INTERNATIONAL INSTITUTE OF TECHNOLOGY & MANAGEMENT, MURTHAL SONEPAT
E-NOTES , Subject : Engineering Graphics, Course: Diploma , Branch- Common to all Sem. 1st year ,
Chapter Name: Development of surface
(Prepared By: Ms. Promila , Assistant Professor , MED)

INTRODUCTION

In industrial world, an engineer is frequently confronted with problems where the development of surfaces of an object has to be made to help him to go ahead with the design and manufacturing processes. For example, in sheet metal work, it plays a vital role, thus enabling a mechanic to cut proper size of the plate from the development and then to fold at proper places to form the desired objects, namely, boilers, boxes, buckets, packing boxes, chimneys, hoppers, air-conditioning ducts etc.

"**The development of surface of an object means the unrolling and unfolding of all surfaces of the object on a plane.**"

"If the surface of a solid is laid out on a plain surface, the shape thus obtained is called the development of that solid." In other words, the development of a solid is the shape of a plain sheet that by proper folding could be converted into the shape of the concerned solid.

Importance of Development:

Knowledge of development is very useful in **sheet metal work**, **construction of storage vessels**, **chemical vessels**, **boilers**, and **chimneys**. Such vessels are manufactured from plates that are cut according to these developments and then properly bend into desired shaped. The joints are then **welded or riveted**.

Principle of Development:

Every line on the development should show the true length of the corresponding line on the surface which is developed.

Methods of Development:

(a) Parallel-line development
(b) Radial-line development
(c) Triangulation development
(d) Approximate development

- **Parallel-line Method:**

It is used for developing prisms and single curved surfaces like cylinders, in which all the edges/generation of lateral surfaces are parallel in each other.

- **Radial-line Method:**

It is employed for pyramids and single curved surfaces like cones in which the apex is taken as centre and the slant edge or generator as radius of its development.

- **Triangulation Method:**

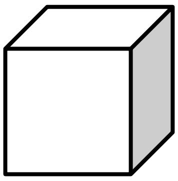
It is used for developing transition pieces.

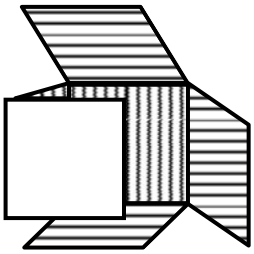
- **Approximate Method:**

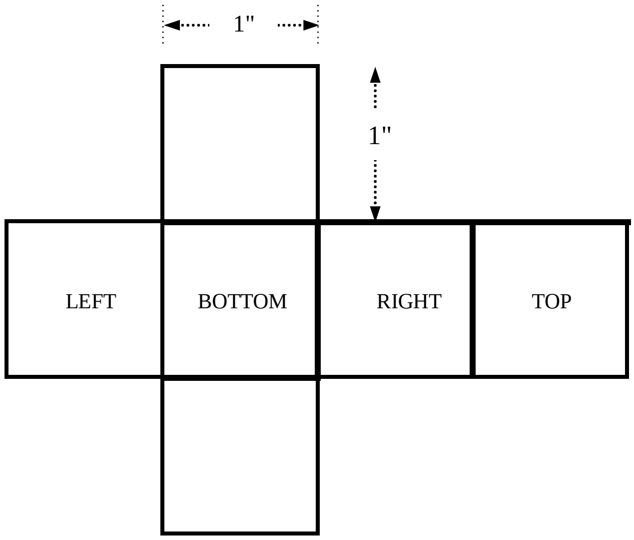
It is employed for double curved surfaces like spheres, as they are theoretically not possible to develop. The surface of the sphere is developed by approximate method. When the surface is cut by a series of cutting planes, the cut surfaces is called a zone.

DEVELOPMENT OF SECTIONS

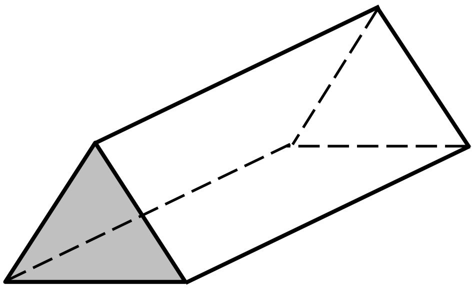
Develop the surfaces of the cube of 1"

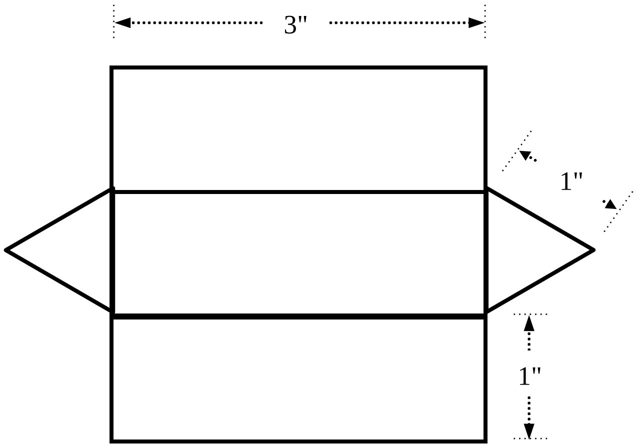




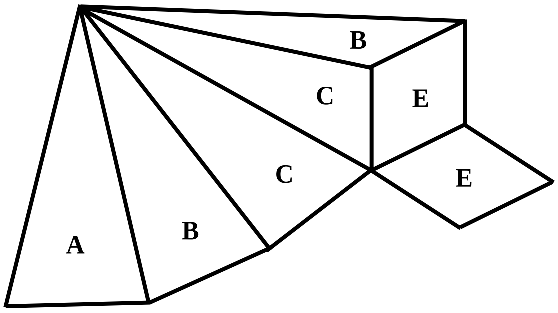


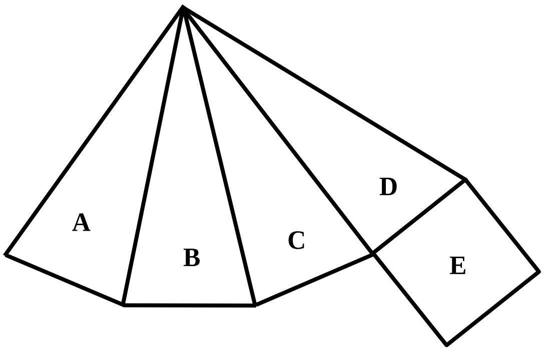
Develop the surfaces of a triangular prism



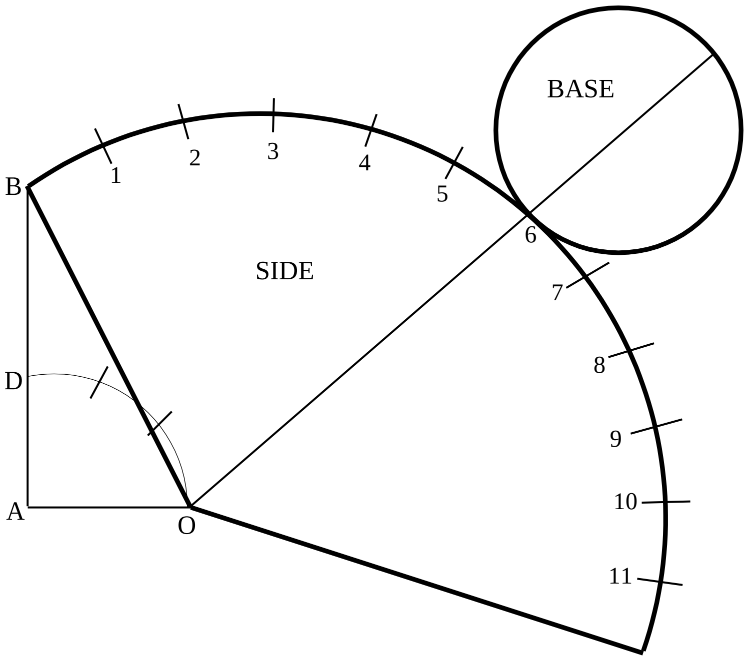


Develop the surfaces of a pyramid



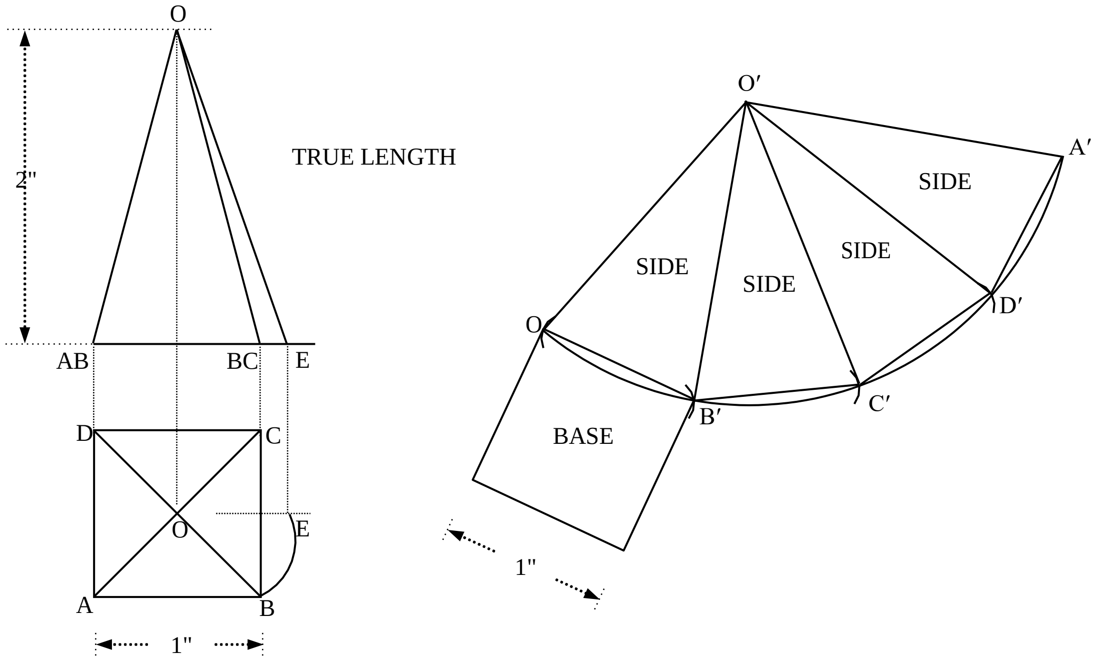


- **Draw the development of a cone of diameter 1.5" and inclined height of 2"**



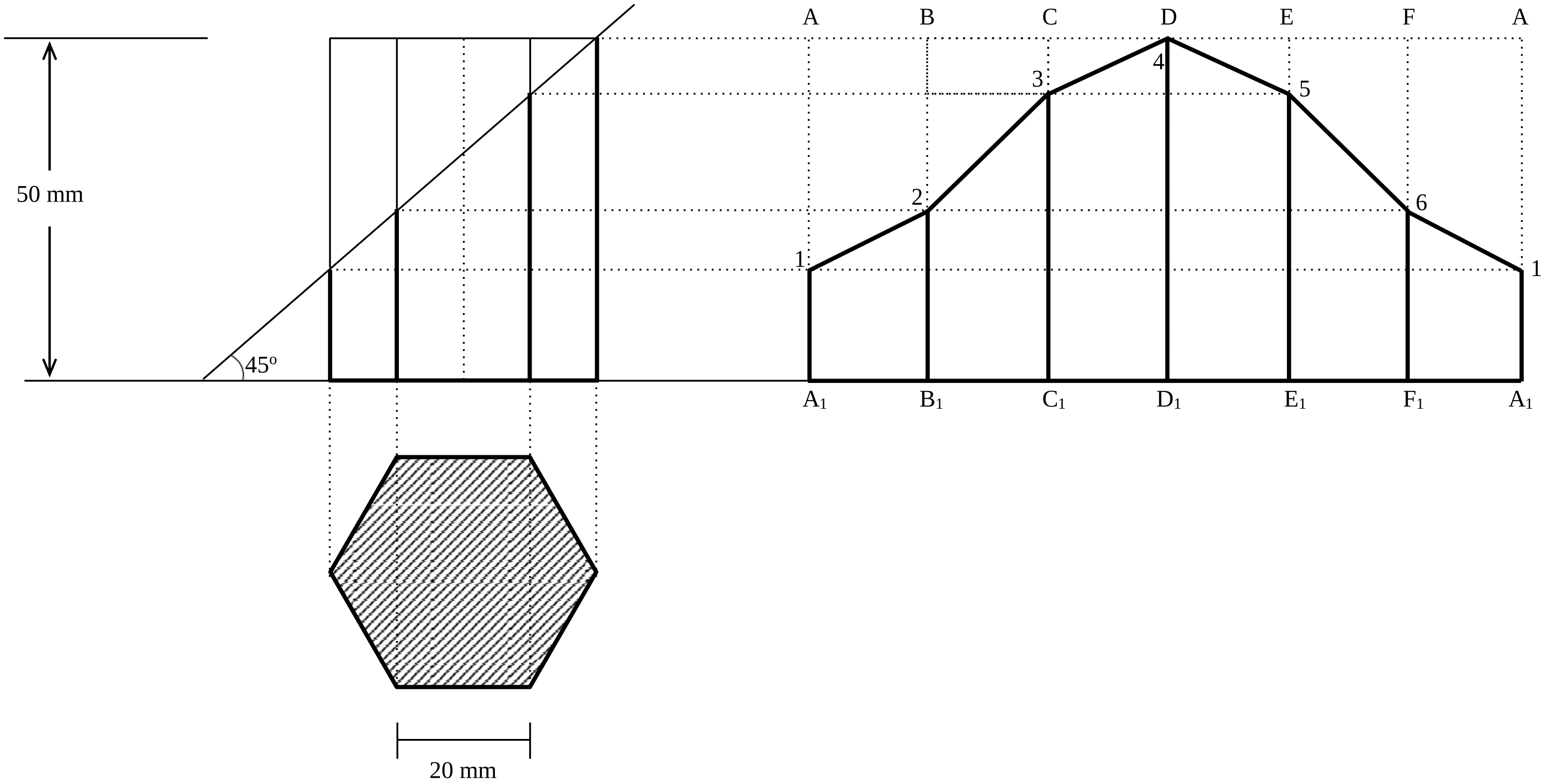
AO = Radius of the base = 3/4"
BO = Inclined height of cone = 2"

- **Draw the development of a square pyramid from its plan and front elevation which stands vertically on its base on H.P with one edge of the base parallel to V.P.**



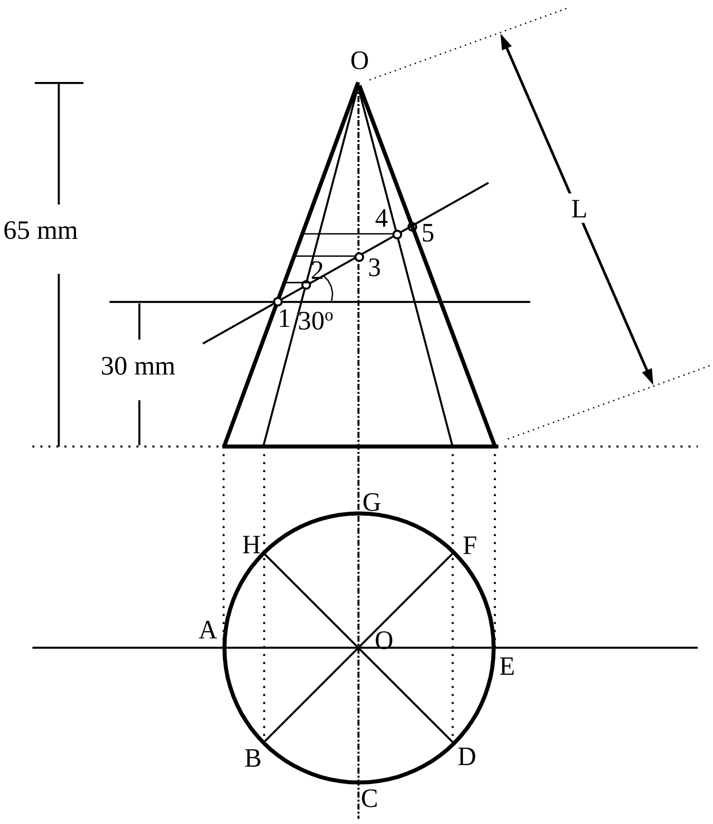
Problem:

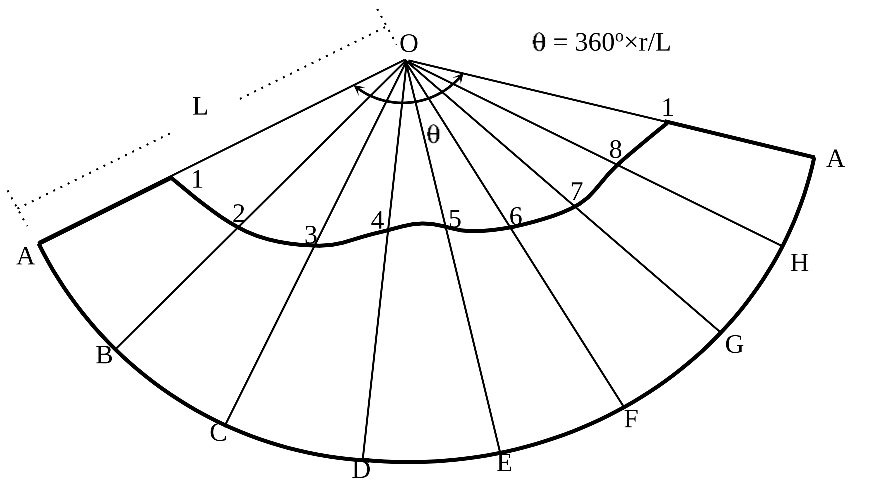
A hexagonal prism, edge of base 20 mm and axis 50 mm long, rests with its base on H.P such that one of its rectangular faces is parallel to V.P. It is cut by a plane perpendicular to V.P, inclined at 45º to H.P and passing through the right corner of the top face of the prism. Draw the sectional top view and develop the lateral surface of the truncated prism.



Problem:

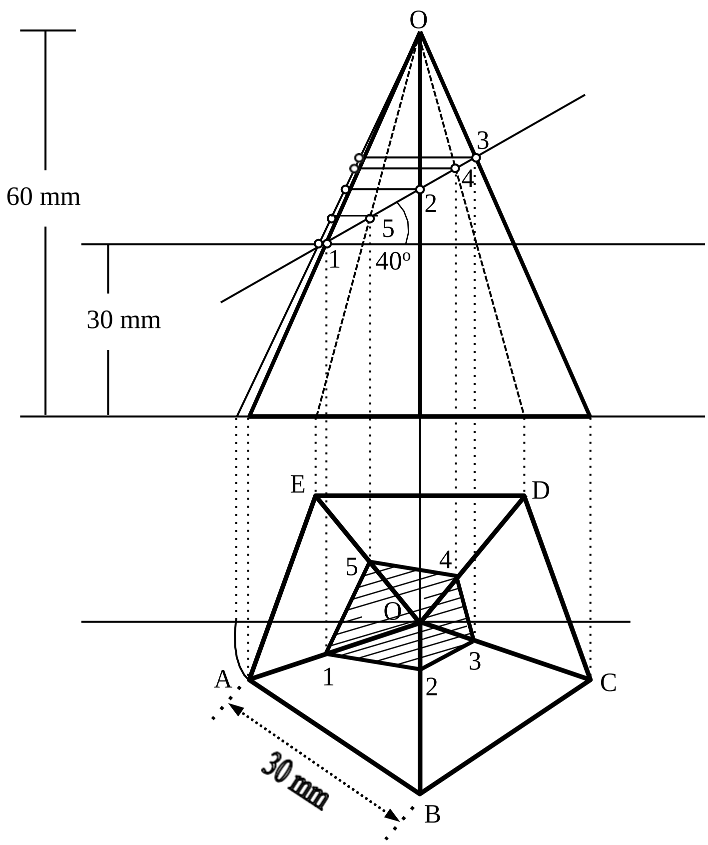
A cone of base 50 mm diameter and height 65 mm rests with its base on H.P. A section plane perpendicular to V.P and inclined at 30º to H.P bisects the axis of the cone. Draw the development of the lateral surface of the truncated cone.

