Drills - different parts, types and sizes

Objectives : At the end of this lesson you shall be able to
- **state what is drilling**
- **state the necessity of drilling**
- **list the types of drills used**
- **name the parts of a twist drill**
- **list the defects in a drilled hole**
- **state the causes and the remedies for the defects.**

Drilling

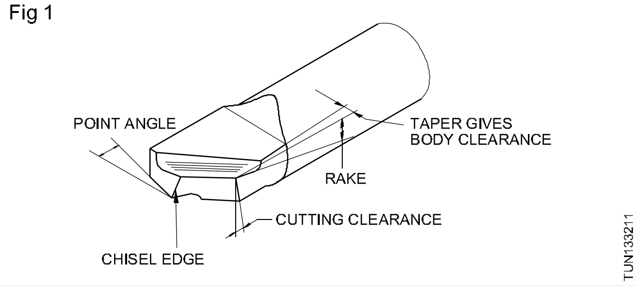
Drilling is the production of cylindrical holes of definite diameters in workpieces by using a multi-point cutting tool called a 'drill'. It is the first operation done internally for any further operation.

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. But it is difficult to hold during operation, and the chip removal is poor. Its operating efficiency is very low.

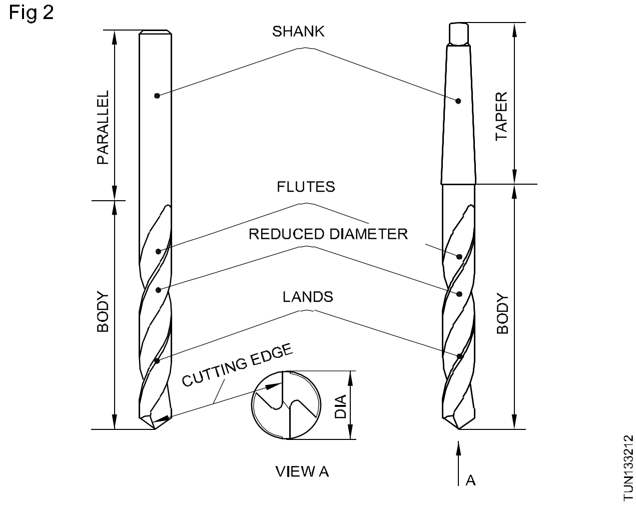
Fig 1



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. Twist drills are available in standard sizes. Parallel shank twist drills are available below 13mm size. (Fig 2)

Fig 2

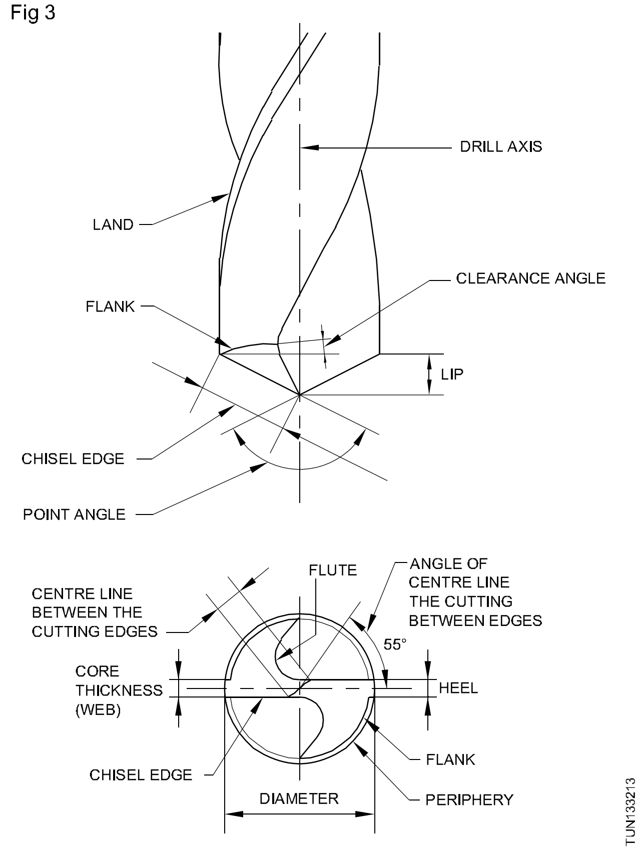


Parts of a twist drill

Drills are made from high speed steel. The spiral flutes are machined at an angle of 271/2° 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



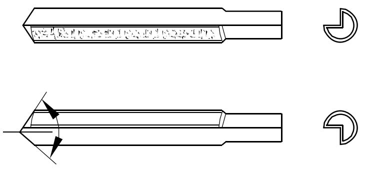
The portions left between the flutes are called 'lands'. The size of a drill is determined and governed by the diameter over the lands.

The point angle is the cutting angle, and for general purpose work, it is 118°. The clearance angle 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)

Fig 4



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)

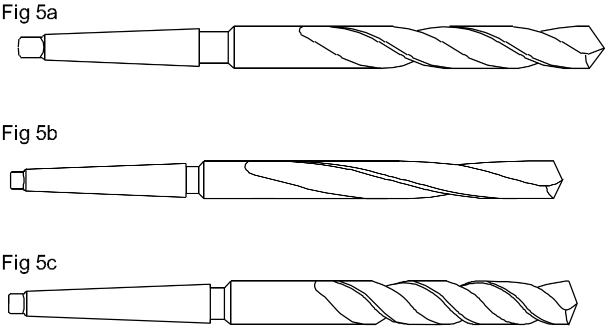
A slow helix drill is used on materials like brass, gun-metal, phosphor-bronze and plastics. (Fig 5b)

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

Fig 5a

Fig 5b

Fig 5c



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

Parallel shank drills can be held only in drill chucks. Taper shank twist drills are mounted directly in the tailstock barrel or fitted to a sleeve and mounted. The tang in the taper shank twist drill provides a positive drive. When inserting a taper shank twist drill in a socket or sleeve, tap the sleeve with a hide mallet. To remove the drill from the socket, a drift is used.

