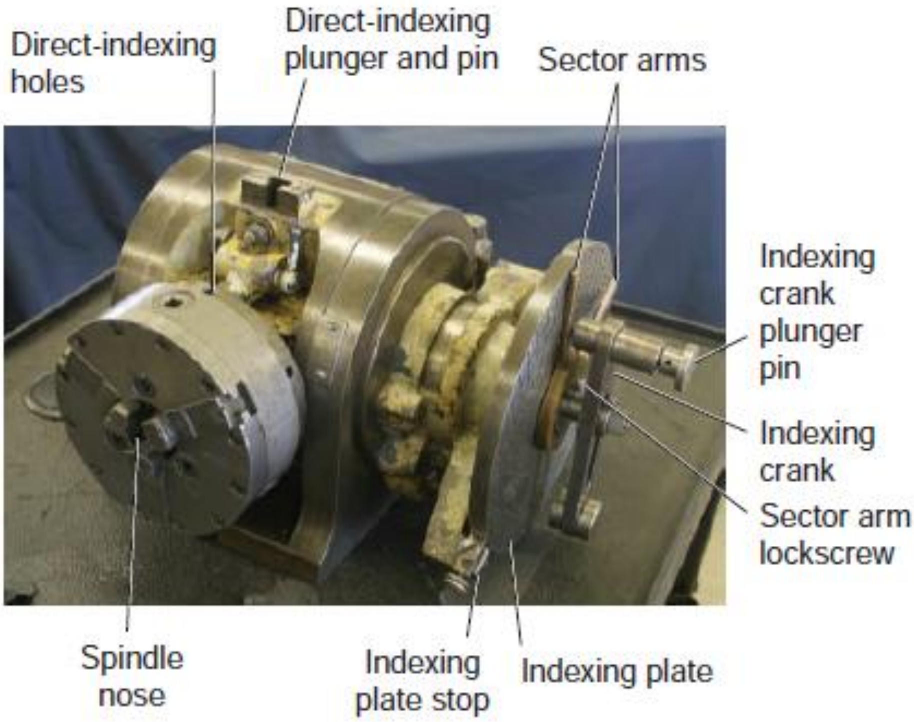


Figure 3.22. The parts of the indexing or dividing head. A three jaw chuck is mounted on the spindle of this model.

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

Self-Check -3	Written Test

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

Each term with the correct sentence below.

A

1. Can only be mounted parallel to or at right angles on worktable
2. Has a circular base graduated in degrees
3. Permits compound angles (angles on two planes) to be machined without complex or multiple setups.
4. Can only be used with ferrous metals.
5. Needed when cutting segments of circles, circular slots, and irregular-shaped slots.
6. Permits rapid positioning of circular work in 15° increments and can be locked at any angular setting.
7. Used to divide circumference of round work into equally spaced divisions.
8. Arms that are loosened and repositioned to divide a circumference equally.

B

A. Index fingers
B. Flanged vise
C. Rotary table
D. Magnetic chuck
E. Swivel vise
F. Index table
G. Dividing head
H. Universal vise

Information Sheet-4	Performing milling operations produce component including gears

4.1. 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.

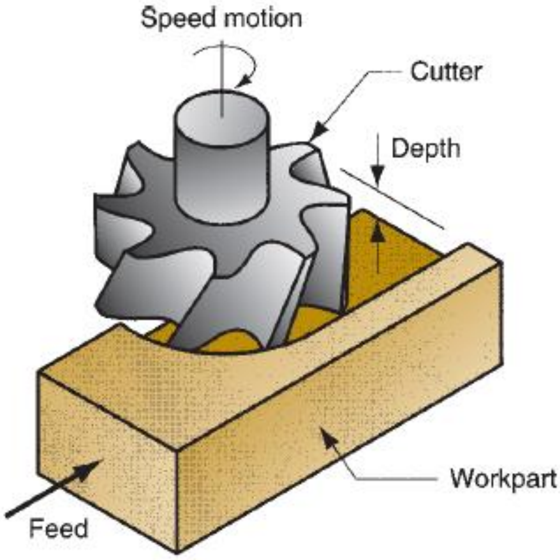


Figure 4.1. Face milling

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

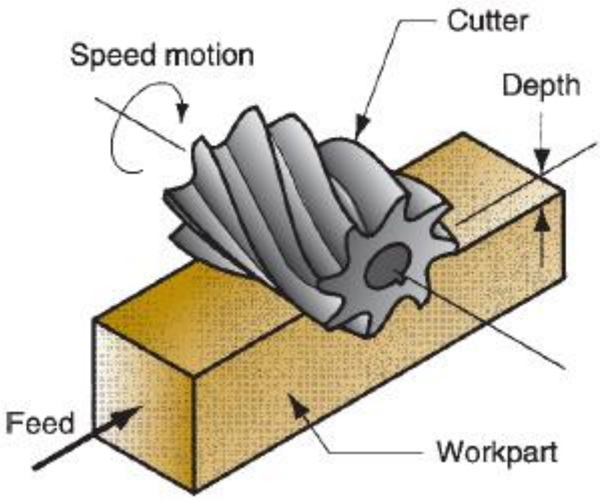


Figure 4.2. Peripheral or plain milling

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.

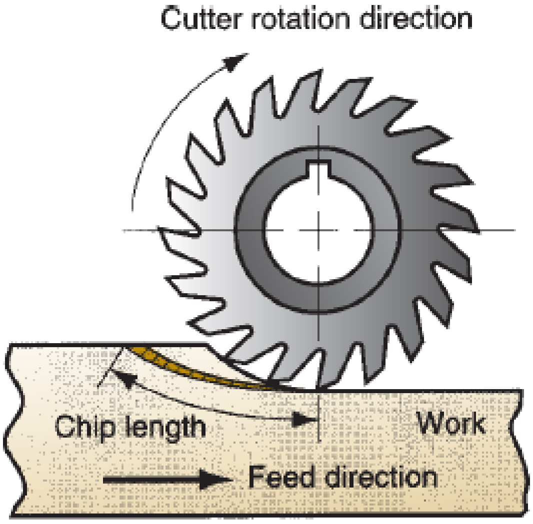


Figure 4.3.up milling

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 antibacklash 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.

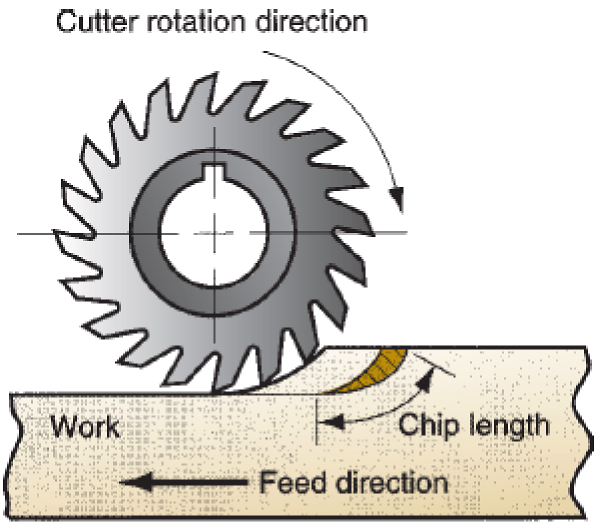


Figure 4.5.down milling

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:

- **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 workpiece to other end to complete the one pairs of plain milling operation.

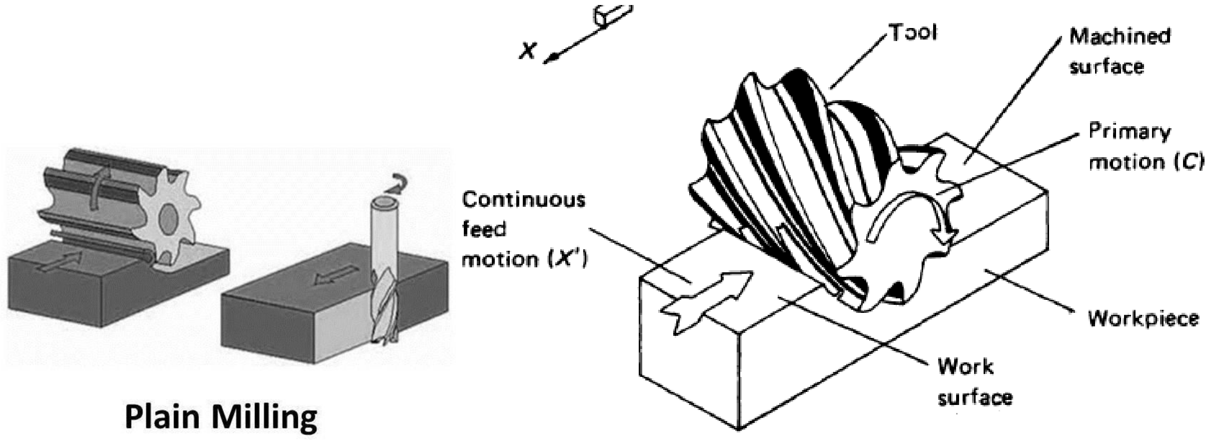


Figure 4.6. Plain milling or slab milling

Face Milling Operation

This operation produces flat surface at the face of the workpiece. 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.

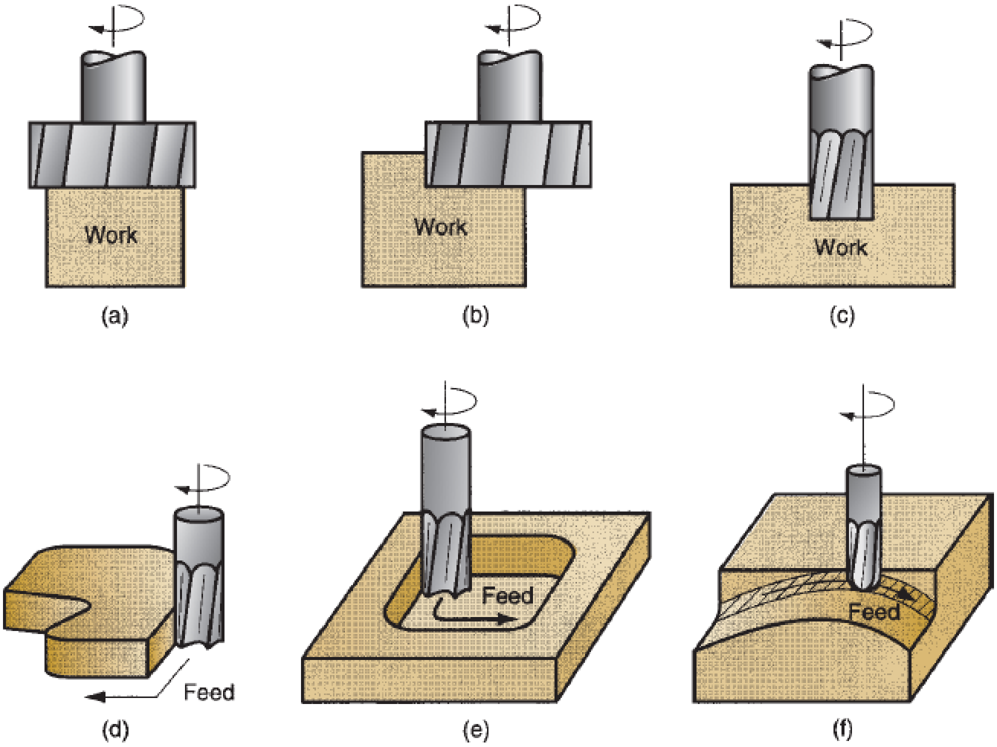


Figure 4.7. Face milling: (a) conventional face milling, (b) partial face milling, (c) end milling, (d) profile milling, (e) pocket milling, and (f) surface contouring

- **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 workpiece.