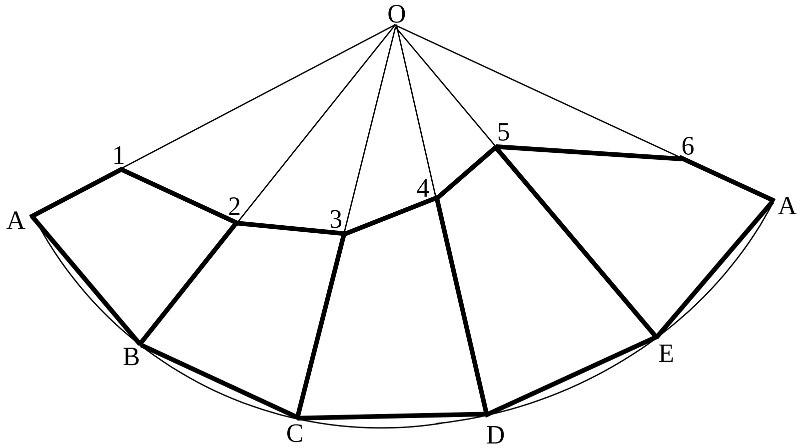




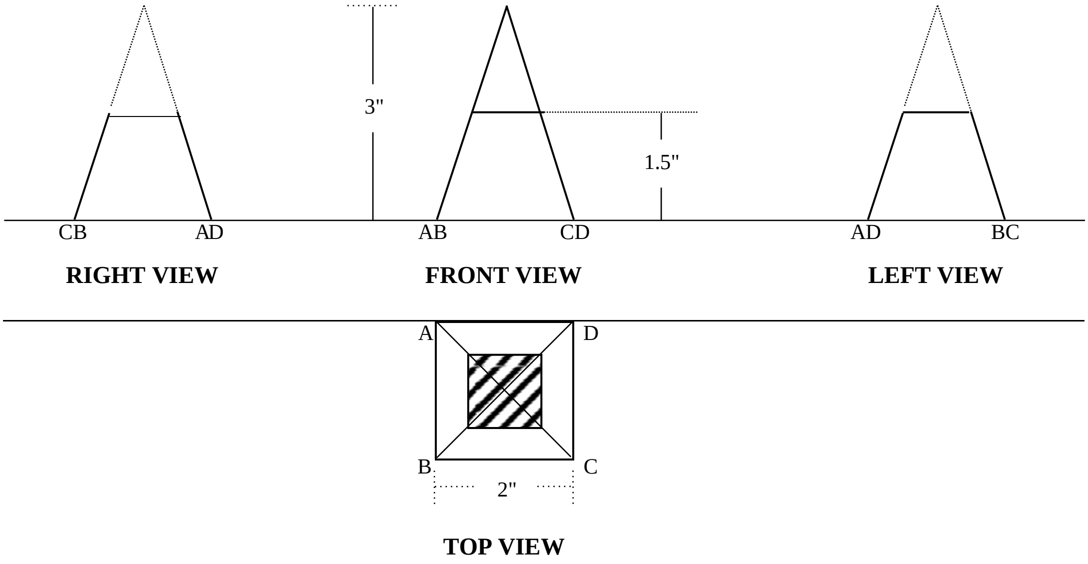
Problem:

A pentagonal pyramid, side of base 30 mm and height 60 mm, stands with its base on H.P and an edge of the base is parallel to V.P. It is cut by a plane perpendicular to V.P, inclined at 40º to H.P and passing through a point on the axis, 32 m above the base. Draw the sectional top view and develop the lateral surface of the truncated pyramid.

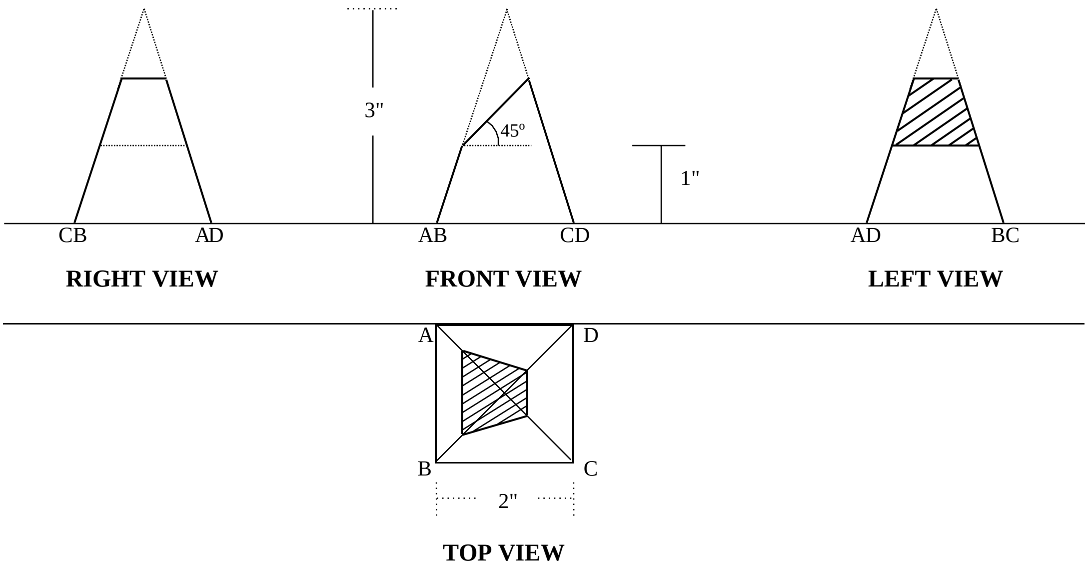




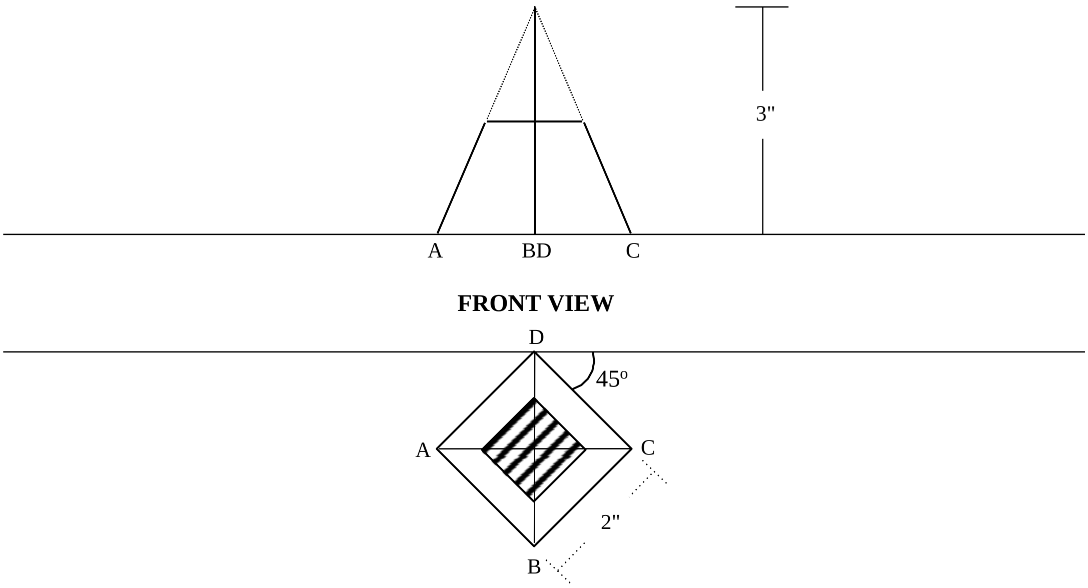
- Draw the development of a square pyramid from its plan and front elevation which stands vertically on its base on H.P with one edge of the base parallel to V.P.



- Draw the development of a square pyramid from its plan and front elevation which stands vertically on its base on H.P with one edge of the base parallel to V.P.



- Draw the development of a square pyramid from its plan and front elevation which stands vertically on its base on H.P with one edge of the base parallel to V.P.



FRONT VIEW

TOP VIEW

KEYS, COTTERS

INTROOUSTION

Keys, cotters and pin joints discussed in this chapter are some examples of removable (temporary) fasteners. Assembly and removal of these joints are easy as they are simple in shape. The standard proportions of these joints are given in the figures.

KEYS

Keys are machine elements used to prevent relative rotational movement between a shaft and the parts mounted on it, such as pulleys, gears, wheels, couplings, etc. Figure shows the parts of a keyed joint and its assembly.

For making the joint, grooves or keyways are cut on the surface of the shaft and in the hub of the part to be mounted. After positioning the part on the shaft such that, both the keyways are properly aligned, the key is driven from the end, resulting in a firm joint.

For mounting a part at any intermediate location on the shaft, first the key is firmly placed in the keyway of the shaft and then the part to be mounted is slid from one end of the shaft, till it is fully engaged with the key.

Keys are classified into three types, viz., saddle keys, sunk keys and round keys.

Saddle Keys

These are taper keys, with uniform width but tapering in thickness on the upper side. The magnitude of the taper provided is 1:100. These are made in two forms: hollow and flat.

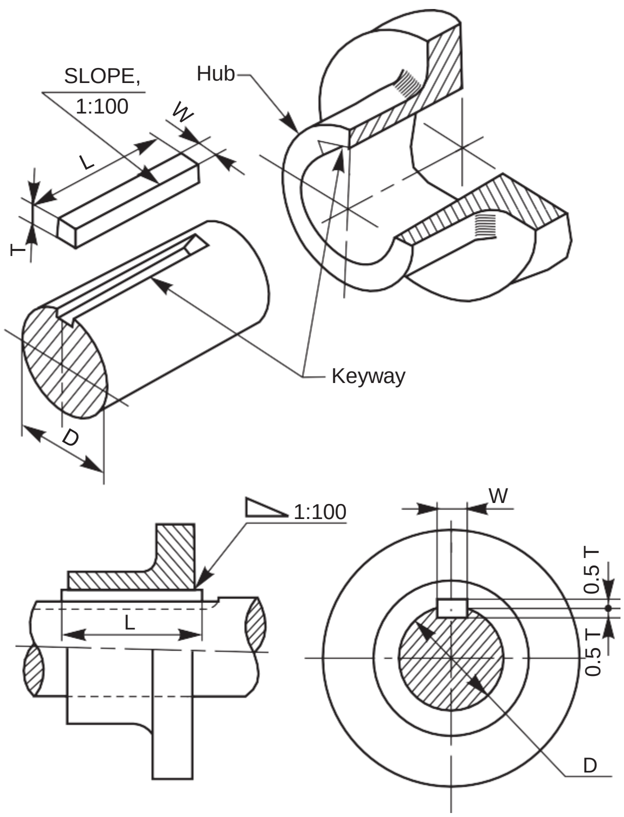


Fig. Keyed joint

Hollow Saddle Key

A hollow saddle key has a concave shaped bottom to suit the curved surface of the shaft, on which it is used. A keyway is made in the hub of the mounting, with a tapered bottom surface. When a hollow saddle key is fitted in position, the relative rotation between the shaft and the mounting is prevented due to the friction between the shaft and key (Fig.).

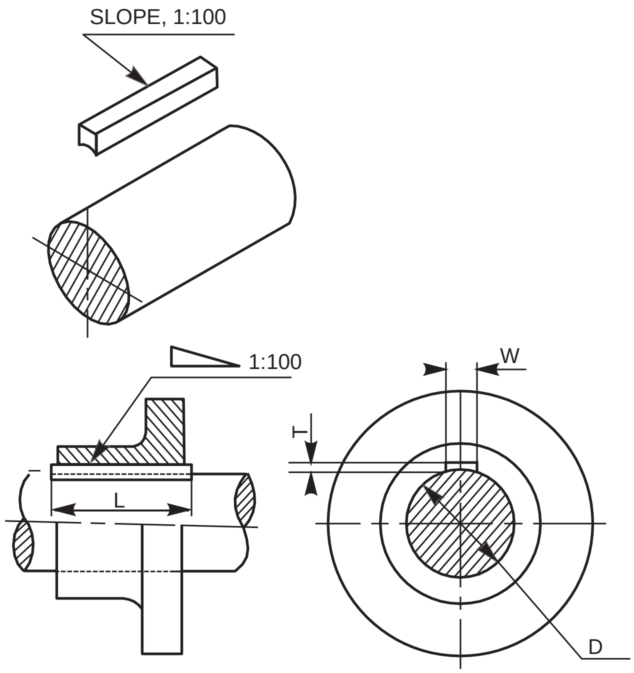


Fig. Hollow saddle key

Flat Saddle Key

It is similar to the hollow saddle key, except that the bottom surface of it is flat. Apart from the tapered keyway in the hub of the mounting, a flat surface provided on the shaft is used to fit this key in position (Fig).

The two types of saddle keys discussed above are suitable for light duty only. However, the flat one is slightly superior compared to the hollow type. Saddle keys are liable to slip around the shaft when used under heavy loads.

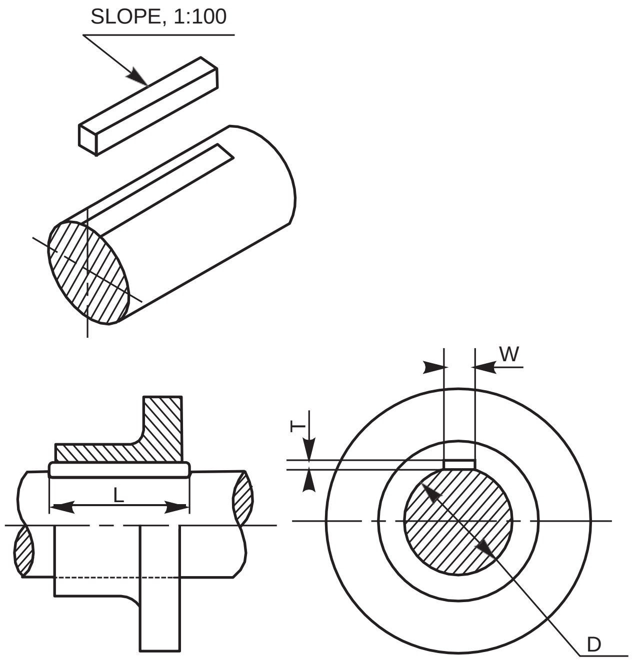


Fig. Flat saddle key

Sunk Keys

These are the standard forms of keys used in practice, and may be either square or rectangular in cross-section. The end may be squared or rounded. Generally, half the thickness of the key fits into the shaft keyway and the remaining half in the hub keyway. These keys are used for heavy duty, as the fit between the key and the shaft is positive.

Sunk keys may be classified as:

- taper keys,
- parallel or feather keys and
- woodruff keys.

Taper Sunk Keys

These keys are square or rectangular in cross-section, uniform in width but tapered in thickness. The bottom surface of the key is straight and the top surface is tapered, the magnitude of the taper being 1:100. Hence, the keyway in the shaft is parallel to the axis and the hub keyway is tapered (Fig.).

A tapered sunk key may be removed by driving it out from the exposed small end. If this end is not accessible, the bigger end of the key is provided with a head called gib. Figure 6.4 shows the application of a key with a gib head. Following are the proportions for a gib head:

If D is the diameter of the shaft, then,

Width of key, W	= 0.25 D + 2 mm
Thickness of key, T	= 0.67 W (at the thicker end)
Standard taper	= 1:100
Height of head, H	= 1.75 T
Width of head, B	= 1.5 T

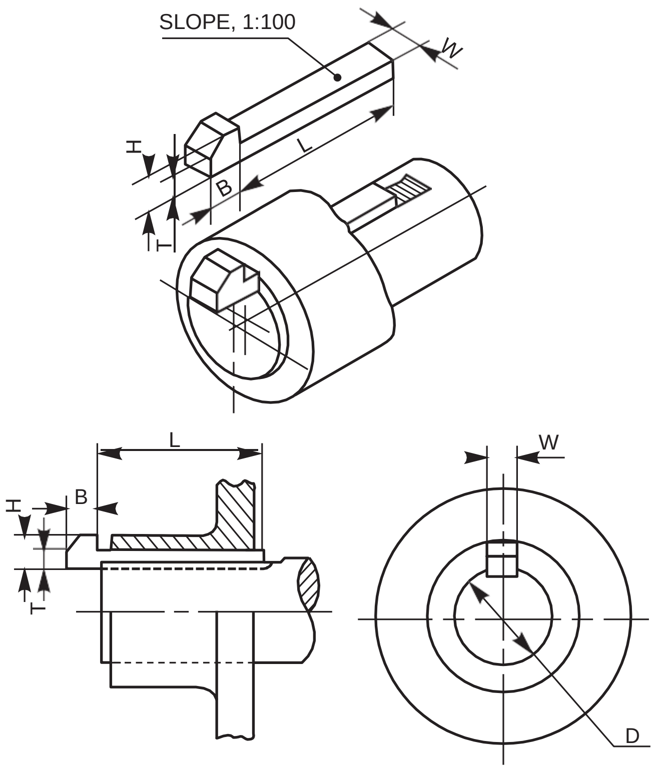


Fig. 6.4 Key with gib head

Table 6.1 gives the dimensions of taper sunk keys, for various shaft sizes.

Table 6.1 Proportions of taper sunk keys for various shaft sizes (contd.)

Shaft diameter (mm)			
Over	*Upto and including*	*Width, W (mm)*	*Thickness, T (average value) (mm)*
6	8	2	2
8	10	3	3
10	12	4	4

Table 6.1 Proportions of taper sunk keys for various shaft sizes

Shaft diameter (mm)		*Width, W (mm)*	*Thickness, T (average value) (mm)*
Over	*Upto and Including*		
12	17	5	5
17	22	6	6
22	30	8	7
30	38	10	8
38	44	12	8
44	50	14	9
50	58	16	10
58	65	18	11
65	75	20	12
75	85	22	14
85	95	25	14
95	110	28	16

Parallel or Feather Keys

A parallel or feather key is a sunk key, uniform in width and thickness as well. These keys are used when the parts (gears, clutches, etc.) mounted are required to slide along the shaft; permitting relative axial movement. To achieve this, a clearance fit must exist between the key and the keyway in which it slides.

The feather key may be fitted into the keyway provided on the shaft by two or more screws (Fig. 6.5) or into the hub of the mounting (Fig. 6.6). As seen from Fig. 6.6, these keys are of three types: (*i*) peg feather key, (*ii*) single headed feather key and (*iii*) double headed feather key.

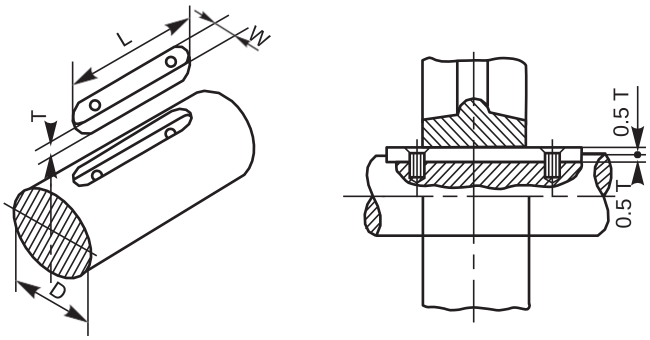


Fig. Parallel sunk key

Peg Feather Key

In this key, a projection known as peg is provided at the middle of the key. The peg fits into a hole in the hub of the sliding member (Fig. 6.6 *a*). Once placed in a position, the key and the mounting move axially as one unit.

Single Headed Feather Key

In this, the key is provided with a head at one end. The head is screwed to the hub of the part mounted on the shaft (Fig. 6.6 *b*).

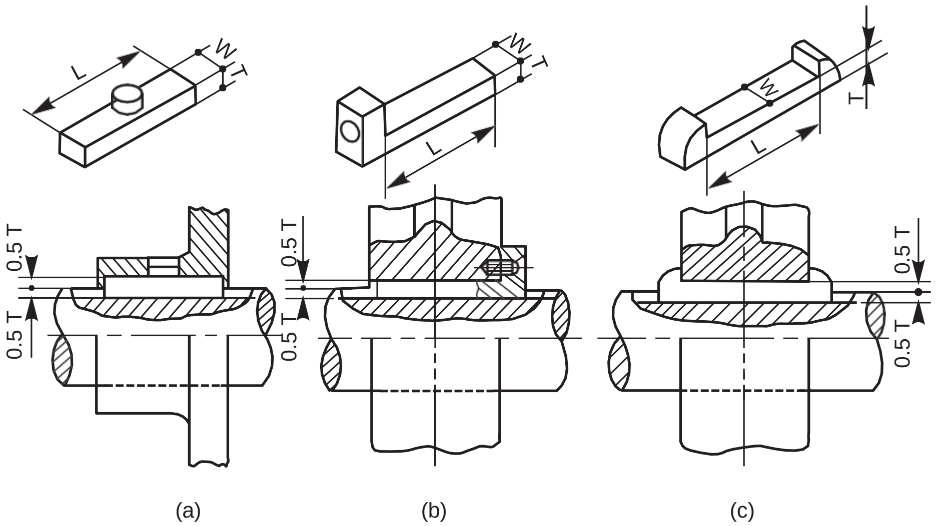


Fig. 6.6 Feather keys

Double Headed Feather Key

In this, the key is provided with heads on both ends. These heads prevent the axial movement of the key in the hub. Here too, once placed in position, the key and the mounting move as one unit (Fig.).

Splines

Splines are keys made integral with the shaft, by cutting equi-spaced grooves of uniform cross-section. The shaft with splines is called a splined shaft. The splines on the shaft, fit into the corresponding recesses in the hub of the mounting, with a sliding fit, providing a positive drive and at the same time permitting the latter to move axially along the shaft (Fig. 6.7).

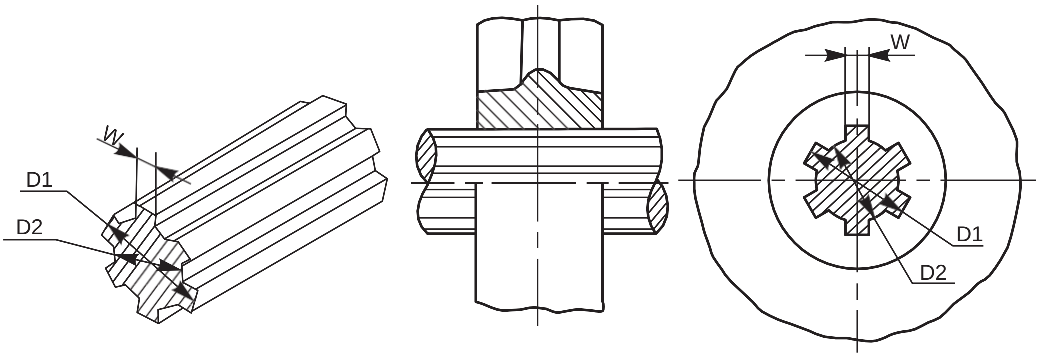


Fig. 6.7 Splined shaft and hub

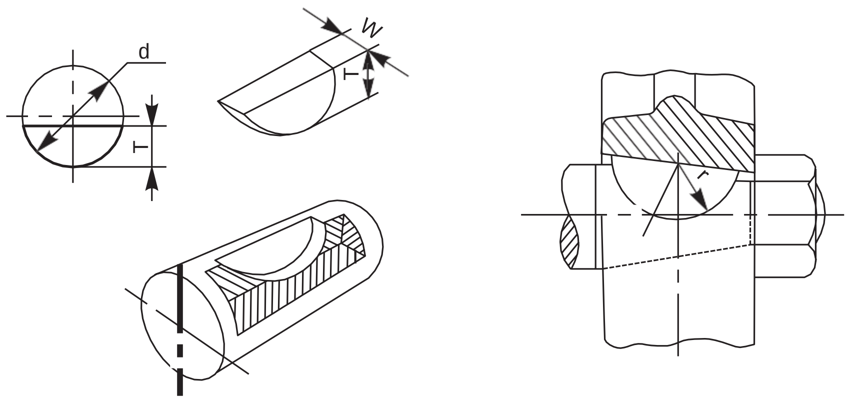
Table 6.2 gives the proportions for splined shafts of various sizes.