Factors governing drilling operations

The three factors governing drilling operations are:

- cutting speed
- feed pressure
- cooling method.

Cutting speed for drilling

The cutting speed for drilling is the peripheral speed of the drill, and it is stated in metres per minute. The cutting speed depends upon the machinability of the work material. When the cutting speed for drilling a material is determined, the revolutions for which the lathe has to be set during drilling is calculated by the formula.

$$V = \frac{\pi \times D \times N}{1000}$$

Feed

The rate at which the drill advances into the material for each revolution of the drill is known as the feed rate and it is expressed in mm/rev. The feed rate selection also depends upon the machinability of the metal being drilled:

Drill grinding

Any one of the following indicates the sign that the drill needs re-sharpening.

- A need for high feed pressure to make the drill to cut and advance.
- Chattering or screaming of the drill when pressure is applied.

It is recommended that a drill grinding jig is to be used for re-sharpening purposes as it is almost impossible to grind the drills to the correct angles by off-hand grinding. But for general purpose drilling, off- hand grinding may also be done, taking care to avoid the following faults.

Grinding faults

Faulty grinding is indicated by the following.

The two cutting edges are of unequal length. This fault causes two cuttings of unequal thickness, or one cutting being ejected and an oversize hole results. (Figs 6 and 7)

Fig 6

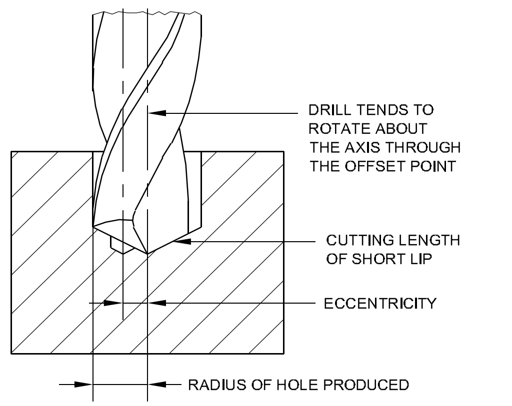
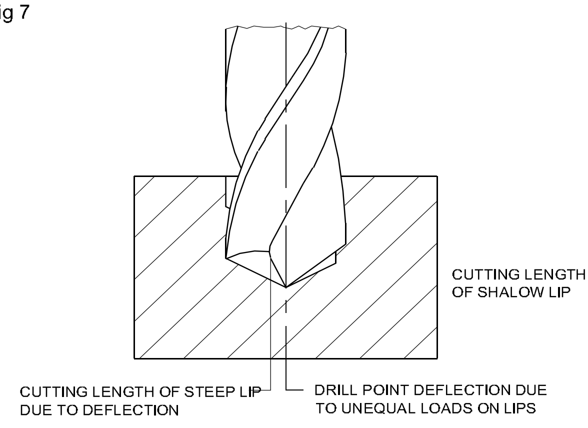
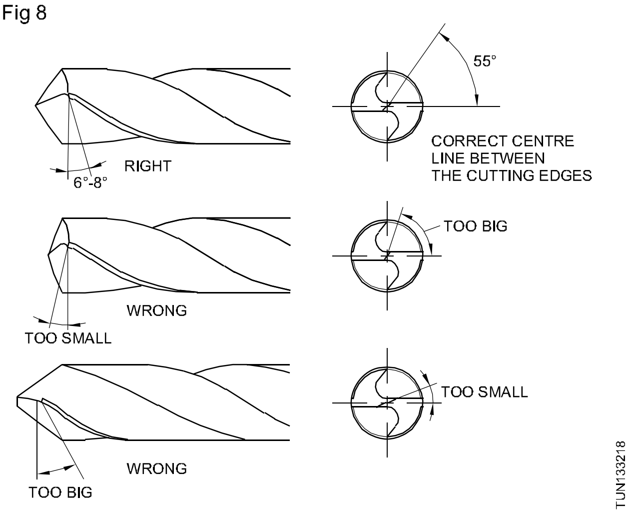


Fig 7



Excessive clearance angle ground. This fault will cause the cutting edges to chip off and break. This in turn, will cause the drill to dig into the workpiece. (Fig 8)

Fig 8



Insufficient clearance angles. The drill rubs rather than cuts. When a drill gets badly worn out, it cuts poorly and the three signs of a badly worn out drill are :

blunt cutting edges'at the point of the drill

- too much feeding pressure to make the drill to cut
- work and drill getting heated up.

It is essential to provide a bigger point angle of the drill for drilling hard metals.

