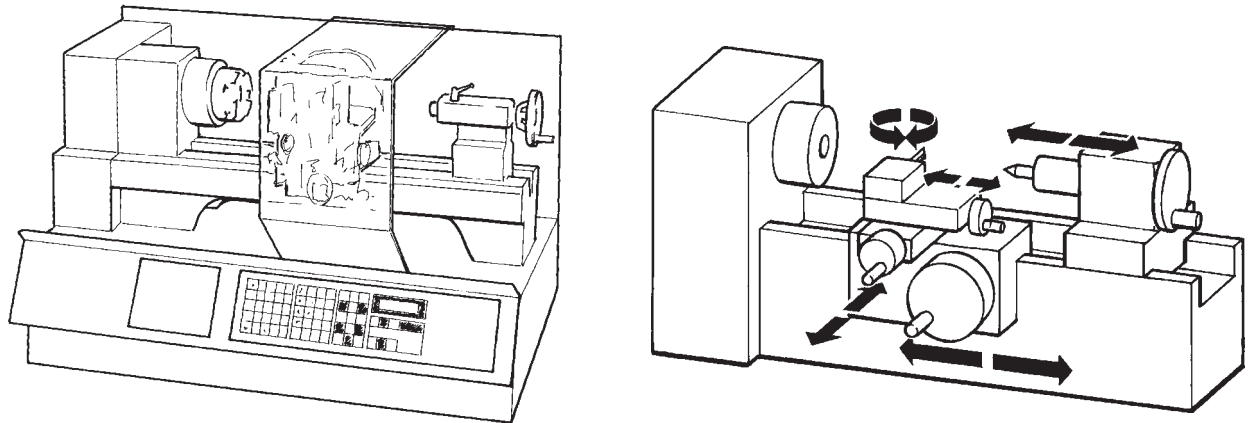


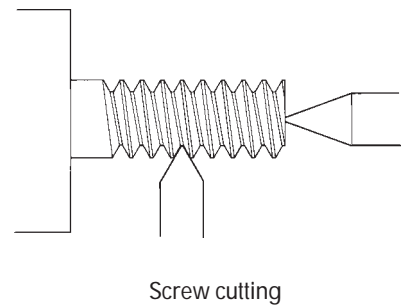
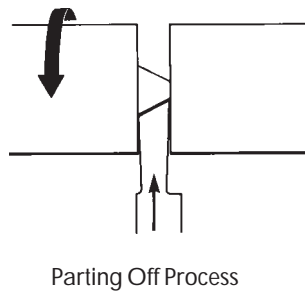
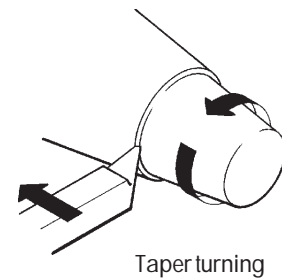
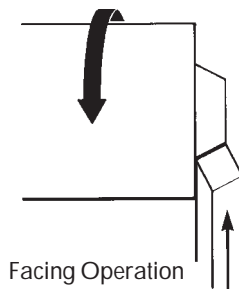
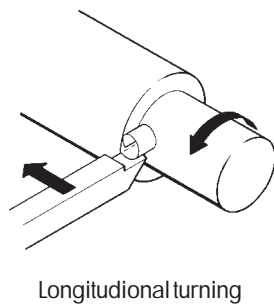
MACHINING

LATHE TURNING



Lathe turning is one of the most common manufacturing processes in engineering and normally uses a single point cutting tool or bit held in a tool post. The tool bit is specially shaped and angled. The tool may be made of high speed steel (HSS) or tool steel with a very hard replaceable tip. There are several basic turning operations:

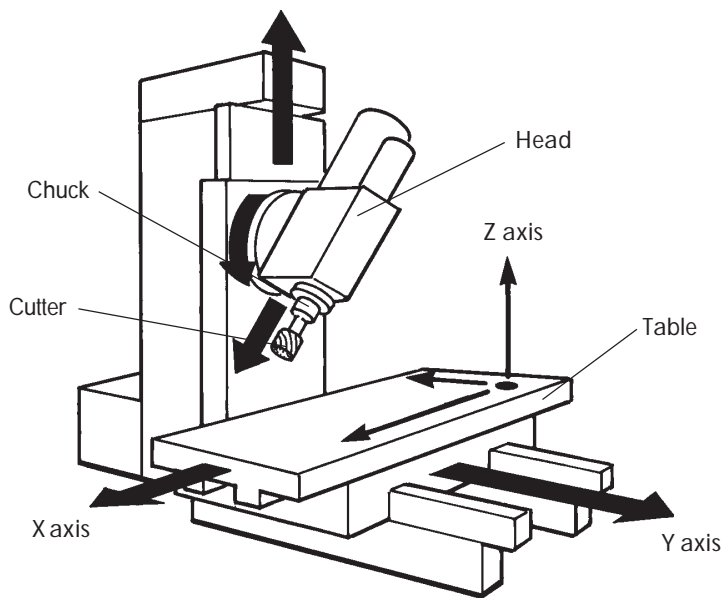
- longitudinal turning
- facing
- parting off
- taper turning
- screw cutting



In each case the rotating workpiece must be rotating at the correct speed. This can be determined from a look-up table. In the case of screw cutting, the speed of the workpiece is synchronized with the speed of the moving tool - which is ground to the profile of the thread being cut.

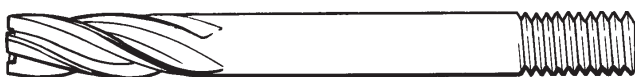
Many lathes used in industry are now computer numerically controlled (CNC). A computer is programmed with machining instructions and the two axes of movement of the tool (and the speed of the workpiece) are then controlled entirely by the computer.