Table 6.2 Proportions for splined shafts of various sizes

Nominal (major) diameter, D_1 (mm)	*Number of splines*	*Minor (root) diameter, D_2 (mm)*	*Width of spline, W (mm)*
14	6	11	3
16	6	13	3.5
20	6	16	4
22	6	18	5
25	6	21	5
28	6	23	6
32	6	26	6
34	6	28	6
38	8	32	7
42	8	36	7
48	8	42	8
54	8	46	9
60	8	52	10
65	8	56	10
72	8	62	12
82	10	72	12
92	10	82	12
102	10	92	14

Woodruff Key

It is a sunk key, in the form of a segment of a circular disc of uniform thickness . As the bottom surface of the key is circular, the keyway in the shaft is in the form of a circular



(a) (b)

Fig. Woodruff key

recess to the same curvature as the key. A keyway is made in the hub of the mounting, in the usual manner. Woodruff key is mainly used on tapered shafts of machine tools and automobiles. Once placed in position, the key tilts and aligns itself on the tapered shaft (Fig. 6.8 *b*). The following are the proportions of woodruff keys:

If D is the diameter of the shaft,

Thickness of key, W = 0.25 D

Diameter of key, *d* = 3 W

Height of key, T = 1.35 W

Depth of the keyway in the hub, T_1 = 0.5 W + 0.1 mm

Depth of keyway in shaft, T_2 = 0.85 W

Round Keys

Round keys are of circular cross-section, usually tapered (1:50) along the length. A round key fits in the hole drilled partly in the shaft and partly in the hub (Fig. 6.9). The mean diameter of the pin may be taken as 0.25 D, where D is shaft diameter. Round keys are generally used for light duty, where the loads are not considerable.

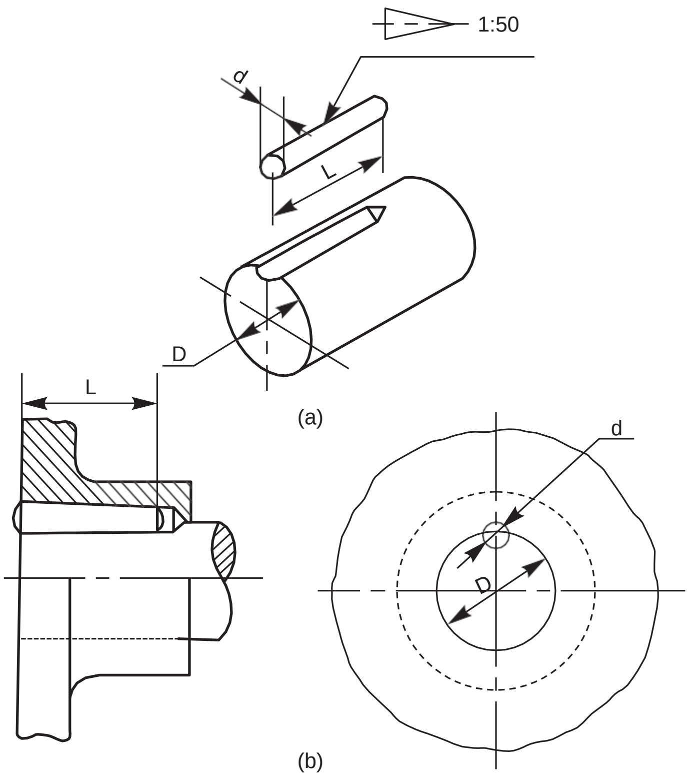


Fig. 6.9 Round key

SOTTER JOINTS

A cotter is a flat wedge shaped piece, made of steel. It is uniform in thickness but tapering in width, generally on one side; the usual taper being 1:30. The lateral (bearing) edges of the cotter and the bearing slots are generally made semi-circular instead of straight (Fig. 6.10).

This increases the bearing area and permits drilling while making the slots. The cotter is locked in position by means of a screw as shown in Fig. 6.11.

Cotter joints are used to connect two rods, subjected to tensile or compressive forces along their axes. These joints are not suitable where the members are under rotation. The following are some of the commonly used cotter joints:

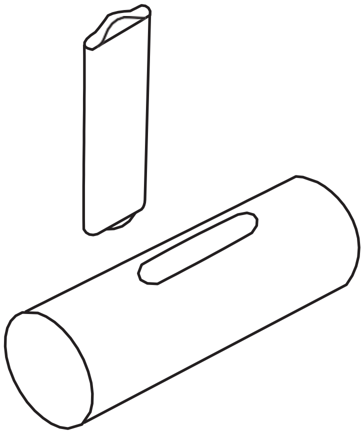


Fig. 6.10 Cotter and the bearing slot

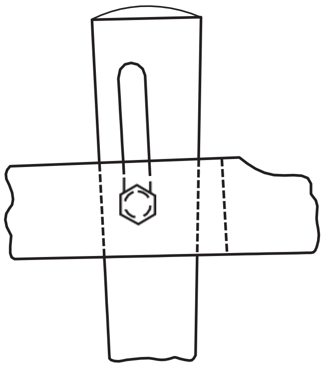


Fig. 6.11 Locking arrangement of cotter

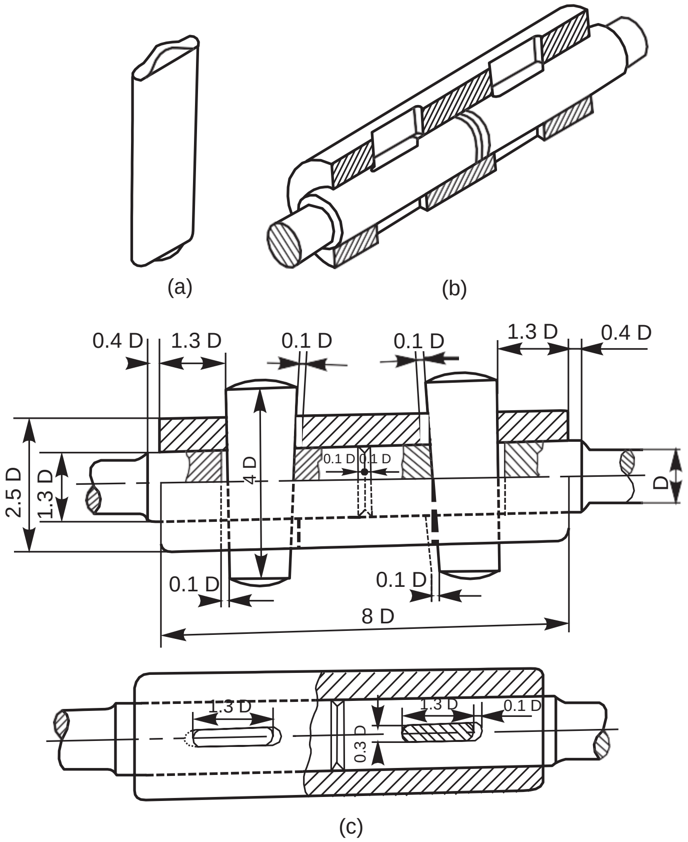


Fig. 6.12 Cotter joint with sleeve

Sotter Joint with Sleeve

This is the simplest of all cotter joints, used for fastening two circular rods. To make the joint, the rods are enlarged at their ends and slots are cut. After keeping the rods butt against each other, a sleeve with slots is placed over them. After aligning the slots properly, two cotters are driven-in through the slots, resulting in the joint (Fig. 6.12). The rod ends are enlarged to take care of the weakening effect caused by the slots.

The slots in the rods and sleeve are made slightly wider than the width of cotter. The relative positions of the slots are such, that when a cotter is driven into its position, it permits wedging action and pulls the rod into the sleeve.

Sotter Joint with Socket and Spigot Ends

This joint is also used to fasten two circular rods. In this, the rod ends are modified instead of using a sleeve. One end of the rod is formed into a socket and the other into a spigot (Fig. 6.13) and slots are cut. After aligning the socket and spigot ends, a cotter is driven-in through the slots, forming the joint.

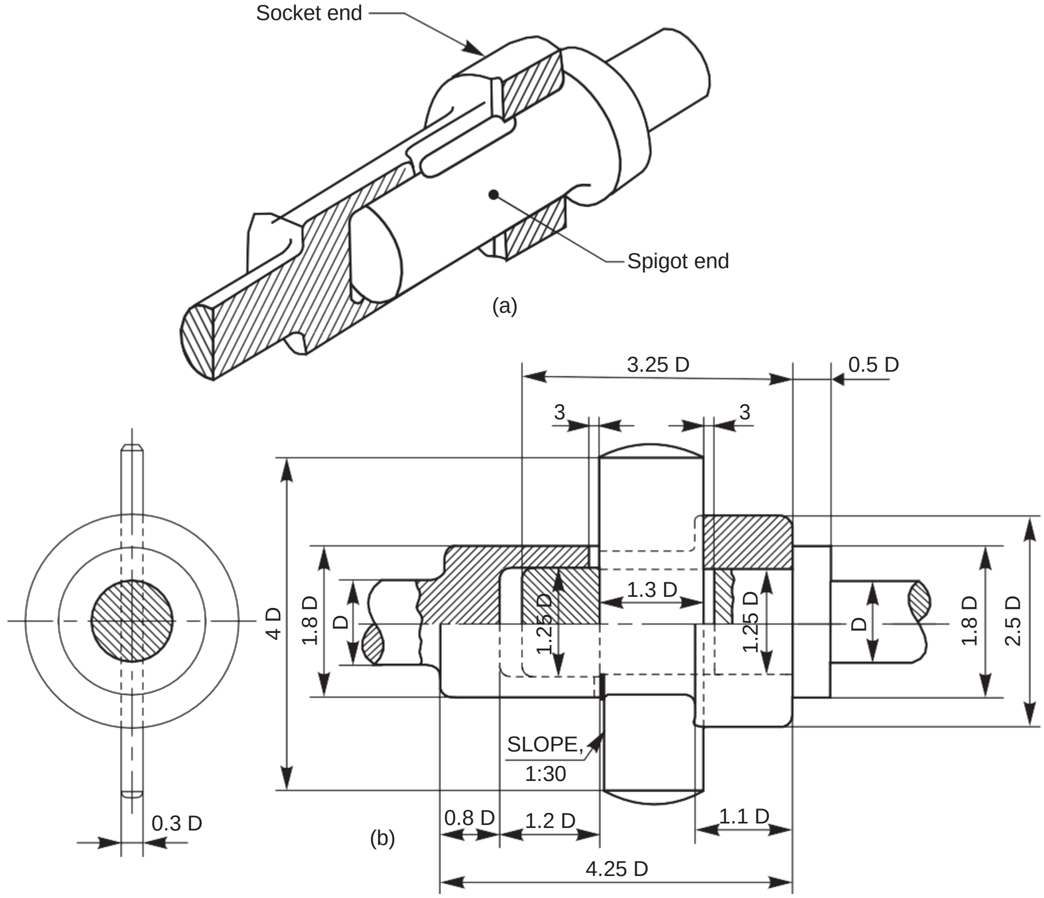


Fig. 6.13 Cotter joint with socket and spigot ends

6.3.2 Sotter Joint with a Cib

This joint is generally used to connect two rods of square or rectangular cross-section. To make the joint, one end of the rod is formed into a U-fork, into which, the end of the other rod fits in.

When a cotter is driven-in, the friction between the cotter and straps of the U-fork, causes the straps to open. This is prevented by the use of a gib.

A gib is also a wedge shaped piece of retangular cross-section with two rectangular projections called lugs. One side of the gib is tapered and the other straight. The tapered side of the gib bears against the tapered side of the cotter such that, the outer edges of the cotter and gib as a unit are parallel. This facilitates making of slots with parallel edges, unlike the tapered edges in case of ordinary cotter joint. Further, the lugs bearing against the outer surfaces of the fork, prevents the opening tendency of the straps.

Figure 6.14 shows a cotter joint with a gib. For making the joint, first the gib is placed in position and then the cotter is driven-in.

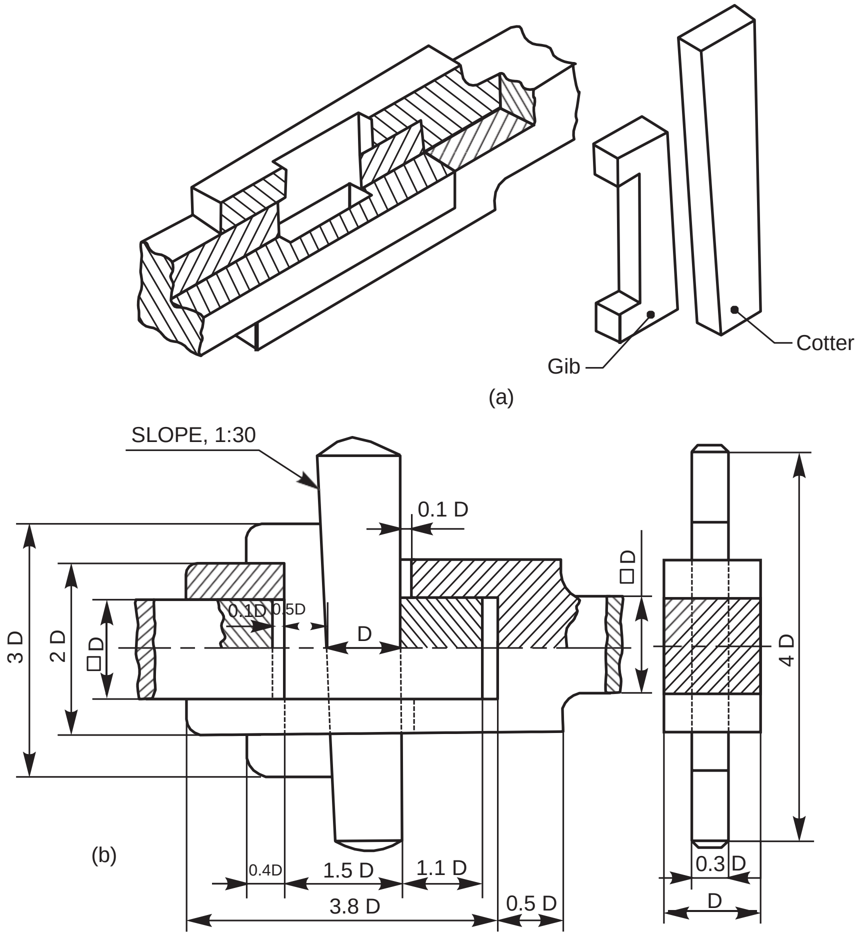


Fig. 6.14 Cotter joint with a gib

6.3 PIN JOINTS

In a pin joint, a pin is used to fasten two rods that are under the action of a tensile force; although the rods may support a compressive force if the joint is guided. Some pin joints such as universal joints, use two pins and are used to transmit power from one rotating shaft to another (the universal joint is discussed under Chapter 7). A pin joint permits a small amount of flexibility or one rod may be positioned at an angle (in the plane containing the rods) with respect to the other rod, after providing suitable guides.

Unlike in cotter joints, the pin in a pin joint is not driven-in with a force fit, but is inserted in the holes with a clearance fit. The pin is held in position, by means of a taper pin or a split pin provided at its end.

Knuckle Joint

A knuckle joint is a pin joint used to fasten two circular rods. In this joint, one end of the rod is formed into an eye and the other into a fork (double eye). For making the joint, the eye end of the rod is aligned into the fork end of the other and then the pin is inserted through the holes and held in position by means of a collar and a taper pin (Fig. 6.15). Once the joint is made, the rods are free to swivel about the cylindrical pin.

Knuckle joints are used in suspension links, air brake arrangement of locomotives, etc.

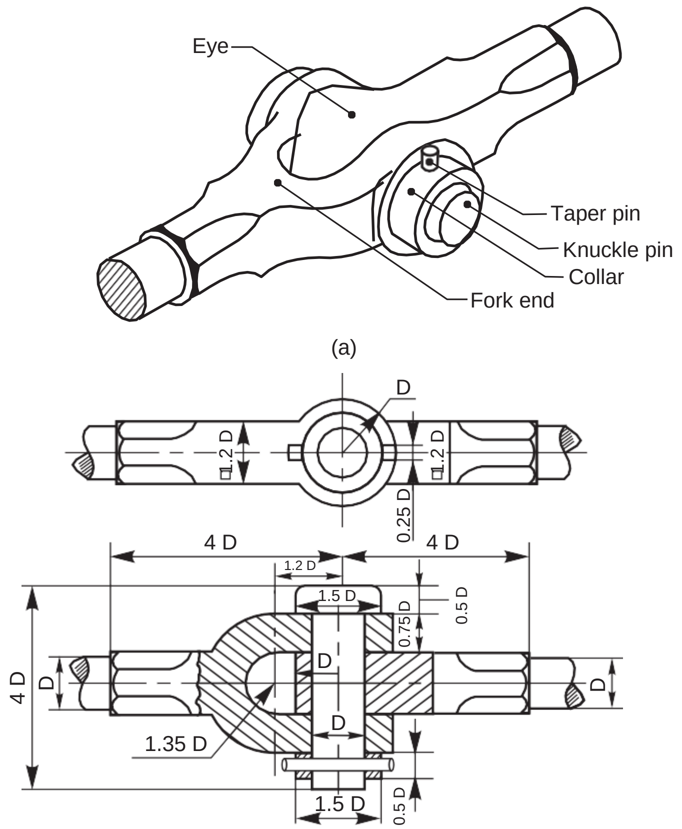


Fig. Knuckle joint

SHAFT COUPLINGS

INTRODUCTION

Shaft couplings are used to join or connect two shafts in such a way that when both the shafts rotate, they act as one unit and transmit power from one shaft to the other. Shafts to be connected or coupled may have collinear axes, intersecting axes or parallel axes at a small distance. Based on the requirements, the shaft couplings are classified as:

(*i*) rigid couplings,
(i) flexible couplings,
(ii) (*iii*) loose or dis-engaging couplings and
(iii) (*iv*) non-aligned couplings.

RIGID COUPLINGS

Rigid shaft couplings are used for connecting shafts having collinear axes. These are further sub-classified into muff or sleeve couplings and flanged couplings.

Sleeve or Muff Couplings

This is the simplest of all couplings. It consists of a sleeve called muff, generally made of cast iron, which is fitted over the ends of the shafts to be connected. After properly aligning the keyways in the shafts and sleeve, a sunk key is driven-in; thus making the coupling. Instead of a single key running the entire length of the sleeve, it is desirable to use two keys, which may be inserted from the outer ends of the sleeve; thus overcoming the possible mis-alignment between the keyways. The following are the types of muff couplings:

Butt-muff Coupling

In this, the ends of the two shafts to be coupled butt against each other, with the sleeve keyed to them,

Hall-lap muff Coupling

In this, the ends of the shafts overlap each other for a short length. The taper provided in the overlap prevents the axial movement of the shafts. Here too, after placing the muff over the overlapping ends of the shafts, a saddle key(s) is(are) used to make the coupling (Fig. 7.2).

Split-muff Coupling

In this, the muff is split into two halves and are recessed. A number of bolts and nuts are used to connect the muff halves and the recesses provided accommodate the bolt heads and nuts.

For making the coupling, a sunk key is first placed in position and then the muff halves are joined by bolts and nuts . This type of coupling is used for heavy duty work, since both the key and friction grip transmit the power (torque).