Recommended helix point angles for drilling different : Work materialsPilot hole drilling

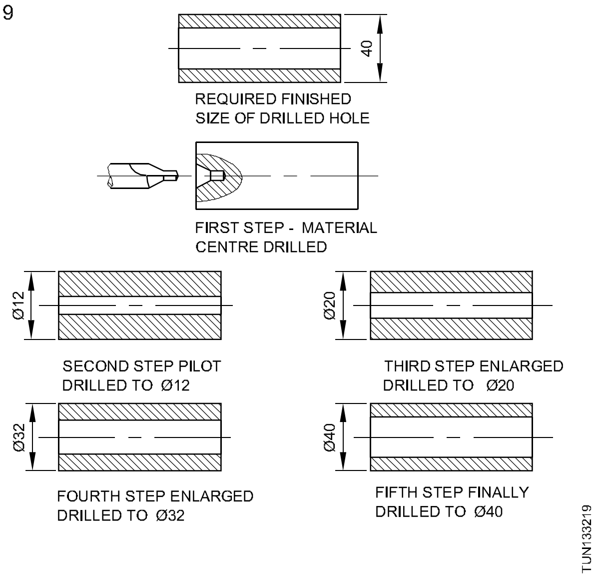
(For guidance only)

Material of the workpiece	Helix angle angle	Point angle
Steel, alloyed and unalloyed cast steel, cast iron, malleable iron	28°	118°
White cast iron	28°	150°
Brass, bronze	15°	118° to 140°
Bakellite	40°	118° to 140°

For producing large size holes by drilling, it is always advantageous to drill with smaller drills and finally use the drill of the required size. This operation is known as pilot drilling.

The steps to finish a required size of a drilled hole are shown in Fig 9.

Fig 9



Cutting fluids for drilling

The use of an appropriate cutting fluid will always give a better surface finish; it permits the use of a higher cutting speed, and extends the tool life.

The cutting fluids generally used for drilling operation are the same that are used for other lathe operations.

Soluble oil is the most commonly used cutting fluid.

Recommended cutting fluids for drilling different metals

- Aluminium and its alloys - dry or kerosene.
- Copper - soluble oil.
- Brass - dry or soluble oil.
- Cast iron- dry or cooled with compressed air.
- Chilled cast iron - soluble oil.
- Mild steel - soluble oil, sulphurised oil.
- Alloy steels - soluble oil, sulphurised oil.

Drill cutting angle, cutting speed

Objectives : At the end of this lesson you shall be able to
- **state what is drilling**
- **state the necessity of drilling**
- **name the types of drills used**
- **name the parts of a twist drill**
- **list out the defects in a drilled hole**
- **state the causes and the remedies for the defects.**

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

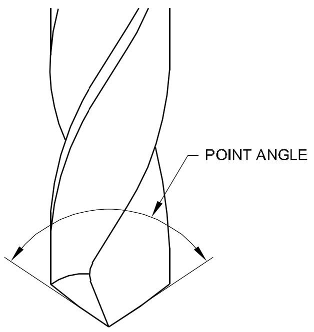
Angles

There 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)

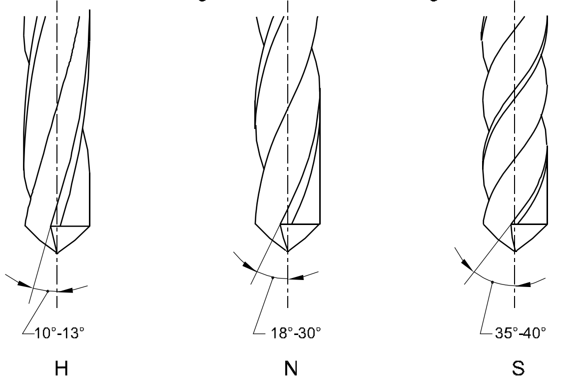
Fig 1



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.

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.

Fig 5

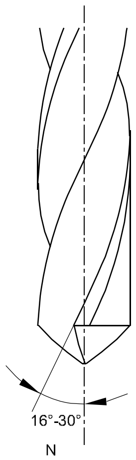
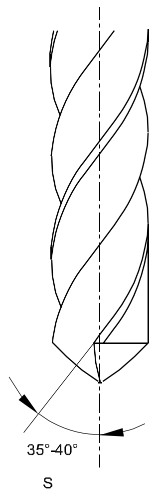


Fig 6



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

Type N = for normal low carbon steel.

Type H = for hard and tenaceous materials.

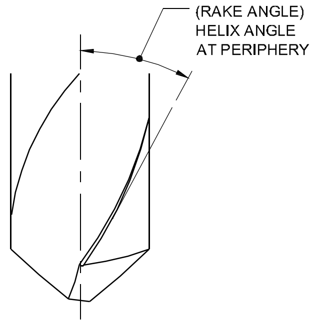
Type S = for soft and tough materials.

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

Rake angle (Fig 7)

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

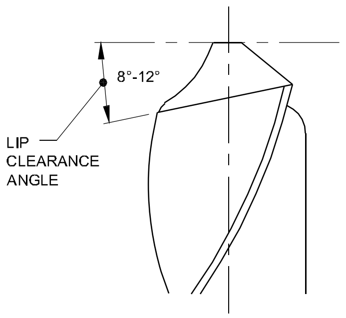
Fig 7



Clearance angle (Fig 8)

Clearance angle is to prevent the friction of the land 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, if it is too small, the drill will not cut.

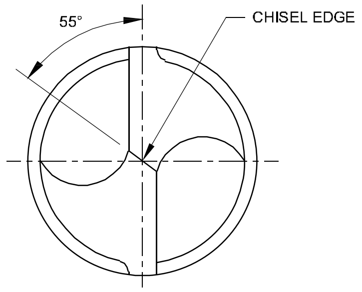
Fig 8



Chisel edge angle/Web angle (Fig 9)

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

Fig 9



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

Diameter of drill

IS NO.

