Drills

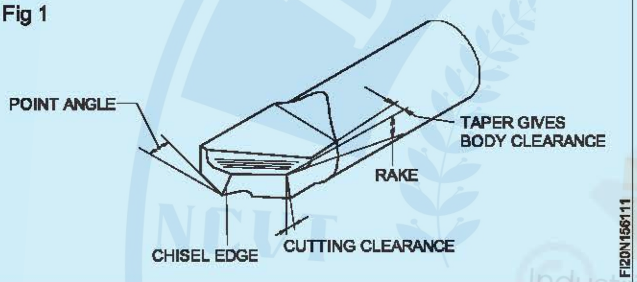
Objectives: At the end of this lesson you shall be able to
- **state drilling and drill material**
- **state the necessity of drilling**
- **name the types of drills used**
- **list the parts of a twist drill.**

Drilling: Drilling is the production of cylindrical holes of definite diameters in workpieces by using a multi-point cutting tool called a 'drill'. It is the first operation done internally for any further operation. The fluted part (or) body of a drill is made of either high carbon steel (or) High speed steel.

Types of drills and their specific uses

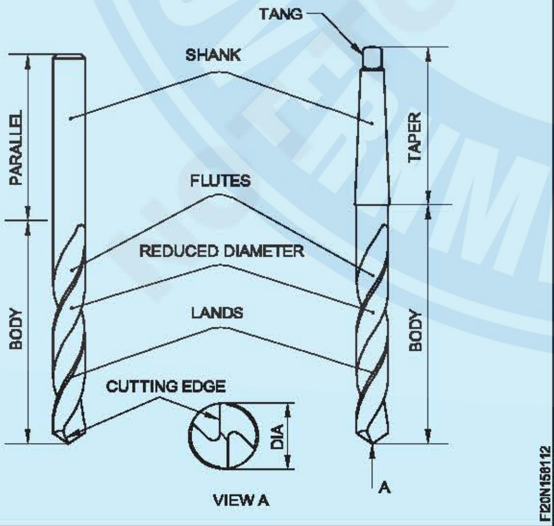
Flat drill (Fig 1) **:** The earliest form of drill was the flat drill which is easy to operate, besides being inexpensive to produce. The chip removal is poor and its operating efficiency is very low.

Fig 1



Twist drill : Almost all drilling operation is done using a twist drill. It is called a twist drill as it has two or more spiral or helical flutes formed along its length. The two basic types of twist drills are, parallel shank and taper shank. Parallel shank twist drills are available below 13mm size (Fig 2).

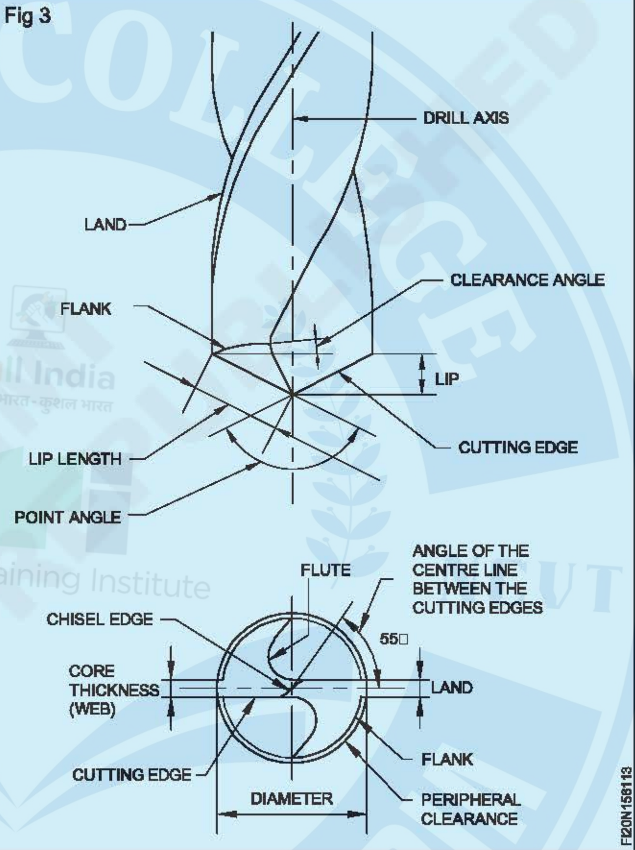
Fig 2



Parts of a twist drill : Drills are made out of high speed steel. The spiral flutes are machined at an angle of 27 1/2° to its axis.

The flutes provide a correct cutting angle which provides an escape path for the chips. It carries the coolant to the cutting edge during drilling. (Fig 3)

Fig 3



The portions left between the flutes are called 'lands'. The size of a drill is determined and governed by the diameter over the lands.

The point angle is the cutting angle, and for general purpose work, it is 118°. The clearance serves the purpose of clearing the back of the lip from fouling with the work. It is mostly 8°.

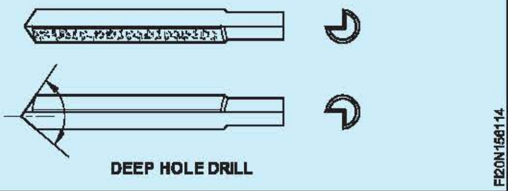
Deep hole drills

Deep hole drilling is done by using a type of drill known as 'D' bit (Fig 4)

Drills are made of high speed steel.

Drills are manufactured with varying helix angles for drilling different materials. General purpose drills have a standard helix angle of 27 1/2°. They are used on mild steel and cast iron. (Fig 5a)

Fig 4

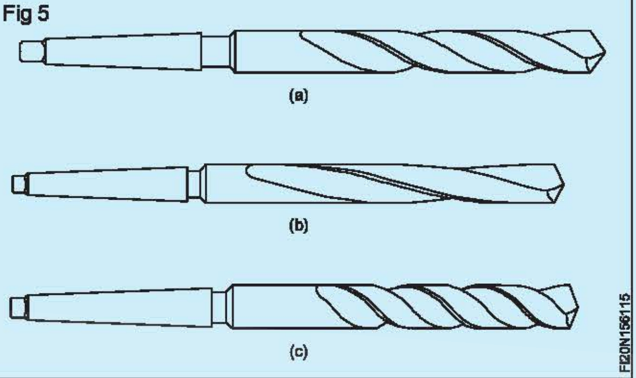


A slow helix drill is used on materials like brass, gun metal, phosphor-bronze and plastics. (Fig 5b)

A quick helix drill is used for copper, aluminium and other soft metals (Fig 5c)

A quick helix drill should never be used on brass as it will 'dig in' and the workpiece may be thrown from the machine table.

Fig 5

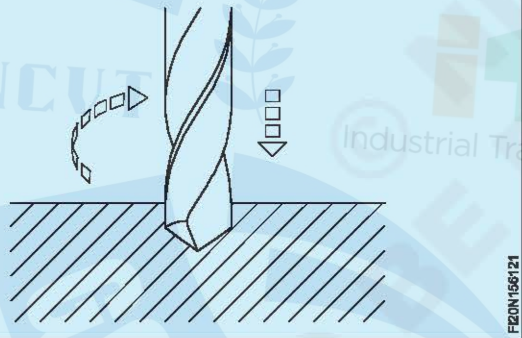


Drill (Parts and functions)

Objectives: At the end of this lesson you shall be able to
- **state the functions of drills**
- **identify the parts of a drill**
- **state the functions of each part of a drill.**

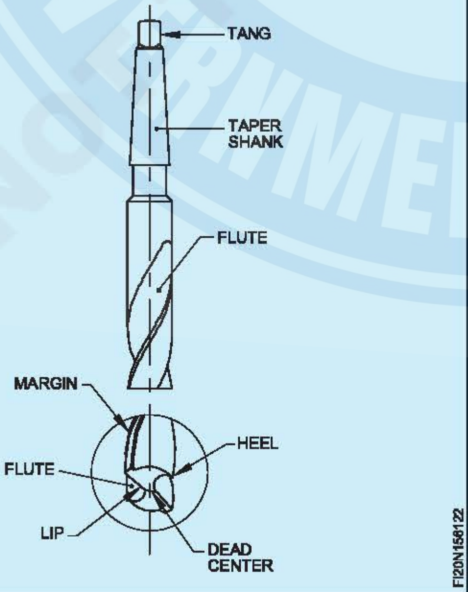
Drilling is a process of making holes on workpieces. The tool used is a drill. For drilling, the drill is rotated with a downward pressure causing the tool to penetrate into the material. (Fig 1)

Fig 1



Parts of a Drill (Fig 2)

Fig 2



The various parts of a drill can be identified from figure 2.

Point

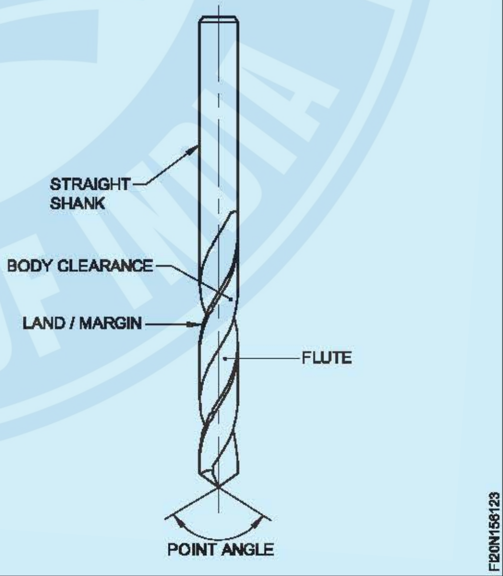
The cone shaped end which does the cutting is called the point. It consists of a dead centre, lips or cutting edges, and a heel.

Shank

This is the driving end of the drill which is fitted on to the machine. Shanks are of two types.

Taper shank, used for larger diameter drills, and straight shank, used for smaller diameter drills. (Fig 3)

Fig 3



Tang

This is a part of the taper shank drill which fits into the slot of the drilling machine spindle.

Body

The portion between the point and the shank is called the body of a drill.

The parts of the body are flute, land/margin, body clearance and web.

Flutes (Fig 3)

Flutes are the spiral grooves which run to the length of the drill. The flutes help

- To form the cutting edges
- To curl the chips and allow these to come out
- The coolant to flow to the cutting edge.

Land/Margin (Fig 3)

The land/margin is the narrow strip which extends to the entire length of the flutes.

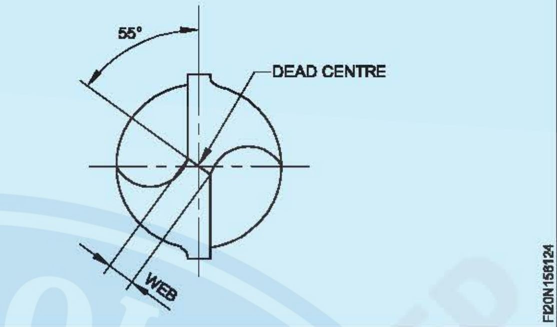
The diameter of the drill is measured across the land/ margin.

Body clearance (Fig 3)

Body clearance is the part of the body which is reduced in diameter to cut down the friction between the drill and the hole being drilled.

Web (Fig 4)

Fig 4



Web is the metal column which separates the flutes. It gradually increases in thickness towards the shank.

Drill angles

Objectives: At the end of this lesson you shall be able to
- **list the various angles of a twist drill**
- **state the functions of each angle**
- **list the types of helix for drills as per ISI**
- **distinguish the features of different types of drills**
- **designate drills as per ISI recommendations.**

Like all cutting tools the drills are provided with certain angles for efficiency in drilling.

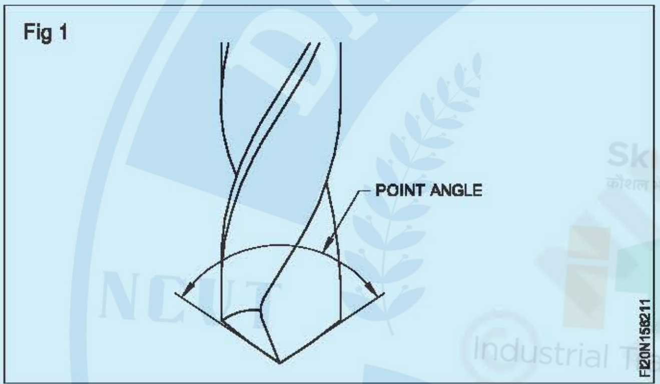
Drill angles

They are different angles for different purposes. They are listed below.

Point angle, helix angle, rake angle, clearance angle and chisel edge angle.

Point angle/ cutting angle (Fig 1)

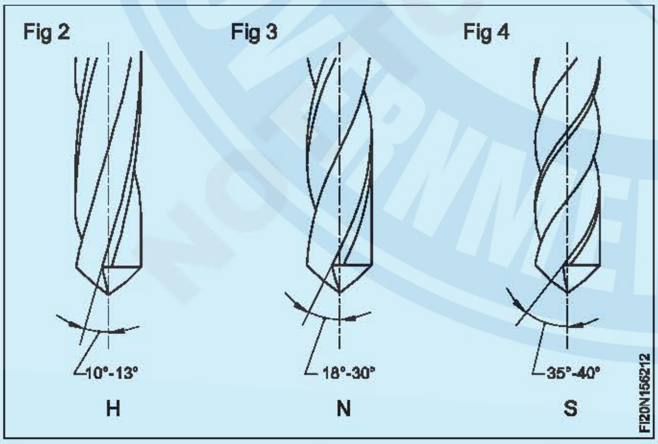
Fig 1



The point angle of a general purpose (standard) drill is 118°. This is the angle between the cutting edges (lips). The angle varies according to the hardness of the material to be drilled. (Fig 1)

Helix angle (Figs 2,3 and 4)

Fig 2 Fig 3 Fig 4



Twist drills are made with different helix angles. The helix angle determines the rake angle at the cutting edge of the twist drill.

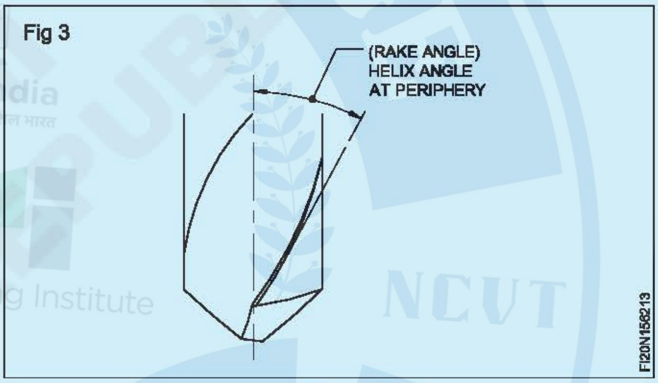
The helix angles vary according to the material being drilled. According to indian standards, three types of drills are used for drilling various materials.

- Type N - For normal low carbon steel.
- Type H - For hard and tenacious materials.
- Types S - For soft and tough materials.

The type of drill used for general purpose drilling work is type N.

Rake angle (Fig 5)

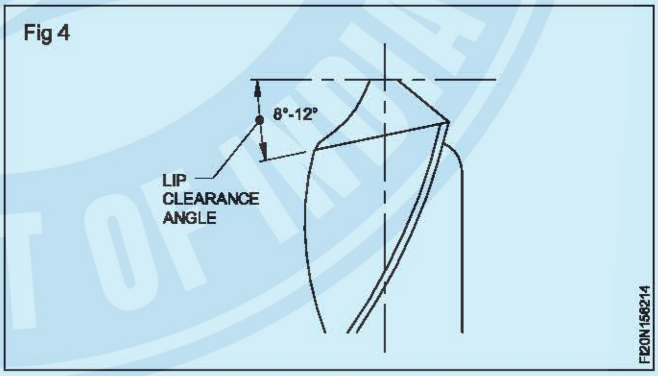
Fig 3



Rake angle is the angle of flute (helix angle).

Clearance angle (Fig 6)

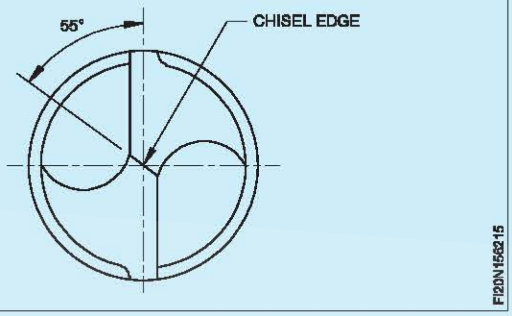
Fig 4



The clearance angle is meant to prevent the friction of the tool behind the cutting edge. This will help in the penetration of the cutting edges into the material. If the clearance angle is too much, the cutting edges will be weak, and if it is too small, the drill will not cut.

Chisel edge angle/web angle (Fig 7)

Fig 5



This is the angle between the chisel edge and the cutting lip.

Designation of drills

Twist drills are designated by the

- Diameter
- Tool type
- Material

Example

A twist drill of 9.50 mm dia. of tool type 'H' for right hand cutting and made from HSS is designated as:

Twist drill 9.50 - H - IS5101 - HS

where H = tool type

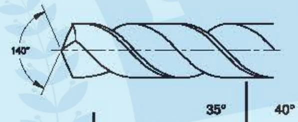
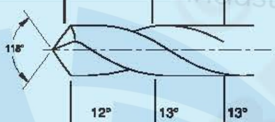
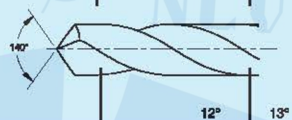
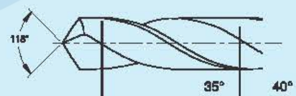
IS5101 = IS Number

HS = tool material

9.5 = diameter of the drill.

If the tool type is not indicated in the designation, it should be taken as type 'N' tool.

DRILLS FOR DIFFERENT MATERIALS

Recommended drills					
Material to be drilled	**Point angle**	**Helix angle** d=3.2-5 5-10 10-	**Material to be drilled**	**Point angle**	**Helix angle** d=3.5-5 5-
Steel and cast steel up to 70 kgf/mm^2 strength Gray cast iron Malleable cast iron Brass German silver, nickel.			Copper (up to 30 mm drill diameter) Al-alloys, forming curly chips Celluloid		
Brass, CuZn 40			Austenitic steels Magnesium alloys		
Steel and cast steel 70...120 Kgf/mm^2			Moulded plastics (with thickness s>d)		
Stainless steel; Copper (drill diameter more than 30 mm) Al-alloy, forming short broken chips			Moulded plastics, with thickness s<d Laminated plastics, hard rubber (ebonite) marble, slate, coal		
			Zinc alloys		

Drilling - Cutting speed, feed and r.p.m , drill holding devices

Objectives: At the end of this lesson you shall be able to
- **define cutting speed**
- **state the factors for determining the cutting speed**
- **determine r.p.m/spindle speed.**

Cutting speed is the speed at which the cutting edge passes over the material while cutting, and is expressed in metres per minute.

Cutting speed is also sometimes stated as surface speed or peripheral speed.

The selection of the recommended cutting speed for drilling depends on the materials to be drilled, and the tool material.

Tool manufacturers usually provide a table of cutting speeds required for different materials.

The recommended cutting speeds for different materials are given in the Table 1. Based on the cutting speed recommended, the r.p.m, at which a drill has to be driven is determined.

TABLE 1

Recommended cutting speeds

Materials being drilled (HSS Tool)	
Aluminium	70 - 100
Brass	35 - 50
Bronze(phosphor)	20 - 35
Cast iron (grey)	25 - 40
Copper	35 - 45
Steel (medium carbon/mild steel)	20 - 30
Steel (alloy, high tensile)	5 - 8
Thermosetting plastic (low speed due to abrasive properties)	20 - 30

Cutting speed calculation

Cutting speed (V) $\pi \times d \times h$

$$r.p.m(n) = \frac{V \times 1000}{d \times \pi}$$

n - r.p.m.

v - Cutting speed in m/min.

d - diameter of the drill in mm.

π = 3.14

Examples

Calculate the r.p.m for a high speed steel drill Ø 24 to cut mild steel.

The cutting speed for mild steel is taken as 30 m/min from the table.

$$n = \frac{1000 \times 30}{3.14 \times 24} = 398 \text{ r.p.m}$$

It is always preferable to set the spindle speed to the nearest available lower range.

The r.p.m. will differ according to the diameter of the drills. The cutting speed being the same, larger diameter drills will have lesser r.p.m and smaller diameter drills will have higher r.p.m.

The recommended cutting speeds are achieved only by actual experiment.

Feed in drilling

Objectives: At the end of this lesson you shall be able to
- **state what is meant by feed**
- **state the factors that contribute to an efficient feed rate.**

Feed is the distance a drill advances into the work in one complete rotation. (Fig 1)

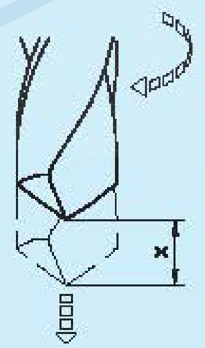
Feed is expressed in hundredths of a millimeter.

Example - 0.040mm / rev

The rate of feed is dependent up on a number of factors.

- The finish required
- Type of drill (drill material)
- Material to be drilled

Fig 1



Factors like rigidity of the machine, holding of the work-piece and the drill, will also have to be considered while determining the feed rate. If these are not to the required standard, the feed rate will have to be decreased.

It is not possible to suggest a particular feed rate taking all the factors into account.

The table gives the feed rate which is based on the average feed values suggested by the different manufacturers of drills. (Table 1)

Too coarse a feed may result in damage to the cutting edges or breakage of the drill.

Too slow a rate of feed will not bring improvement in surface finish but may cause excessive wear of the tool point, and lead to chattering of the drill.

For optimum results in the feed rate while drilling, it is necessary to ensure the drill cutting edges are sharp. Use the correct type of cutting fluid.

TABLE 1

Drill diameter (mm) H.S.S	Rate of feed (mm/rev)
1.0 - 2.5	0.040 - 0.060
2.6 - 4.5	0.050 - 0.100
4.6 - 6.0	0. 075 - 0.150
6.1 - 9.0	0.100 - 0.200
9.1 - 12.0	0.150 - 0.250
12.1 - 15.0	0.200 - 0.300
15.1 - 18.0	0.230 - 0.330
18.1 - 21.0	0.260 - 0.360
21.1 - 25.0	0.280 - 0.380

Cutting Tool	Mild Steel	Carbon steel	Aluminium	Brass	Cast iron	Stainless steel
HSS	100	80	250 to 350	175	100	80 to 100
Carbide	300	200	750 to 1000	500	250	200 to 250

Drill-holding devices

Objectives: At the end of this lesson you shall be able to
- **name the different types of drill-holding devices**
- **state the features of drill chucks**
- **state the functions of drill sleeves**
- **state the function of drift.**

For drilling holes on materials, the drills are to be held accurately and rigidly on the machines.

The common drill-holding devices are drill chucks, sleeves and sockets.

Drill chucks: Straight shank drills are held in drill chucks. (Fig 1A) For fixing and removing drills, the chucks are provided either with a pinion and key or a knurled ring.

The drill chucks are held on the machine spindle by means of an arbor (Fig 1B) fitted on the drill chuck.

Taper sleeves and sockets (Fig 2): Taper shank drills have a Morse taper.

Sleeves and sockets are made with the same taper so that the taper shank of the drill, when engaged, will give a good wedging action. Due to this reason Morse tapers are called self-holding tapers.

The drills are provided with five different sizes of Morse tapers, and are numbered from MT 1 to MT 5.

In order to make up the difference in sizes between the shanks of the drills and the bore of machine spindles, sleeves of different sizes are used. When the drill taper shank is bigger than the machine spindle, taper sockets are used. (Fig 2)

Fig 1

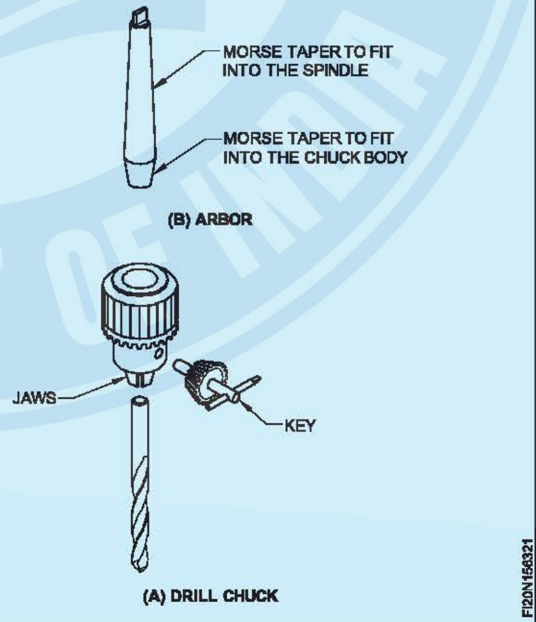
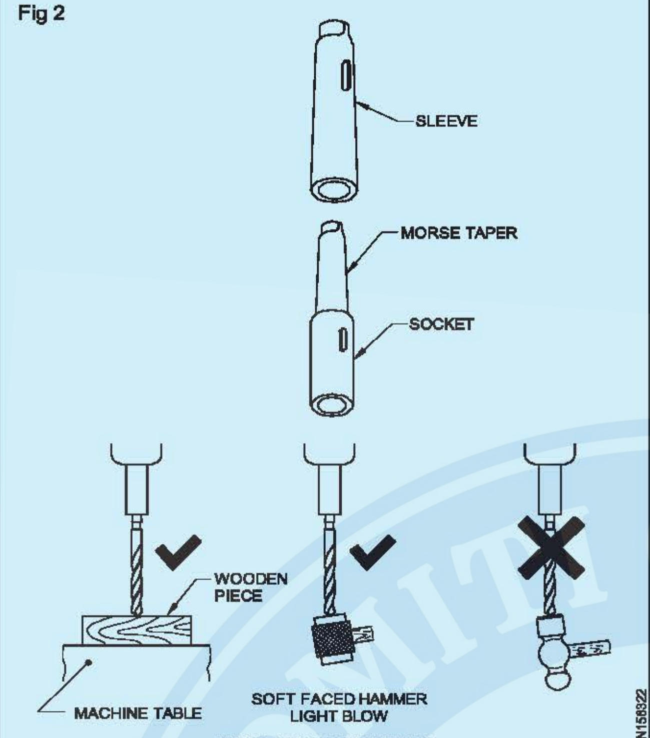


Fig 2



FIXING DRILLS ON SPINDLE

While fixing the drill in a socket or sleeve, the tang portion should align in the slot. This will facilitate the removal of the drill or sleeve from the machine spindle.