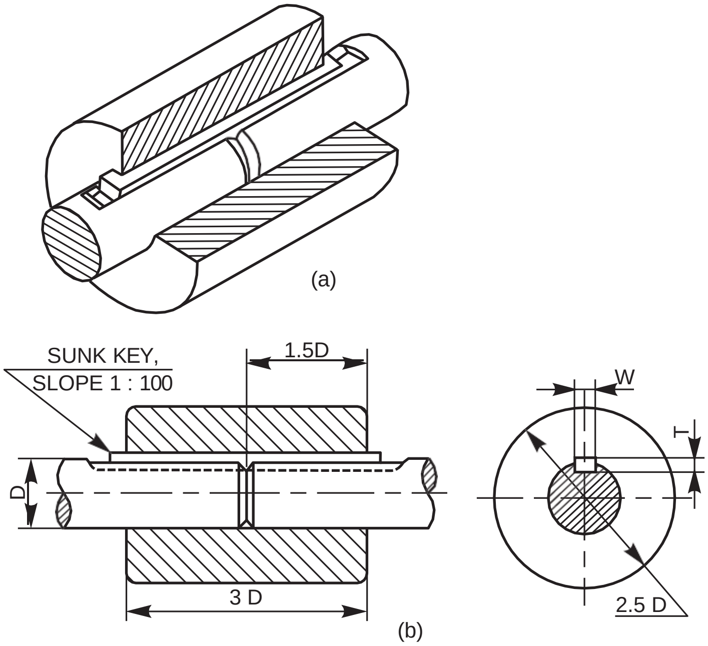


Fig. 7.1 Butt-muff coupling

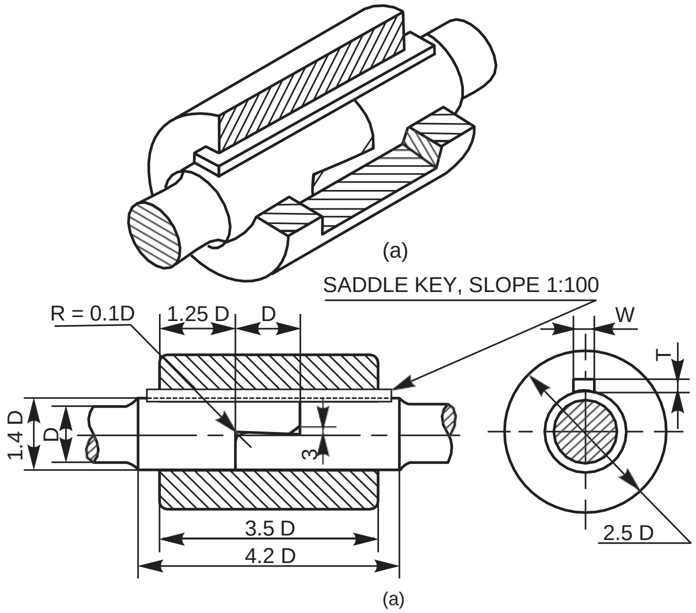


Fig. 7.2 Half-lap muff coupling

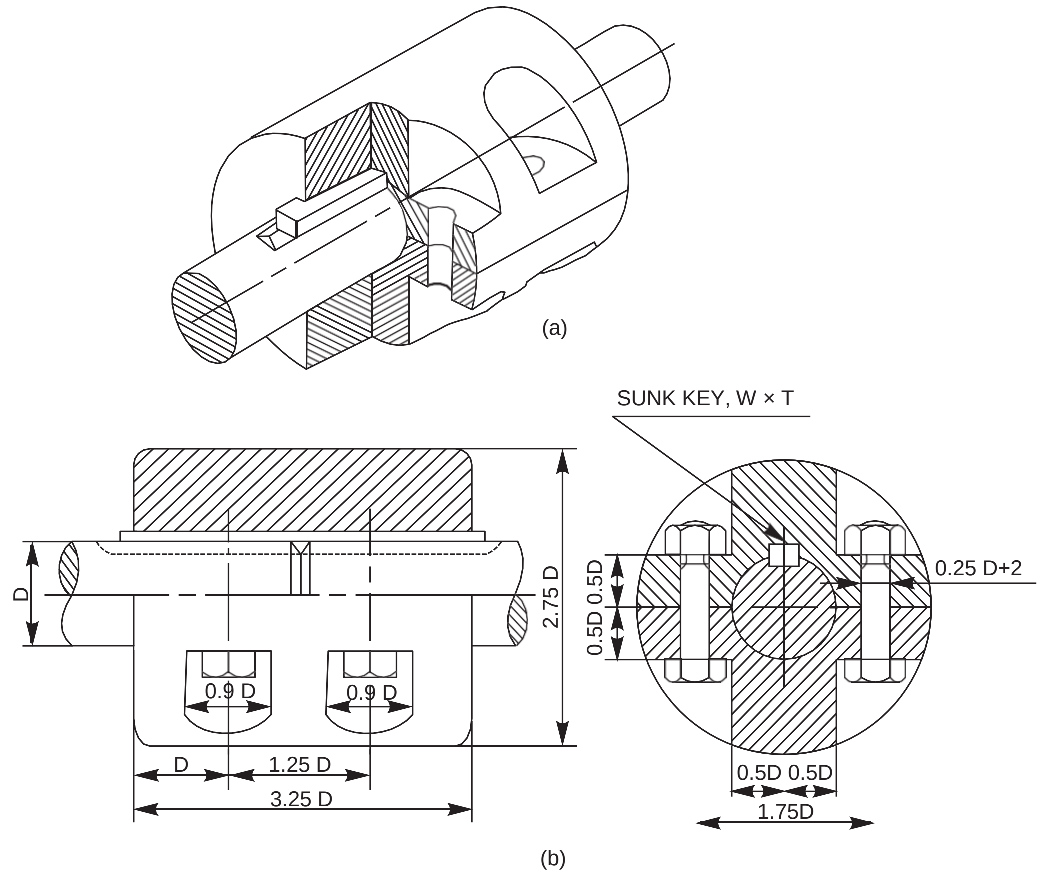


Fig. 7.3 Split-muff coupling

Flanged Couplings

These are the standard forms of couplings, most extensively used. In a flanged coupling, flanges are either fitted or provided at the ends of shafts. The flanges are fastened together by means of a number of bolts and nuts. The number and size of the bolts depend upon the power to be transmitted and hence, the shaft diameter.

Flanged coupling with detachable Flanges

In this, two flanges are keyed, one at the end of each shaft, by means of sunk keys (Fig. 7.4). For ensuring correct alignment, a cylindrical projection may be provided on one flange which fits into the corresponding recess in the other.

In the design shown in figure, the bolt heads and nuts are exposed and liable to cause injury to the workman. Hence, as a protection, the bolt heads and nuts may be covered by providing an annular projection on each flange. A flanged coupling, using these flanges is called a protected flanged coupling .

Solid Flanged Coupling

Couplings for marine or automotive propeller shafts demand greater strength and reliability. For these applications, flanges are forged integral with the shafts. The flanges are joined together by means of a number of headless taper bolts .

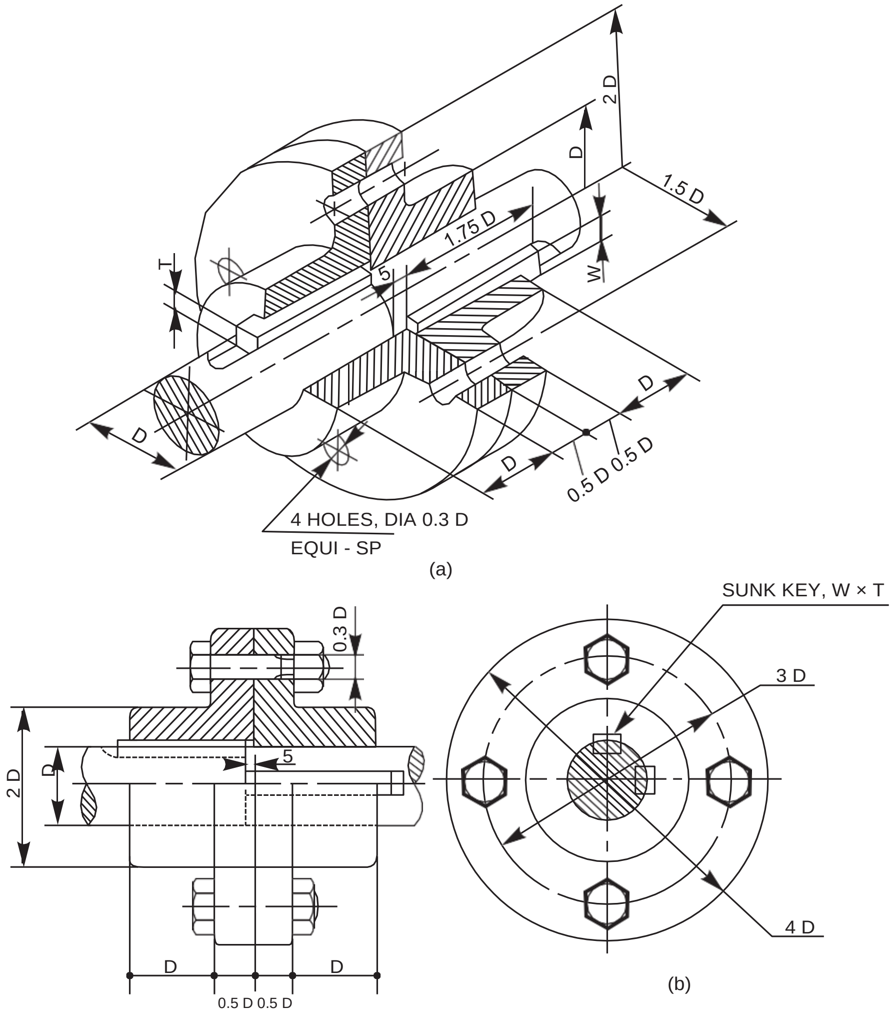


Fig. Flanged coupling

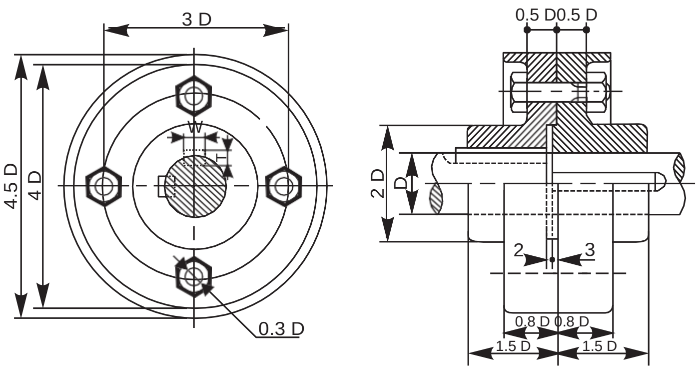


Fig. Protected flanged coupling

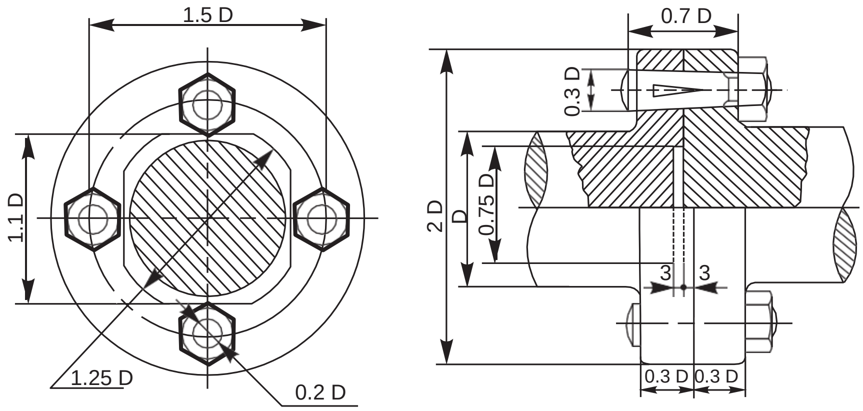


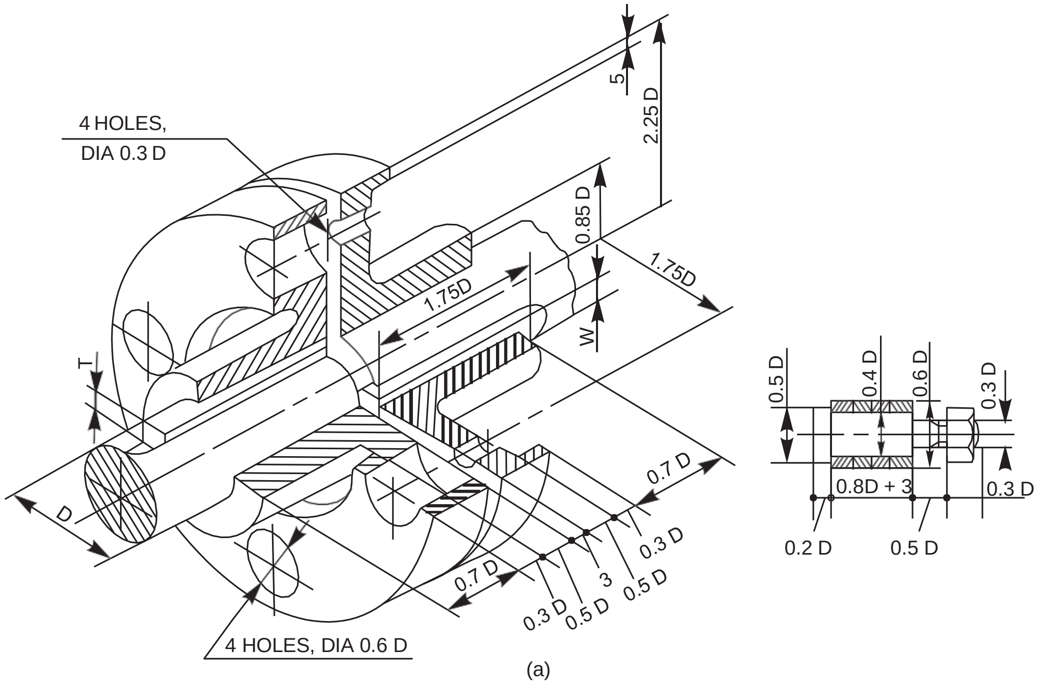
Fig. Solid flanged coupling

FLEXIBLE SOUPLINCS

Perfect alignment of two shafts is impossible to achieve and difficult to maintain, because of inevitable bearing wear and other reasons. To overcome the trouble, flexible couplings are employed. These permit relative rotation or variation in the alignment of shaft axes within certain limits. The following are the types of flexible couplings:

Bushed Pin Type Flanged Coupling

It is the modified version of a protected flanged coupling. In this, bolts are replaced by bushed pins. The smaller ends of the pins are rigidly fastened by nuts to one of the flanges, while the enlarged ends are covered with flexible material like leather or rubber bushes, in the other flange. The flexible medium takes care of mis-alignment, if any, and acts as a shock absorber. These couplings are used to connect prime mover or an electric motor and a centrifugal pump.



(a)

Fig. 7.7 Bushed pin type flanged coupling (*Contd.*)

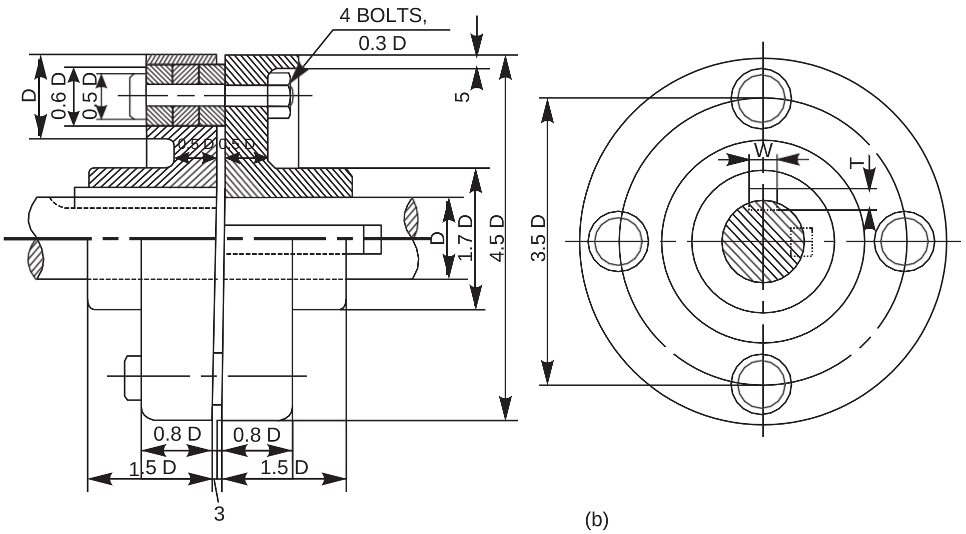


Fig. Bushed pin type flanged coupling

Compression Coupling

This consists of a compressible steel sleeve which fits on to the ends of the shafts to be coupled. The sleeve corresponds to the shaft diameter and its outer surface is of double conical form. The sleeve has one through cut longitudinally and five other cuts, equi-spaced, but running alternately from opposite ends to about 85% of its length; making it radially flexible.

The two flanges used have conical bores and are drawn towards each other by means of a number of bolts and nuts, making the sleeve firmly compressed onto the shafts. Here, the friction between the shafts and sleeve assists power transmission and the bolts do not take any load. Because of the presence of flexible sleeve, the coupling takes care of both axial and angular mis-alignment of shafts .

DISENGAGINC COUPLINCS

Disengaging couplings are used when power transmission from one shaft to another is intermittent. With this, the shafts can be engaged or disengaged as and when required, even during rotation. A dis-engaging coupling in general consists of one part firmly fixed to the driving shaft and another one mounted with provision for sliding over the driven shaft. The part that is mounted on the driven shaft, can be made to slide at will to engage or dis-engage from the rotating driving shaft. The following are the examples of dis-engaging couplings.

Slaw Coupling

In this, each flange has a number of identical claws which engage into the corresponding recesses in the flange. One flange is firmly fitted to the driving shaft by means of a taper sunk key. The other one is placed over the driven shaft by two feather keys, so that it can slide freely on it. The sliding flange has a groove on the boss, into which the forked end of a lever fits. By operating the lever, the sliding flange may be moved so as to engage with or dis-engage from the fixed flange . This type of coupling is generally used on slow speed shafts.

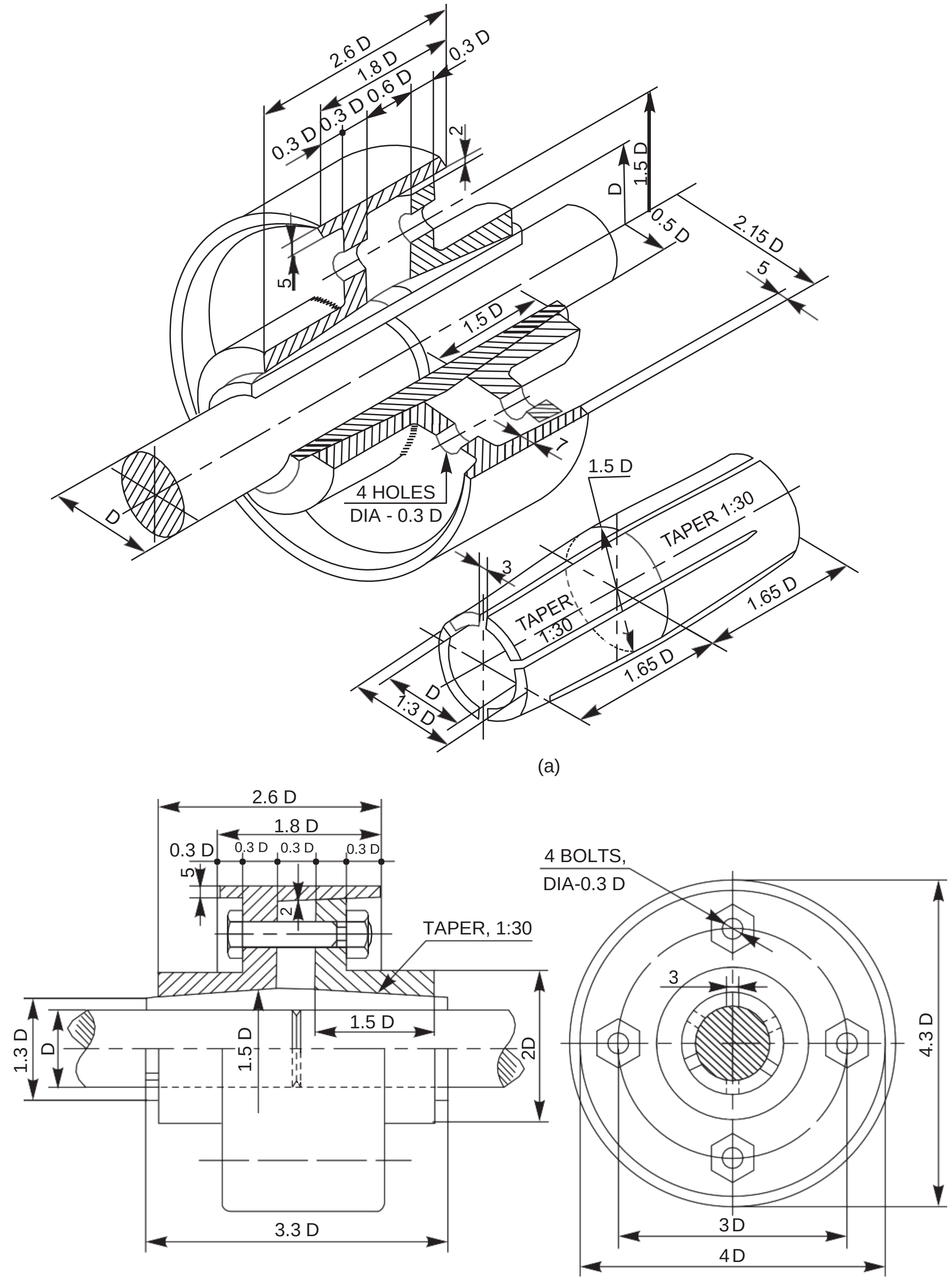


Fig. 7.8 Compression coupling

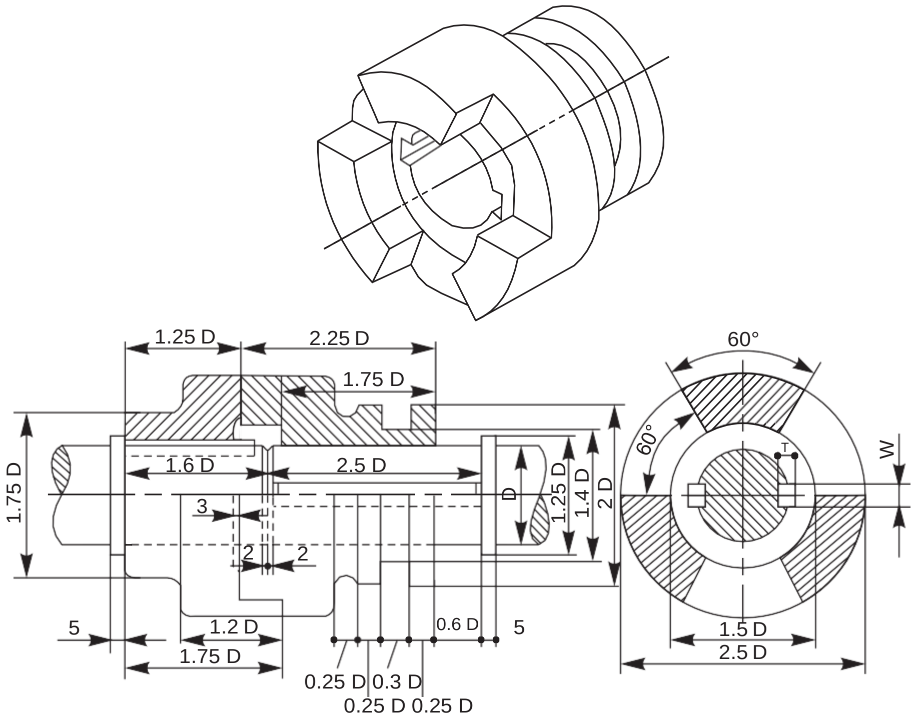


Fig. 7.9 Claw coupling

Cone Coupling

In this, two shafts may be coupled together by means of two flanges with conical surfaces (on the inside of one and on the outside of the other) by virtue of friction. Here too, one flange is firmly fitted to the driving shaft by means of a taper sunk key, whereas the other slides freely over a feather key fitted to the driven shaft. The sliding flange may be moved by means of a forked lever fitted into the groove provided on it..