Twist drill 9.50 - H - IS5101 - HS

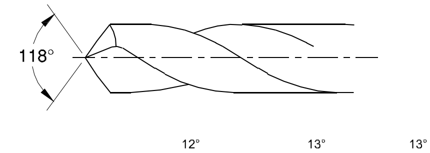
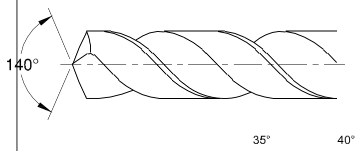
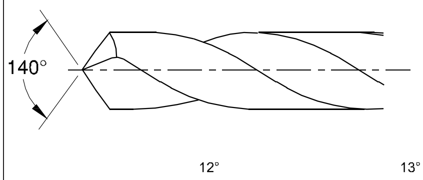
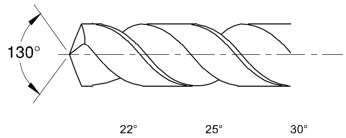
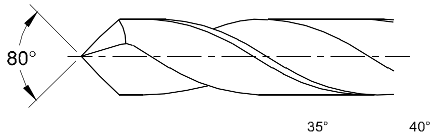
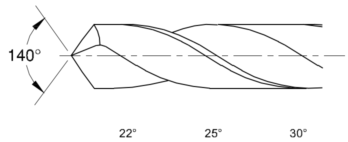
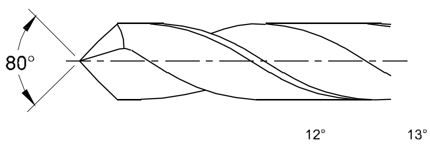
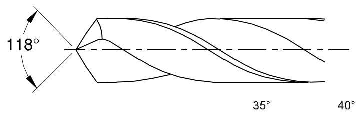
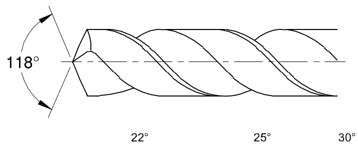
Tool type

Material

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

Drills for different materials

Recommeded drills

Material to be drilled	Point angle / Helix angle d=3.2.5 5-10 10	Material to be drilled	Point angle / Helix d=3.5 mm 5 mm
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 - alloy, forming Curly chips Celluloid	
Brass, Cuzn 40		Austenitic steels Magnesium alloys	
Steel, and cast steel 70... 120 Kgf/mm^2		Moulded steels (with thickness S>d)	
Stainless steel Copper (drill diameter more than 30 mm) Al-alloy, forming short borken chips		Moulded plastics, with thickness s>d Laminated plastics Hard rubber (ebonite) Marble, state, coal	
		Zinc alloys	

Cutting speed and RPM

Objectives : At the end of this lesson you shall be able to
- **define cutting speed**
- **state the factors for determining the cutting speed**
- **differentiate between cutting speed and r.p.m.**
- **determine r.p.m. spindle speed**
- **select r.p.m. for drill sizes from tables.**

For a drill to give satisfactory performance, it must operate at the correct cutting speed and feed.

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. Based on the cutting speed recommended, the r.p.m., at which a drill has to be driven, is determined.

Materials being drilled for HSS	Cutting speed(m/min)
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

Calculating r.p.m.

$$v = \frac{n \times d \times \pi}{1000} \text{ m/min}$$

$$n = \frac{v \times 1000}{d \times \pi} \text{ r.p.m.}$$

n - r.p.m.

v - Cutting speed in m/min.

d - diameter of the drill in mm

π = 3.14

Example

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

The cutting speed for MS 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 selected spindle speed is 300 r.p.m.

The r.p.m. will differ according to the diameter of 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 experiments.

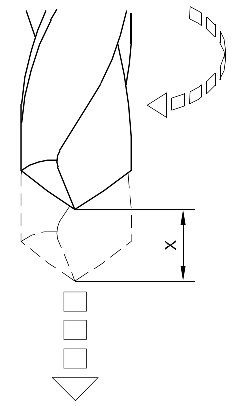
Feed in drilling

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 (X) a drill advances into the work in one complete rotation. (Fig 1)

Fig 1



Feed is expressed in hundredths of a millimetre.

Example - 0.040mm.

The rate of feed is dependent upon a number of factors.

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

Factors like rigidity of the machine, holding of workpiece 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 for the feed rate given here is based on the average feed values suggested by the different manufacturers of drills. (Table 1)

Table 1

Drill diameter	Rate of feed
(mm) H.S.S.	(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 -15.1	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

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.

Boring tools, counter sinking

Objectives : At the end of this lesson you shall be able to
- **identify and name the different types of boaring tools**
- **list out the advantages of the different boring tools.**

Boring is the process of enlarging and truing an existing drilled or core hole with a single point cutting tool.

