

Power Cutting Tools

6.1 Drills and Drilling

Introduction

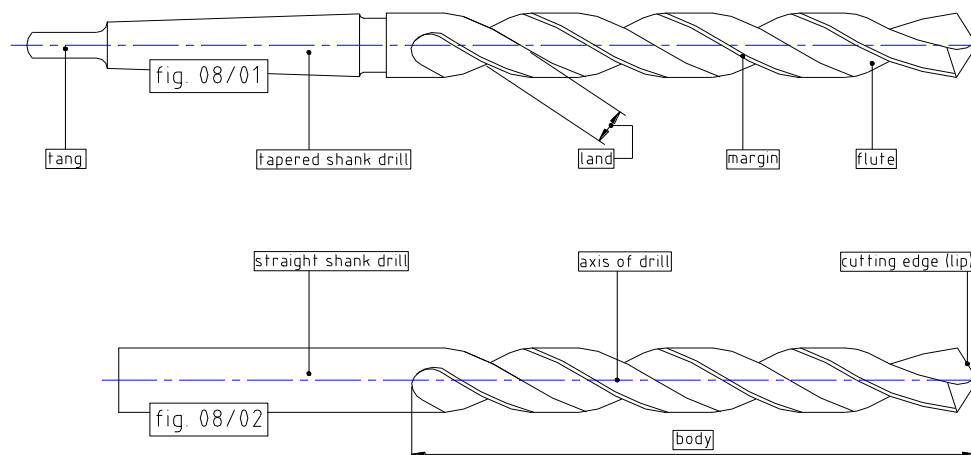
Twist drills are end-cutting tools used to produce holes in most types of material. On standard drills, two helical grooves, or flutes, are cut lengthwise around the body of the drill. They provide cutting edges and space for the cuttings to escape in the drilling process. Since drills are one of the most efficient tools, it is necessary to know the main parts, how to sharpen the cutting edges, and the correct speeds and feeds for drilling various metals in order to use them most efficiently and prolong their life.

6.1.1 Shank

Most twist drills used in machine shop work today are made of high-speed steel. High-speed drills have replaced carbon-steel drills since they can be operated at double the cutting speed and the cutting edge lasts longer. A drill may be divided into three main parts: the shank, the body and the point.

6.1.1.1 Generally drills up to 13mm in diameter have straight drill shanks, while those over this diameter usually have tapered shanks. *Straight-shank drills* are held in a drill chuck; *tapered-shank drills* fit into the internal taper of the drill press spindle.

6.1.1.2 A *tang* (**fig. 08/01**) is provided on the end of tapered-shank drills to prevent the

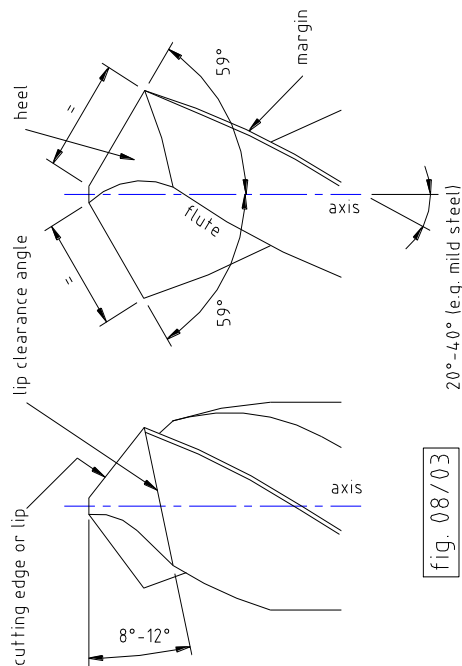
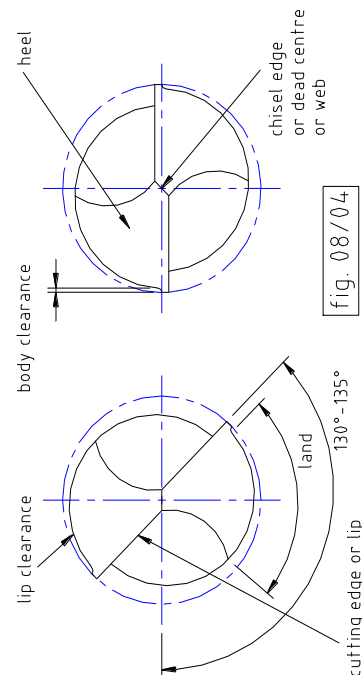


drill from slipping while it is cutting and to allow the drill to be removed from the spindle or socket without the shank being damaged by using a drill drift.

