Drilling & milling machines

Objectives: At the end of this lesson you shall be able to
- **name the types of drilling machines**
- **identify the parts of bench and pillar type drilling machine.**

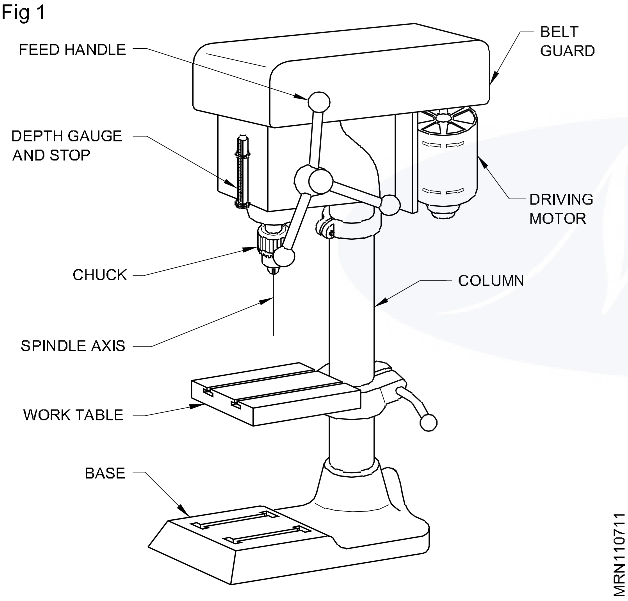
The principal types of drilling machines are the sensitive bench drilling machine the pillar drilling machine the column drilling machine and the radial arm drilling machine. (Radial Drilling Machine).

(You are not likely to use the column and radial type of drilling machines now. Therefore, only the sensitive and pillar type machines are explained here.)

The Sensitive Bench Drilling Machine

The simplest type of sensitive drilling machine is shown in the figure with its various parts marked. This is used for light duty work. (Fig 1)

Fig 1



This machine is capable of drilling holes up to 12.5 mm diameter. The drills are fitted in the chuck or directly in the tapered hole of the machine spindle.

For normal drilling, the work- surface is kept horizontal. If the holes are to be drilled at an angle, the table can be tilted.

Different spindle speeds are achieved by changing the belt position in the stepped pulley. (Fig 2)

The pillar drilling machine

This is an enlarged version of the sensitive bench drilling machine. These drilling machines are mounted on the floor and driven by more powerful electric motors. They are used for heavy duty work. Pillar drilling machines are available in different sizes .(Fig 3)

Fig 2

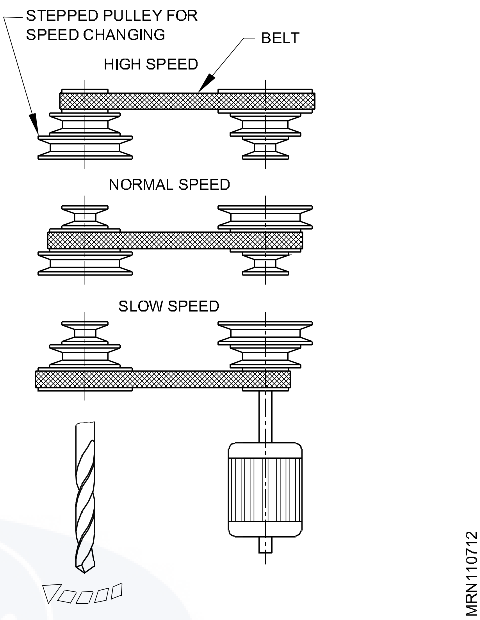
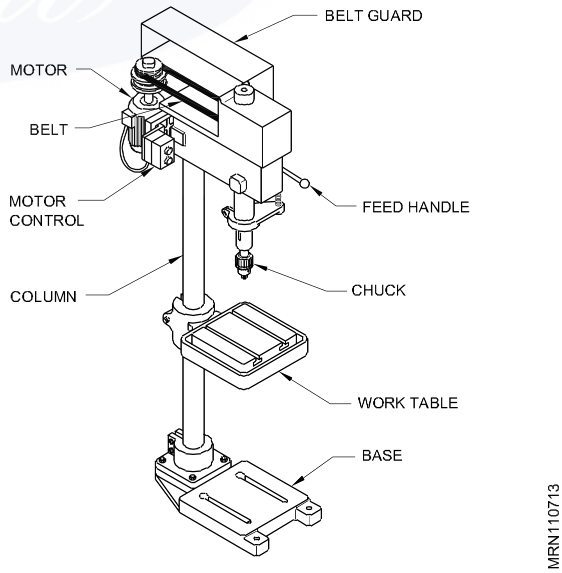


Fig 3



Large machines are provided with a rack and pinion mechanism for moving the table for setting the work.