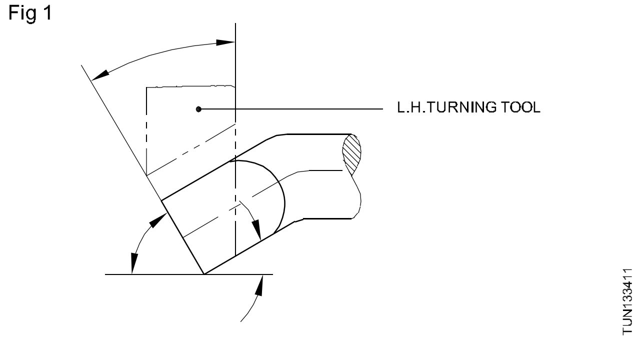
Necessity of boring a hole

- To enlarge a drilled hole larger than the drill size as drills are available in standard sizes only.
- To obtain concentricity of the hole.
- To maintain accuracy of the hole.
- To obtain better surface finish.
- To remove the error formed by drilling, and to facilitate the reaming operation.

Boring tools and holders

Boring is an internal operation performed on the drilled or cored holes. The cutting edge of a boring tool is ground similar to the left hand plain turning tool. But the operation being performed is from right to left. (Fig 1)

Fig 1



Parts of a rough boring tool (Figs 1,2 & 3)

Fig 2

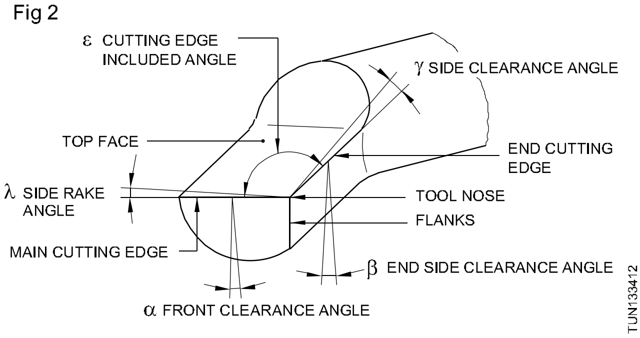
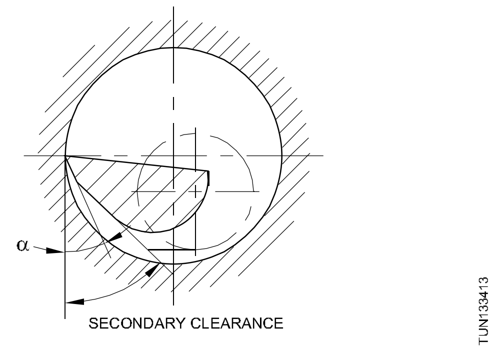


Fig 3

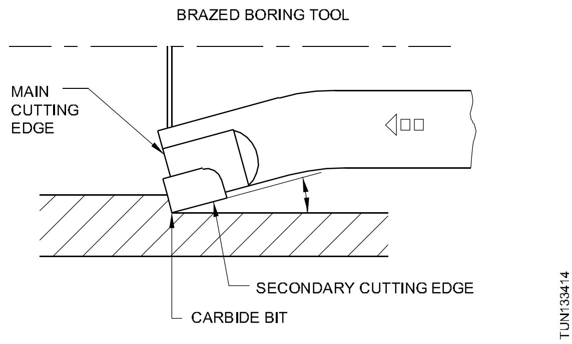


Types of boring tools

The following are the different types of boring tools.

- Solid forged tools
- Boring bars with bits
- Brazed tools (Fig.4)

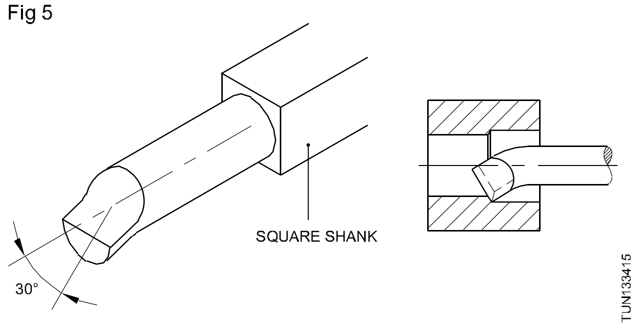
Fig 4



- Throw-away bits inserted in special holders.

Solid forged tools (Fig 5)

Fig 5

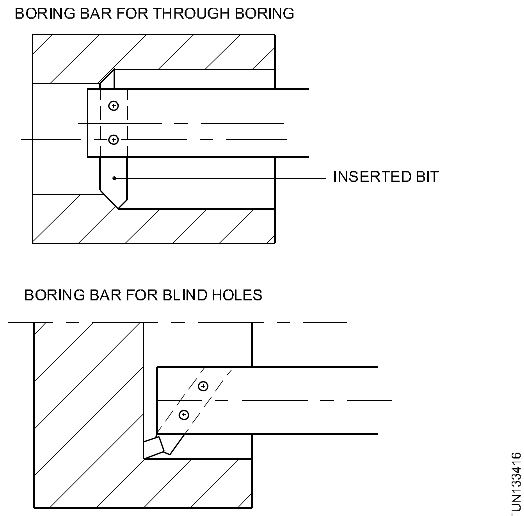


The solid forged boring tool is generally made of high speed steel, with the end forged and ground to resemble a left hand turning tool. They are light duty tools and are used on small diameter holes. They are held in special tool holders which are mounted in the tool post.

Occasionally tungsten carbide or high speed steel tips are brazed to low carbon bars, for economy.

Boring bars with inserted bit (Fig 6)

Fig 6



The boring bar tool-holder is mounted in the tool post and is used for heavier cuts than those for the forged boring tool.

The square tool bits are set at angles of 30°, 45° or 90° in the broached holes in the bar.

The boring bars may be plain type or end cap type. The cutting tool of the plain type is held in position by a set screw. The cutting tool of the end-cap type is held in position by the wedging action of a hardened plug.