6.1.2 Body

The *body* is the portion of the drill between the shank and the point. It consists of a number of parts important to the efficiency of the cutting action.

6.1.2.1 The *flutes* are two or more helical grooves cut around the body of the drill. They form the cutting edges, admit cutting fluid, and allow the chips to escape from the



hole.

6.1.2.2 The *margin* is the narrow, raised section on the body of the drill. It is immediately next to the flutes and extends along the entire length of the flutes. Its purpose is to provide a full size to the drill body and cutting edges.

6.1.2.3 The *lip clearance* (**fig. 08/04**) is the undercut portion of the body between the margins and the flutes. It is made smaller to reduce friction between the drill and the hole during the drilling operation.

6.1.2.4 The *web* (**fig. 08/04**) is the thin partition in the center of the drill which extends the full length of the flutes. This part forms the chisel edge (8.4.1.) at the cutting end of the drill. The web gradually increases in thickness toward the shank to give the drill strength.

6.1.3 Point

6.1.3.1 The *point* (**fig. 08/04**) of a twist drill consists of the chisel edge, the lips, the lip clearance angle and the heel.

The *chisel edge* (web) (**fig. 08/04**) is the chisel-shaped portion of the drill point.

6.1.3.2 The *lip* (cutting edge) (**fig. 08/02 and fig. 08/03**) is formed by the intersection of the flutes. The lips must be equal length and have the same angle so that the drill will run true and will not cut a hole larger than the size of the drill.

6.1.3.3 The *lip clearance angle* (**fig. 08/03**) is the relief which is ground on the point of the drill extending from the cutting lips back to the *heel* (**fig. 08/03**). The average lip clearance is from 8° to 12°, depending upon the hardness or softness of the material to be drilled.

6.1.4 SPEED

A wide range of drills and drill sizes is used to cut various metals; an equally wide range of speeds is required for the drill to cut efficiently. For every job, there is the problem of choosing the drill speed which will result in the best production rates and the least amount of downtime for regrinding the drill. The recommended cutting speeds for drilling various types of materials may be found in the table shown below.

It is essential to select the correct cutting speed and the feed. Followings are the most common used cutting speed and feed rate.

Cutting Speed	
Material	Cutting Speed
Mild steel	6 - 9 m/min
Stainless Steel	4 - 9 m/min
Aluminium	30 - 36 m/min

Feed Rate	
5.5 mm diameter twist drill	0.08 - 0.15 mm/rev
30 mm diameter twist drill	0.04 - 0.55 mm/rev

The most economical drilling speed depends upon many variables such as:

- The type and hardness of the material
- The diameter and material of the drill
- The type and condition of the drill press
- The efficiency of the cutting fluid employed

To determine the correct number of r/min (revolution per minute) of a drill press spindle for a given size drill, the following should be known:

- The type of material to be drilled
- The recommended cutting speed of the material
- The type of material from which the drill is made

Stainless steel	Tool steel	Cast Iron	Machine steel	Aluminium
CS = 12	CS = 18	CS = 24	CS = 30	CS = 60

Formula: $r/min = \frac{\text{CS in m/min}}{\pi \times D \text{ in mm}} \Rightarrow \frac{\text{CS} \times 1000}{3.14 \times D} \Rightarrow \frac{\text{CS} \times 320}{D}$

6.1.5 Feed

Feed is the distance that a drill advances into the work for each revolution. Drill feeds may be expressed in decimals, fractions of millimeters. Since the feed rate is a determining factor in the rate of production and the life of the drill, it should be carefully chosen for each job. The rate of feed is generally governed by:

6.1.5.1 the diameter of the drill

6.1.5.2 the material of the work piece

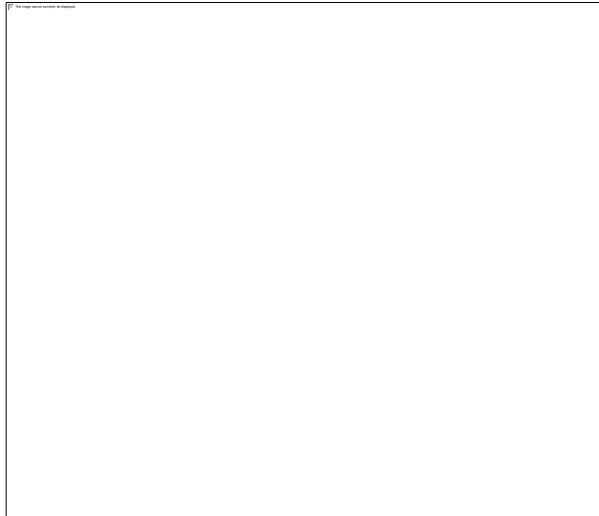
6.1.5.3 the condition of the machine

<u>Drill size</u>	<u>Feed per revolution</u>
1 to 3	0.02 to 0.05
3 to 6	0.05 to 0.10
6 to 13	0.10 to 0.18
13 to 25	0.18 to 0.38