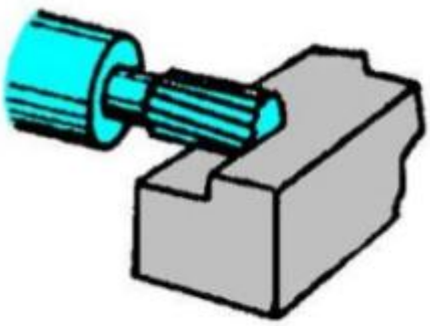


Figure 4.8. Side milling

- **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

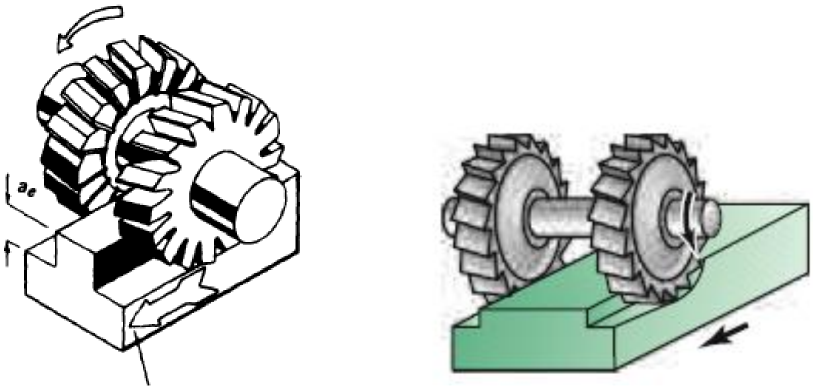


Figure 4.9. Straddle Milling

- **Angular Milling Operation**

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

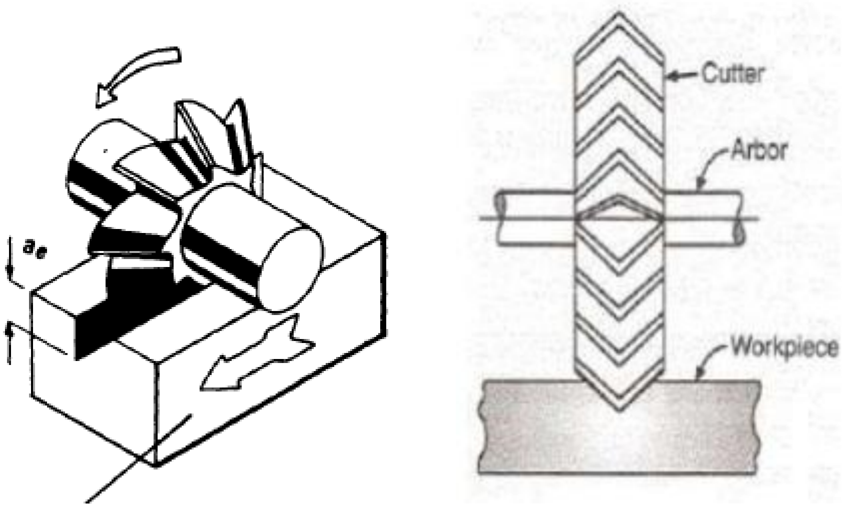


Figure 4.10. Angular milling

- **Gang Milling Operation**

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

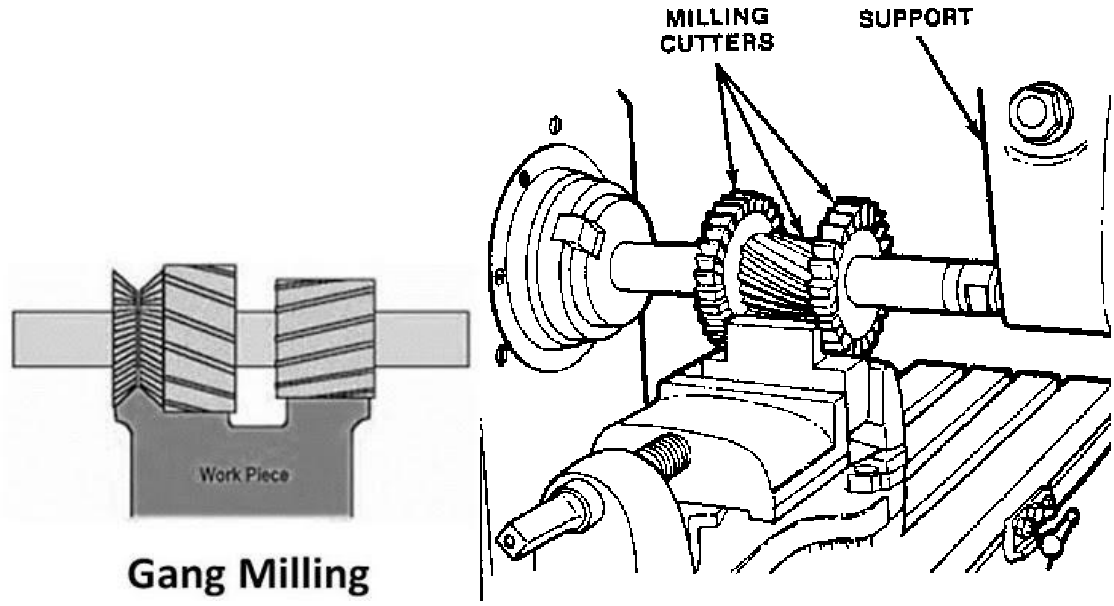


Figure 4.10. Gang milling

- **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.

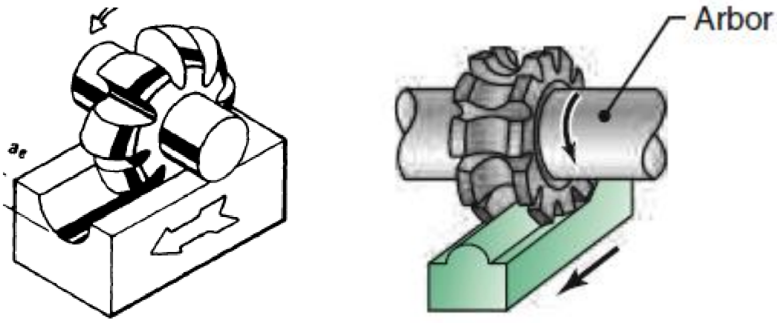


Figure 4.12. Form milling

- **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.

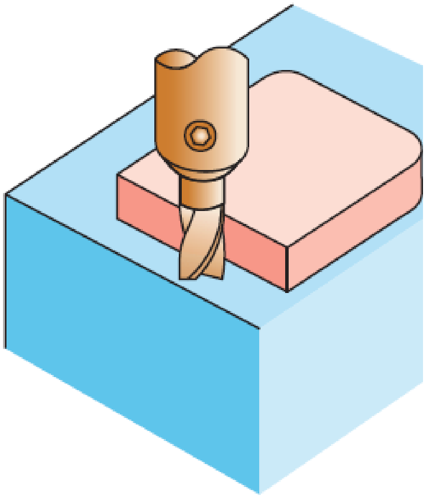


Figure 4.13. Profile milling

- **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.

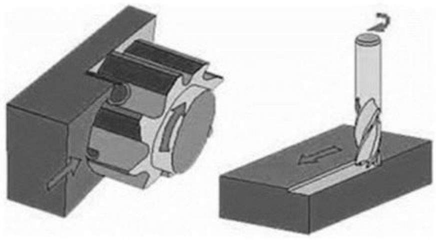


Figure 4.14. End milling

- **Saw Milling Operation**

Saw milling operation produces narrow slots or grooves into the workpiece using saw milling cutter. This operation is also used to cut the workpiece into two equal or unequal pieces which cut is also known as “parting off”. In case of parting off operation cutter and workpiece 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.

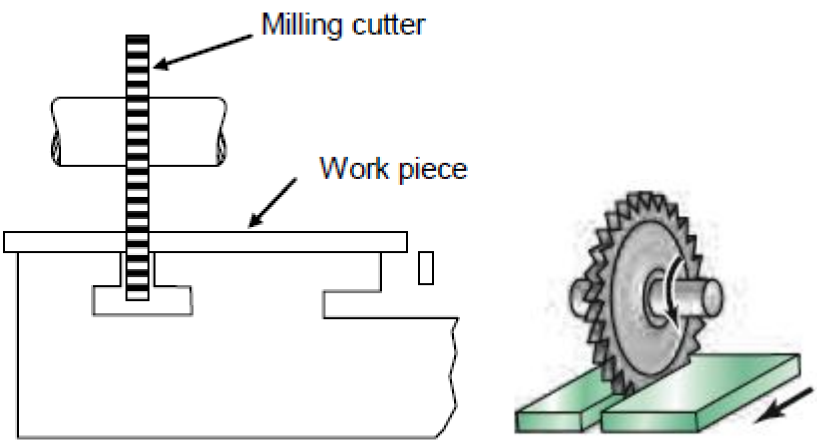


Figure 4.15. Saw milling

- **Slot Milling Operation**

The operation of producing keyways, grooves, slots of varying shapes and sizes is called slot milling operation. Slot milling operation can use any type of milling cutter like plain milling cutter, metal slitting saw or side milling cutter. Selection of a cutter depends upon type and size of slot or groove to be produced. Right placement of milling cutter is very important in this operation as axis of cutter should be at the middle of geometry of slot or groove to be produced.

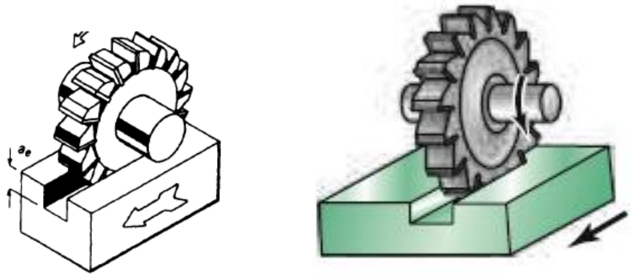


Figure 4.16. Slot Milling

- **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.

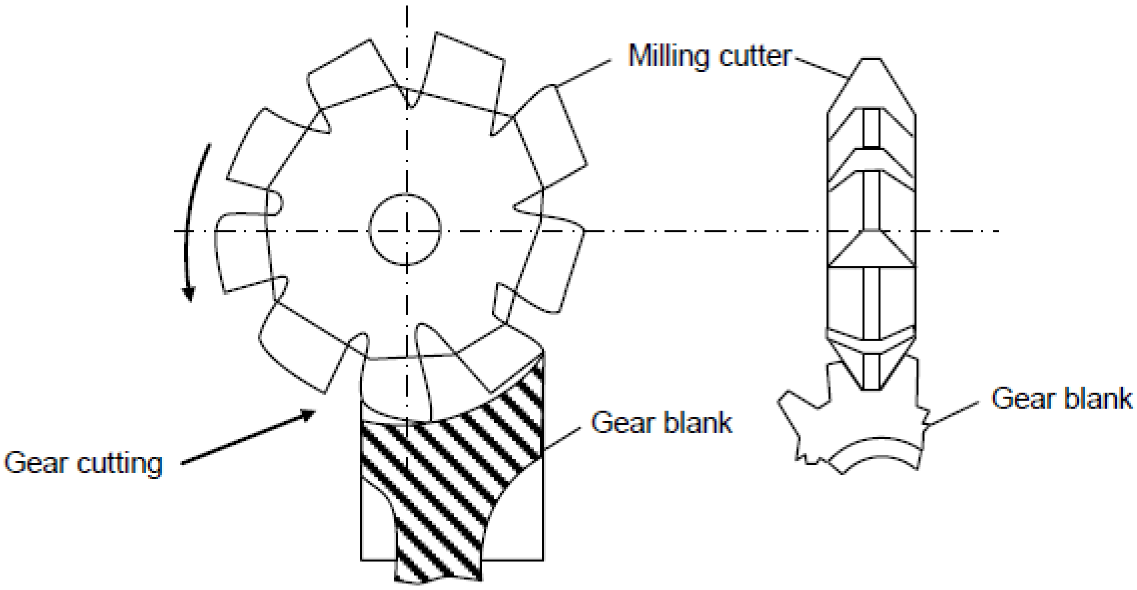


Figure 4.17. Gear cutting

- **Helical Milling Operation**

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

Figure 4.18. Helical milling

- **Cam Milling Operation**

The operation cam milling is used to produce the cam on milling machine. In this operation cam blank is mounted at the end of the dividing head spindle and the end mill is held in the vertical milling attachment.

Figure 4.19. Cam shaft

- **Thread Milling Operation**

The operation thread milling produces threads using thread milling centres. 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.

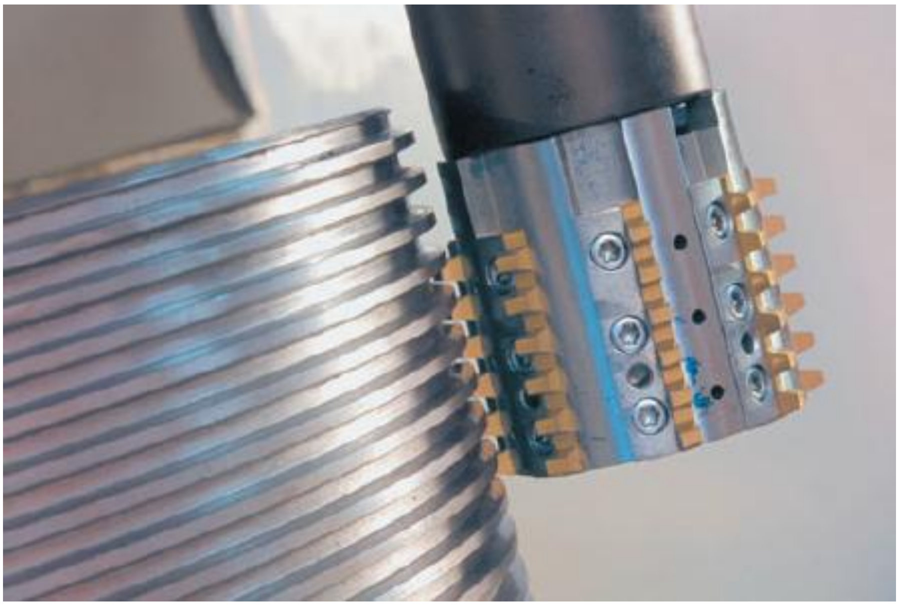


Figure 4.20. Typical thread milling cutters. The pitch of the cutter teeth must match the thread pitch

- **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 rachet 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.

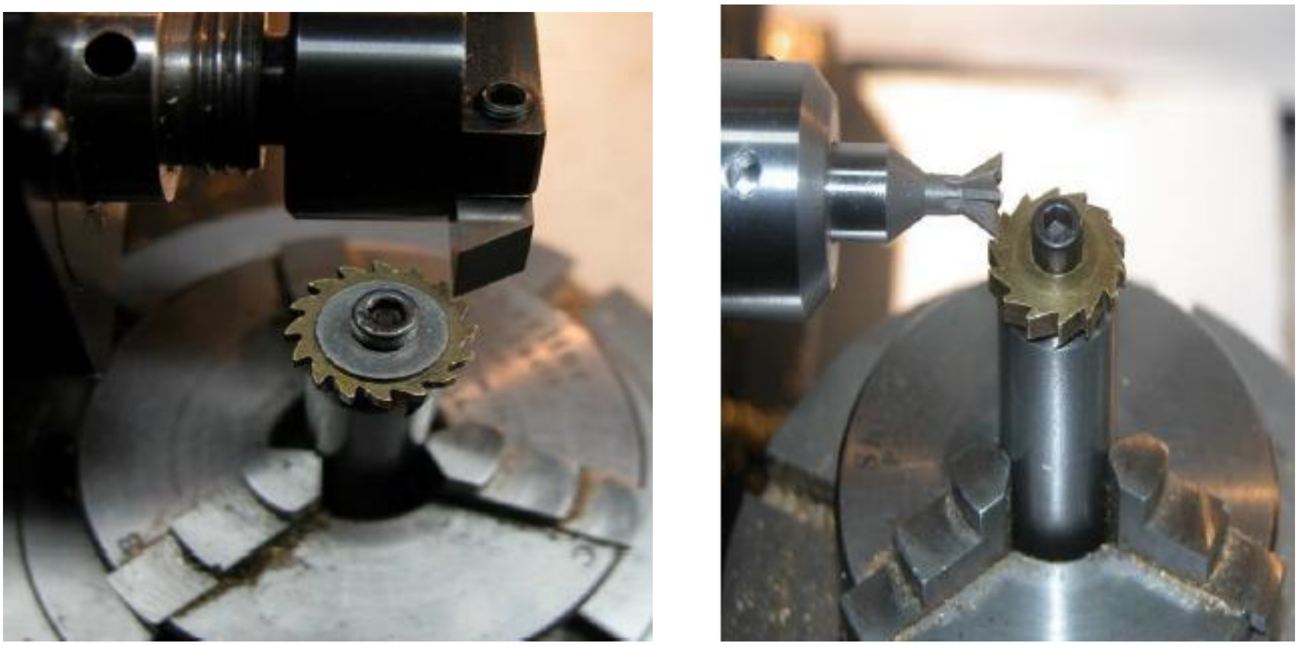


Figure 4.21. Ratchet milling

4.4. 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 workpiece. 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.