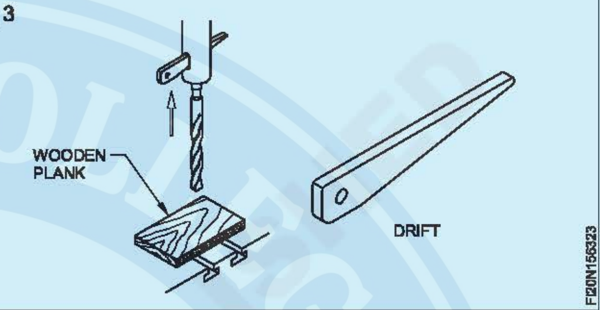
Use a drift to remove drills and sockets from the machine spindle. (Fig 3)

While removing the drill from the sockets/sleeves don't allow it to fall on the table or jobs.

Drill chucks are made from special alloy steel

Drill sleeves are made from case hardened steel

Fig 3



— — — — — — —

Counter sinking

Objectives: At the end of this lesson you shall be able to
- **What is countersinking**
- **list the purposes of countersinking**
- **state the angles of countersinking for the different applications**
- **name the different types of countersinks**
- **distinguish between Type A and Type B counter sink holes.**

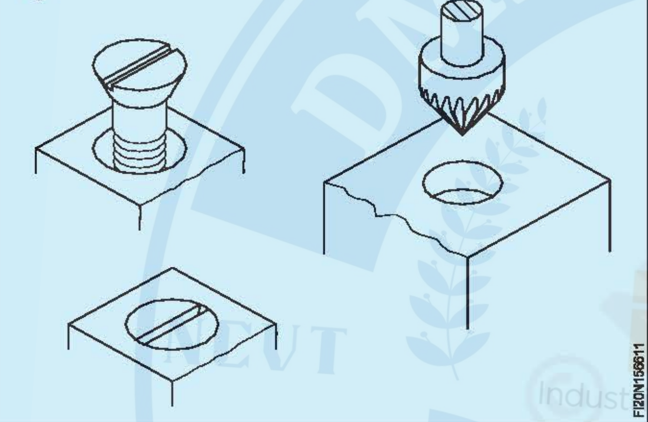
What is countersinking?

Countersinking is an operation of beveling the end of a drilled hole. The tool used is called a countersink.

Countersinking is carried out for the following purposes:

- To provide a recess for the head of a countersink screw, so that it is flush with the surface after fixing (Fig 1)

Fig 1



- To deburr a hole after drilling
- For accommodating countersink rivet heads
- To chamfer the ends of holes for thread cutting and other machining processes.

Angles for countersinking

Countersinks are available in different angles for different uses.

75° countersink riveting

80° countersink self tapping screws

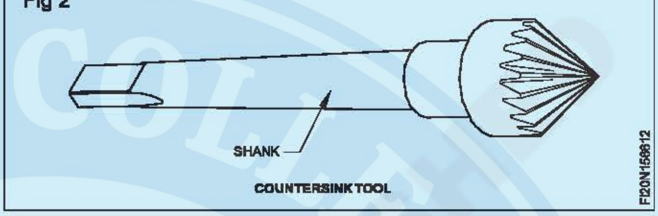
90° countersink head screws and deburring

120° chamfering ends of holes to be threaded or other machining processes.

Countersinks: Countersinks of different types are available.

The commonly used countersinks have multiple cutting edges and are available in taper shank and straight shank. (Fig 2)

Fig 2

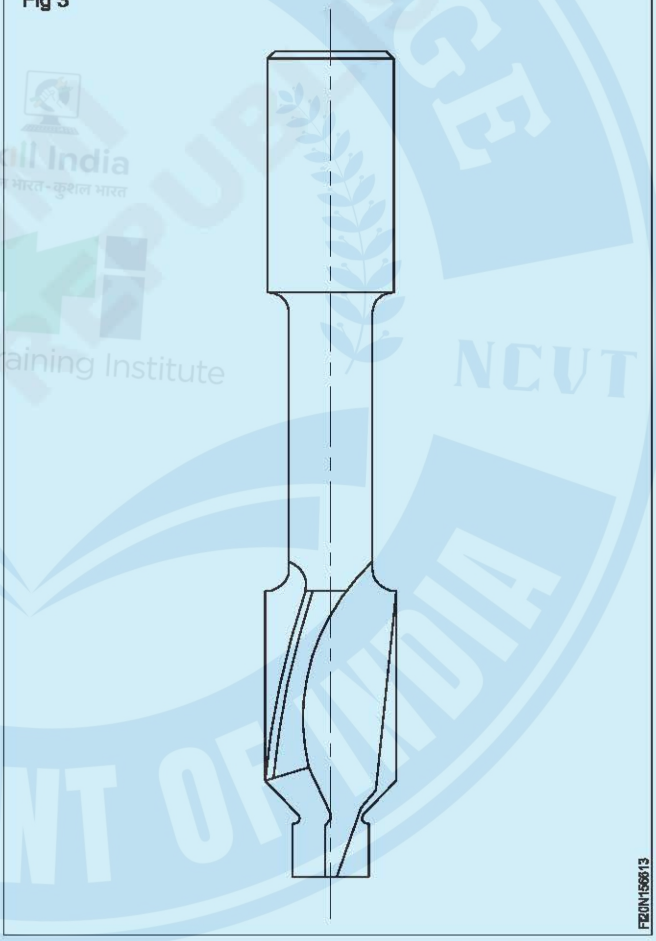


For countersinking small diameter holes special countersinks with two or one flute are available. This will reduce the vibration while cutting.

Countersinks with Pilot (Fig 3)

For precision countersinking, needed for machine tool assembling and after machining process, countersinks with pilots are used.

Fig 3



They are particularly useful for heavy duty work.

The pilot is provided at the end for guiding the countersink concentric to the hole.

Countersinks with pilots are available with interchangeable and solid pilots.

Countersink hole sizes: The countersink holes according to Indian Standard IS 3406 (Part 1) 1986 are of four types: Type A, Type B, Type C and Type E.

Type A is suitable for slotted countersink head screws, cross recessed and slotted raised countersink head screws.

These screws are available in two grades i.e. medium and fine.

The dimensions of various features of the Type 'A' countersink holes, and the method of designation are given in Table 1. (Fig 4 & 5)

Type 'B' countersink holes are suitable for countersink head screws with hexagon socket.

The dimensions of the various features and the method of designation are given in Table II. (Fig 6)

Type 'C' countersink holes are suitable for slotted raised countersink (oval) head tapping screws and for slotted countersink (flat) head tapping screws.

The dimension of the various features and the method of designation are given in Table III. (Fig 7)

Type 'E' countersinks are used for slotted countersink bolts used for steel structures.

The dimensions of the various features and the method of designation are given in Table IV. (Fig 8)

Table I

Dimensions and designation of countersink - Type A according to IS 3406 (Part 1) 1986

FIG 4

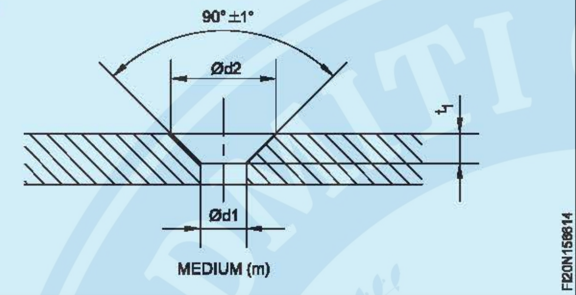


Fig 5

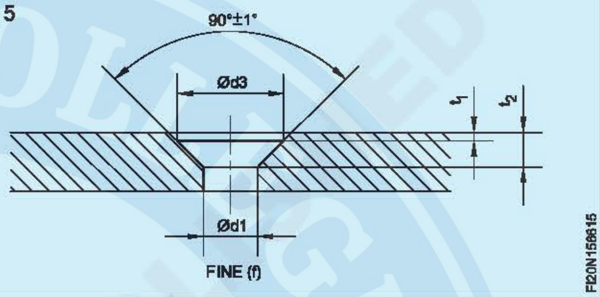


Table I

For Nominal Size		1	1.2	(1.4)	1.6	(1.8)	2	2.5	3	3.5	4	(4.5)
Medium	d_1 H13	1.2	1.4	1.6	1.8	2.1	2.4	2.9	3.4	3.9	4.5	5
Series	d_2 H13	2.4	2.8	3.3	3.7	4.1	4.6	5.7	6.5	7.6	8.6	9.5
(m)	$t_1{}^3$	0.6	0.7	0.8	0.9	1	1.1	1.4	1.6	1.9	2.1	2.3
Fine	d_1 H12	1.1	1.3	1.5	1.7	2	2.2	2.7	3.2	3.7	4.3	4.8
Series	d_3 H12	2	2.5	2.8	3.3	3.8	4.3	5	6	7	8	9
(f)	$t_1{}^3$	0.7	0.8	0.9	1	1.2	1.2	1.5	1.7	2	2.2	2.4
	t_2 + 0.1 0	0.2	0.15	0.15	0.2	0.2	0.15	0.35	0.25	0.3	0.3	0.3

For Nominal Size		5	6	8	10	12	(14)	16	(18)	20
Medium	d_1 H13	5.5	6.6	9	11	13.5	15.5	17.5	20	22
Series	d_2 H13	10.4	12.4	16.4	20.4	23.9	26.9	31.9	36.4	40.4
(m)	$t_1{}^3$	2.5	2.9	3.7	4.7	5.2	5.7	7.2	8.2	9.2
Fine	d_1 H12	5.3	6.4	8.4	10.5	13	15	17	19	21
Series	d_3 H12	10	11.5	15	19	23	26	30	34	37
(f)	$t_1{}^3$	2.6	3	4	5	5.7	6.2	7.7	8.7	9.7
	t_2 + 0.1 0	0.2	0.45	0.7	0.7	0.7	0.7	1.2	1.2	1.7

Note 1 : **Size shown in brackets are of second preference.**

Note 2 : Clearance hole d_1 according to medium and fine series of IS : 1821 ' Dimensions for clearance holes for bolts and screws (second revision)'

Designation : A countersink Type A with clearance hole of fine (f) series and having nominal size 10 shall be designated as – Countersink A f 10 - IS : 3406.

Table II

Dimensions and designation of countersink - Type B according to IS 3406 (Part 1) 1986

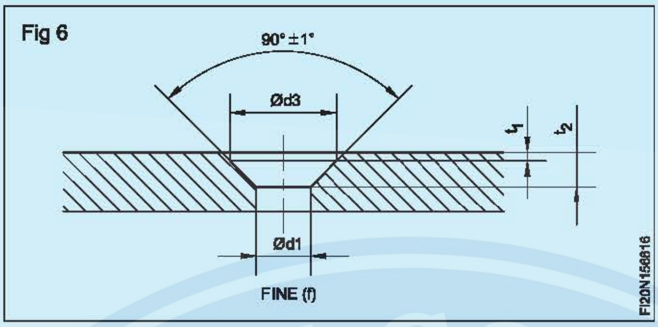


Fig 6

For Nominal Size		**3**	**4**	**5**	**6**	**8**	**10**	**12**	**(14)**	**16**	**(18)**	**20**	**22 24**
Fine	d1 H12	3.2	4.3	5.3	6.4	8.4	10.5	13	15	17	19	21	23 25
Series	d2 H12	6.3	8.3	10.4	12.4	16.5	20.5	25	28	31	34	37	48.2 52
(f)	t_1 ³	1.7	2.4	2.9	3.3	4.4	5.5	6.5	7	7.5	8	8.5	13.1 14
	t_2 + 0.1	0.2	0.3			0.4	0.5						1

Note 1: Sizes shown in brackets are of second preference.
Note 2: Clearance hole d1 according to medium and fine series of IS : 1821- 1982.

Designation : A countersink Type A with clearance hole of fine (f) series and having nominal size 10 shall be designated as – Countersink A f 10 - IS : 3406.

Table III

Dimensions and designation of countersink - Type C according to IS 3406 (Part 1) 1986

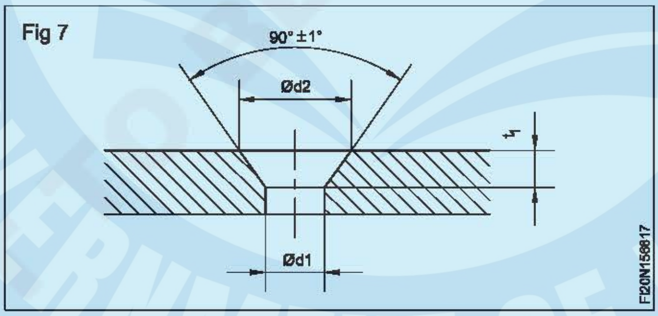


Fig 7

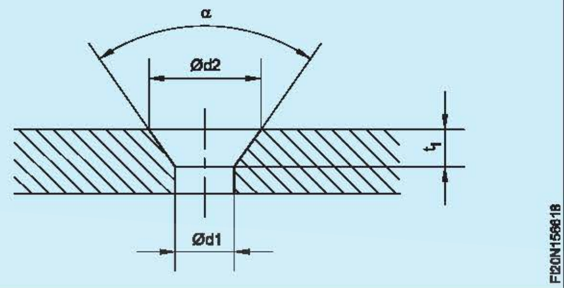
For Screw Size No.	**(0)**	**(1)**	**2**	**(3)**	**4**	**(5)**	**6**	**(7)**	**8**	**10**	**(12)**	**14**	**(16)**
d1 H12	1.6	2	2.4	2.8	3.1	3.5	3.7	4.2	4.5	5.1	5.8	6.7	8.4
d2 H12	3.1	3.8	4.6	5.2	5.9	6.6	7.2	8.1	8.7	10.1	11.4	13.2	16.6
t_1 ³	0.9	1.1	1.3	1.5	1.7	1.9	2.1	2.3	2.6	3	3.4	3.9	4.9

Note : Sizes given in brackets are of second preference.

Designation : A countersink Type C for screw size 2 shall be designated as – Countersink C 2 - IS : 3406.

Table IV

Fig 8



Dimension and designation of countersink - Type E according to IS 3406 (Part 1) 1986

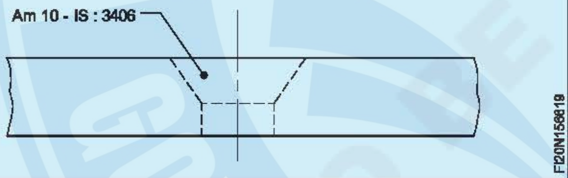
For Nominal No.	10	12	16	20	22	24
d_1 H12	10.5	13	17	21	23	25
d_2 H12	19	24	31	34	37	40
t_1 [3]	5.5	7	9	11.5	12	13
$\alpha \pm 1°$		75°			60°	
Note: Clearance hole d_1 according to fine series of IS : 1821 - 1982						

Designation : A countersink Type E for nominal size 10 shall be designated as – Countersink E 10 - IS : 3406.

Methods of representing countersink holes in drawings

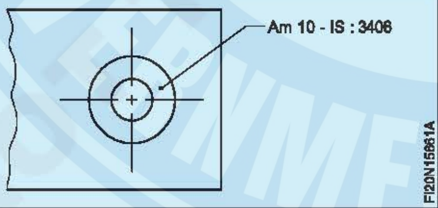
Countersink hole sizes are identified by code designation or using dimension. (Fig 9 - 12)

Fig 9



Use of code designation

Fig 10



Use of dimension

The dimension of the countersink can be expressed by the diameter of the countersink and the depth of the countersink.

Fig 11

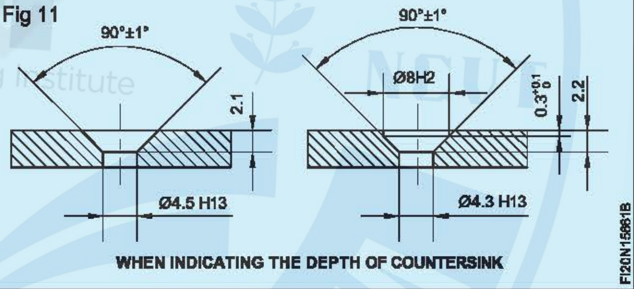
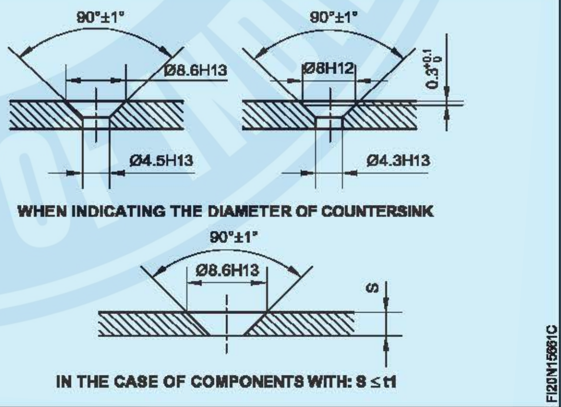


Fig 12



Counterboring and spot facing

Objectives: At the end of this lesson you shall be able to
- **differentiate counterboring and spot facing**
- **state the types of counterbores and their uses**
- **determine the correct counterbore sizes for different holes.**

Counterboring

Counterboring is an operation of enlarging a hole to a given depth, to house heads of socket heads or cap screws with the help of a counterbore tool. (Fig 1)

Fig 1

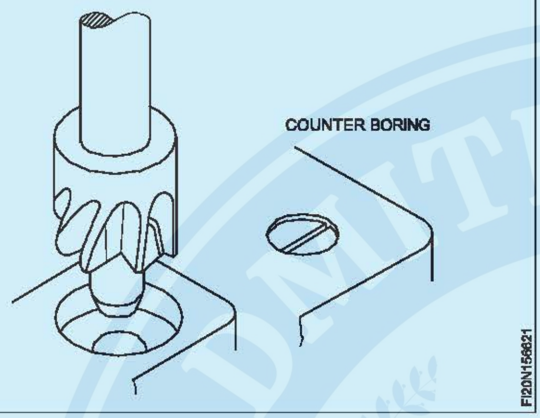
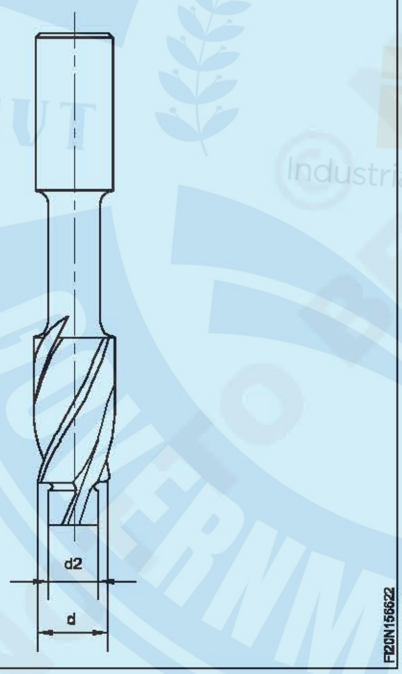


Fig 2



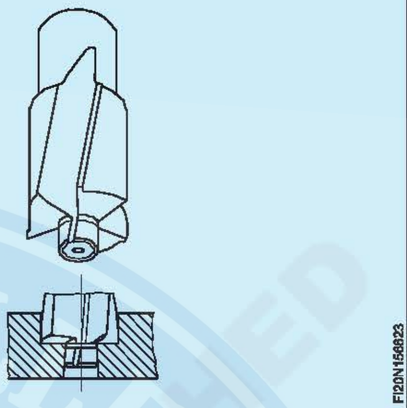
Counterbore (Tool)

The tool used for counterboring is called a counterbore. (Fig 2) Counterbores will have two or more cutting edges.

At the cutting end, a pilot is provided to guide the tool concentric to the previously drilled hole. The pilot also helps to avoid chattering while counterboring. (Fig 3)

Counterbores are available with solid pilots or with interchangeable pilots. The interchangeable pilot provides flexibility of counterboring on different diameters of holes.

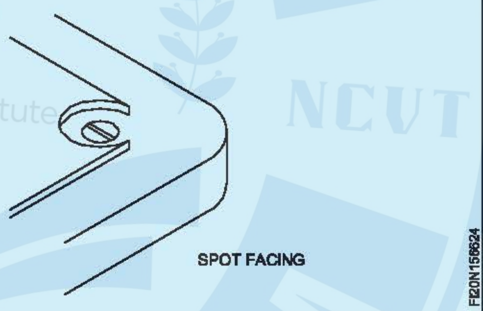
Fig 3



Spot facing

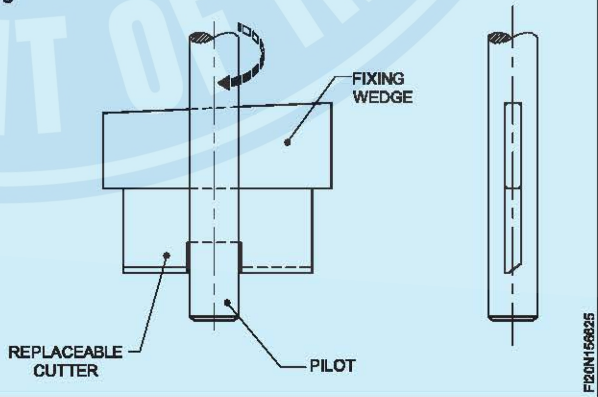
Spot facing is a machining operation for producing a flat seat for bolt head, washer or nut at the opening of a drilled hole. The tool is called a spot facer or a spot facing tool. Spot facing is similar to counterboring, except that it is shallower. Tools that are used for counterboring can be used for spot facing as well. (Fig 4)

Fig 4



Spot facing is also done by fly cutters by end-cutting action. The cutter blade is inserted in the slot of the holder, which can be mounted on to the spindle. (Fig 5)

Fig 5



Counterbore sizes and specification

Counterbore sizes are standardised for each diameter of screws as per BIS.

There are two main types of counterbores. Type H and Type K.

The type H counterbores are used for assemblies with slotted cheese head, slotted pan head and cross recessed pan head screws. The type K counterbores are used in assemblies with hexagonal socket head capscrews.

For fitting different types of washers the counterbore standards are different in Type H and Type K.

The clearance hole d1 are of two different grades i.e. medium (m) and fine (f) and are finished to H13 and H12 dimensions.

The table given below is a portion from IS 3406 (Part 2) 1986. This gives dimensions for Type H and Type K counterbores.

Counterbore and Clearance Hole Sizes for Different Sizes of Screws

Dimensions for H and K Type counter bores

While representing counterbores in drawings, counterbores can be indicated either by code designation or using the dimensions.

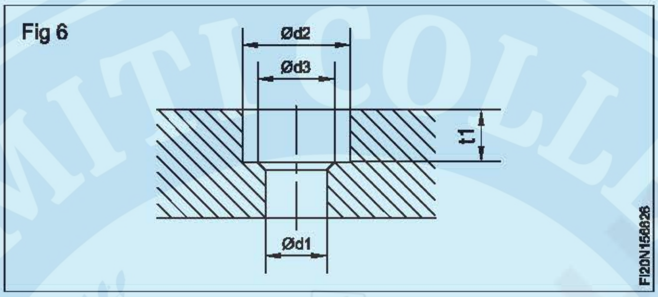


Fig 6

Table - 1

	For Nominal size	1	1.2	1.4	1.6	1.8	2	2.5	3	(3.5)	4	5	6	8	10	12	(14)	16	18	20	22	24	27	30	33	36
d1	Medium (m) H13	1.2	1.4	1.6	1.8	2.1	2.4	2.9	3.4	3.9	4.5	5.5	6.6	9	11	13.5	15.5	17.5	20	22	24	26	30	33	36	39
	fine (f) H12	1.1	1.3	1.5	1.7	2	2.2	2.7	3.2	3.7	4.3	5.3	6.4	8.4	10.5	13	15	17	19	21	23	25	-	-	-	-
	d2 H13	2.2	2.5	2.8	3.3	3.8	4.3	5	6	6.5	8	10	11	15	18	20	24	26	30	33	36	40	43	48	53	57
	d3	-	-	-	-	-	-	-	-	-	-	-	-	-	-	15.5	17.5	19.5	22	24	26	28	33	36	39	42
t1	Type H	0.8	0.9	1	1.2	1.5	1.6	2	2.4	2.9	3.2	4	4.7	6	7	8	9	10.5	11.5	12.5	13.5	14.5	-	-	-	-
	Type K	-	-	1.6	1.8	-	2.3	2.9	3.4	-	4.6	5.7	6.8	9	11	13	15	17.5	19.5	21.5	23.5	25.5	28.5	32	35	38
	Tolerances		+0.1 / 0					+0.2 / 0								+0.4 / 0						+0.6 / 0				

Note : Sizes given in brackets are of second preference. For details refer IS : 3406 (Part2) 1986.