Drill-holding devices

Objectives: At the end of this lesson you shall be able to
- **name the 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 and sleeves and sockets.

Drill chuck

Straight shank drills are held in drill chucks. 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 fitted on the drill chuck. (Fig 1)

Taper sleeves and sockets (Fig 1)

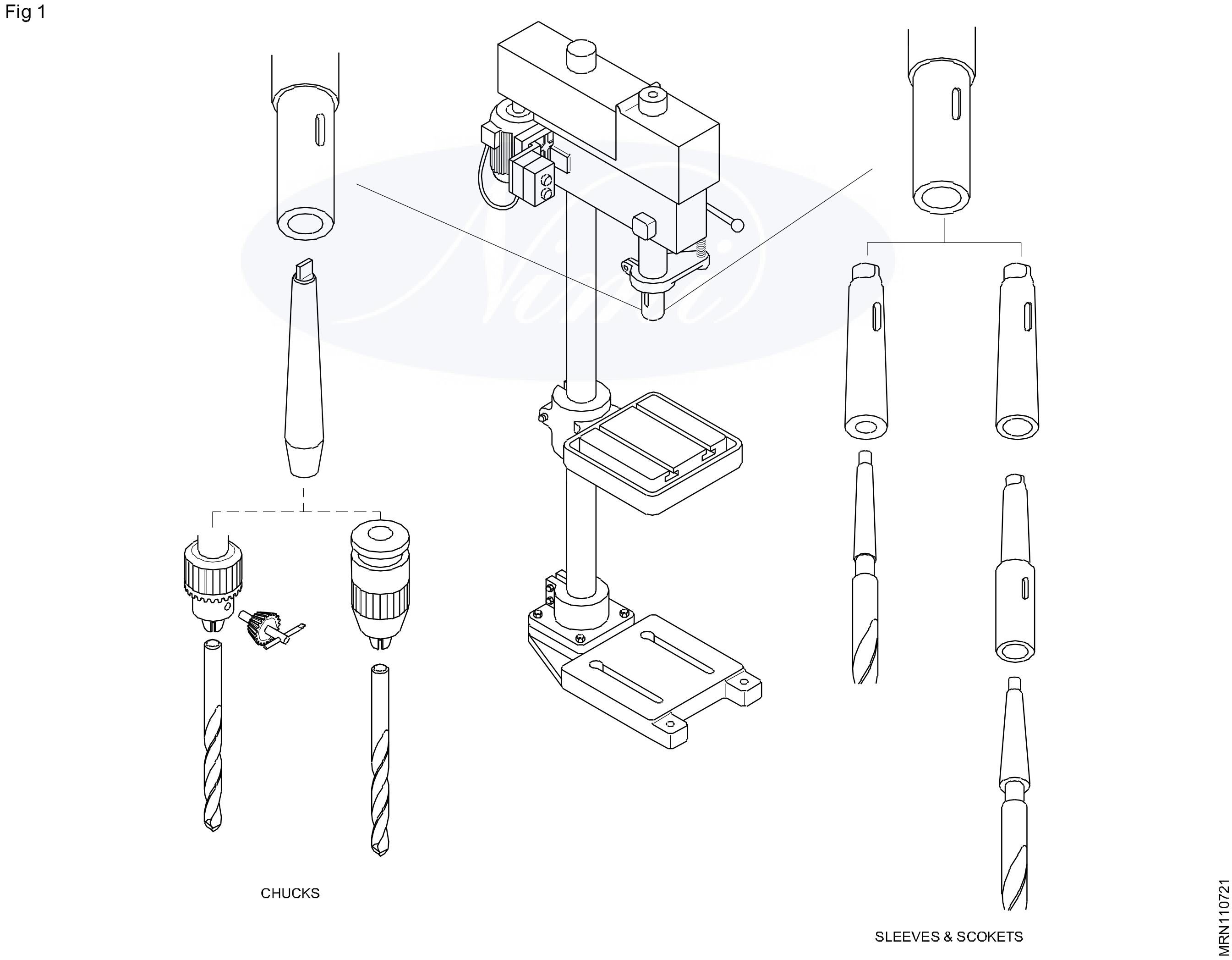
Taper shank drills have a morse taper. Sleeves and sockets are made with the same taper so that the taper shank of the drill, when engaged, will give a good wedging action. Due to this reason morse tapers are called self-holding tapers.