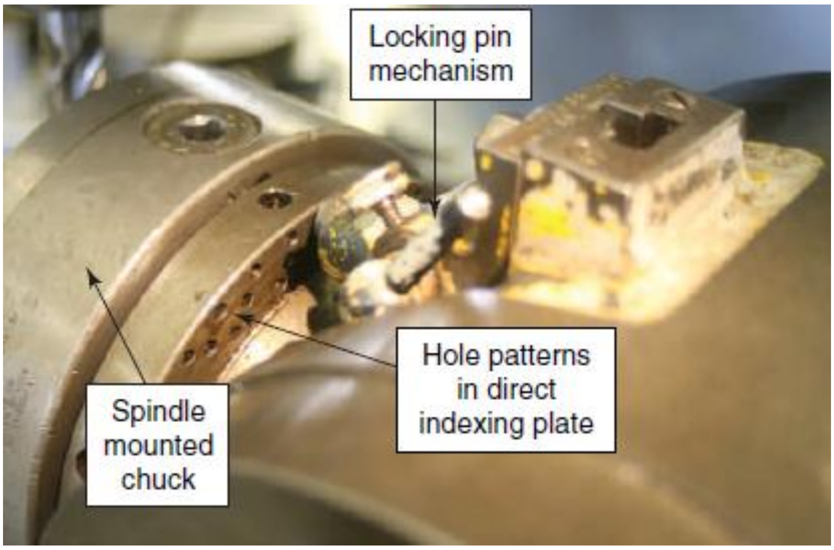


Figure 4.22. The proper indexing plate and plunger positions must be selected for direct indexing.

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÷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.

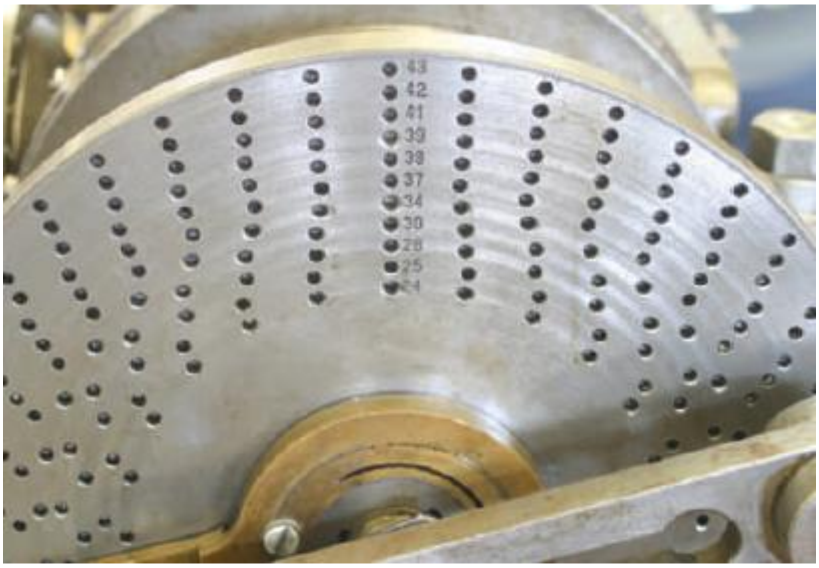


Figure 4.23. A simple indexing plate. Note that each circle has a different number of equally spaced holes

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 workpiece: 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 involute gear cutters. These cutters are manufactured in many pitch sizes and shapes for different numbers of teeth per gear. If involute 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 involute 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.

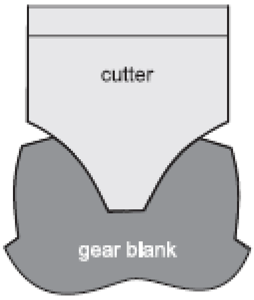
Fasten the cutter bit that has been ground to the shape of the gear tooth spaces in the fly cutter arbor. Adjust the cutter centrally with the axis of the gear. Rotate the milling machine spindle to position the cutter bit in the fly cutter so that its cutting edge is downward.

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

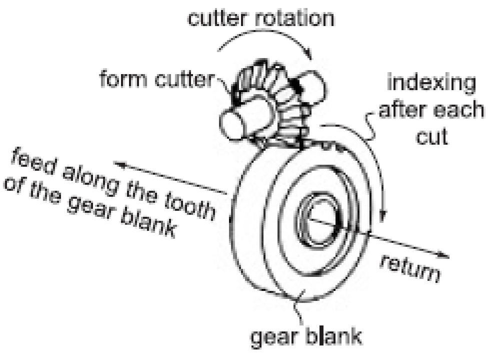
In gear form cutting, the cutting edge of the cutting tool has a shape identical with the shape of the space between the gear teeth.



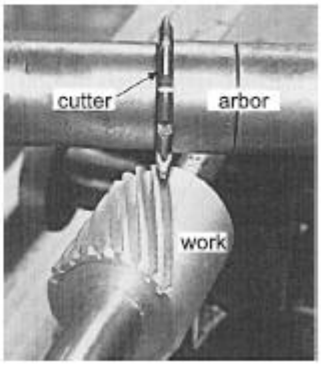
The principle of gear forming.

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.

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. Gears are compact, Positive-engagement, power transmission elements that determine the speed, torque, and direction of rotation of driven machine elements. Gear types may be grouped into five main categories: Spur, Helical, Bevel, Hypoid, and Worm. Typically, shaft orientation, efficiency, and speed determine which of these types should be used for a particular application.

Spur Gear

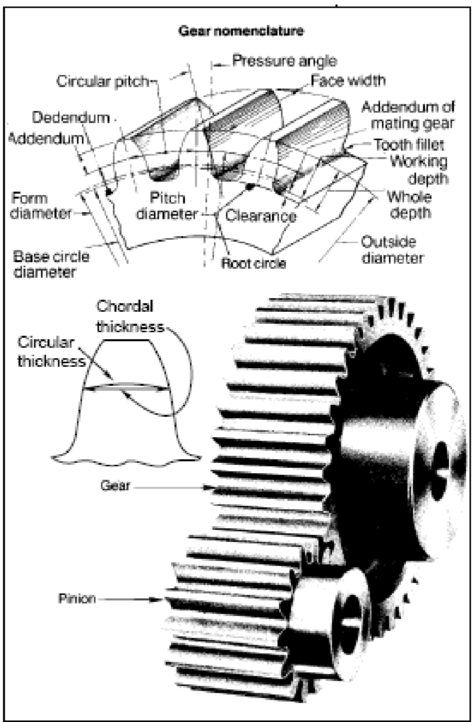
Helical Gear

Bevel Gear

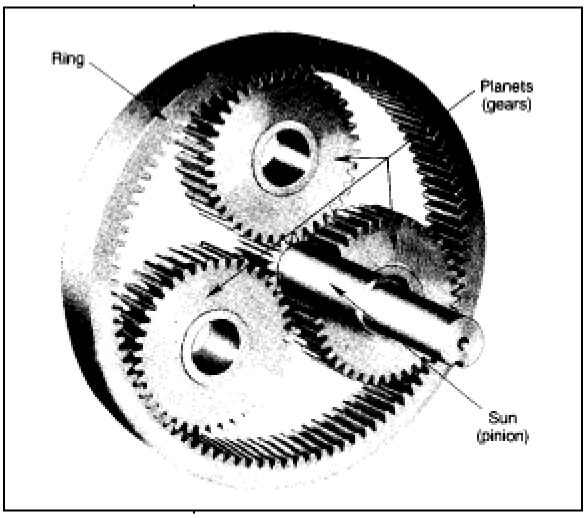
Worm and Worm Wheel

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.



Spur gears have straight teeth cut parallel to the rotational axis.



Internal (ring) gears produce a complex form of output with a planetary configuration of sun, planets, and ring.

Spur gears nomenclature

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

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.	$c = b - a =$ (
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_p	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_p	Diameter of pith circle of gear and pinion	$D = N/P$
Pressure angle	$\emptyset$	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 dedendim.	$D_R = D - 2b$
Whole depth	h_t	Total height of the tooth; equal to the addendum plus the dedendum.	$h_t = a + b =$
Working depth	h_k	Distance a tooth projects into mating space; equal to twice the addendum.	$h = 2a = 2/$

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$(\boldsymbol{m})$**: -** is defined as the ratio of the pitch circle diameter to the number of teeth on the gear. $\boldsymbol{m} = \frac{D}{N}$ 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

$B = 2p_c \text{ to } 4p_c \text{ or } 2\pi m \text{ to } 4\pi m$ 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 $D_a = D + 2a = D + 2m = 128 + 2 \times 4 = 136$

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

Hub diameter $\mathrm{D_h} = 1.5\mathrm{D_s}$ to $2\mathrm{D_s}$ where $\mathrm{D_s}$ is the shaft diameter

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

Hub length L = B to 1.4B

Assume L = 1.2B = 1.2 x 37.7 = 45.24mm

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

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

Spur gearing

Given	**gear m = 5**	**pinion m = 5**
	$\mathbf{N_G = 40}$	$\mathbf{N_P = 16}$
	$\mathbf{D_{sg}=40}$	$\mathbf{D_{sp}=40}$

Pitch diameter of pinion $D_p = mN_p = 5 \times 16 = 80mm$

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

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

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

Denendum circle dia. of pinion

$\mathrm{D_{bP}} = \mathrm{D_P} - 2\mathrm{b} = 80 - 2\mathrm{m}\left(1 + \frac{\pi}{20}\right) = 80 - 2 \times 5\left(1 + \frac{\pi}{20}\right) = 68.43 \cong 68\mathrm{mm}$

Denendum circle dia. of gear

$\mathrm{D_{bG}} = \mathrm{D_G} - 2\mathrm{b} = 200 - 2\mathrm{m}\left(1 + \frac{\pi}{20}\right) = 200 - 2 \times 5\left(1 + \frac{\pi}{20}\right) = 188.43 \cong 188\mathrm{mm}$

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

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

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

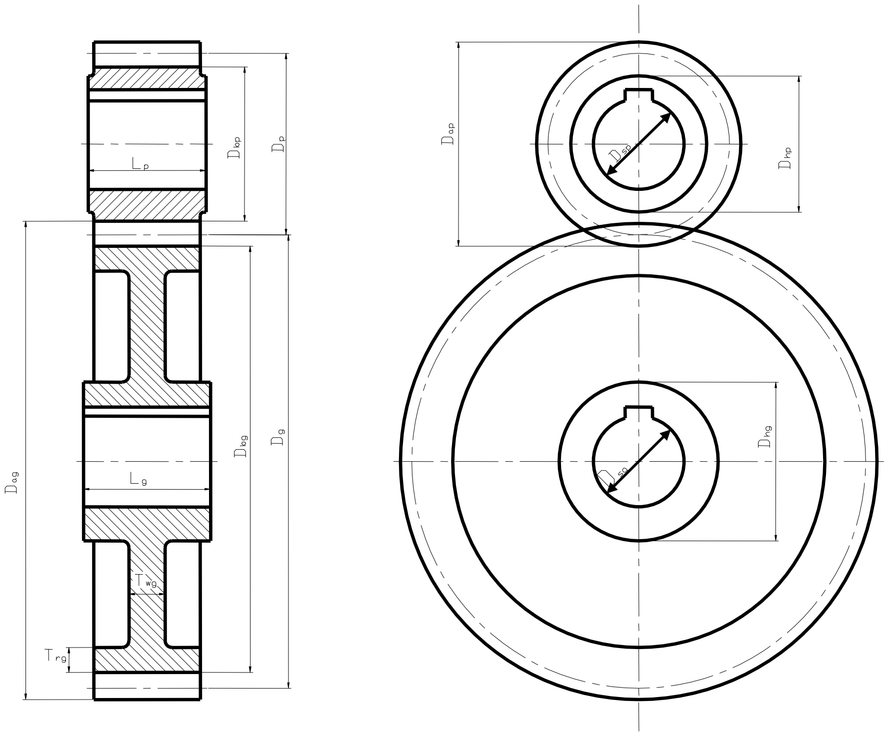
Hub length $L_P = 1.1B = 1.1 \times 47 = 51.7 \cong 52\text{mm}$

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

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

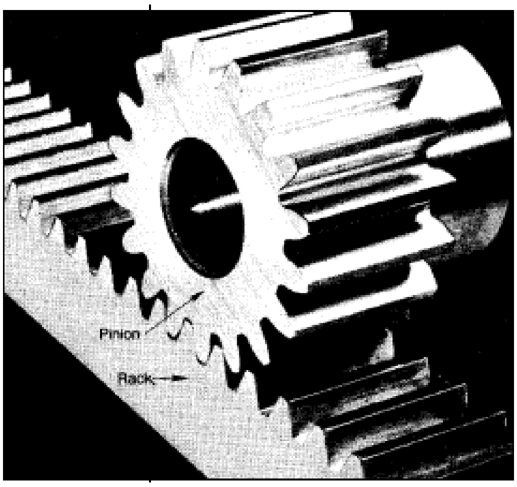
Rim thickness $T_r = a + b = m + \left(1 + \frac{\pi}{20}\right)m = 5 + \left(1 + \frac{\pi}{20}\right) \times 5 = 10.75 \cong 11\text{mm}$

Key way width= 12mm depth = 4mm



Rack-and-pinion

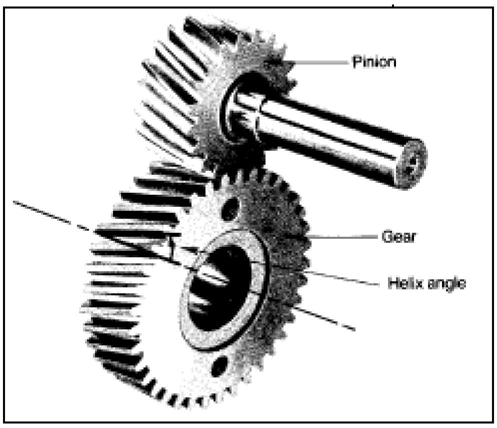
Gears — a straight bar with teeth cut straight across it, is called a rack. Basically, this rack is considered to be a spur gear unrolled and laid out fiat. Thus, the rack-and pinion is a special case of spur gearing. The rack-and-pinion is useful in converting rotary motion to linear and vice versa. 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. Generally, the rack is operated without a sealed en-closure in these applications, but some type of cover may be provided to keep dirt and other contaminants from accumulating on the working surfaces.



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 use 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. The increased amount of sliding action between helical gear teeth, however, places greater demands on the lubricant to prevent metal-to-metal contact and resulting premature gear failure. Also, since the teeth mesh at an angle, a side thrust load is produced along each gear shaft. Thus, thrust bearings must be used to absorb this load so that the gears are held in proper alignment.