Using code designation (Fig 7)

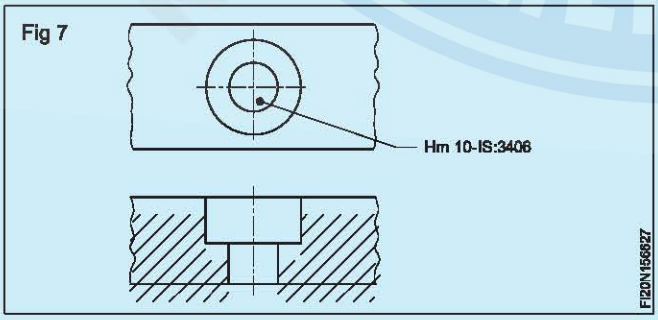


Fig 7

Using dimensions (Fig 8)

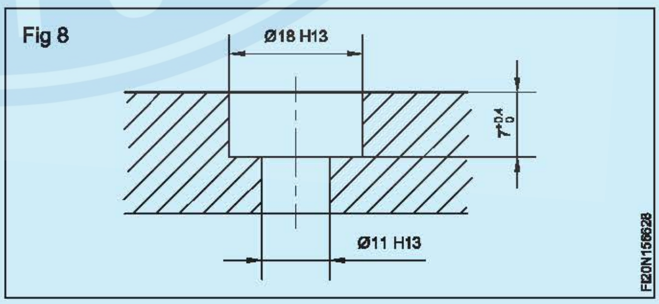


Fig 8

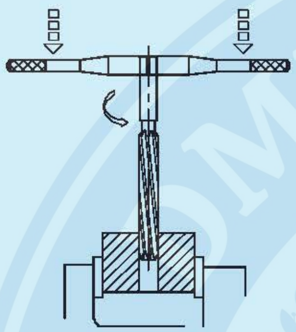
Reamers

Objectives: At the end of this lesson you shall be able to
- **state the use of reamers**
- **state the advantages of reaming**
- **distinguish between hand and machine reaming**
- **name the elements of a reamer and state their functions.**

What is a reamer?

A reamer is a multipoint cutting tool used for enlarging by finishing previously drilled holes to accurate sizes. (Fig 1)

Fig 1



F120N156711

Advantages of 'reaming'

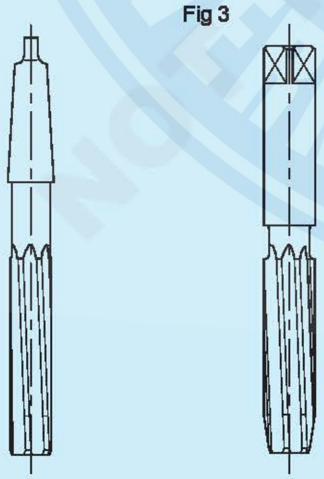
Reaming produces

- High quality surface finish
- Dimensional accuracy to close limits.
- Also small holes which cannot be finished by other processes can be finished.

Classification of reamers

Reamers are classified as hand reamers and machine reamers. (Figs 2a and 2b)

Fig 2 Fig 3



F120N156712

Reaming by using hand reamers is done manually for which great skill is needed.

Machine reamers are fitted on spindles of machine tools and rotated for reaming.

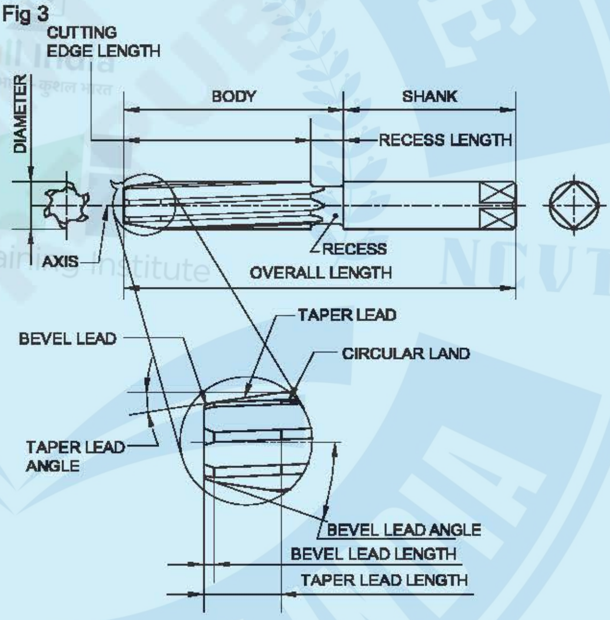
Machine reamers are provided with mores taper shanks for holding on machine spindles.

Hand reamers have straight shanks with 'square' at the end, for holding with tap wrenches. (Figs 2 (a) and (b)

Parts of a hand reamer

The parts of a hand reamer are listed hereunder. Refer to Fig 3.

Fig 3



F120N156713

Axis: The longitudinal centre line of the reamer.

Body: The portion of the reamer extending from the entering end of the reamer to the commencement of the shank.

Recess: The portion of the body which is reduced in diameter below the cutting edges, pilot or guide diameters.

Shank: The portion of the reamer which is held and driven. It can be parallel or taper.

Circular land: The cylindrically ground surface adjacent to the cutting edge on the leading edge of the land.

Bevel lead: The bevel lead cutting portion at the entering end of the reamer cutting its way into the hole. It is not provided with a circular land.

Taper lead: The tapered cutting portion at the entering end to facilitate cutting and finishing of the hole. It is not provided with a circular land.

Bevel lead angle: The angle formed by the cutting edges of the bevel lead and the reamer axis.

Taper lead angle: The angle formed by the cutting edges of the taper and the reamer axis.

Terms relating to cutting geometry

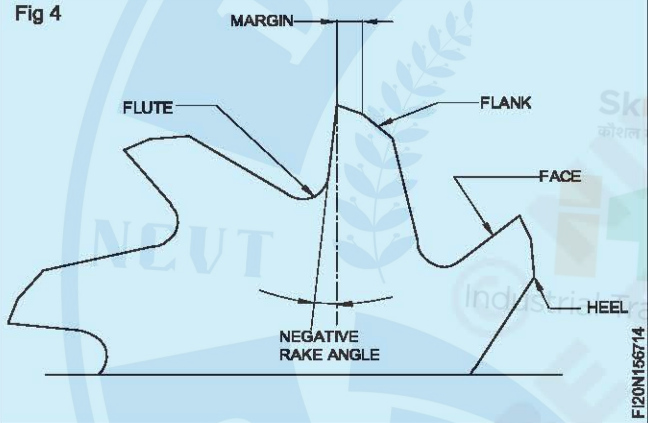
Flutes: The grooves in the body of the reamer to provide cutting edges, to permit the removal of chips, and to allow the cutting fluid to reach the cutting edges. (Fig 4)

Heel: The edge formed by the intersection of the surface left by the provision of a secondary clearance and the flute. (Fig 4)

Cutting edge: The edge formed by the intersection of the face and the circular land or the surface left by the provision of primary clearance. (Fig 4)

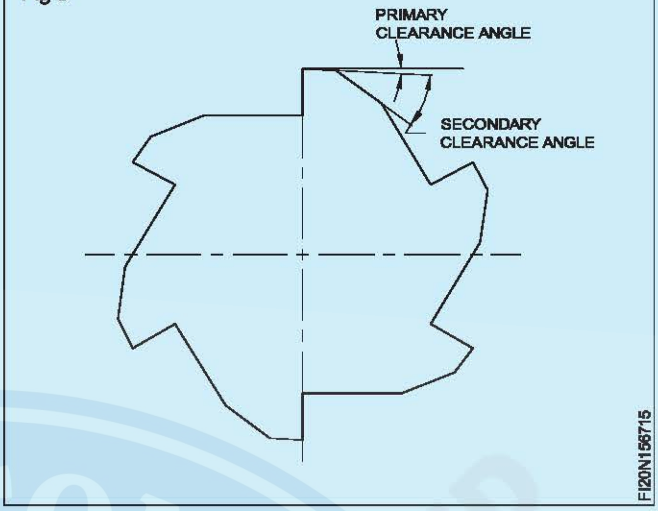
Face: The portion of the flute surface adjacent to the cutting edge on which the chip impinges as it is cut from the work. (Fig 4)

Fig 4



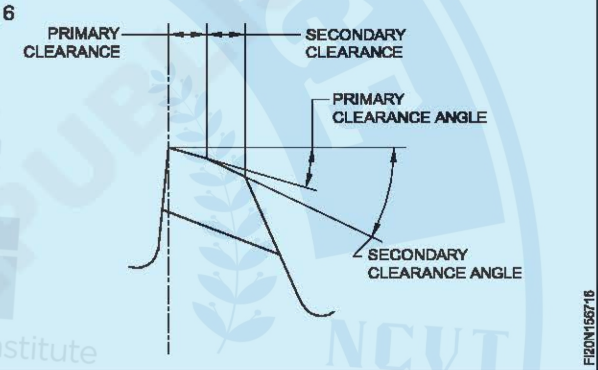
Rake angles: The angles in a diametric plane formed by the face and a radial line from the cutting edge. (Fig 5)

Fig 5



Clearance angle: The angles formed by the primary or secondary clearances and the tangent to the periphery of the reamer at the cutting edge. They are called primary clearance angle and secondary clearance angle respectively. (Fig 6)

Fig 6



Helix angle: The angle between the edge and the reamer axis. (Fig 7)

Fig 7

Hand reamers

Objectives: At the end of this lesson you shall be able to
- **state the general features of hand reamers**
- **identify the types of hand reamers**
- **distinguish between the uses of straight fluted and helical fluted reamers**
- **name the materials from which reamers are made and specify reamers.**

General features of hand reamers (Fig 1)

Hand reamers are used to ream holes manually using tap wrenches.

These reamers have a long taper lead. (Fig 2) This allows to start the reamer straight and in alignment with the hole being reamed.

Most hand reamers are for right hand cutting.

Helical fluted hand reamers have left hand helix. The left hand helix will produce smooth cutting action and finish.

Most reamers, machine or hand, have uneven spacing of teeth. This feature of reamers helps to reduce chattering while reaming. (Fig 3)

Types, features and functions: Hand reamers with different features are available for meeting different reaming conditions. The commonly used types are listed here under:

Fig 1

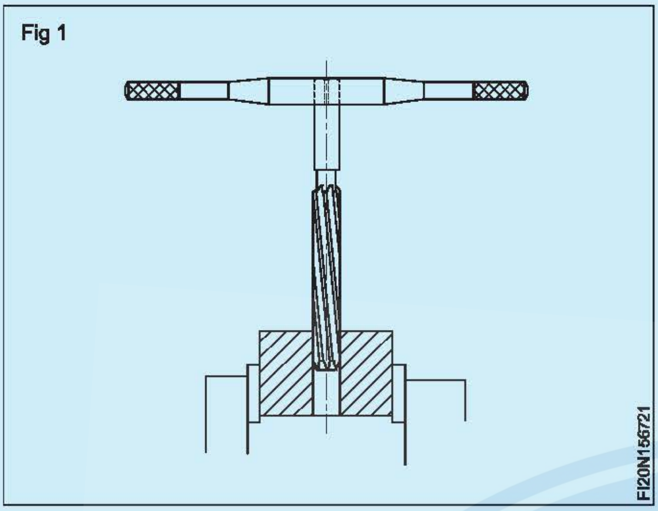


Fig 2

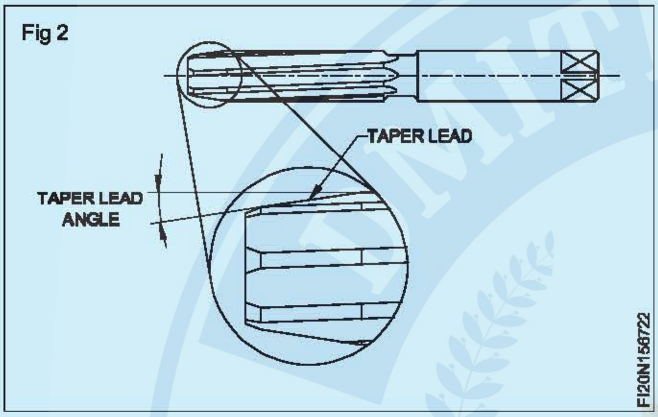
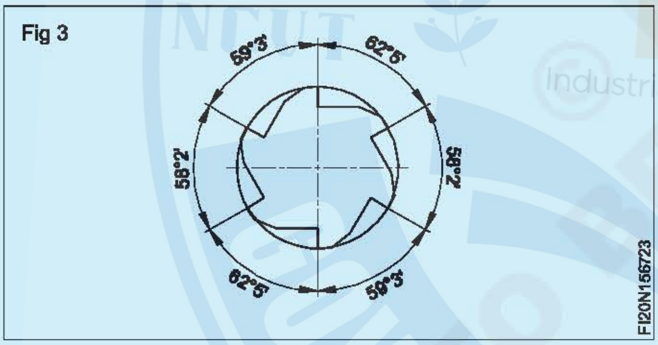


Fig 3



Parallel hand reamer with parallel shank (Fig 4a)

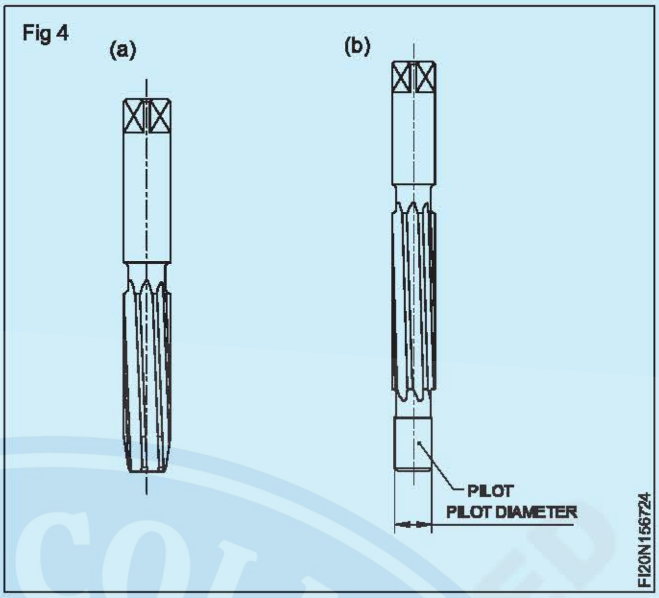
A reamer which has virtually parallel cutting edges with taper and bevel lead. The body of the reamer is integral with a shank. The shank has the nominal diameter of the cutting edges. One end of the shank is square shaped for tuning it with a tap wrench. Parallel reamers are available with straight and helical flutes. This is the commonly used hand reamer for reaming holes with parallel sides.

Reamers commonly used in workshop produce H7 holes.

Hand reamer with pilot (Fig 4b)

For this type of reamer, a portion of the body is cylindrically ground to form a pilot at the entering end. The pilot keeps the reamer concentric with the hole being reamed.

Fig 4



Socket reamer with parallel shank (Figs 5a and 5b)

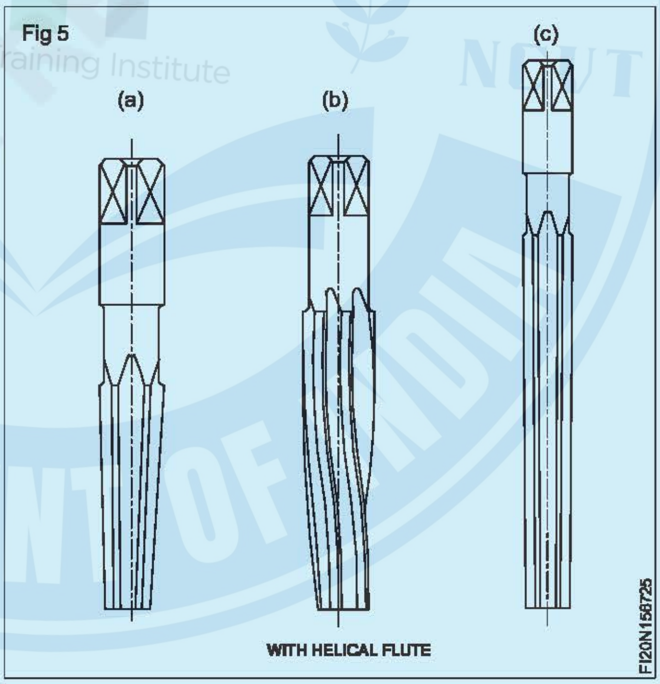
This reamer has tapered cutting edges to suit metric morse tapers. The shank is integral with the body, and is square shaped for driving. The flutes are either straight or helical.

The socket reamer is used for reaming internal morse tapered holes.

Taper pin hand reamer (Fig 5c)

This reamer has tapered cutting edges for reaming taper holes to suit taper pins. A taper pin reamer is made with a taper of 1 in 50. These reamers are available with straight or helical flutes.

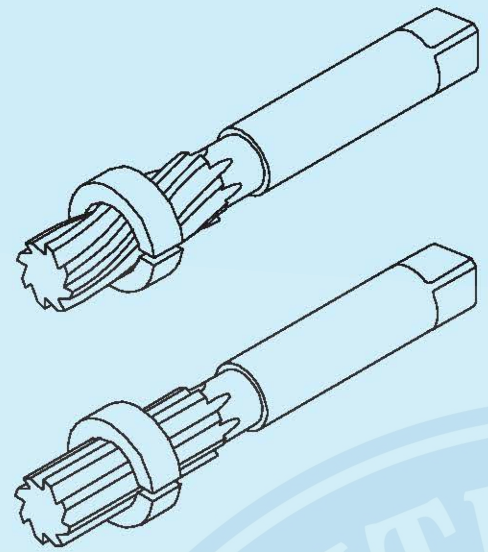
Fig 5



Use of straight and helical fluted reamers (Fig 6)

Straight fluted reamers are useful for general reaming work. Helical fluted reamers are particularly suitable for reaming holes with keyway grooves or special lines cut into them. The helical flutes will bridge the gap and reduce binding and chattering.

Fig 6



Material of hand reamers

When the reamers are made as a one-piece construction, high speed steel is used. When they are made as two-piece construction then the cutting portion is made of high speed steel while the shank portion is made of carbon steel. They are butt-welded together before manufacturing.

Specifications of a reamer: To specify a reamer the following data is to be given.

- Type
- Flute
- Shank end
- Size

Example : Hand reamer, Straight flute, Parallel shank of Ø 20 mm.

Drill size for reaming

Objective: At the end of this lesson you shall be able to
- **determine the hole size for reaming.**

For reaming with a hand or a machine reamer, the hole drilled should be smaller than the reamer size.

The drilled hole should have sufficient metal for finishing with the reamer. Excessive metal will impose a strain on the cutting edge of the reamer and damage it.

Calculating drill size for reamer: A method generally practised in workshop is by applying the following formula.

Drill size = Reamed size – (Undersize + Oversize)

Finished size: Finished size is the diameter of the reamer.

Undersize: Undersize is the recommended reduction in size for different ranges of drill diameter. (Table 1)

Table 1

Undersizes for reaming

Diameter of ready reamed hole (mm)	**Undersize of rough bored hole (mm)**
under 5	0.1.....0.2
5......20	0.2.....0.3
21....50	0.3.....0.5
over 50	0.5.....1

Oversize: It is generally considered that a twist drill will make a hole larger than its diameter. The oversize for calculation purposes is taken as 0.05 mm - for all diameters of drills.

For light metals the undersize will be chosen 50% larger.

Example: A hole is to be reamed on mild steel with a 10 mm reamer. What will be the diameter of the drill for drilling the hole before reaming?

Drill size = Reamed size – (Undersize + Oversize)

(Finished size) = 10 mm

Undersize as per table = 0.2 mm

Oversize = 0.05 mm

Drill size = 10 mm -- 0.25 mm

= 9.75 mm

Determine the drill hole sizes for the following reamers:

i 15 mm

ii 4 mm

iii 40 mm

iv 19 mm

Answer

i ______________________

ii ______________________

iii ______________________

iv ______________________

Note: If the reamed hole is undersize, the cause is that the reamer is worn out.

Always inspect the condition of the reamer before commencing reaming.

For obtaining good surface finish

Use a coolant while reaming. Remove metal chips from the reamer frequently. Advance the reamer slowly into the work.

Defects in reaming - Causes and Remedies

- **Reamed hole undersize**
  - If a worn out reamer is used, it may result in the reamed hole bearing undersize. Do not use such reamers.

- Always inspect the condition of the reamer before using.

- **Surface finish rough**
  - The causes may be any one of the following or a combinations thereof.
  - Incorrect application
  - Swarf accumulated in reamer flutes
  - Inadequate flow of coolant
  - Feed rate too fast
  - While reaming apply a steady and slow feed-rate.
  - Ensure a continuous supply of the coolant.
  - Do not turn the reamer in the reverse direction.

Determining the drill size for reaming

Use the formula,

drill diameter = reamed hole size. (undersize + oversize)

Refer to the Table 1 for the recommended undersizes in Related Theory on DRILL SIZES FOR REAMING.

Reaming

Objective : At the end of this lesson you shall be able to
- **state the procedure for hand reaming and machine reaming.**

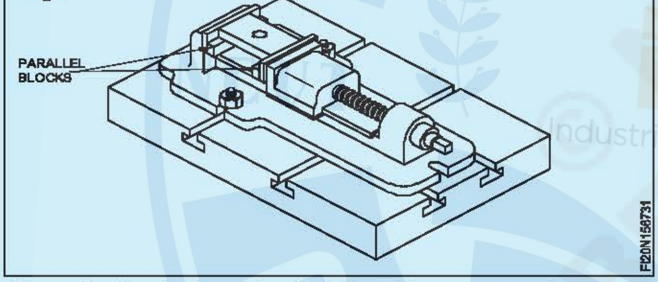
Reaming: Reaming is the operation of finishing and sizing a hole which has been previously drilled, bored, casteed holes. The tool used is called a reamer, which has multiple cutting edges. Manually it is held in a tap wrench and reamed. Machine reamer are used in drilling machine using sleeves (or) socket. Normally the speed for reaming will be 1/3rd speed of drilling.

Hand Reaming

Drill holes for reaming as per the sizes determined.

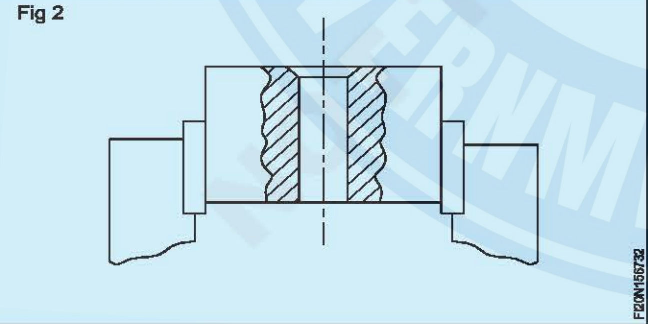
Place the work on parallels while setting on the machine vice. (Fig 1)

Fig 1



Chamfer the hole ends slightly. This removes burrs and will also help to align the reamer vertically. (Fig 2) Fix the work in the bench vice. Use vice clamps to protect the finished surfaces. Ensure that the job is horizontal. (Fig 2)

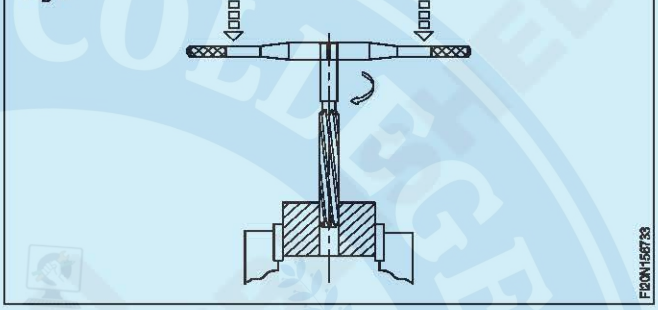
Fig 2



Fix the tap wrench on the square end and place the reamer vertically in the hole. Check the alignment with a try square. Make corrections, If necessary. Turn the tap wrench in a clockwise direction applying a slight downward pressure at the same time. (Fig 3) Apply pressure evenly at both ends of the tap wrench.

Apply cutting force: Turn the tap wrench steadily and slowly, maintaining the downward pressure.

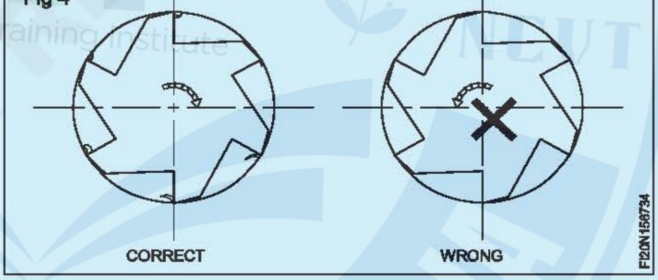
Fig 3



Do not turn in reverse direction it will scratch the reamed hole. (Fig 4)

Ream the hole through, ensure that the taper lead length of the reamer comes out well and clear from the bottom of the work. Do not allow the end of the reamer to strike on the vice.

Fig 4

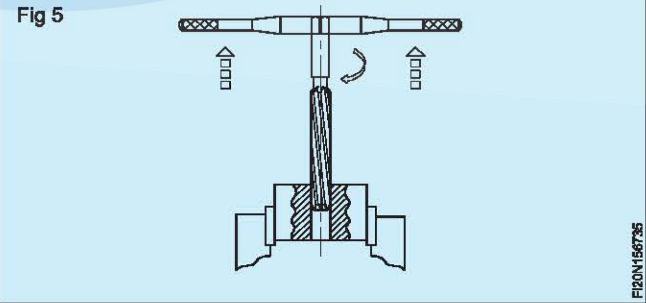


Remove the reamer with an upward pull until the reamer is clear of the hole. (Fig 5)

Remove the burrs from the bottom of the reamed hole.

Clean the hole. Check the accuracy with the cylindrical pins supplied.

Fig 5



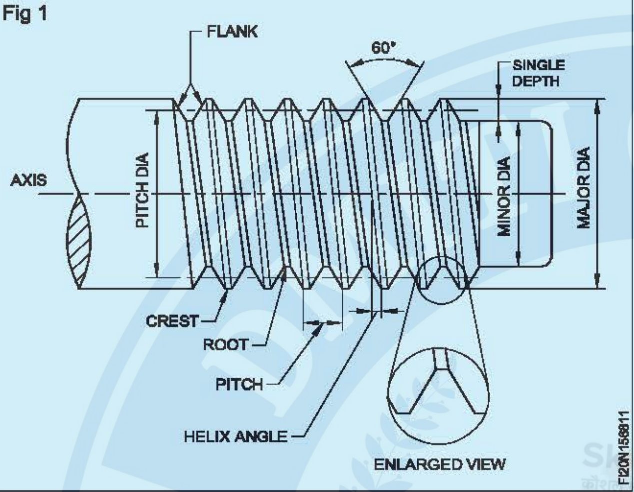
Screw thread and elements

Objectives: At the end of this lesson you shall be able to
- **state the terminology of screw threads**
- **state the types of screw threads.**

Screw thread terminology

Parts of screw thread (Fig 1)

Fig 1



Crest: The top surface joining the two sides of a thread.