Drills are provided with five different sizes of morse tapers, and are numbered from MT1 to MT5.

In order to make up the difference in sizes between the shanks of the drills and the type 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 1)

Fig 1



While fixing the drill in a socket or sleeve, the tang portion should align in the slot. (Fig 2) This will facilitate the removal of 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. (Fig 4)

Fig 2

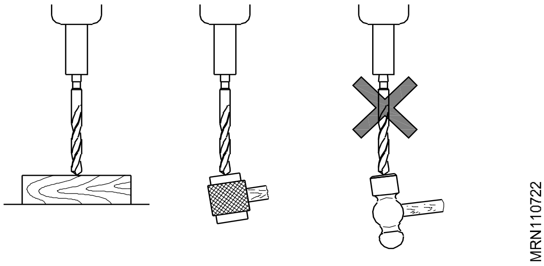


Fig 3

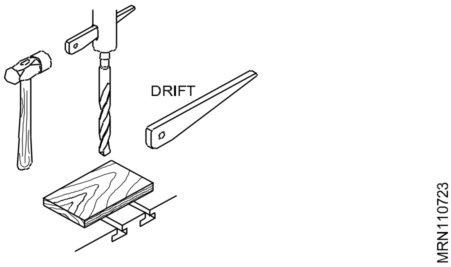
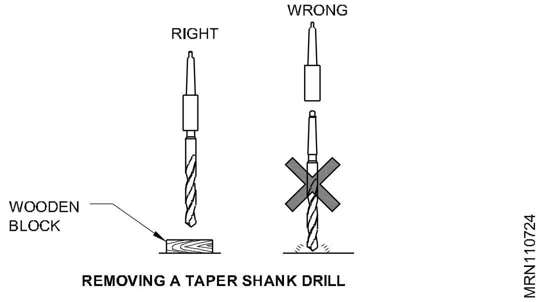


Fig 4



Work-holding devices

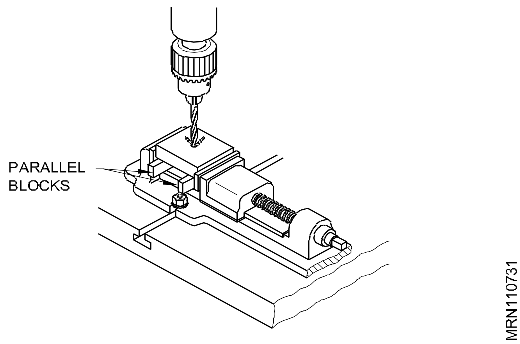
Objectives: At the end of this lesson you shall be able to
- **state the purpose of work-holding devices**
- **name the devices used for holding work**
- **state the precautions to be observed while using work-holding devices.**

Work pieces to be drilled should be properly held or clamped to prevent from rotating along with the drill. Improperly secured work is not only a danger to the operator but can also cause inaccurate work, and breakage to the drill. Various devices are used to ensure proper holding.

The machine vice

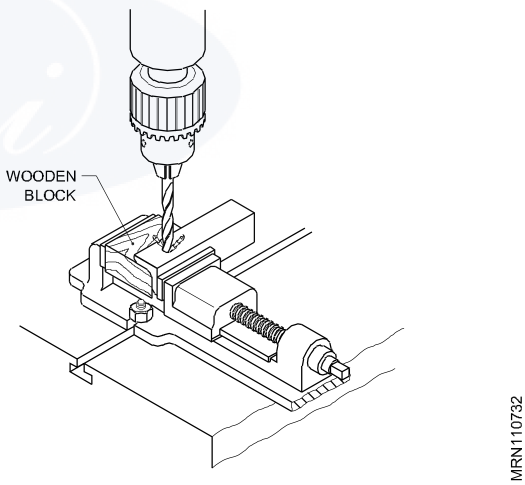
Most of the drilling work can be held in a machine vice. Ensure that the drill does not drill through the vice after it has passed through the work. For this purpose, the work can be lifted up and secured on parallel blocks providing a gap between the work and the bottom of the vice.(Fig 1)