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

The following are the new terms added in helical gears

Normal pitch P_n this is measured normal to tooth faces

$P_n = P_c cos\varphi$ *where* P_c *is the circular pitch in the diameteral plane and* φ *is the helix angle*

Then, pitch circle diameter $D = \frac{NP_c}{\pi} = \frac{NP_n}{\pi cos\varphi}$ *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 $\varphi = 20^0$

Normal circular pitch $P_n = 10mm$

Gear shaft diameter $D_s = 30mm$

$m = 5mm$

Thus, pitch circle diameter $D = \frac{NP_n}{\pi cos\varphi} = \frac{32 \times 10}{\pi cos20°} = 108.4 \cong 108mm$

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

Dedendum circle diameter

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

Face width $assume\ B = 3P_c = 3 \times \frac{P_n}{cos\varphi} = 3 \times \frac{10}{cos20°} = 32mm$

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

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

Web thickness $T_w = P_c = \frac{P_n}{cos\varphi} = \frac{10}{cos20°} = 10.6mm$

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

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

Bevel gears

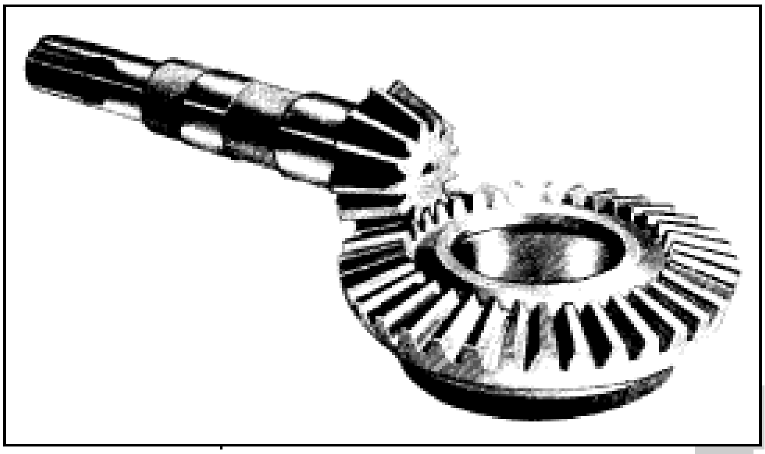
Unlike spur and helical gears with teeth cut from a cylindrical blank, bevel gears have teeth cut on an angular or conical surface. Bevel gears are used when input and output shaft centerlines intersect. Teeth are usually cut at an angle so that the shaft axes intersect at 90 deg, but any other angle 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.

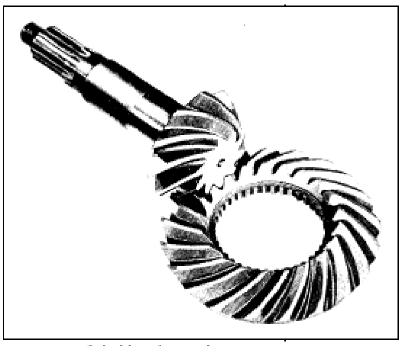
Spiral-bevels: — Curved teeth provide an action somewhat like that of a helical gear. This produces smoother, quieter operation than straight-tooth bevels. Thrust loading depends on the direction of rotation and whether the spiral angle at which the teeth are cut is positive or negative.

Term	Symbol	Definition	Formula
Addendum	a	Distance from pitch cone to top of tooth measured at larger end	$a = {}^{1}/_{P}$
Addendum angle	a	Angle subtended by addendum; same for gear and pinion.	$\tan a = {}^{a}/_{A}$
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.	$Z = Y + a \sin \Gamma$
Crown height	X	Distance, parallel to gear axis, from cone apex to crown of the gear.	$X = \frac{1}{2} D_o / \tan \Gamma_o$
Dedendum	b	Distance form 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	Γ_o	Angle between top of the teeth and the gear axis.	$\Gamma_o = \Gamma + a$
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.	$M = Y + \frac{1}{2} D_o / \tan \Gamma$
Outer cone distance	A	Slant height of pitch cone; same for gear and pinion.	$A = D/2\sin\Gamma$
Outer diameter	D_o	Diameter of outside or crown circle of the gear.	$D_o = D + 2a\sin\Gamma$
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	$\Gamma_R = \Gamma - \delta$

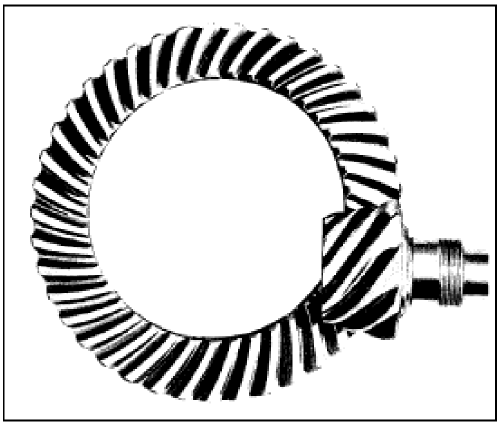
Pitch angle is the angle between the pitch line and the axis of the gear.
Pitch line is a straight line joining the vertex and the point on the outer edge of the tooth at pitch circle radius.



Straight-tooth bevel gears are efficient but somewhat noisy.



Spiral bevel-gears have curved teeth for smoother operation.



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

Bevel gearing

Pinion	**gear**
m=5mm	m=5mm
N_P=18	N_g = 54
D_s =30mm	D_s = 40mm
Keyway width x thickness = 10 x 4	Keyway width x thickness = 12 x 4

Shaft angle 90^0

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

$$\left(\begin{matrix}\text{Pitch angle of Gear or}\\ \text{Back angle of gear}\end{matrix}\right)\alpha_G = tan^{-1}\frac{sin\theta_s}{\frac{N_P}{N_G}+cos\theta} = tan^{-1}\frac{N_G}{N_P} = tan^{-1}\frac{54}{18} = tan^{-1}3 = 71.57° \cong 72°$$

Pitch diameter of pinion $D_p = mN_p$ = 5 x 18 = 90mm

Pitch diameter of Gear $D_G = mN_G$ = 5 x 54 = 270mm

Outer cone distance of pinion $A = \frac{D_P}{2sin\,\alpha_P} = \frac{90}{2sin\,18.43} = 142mm$

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

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

Addendum a = m = 5mm

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

Hub diameter D_{hP} =1.5 x D_s = 1.5 x 30 = 45mm

Hub diameter D_{hG} =2 x D_s = 2 x 40 = 80mm

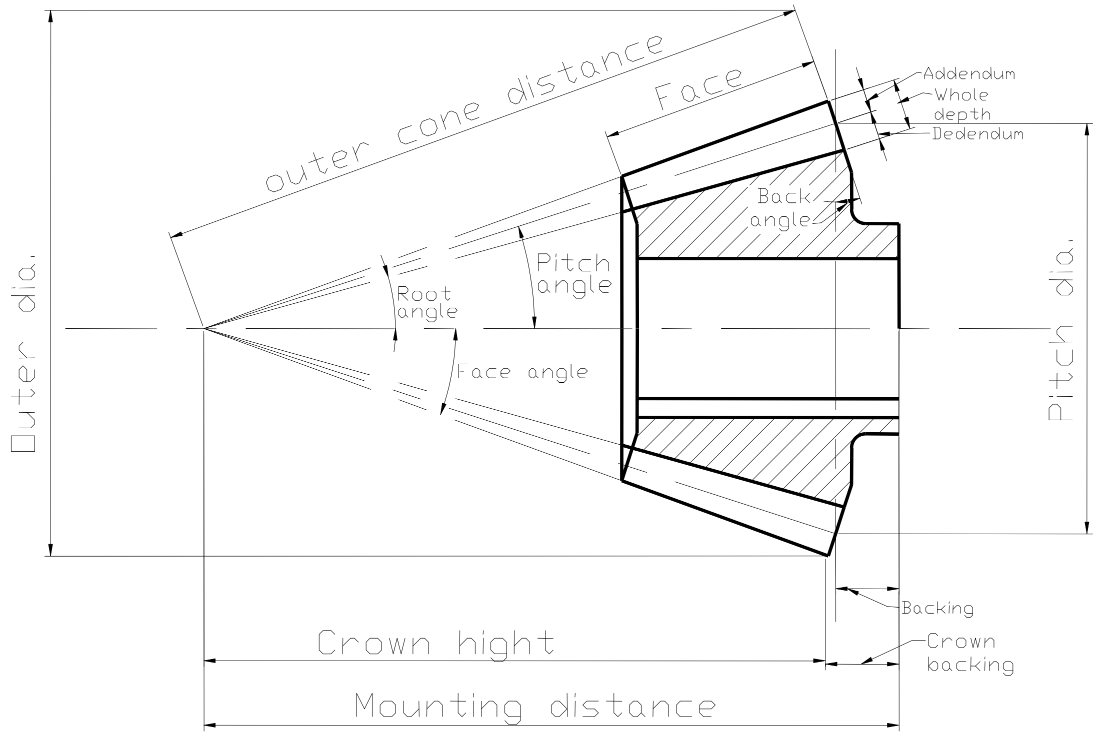
Hub length L_P = 1.2 x B = 1.2 x 47 ≅ 56.4≅ $56mm$

Hub length L_G = 1.4 x B = 1.4 x 47 ≅ 65.8≅ $66mm$

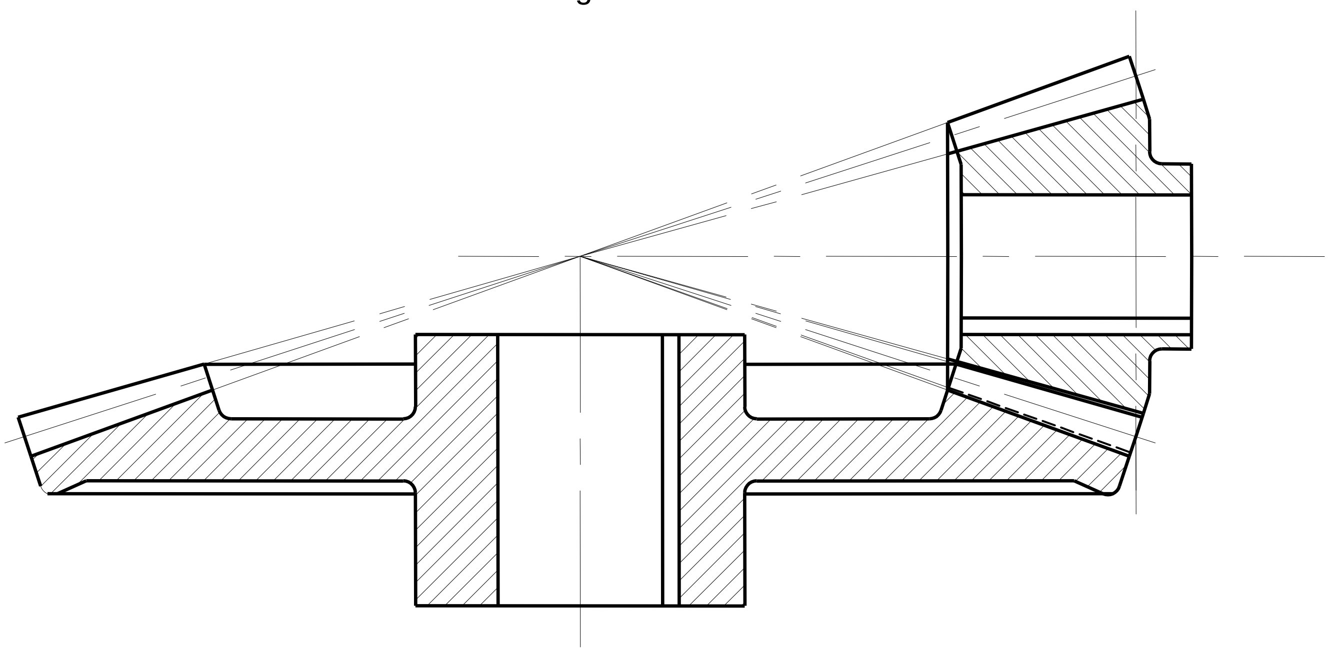
Web thickness of gear $T_{wG} = P_C = \frac{\pi D_G}{N_G} = \frac{\pi \times 270}{54} = 15mm$

Rim thickness of gear T_{rG} = a + b = 10.785 ≅ 11mm

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

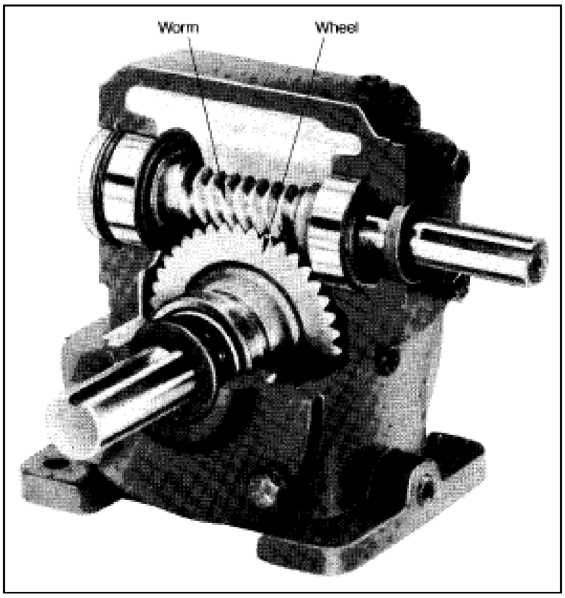


Bevel gear nomenclature



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. However, a reversible worm-gear has the worm and wheel pitches so proportioned that movement of the wheel rotates the worm. In most worm gears, the wheel has teeth similar to those of a helical gear, but the tops of the teeth curve inward to envelop the worm. As a result, the worm slides rather than rolls as it drives the wheel. Because of this high level of rubbing between the worm and wheel teeth, the efficiency of worm gearing is lower than other major gear types. One major advantage of the worm gear is low wear, due mostly to the full-fluid lubricant film that tends to be formed between tooth surfaces by the worm sliding action. A continuous film that separates the tooth surfaces and prevents direct metal-to-metal contact is typically provided by relatively heavy oil, which is often compounded with fatty or fixed oils such as acid-less tallow oil. This adds film strength to the lubricant and further reduces friction by increasing the oiliness of the fluid.



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.