Root: The bottom surface joining the two sides of adjacent threads.

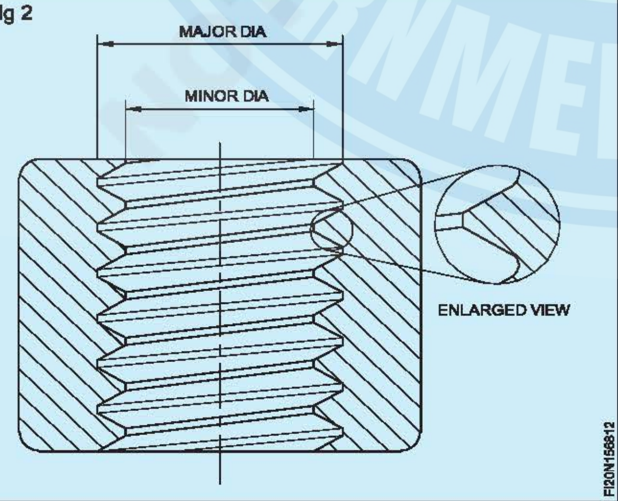
Flank: The surface joining the crest and the root.

Thread angle: The included angle between the flanks of adjacent threads.

Depth: The perpendicular distance between the roots and crest of the thread.

Major Diameter: In the case of external threads it is the diameter of the blank on which the threads are cut and in the case of internal threads it is the largest diameter after the threads are cut that are known as the major diameter. (Fig 2)

Fig 2



This is the diameter by which the sizes of screws are stated.

Minor Diameter: For external threads, the minor diameter is the smallest diameter after cutting the full thread. In the case of internal threads, it is the diameter of the hole drilled for forming the thread which is the minor diameter.

Pitch Diameter (effective diameter): The diameter of the thread at which the thread thickness is equal to one half of the pitch.

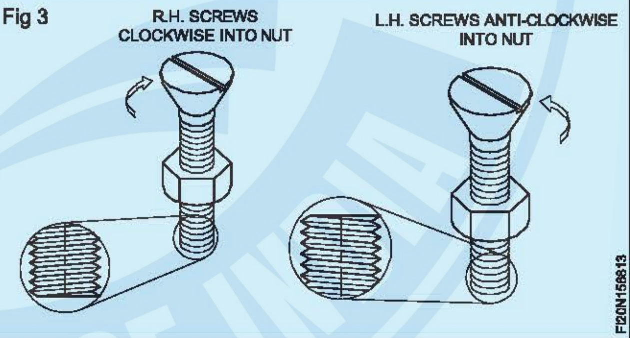
Pitch: It is the distance from a point on one thread to a correspond ing point on the adjacent thread measured parallel to the axis.

Lead: Lead is the distance of a threaded component moves along the matching component during one complete revolution. For a single start thread the lead is equal to the pitch.

Helix Angle: The angle of inclination of the thread to the imaginary perpendicular line.

Hand: The direction in which the thread is turned to advance. A right hand thread is turned clockwise to advance, while a left hand thread is turned anticlockwise.(Fig 3)

Fig 3



Screw threads - types of V threads and their uses

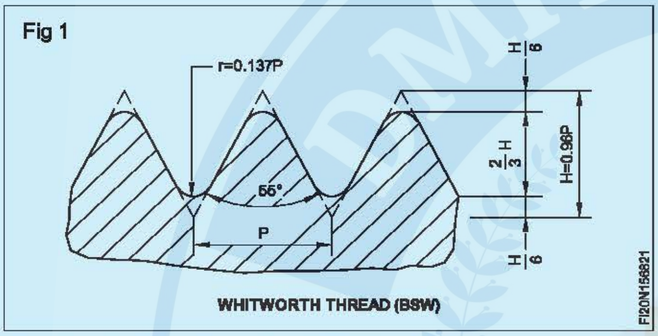
Objectives: At the end of this lesson you shall be able to
- **state the different standards of V threads**
- **indicate the angle and the relation between the pitch with the other elements of the thread**
- **state the uses of the different standards of V threads.**

The different standards of V threads are:

- BSW thread: British Standard Whitworth thread
- BSF thread: British Standard fine thread
- BSP thread: British Standard pipe thread
- B.A thread: British Association thread
- I.S.O Metric thread: International Standard Organisation metric thread
- ANS: American National or sellers' thread
- BIS Metric thread: Bureau of Indian Standard metric thread.

BSW thread (Fig 1): It has an included angle of 55° and

Fig 1



WHITWORTH THREAD (BSW)

the depth of the thread is 0.6403 x P. The crest and root are rounded off to a definite radius . The Fig 1 shows the relationship between the pitch and the other elements of the thread.

BSW thread is represented in a drawing by giving the major diameter. For example : 1/2" BSW, 1/4" BSW. The table indicates the standard number of TPI for different diameters. BSW thread is used for general purpose fastening threads.

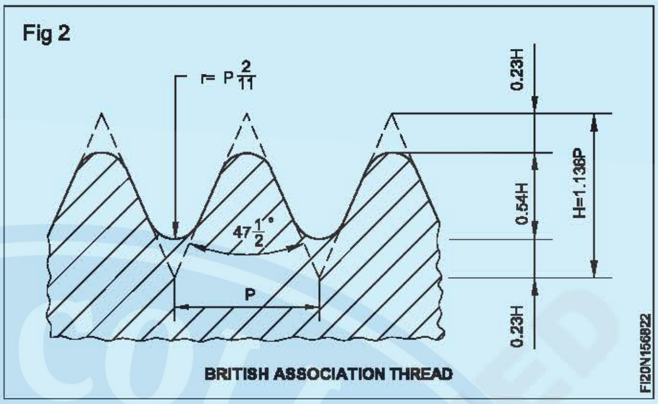
BSF thread: This thread is similar to BSW thread except the number of TPI for a particular diameter. The number of threads per inch is more than that for the BSW thread for a particular diameter. For Example, 1" BSW has 8 TPI and 1 "BSF has 10 TPI. The table indicates the standard number of TPI for different dia. of BSF threads. It is used in automobile industries.

BSP thread: This thread is recommended for pipe and pipe fittings. The table shows the pitch for different diameters. It is also similar to BSW thread. The thread is cut externally with a small taper for the threaded length. This avoids the leakage in the assembly and provides for further adjustment when slackness is felt.

BA thread (Fig 2): This thread has an included angle of 47 1/2°. Depth and other elements are as shown in the figure. It is used in small screws of electrical appliances, watch screws, screws of scientific apparatus.

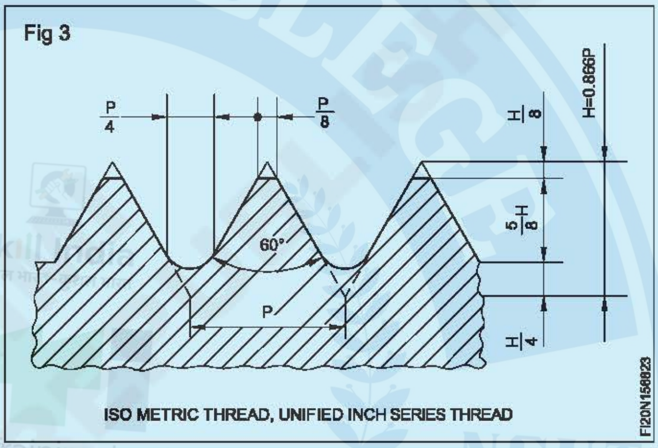
Unified thread (Fig 3): For both the metric and inch series, ISO has developed this thread. Its angle is 60°. The crest and root are flat and the other dimensions are as shown in the Fig 3. This thread is used for general fastening purposes.

Fig 2



BRITISH ASSOCIATION THREAD

Fig 3



ISO METRIC THREAD, UNIFIED INCH SERIES THREAD

This thread of metric standard is represented in a drawing by the letter 'M' followed by the major diameter for the coarse series.

Ex : M14, M12 etc.

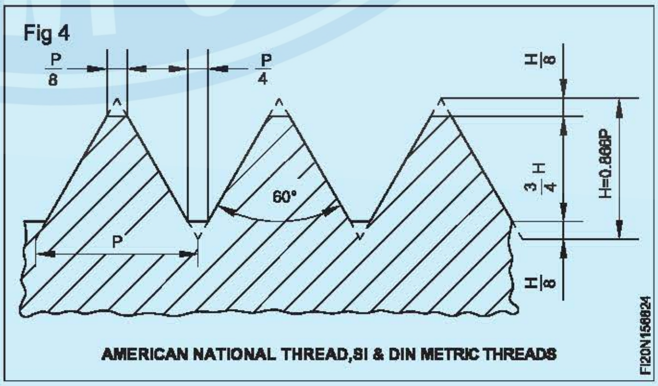
For the fine series, the letter 'M' is followed by the major diameter and pitch.

Ex : M14 x 1.5

M24 x 2

American National Thread (Fig 4)**:** These threads are also called as seller's threads. It was more commonly used prior to the introduction of the ISO unified thread.

Fig 4



AMERICAN NATIONAL THREAD,SI & DIN METRIC THREADS

Screw pitch gauge

Objectives: At the end of this lesson you shall be able to
- **state the purpose of a screw pitch gauge**
- **state the features of a screw pitch gauge.**

Purpose

A screw pitch gauge is used to determine the pitch of a thread.

It is also used to compare the profile of threads.

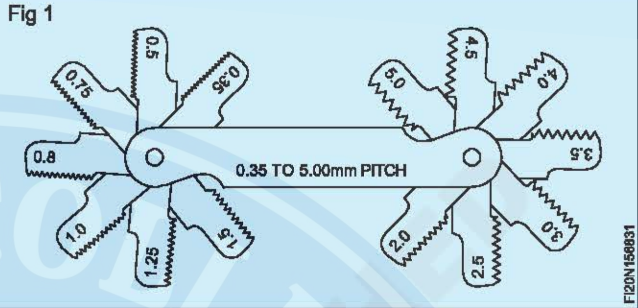
Constructional features

Pitch gauges are available with a number of blades assembled as a set. Each blade is meant for checking a particular standard thread pitch. The blades are made of thin spring steel sheets, and are hardened.

Some screw pitch gauge sets will have blades provided for checking British Standard threads (BSW, BSF etc.) at one end and the metric standard at the other end.

The thread profile on each blade is cut for about 25 mm to 30 mm. The pitch of the blade is stamped on each blade. The standard and range of the pitches are marked on the case. (Fig 1)

Fig 1



Taps

Objectives: At the end of this lesson you shall be able to
- **state the uses of hand taps**
- **state the features of hand taps**
- **distinguish between the different taps in a set.**

Use of hand taps: Hand taps are used for internal threading of components.

Features (Fig 1): They are made from high speed steel.

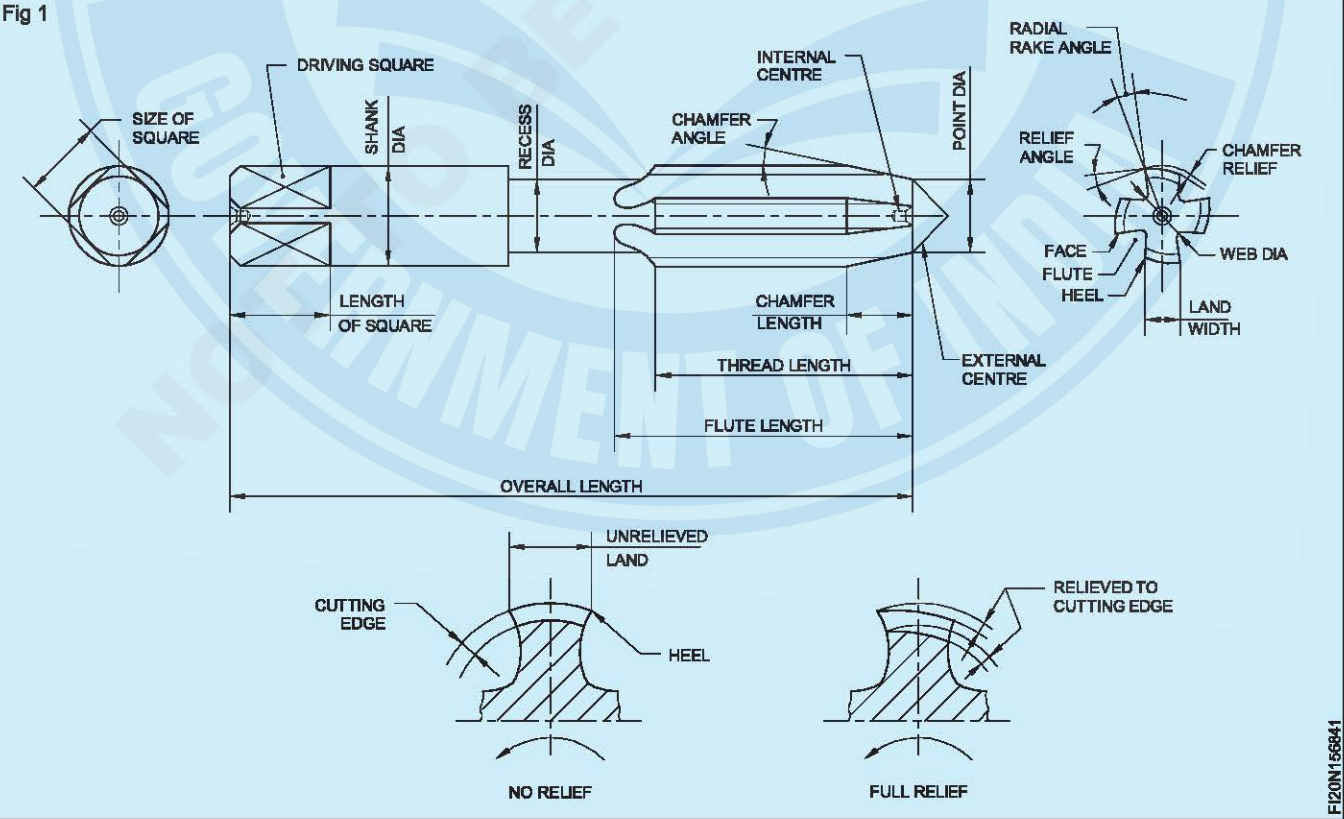
The threads are cut on the periphery and are accurately finished.

To form the cutting edges, flutes are cut across the thread.

The end of the shank of the tap is made of square shape for the purpose of holding and turning the taps.

The end of the taps are chamfered (taper lead) for assisting, aligning and starting of the thread.

Fig 1



The size of the taps, the thread standard, the pitch of the thread, the dia. of the tapping hole are usually marked on the shank.

Marking on the shank are also made to indicate the type of tap i.e. first, second and plug.

Types of taps in a set : Hand taps for a particular thread are available as a set consisting of three pieces. (Fig 2)

These are:

– First tap or taper tap

– Second tap or intermediate tap

– Plug or bottoming tap.

These taps are identical in all features except in the tap lead.

The taper tap is to start the thread. It is possible to form full threads by the taper tap in through holes which are not deep.

The bottoming tap (plug) is used to finish the threads of a blind hole to the correct depth.

For identifying the type of taps quickly - the taps are either numbered 1,2 and 3 or rings are marked on the shank.

The taper tap has one ring, the intermediate tap has two and the bottoming tap has three rings. (Fig 2)

Fig 2

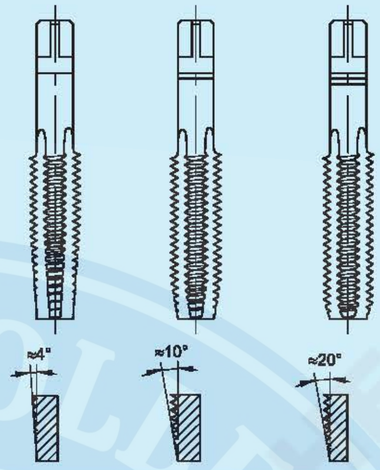


Table for tap drill size

B.S.W. (55°)		
Tap size (inch)	**Threads per inch**	**Tap drill size (mm)**
3/16	24	3.7mm
7/32	24	4.5mm
1/4	20	5.1mm
5/16	18	6.5mm
3/8	16	7.94mm
7/16	14	9.3mm
1/2	12	10.5mm
9/16	12	12.1mm
5/8	11	13.5mm
11/16	11	15mm
3/4	10	16.257mm
7/8	9	19.25mm
1"	8	22mm

B.S.F. (55°)		
Tap size (inch)	**Threads per inch**	**Tap drill size (mm)**
3/16	32	3.97mm
7/32	28	4.6mm
1/4	26	5.3mm
5/16	22	6.75mm
3/8	20	8.2mm
7/16	18	9.7mm
1/2	16	11.11mm
9/16	16	12.7mm
5/8	14	14mm
11/16	14	15.5mm
3/4	12	16.75mm
7/8	11	19.84mm
1"	10	22.75mm

NPT National pipe thread

Tap size (inch)	Threads per inch	Tap drill size inch	Tap size (inch)	Threads per inch	Tap drill size inch
1/8	27	11/32	1	11 1/2	1 5/32
1/4	18	7/16	1 1/4	11 1/4	1 1/2
3/8	18	19/32	1 1/2	11 1/2	1 23/32
1/2	14	23/32	2	11 1/2	2 23/16
3/4	14	15/16	2 1/2	8	2 5/8

Tap drill sizes ISO Inch (Unified) thread

NC National coarse			NF National Fine		
Tap size (inch)	Threads per inch	Tap drill size inch	Tap size (inch)	Threads per inch	Tap drill size inch
1/4	20	13/64	1/4	28	7/32
5/16	18	17/64	5/16	24	17/64
3/8	16	5/16	3/8	24	21/64
7/16	14	3/8	7/16	20	25/64
1/2	13	27/64	1/2	20	29/64
9/16	12	31/64	9/16	18	33/64
5/8	11	17/32	5/8	18	37/64
3/4	10	21/32	3/4	16	11/16
7/8	9	49/64	7/8	14	13/16
1"	8	7/8	1"	14	15/16
1 1/8	7	63/64	1 1/8	12	1 3/6
1 1/4	7	17/64	1 1/4	12	1 11/6
1 3/8	6	17/32	1 3/8	12	1 19/64
1 1/2	6	1 11/32	1 1/2	12	1 27/64
1 3/4	5	1 9/16			
2"	4 1/2	1 25/32			

Machine taps

Objectives: At the end of this lesson you shall be able to
- **state the characteristics of machine taps**
- **name the different types of machine taps**
- **state the features and uses of different types of machine taps.**

Machine taps: Machine taps of different types are available. The two important features of machine taps are

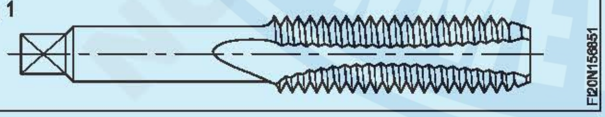
– Ability to withstand the torque needed for threading holes

– Provision for eliminating chip jamming.

Types of machine taps

Gun tap (Spiral pointed tap) (Fig 1)

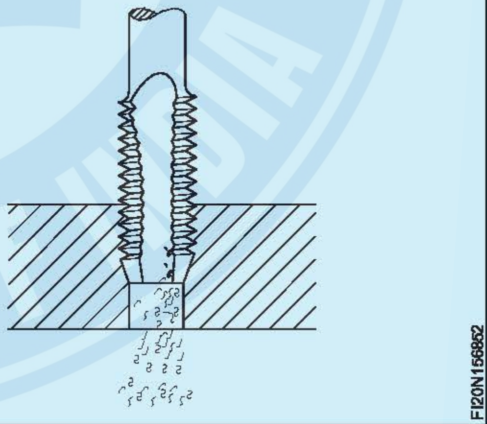
Fig 1



These taps are especially useful for machine tapping of through holes. In the case of blind hole tapping, there should be sufficient space below to accommodate the chips. While tapping, the chips are forced out ahead of the tap. (Fig 2)

This prevents the clogging of the chips and thus reduces the chances of tap breakage. These taps are stronger since the flutes are shallow. The flutes of these taps do not convey chips.

Fig 2



Flute-less spiral pointed tap (Stub flute taps) (Fig 3)

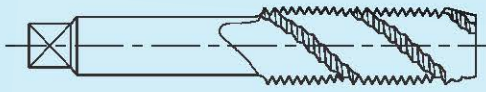
These taps have short angular flutes ground on the chamfered end, and the rest of the body is left solid. These taps are stronger than gun taps.

Flute-less taps are used for tapping through holes on materials which are not thicker than the diameter of the holes. Flutes spiral point taps are best suited for tapping soft materials or thin metal sections.

Helical fluted taps/spiral fluted taps: These taps have spiral flutes which bring out the chips from the hole being tapped. (Fig 4)

Fig 3

Fig 4



These are useful for tapping holes with slots. The helical land of the tap will bridge the interruption of the surface being threaded. The helical flutes of the tap provide a shear cutting action, and are mostly used to tap holes in ductile materials like aluminium, brass, copper etc.

Spiral fluted taps are also available with fast spiral. (Fig 5) These taps are best suited for tapping deep holes as these can clear the chips faster from the hole. (Fig 6)

Thread forming taps (Fluteless taps)

These taps form threads in the hole by displacing the material and not by cutting action. (Fig 7)

Fig 5

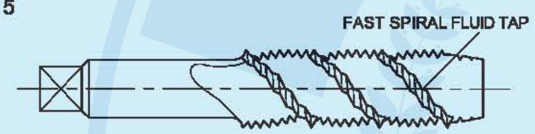


Fig 6

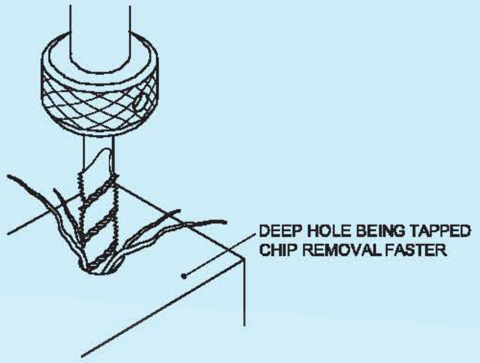
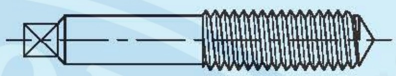
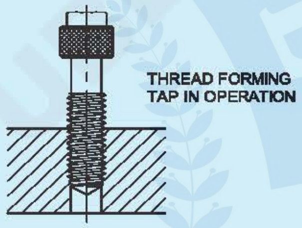


Fig 7



These taps have projecting lobes which actually help in forming the thread. (Fig 8) Since there are no chips in the process, it is very valuable in places where chip removal poses problems. These taps are excellent for tapping copper, brass, aluminium, lead etc. The thread finish is also comparatively better than in the fluted taps.

Fig 8

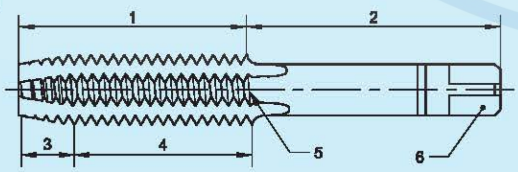


General informative points on taps

Objectives: At the end of this lesson you shall be able to
- **differentiate between hand tap and machine tap**
- **identify the parts of a machine tap**
- **state the constructional features of a machine tap.**

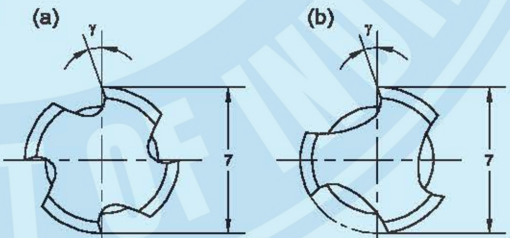
Unlike tapping with the three piece set of hand taps, the machine tap cuts the entire threaded profile in one operation. The machine tap is normally made of tool steel and consists of the shank (2) and the cutting section (1) as shown in (Fig 1). The cutting section itself is subdivided into two areas. The start (3), which serves for cutting, and the guiding section (4) for the feeding motion and smoothing of the newly cut thread. (Fig 1)

Fig 1



The number of flutes (5), may be even or odd. With an even number of flutes, measuring of the diameter (7) is easier. (Figs 2a and 2b)

Fig 2



Straight and spiral groove machine taps are available. The diameter of the shank and the shape of its end vary between the various standards. The shank diameter may be smaller, equal to or larger than the thread diameter. The shank ends are available in straight design, with square ends as shown in (6) or with driving shoulders.

Chip removal (flow) takes place at the start of the tap. The rake angle must be adapted to the material to be machined. Hard and brittle materials require a small rake angle and soft materials need a larger rake angle.

Accordingly three types of taps are available.

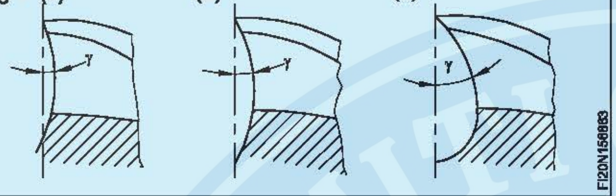
Type normal (Fig 3b) with a rake angle of approximately 12°.

Type soft (Fig 3c) with a rake angle of approximately 20°.

Type hard (Fig 3a) with a rake angle of approximately 3°.

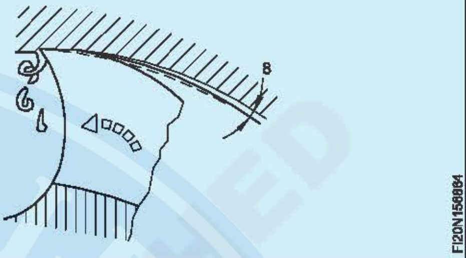
The normal type of rake angle taps can be used in most cases. The start must be ground symmetrical. Before using the tap, it is necessary to check that the cutting edges are not chipped, and all the edges are sharp.