Fig 1



Work pieces which are not accurate may be supported by wooden pieces. (Fig 2)

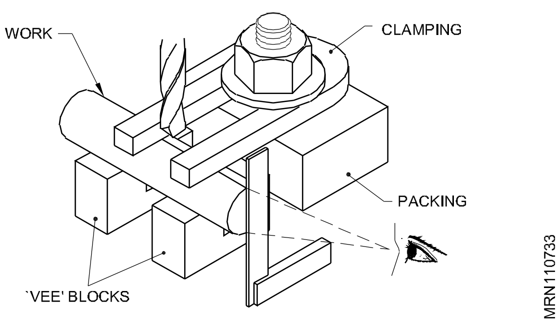
Fig 2



Clamps and bolts

Drilling machine tables are provided with T-slots for fitting bolt heads. Using clamps and bolts, the workpieces can be held very rigidly. (Fig 3) While using this method, the packing should be, as far as possible, of the same height as the work, and the bolt nearer to the work. (Fig 4)

Fig 3



There are many types of clamps and it is necessary to determine the clamping method according to the work. (Figs 5 & 6).

Fig 4

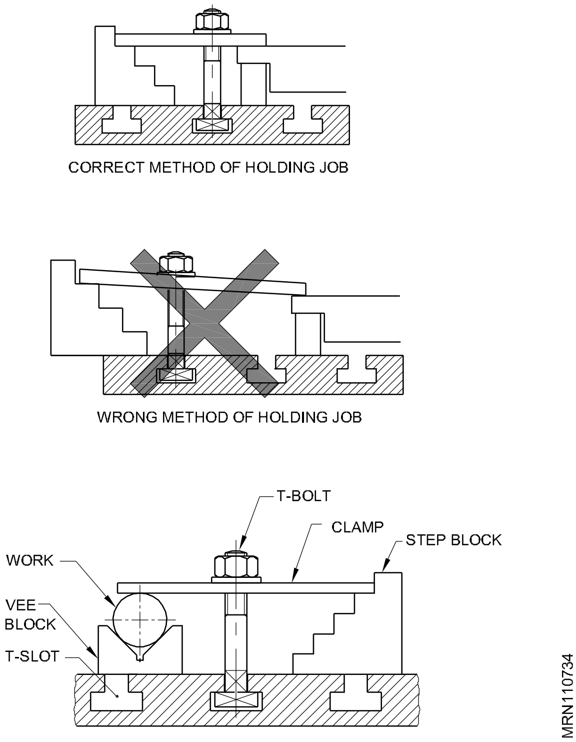


Fig 5

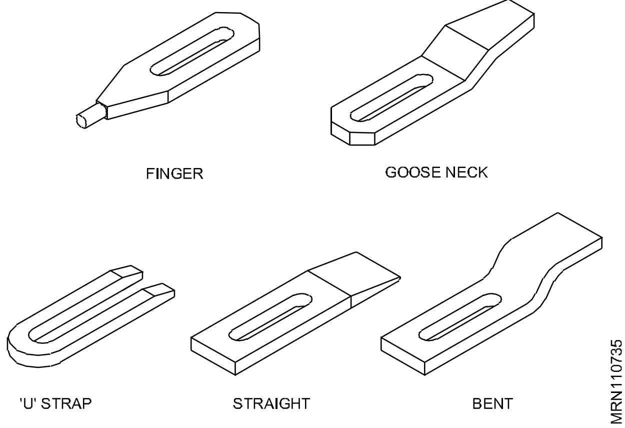
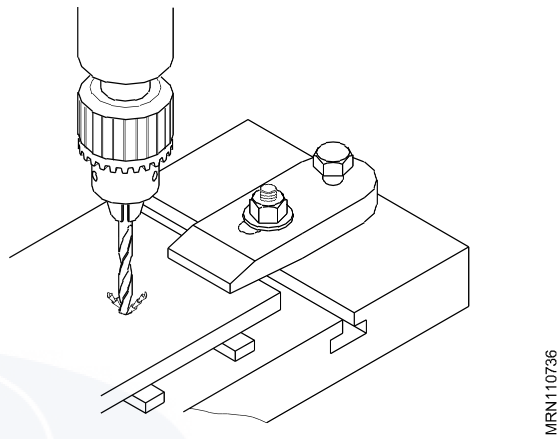


Fig 6



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 RPM**
- **determine RPM/spindle speed**
- **select RPM for drill sizes from tables.**

For a drill to give a 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 RPM, at which a drill has to be driven, is determined.