Vertical milling



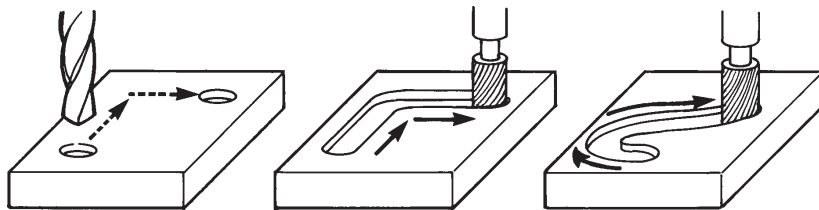
Vertical milling machine

A vertical milling machine uses a multi-point cutter held in a chuck. The head remains stationary while a moving table holding the workpiece moves along X and Y axes. The table can also be moved up and down (Z axis). If necessary, the head of some vertical milling machines can be tilted at an angle.



End mill

There are a wide variety of milling cutters for vertical milling machines but the most common ones are end mills. These can be used for a variety of surfacing operations or for cutting channels or pathways. Because an end mill is not designed to drill down into the workpiece, holes have to be drilled prior to milling out a channel.

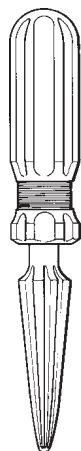
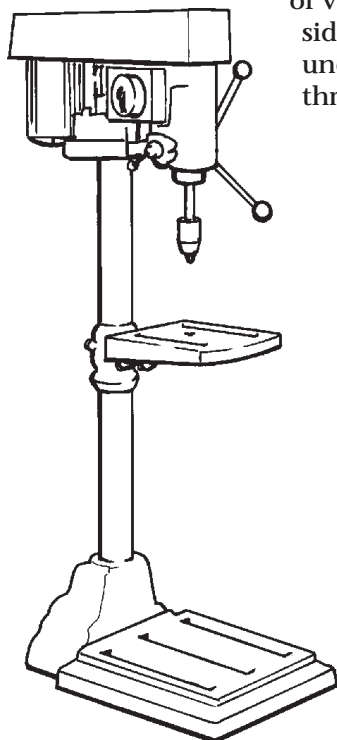


Milling a channel

Many milling machines now used in industry are computer numerically controlled (CNC). A computer is programmed with machining instructions and the three axes of movement of the table (and the speed of the cutter) are then controlled entirely by the computer. It is almost impossible to produce a curved profile using a conventional milling machine, but it is very easy with a CNC machine.

Drilling

This is perhaps the most common of all machining operations. The bulk of the holes we drill are produced to within fairly wide tolerances; when we need to produce a hole of very accurate diameter with smooth sides, the workpiece is drilled slightly undersize and then finished by passing through a reamer.

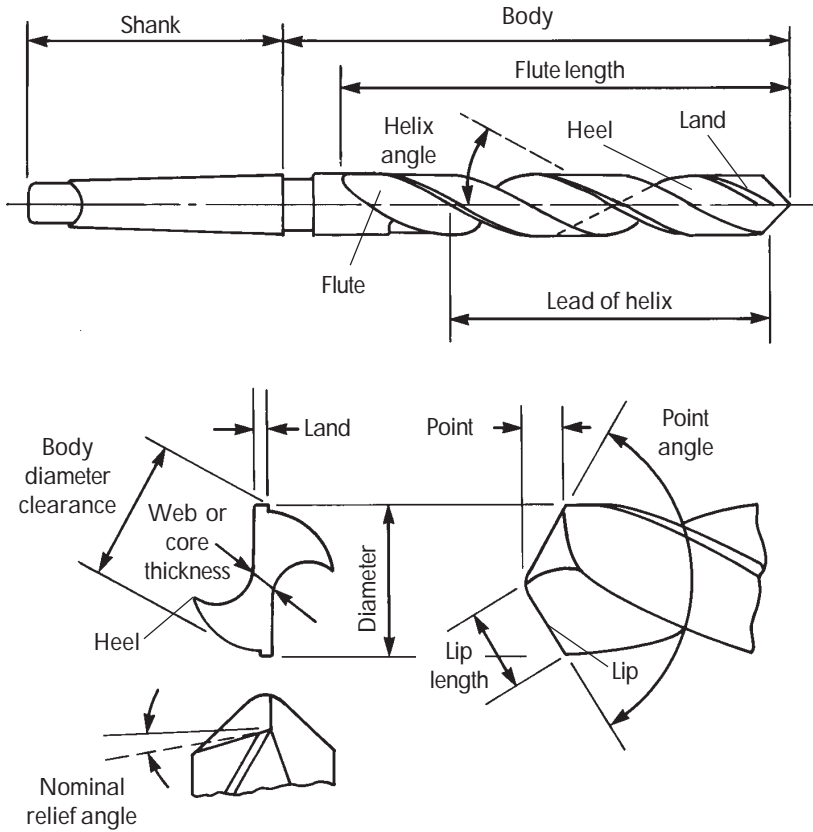


Hand reamer



Reamer

An ordinary drill bit is normally made from high speed steel (HSS) and has a complex shape or geometry. The actual cutting is done at the drill tip and the waste material or swarf passes up through the flutes. The drill illustrated is a larger diameter type with a tapered shank that fits directly into a drilling machine or lathe tailstock in place of the chuck. For drills under about 13 mm in diameter, most drills have a parallel sided shank which fits in a chuck.



The Geometry of a Drill

When a larger drill is used, a small pilot hole should be drilled first so that the core or web of the larger drill can pass through. Look-up tables show the recommended speeds of different diameter drills in relation to the materials being drilled. Generally speaking, the larger the diameter the drill, the slower the drilling speed.