Fig 3



The 'hard' type tap is used for tapping brittle materials like cast iron. In case a 'normal' type tap is used on cast iron, the tap cutting edges get blunt soon and the tap cannot be used again on ductile materials like mild steel. The fine cast iron splinters wear the external diameter of the cutting edges of the tap causing them to tend to become blunt, and when the same tap is used on steel which is more flexible it is elastically pressed away (8) at the cutting point. Behind the cutting edge the material returns to the machined diameter. The depth of the groove also causes jamming of the guiding section of the tap. (Fig 4)

Fig 4



Pipe Threads and Pipe Taps

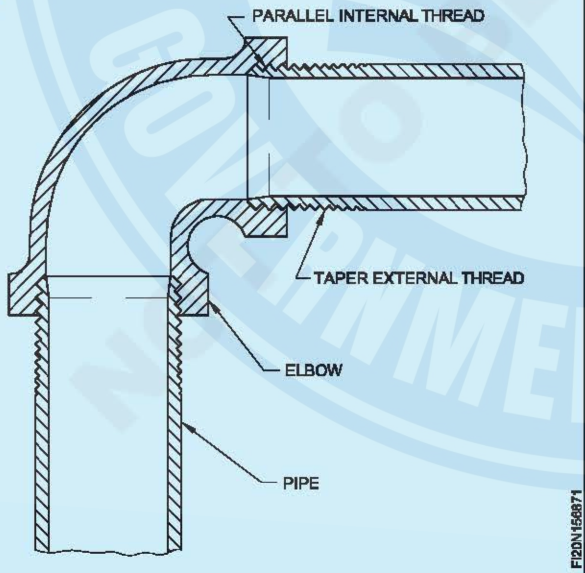
Objectives: At the end of this lesson you shall be able to

- **state parallel and taper pipes threads**
- **determine the wall thickness and threads per inch (TPI) of BSP threads**
- **state the method of sealing pipe joints**
- **determine blank sizes for threading as per B.S 21 - 1973 and I.S. 2643 - 1964.**

Pipe threads

The standard pipe fittings are threaded to British Standard pipe (BSP). The internal pipe threads have parallel threads whereas the external pipes have tapered threads as shown in Fig 1.

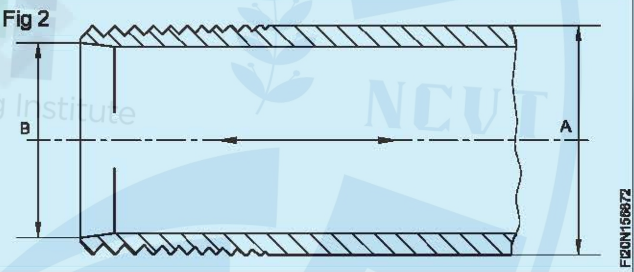
Fig 1



B.S.P. threads

Galvinized iron pipes are available in sizes ranging from 1/2" to 6" in several different wall thickness. The table 1 shows outside diameters and threads per inch from 1/2" to 4". (Fig 2)

Fig 2



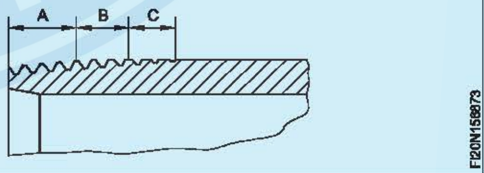
The next two threads have fully formed bottoms but that tops. (B)

The last four threads have flat tops and bottoms. (C)

Sealing pipe joint

Fig 3 shows that the pipe has several fully formed threads at the end. (A)

Fig 3



The pipe joint shown in Fig 4 consists of the following:

1 Parallel female thread

2 Tapered male thread

3 Hemp packing

Fig 4

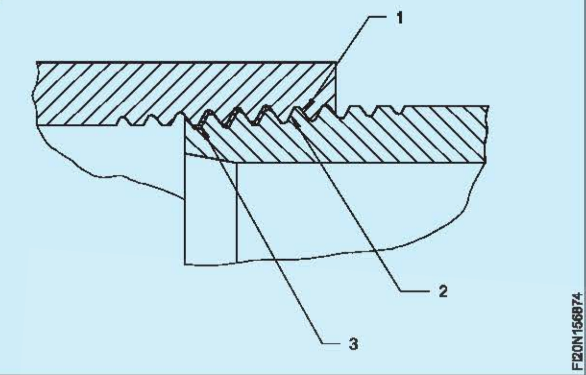


Table 1

BSP - Pipe sizes or DIN 2999 (inside) (B)	Threads inch	Outside diameter/ mm of the pipe (A)
1/2"	14	20.955 mm
3/4"	14	26.441
1"	11	33.249
11/4"	11	41.910
11/2"	11	47.803
2"	11	59.614
2 1/2"	8	75.184
3"	8	87.884
4"	8	113.030

The hemp packing issued to ensure that any small space between two metal threads (male and female threads) is sealed to prevent any leakage.

Pipe taps

Internal pipe threads are usually cut with standard taper pipe taps. (Fig 5)

Fig 5

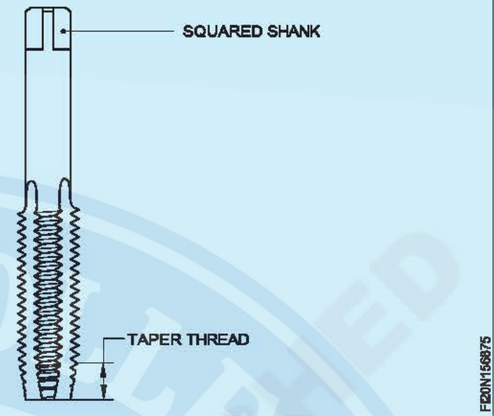
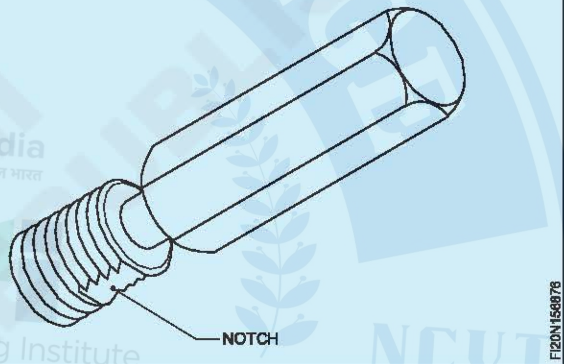


Fig 6



Tap wrenches, removal of broken tap, studs

Objectives : At the end of this lesson you shall be able to
- **name the different types of tap wrenches**
- **state the uses of the different types of wrenches.**

Tap wrenches: Tap wrenches are used to align and drive the hand taps correctly into the hole to be threaded.

Tap wrenches are of different types, such as double-ended adjustable wrench, T- handle tap wrench, solid type tap wrench etc.

Double - ended adjustable tap wrench or bar type tap wrench (Fig 1)

Fig 1

This is the most commonly used type of tap wrench. It is available in various sizes - 175, 250, 350 mm long. These tap wrenches are more suitable for large diameter taps, and can be used in open places where there is no obstruction to turn the tap.

It is important to select the correct size of wrench.

T- handle tap wrench (Fig 2)

These are small, adjustable chucks with two jaws and a handle to turn the wrench.

This tap wrench is useful to work in restricted places, and is turned with one hand only. Most suitable for smaller sizes of taps.

Solid type tap wrench (Fig 3)

These wrenches are not adjustable.

They can take only certain sizes of taps. This eliminates the use of wrong length of the tap wrenches, and thus prevents damage to the taps.

Tap Material: Made from a single piece of solid Cast iron (or) steel. Cast iron and steel are used because of strong, durable and unlikely to deform under pressure.

Fig 2

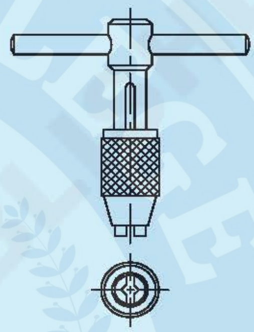
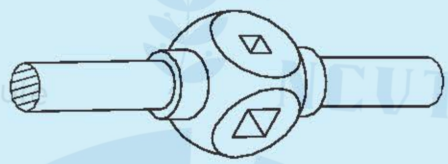


Fig 3



Removing broken taps

Objectives: At the end of this lesson you shall be able to
- **name the different methods of removing broken taps**
- **state the methods of removing broken taps.**

A tap broken above the surface of the workpiece can be removed using gripping tools like pliers.

Taps broken below the surface pose a problem for removing. Any one of the several methods given below can be used.

Use of tap extractor (Fig 1)

This is a very delicate tool and need very careful handling.

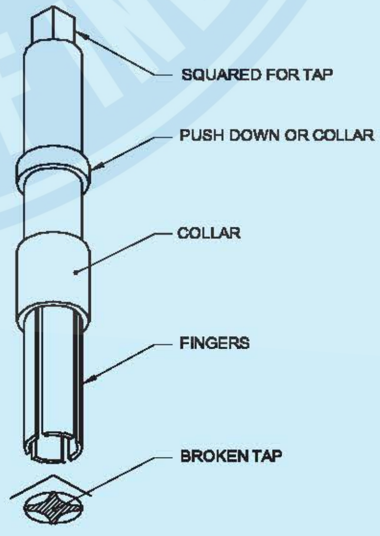
This extractor has fingers which can be inserted on the flutes of the broken tap. The sliding collar is then brought to the surface of the work and the extractor turned anticlockwise to take out the broken tap.

A light blow on the broken tap with a punch will help to relieve the tap if it is jammed inside the hole.

Use of punch (Fig 2)

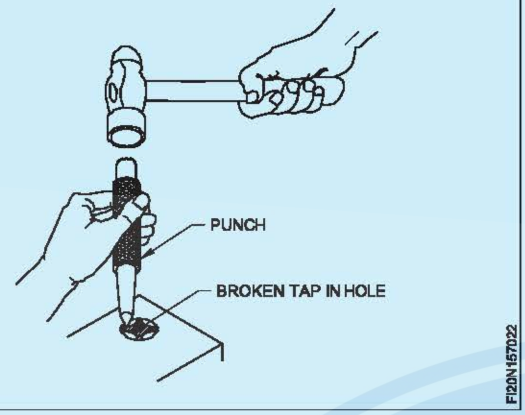
In this method the point of the punch is placed in the flute of the broken tap in an inclination and struck with a hammer

Fig 1



the positioning of the punch should be such that the broken tap is rotated anticlockwise when struck.

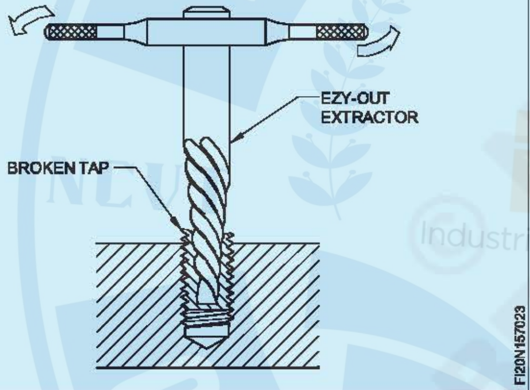
Fig 2



Annealing and drilling the tap

This is a method adopted when other method fail. In the process the broken tap is heated by flame or by other methods for annealing. A hole is then drilled on the annealed tap. The remaining piece can be removed either by using a drift or using an EZY - OUT (extractor). This method is not suitable for workpieces with low melting temperatures such as aluminium, copper etc. (Fig 3)

Fig 3



Use of arc welding

This is a suitable method when a small tap is broken at the bottom of materials like copper, aluminium etc. In this method the electrode is brought in contact with the broken tap and stuck so that it is attached with the broken tap. The tap may be removed by rotating the electrode.

Use of nitric acid

In this method nitric acid is diluted in a proportion of about one part acid to five parts of water is injected inside. The action of the acid loosens the tap and then it is removed with an extractor or with a nose plier. The workpiece should be thoroughly cleaned for preventing further action of the acid.

While diluting acid mix acid to water.

Use of spark erosion

For salvaging certain precision components damaged due to breakage of taps, spark erosion can be used. In this process, the metal (broken tap) is removed by means of repetitive spark discharges. The electrical discharge occurs between an electrode and the electro - conductive workpiece (tap) and the minute particles are eroded both from the electrode and the workpiece. In many cases it may not be necessary to remove the broken tap completely. (After a small portion has been eroded, a screw- driver or punch can be used to remove the remaining portion of the tap.) The shape of the electrode also need not be round. It can be for assisting the tools for routing the broken tap.

Removing broken stud

Objectives: At the end of this lesson you shall be able to
- **state the reasons for breakage of stud**
- **state different methods for removing broken stud.**

The stud is used in place of a bolt, when there is insufficient space to accommodate the bolt head or to avoid use of an unnecessarily long bolt. Studs are generally used to fix up cover plates or to connect cylinder covers to engine cylinders.

Reasons for breakage of stud/bolt.

Excessive torque is applied while screwing the stud into the hole.

Corrosive attack on the thread.

Matching threads are not of proper formation.

Threads are seized.

Methods of removing broken studs

Prick punch method

If the stud is broken very near to the surface, drive it in an anticlockwise direction, using a prick punch and hammer to remove it. (Fig 1)

Filing square form

When the stud is broken a little above the surface form a square on the projecting portion to suit a standard spanner. Then turn it anticlockwise using a spanner to remove it. (Fig 2)

Fig 1

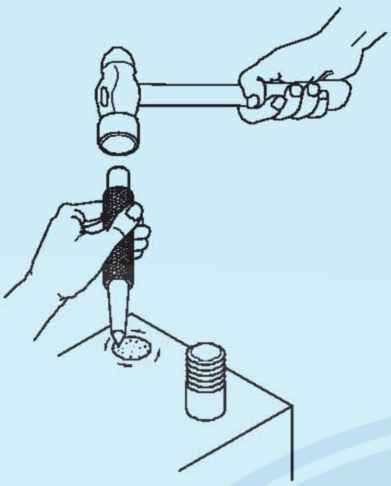
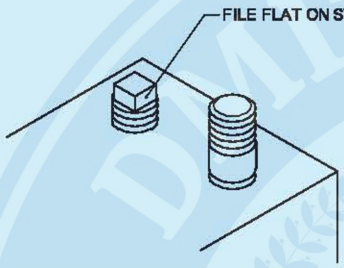


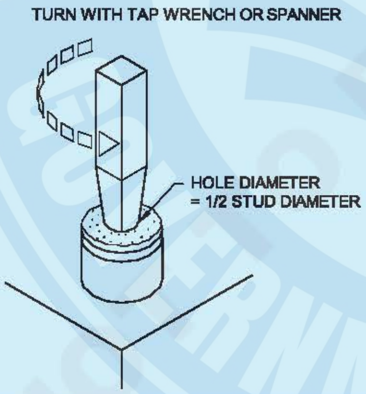
Fig 2



Using square taper punch

Broken stud can also be removed by drilling a blind hole (hole diameter equals to half of stud diameter) and driving a square taper punch into the hole as shown in Fig 3. Turn the punch using a suitable spanner in an anti - clock-wise direction to unscrew the stud.

Fig 3

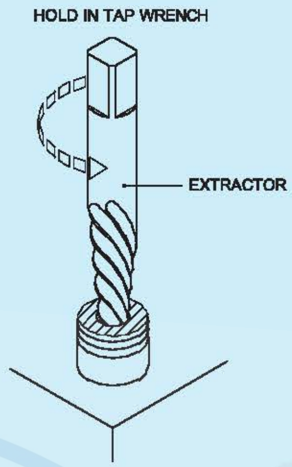


EZY - out method (Fig 4)

Ezy - out or a stud extractor is a hand tool, somewhat similar to the form of a taper reamer but has left hand spiral. It is available in a set of 5 pieces. The recommended drill size is punched on each ezy - out .

After drilling the hole recommended ezy - out is set on it and turned in an anti - clockwise direction by a tap wrench. As it is rotated it penetrates into the hole increasing its grip and in the process the broken stud gets unscrewed. (Fig 4)

Fig 4



Making drill hole: Correctly find out the centre of the broken stud and drill hole nearly equal to the core diameter of the stud down the centre so that the threads only remain. Remove the thread portion by the point of a scriber in the form of broken chips. Re - tap the drill the hole to clear the threads. (Fig 5)

If all other method fail, drill a hole equal to the size of the stud size or a little over and tap the hole with an oversize tap. Now a special over size stud as shown in Fig 6 is to be made and fitted in position.

Fig 5

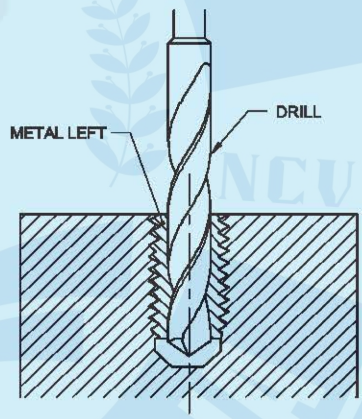
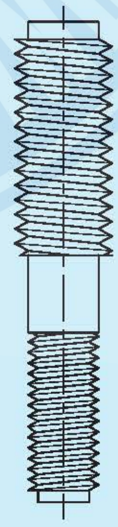


Fig 6



OVERSIZE SPECIAL STUD TO SUIT THE TAPPED HOLE

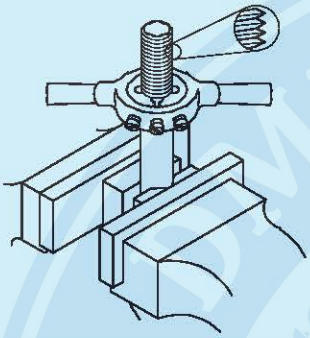
Dies and die stock

Objectives: At the end of this lesson you shall be able to
- **list the different types of dies**
- **state the features of each type of die**
- **state the use of each type of die**
- **name the type of diestock for each type of die.**

Uses of dies

Threading dies are used to cut external threads on cylindrical workpieces. (Fig 1)

Fig 1



FI20N157111

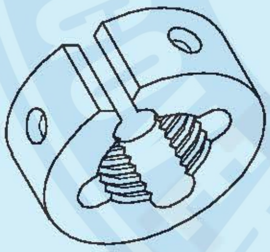
Types of dies

The following are the different types of dies.

- Circular split die (Button die)
- Half die
- Adjustable screw plate die

Circular split die/button die (Fig 2)

Fig 2



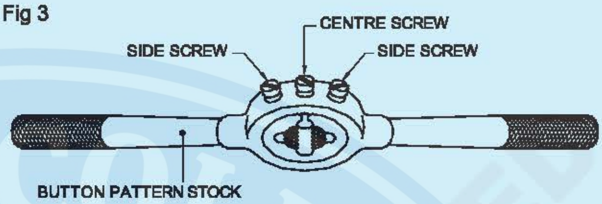
FI20N157112

This has a slot cut to permit slight variation in size.

Dies are made of high speed steel

When held in the diestock, variation in the size can be made by using the adjusting screws. This permits increasing or decreasing of the depth of cut. When the side screws are tightened the die will close slightly. (Fig 3) For adjusting the depth of the cut, the centre screw is advanced and locked in the groove. This type or die stock is called button pattern stock

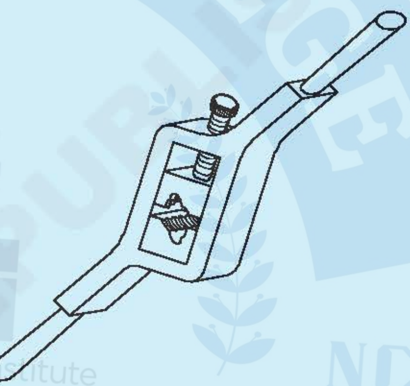
Fig 3



FI20N157113

Half die (Fig 4)

Fig 4



FI20N157114

Half dies are stronger in construction.

Adjustments can be made easily to increase or decrease the depth of cut.

These dies are available in matching pairs and should be used together.

By adjusting the screw of the diestock, the die pieces can be brought closer together or can be moved apart.

They need a special die holder.

Adjustable screw plate die (Fig 5)

This is another type of a two piece die similar to the half die.

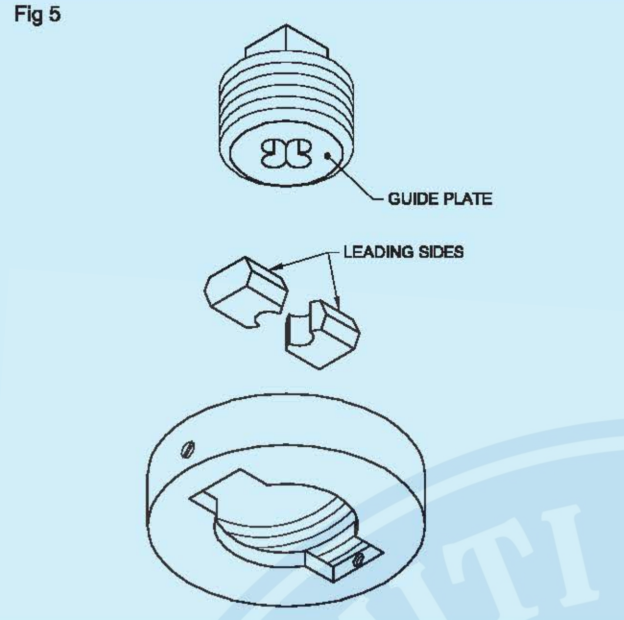
This provides greater adjustment than the split die.

The two die halves are held securely in a collar by means of a threaded plate (guide plate) which also acts as a guide while threading.

When the guide plate is tightened after placing the die pieces in the collar, the die pieces are correctly located and rigidly held.

The die pieces can be adjusted, using the adjusting screws on the collar. This type of die stock used is called quick cut diestock. (Fig 6)

Fig 5



The bottom of the die halves is tapered to provide the lead for starting the thread. On one side of each die head, the serial number is stamped.

Both pieces should have the same serial numbers.

Die Nut (Solid Die) (Fig 7)

Fig 6

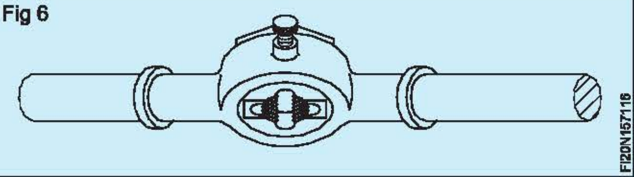
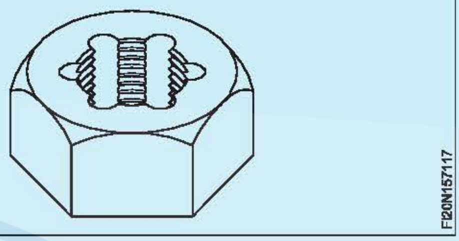


Fig 7



The die nut is used for chasing or reconditioning the damaged threads.

Die nuts are not to be used for cutting new threads.

The die nuts are available for different standards and sizes of threads.

The die nut is turned with a spanner.

Blank size for external threading

Objective: At the end of this lesson you shall be able to
- **determine the diameter of blank size for external thread cutting.**

Why should the blank size be less?

It has been observed from practice that the threaded diameters of steel blank**s** show a slight increase in diameter. such increase in the diameter will make assembly of external and internal threaded components very difficult. To overcome this, the diameter of the blank is slightly reduced before commencing the threading.

What should be the blank size?

The diameter of the blank should be less by 1/10th of the pitch of the thread.

Example

For cutting the thread of M12 with 1.75mm pitch the diameter of the blank is 11.80.

Formula, D = d - p/10

= 12mm - 0.175mm

= 11.825 or 11.8 mm.

d = diameter of bolt

D = the blank diameter

p = pitch of thread

Calculate the blank size for preparing a bolt of M16 x 1.5?

Answer

....

....

....

External threading using dies

Objective: At the end of this lesson you shall be able to
- **cut external threads using dies.**

Check blank size.

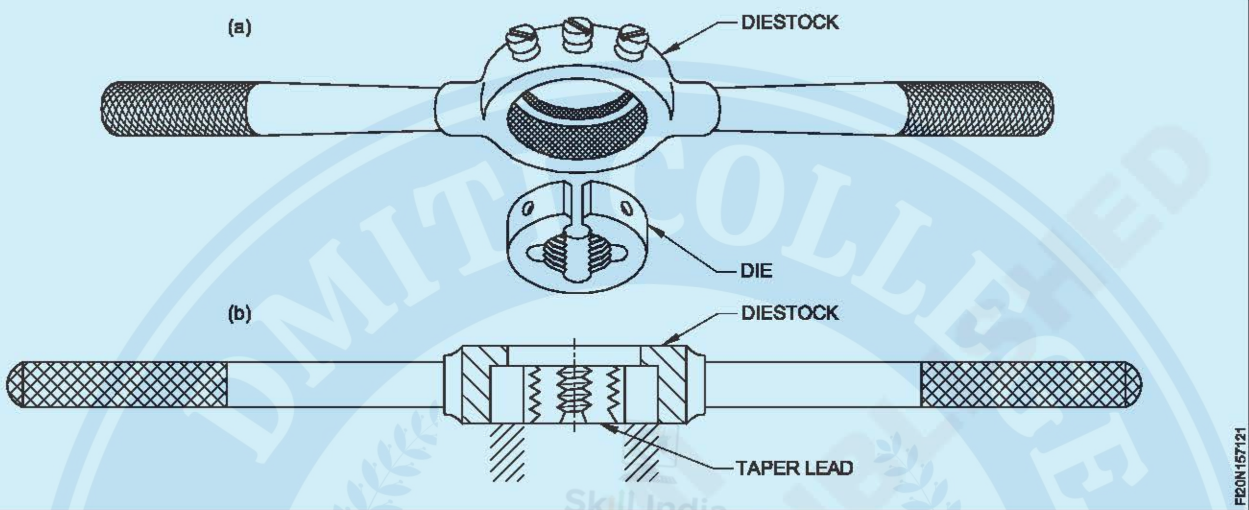
Blank size = Threads size -0.1 $\times$ pitch of thread

Procedure: Fix the die in the diestock and place the leading side of the die opposite to the step of the diestock. (Figs 1a & 1b)

Use false jaws for ensuring a good grip in the vice.