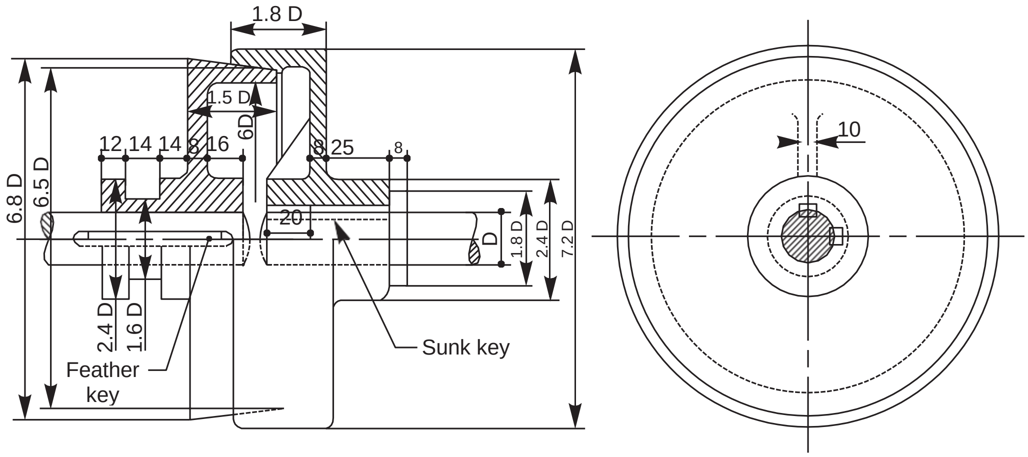


Fig. 7.10 Cone coupling

NON•ALIGNED COUPLINCS

Non-aligned couplings are used to transmit power between two shafts which are not coaxial. The following are the examples of non-aligned couplings:

Universal Coupling (Hooke's Joint)

It is a rigid coupling that connects two shafts, whose axes intersect if extended. It consists of two forks which are keyed to the shafts. The two forks are pin joined to a central block, which has two arms at right angle to each other in the form of a cross . The angle between the shafts may be varied even while the shafts are rotating.

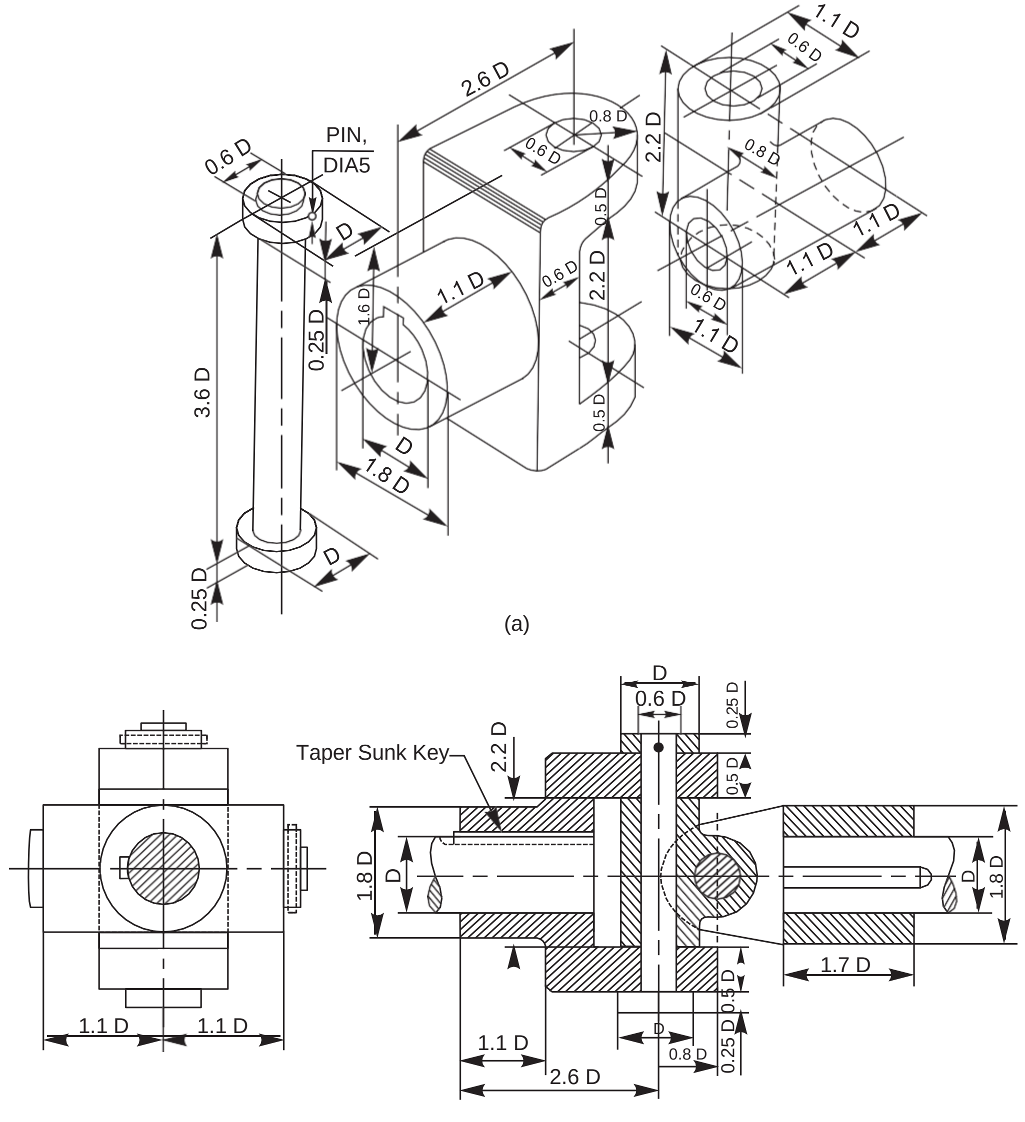


Fig. 7.11 Universal coupling

Oldham Coupling

It is used to connect two parallel shafts whose axes are at a small distance apart. Two flanges, each having a rectangular slot, are keyed, one on each shaft. The two flanges are positioned such that, the slot in one is at right angle to the slot in the other.

To make the coupling, a circular disc with two rectangular projections on either side and at right angle to each other, is placed between the two flanges. During motion, the central disc, while turning, slides in the slots of the flanges. Power transmission takes place between the shafts, because of the positive connection between the flanges and the central disc .

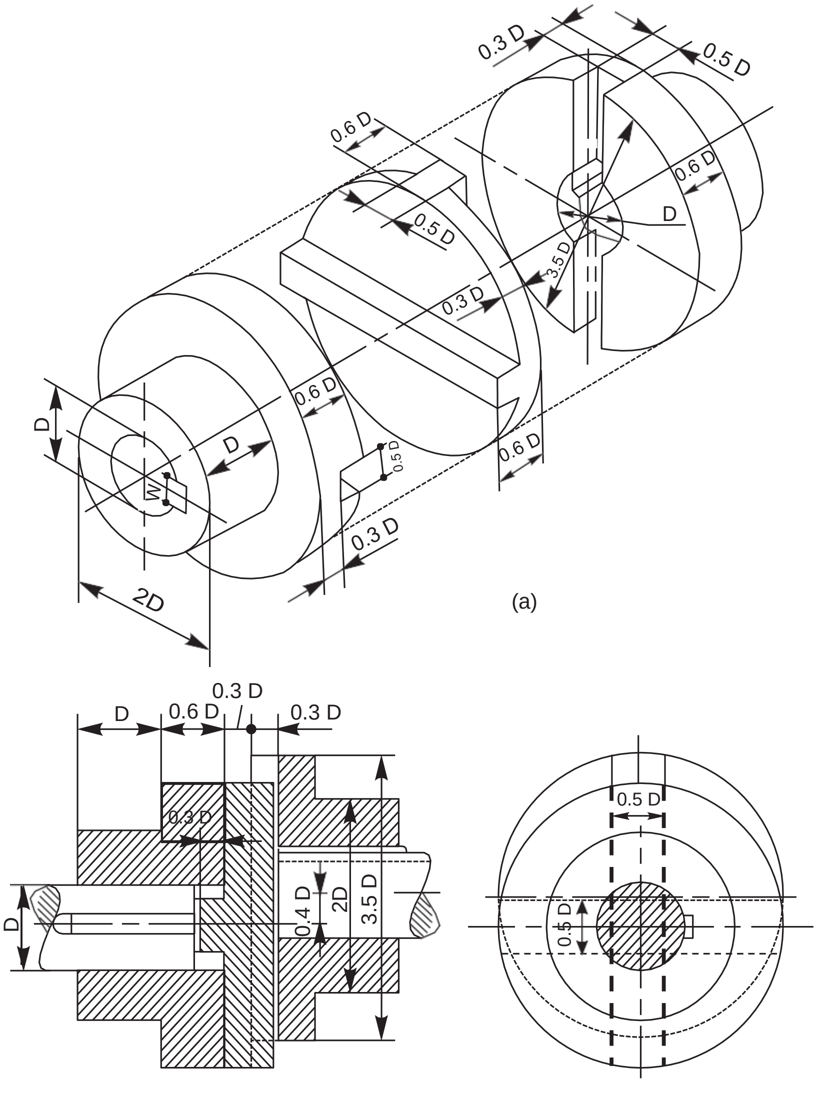


Fig. 7.12 Oldham coupling

Cushion Coupling

One of the most commonly used flexible couplings now-a-days is the cushion coupling. The H.P rating of these couplings for various speeds, range from 0.2 to 450.

The tyre of the coupling is made of natural and synthetic rubber, impregnated with canvas or rayon. The hubs are made of C I or steel (FIG).

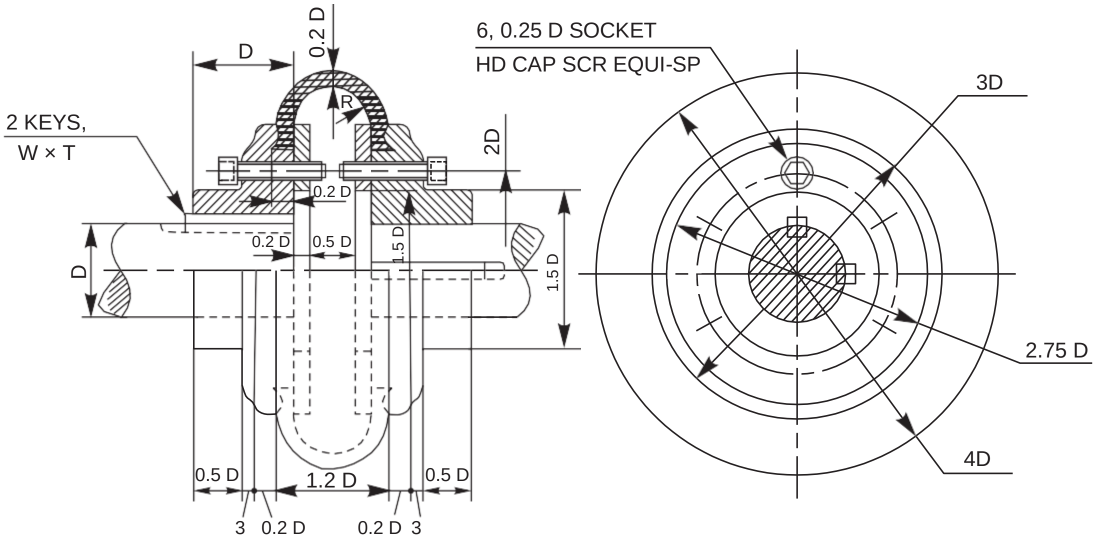


Fig. Cushion coupling

Engineering Graphics
Diploma

First Year,

Threads and fasteners

Threads & Fasteners:

Summary

1) Fasteners
2) Screw Thread Definitions
3) Types of Thread
4) Manufacturing Screw Threads
5) Drawing Screw Threads
6) Unified Threads
7) Metric Threads
8) Drawing Bolts
9) Bolt and Screw Clearances

Summary

- What will we learn in this topic?
  - How to represent threads on an engineering drawing.
  - How to calculate bolt and screw clearance holes.

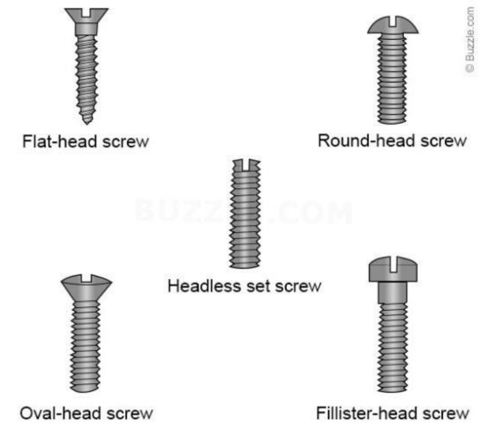
- Key points
  - Threads are represented by thread symbols, not by a realistic drawing.

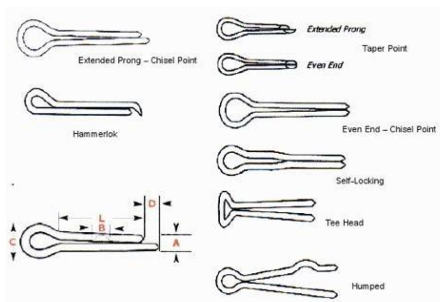
Threads and Fasteners

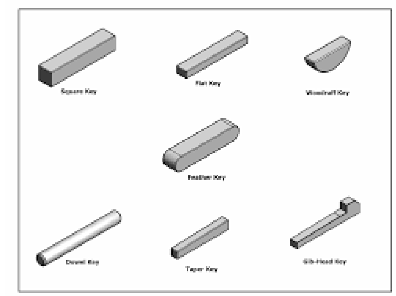
1) Fasteners

Fastener

- Fasteners include:
  - bolts and nuts (threaded
  - set screws (threaded)
  - washers
  - keys
  - Pins







- Fasteners are not a permanent means of assembly such as welding or adhesives.

Fastener

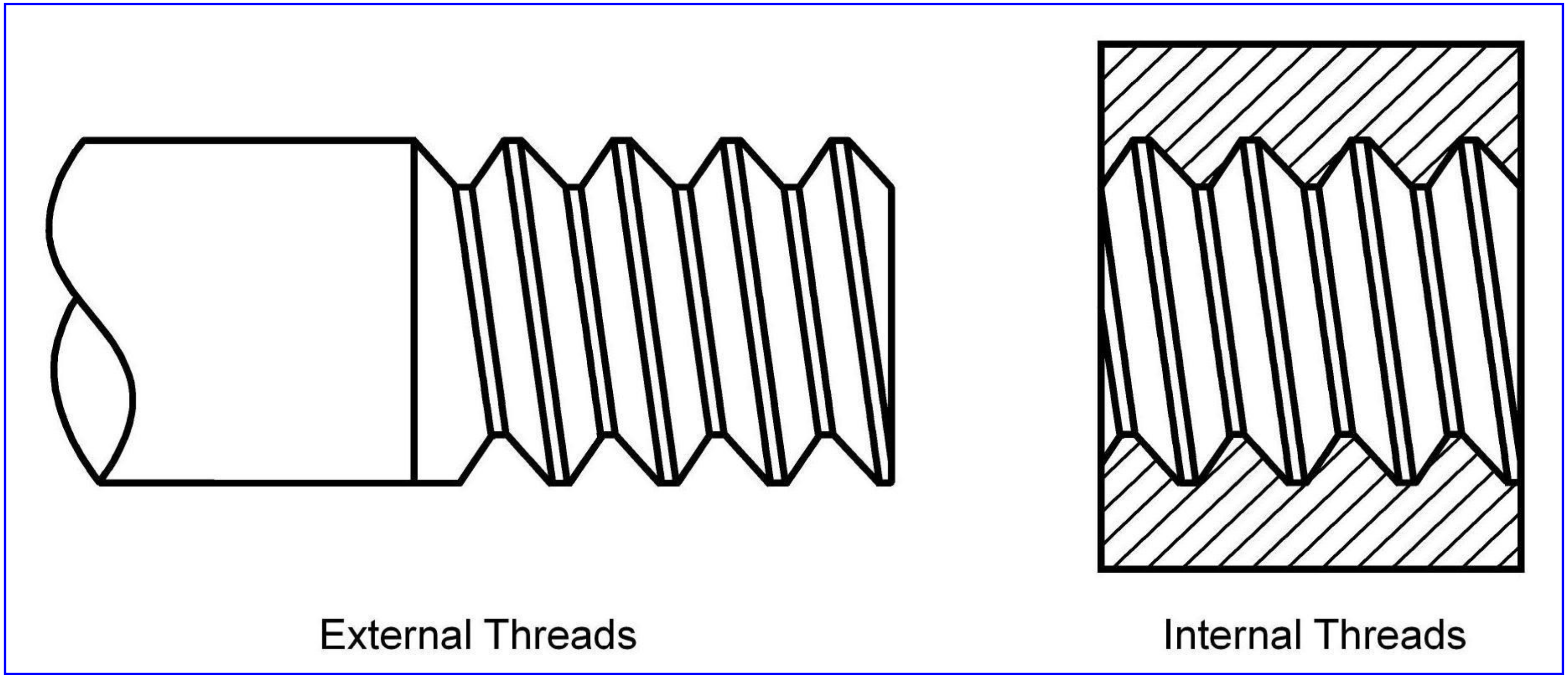
- Fasteners and threaded features must be specified on your engineering drawing.
  - Threaded features: Threads are specified in a thread note.
  - General Fasteners: Purchasing information must be given to allow the fastener to be ordered correctly.

Threads and Fasteners

2) Screw Thread Definitions

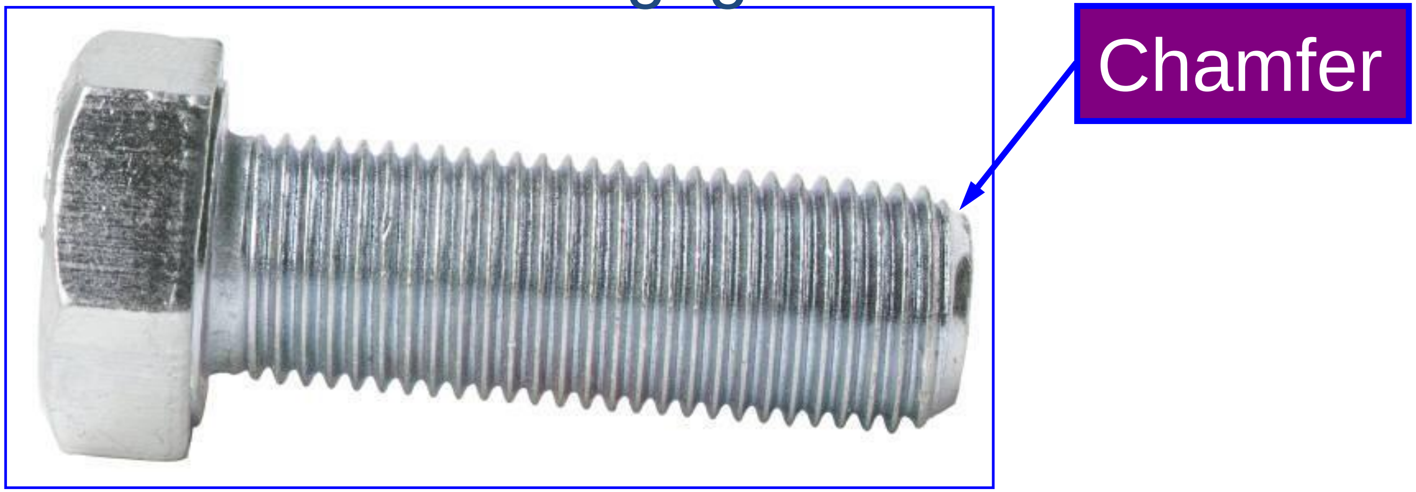
Thread

- Screw Thread: A ridge of uniform section in the form of a helix.



Thread

- External Thread: External threads are on the outside of a member.
  - A chamfer on the end of the screw thread makes it easier to engage the nut.



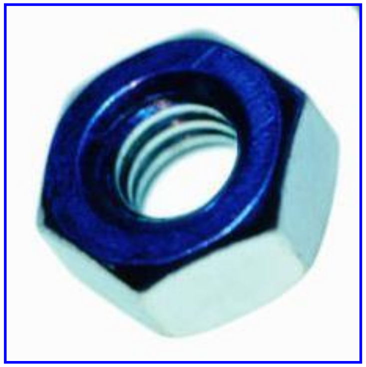
Thread

- External Thread:

  – An external thread is cut using a die or a lathe.

Thread

- Internal Thread: Internal threads are on the inside of a member.



  - An internal thread is cut using a tap.

Thread

- Major DIA (D): The largest diameter (For both internal and external threads).

- Minor DIA (d): The smallest diameter.