Proportions

A worm and worm gear have a circular pitch of 15.748mm and the gear has 32 teeth of $14\,{}^{1}/_{2}\,^{\circ}$ 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 =15.748mm	P = 15.748mm
z = 2	N_G = 32
D_{sw} = 59mm	D_{sG} 41mm

For the worm

Pitch diameter D_W =2.4P + 27.94 = 2.4 x 15.748 + 27.94 = 65.74 $\cong 66mm$

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

Outside diameter $D_O = D_W + 0.636p = 66 + 0.636 \times 15.748 \cong 76mm$

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

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

Keyway width X thickness = 8 x 3.5mm

For the worm wheel

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

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

Outside diameter $D_O = D_t + 0.4775P = 170 + 0.4775 \times 15.748 = 177.9 \cong 178mm$

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

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

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

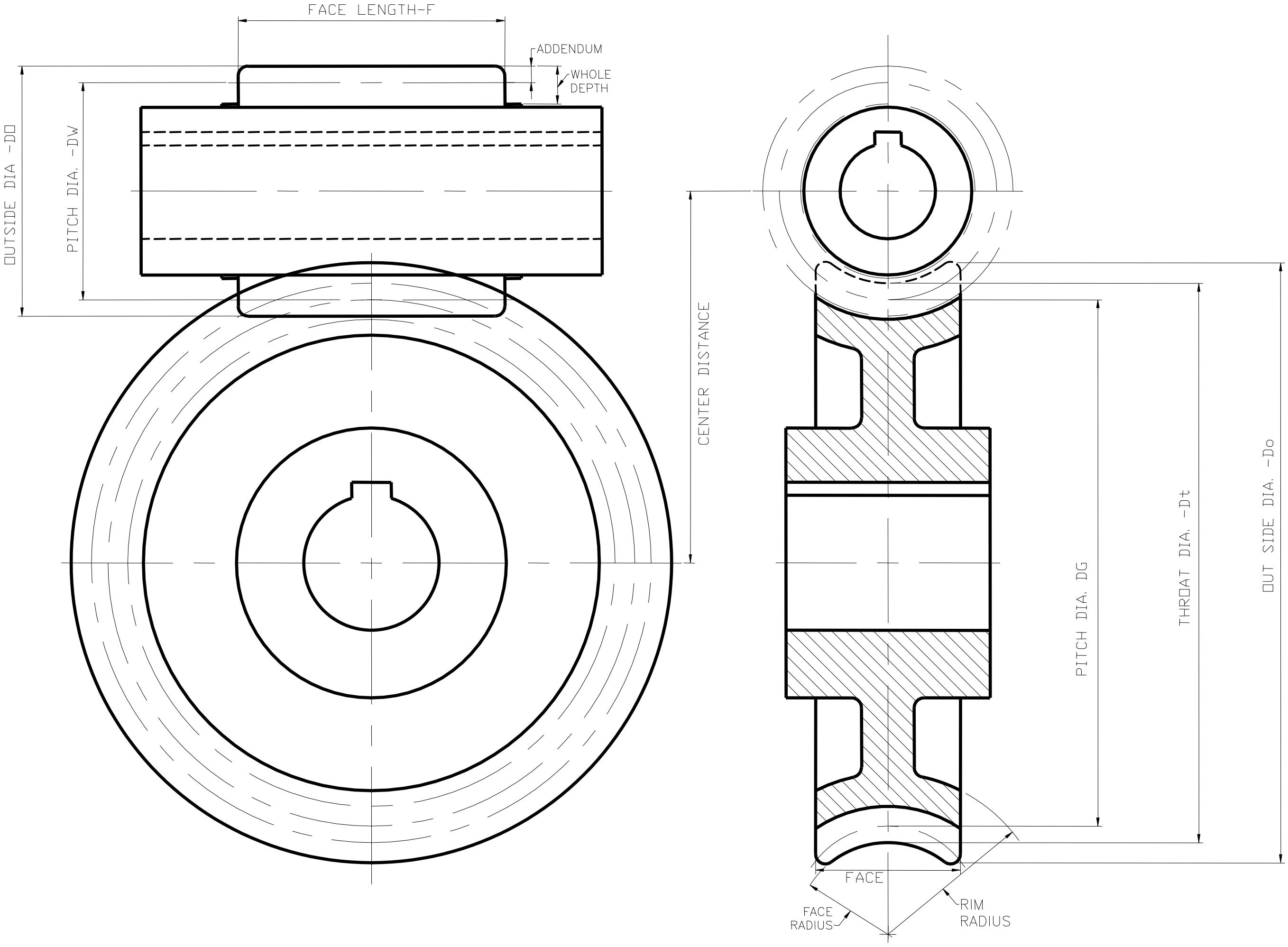
Center distance $C = \frac{1}{2}(D_G + D_w) = \frac{1}{2}(160 + 66) = 113mm$

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

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

Web thickness$T_w = P = 15.748 \cong 16mm$

Keyway width x thickness = 12 x 4mm



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.

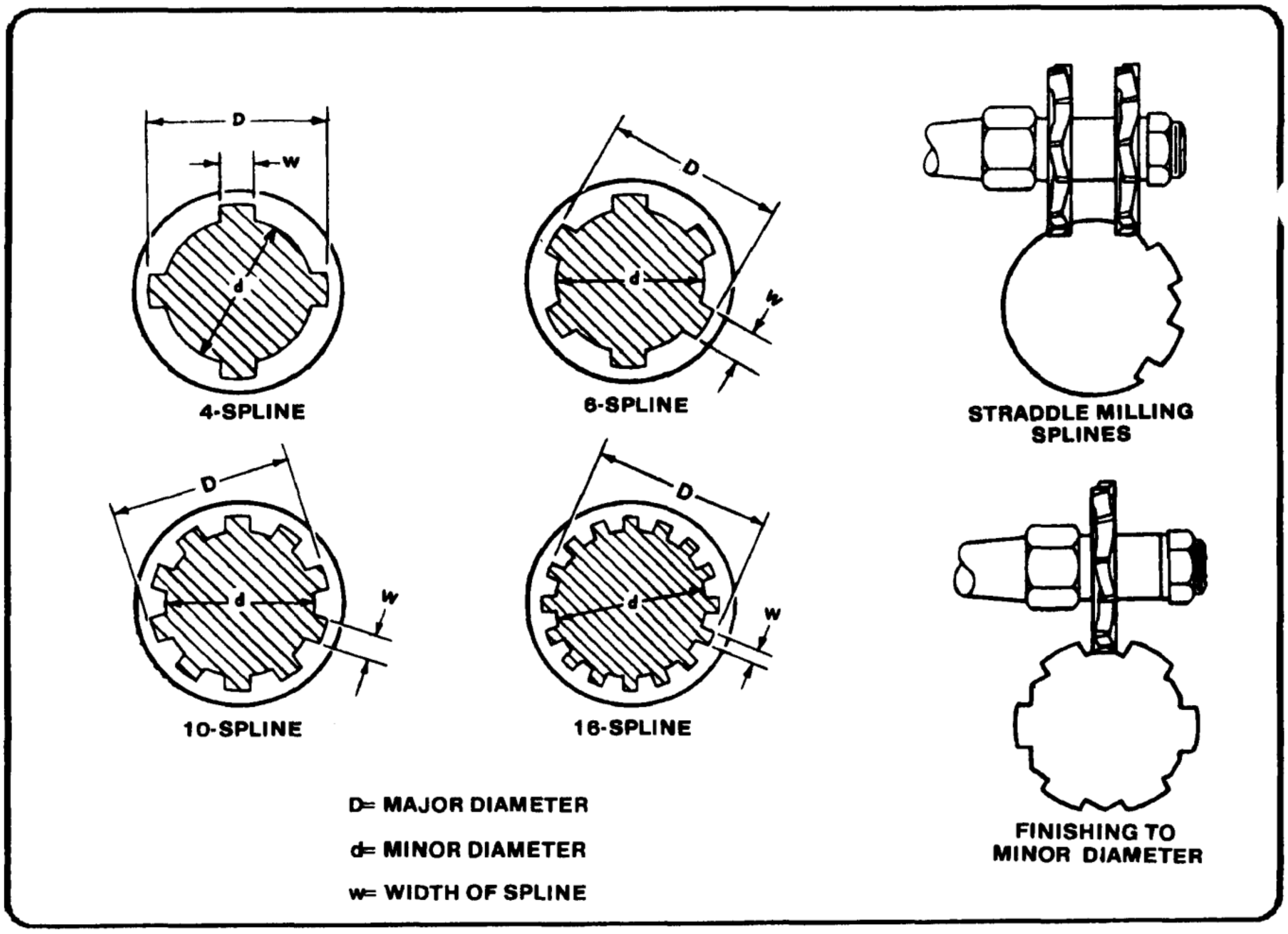
The shaft to be splined is set up between centers in the indexing fixture.

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.

The splines are cut by straddle milling each spline to the required depth and using the index head of the indexing fixture to rotate the workpiece the correct distance between each spline position.

After the splines are milled to the correct depth, mount a narrow plain milling cutter in the arbor and mill the spaces between the splines to the proper depth. It will be necessary to make several passes to cut the groove uniformly so that the spline fitting will not interfere with the grooves. A formed spline milling cutter, if available, can be used for this operation.



Milling spines on shafts

Self-Check - 4	Written Test

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

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) All of the above.
   e) None of the above

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 true 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. All
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 B. slotters 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 can be done by milling machine?
   a) cutting keyways
   b) slots and grooves
   c) gears
   d) all of the mentioned

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 1	Calculating speeds and feeds

Computation of feed, cutting speed and machine rpm

1. Select cutting speeds and calculate the r/min for various cutters and materials
2. Select and calculate the proper feeds for various cutters and materials
3. Follow the correct procedure for taking roughing and finishing cuts

Operation Sheet 2	Performing milling operations produce component including gears

Cutting spur gear on milling machine (gear cutting operation)

Aim:

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

Tools required:

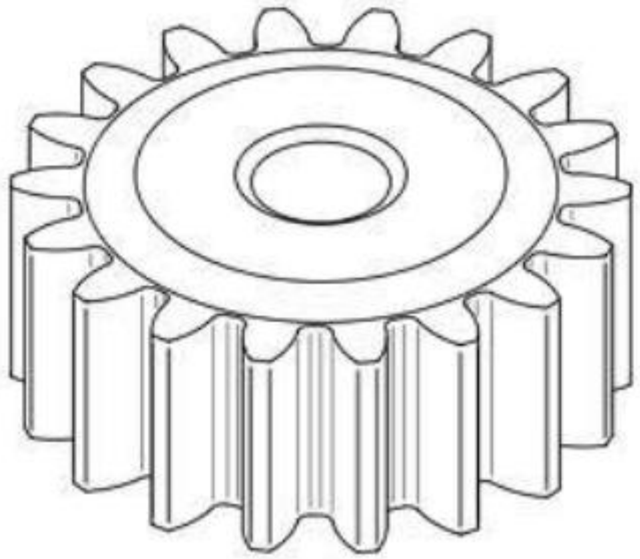
Horizontal milling machine, Spur gear cutter, vernier caliper, gear tooth vernier, indexing attachment.

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 form 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.
12. Form dividing head suitable method of indexing is adopted to get the equal number is visions 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 devises while machining is going on.

Result:

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



Operation Sheet 3	Performing milling operations produce component 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.

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-4	Performing milling operations produce component including gears

Milling key way

1. Copied the drawing.
2. Fixed the work piece on machine vice.
3. Fixed the side milling cutter on the arbor.
4. Performed key way milling with proper depth of cut.
5. 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-5	Performing milling operations produce component including gears

Aim

To perform the slotting on milling machine

Tools and equipment

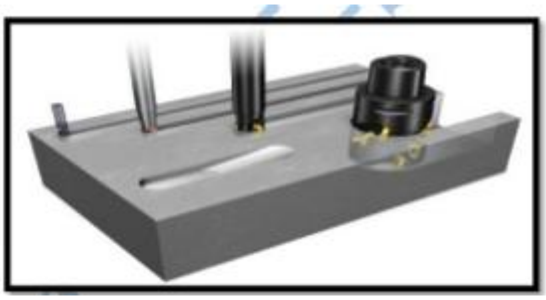
Milling machine, key slot milling cutter, vernier caliper

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 seed and feed
6. Mark the dimensions on the work piece perform the slotting operation



Operation Sheet-6	Performing milling operations produce component including gears

To mill T-slots

Tools and equipment

Vertical Milling machine, key slot milling cutter, vernier caliper

Material

Aluminum or m/steel block/shaft

Sequence of Operations

1. Use an end mill first to mill the clearance for the bolt body.
2. This size should be slightly larger than the bolt body to allow proper clearance so that the bolt body will fit almost snugly while permitting it to be freely moved in and out of the slot.
3. After milling the body clearance, remove the end mill and mount the appropriate T-slot cutter.
4. Locate the correct depth and, using plenty of cutting fluid, mill out the slot.

LAP Test	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

Instruction Sheet	Learning Guide #- 31

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

- Checking /measuring work-piece
  - ✓ using techniques, measuring tools and equipment
- Checking work-piece conformance to finished quality
- Handling deviation according to organization

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, upon completion of this Learning Guide, **you will be able to –**

- Check/measure work piece using appropriate techniques, measuring tools and equipment in conformance to specification
- Check work piece in conformance with finished quality
- handle deviations appropriately in accordance with organization procedures and standard

Learning Instructions:

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below.
3. Read the information written in the "Information Sheets". Try to understand what are being discussed. Ask your trainer for assistance if you have hard time understanding them.
4. Accomplish the "Self-checks" which are placed following all information sheets.
5. Ask from your trainer the key to correction (key answers) or you can request your trainer to correct your work. (You are to get the key answer only after you finished answering the Self-checks).
6. If you earned a satisfactory evaluation proceed to "Operation sheets
7. Perform "the Learning activity performance test" which is placed following "Operation sheets" ,
8. If your performance is satisfactory proceed to the next learning guide,
9. If your performance is unsatisfactory, see your trainer for further instructions or go back to "Operation sheets".

Information Sheet-1	Checking /measuring work-piece

1.1. Introduction

Performing machining operations is only one part of a career in the machining field. Measurement is another core duty that is performed many times every day in the industry. During machining, dimensions, or specified print sizes, must be measured to be sure they are within required specifications. Every dimension shown on a print is given a certain amount of allowable variation. That allowable variation is called a tolerance. A tolerance gives a size range that is acceptable. Why do tolerances exist? They are an engineer's or designer's way of saying "so long as this print dimension is within this range, the machined part will function correctly."

1.2. Steel rules

The rule is by far the most common semi-precision measuring tool utilized in the machining field. It is a flat piece of steel with graduations that divide inches or millimeters into fractional parts.

Figure 1.1. Steel rule

- **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.

1.3. Calipers

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 1.2. Calipers

1.4. Screw Pitch Gage

A screw pitch gage determines the distance between threads. Each leaf is used for a different size. They are available in inch and metric versions. shows a typical screw pitch gage and how it is used.

Figure 1.3. (A) An example of a screw pitch gage. (B) Checking threads with a screw pitch gage.

1.5. Verniercaliper and digital calipers

- **Vernier 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.