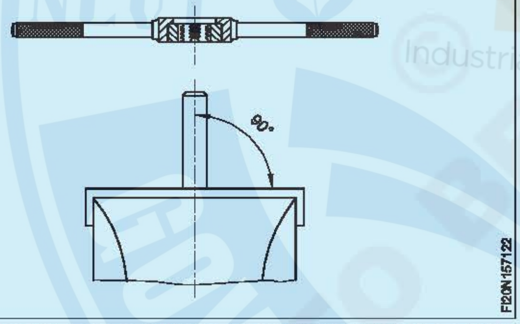
Project the blank above the vice - just the required thread length only.

Fig 1



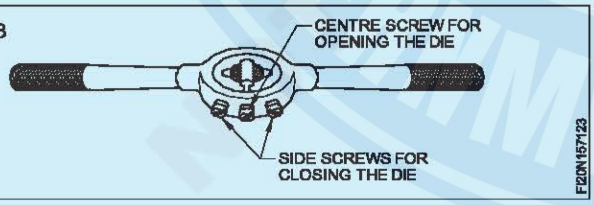
Place the leading side of the die on the chamfer of the work (Fig 2)

Fig 2



Make sure that the die is fully open by tightening the centre screw of the diestock. (Fig 3)

Fig 3



Start the die, square to the bolt centre line. (Fig 4)

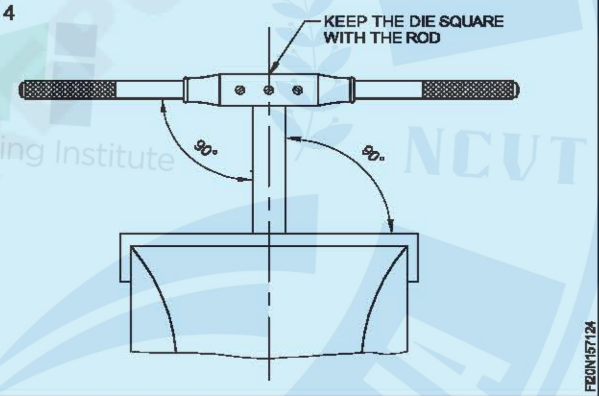
Apply pressure on the diestock evenly and turn clockwise direction to advance the die on the bolt blank. (Fig 5)

Cut slowly and reverse the die for a short distance in order to break the chips

Use a cutting lubricant.

Increase the depth of the cut gradually by adjusting the outer screws.

Fig 4



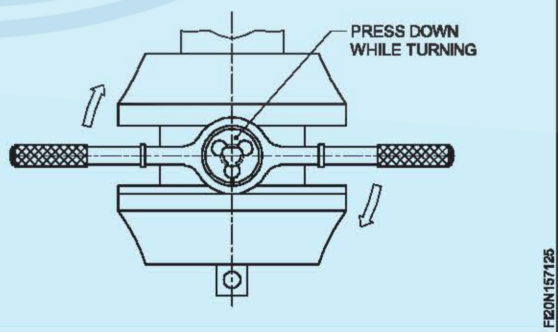
Check the thread with a matching nut.

Repeat the cutting until the nut matches.

Too much depth of cut at one time will spoil the threads. It can also spoil the die.

Clean the die frequently to prevent the chips from clogging and spoiling the thread.

Fig 5



Drill troubles - Causes and remedy, drill kinds

Objectives: At the end of this lesson you shall be able to
- **list the common drilling defects**
- **explain the causes of drilling defects**

The common defects in drilling are listed below.

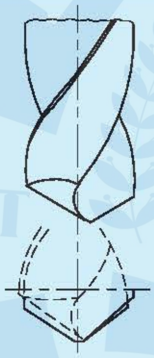
- Oversized holes
- Overheated drills
- Rough holes
- Unequal and interrupted flow of chips
- Split webs or broken drills

Oversized holes

Oversized holes can be due to:

- The unequal length of the cutting edges (Fig 1)

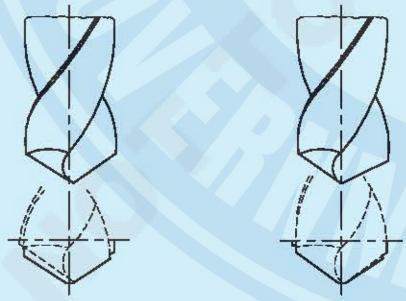
Fig 1



UNEQUAL LIP LENGTH

- The unequal angle of the cutting edges (Fig 2)

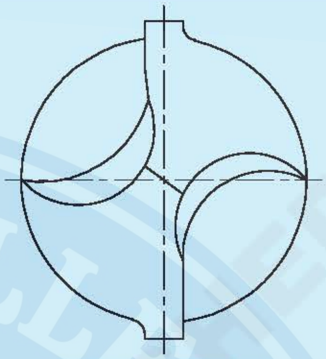
Fig 2



UNEQUAL ANGLE   UNEQUAL ANGLE AND LENGTH

- The unequal thinning of the point (Fig 3)
- The spindle running out of centre
- The drill point not being in centre. (Fig 4)

Fig 3



UNEQUAL POINT THINNING

Fig 4

DRILL POINT NOT IN THE CENTRE OF THE JOB

Overheated drills

The drills may get overheated if the:

- Cutting speed is too high
- Feed rate is too high
- Clearance angle is incorrect
- Cooling is ineffective
- Point angle is incorrect
- Drill is not sharp.

Rough holes

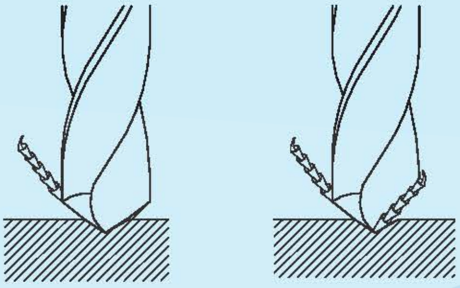
Rough holes are caused if the:

- Feed rate is too much
- Drill cutting edges are not sharp
- Cooling is ineffective.

Unequal flow of chips (Fig 5)

Unequal flow of chips is caused if the cutting edges are not equal and the point angle is not in the centre of the drill.

Fig 5



FI20N157215

Broken drill or split web

Broken drill or split web occurs when the:

- Cutting speed is too high
- Feed rate is too high
- Work is not held rigidly
- Drill is not held correctly
- Drill is not sharp
- Point angle is incorrect
- Cooling is insufficient
- Flutes are clogged with chips.

Letter and number drills

Objectives: At the end of this lesson you shall be able to
- **state the range of drill sizes in number and letter drill series**
- **determine the number and letter drills for given diameters referring to the chart.**

Generally drills are manufactured to standard sizes in the metric system. These drills, are available in specified steps. The drills, which are not covered under the above category, are manufactured in number and letter drills.

These drills are used where odd sizes of holes are to be drilled.

Letter drills: The letter drill series consists of drill sizes from 'A' to 'Z'. The letter 'A' drill is the smallest with 5.944 mm diameter, and the letter 'Z' is the largest, with a 10.490 mm diameter. (Table 1)

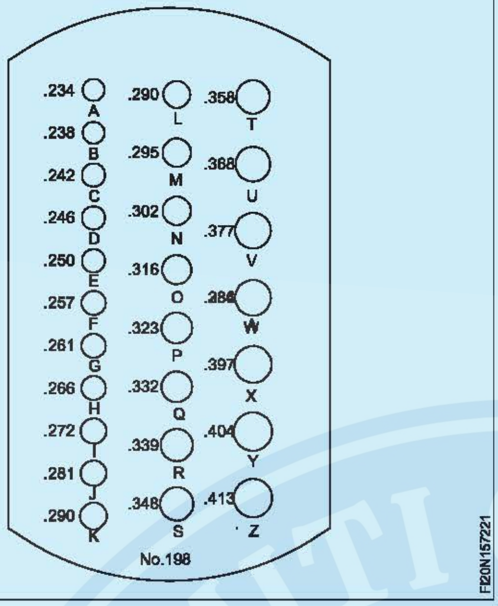
Table 1

Letter drill sizes

Letter	Diameter	
	Inches	mm
A	.234	5.944
B	.238	6.045
C	.242	6.147
D	.246	6.248
E	.250	6.35
F	.257	6.528
G	.261	6.629
H	.266	6.756
I	.272	6.909
J	.277	7.036
K	.281	7.137
L	.290	7.366
H	.266	6.756
I	.272	6.909
J	.277	7.036
K	.281	7.137
L	.290	7.366
M	.295	7.493
N	.302	7.671
O	.316	8.026
P	.323	8.204
Q	.332	8.433
R	.339	8.611
S	.348	8.839
T	.358	9.093
U	.368	9.347
V	.377	9.576
W	.386	9.804
X	.397	10.084
Y	.404	10.262
Z	.413	10.490

In the number drill and the letter drill series, the correct diameter of the drill is gauged with the help of the respective drill gauges. A drill gauge is a rectangular or square shaped metal piece containing a number of different diameter holes. The size of the hole is stamped against each hole. (Fig 1)

Fig 1



Number drills:

The number drill series consists of drills numbered from 1 to 80. The No.1 drill is the largest, with 5.791 mm diameter, and the No.80 drill is the smallest, with 0.35 mm diameter. (Table 2). There is no uniform variation in the drill diameters from number to number. To find the correct diameter of a number drill, refer to a drill Size Chart or a Hand-book. Number drill series are also known as 'wire gauge' series.

Table 2

Number drill sizes

No.	Diameter	
	Inches	mm
1	228	5.791
2	.221	5.613
3	.213	5.410
4	.209	5.309
5	.2055	5.220
6	.204	5.182
7	.201	5.105
8	.199	5.055
9	.196	4.978
10	.1935	4.915
11	.191	4.851
12	.189	4.801
13	.185	4.699
14	.182	4.623
15	.180	4.572
16	.177	4.496
17	.173	4.394
18	.1695	4.305
19	.166	4.216
20	.161	4.089
21	.159	4.039
22	.157	3.988
23	.154	3.912
24	.152	3.861
25	.1495	3.797
26	.147	3.734
27	.144	3.658
28	.1405	3.569
29	.136	3.454
30	.1285	3.264
31	.120	3.048
32	.116	2.946
33	.113	2.870
34	.111	2.819
35	.110	2.794
36	.1065	2.705
37	.104	2.642
38	.1015	2.578
39	.0995	2.527
40	.098	2.489
41	.096	2.438
42	.0935	2.375
43	.089	2.261
44	.086	2.184
45	.082	2.083
46	.081	2.057
47	.0785	1.994
48	.076	1.930
49	.073	1.854
50	.070	1.778
51	.067	1.702
52	.0635	1.613
53	.0595	1.511
54	.055	1.395

No.	Diameter	
	Inches	mm
55	.052	1.321
56	.0465	1.181
57	.043	1.092
58	.042	1.067
59	0.41	1.041
60	.040	1.016
61	0.0390	1.00
62	0.0380	0.98
63	0.0370	0.95
64	0.0360	0.92
65	0.0350	0.90
66	0.033	0.85
67	0.032	0.82

No.	Diameter	
	Inches	mm
68	0.031	0.79
69	0.0292	0.75
70	0.0280	0.70
71	0.0260	0.65
72	0.0240	0.65
73	0.0240	0.60
74	0.0225	0.58
75	0.0210	0.52
76	0.0200	0.50
77	0.0180	0.45
78	0.0160	0.40
79	0.0145	0.38
80	0.0135	0.35

Fraction & Metric sizes of drills conversion table

Inches and millimeteres

(a) Inches to millimteres — Basic: 1 inch = 25.4 millimetres

Inch	0	1/16	1/8	3/16	1/4	5/16	3/8	7/16
0		1.59	3.18	4.76	6.35	7.94	9.53	11.11
1	25.40	26.98	25.58	30.16	31.75	33.34	34.93	36.51
2	50.80	52.39	53.97	55.56	57.15	58.74	60.33	61.91
3	76.20	77.79	79.38	80.96	82.55	84.14	85.73	87.31
4	101.60	103.19	104.78	106.36	107.95	109.54	111.13	112.71
5	127.00	128.59	130.18	131.76	133.35	134.94	136.53	138.11
6	152.40	153.99	155.58	157.16	158.75	160.34	161.93	163.51
7	177.80	179.39	180.98	182.56	184.15	185.74	187.33	188.91
8	203.20	204.79	206.38	207.96	209.55	211.14	212.73	214.31
9	228.60	230.19	231.78	233.36	234.95	236.54	238.13	239.71
10	254.00	255.59	257.18	258.76	260.35	261.94	263.53	265.11
Inch	**1/2**	**9/16**	**5/8**	**11/16**	**3/4**	**13/16**	**7/8**	**15/16**
0	12.70	14.29	15.88	17.46	19.05	20.64	22.23	23.81
1	38.10	39.69	41.28	42.86	44.45	46.04	47.63	49.21
2	63.50	65.09	66.68	68.26	69.85	71.44	73.03	74.61
3	88.90	90.49	92.08	93.66	95.25	96.84	98.43	100.01
4	114.30	115.89	117.48	119.06	120.65	122.24	123.83	125.41
5	139.70	141.29	142.88	144.46	146.05	147.64	149.23	150.81
6	165.10	166.69	168.28	169.86	171.45	173.04	174.63	176.21
7	190.50	192.09	193.68	195.26	196.85	198.44	200.03	201.61
8	215.90	217.49	219.08	220.66	222.25	223.84	225.43	227.01
9	241.30	242.89	244.48	246.06	247.65	249.24	250.83	252.41
10	266.70	268.29	269.88	271.46	273.05	274.64	276.23	277.81

Example: 25 3/4" = { 20' = (10 x 2" = 10 x 50.8 =) 508.00
5 3/4" = 146.05 } = 654.05 mm

(b) Millimteres to Inches — Basic: 1 Millimetre = 0.039369 inch

mm	0	1	2	3	4	5	6	7	8	9
0		0.039	0.079	0.118	0.157	0.197	0.236	0.276	0.315	0.354
10	0.394	0.433	0.472	0.512	0.551	0.591	0.630	0.669	0.700	0.748
20	0.787	0.827	0.866	0.905	0.945	0.984	1.024	1.063	1.102	1.142
30	1.181	1.220	1.259	1.299	1.338	1.378	1.417	1.457	1.496	1.535
40	1.575	1.614	1.653	1.693	1.732	1.772	1.811	1.850	1.890	1.929
50	1.968	2.007	2.047	20.87	2.126	2.165	2.205	2.244	2.283	2.323
60	2.362	2.401	2.441	2.480	2.520	2.559	2.598	2.638	2.677	2.716
70	2.756	2.795	2.835	2.874	2.913	2.953	2.992	3.031	3.074	3.110
80	3.149	3.189	3.228	3.268	3.307	3.346	3.386	3.425	3.464	3.504
90	3.543	3.583	3.622	3.661	3.701	3.740	3.779	3.819	3.858	3.897
mm	**0**	**100**	**200**	**300**	**400**	**500**	**600**	**700**	**800**	**900**
0		3.94	7.87	11.81	15.75	19.68	23.62	27.56	31.49	35.43
1000	39.37	43.30	47.24	51.18	55.12	59.05	62.99	66.93	70.86	74.80
2000	78.74	82.67	86.61	90.55	94.48	98.42	102.36	106.30	110.23	114.17
3000	118.11	122.04	125.98	129.92	133.85	137.79	141.73	145.66	149.60	153.54
4000	157.47	161.41	165.35	169.20	173.22	177.16	181.10	185.03	188.97	192.91
5000	196.84	204.71	212.59	220.38	228.34	236.21	244.09	251.96	259.83	267.71

Example: 2256 mm = { 2200 mm = 86.61'
56 mm = 2.204" } =88.814"

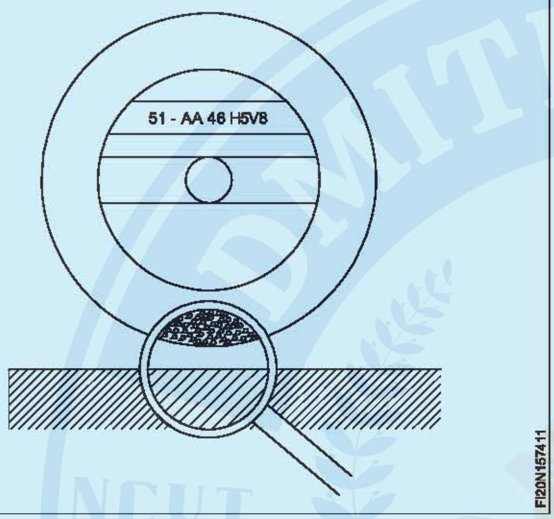
Standard marking system for grinding wheels

Objectives: At the end of this lesson you shall be able to
- **interpret the marking on a grinding wheel**
- **specify a grinding wheel.**

Introduction

Standard wheel - markings specify all the important wheel characteristics. The marking system comprises of seven symbols which are arranged in the following order. (Fig 1)

Fig 1



Example (Marking system)

51 - A 46

Specification of grinding wheels

A grinding wheel is specified by the standard wheel markings like diameter of th

e wheel, bore diameter of the wheel, thickness of the wheel type (Shape) of the wheel.

Example

32 A 46 H8V

250X20X32-

Straight wheel

Table 1 shows the relative position measuring of the marking system

Table 1

Position 0	**Position 1**	**Position 2**	**Position 3**	**Position 4**	**Position 5**	**Position 6**
Manufac-turer's symbol for abrasive (Optional)	Type of abra-sive grit size	Grain size	Grade	Structure (Optional)	Type of bond	Manufac-turer's own mark (Optional)
51	A	46	H	5	V	8

Chart illustrating the standard marking system is : 551-1966 (Table - 2)

Table 2

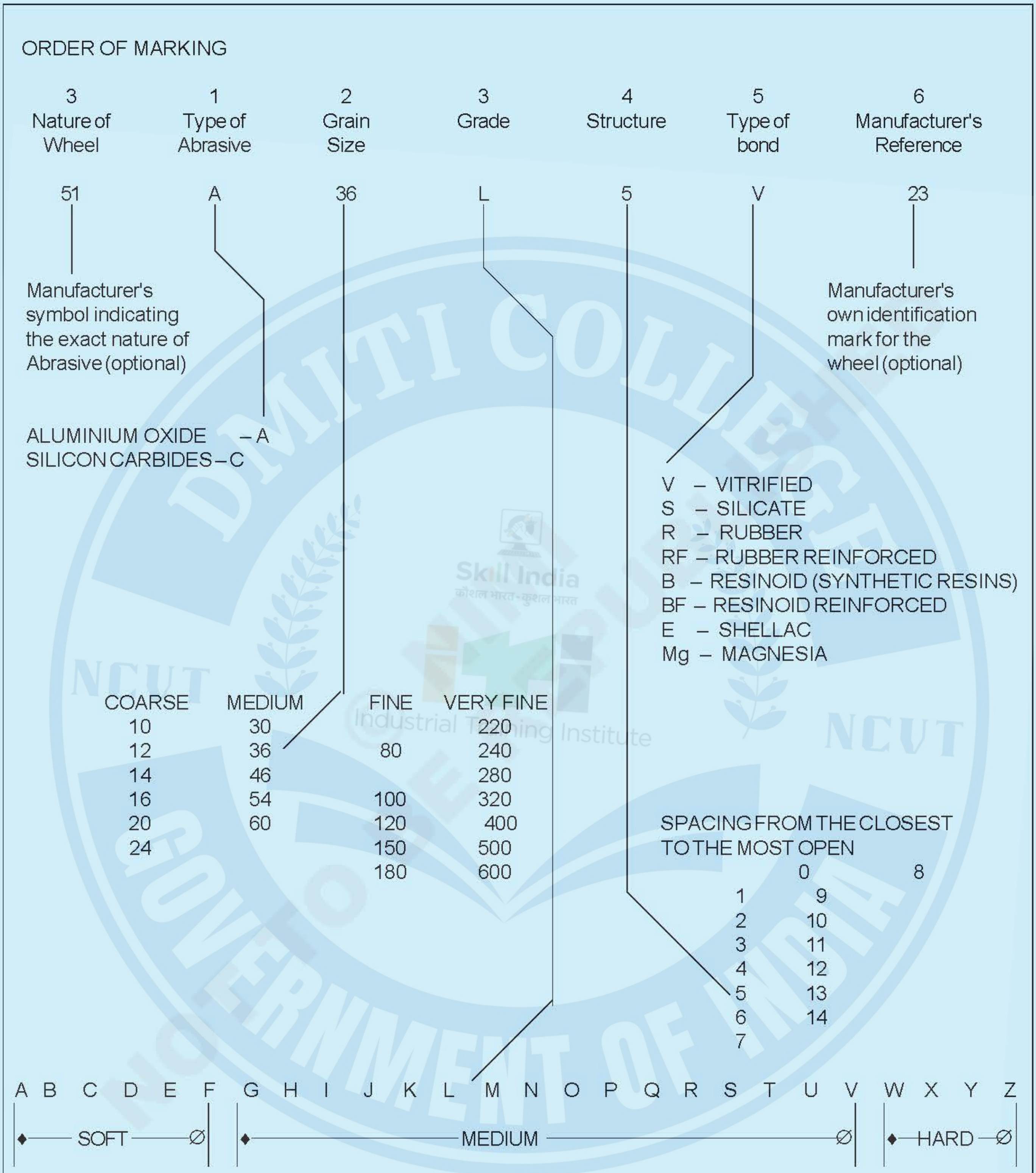


CHART ILLUSTRATING STANDARD MARKING SYSTEM IS : 551 – 1966

Construction of the grinding wheel

Objectives: At the end of this lesson you shall be able to
- **state the different types of abrasives and their uses**
- **state the different grain sizes and their uses**
- **state the different grades of grinding wheels**
- **state the structure of a grinding wheel**
- **name the bonding materials used for grinding wheels.**

In order to suit the grinding wheel for different work situations, the features such as abrasive, grainsize, grade, structure and bonding materials can be varied.

A grinding wheel consists of the abrasive that does the cutting, and the bond that holds the abrasive particles together.

Abrasives

There are two types of abrasives.

- Natural abrasive
- Artificial abrasive

The natural abrasives are emery and corrundum,These are impure forms of aluminium oxide.

Artificial abrasives are silicon carbide and aluminium oxide.

The abrasives are selected depending upon the material being ground.

'Brown' aluminium oxide is used for general purpose grinding of tough materials.

'White aluminium oxide is used for grinding ferrous and ferrous alloys.

'Green' silicon carbide is used for very hard materials with low tensile strength such as cemented carbides.

Grain size (Grit size): The number indicating the size of the grit represents the number of openings in the sieve used to size the grain. The larger the grit size number, the finer the grit.

Grade: Grade indicates the strength of the bond and, therefore, the 'hardness' of the wheel. In a hard wheel the bond is strong, and securely anchors the grit in place and, therefore, reduces the rate of wear. In a soft wheel, the bond is weak and the grit is easily detached resulting in a high rate of wear.

Structure: This indicates the amount of bond present between the individual abrasive grains and the closeness of the individual grains to each other. An open structure wheel will cut more freely. That is, it will remove more metal in a given time and produce less heat. It will not produce such a good finish as a closely structured wheel.

Bond: The bond is the substance which, when mixed with abrasive grains, hold them together, enabling the mixture to be shaped to the form of the wheel, and after suitale treatment to take on the necessary mechanical strength for its work. The degree of hardness possessed by the bond is called the 'grade' of the wheel, and indicates the ability of the bond to hold the abrasive grains in the wheel. There are several types of bonding materials used for making wheels.

Vitrified bond: This is the most widely used bond. It has high porosity and strength which makes this type of wheel suitable for high rate of stock removal. It is not adversely affected by water, acid, oils or ordinary temperature conditions.

Silicate bond: Silicate wheels have a milder action and cut with less harshness than vitrified wheels. For this reason they are suitable for grinding fine edge tools, cutters etc.

Shellac bond: This is used for heavy duty, large diameter wheels where a fine finish is required. For example, the grinding of mill rolls.

Rubber bond: This is used where a small degree of flexibility is required on the wheel as in the cutting off wheels.

Retinoid bond: This is used for speed wheels. Such wheels are used in foundries for dressing castings. Retinoid bond wheels are also used for cutting off. They are strong enough to withstand considerable abuse.

Wheel inspection and wheel mounting

Objectives: At the end of this lesson you shall be able to
- **brief steps involved in grinding wheel inspection**
- **state the procedure for mounting of grinding wheel.**

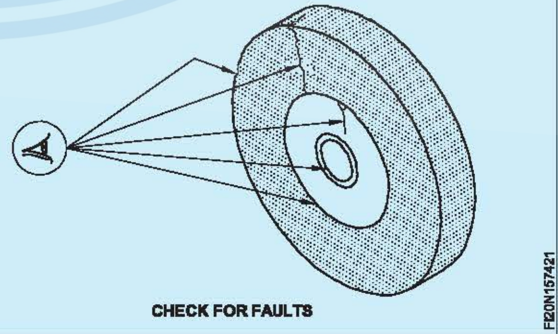
Wheel inspection: The wheel selected may have been damaged during transport or storage and must be carefully inspected before use.

Visual inspection (Fig 1)

Look for

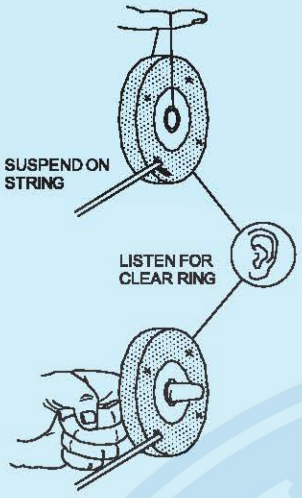
- Broken or chipped edges.
- Cracks
- Damaged mounting bushing
- Damaged paper washers

Fig 1



Testing for cracks (Fig 2)

Fig 2



TAP GENTLY WITH WOODEN HANDLE AT SEVERAL POINTS

Test a wheel for cracks by the following method

- Suspend the wheel on a piece of string or support it with one finger through the bushing.
- Allow the wheel to hang free.
- Tap the wheel with a non-metallic object such as a small wooden mallet or tool handle.
- A clear ringing sound indicates that the wheel is not cracked.
- A dull sound means that the wheel is cracked and must not be used.