Drill Features

The point of the drill is ground to an angle of 59° to the centre line to give two equal cutting edges, and each side is ground back to give "relief" of about 12° to each cutting edge as shown in figure 29.

It is very important that drill points are central and that the lip angles are equal and that the cutting edges are unchipped and the clearance angle correct. To obtain this state and ensure correct angles it is important that drills are ground in a grinding machine.



2. Hand thread cutting

4.1 Safety

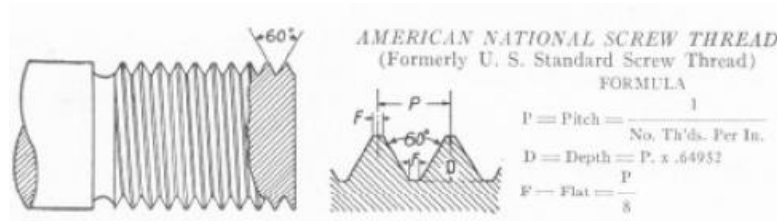
1. Do not use either the stock or the die as a hammer; the threads may be broken.
2. Remove the die from the stock after every thread cutting, clean them and pack them into their boxes.
3. Use plenty of oil during cutting to reduce friction.

4.2 Definition of threads and thread types

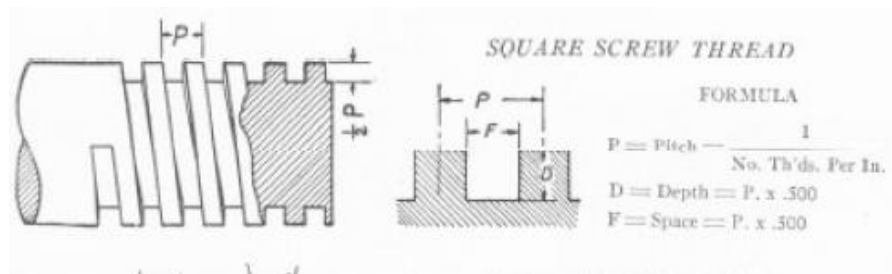
Definition of a thread: A thread is a spiral or helical groove cut on the inside or outside of a cylinder or cone. The thread on a rod is called an external thread, the thread in side of a hole or nut is called an internal thread. Threads are used to hold parts together, transmit power and provide accurate measurements.

Thread types

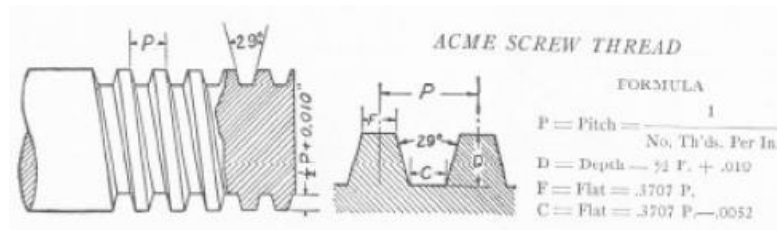
V-thread: Is used in metric bolts, nuts and machine building parts. It has an included angle of 60° .



Square thread: Is formed like square; the depth and width of the groove are equal. It is used in vises and screw jacks.



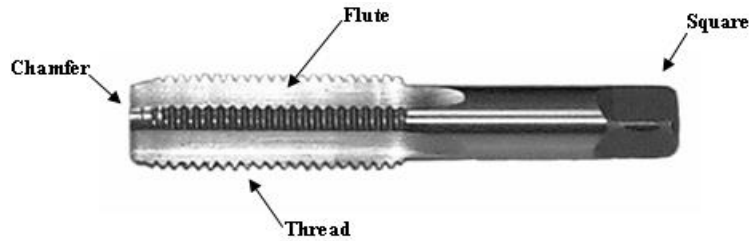
Acme thread: Has an included angle of 29° and is stronger than square thread.