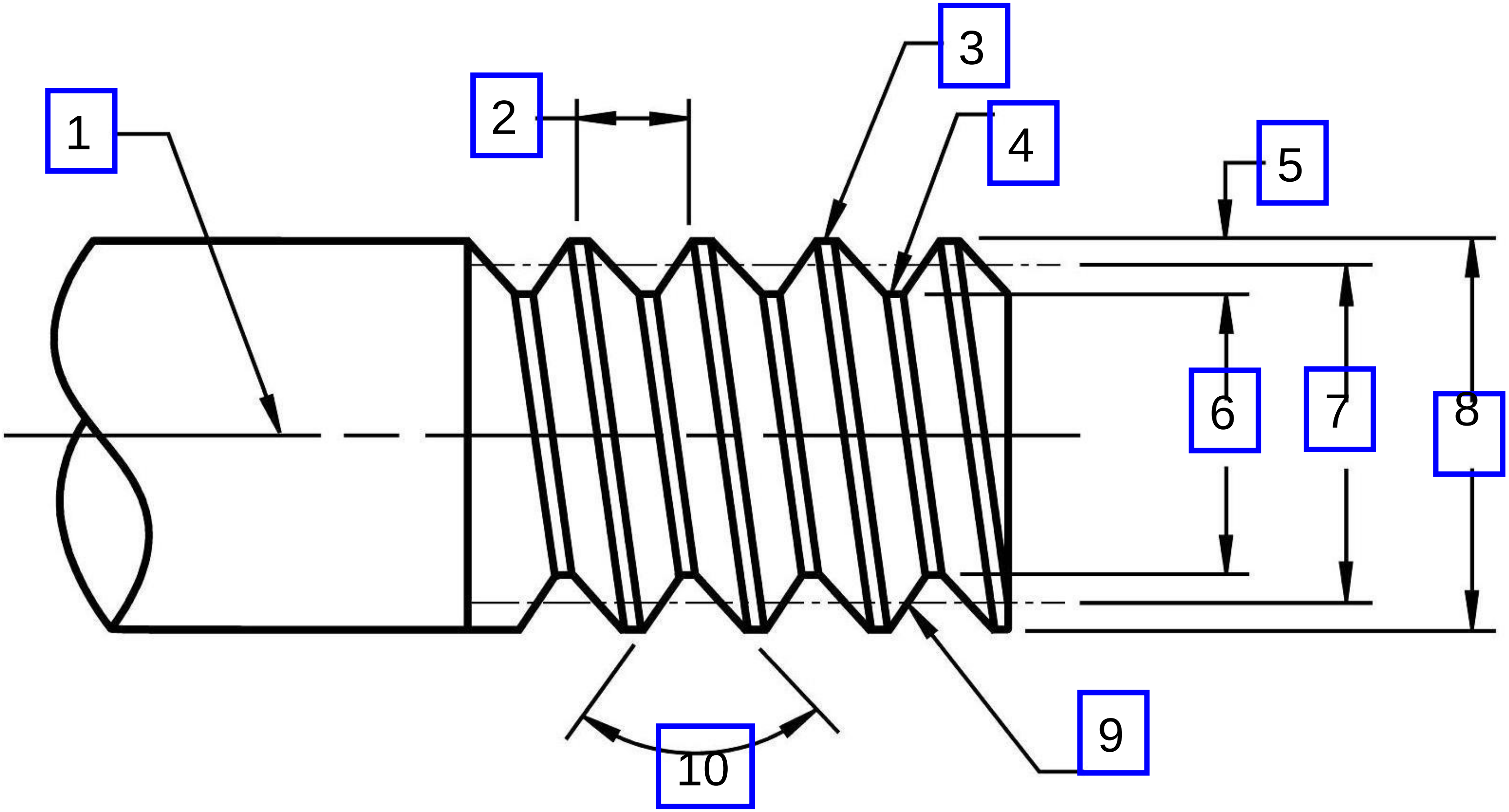
- Depth of thread: (D-d)/2

- Pitch DIA (d_P): The diameter at which a line cuts the spaces and threads equally.

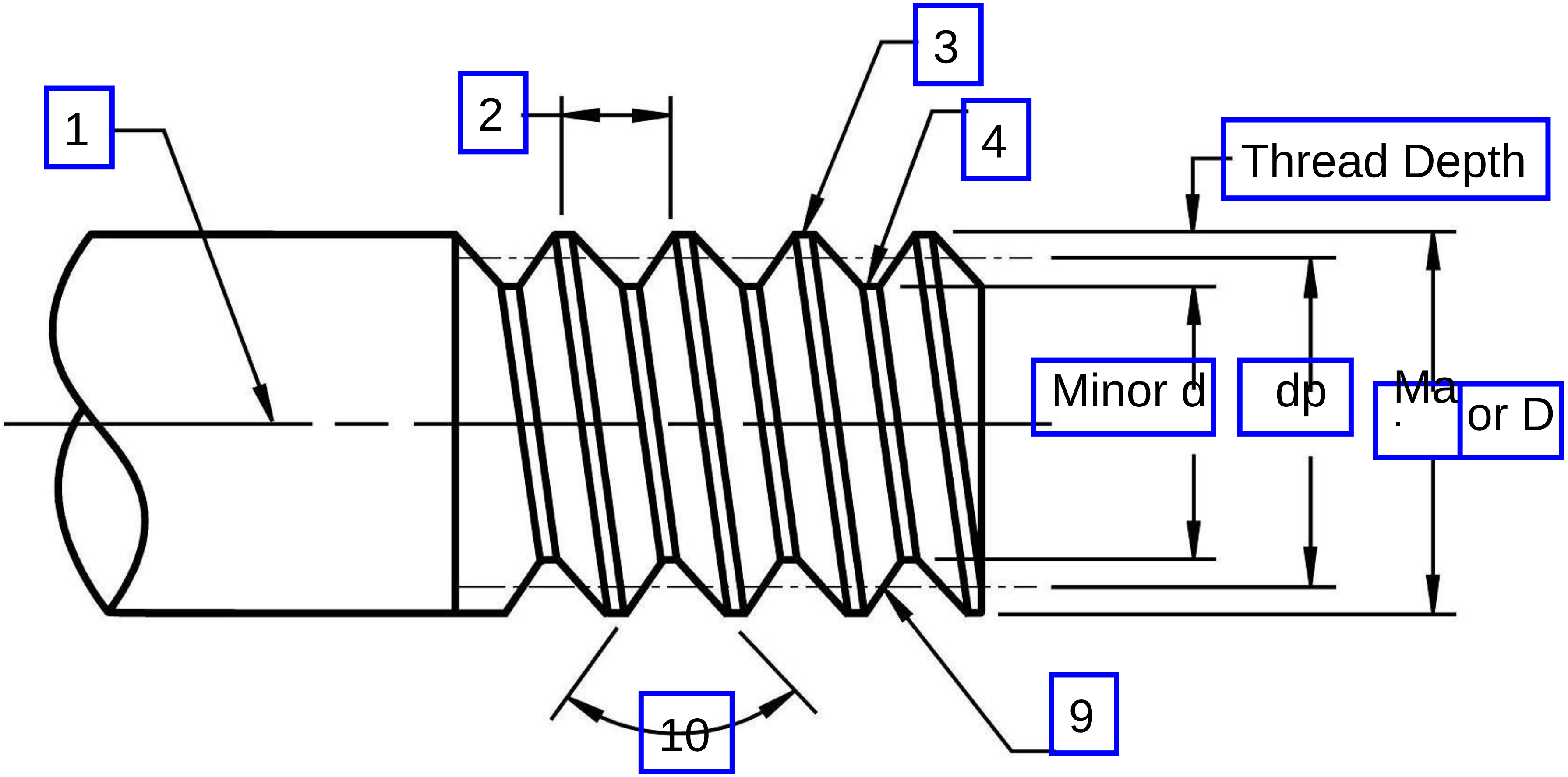
Exercise 1

Screw thread features

Identify the *Major, Minor & Pitch*



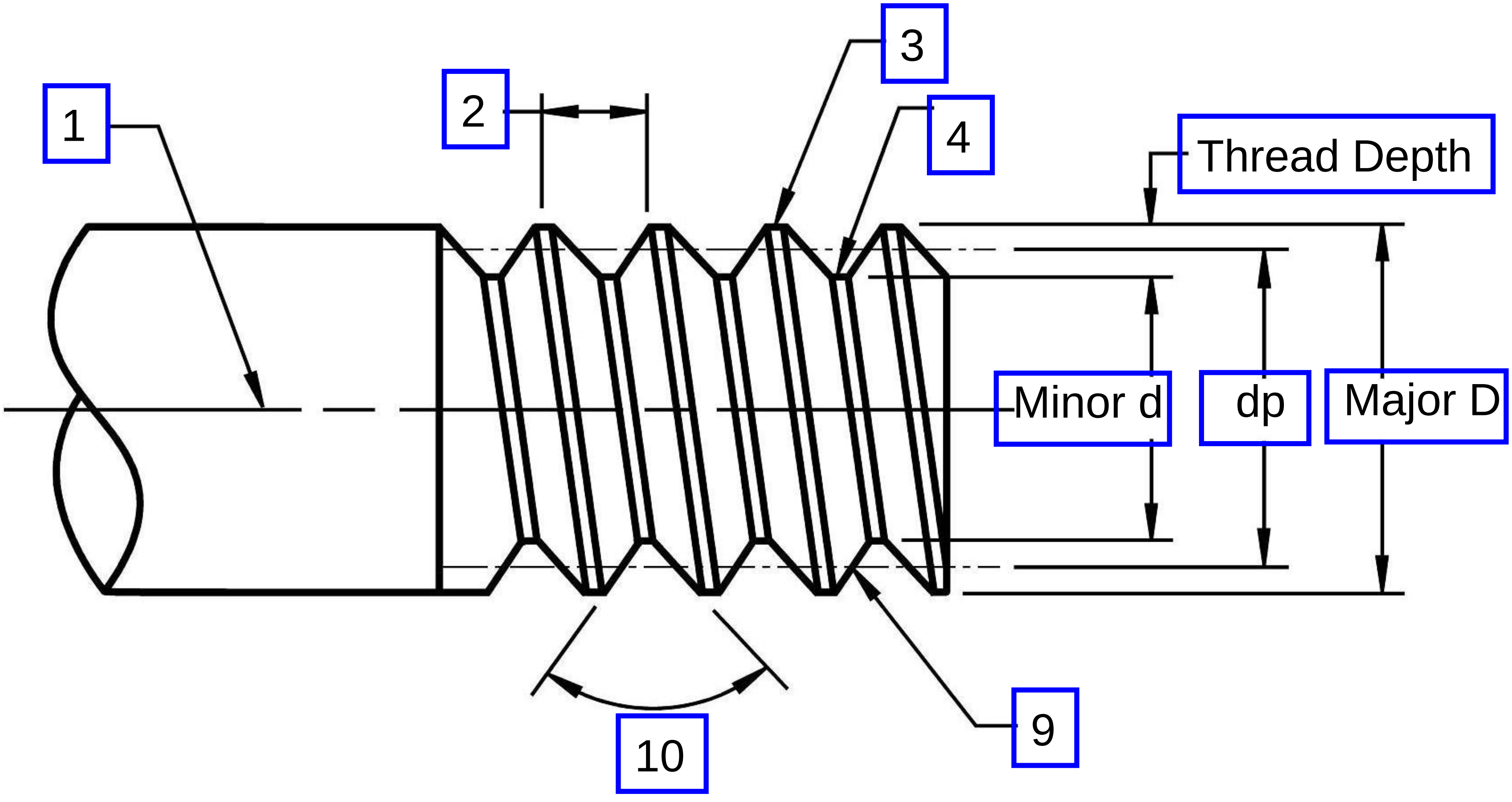
Identify the *Major, Minor & Pitch*



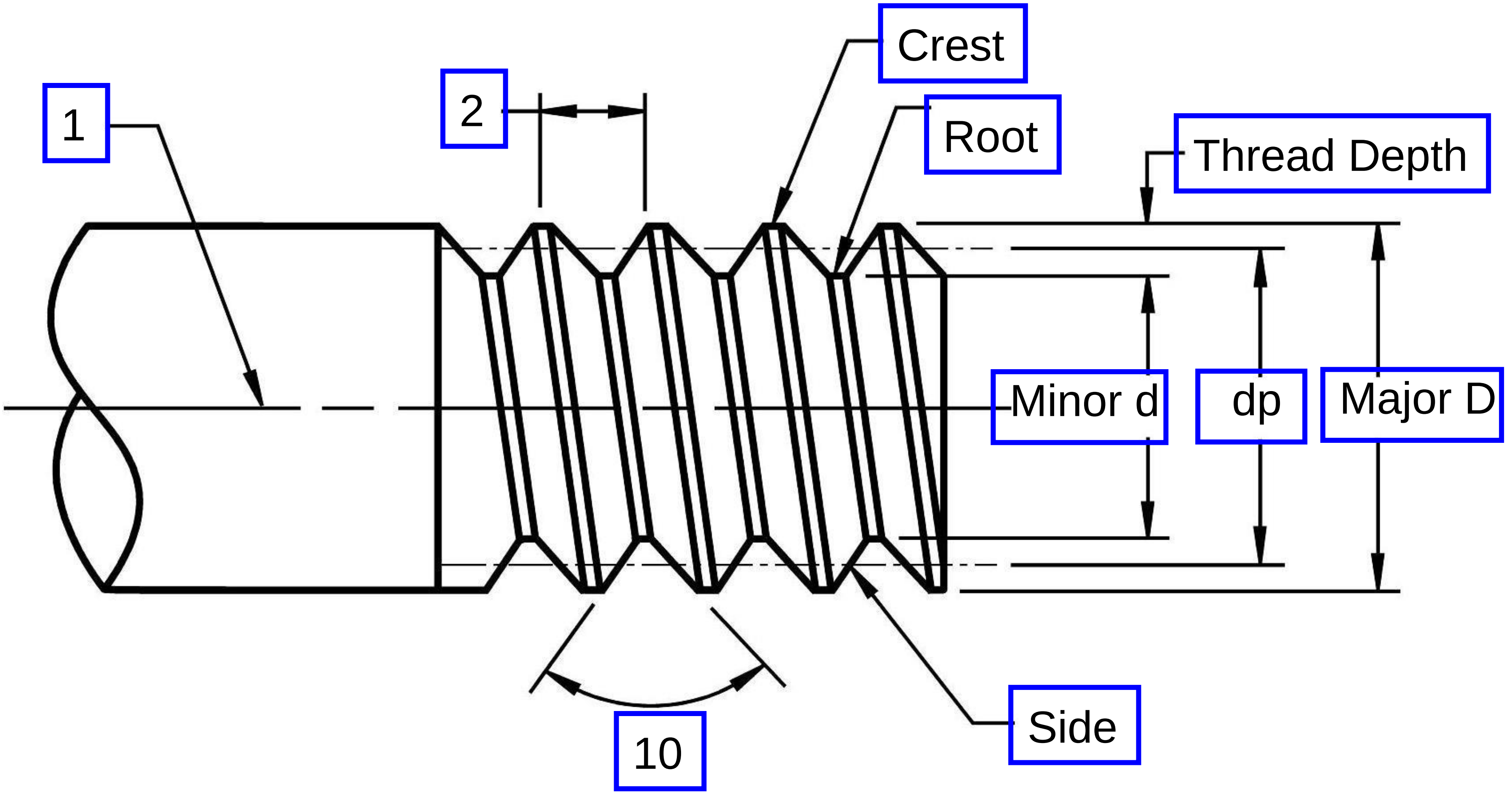
Thread Definitions

- Crest: The top surface.
- Root: The bottom Surface.
- Side: The surface between the crest and root.

Identify the *Crest*, *Root* and



Identify the *Crest*, *Root* and



Thread

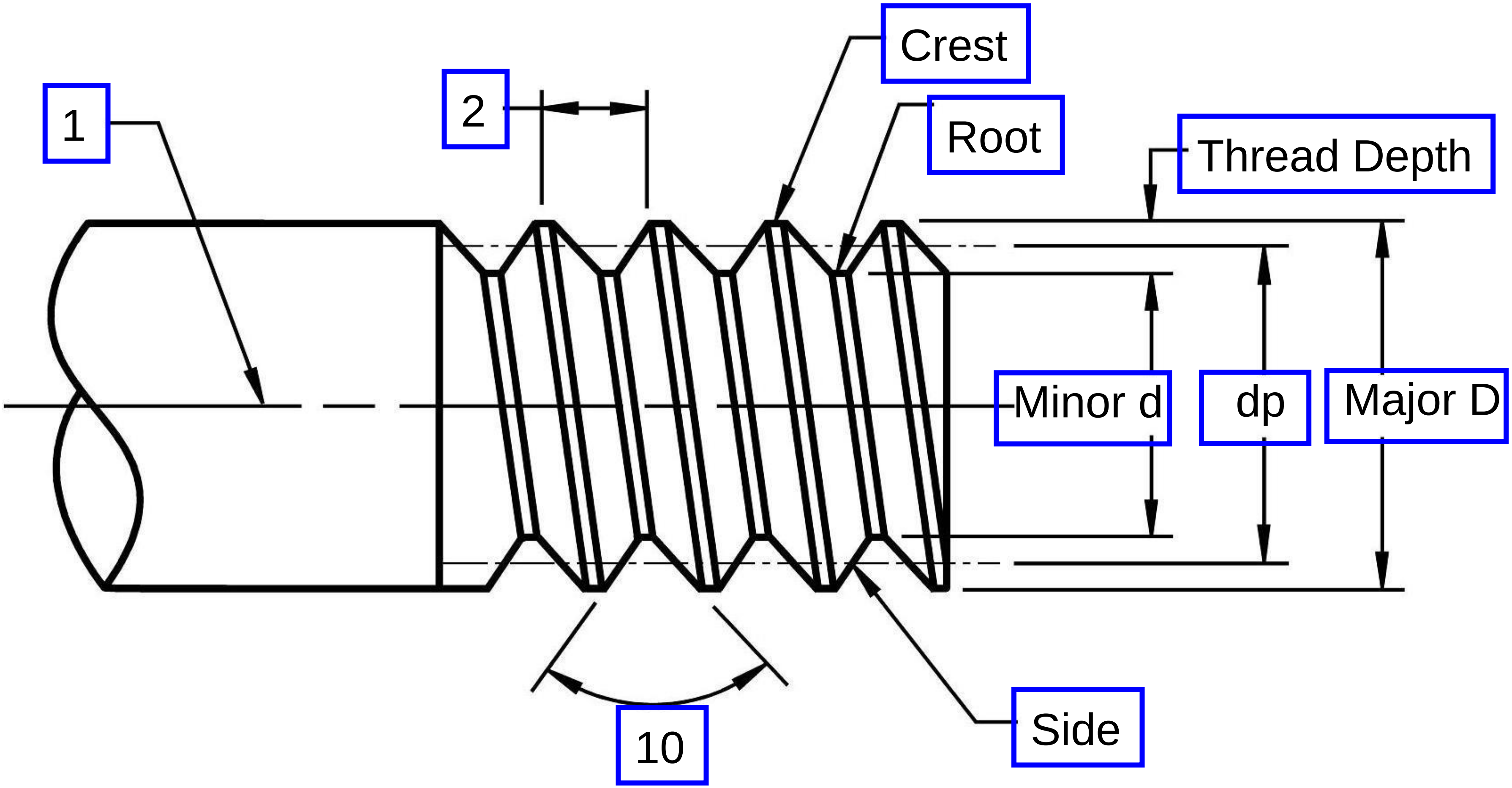
- Pitch (P): The distance from a point on a screw thread to a corresponding point on the next thread (in/Threads).

- Angle of Thread (A): The angle between the threads.

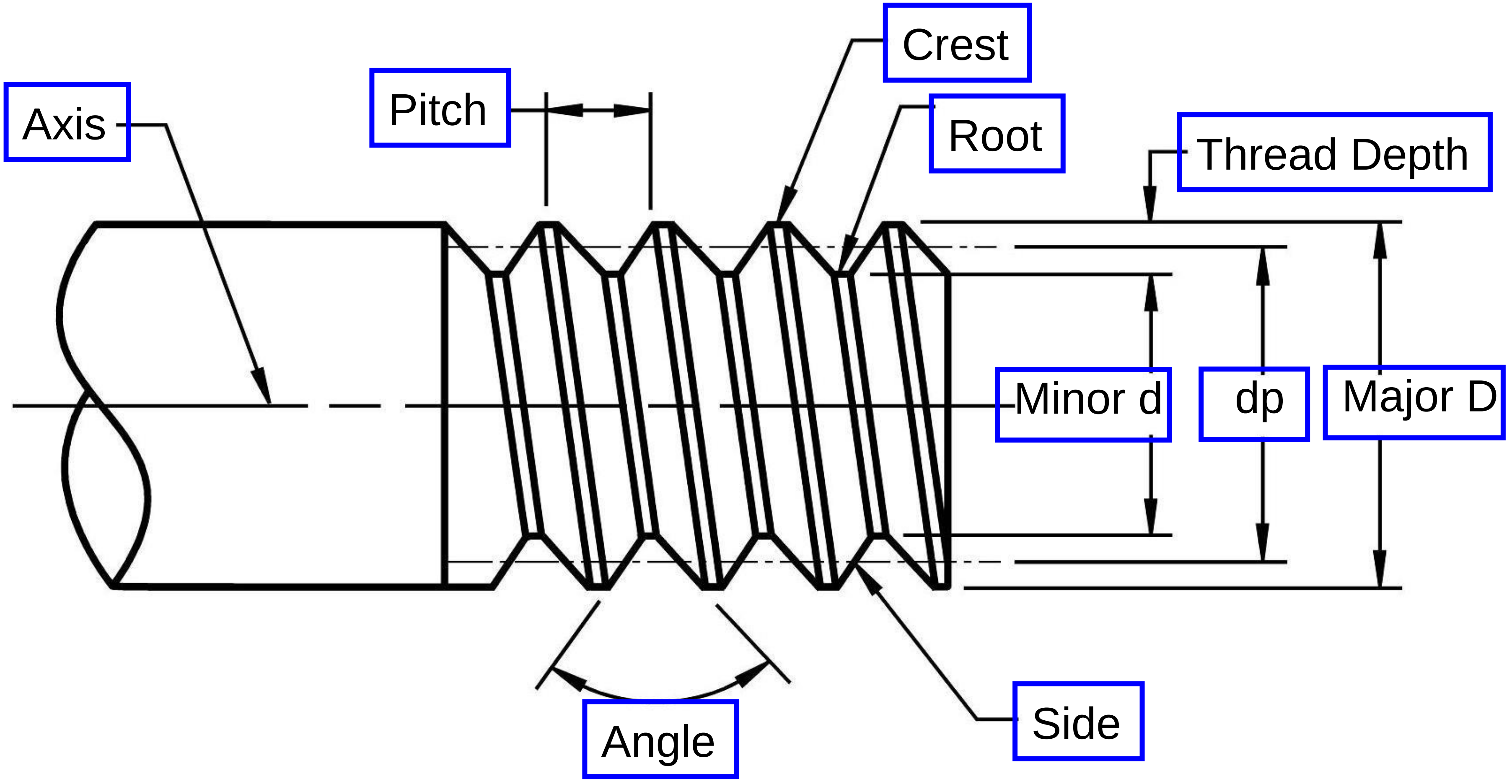
Thread

- Screw Axis: The longitudinal centerline.
- Lead: The distance a screw thread advances axially in one turn.