> **Warning**
>
> **Discard any wheel that:**
>
> - **Shows any sign of damage.**
> - **Does not ring clearly when struck.**
>
> **If you are in doubt, do not use the wheel. Clearly mark it and seek advice from your supervisor. (Fig 3)**

Mounting the grinding wheel (Fig 4)**:** For correct and safe operation of a grinding machine it is essential to mount the grinding wheel correctly on the spindle.

Before fitting a new wheel, make sure that the spindle is completely clean and free from surface irregularities.

The spindle of the grinding machine includes an inner flange, an outer flange and a nut threaded on the spindle to hold the grinding wheel in position.

The inner flange must be fixed to rotate with the spindle.

Each flange has a dished face towards the surface of the wheel and has a true bearing surface at its area of contact.

Suitable paper discs are normally fitted to the wheel by the manufacturer.

Fig 3

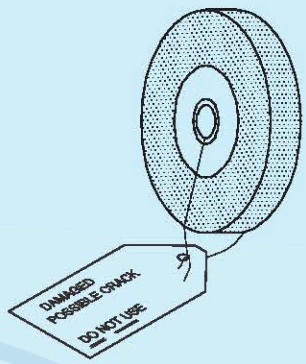
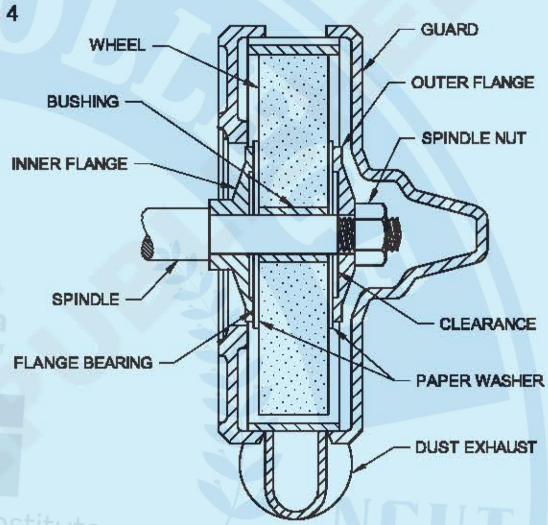


Fig 4



GRINDING WHEEL CORRECTLY MOUNTED

Mounting procedure (Fig 5)

Mount the wheel on the spindle of the grinding machine as follows:

Check that the spindle surface is clean and free of irregularities. Clean with a dry cloth, if necessary.

Check that the inner flange is fixed to the spindle and that its bearing surface is clean and true.

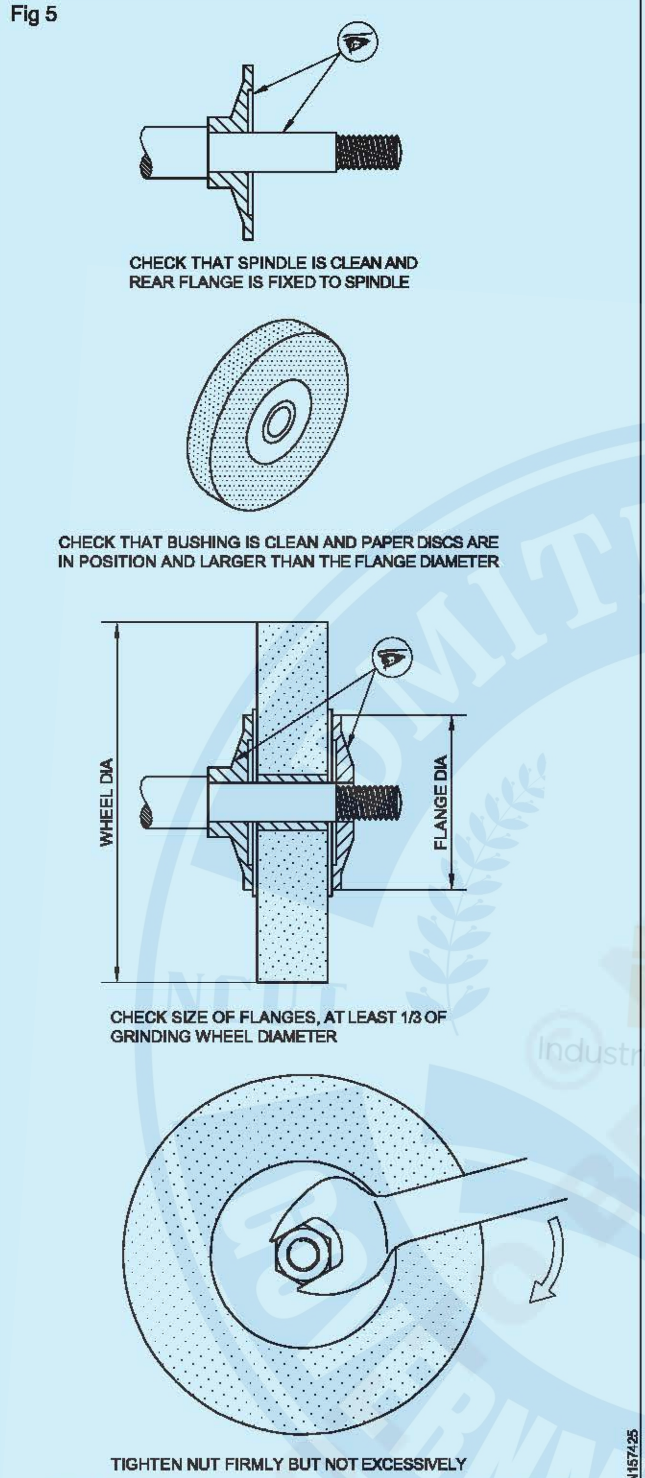
Check that the wheel bush surface is clean and that it can fits easily, but not loosely, onto the spindle. Clean the bush before fitting the wheel on the spindle, if necessary.

Check that each side of the grinding wheel is fitted with a soft paper disc of slightly larger diameter than the spindle flanges.

Check that the diameter of each spindle flange is at least one third the diameter of the grinding wheel.

Fit the grinding wheel to the spindle and place the outer spindle flange in position.

Fig 5



MOUNTING A GRINDING WHEEL ON A GRINDING MACHINE SPINDLE

FI20N157425

Tighten the spindle nut against the outer spindle flange with a spanner of the correct size.

Replace the wheel guard correctly

> **Caution**
>
> **The nut should only be tightened sufficiently to hold the wheel firmly. If it is tightened excessively, the wheel may break.**
>
> **The nut is threaded onto the spindle in a direction opposite to the direction of rotation of the spindle.**

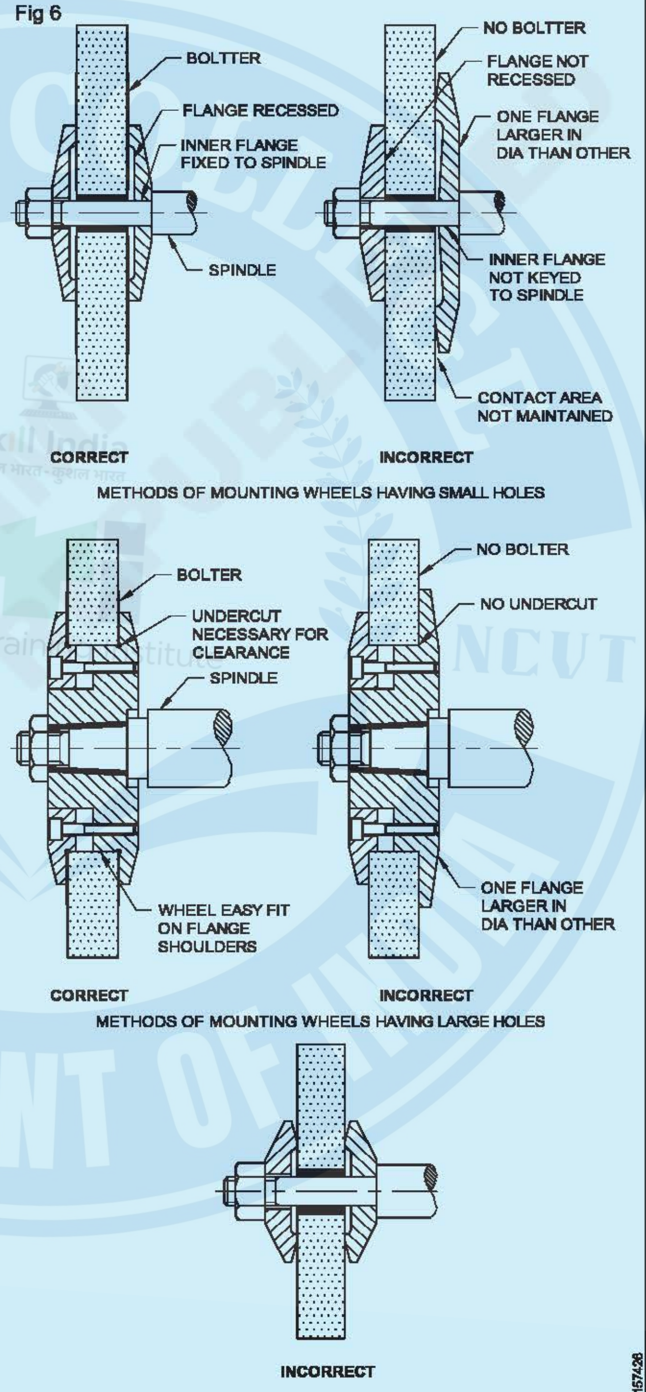
- Run the wheel at its recommended speed in the grinding machine for at least a minute. Do not use the wheel during this period.

Points to note

Study these illustrations carefully and note the points to watch when mounting grinding wheels. (Fig 6)

Washer of compressible material such as card board, leather, rubber etc, not more than 1.5mm thick should be fitted between the wheel and flanges. This prevents any unevenness of the wheel surface is balanced and the tight joint is obtained.

Fig 6



RESULT WHEN SPINDLE END NUT IS EXCESSIVELY TIGHTENED

FI20N157428

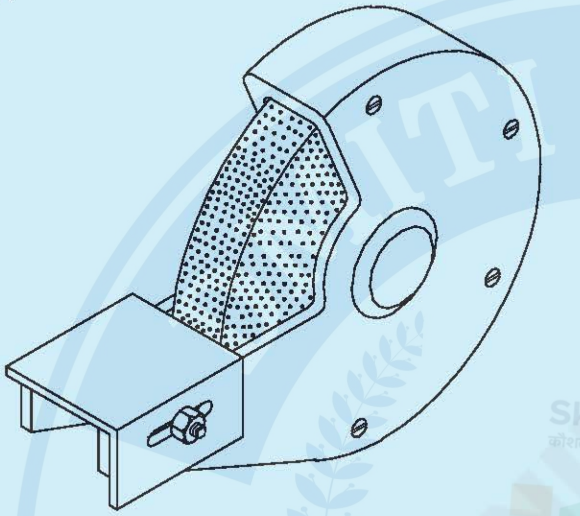
Grinding wheel dressing

Objectives : At the end of this lesson you shall be able to
- **differentiate between loading and glazing**
- **state the effects of loading and glazing**
- **differentiate between dressing and truing.**

Grinding wheels become inefficient due to two main causes known as loading and glazing.

Loading: When soft materials such as aluminium, copper, lead etc. are ground, the metal particles get clogged in the pores of the wheel. This condition is called loading. (Fig 1)

Fig 1



Glazing: When a surface of the wheel develops a smooth and shining appearance, it is said to be glazed. This indicates that the wheel is blunt, i.e. the abrasive grains are not sharp.

When such grinding wheels are used, there is a tendency to exert extra pressure in order to make the wheels cut. Excessive pressure on the grinding wheel will lead to the fracture of the wheel, excessive heating of the wheel, weakening of bonding of the wheel and bursting of the wheel.

Dressing: The purpose of dressing is to restore the correct cutting action of the wheel. Dressing removes the clogs on the surface of the wheel and the blunt grains of the abrasive, exposing the new sharp abrasive grains of the wheel which can be cut and brought to shape efficiently.

Truing: Truing refers to the shaping of the wheel to make it run concentric with the axis. When a new grinding wheel is mounted, it must be trued before use. The cutting surface of a new wheel may run out slightly due to the clearance between the bore and the machine spindle. Grinding wheels, which are in use, also can run out of true, due to uneven loading while grinding.

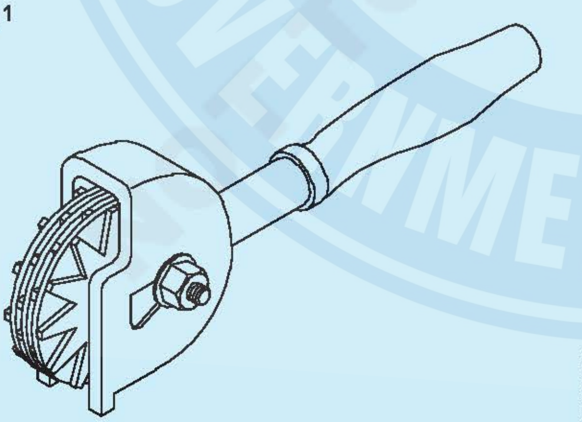
Dressing and truing are done at the same time.

Grinding wheel dressers

Objectives: At the end of this lesson you shall be able to
- **name the common types of wheel dressers**
- **state the uses of each type of wheel dressers.**

The wheel dressers used for off-hand grinders are star wheel dressers (Fig 1) (Huntington type wheel dresser) and diamond dressers.

Fig 1



The star wheel dresser consists of a number of hardened star-shaped wheels mounted on a spindle at one end and a handle at the other end.

While dressing, the star wheel is pressed against the face of the revolving grinding wheel. The star wheel revolves and digs into the surface of the grinding wheel. This releases the wheel loading and dull grains, exposing sharp new abrasive grains.

Star wheels are useful for pedestal grinders in which a precision finish is not expected.

> **Star wheel dressers should be used only on wheels which are large enough to take the load.**

Diamond Dressers (Fig 2)

Bench type off-hand grinders used for sharpening cutting tools are usually fitted with smaller and rather delicate wheels.

These wheels are dressed and trued with diamond dressers.

Diamond dressers consist of a small diamond mounted on a holder which can be held rigidly on the work-rest.

Fig 2

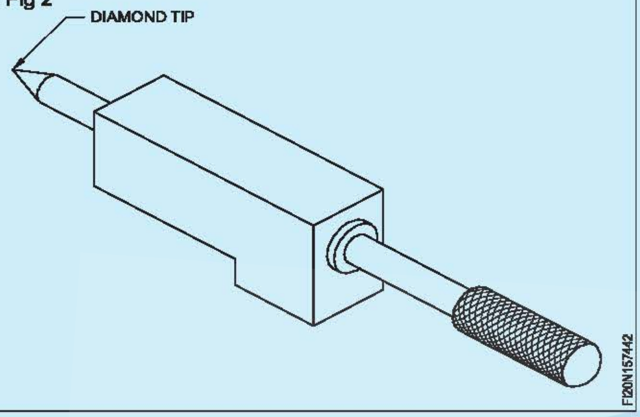
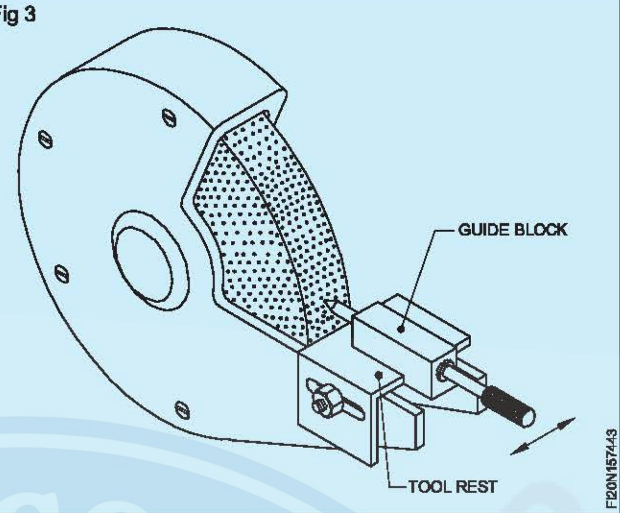


Fig 3



How to use a wheel dresser (Fig 3)

For dressing and truing, the dresser is slowly brought in to contact with the wheel face and moved across.

The finish obtained depends on the rate at which the dresser is moved across the face.

For roughing, the dresser is moved faster.

For fine finish, the dresser is moved slowly.

Roughing will be efficient with a dresser that has a sharp point, while, for fine finishing, a blunt diamond dresser is more suitable.

Abrasive stick: When only a light dressing is required, abrasive sticks can also be used. There are abrasive materials made in the form of sticks for the convenience of handling.

Diamond dressers, if moved too slowly, can glaze the wheel.

Off-hand grinding with bench and pedestal grinders

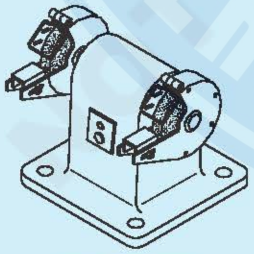
Objectives: At the end of this lesson you shall be able to
- **state the purposes of off-hand grinding**
- **name the machines with which off-hand grinding is done**
- **state the features of bench and pedestal grinders.**

Off-hand grinding is the operation of removing material which does not require great accuracy in size or shape. This is carried out by pressing the workpiece by hand against a rotating grinding wheel.

Off-hand grinding is performed for rough grinding of jobs and resharpening of scribers, punches, chisels, twist drills, single point cutting tools etc.

Off-hand grinders are fitted to a bench and pedestal (Figs 1 and 2)

Fig 1

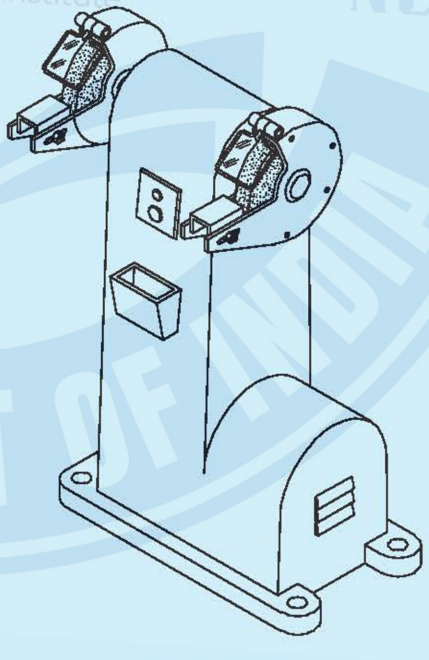


BENCH GRINDER

Bench grinders: Bench grinders are fitted on a bench or table, and are useful for light duty work.

Pedestal grinders: Pedestal grinders are mounted on a base (pedestal), which is fastened to the floor. They are used for heavy duty work.

Fig 2

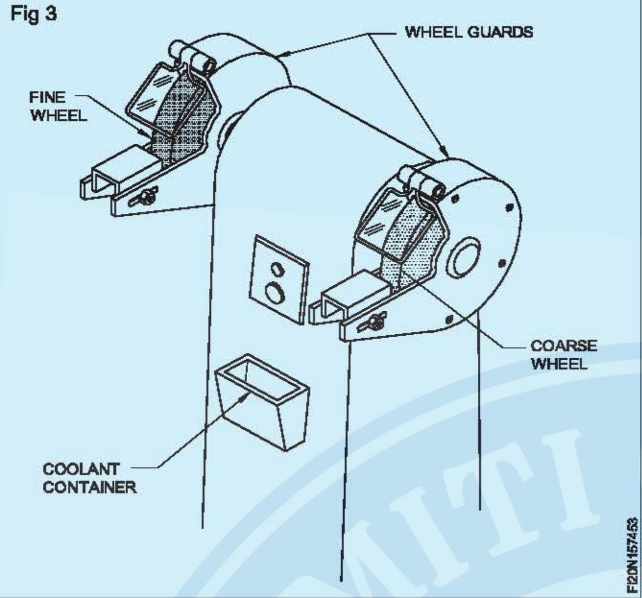


PEDESTAL GRINDER

These grinders consist of an electric motor and the spindle for mounting the grinding wheels. On one end of the spindle a coarse-grained wheel is fitted, and on the other end, a fine-grained wheel. For safety while working, wheel guards are provided.

A coolant container (Fig 3) is provided for frequent cooling of the work.

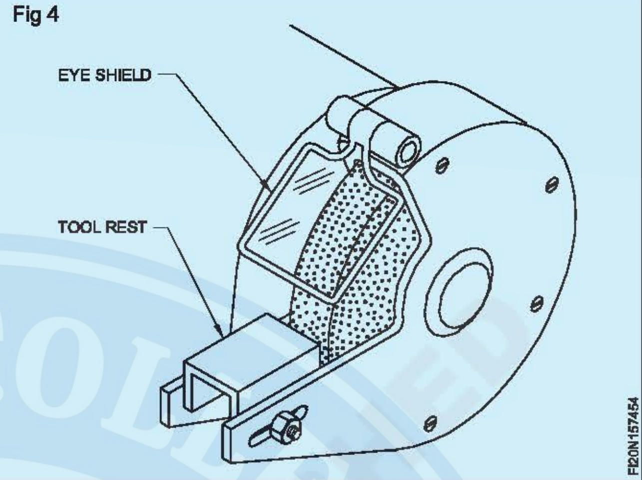
Fig 3



Adjustable work-rests are provided for both the wheels to support the work while grinding. These work-rests must be set very close to the wheels. (Fig 4)

Extra eye shields are also provided for the protection of the eyes. (Fig 4)

Fig 4



DMITI COLLEGE
NCVT
Skill India
Industrial Training Institute
NOT TO BE REPUBLISHED
GOVERNMENT OF INDIA

Gauges and types of gauges

Objectives: At the end of this lesson you shall be able to
- **define template with its uses and advantages**
- **define gauges their necessity and types.**

Gauge: Gauge is an inspection tool used to check product dimension with reference to its maximum and minimum acceptable limits. It is, generally, used to segregate acceptable and non-acceptable products in mass production, without the exact dimensions. It is made of tool steel and is heat treated.

Advantages of gauging

Faster checking of the product is within the specified limits.

Less dependence on operator skill and getting affected by operator judgement.

Gauges are economical when compared to measuring instruments.

Instrument used for gauging

1 Snap and ring gauge
2 Combined gauge
3 Plug gauge
4 Screw pitch gauge
5 Template and form gauge
6 Taper gauge

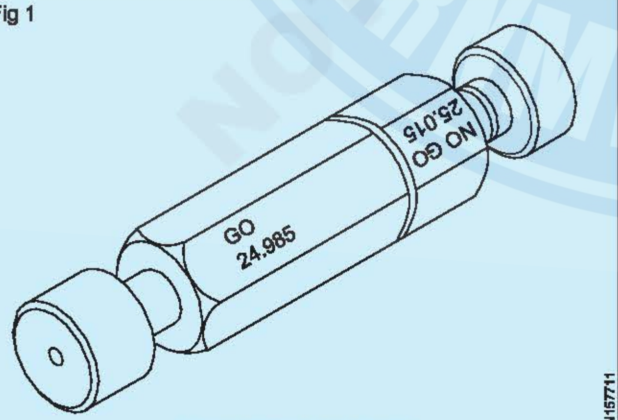
Types of cylindrical plug gauges

Double-ended plug gauge (Fig 1 and 2)

Progressive plug gauge (Fig 3)

Plain cylindrical gauges are used for checking the inside diameter of a straight hole. The 'Go' gauge checks the lower limit of the hole and the 'No- Go' gauge checks the upper limit. The plugs are ground and lapped. (Fig 3)

Fig 1



DOUBLE ENDED PLUG GAUGE

Fig 2

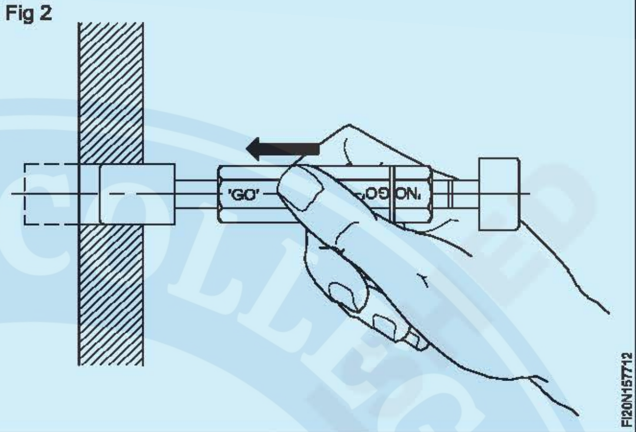
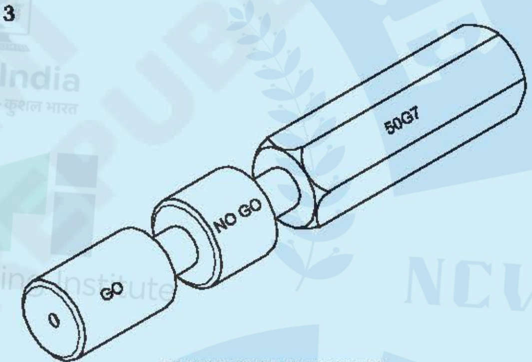


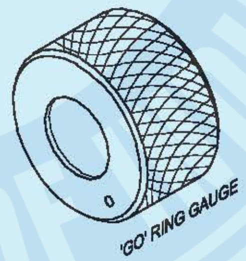
Fig 3



PROGRESSIVE PLUG GAUGE

Plain ring gauge (Fig 4)

Fig 4

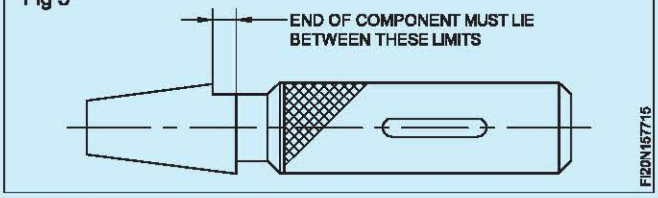


Plain ring gauges are used to check the outside diameter of pieces. Separate gauges are used for checking 'Go' and 'No- Go' sizes. A `No-Go' gauge is identified by an annular groove on the knurled surface.