Calculating RPM

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

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

$$n = \frac{v \times 1000}{d \times \pi}$$

n - RPM

v - cutting speed in m/min.

d - diameter of the drill in mm

π = 3.14

b= <u>Examples:</u> Calculate the RPM for a high speed steel drill Ø24 to cut mild steel.

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

$$n = \frac{1000 \times 30}{3.14 \times 24} = 398 \text{ RPM}$$

It is always preferable to set the spindle speed to the nearest available lower range. The selected spindle speed is300 RPM.

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

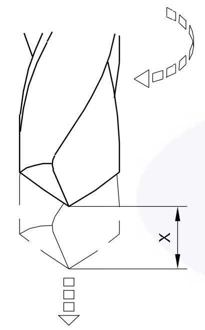
The recommended cutting speeds are achieved only by actual experiments.

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 millimeter.

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 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 for the feed rate given here 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.

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

- **Ensure that the spindle-head and the arms are locked properly to avoid vibration. (Fig 2)**
- **The work piece and the drill should be rigidly held.**
- **Bring back the spindle head nearer to the pillar after use.**
- **Switch off power when not in use.**
- **Use the drill drift for removing the drills, chucks or sockets.**
- **Use a minimum number of sockets and sleeves to make for the spindle bore size.**
- **Clean and oil the machine after use.**
- **Stop the machine to remove the swarf.**
- **Use a brush to clean the chips and swarf.**

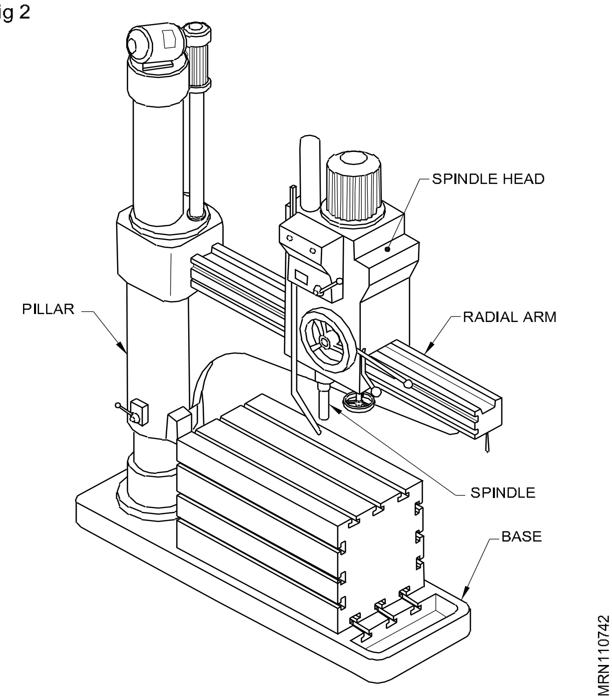


Fig 2

Drill angles

Objectives: At the end of this lesson you shall be able to
- **identify the various angles of a twist drill**
- **state the functions of each angle**
- **list the tool types for drill 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.

Angles

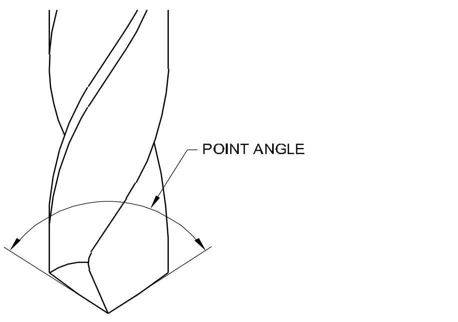
They are different angles for different purposes. They are listed below.

Point Angle, Helix angle, Rake angle, Clearance angle and Chisel edge angle.