Identify the *Pitch*, *Screw Axis* and *Thread*



Identify the *Pitch*, *Screw Axis* and *Thread*



Thread Definitions

- Right Handed Thread: Advances when turned CW. (Threads are assumed RH unless specified otherwise.)

- Left Handed Thread: Advances when turned CCW.

Application Question 1

- Name an example of a left handed thread.

  Left peddle of a bike

Threads and Fasteners

3) Types of Thread

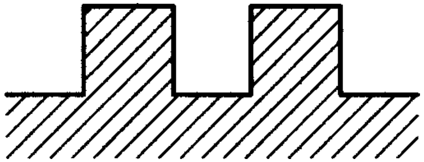
Types of

- There are many different types of thread forms (shape) available. The most common are;
  - Unified
  - Metric

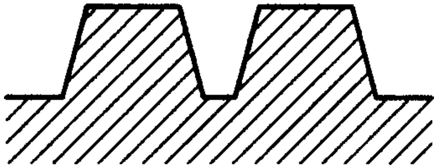
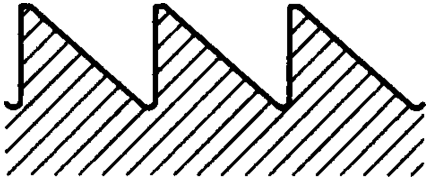
Types of

- Thread form choice depends on;
  - what it will be used for
  - length of engagement
  - load
  - etc…

Types of Thread

Thread Name	Figure	Uses
Unified screw thread		General use.
ISO metric screw thread		General use.
Square		Ideal thread for power transmission.

Types of Thread

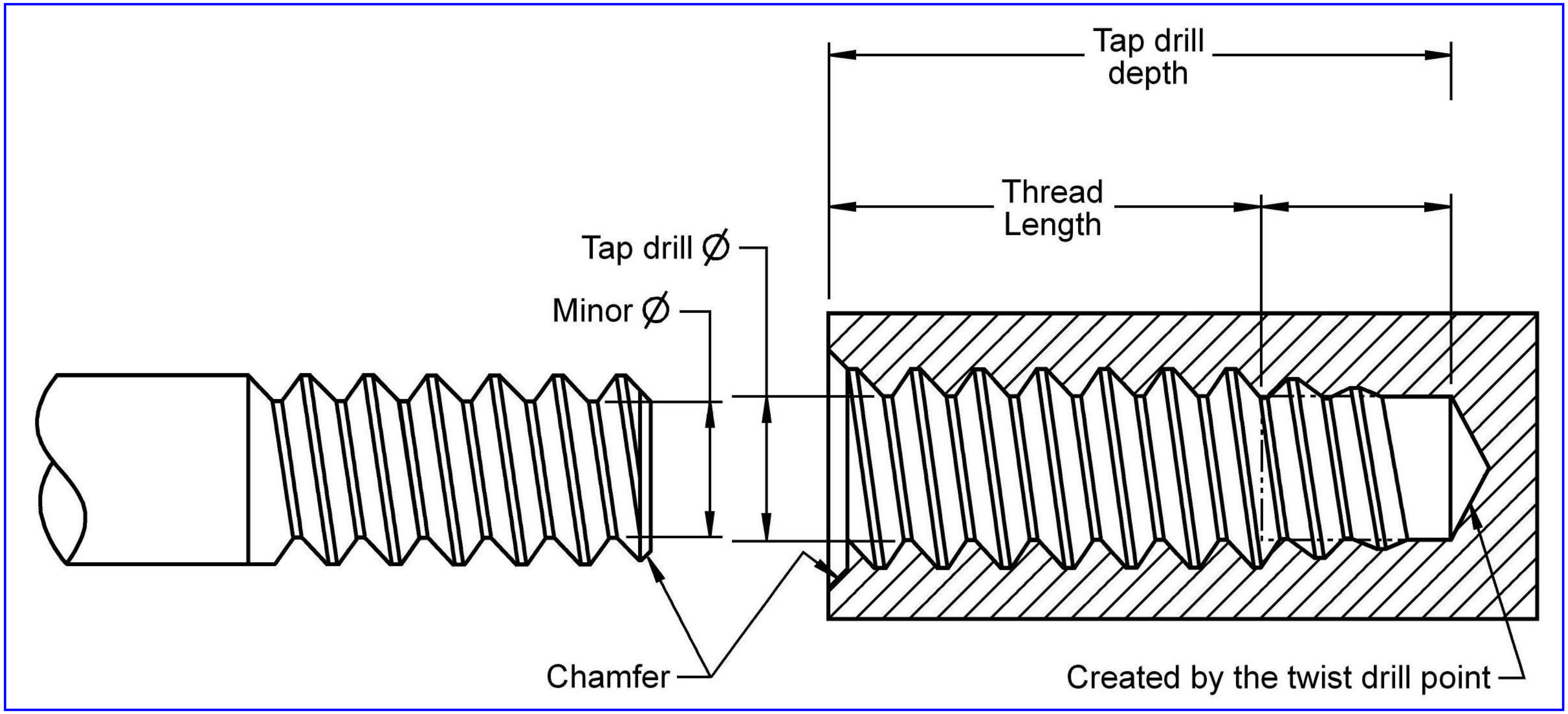
Thread Name	**Figure**	**Uses**
ACME		Stronger than square thread.
Buttress		Designed to handle heavy forces in one direction. (Truck jack)

Threads and Fasteners

4) Manufacturing Screw Threads

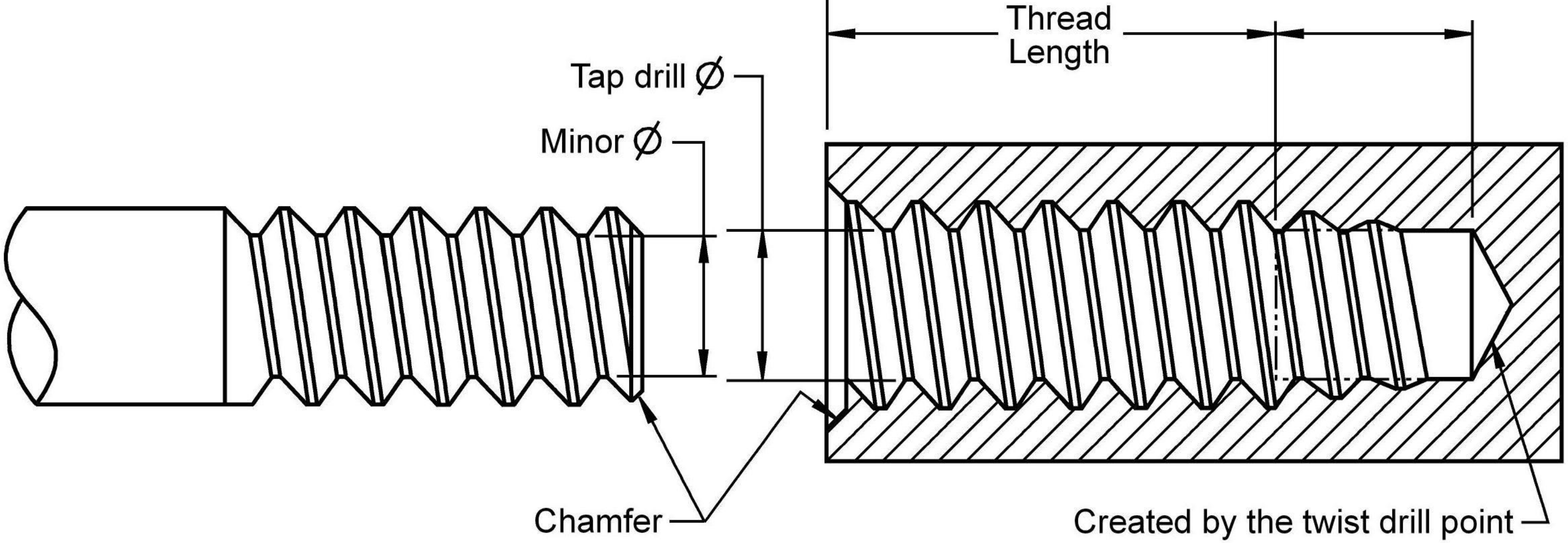
Manufacturing

- Internal Threads
  - First a tap drill hole is cut with a twist drill.



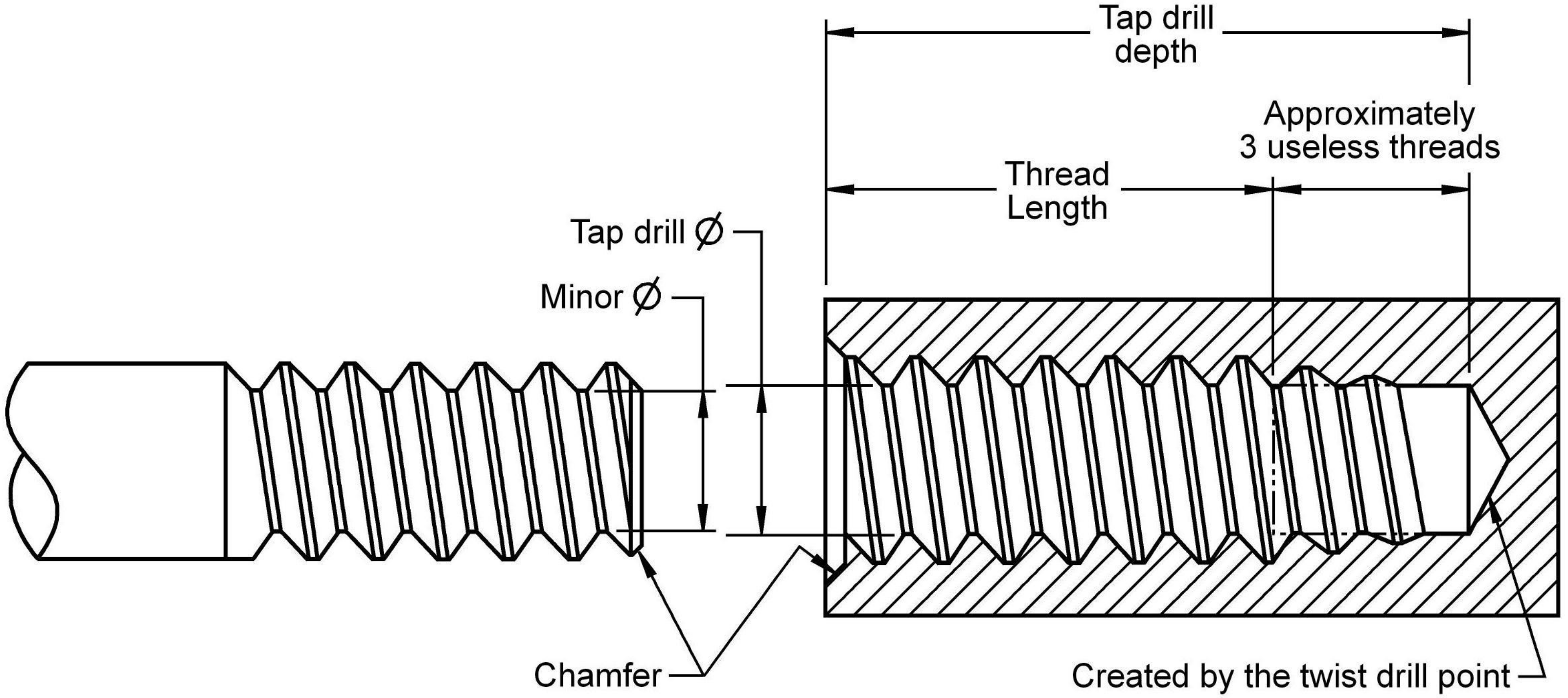
Manufacturing

- Internal Threads
  - Then the threads are cut using a tap.



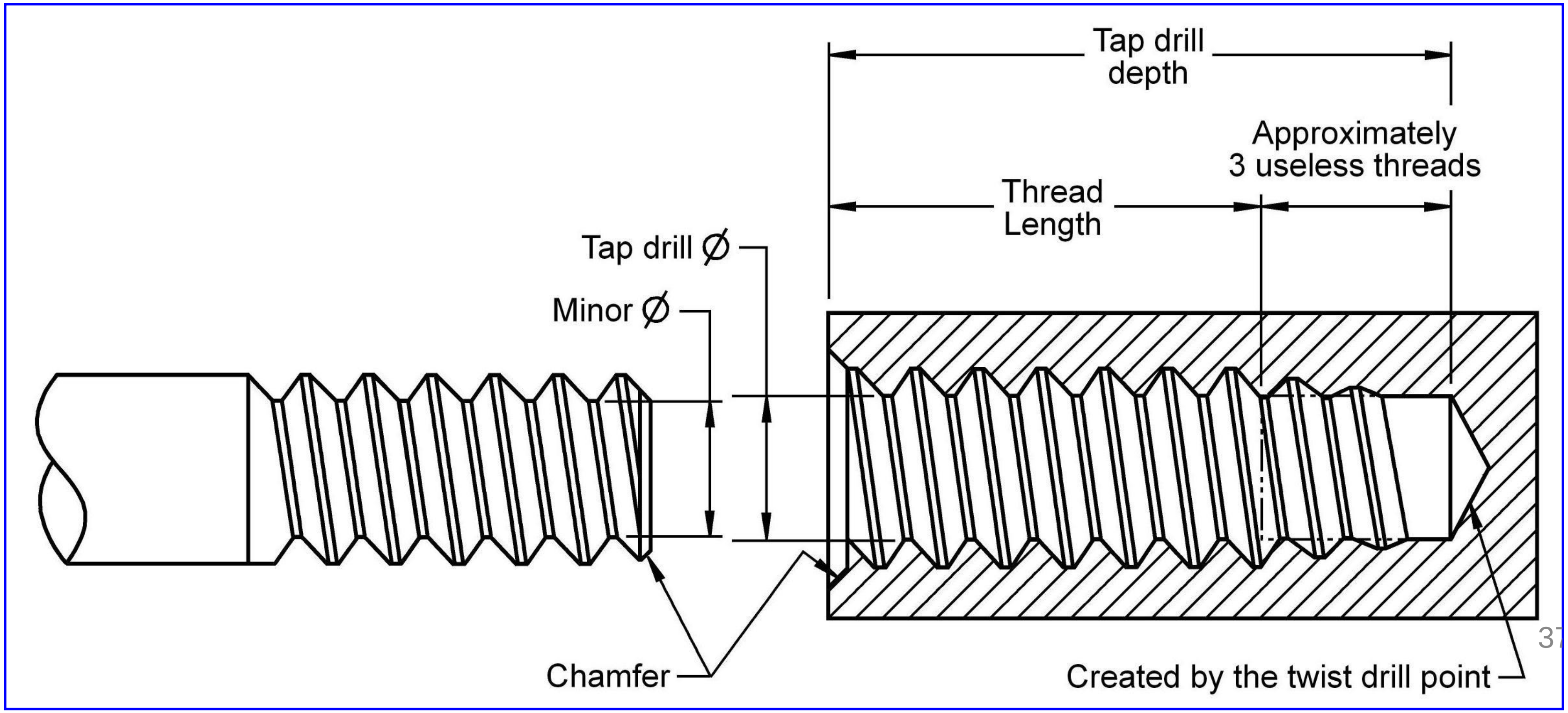
Manufacturing

- Internal Threads
  - Chamfers are sometimes cut to allow for easy engagement.



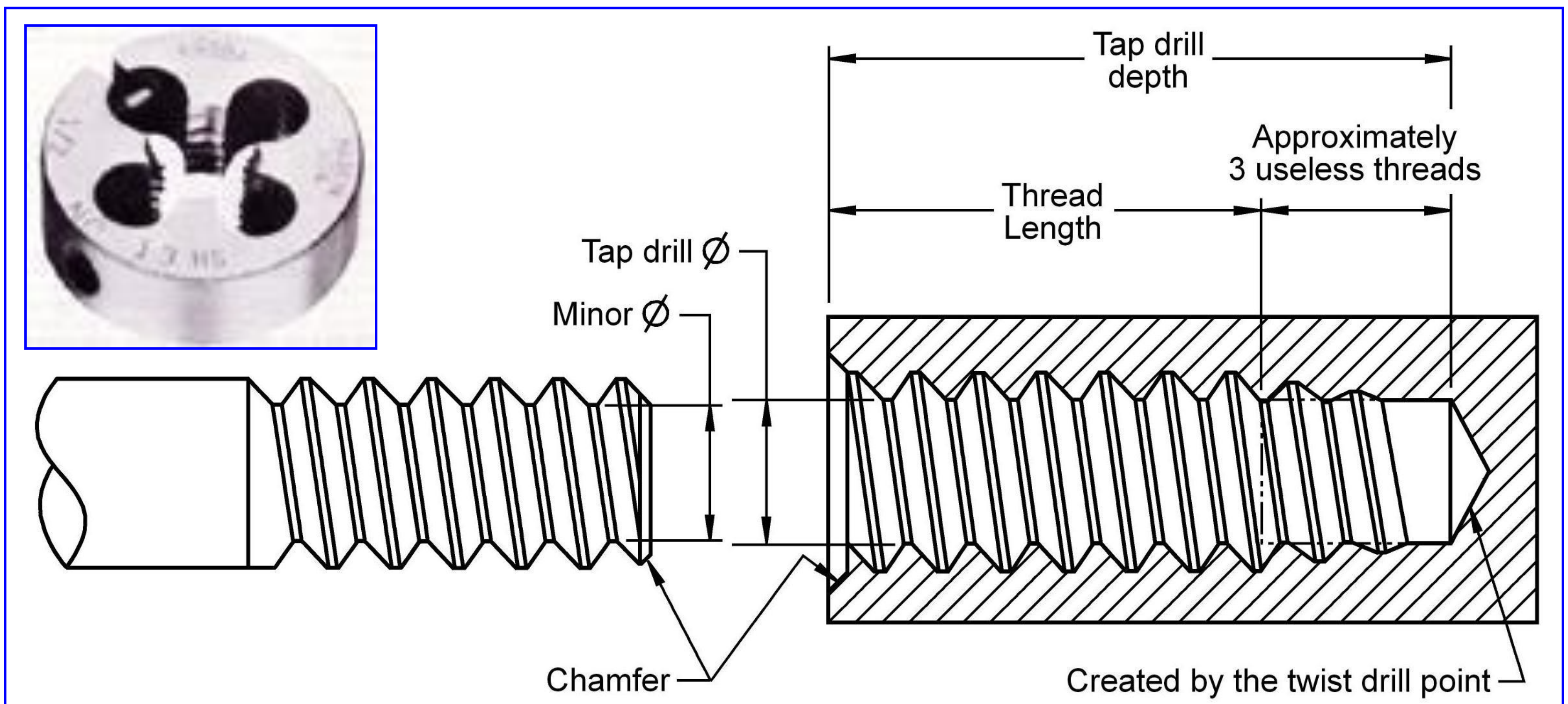
Manufacturing

- External Threads
  - You start with a shaft the same size as the major diameter.



Manufacturing

- External Threads
  - The threads are then cut using a die or on a lathe.



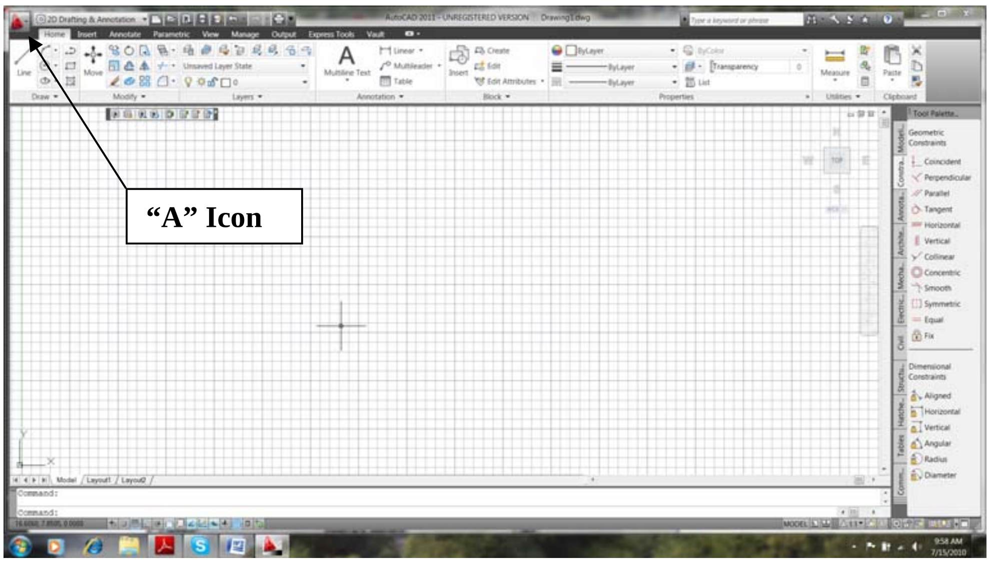
Manufacturing

The term CAD (Computer Aided Design) applies to a wide range of programs that allow th user to created drawings, plans, and designs electronically. AutoCAD is one such program and it main claim to fame is that it is relatively easy to use, it is very comprehensive in its ability to create 2D and some 3D drawings, and it is very popular. Seventy percent of the CAD users in the world use AutoCAD.