The round or square section tool bits may be inserted in boring bars, the size depending on the diameter of the bar.

The tool bit may be square to the axis of the bar for plain boring or at an angle for facing shoulder, or threading up to a shoulder.

The bar is held in a split or 'V' block holder.

The advantages of different boring tools

Solid boring tools

Available with square and round shank.

Enables to mount on the tool post easily.

Re-grinding is easy.

As the tool is integral, alignment is easy.

Can be easily forged to the required shape and angle.

Boring bars and inserted bits

Used for heavy duty boring operation.

Used for deep boring operation.

Tool changing is faster, thereby re-sharpening time is avoided.

Cost is less because the boring bar is made out of low carbon steel.

Boring tools can be set square to the axis of the boring bar or at an angle very quickly.

Counter sinking

Objectives: At the end of this lesson you shall be able to
- **state what is countersinking**
- **list the purpose of countersinking**
- **state the angles of countersinking for the different applications**
- **name the different types of countersinks**
- **distinguish between Type A and Type B countersink 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)

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.

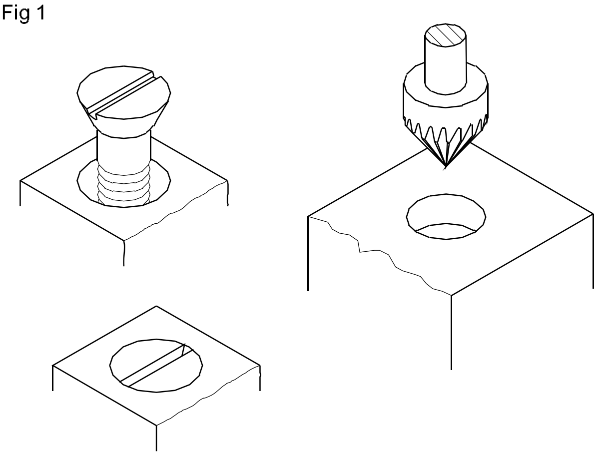
75° countersink riveting

80° countersink self tapping screws

90° countersink head screws and deburring

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

Fig 1

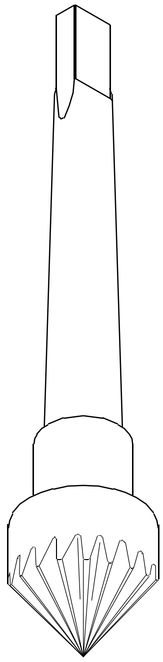


Countersinks

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 counter-sinks 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.

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 interchange able and solid pilots.

Countersink hole sizes

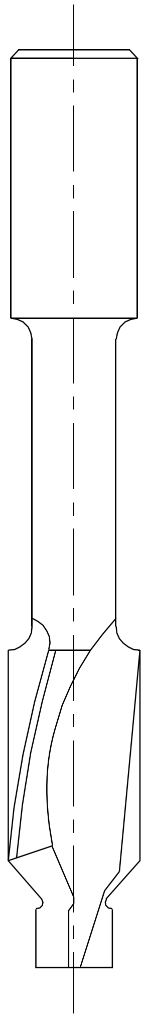
The countersink holes according to Indian Standard IS 3406 (Part 1) 1986 are of four types: Type A, Type B, Type C and Type E.

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 3



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.

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.

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.

Methods of Representing countersink holes in drawings

Countersink hole sizes are identified by code designation or using dimension. IS details are given in Table 1 toTable IV

TABLE I

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

Fig 4

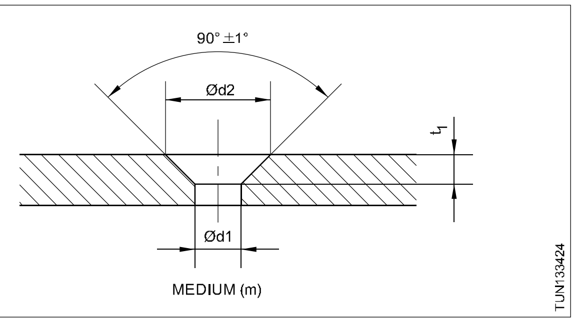
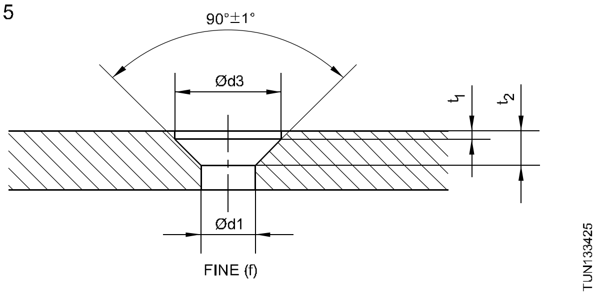


Fig 5



For Norminal 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

TABLE I (Contd)

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.

For Norminal 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	5.5	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	0.2	0.7	0.7	1.2	1.2	1.7

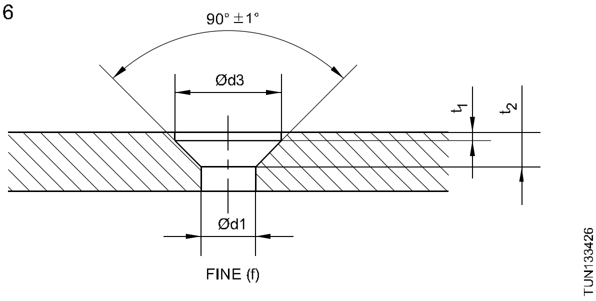
Note 1 : Size shown in brackets are of second preference.