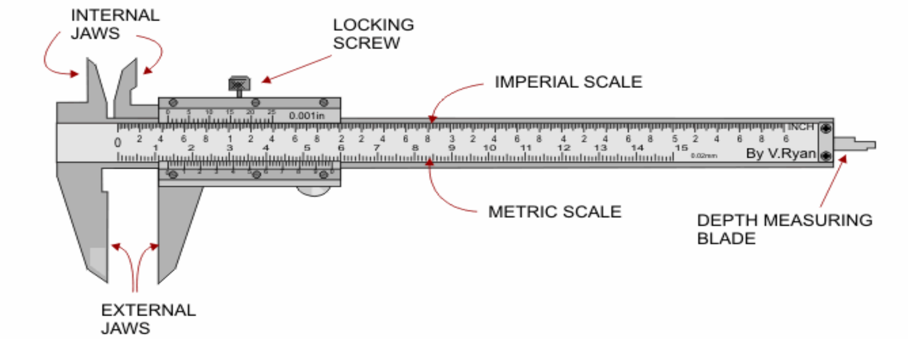


Figure 1.4. Parts of the Vernier caliper

- ✓ **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 main scale division –1vernier scale division

L.C = 1mm – 0.9mm

L.C = 0.1mm = 0.01cm

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

L.C = 1mm / 50 = 0.02mm

✓ **Reading the Vernier Scale**

Read the main reading up to "0" position of the vernier scale to get the main reading. Thus main reading = 18mm

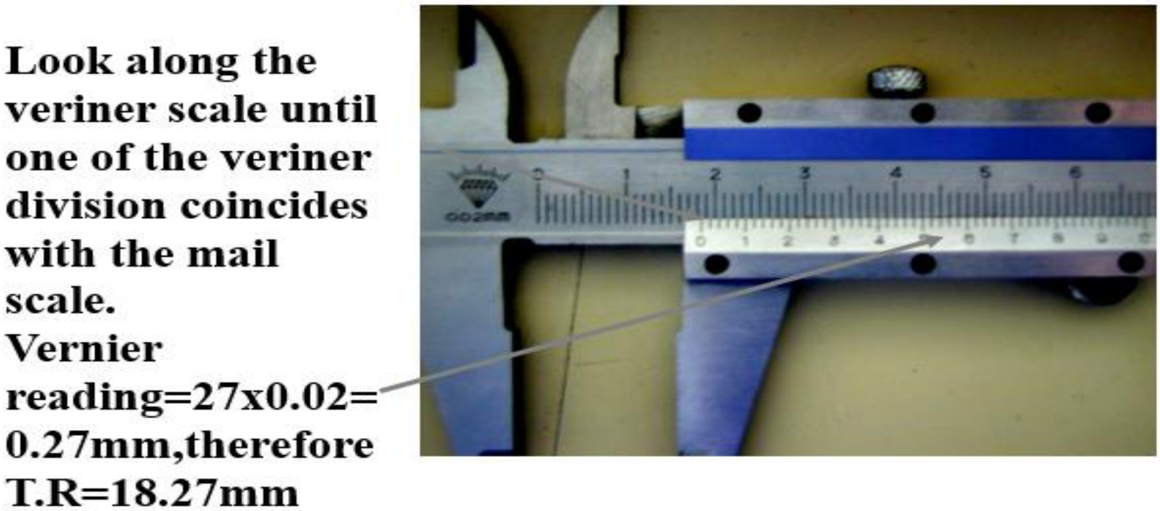


Figure 1.5. vernier caliper reading example

✓ **Steps to know the diameter of a piece of round section steel**

A. The main metric scale is read first and this shows that there are 13 whole divisions before the 0 on the vernier (hundredths) scale. Therefore, the first number is 13.

B. The vernier (' hundredths of mm') scale is then read. Only one division on the main metric scale lines up with a division on the vernier (hundredths) scale below it, whilst others do not. In the example below, 21 division on the vernier scale lines up exactly with a division on the main metric scale above.

C. Therefore 21 is multiplied by 0.02 giving 0.42 as the answer (each division on the vernier scale is equivalent to 0.02mm).

D. 13 and 0.42 are added together to give the final measurement of 13.42mm (the diameter of the piece of round section steel).

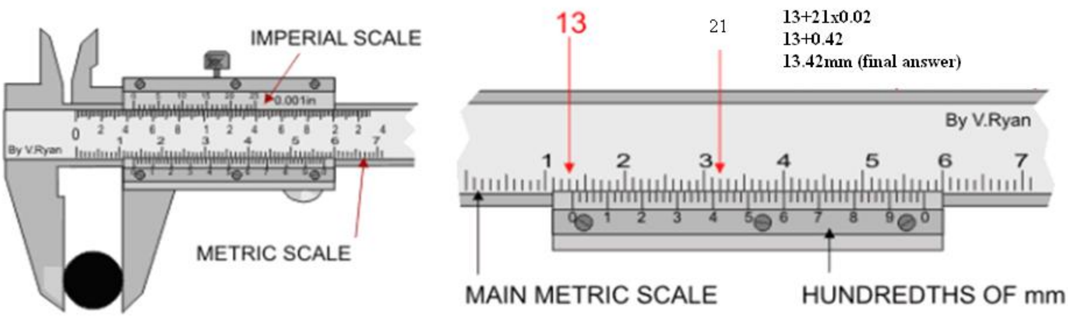


Figure 1.6. vernier caliper reading example 2

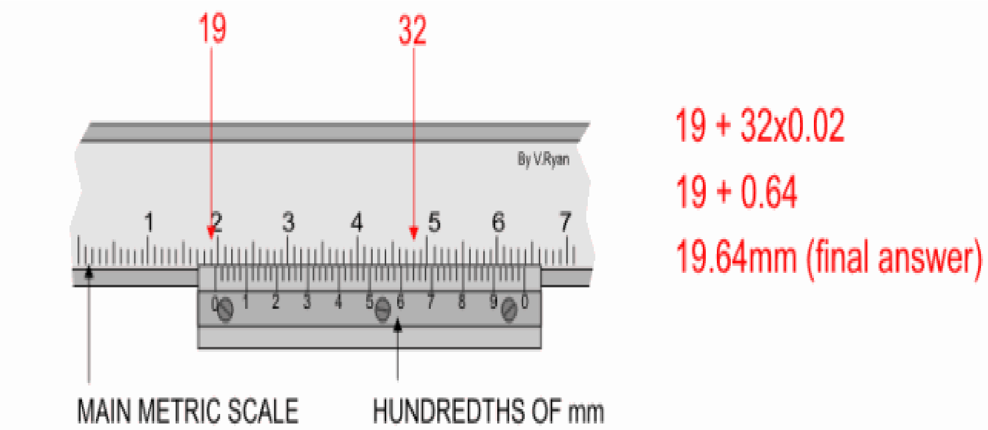


Figure 1.7. vernier caliper reading example 3

- **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.

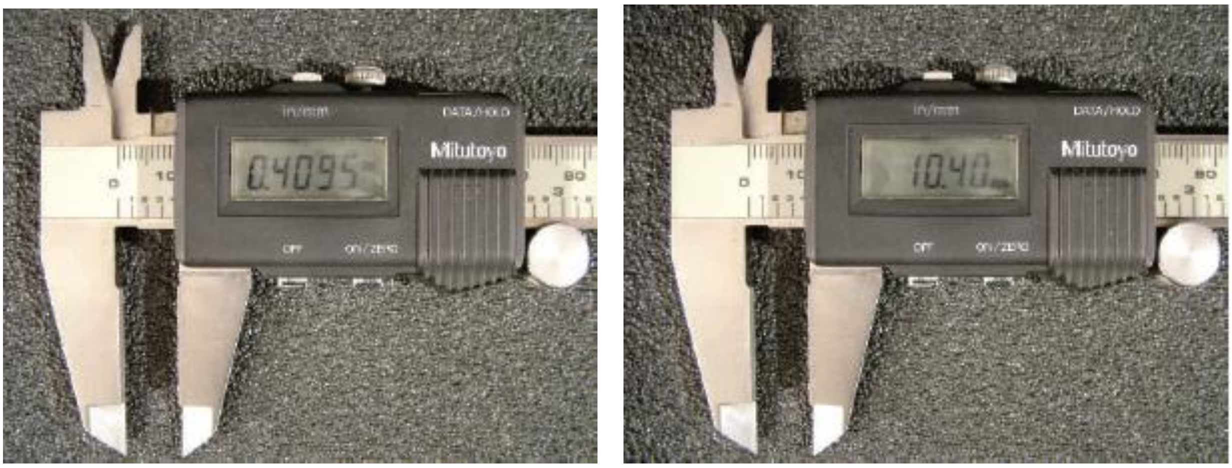


Figure 1.8. Digital calipers

1.6. Micrometer(internal, external and depth micrometer)

- **Internal Micrometer**

The internal micrometer uses a three-point measuring contact system to determine the size of a bore or hole. The instrument is direct reading and is more likely to yield a reliable reading because its three-point measuring contacts make the instrument self-centering as compared with a tool that uses only two contacts.

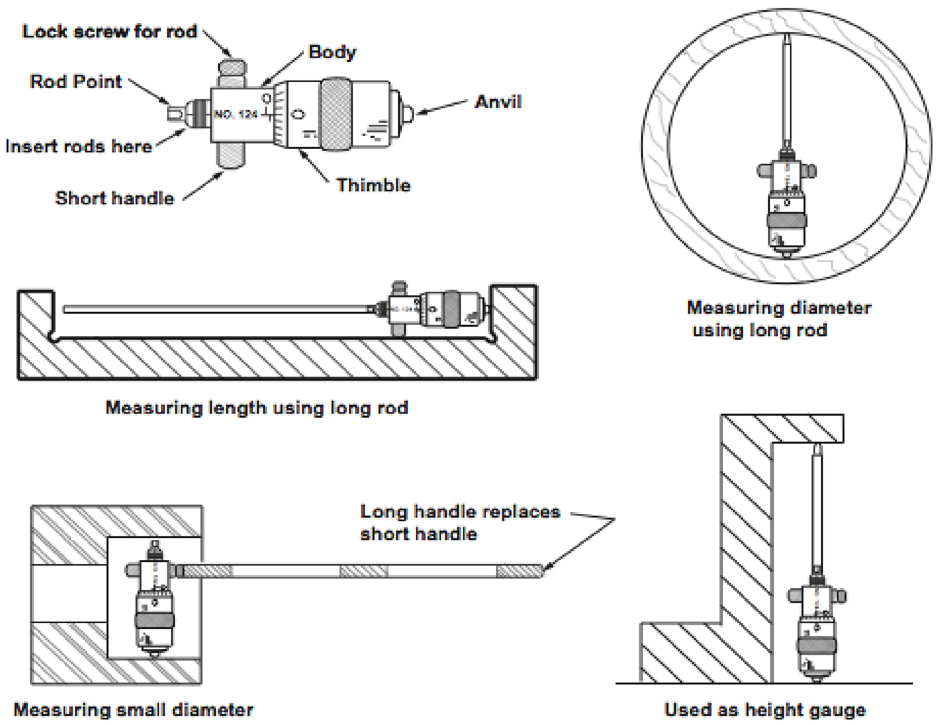


Figure 1.9. Internal micrometer

- **External (outside) Micrometer**

A micro meter is a precision instrument used to measure a job, generally within an accuracy of 0.01mm.Micrometers used to take the outside measurements are known as outside micrometers.

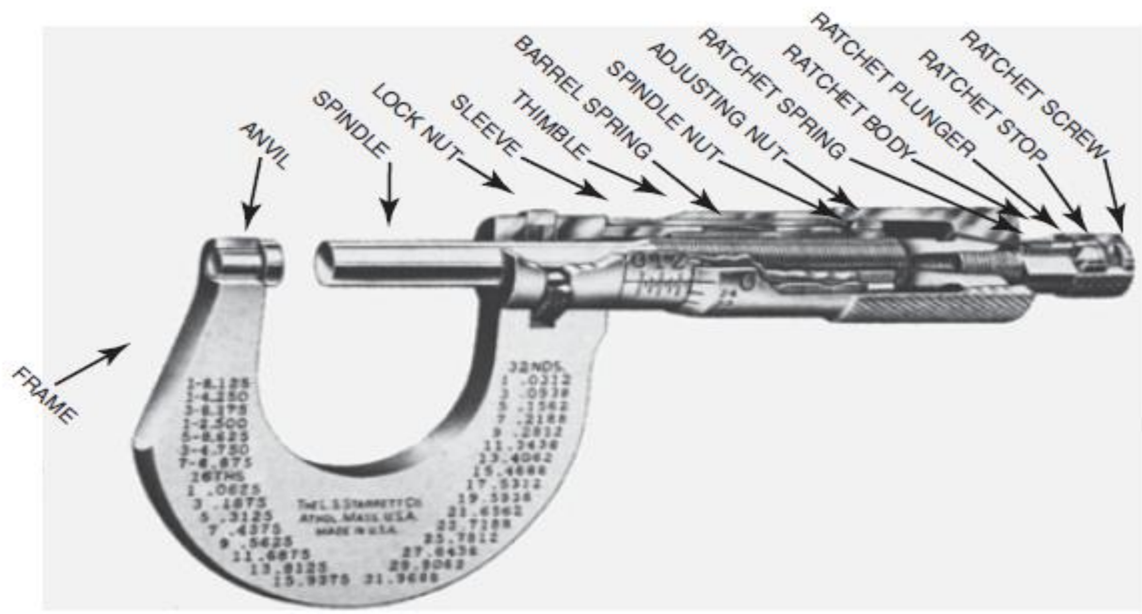


Figure 1.10. Parts of the outside micrometer

WOKING 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 to75mm, 75 to 100mm,100to125mm, 125 to 150mm
- 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.

-See below fig.
Main reading = 13.0mm
Half mm. reading= 0.5mm
Thimble reading=13x0.01=0.13mm
Total reading= 13.0+ 0.5 + 0.13
= 13.63mm

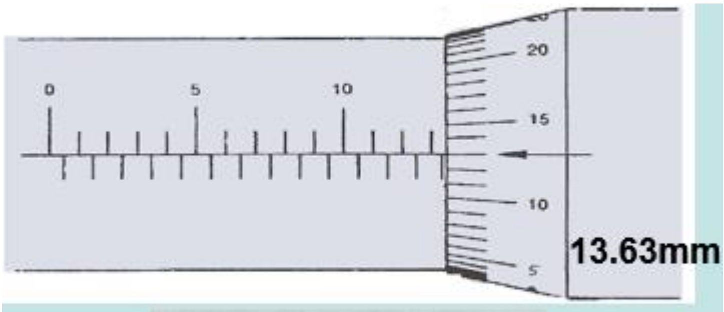


Figure 1.11. Micrometer reading example

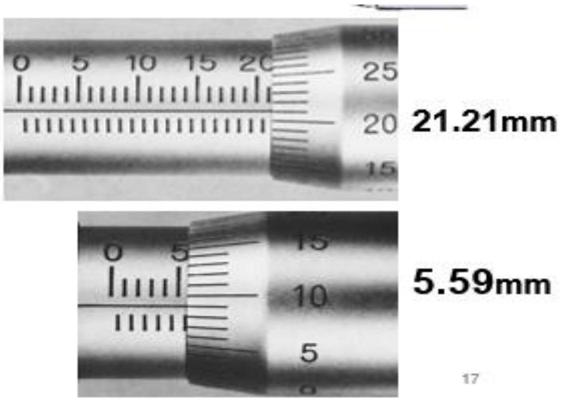


Figure 1.12. Micrometer reading exercises

- **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.