Point angle/cutting angle

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

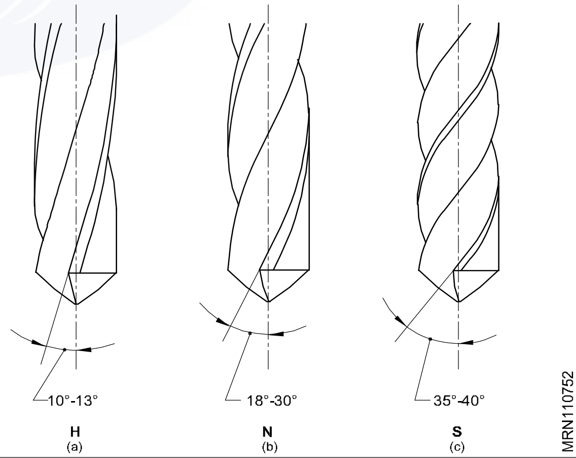
Fig 1



Helix angle (Figs 2a, 2b, 2c)

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

Fig 2



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

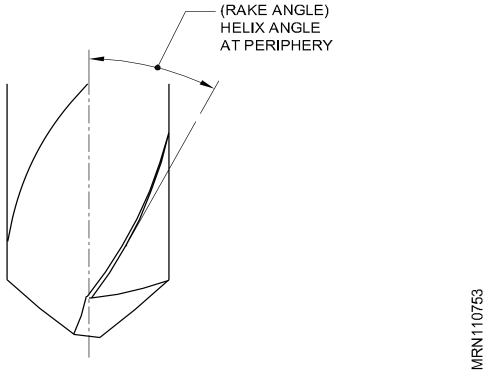
- Type N - for normal low carbon steel.
- Type H - for hard and tenacious materials.
- Type S for soft and tough materials.

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

Rake angle (Fig 3)

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

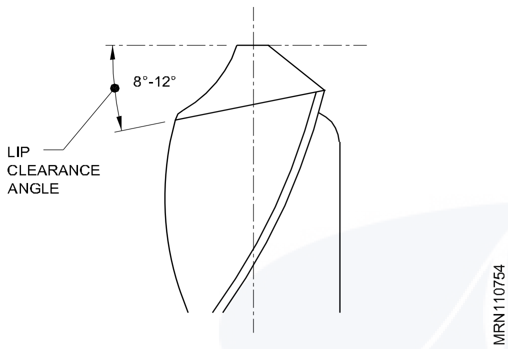
Fig 3



Clearance angle (Fig 4)

The clearance angle is 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.

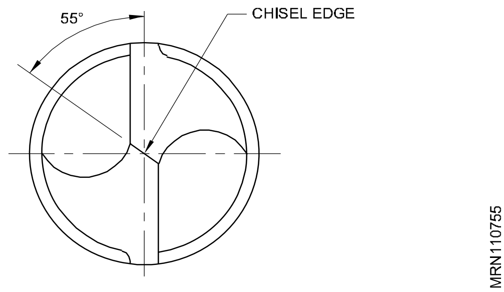
Fig 4



Chisel edge angle/web angle (Fig 5)

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

Fig 5



Designation of drills

Twist drills are designated by the

- diameter
- tool type
- material

Example

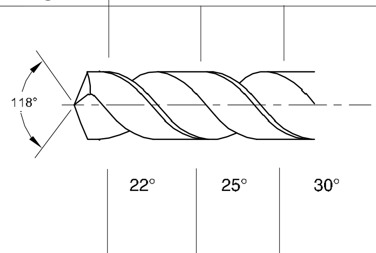
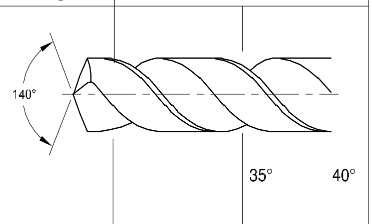
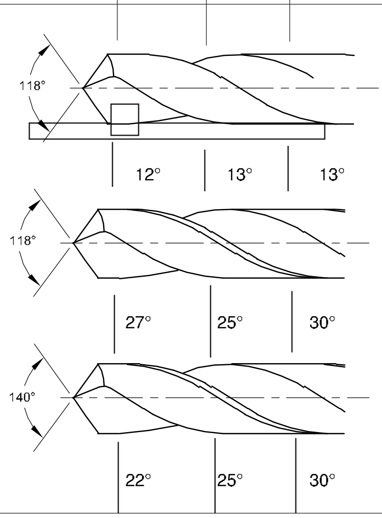
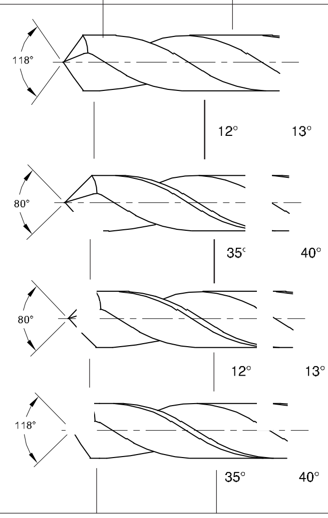
A twist drill of 9.50 mm dia. of tool type `H' for right hand cutting and made from HSS is designated as