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

TABLE II

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

Fig 6



For Norminal Size		3	4	5	6	8	10	12	(14)	16	(18)	20	22 24
Medium	d_1 H12	4.3	5.3	6.4	8.4	10.5	13	15	17	19	21	23	25
Series	d_2H12	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	8.513.1
14	t_2 + 0.1												
	0	0.2		0.3		0.4			0.5				1

Note 1 : Sizes shown in brackets are of second perference.

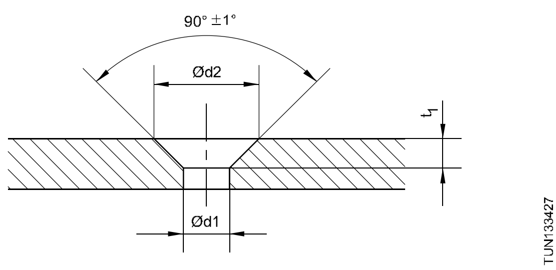
Note 2 : Clearance hole d_1 accodring to medium and fine series of IS : 1821 - 1982.

Designation : A countersink Type B with clearance hole of fine (f) series and having nominal size 10 shall be designated as - Countersink B 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)
d_1 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
d_1 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

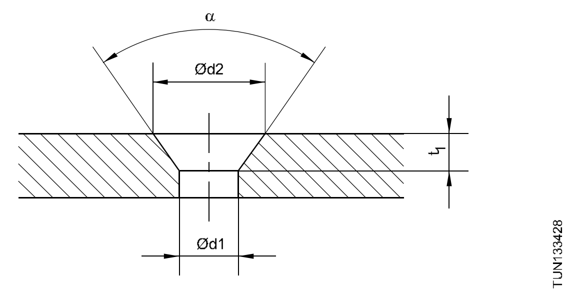
Note : Size given in brackets are of second perference.

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

TABLE IV

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

Fig 8



For Screw Size 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
	75°			60°		
Note : Clearance hold d_1 according to fine series of IS : 1821 - 1982						

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

Use of code designation

Use of dimension

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

Fig 9

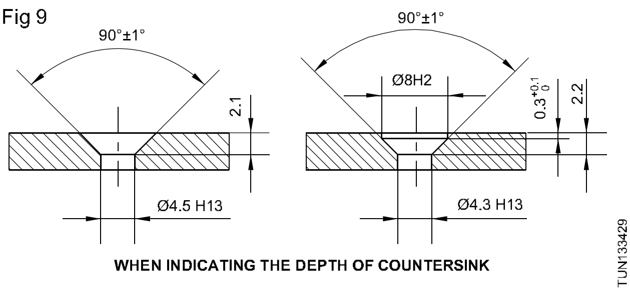
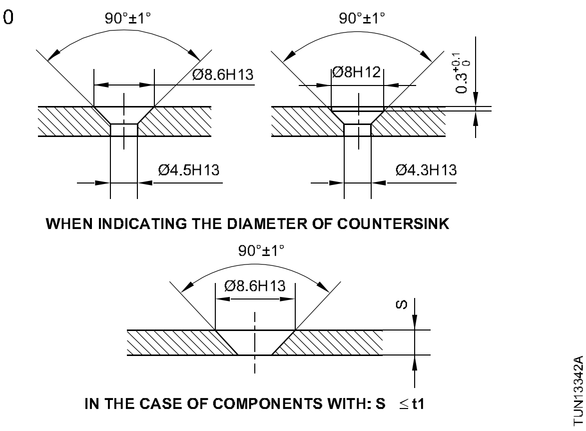


Fig 10



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 even depth to house heads of socket heads or cap screws with the help of a counterbore tool. (Fig 1)

Counterbore (Tool)

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

Fig 1

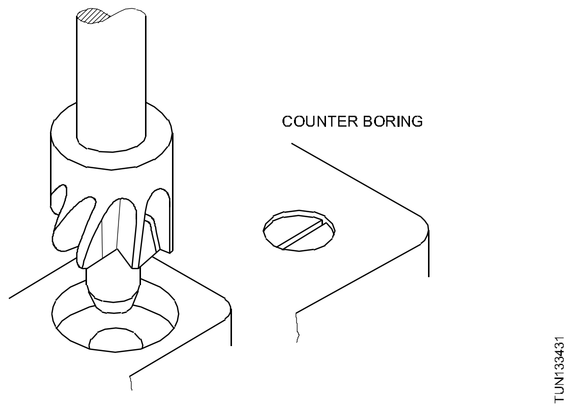
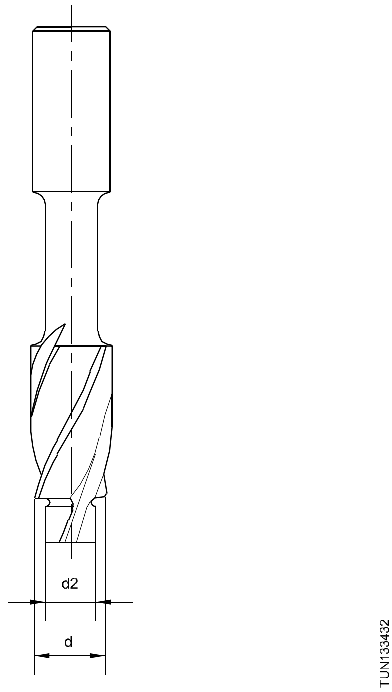
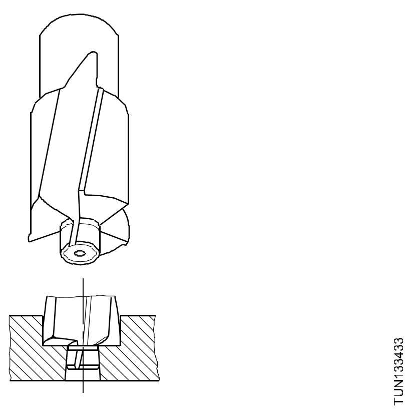