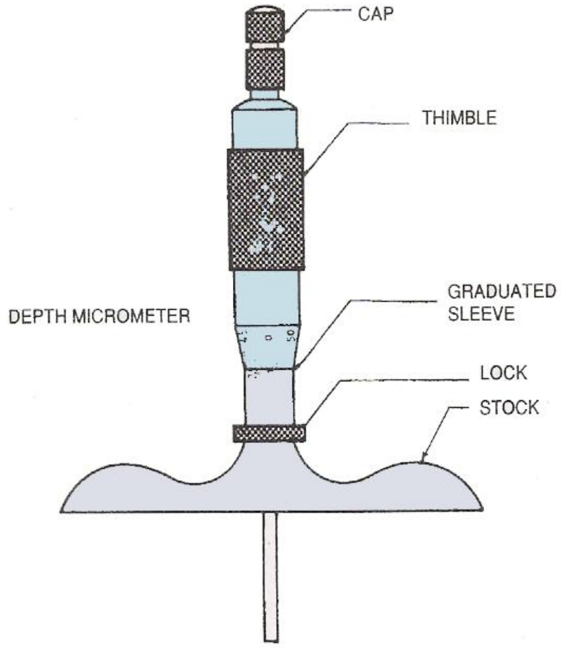


Figure 1.13. Depth micrometers

- **Vernier micrometer**

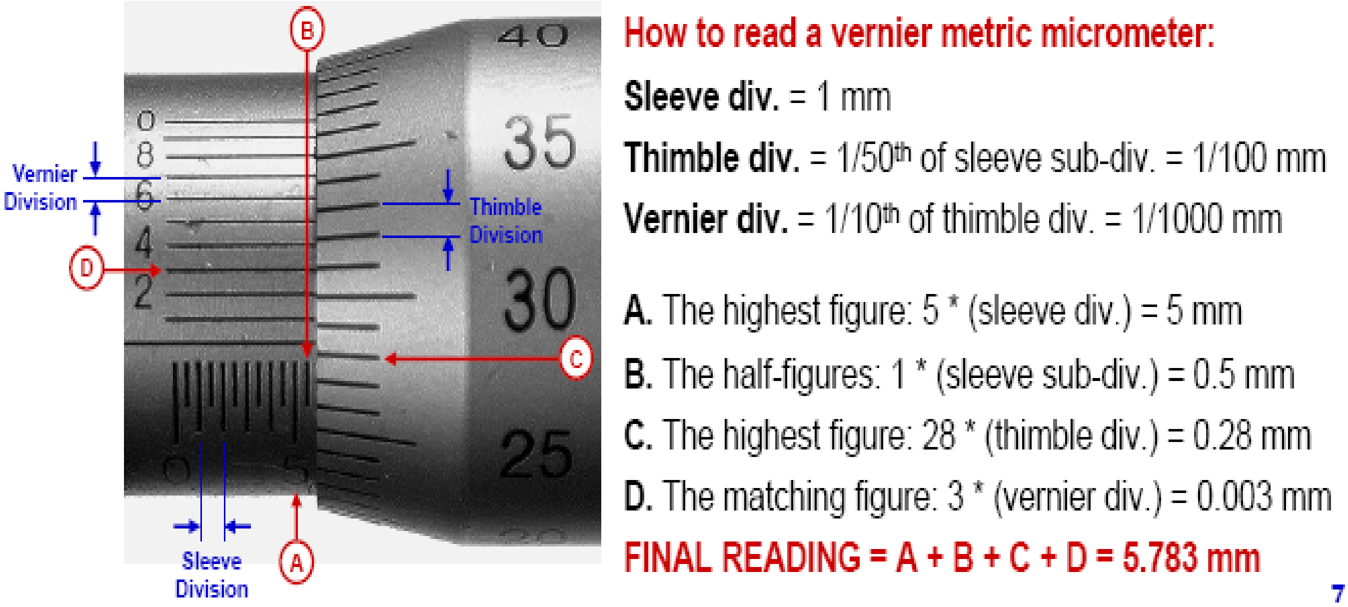


Figure 1.14. Vernier micrometer reading example

1.7. Gauges (height ,bore, surface finish/comparator, radius, depth, blocks)

- **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. The point acts just like the moveable jaw of a caliper. When the point is removed, other measuring attachments can be mounted to the tool. The pointer can also be used as a scribe for marking purposes.

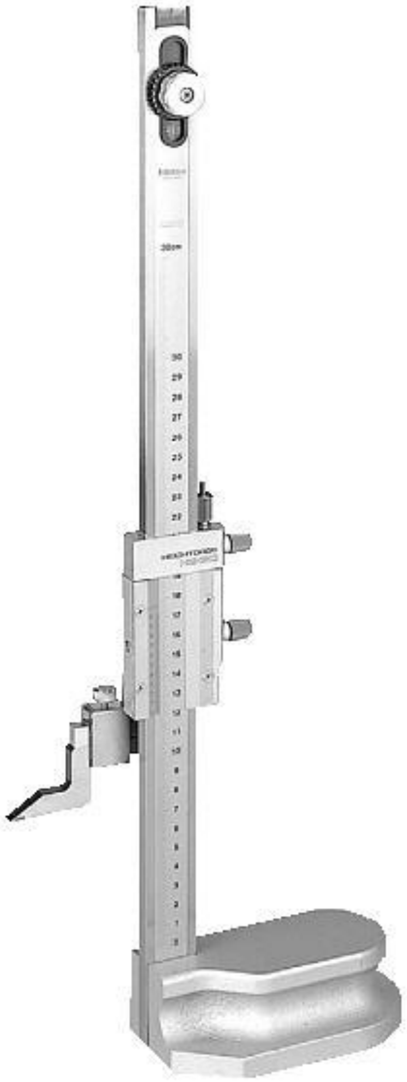


Figure 1.15. vernier height gage

- **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 1.16. (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.

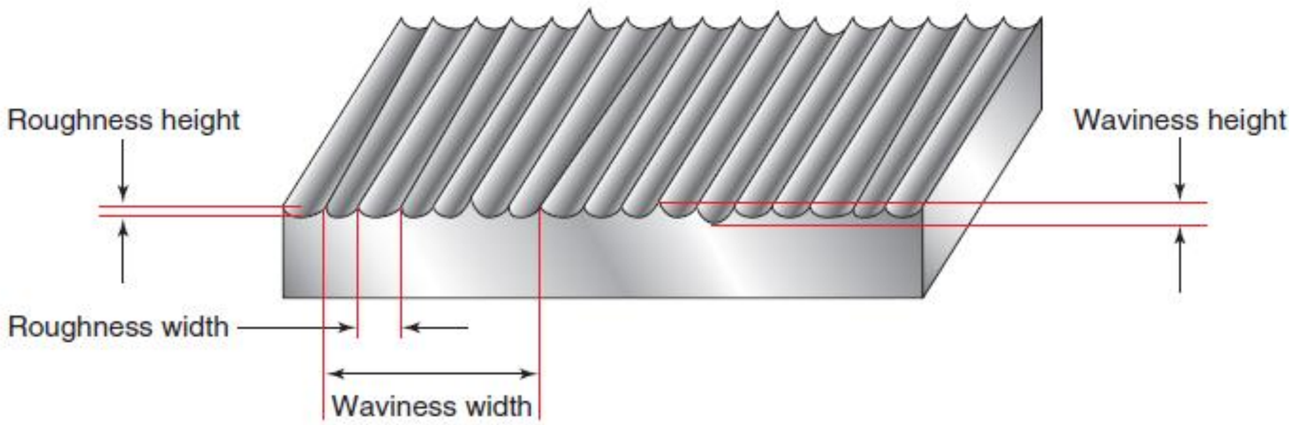


Figure 1.17. Surface characteristics

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.

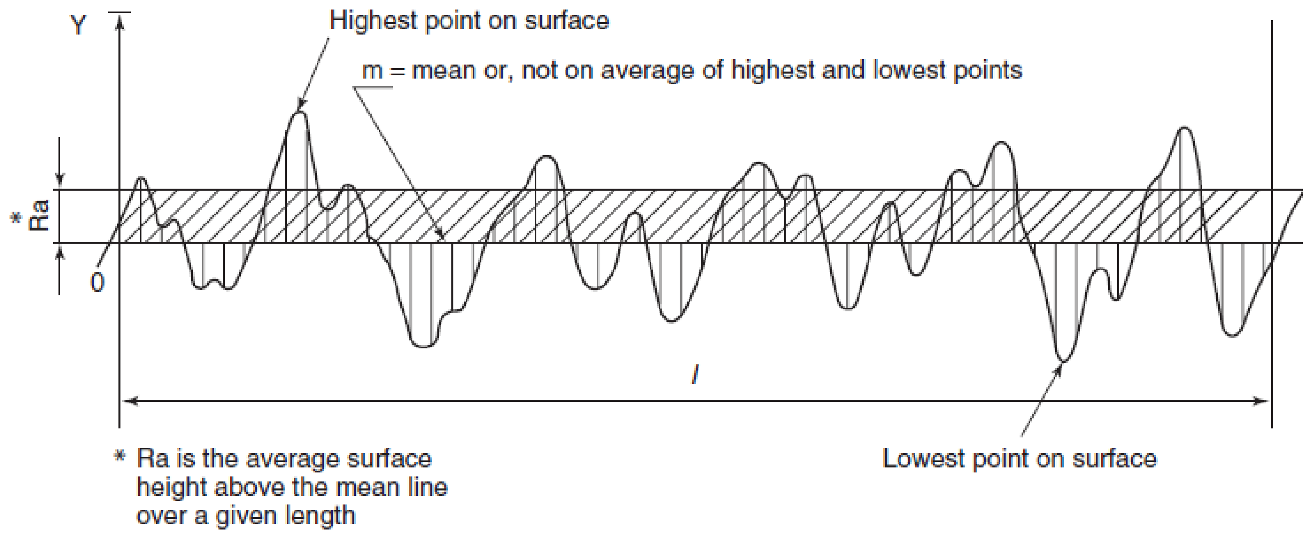


Figure 1.18. The basic concept of Ra surface finish measurement

- ✓ **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 microinches or micrometers, but it is simple, highly portable, cost effective, and adequate for many applications.

Figure 1.18. Surface roughness comparator

- ✓ **Profilo meter**

A more accurate measurement of Ra surface roughness can be obtained with a profilo meter. This electronic tool moves a stylus, or contact point, across a surface and actually measures the height of the peaks and valleys. An actual value in micro inches or micrometers is then automatically calculated and shown on a display.

Figure 1.19. profilometer

- **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.

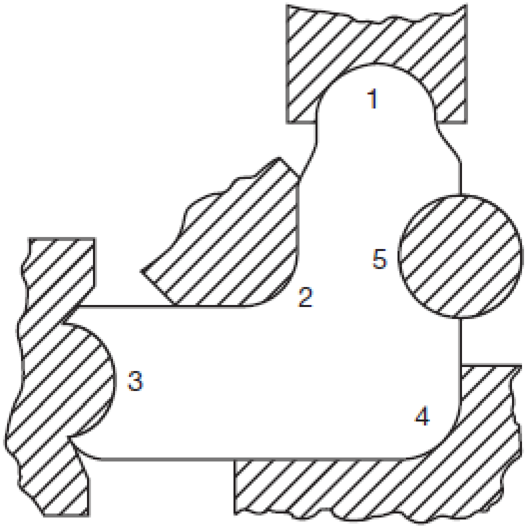


Figure 1.19. A. A radius and fillet gage set B. Uses of radius and fillet gages

- **Vernier Depth Gage**

A vernier depth gage uses a sliding rod similar to a vernier caliper with a depth rod. Also using 0.001" or 0.02-mm graduations, it usually has a measuring range of 0–6" (0–150 mm) or 0–12" (0–300 mm).

Figure 1.20. vernier depth gage

- **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.

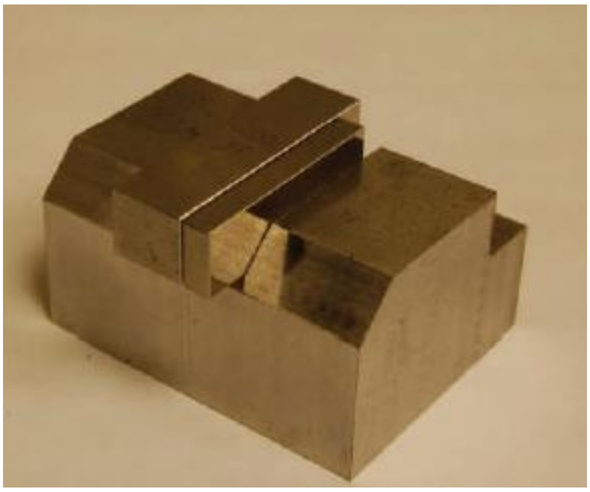


Figure 1.21. 21 A typical rectangular gage block set   Figure 1.21.Measuring a slot with gage blocks

1.8. 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.

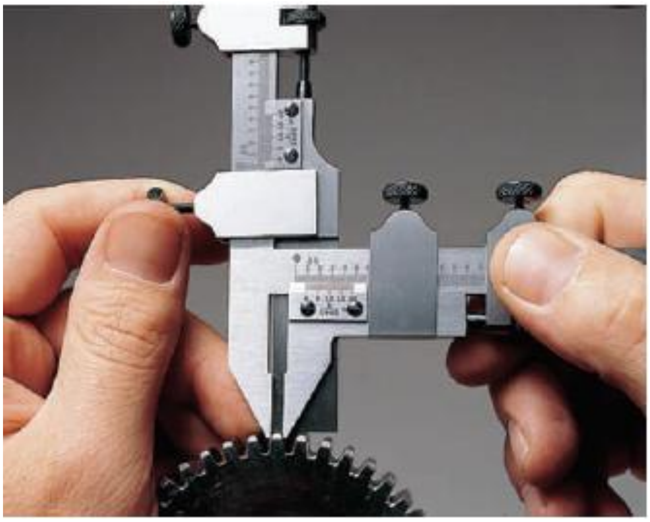


Figure 1.22. Vernier gear tooth caliper

1.9. 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.