I Starting AutoCAD

You can start AutoCAD by either double clicking on the program Icon on the desktop or by clicking on the program name in the Start menu.

The program will start and after a minute or so should display a screen similar to the one shown below. The dialog box in the middle will aid you in getting started at either creating a new drawing or continuing your work on a drawing that is not finished.



If you are continuing work on a drawing, click on the "**A**" icon in the extreme upper left corner of the window and Open->Drawing. A "Select File" dialog box will open allowing you to select the drawing file you want to open.

II The Initial Screen

AutoCAD has a very versatile user interface that allows you to control the program in several different ways. At the top of the window is a row of menus. Clicking on the Home, Insert, or Annotate causes another selection of menus to appear. This new selection of commands is frequently called a Ribbon or a Dashboard. You can operate the program by clicking on the icons in these menus.

Another method of using the program is typing in the command names. This is frequently faster than using drop down menus for frequently used commands because you do not have to search for the correct menu or icon. You just type in the command name. For the most part, we will use this approach in this series of tutorials. The commands that you type will appear at the bottom of the of the AutoCAD window.

III The LINE Command

Now that you have started AutoCAD and configured tool bars you want, you are ready to start learning to use the program. We will start with relatively simple commands and eventually, in later lessons, look at some of the more complex things that AutoCAD can do. The first command we will look at drawing straight lines. At the keyboard, type:

```
line
```

and press the ENTER key. You can use either upper or lower case when you type in AutoCAD commands.

The program will respond with:

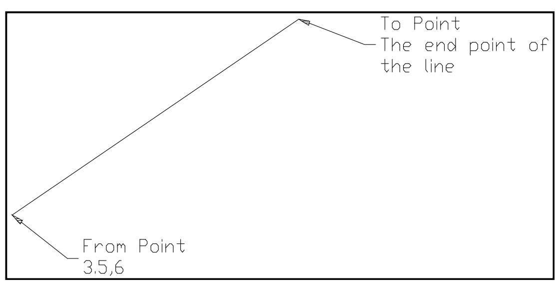
```
Specify First Point:
```

Each line has a beginning and ending point and the program wants you to specify the beginning point of the line. You enter the beginning point by either typing the point coordinates at the keyboard or by clicking the mouse on a location of the screen where you want the line to begin. It is certainly much simpler to click with the mouse than it is to type in coordinates but engineering drawings are drawn precisely to scale and for the most part we will have to enter coordinates from the keyboard.

When you type a coordinate, enter the X or horizontal coordinate first followed by a comma and the Y or vertical coordinate. You cannot enter a space between the two coordinates. AutoCAD interprets a space as the ENTER key and assumes that you have finished entering the coordinates.

For Example, you could type:

```
Specify First Point:  3.5,6
```



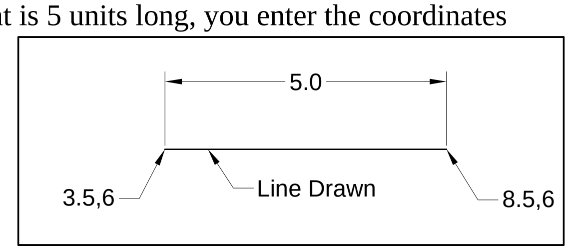
The 3.5 coordinate is the X or horizontal coordinate and the 6 is the vertical coordinate.

After you enter the coordinates, press the enter key. The enter key tells the program that you have entered the first coordinate and are ready to enter the coordinates for the next which will be the end of the line. The program responds by displaying:

```
To Point:
```

If you want a horizontal line that is 5 units long, you enter the coordinates @5,0 which is shown below.

```
To Point:        @5,0
```



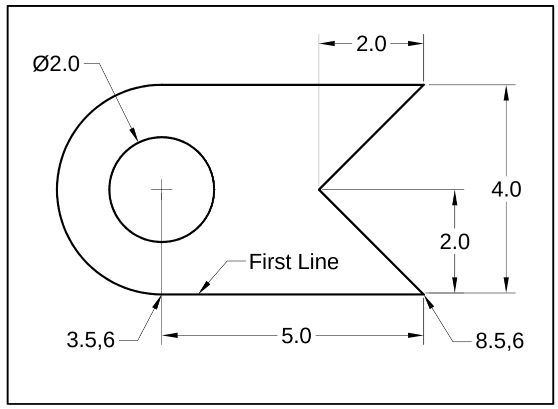
The @ sign tells the program this coordinate is measured from the last coordinate entered. In other words, it says place the end if the line 5 units horizontally from the beginning point and 0 units vertically. The line drawn is shown above.

Using the @ sign to specify relative coordinates is easier than specifying absolute coordinates without the @ sign. The first point we drew had an absolute coordinate of 3.5,6 and the second point had an absolute coordinate of 8.5,6 since it is displaced 5 units horizontally from the first point.

We will continue with this to create the object shown on the right. It has lines, an arc, and a circle. We have drawn the first and we will continue drawing the rest of the lines.

As a shortcut, you can start the **`LINE`** command by typing **`L`** instead of the entire word **`LINE`**. Many AutoCAD commands can be abbreviated to just the first letter of the command.

IV Continue Drawing the Object

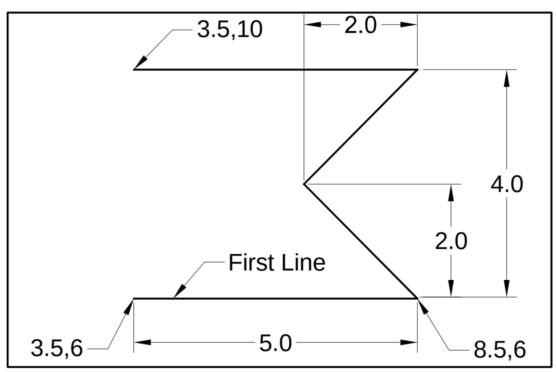


We can continue drawing the object shown on the right by adding more lines. If the line command is still operating, press ENTER to end it. We will start it again to draw the remaining lines.

You can draw the remaining lines by typing:

`line`		
`Specify First Point:`	**`8.5,6`**	*{these are the coordinates of the end of the first line we drew}*
`To Point:`	**`@-2,2`**	
`To Point`	**`@2,2`**	
`To Point`	**`@-5,0`**	
`To point`		*{Press ENTER without entering coordinates. This will end the line command}*

When you have finished entering all of the coordinates, you should have the object shown on the right.



V Erasing Objects

AutoCAD calls lines, circles, arcs and other things that you draw objects. You can erase any of these objects by typing the command:

ERASE

The program will respond with:

```
Select Objects:
```

You select the objects (lines, arcs, circles, etc.) in several different ways. The easiest way is to click on the object you want to erase. When you do, the object is redrawn as a dashed line. This shows the object has been selected for deletion. Click on all of the objects that you want to erase then press the ENTER key to terminate the command and erase the objects.

AutoCAD commands frequently have command modifiers that change the way the command works. For the ERASE command, you can type:

`ERASE ALL`

and AutoCAD selects all of the objects in the drawing for erasure. The word **ALL** modifies the way command works.

Another option is:

`ERASE W`

The **W** stands for window which allows you to select the objects by drawing a box around them. First click above and to the right of the objects that you want to erase. When you do, the mouse pointer changes to an elastic box with one corner fixed at the place where you clicked. Move the mouse until the box completely covers the information you want selected and click the mouse button again. All of the objects inside the box will be selected for erasure. Press the ENTER key to erase the objects.

You can type **E** to start the ERASE command.

VI Oops

If you make a mistake and erase something that you did not want to erase, type:

`OOPS`

to undo the last erasure. OOPS always undoes the last erasure even though you have continued with other commands since the objects were erased.

VII Canceling a Command

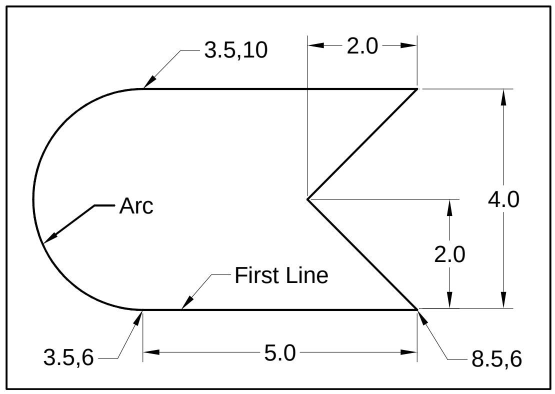
If you start a command and do not want to complete it, you can press the **Esc** key to cancel the command. For some commands, you may have to press the key more than once. Keep pressing the **Esc** key until you see the **`Command:`** prompt at the bottom of the screen.

VIII Drawing Arcs

The **ARC** command is used to draw arcs. We can use this command to draw the semicircle on the left side of the object. Enter:

Arc		
`Specify start point of arc or [Center]:`	**`3.5,10`**	*{The end point of the last line we drew}*
`Specify second point of arc or [Center/End]:`	**`c`**	*{Enter C to tell the program we want to enter the center point instead of the end point of the arc}*
`Specify center point of arc:`	**`@0,-2`**	*{The center of the arc is 2 units below the start point}*
`Specify end point of arc:`	**`@0,-2`**	*{The end of the arc is 2 units below the center}*

The completed arc is shown in the drawing on the right. Unless otherwise specified, AutoCAD will draw arcs in a counterclockwise (anticlockwise) direction.



IX Drawing Circles

Circles are created with the CIRCLE command. Type:

`CIRCLE`

at the command prompt and AutoCAD will respond with:

`3P/2P/TTR/<Center point>:`

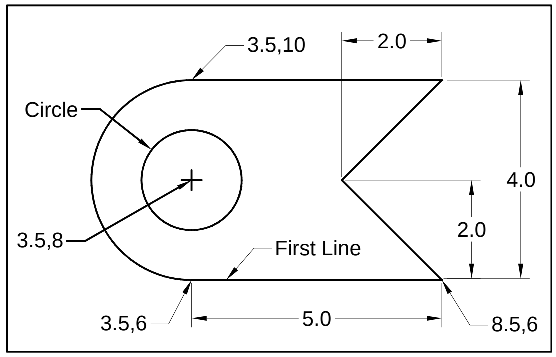
There are several different ways you can define a circle. In the computer response above, the words **Center point** are surrounded by angle brackets and this shows you the program is expecting you to enter the coordinates of the center of the circle. You can either type the coordinates or click with the mouse. The quantity in angle brackets is always the default selection for a command. The letters `3P/2P/TTR/`are options you can use to modify the input required to create a circle. These options are:

3P	Define the circle with 3 non-collinear points.
2P	Define the circle with points on either end of the circle diameter.
TTR	Define the circle by specifying two other objects that are tangent to the circle and the radius of the circle.

We can complete the drawing by drawing a circle. The center of the circle is two units vertically above the beginning point where we started the drawing. The coordinates for the center of the circle are 3.5,8. The circle has a diameter of 2.0.

```
circle
Specify center point for circle or [3P/2P/Ttr]: 3.5,8
Specify radius of circle or [Diameter]:         d
Specify diameter of circle:                     2
```

The completed object is shown in the figure on the right.



X Program Help

If you need more information on the various options for drawing an arc, park the mouse over the icon and after a few seconds, a help message will pop up. The help message will stay on the screen for as long as the mouse is parked over the icon. If you want more help, you can press the F1 key and AutoCAD will open a web page where you can look up the command and read a more in depth description of how it works.

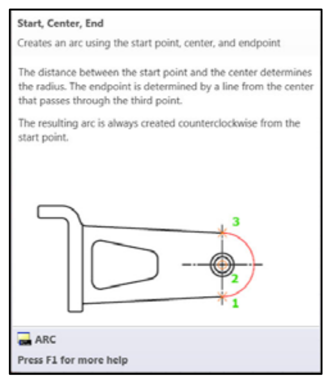


Figure 1 Help message produced when the mouse is parked over the arc command.

XI Undoing Mistakes

If you make a mistake with a command you can undo anything it has done by typing **U** at the command prompt. The entire effects of the last thing you typed will be undone and AutoCAD will return to the state it was in prior to the typing.

If you are inside a LINE or other command, you can enter **U** to undo the last coordinates entered or the last option selected.

You can undo the last command by holding down the Ctrl key and pressing Z. Each time you press Z, AutoCAD will remove the last command. If your press Z five times, it will backup through the last five commands.

XII Zooming in on Detail

You enlarge or reduce the size of an object on the screen with the ZOOM command. At the command prompt type:

ZOOM

and AutoCAD responds with:

```
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:
```

There are many options to the ZOOM command. Some of the more useful are:

`All`	*Change the scale so that all of the drawing is shown in the window.*
`Previous`	*ZOOM to the previous view of the drawing.*
`Scale`	Enter a scale factor in the form nX where n is a scaling factor and X is just the letter X. A number larger than 1 makes the drawing appear larger and a number smaller than 1 makes the drawing smaller. For example 2 makes the drawing twice as large as it currently is and .5 makes the drawing half its current size.
`Window`	Allows you to draw a window around the area of the drawing you want to see enlarged. The window is drawn by selecting opposite diagonal corners with the mouse.

You can start the ZOOM command by just typing **ZOOM** or **Z** at the command prompt.

A very easy to zoom in or out can be done with the mouse. Move the mouse till it is near a location you want to remain on the screen. Roll the wheel between the two mouse buttons forward to make the object larger and roll it backwards (towards you) to make the object on the screen smaller.

If you zoom in to make the object larger, the object you want to see can zoom off of the screen if you did not place your mouse close to it before you started zooming. If it does scroll off of the screen, move your mouse over until it is as close as you can get to the object, hold down on the wheel, and drag the screen over until you can see the object you want to see. This operation is called PANNING.

XIII Size of Drawing Area

The drawing area is as large as you need it to be. The usable drawing area does not just consist of the area that you can see. You can pan around the drawing area to reveal areas of your drawing that are out of view. You can also ZOOM in and out to reveal more or less of the drawing area. Because the drawing area is so large, it is a good idea to indicate the region that you wish to use. This is your drawing size or limits. This is usually the area that will be printed. You can change your drawing size using the LIMITS command.

Setting your drawing size type:

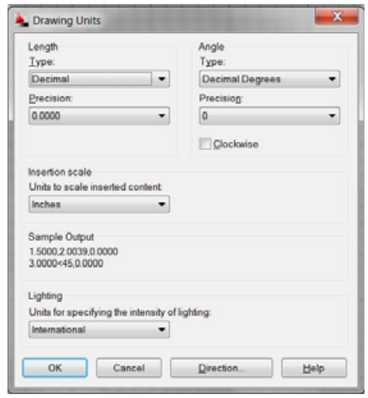
`limits`

`Specify lower left corner or [ON/OFF] <0.0000,0.0000>:` *{Enter the lower left corner of your limits.*

```
Specify upper right corner <420.0000,297.0000>:
```

Usually this should always remain 0,0.} {Enter the coordinates for the upper right corner of the drawing area you want to use. You can use the mouse to select the upper right corner of the limits.}

XIV Specifying the Units for the Drawing



The units (i.e. inches, millimeters, feet) used to draw objects in the drawing area can be selected using the UNITS command. Type UNITS then press enter. The following dialog box will appear allowing you to select the units and the number of decimals displayed in the commands.

XV Shortcuts

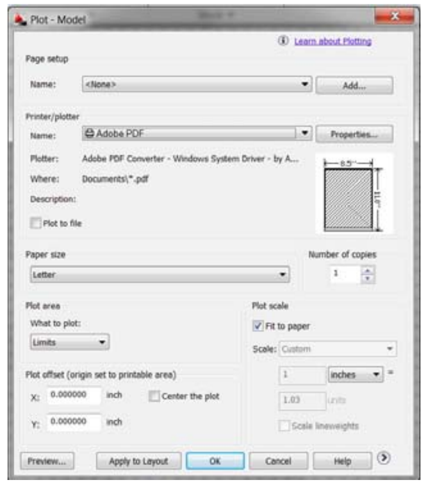
AutoCAD has many short cuts that make the work easier. One very useful one is pressing the ENTER key to repeat the last command.

XVI Saving Your Work

Periodically, you should save your work. You can do this by clicking on the "A" icon then selecting "Save" with your mouse. You can also save your work by typing Ctrl S (holding down the Ctrl key and pressing the S key.)

The first time you do this, AutoCAD produces a dialog box that allows you to select the file folder where you want the files saved and the name you want to use for the drawing. The next time you use the command, AutoCAD saves the file at the same location using the same name.

XVII Plotting



You can plot the drawings you have made by clicking on the “A” icon then selecting print. This creates the dialog box shown on the right.

Select the printer or plotter you are going to use then define the plotting area. If you set limits for the area you want to plot select Limits in the “What to plot:” pull down menu. If you did not define the limits, you can use Extents. The Extents will plot selection will plot everything you have drawn.

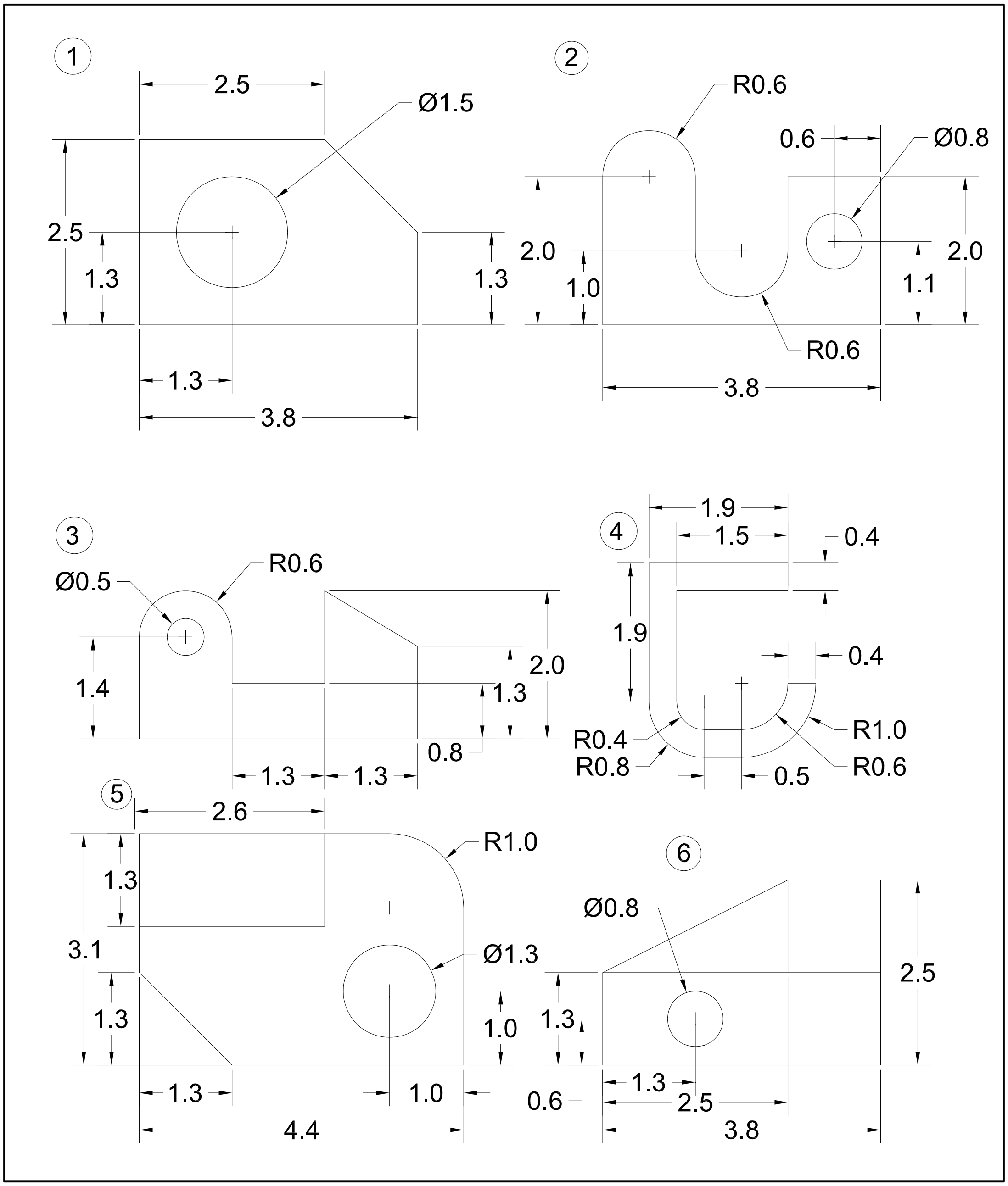
We are not plotting scaled drawings at this time so mark the “Fit to paper” box. This will stretch or shrink the drawing so that it fits on the paper.

Before you click on OK to plot the drawing, it is always a good idea to “Preview” to drawing. Click on this button to see what the drawing will look like on your sheet of paper. If it looks ok, you can click on the printer button in the preview screen. If it does not look ok, click on the circle with an X in it. This will return to the plot Model dialog box so you can make changes that will hopefully produce the plot you want. After you have made changes, be sure to preview it again to make sure it is working correctly.

XVIII Ending the Program

You terminate the program by clicking on “A” then “Exit AutoCAD” in the lower right corner of the pull down menu. The program asks you if you want to save the changes you have made to the drawing. You should click on the “Yes” button to save the changes.

Problems



1
2.5
Ø1.5
2.5
1.3
1.3
1.3
3.8
2
R0.6
0.6
Ø0.8
2.0
1.0
2.0
1.1
R0.6
3.8
3
Ø0.5
R0.6
1.4
2.0
1.3
0.8
1.3
1.3
4
1.9
1.5
0.4
1.9
0.4
R0.4
R0.8
R1.0
R0.6
0.5
5
2.6
R1.0
1.3
3.1
Ø1.3
1.3
1.0
1.3
1.0
4.4
6
Ø0.8
2.5
1.3
0.6
1.3
2.5
3.8