Fig 2



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)

Fig 3

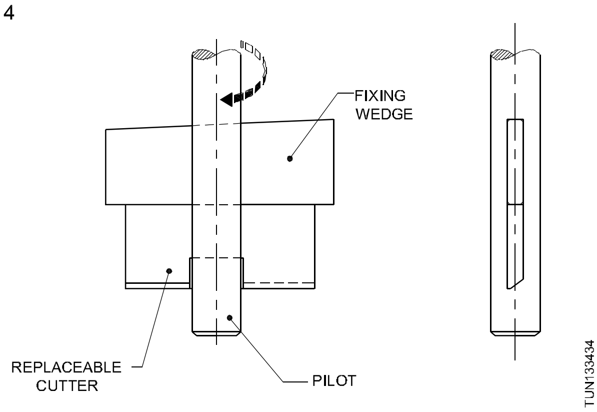


Counterbores are available with solid pilots or with interchangeable pilots. The interchangeable pilot provides flexibility of counterboring on different diameters of holes.

Spot facing

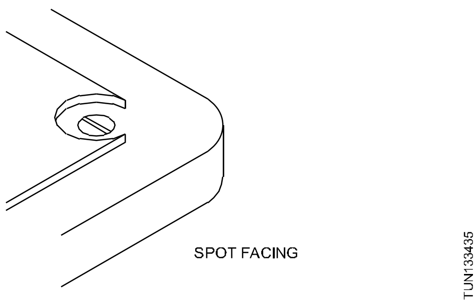
Spot facing is a machining operation for producing a flat seat for bolt head, washer or nut at the opening of a drilled hole. The tool is called a spot facer or a spot facing tool. Spot facing is similar to counterboring, except that it is can be used for spot facing as well. (Fig 4)

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 cap-screws.

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 I S 3406 (Part 2) 1986. This gives dimensions for Type H and Type K counterbores.

Counterbore and Clearance Hole Sizes for Different Sizes of Screws.

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**
- **state the core drill.**

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)

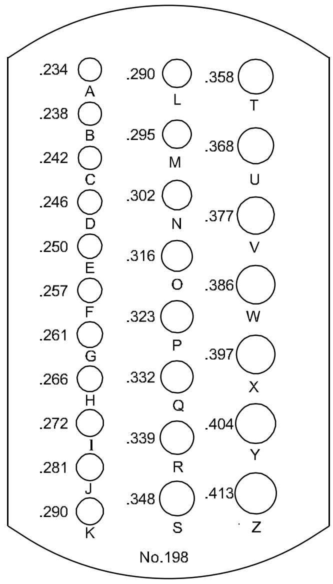
TABLE 1

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
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 Handbook. 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

Number drill sizes (contd)

No.	Diameter	
	Inches	mm
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

Number drill sizes (contd)

No	Diameter	
	Inches	mm
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
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
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

Core drill

A core drill is a specifically designed to remove a cylinder of material, much like a hole saw. The material left inside the drill bit is referred to as the core. The earliest core drills were those used by the ancient egyptians, invented in 3000 BC. core drills are used for many applications, either where the core needs to be preserved, or where drilling can be done more rapidly since much less material needs to be removed than with a standard bit. This is the reason that diamond-tipped core drills are commonly used in construction to create holes for pipes, manholes and other large-diameter penetrations in concrete or stone. Core drills are used frequently in mineral exploration where the coring may be several hundred to several thousand feet in length. The core samples are recovered and examined by geologists for mineral percentage and stratigraphic contact points. This gives exploration companies the information necessary to begin or abandon mining operations in a particular area. Before the start of world war two, Branner Newsom, a California mining engineer, invented a core drill that could take out large diameter cores up to 16 feet in length for mining shafts. This type of core drill is longer in use as modern drill technology allows standard drilling to accomplish the same at a much cheaper cost.

Fig 2

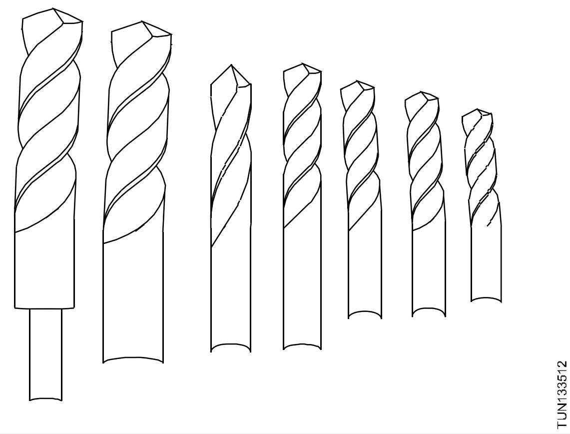


Fig 3