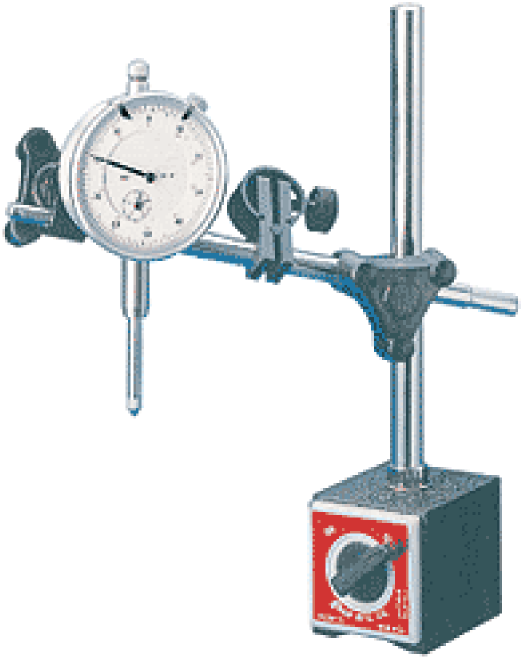


Figure 1.23. A. Dial indicators B. Dial indicators with magnetic stand

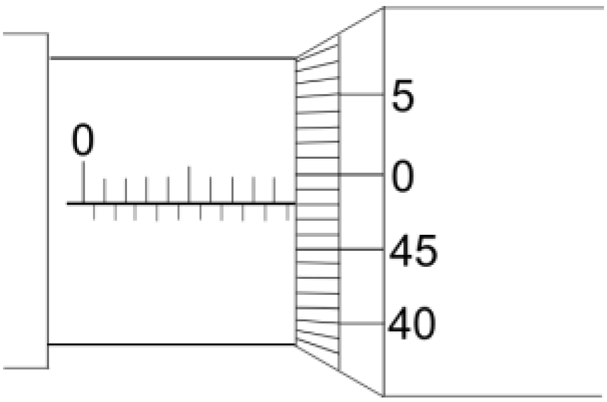
Dial Indicators Discrimination of dial indicators typically ranges from .00005 to .001 in. In metric dial indicators, the discrimination typically ranges from .002 to .01 mm. Indicator ranges, or the total reading capacity of the instrument, may commonly range from .003 to 2.000 in., or .2 to 50 mm for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

Figure 1.24. Dial indicators application

Self-Check -1	Written Test

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

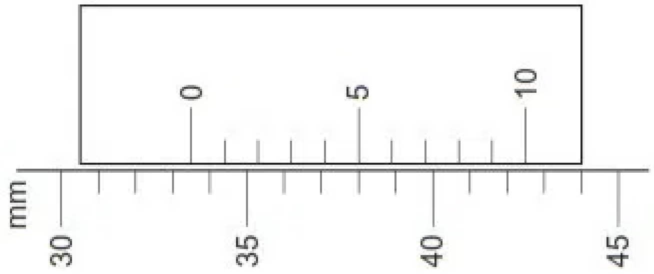
1. A tolerance is which one of the following:
   a) Clearance between a shaft and a mating hole,
   b) Measurement error,
   c) Total permissible variation from a specified dimension,
   d) Variation in manufacturing?
2. An outside micrometer would be appropriate for measuring which of the following (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?
   a) Vernier Caliper
   b) Measuring Tape
   c) Micrometer Screw Gauge
   d) Meter Rule

5. The diagram shows part of a vernier scale.



What is the correct reading?

a) 30.5 mm
b) 33.5 mm
c) 42.5 mm
d) 38.0 mm

6. A vernier caliper is precise to __________?

a) 0.01mm
b) 0.1mm
c) 1mm
d) 0.001mm

Information Sheet-2	Checking work-piece conformance to finished quality

2.1. 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.

This definition for conformance says it is right or it is not right for shipment. In manufacturing, when products conform, we ship them. If products don’t conform, they fail the quality test and we must rework, repair, sort good from bad, or scrap the product. We can’t ship nonconformances “as is”. Conformance sounds like an internal requirement, but it is not. Customers set external standards when they buy an advertised product. Costs and schedules are also important product conformance items. As a consumer, do you want to spend more and receive your purchase later than meets conformance to your requirements?
Conformance by sorting good product from bad is a terminal manufacturing illness. The sorting approach is better than nothing but not much better. Waste from the sort-and-suffer philosophy is insidious. Sorting begins with the best of intentions, but the results are costly. A better manufacturing method is: “Do it right the first time”.

Conformance has no conscience. Conformance doesn’t mean altering, accommodating, adjusting, or altering principles or specifications as if they were made from rubber. Conformance simply means making parts to the requirements. Conformance doesn’t require “more than” or “less than” drawings specify. Conformance is black or white. Conformance is the responsibility of the machine operator who is the only one who can produce conformance no one else. Conformance responsibility doesn’t pass to inspectors.

2.2. 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, workpiece 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.
- 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.

2.3. 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). If the feed per revolution (Feed per tooth ×Number of teeth) of the cutter is smaller than the length of the wiper flat (the land on the tool), then the surface finish on the workpiece will be generated by the highest insert. In finishing cuts, keeping the depth of cut small will limit the axial cutting force, reducing vibrations and producing a superior 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.

Self-Check -2	Written Test

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

True or false

1. Conformance doesn't require "more than" or "less than" drawings specify
2. Conformance is not the responsibility of the machine operator
3. Costs and schedules are important product conformance items.

Information Sheet - 3	Handling deviation according to organization

A machinist is mainly concerned with the measurement of length that is, the distance along a line between two points. A machinist measures length in the basic units of linear measure such as inches, millimeters, and, in advanced metrology, wavelengths of light. In addition, the machinist sometimes needs to measure the relationship of one surface to another, commonly called angularity. Squareness, closely related to angularity, is the measure of deviation from true perpendicularity. A machinist will measure angularity in the basic units of angular measure degrees, minutes, and seconds of arc. In addition to length and angularity, a machinist also needs to measure such things as surface finish, concentricity, straightness, and flatness. He or she also occasionally sees measurements that involve circularity, sphericity, and alignment. However, many of these more specialized measurement techniques are in the realm of the inspector or laboratory metrologist and appear infrequently in general machine shop work.

Measurement Error: Measurements are never perfect. There are always measurement errors. The machinist must be aware of potential sources of measurement errors and try to reduce them. Sources of measurement error include: the measuring tool selected, the procedure used, the temperature of the part, the temperature of the room, the cleanliness of the room air, and the cleanliness of the part at the time of measurement, and many others. One example of a method is the use of a micrometer. A user must develop a consistent "feel" to accurately use a micrometer. A machinist who uses a micrometer, as if it were a c-clamp, will get a much different measurement than a machinist with the correct "feel."

Part designers are aware of measurement error. They are also aware that no part can be made perfectly to match the specification, so they develop part tolerances for the part features.

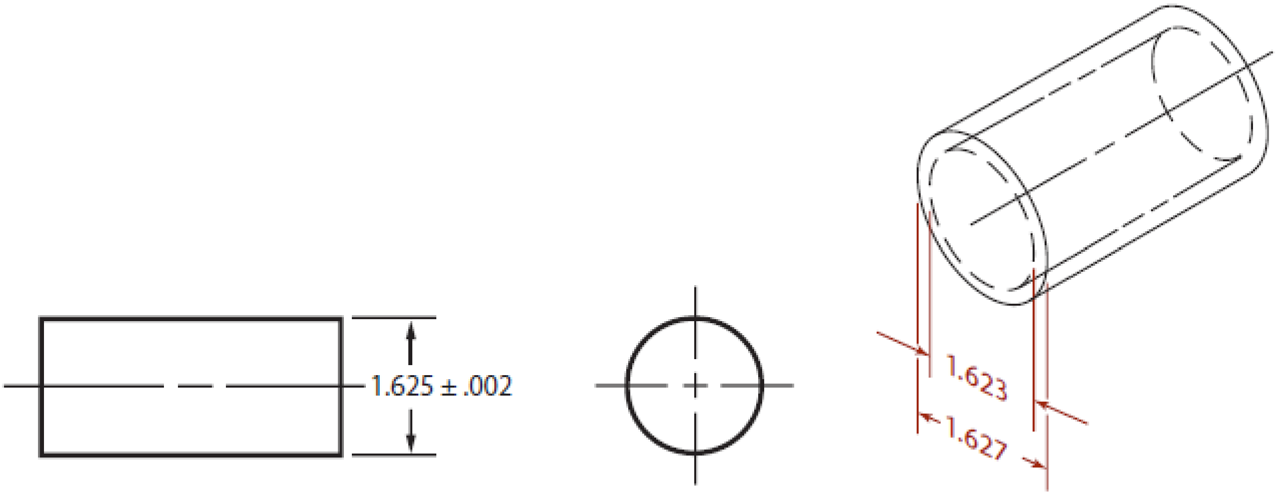
Every measurement has a tolerance, meaning that the measurement is acceptable within a specific range. Tolerance can be quite small depending on the part requirements.

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

Tolerance

Tolerance is the total amount a specific dimension is permitted to vary. Tolerances are specified so that any two mating parts will fit together. To keep part cost low, specify a tolerance as large as possible that still permits satisfactory function of the part. Tolerance can be stated several different ways on a drawing.

One method of providing a tolerance is to specify the dimension and give a plus or minus range after it. Figure A below shows an example of a bilateral tolerance. A dimension given as 1.625 ± 0.002 means that the manufactured part may be 1.627 or 1.623 or anywhere between these maximum and minimum limit dimensions, as shown in Figure B below. The tolerance (the total amount the actual part feature is allowed to vary from what is specified) is 0.004.



A. Direct Limits Used to Specify a Bilateral Tolerance B. Upper and Lower Limits of Dimension

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.

Self-Check -3	Written Test

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

Say true or false

1. Measurements are always perfect.
2. A machinist must understand the basic principles of measurement error and assume the proper responsibility in the selection, calibration, and application of measuring instruments.
3. Tolerance is the total amount a specific dimension is permitted to vary.
4. Part designers are aware of measurement error.

Operation Sheet 1	Checking /measuring work-piece

Reading Metric Vernier Caliper

Step 1 – Look at the main scale

When taking a measurement, you should first read the value on the main scale. On a metric vernier caliper, this will be given in millimetres (mm).

The smallest value that can read from the main scale is 1mm (indicated by a single increment).

The value on the main scale is the number immediately to the left of the 0 marker of the vernier scale.

Step 2 – Look at the vernier scale

Next, look at the vernier scale. The vernier scale of an metric vernier caliper has a measuring range of 1 mm. In the example we will be looking at, the vernier scale is graduated in 50 increments.

Each increment represents 0.02mm. However, some vernier scales are graduated in 20 increments, with each one representing 0.05mm.

When reading the vernier scale, identify the increment that lines up most accurately with an increment on the main scale. This value will make up the second part of your measurement.

Step 3 – Add both values together

To get your total reading, add both the value from the main scale and the value from the vernier scale together.

Operation Sheet 2	Checking /measuring work-piece

Reading a metric micrometer

1. Note the number of the last main division showing above the line to the left of the thimble. Multiply that number by 1 mm.
2. If there is a half-millimeter line showing below the index line, between the whole millimeter and the thimble, then add 0.5 mm.
3. Multiply the number of the line on the thimble that coincides with the index line times 0.01.
4. Add these products.

LAP Test	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. Reading Metric Vernier Caliper

Task 2. Read a metric micrometer

List of Reference Materials

1. Frederick E. Giesecke, Shawna Lockhart, Marla Goodman, Cindy M. Johnson (2016).Technical Drawing with Engineering Graphics (15th edition) Prentice Hall
   J.T. Black, Ronald A. Kohser (2011)DeGarmo'ss Materials and Processes in Manufacturing (11th edition) Wiley
2. John R. Walker (2000). Machining Fundamentals From Basic to Advanced Techniques (7th edition) Goodheart-Willcox Company
3. John R. Walker, Bob Dixon. (2018). Machining Fundamentals (10th edition) Goodheart-Willcox
4. K. G. Swift, J. D. Booker (2013) Manufacturing Process Selection Handbook Butterworth-Heinemann
5. Kok, Chi-Wah_ Schmid, Steven R. Kalpakjian, Serope. (2010). Manufacturing processes for engineering materials (6th edition) Pearson-Prentice Hall
6. Meyers, Arthur R._ Slattery, Thomas J. (2001) Basic Machining Reference Handbook (2nd edition) Industrial Press
7. Michael Fitzpatrick. (2013). Machining and CNC Technology (3rd edition) McGraw-Hill Higher Education
8. Mikell P. Groover (2012). Fundamentals of Modern Manufacturing Materials, Processes, and Systems (5th edition) Wiley
9. Peter J. Hoffman, Eric S. Hopewell, Brian Janes. (2014). Precision Machining Technology (2nd edition) Cengage Learning
10. Richard R. Kibbe, Warren T White, Roland O. Meyer, Kelly Curran, Jon Stenerson. (2014). Machine Tool Practices (10th edition) Pearson
11. Steve F. Krar, Arthur Gill, Peter Smid. (2010). Technology of Machine Tools (7th edition) McGraw-Hill Education
12. http://www.barringer1.com/anvil_files/anvil02_files/anvil02.pdf
13. https://www.youtube.com/watch?v=fUQnHk_LRz0
14. David P. Madsen & David A. Madsen (2011). Engineering Drawing and design (5th edition) Delmar Publishers Inc.
15. Technical Graphics Communications (4th edition) McGraw-Hill Higher Education
   James Leake, Jacob L. Borgerson (2012). Engineering Design Graphics Sketching, Modeling, and Visualization (2nd edition) Wiley.
16. Peter J. Hoffman, Eric S. Hopewell, Brian Janes. (2014). Precision Machining Technology (2nd edition) Cengage Learning
17. Richard R. Kibbe, Warren T White, Roland O. Meyer, Kelly Curran, Jon Stenerson. (2014). Machine Tool Practices (10th edition) Pearson

The trainers who developed the learning guide

No	Name	Qualification	Region	E-mail
1	Wakweya Tolera	A	Oromia	Wakweya.tolera@yahoo.com
2	Mohammed Abdela	A	Dire Dewa	Mahammedabdella49@gmail.com
3	Waliyi Bena	A	Oromia	waliyibena@gmail.com
4	Elias Zemenu	A	Somali	Elijahze26@gmail.com
5	Gurmu Mulatu	A	Adis Ababa	gurmumuulaatuu@gmail.com
6	Muluneh Negussie	B	Amhara	Muluneh3000@gmail.com
7	Teshome W/Yuhanis	A	SNNPR	Tesho1921@gmail.com
8	Solomon Aderaw	B	Amhara	Solo19@gmail.com