Twist drill 9.50 - H - IS5101 - HS

Diameter of drill
IS NO.
Tool type
Material

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/mm2 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 Steel and cast steel 70...120 Kgf/mm2 Stainless steel; Copper (drill diameter more than 30 mm) Al-alloy, forming short broken chips			Austenitic steels Magnesium alloys Moulded plastics (with thickness s>d) Moulded plastics, with thickness s<d Laminated plastics, hard rubber (ebonite) marble, slate, coal Zinc alloys		

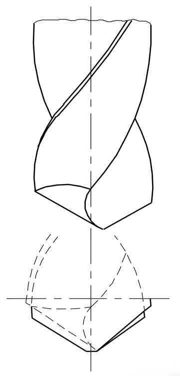
Drilling defects and causes

Objectives: At the end of this lesson you shall be able to
- **state the common drilling defects**
- **identify the causes of drilling defects**
- **suggest remedial steps for preventing drill failures.**

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

Fig 1

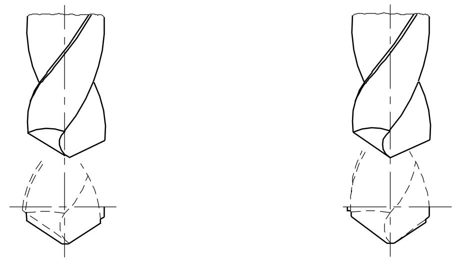


UNEQUAL LIP LENGTH

Oversized holes

- Oversized holes can be due to the
- unequal length of cutting edges
- unequal angle of cutting edges
- unequal point thinning
- spindle running out of centre
- drill point not in centre.

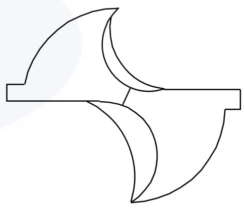
Fig 2



Overheated drills

- The drills may get overheated if the
- cutting speed is too high
- feed rate is too high
- clearance angle is incorrect
- cooling ineffective point angle is incorrect
- drill is not sharp.

Fig 3



UNEQUAL POINT THINNING

Tap drill size

Objectives: At the end of this lesson you shall be able to
- **state what is tap drill size**
- **choose the tap drill sizes for different threads from tables**
- **calculate the tap drill sizes for ISO metric and ISO inch.**

What is a tap drill size?

Before a tap is used for cutting internal threads, a hole is to be drilled. The diameter of the hole should be such that it should have sufficient material in the hole for the tap to cut the thread.

Tap drill sizes for different threads

ISO Metric Thread

Tapping drill size

for M10 x 1.5 thread

Minor diameter = Major diameter – 2 x depth

depth of thread = 0.6134 x pitch of a screw

2 depth of thread = 0.6134 x 2 x pitch

=1.226 x 1.5 mm = 1.839 mm

Minor dia (D1)=10 mm – 1.839 mm

=8.161mm or 8.2 mm

This tap drill will produce 100% thread because this is equal to the minor diameter of the thread. For most fastening purposes a 100% formed thread is not required.

A standard nut with 60% thread is strong enough to be tightened until the bolt breaks without stripping the thread. Further it also requires a greater force for turning the tap if a higher percentage formation of thread is required.

Considering this aspect, a more practical approach for determining the tap drill sizes is

Tap drill size = Major diameter – pitch

= 10 mm - 1.5 mm

= 8.5 mm.

Compare this with the table of tap drill sizes for ISO metric threads.

ISO Inch (Unified) threads Formula

Tap Drill size = 1 Major diameter – Number of threads per inch

For calculating the tap drill size for 5/8" UNC thread

Tap drill size = 5/8" – 1/11"

= 0.625" – 0.091"

= 0.534"

The next drill size is 17/32" (0.531 inches)

Compare this with the table of drill sizes for unified inch threads.

What will be the tapping size for the following threads?

(a) M 20

(b) UNC 3/8