Types of Thread

Types of Taps and dies

Taps are used to cut threads on the inside of a hole, as in a nut. One end of the tap is square so it can be turned with a tap wrench. As the tap turns, it bites into the metal, removing material which flows into the flutes. To form the thread, the hole must be smaller than the outside diameter of the bolt or screw that fits it. In order to cut, the point of the tap is chamfered, or tapered, with the cutting edge higher than the back edge.



There are three basic styles of taps:

Taper Taps - This style has an **8-10 thread chamfer**, it has the longest chamfer of the three to distribute action over the maximum number of teeth; and the taper also acts as a guide in starting the cutting action in the hole. Taper style taps start the thread square with the work-piece. Taper taps are commonly used in through holes and in materials where a tapered guide is necessary.



Figure 4.2 Taper tap

Plug Taps - This style has a **3-5 thread chamfer**, is most widely used in through holes and where there is sufficient room at the bottom in blind holes.



Figure 4.3 Plug tap

Bottoming Taps - This style has **1-2 thread chamfer**, is made with just enough chamfer for starting in the hole; as the name implies, it is designed to thread blind holes to the bottom.



Figure 4.4 Bottoming tap

4.3 Tap holder and die stock

Tap holder or tap wrench is used to hold taps during internal thread (fig 4.5)

Die stock is used to hold dies during cutting of external threads (fig 4.6)



Figure 4.5 Tap holder



Figure 4.6 Die holder

4.4 Cutting internal and external threads

Internal threads

How are Taps used?

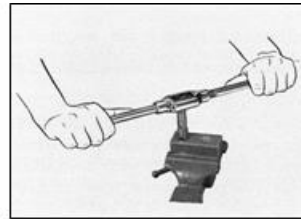
- ✓ First, measure the thread on the screw or bolt you intend to use. You can use a micrometer to measure the outside diameter and a screw pitch gauge to measure the number of threads per inch. If a screw pitch gauge is not available, a scale can also be used by counting the number of threads to an inch.
- ✓ Second, determine the correct size hole to drill before tapping. This is known as the tap-drill size. Tap drill sizes are usually computed for a 75% full thread which is as strong as a 100% full thread and does not put undue strain on the tap. The correct tap drill in each case can easily be found in the “Tap and Drill Chart” in our web page.
- ✓ After the hole has been drilled, clamp the work in a vise with the hole in an upright position.
- ✓ Apply a good grade of cutting oil to the tap.
- ✓ Grasp the tap wrench with your right hand directly over the tap, or with both hands close to the tap, and place the tap in the hole.
- ✓ Press and start to turn the tap clockwise into the hole.
- ✓ Turn the tap forward until resistance is felt and then turn the tap a quarter-turn backwards to break the chip. This will cause the teeth of the tap to take a firm hold

in the side of the hole. **Make sure the tap is square with the surface of the work.** Once the thread is started properly, the tap will draw itself into the work and downward pressure is no longer needed.

- ✓ Grasp the wrench by both handles and with a slow, steady movement, continue turning the tap into the hole, backing up occasionally (when increased resistance is felt) to allow the chips to break.
- ✓ Never force a tap because it can become so tightly wedged that it will break. Using too much pressure and letting the tap get out of line will also break the tap.



Starting Tap



Holding Tap Wrench After Thread is Started

Figure 4.7 Starting Tap and holding Tap

External threads

Dies are just the opposite of taps in that they cut external threads on the outside of pipes, rods or bolts. The die is held in a Die Stock and worked in much the same manner as a tap.

What are Dies?

Dies are made either of high-carbon steel or of high-speed steel. Unlike taps, dies are used for cutting external (male) threads.



Figure 4.8 Dies

Procedures for cutting external threads

1. Square the end of the work and chamfer it (using a file, grinding machine or centre lathe) for an easy start (Figure 4.22).
2. Grip the die, held in the stock, firmly and squarely on the work.
3. Turn clockwise, about a quarter-turn, and ease back to remove chippings.

4. Apply a good supply of oil (lubricant).
5. Make adjustments of the screws after making a full cut until the depth required is achieved.

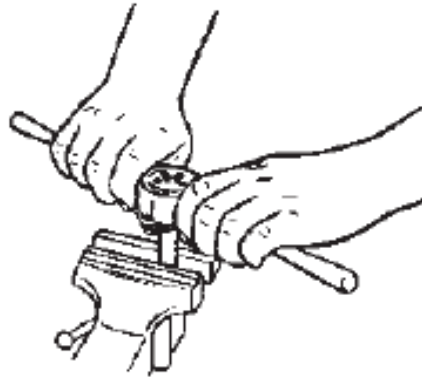


Figure 4.9 Starting Die



Figure 4.10 after die is started