Taper plug gauges (Fig 5)

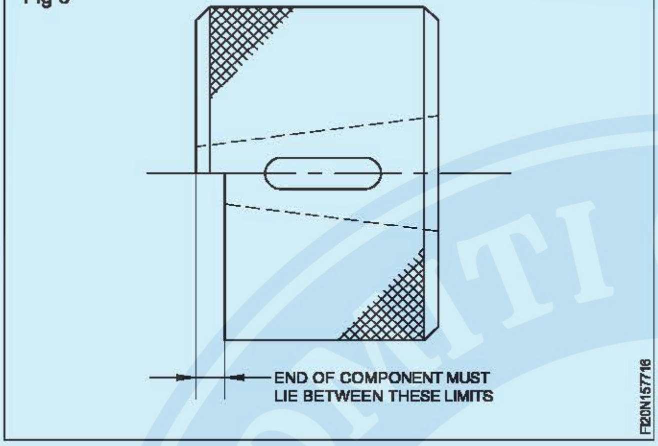
These gauges made with standard or special tapers are used to check the size of the hole and the accuracy of the taper. The gauge must slide into the hole for a prescribed depth and fit perfectly. An incorrect taper is evidenced by a wobble between the plug gauge and the hole.

Fig 5



Taper ring gauges (Fig 6)

Fig 6



They are used to check both the accuracy and the outside diameter of a taper. Ring gauges often have scribed lines or a step ground on the small end to indicate the 'Go' and 'No-Go' dimensions.

Thread plug gauges (Figs 7 and 8)

Fig 7

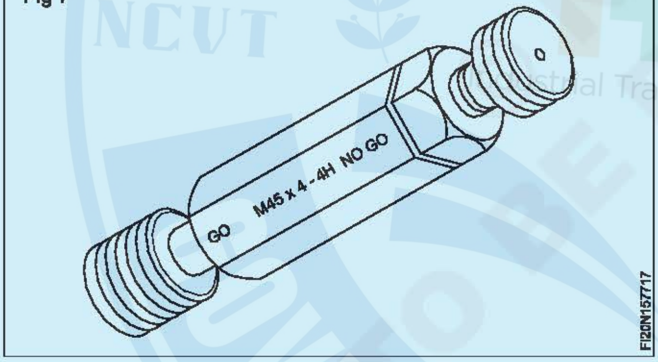
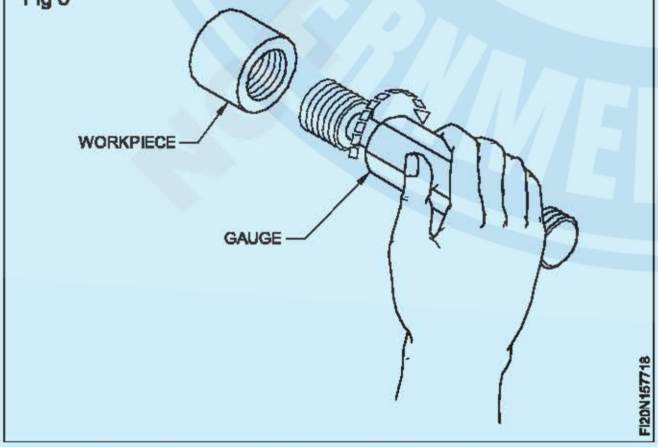


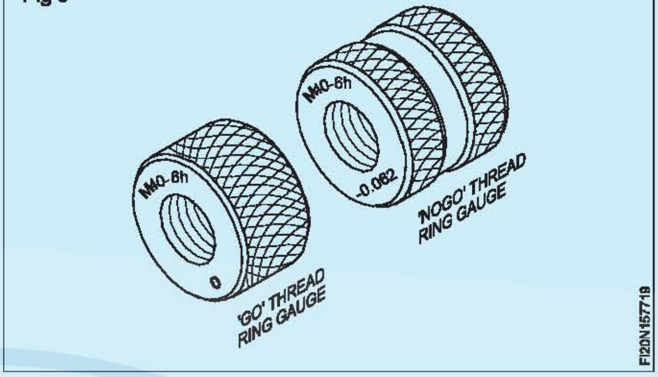
Fig 8



Internal threads are checked with thread plug gauges of 'Go' and `No-Go' variety which employ the same principle as cylindrical plug gauges.

Thread ring gauges (Fig 9)

Fig 9



These gauges are used to check the accuracy of an external thread. They have a threaded hole in the centre with three radial slots and a set screw to permit small adjustments.

Snap gauges (Figs 10, 11, 12 and 13)

Fig 10

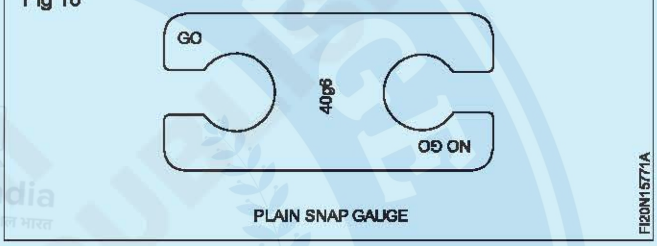
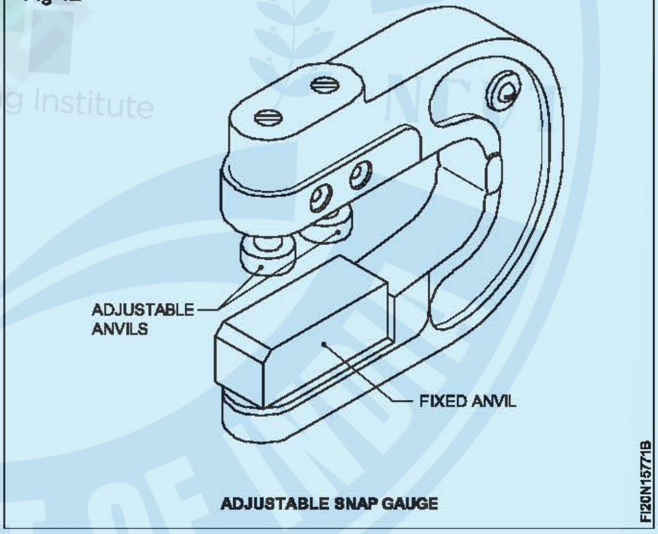


Fig 12



Snap gauges are a quick means of checking diameters and threads to within certain limits by comparing the part's size to the present dimension of the snap gauge.

Snap gauges are generally C-shaped and are adjustable to the maximum and minimum limits of the part being checked. When in use, the work should slide into the 'Go' gauge but not into the 'No-Go' gauging end.

Fig 12

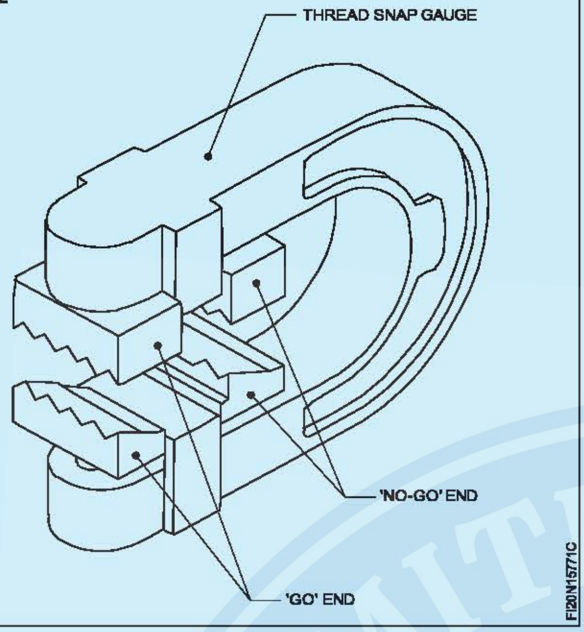


Fig 13

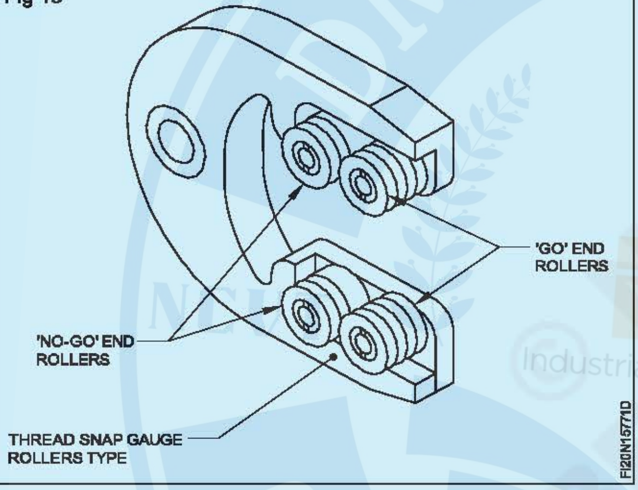


Fig 14

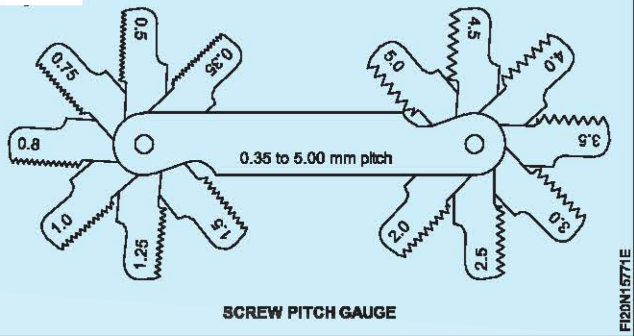
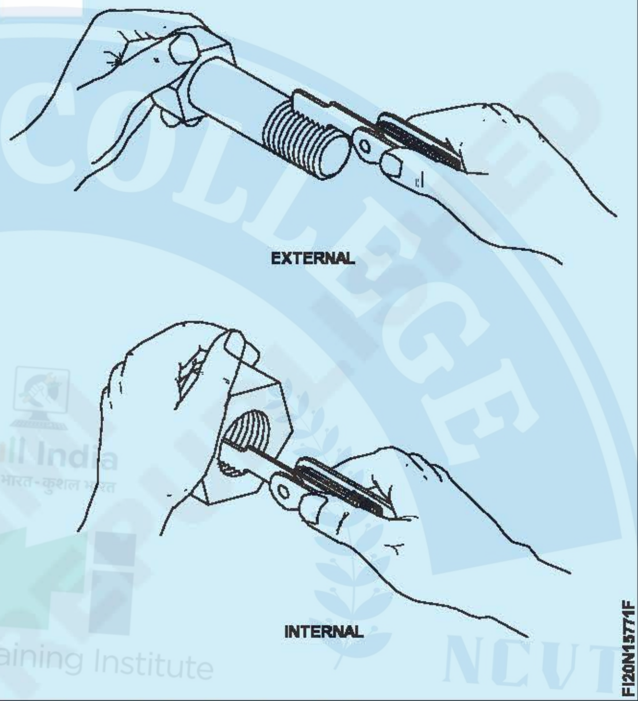


Fig 15



Screw pitch gauge

Purpose

A screw pitch gauge is used to determine the pitch of a thread.

It is also used to compare the profile of threads.

Constructional features

Pitch gauges are available with a number of blades assembled as a set. Each blade is meant for checking a particular standard thread pitch. The blades are made of thin spring steel sheets, and are hardened.

Some screw pitch gauge sets will have blades provided for checking British Standards threads (BSW, BSF etc.) at one end and the metric standard at the other end.

The thread profile on each blade is cut for about 25 mm to 30 mm. The pitch of the blade is stamped on each blade. The standard and range of the pitches are marked on the case. (Fig 14)

For obtaining accurate results while using the screw pitch gauge, the full length of the blade should be placed on the threads. (Fig 15)

Simple and standard workshop gauges

Radius and fillet gauges: Components are machined to have curved formation on the edges or at the junction of two steps. Accordingly they are called radius and fillets. The size of the radius and radius is normally provided on a drawing. The gauges used to check the radius formed on the edges of diameters are fillet and the gauges used to check the fillets are called fillets gauges.

They are made of hardened sheet metal each to a precise radius. They are used to check the radii by comparing the radius on a part with the radius of the gauges.

Fig 16 shows the application of radius gauge to check the radius formed externally. Fig 17 shows the application of a fillet gauge to check the fillet formed on a turned component. The other typical applications are:

Some sets have provisions to check the radius and fillet on each blade. (Fig 18)

And some sets have separate sets of blades to check the radius and fillet. (Fig 19)

Each blade can be swung out of the holder separately, and has its size engraved on it. (Fig 20)

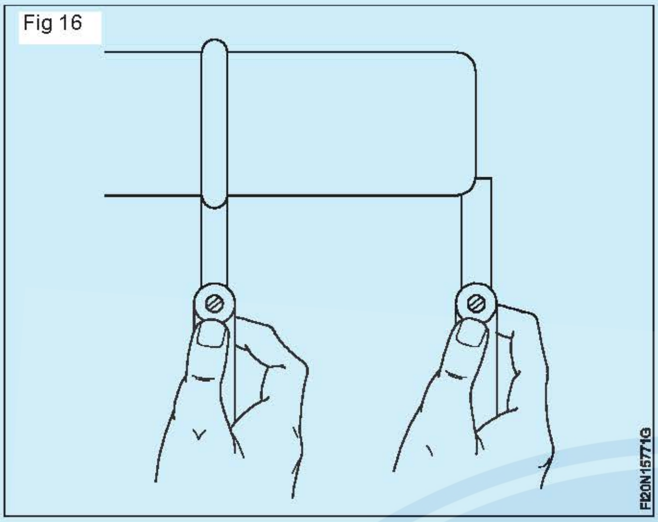


Fig 16

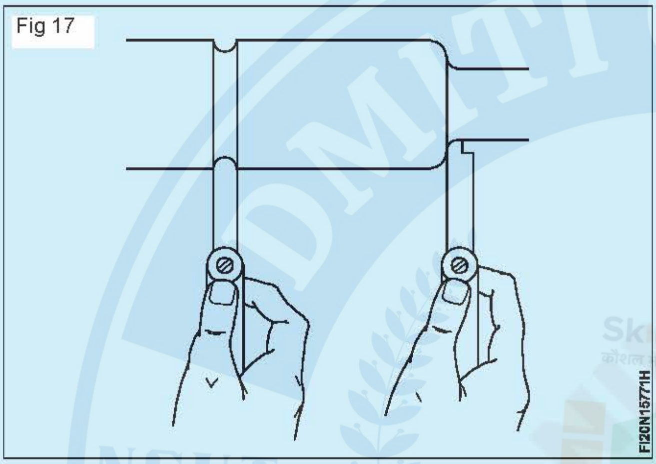


Fig 17

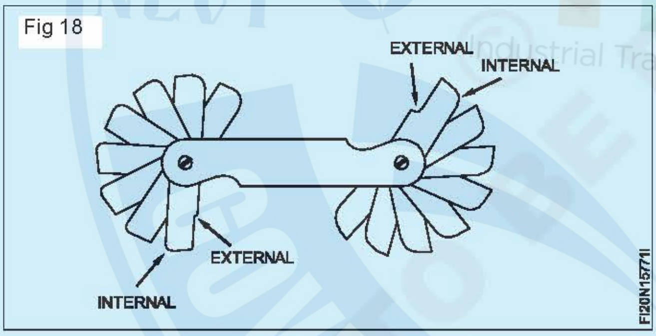


Fig 18

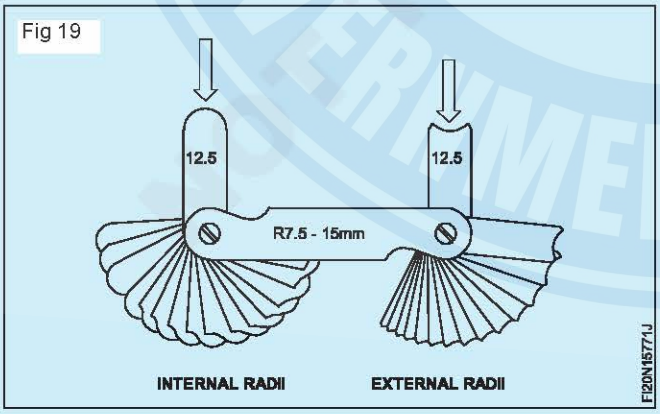


Fig 19

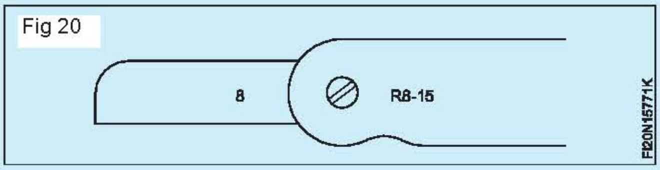


Fig 20

Fillet gauges are available in sets to check the radii and fillets from:

1 to 7 mm in steps of 0.5 mm

7.5 to 15 mm in steps of 0.5 mm

15.5 to 25 mm in steps 0.5 mm.

Individual gauges are also available. They usually have internal and external radii on each gauge and are made in sizes from 1 to 100 mm in steps of 1 mm. (Fig 21)

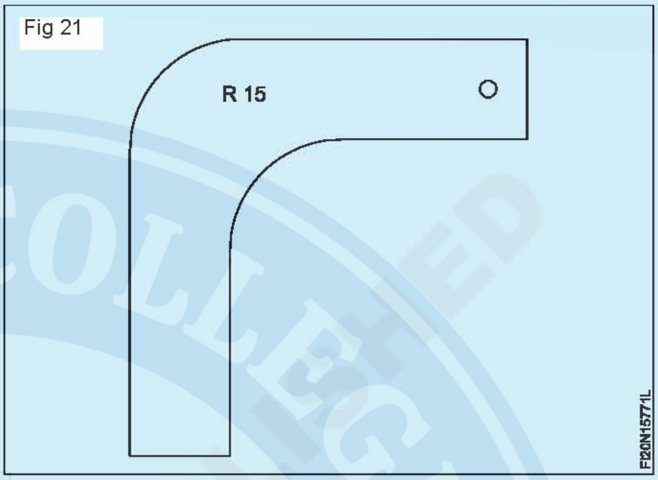


Fig 21

Before using the radius gauge, check that it is clean and undamaged.

Remove burrs from the workpiece.

Select the leaf of the gauge from the set corresponding to the radius to be checked.

Fig 22 shows that the radius of the fillet and that of the external radius are smaller than the gauge.

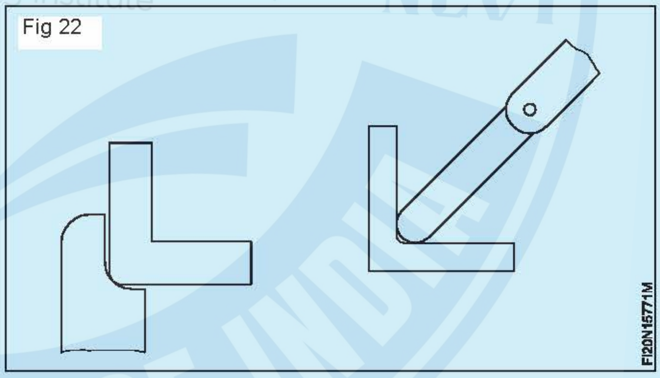


Fig 22

Feeler gauge and uses

Features: A feeler gauge consists of a number of hardened and tempered steel blades of various thicknesses mounted in a steel case. (Fig 23)

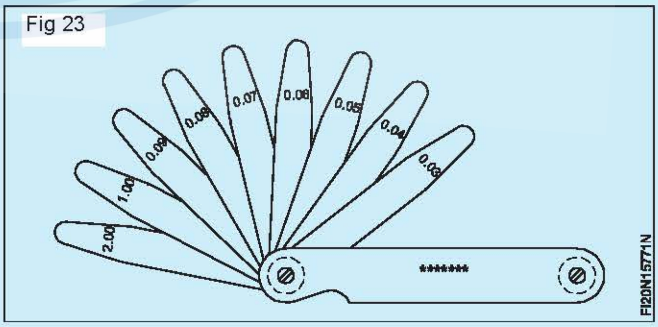


Fig 23

The thickness of individual leaves is marked on it. (Fig 23)

B.I.S. Set: The Indian Standard establishes four sets of feeler gauges Nos. 1,2,3 and 4 which differ by the number of blades in each and by the range of thickness (minimum is 0.03 mm to 1 mm in steps of 0.01 mm). The length of the blade is usually 100 mm.

Example

Set No.4 of Indian Standard consist of 13 blades of different thicknesses.

0.03, 0.04, 0.05, 0.06, 0.07, 0.08, 0.09, 0.10, 0.15, 0.20, 0.30, 0.40, 0.50.

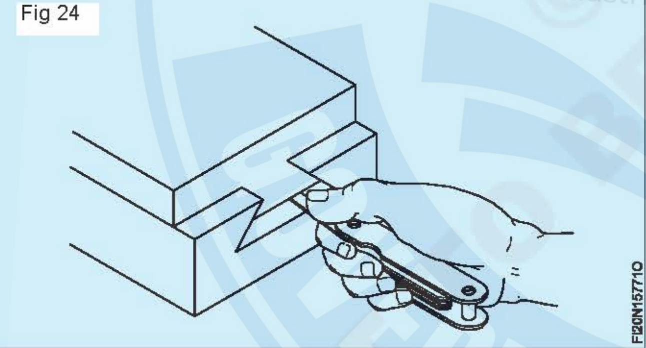
The sizes of the feeler gauges in a set are carefully chosen in order that a maximum number of dimensions can be formed by building up from a minimum number of leaves.

The dimension being tested is judged to be equal to the thickness of the leaves used, when a slight pull is felt while withdrawing them. Accuracy in using these gauge requires a good sense of feel.

Feeler gauges are used:

- To check the gap between the mating parts
- To check and set the spark plug gaps
- To set the clearance between the fixture (setting block) and the cutter/tool for machining the jobs
- To check and measure the bearing clearance, and for many other purposes where a specified clearance must be maintained. (Fig 24)

Fig 24



Drill gauge: A drill gauge is a rectangular or square shaped metal piece containing a number of different diameter holes. The size of the hole is stamped against each hole. (Fig 25)

In the number drill and letter drill series, the diameter of the drill is gauged with the help of the respective drill gauge.

Standard Wire Gauge (SWG): It is used to measure the size of a wire and thickness of sheet shown in Fig.26

Fig 25

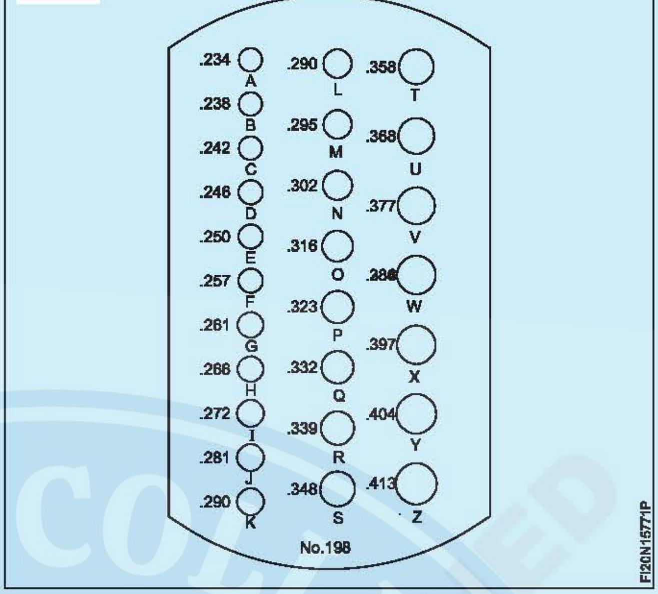
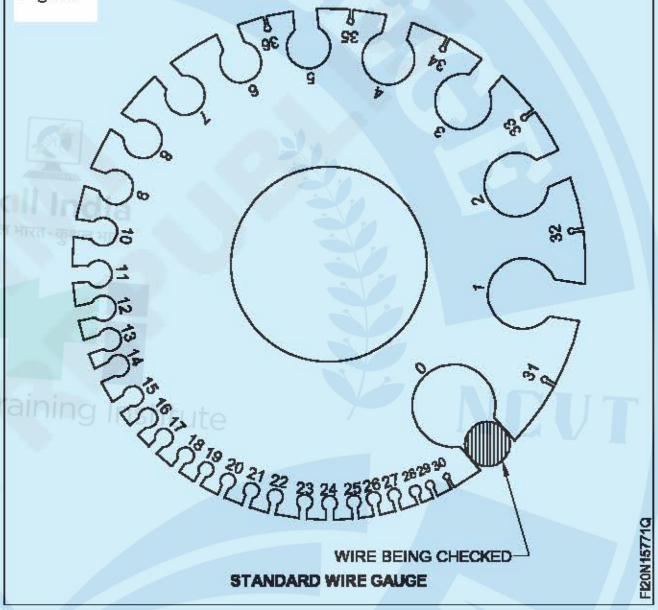


Fig 26



The standard wire gauge is a circular metal disc with varying hole and slot size on its circumference. Each slot size corresponds to a gauge number which is written just below the hole.

The gauge numbers specify the size of a round wire in terms of its diameter.

As the gauge number increase from 0 to 36, the dia size decrease.

The thickness of sheet metal and the diameter of wires confirm to various gauging numbers and the following Table 1 give the decimal equivalents of the different gauge numbers for the diameter of wires, and the thickness of sheets.

Table 1

Standard wire gauge number and equivalent value in mm as per IS 5049-1969

Wire No. according to SWG	Wire Dia according to IS:280-1962 in mm
0	8.00
2	7.10
3	6.30
4	6.00
5	5.60
6	5.00
7	4.50
8	4.00
9	3.55
10	3.15
11	2.80
12	2.50
13	2.24
14	2.00
15	1.80
16	1.60
17	1.40
18	1.25
-	1.12

Wire No. according to SWG	Wire Dia according to IS:280-1962 in mm
19	1.00
20	0.90
21	0.80
22	0.710
23	0.630
24	0.560
25	0.500
26	0.450
27	0.400
29	0.355
30	0.315
32	0.280
33	0.250
34	0.224
36	0.200
37	0.180
38	0.160
39	0.140
40	0.125