MODULE-3

CUTTING TOOL MATERIALS, GEOMETRY AND SURFACE FINISH

LESSON CONTENTS:

Introduction, desirable Properties and Characteristics of cutting tool materials, cutting tool geometry, cutting fluids and its applications, surface finish, effect of machining parameters on surface finish.

Machining equations for cutting operations: Turning, Shaping, Planing, slab milling, cylindrical grinding and internal grinding, Numerical problems

OBJECTIVES:

- To develop the knowledge about Cutting tool materials, geometry and their characteristics.
- To study about the Cutting fluids, their application and surface finish
- Solving numerical on machining equations for various cutting operations

3.0 Introduction:

The selection of cutting-tool materials for a particular application is among the most important factors in machining operations. We will discuss throughout this chapter the relevant properties and performance characteristics of all major types of cutting-tool materials, which will help us in tool selection.

3.1 Characteristics of Cutting tool materials:

The cutting tool is subjected to (a) high temperatures, (b) high contact stresses, and (c) rubbing along the tool-chip interface and along the machined surface. Consequently, the cutting-tool material must possess the following characteristics:

- **Hot hardness:** Hot hardness is the ability of a material to retain its hardness at high temperatures. This is required because of the high-temperature environment in which the tool operates.

- **Toughness and impact Strength:** To avoid fracture failure, the tool material must possess high toughness. Toughness is the capacity of a material to absorb energy without failing. It is usually characterized by a combination of strength and ductility in the material.

- **Wear resistance.** Hardness is the single most important property needed to resist abrasive wear. All cutting-tool materials must be hard. However, wear resistance in metal cutting depends on more than just tool hardness, because of the other tool-wear mechanisms. Other characteristics affecting wear resistance include surface finish on the tool (a smoother surface means a lower coefficient of friction), chemistry of tool and work materials, and whether a cutting fluid is used.

- **Thermal Shock Resistance:** To withstand the rapid temperature cycling encountered in interrupted cutting.

- **Chemical stability and inertness** with respect to the material being machined, to avoid or minimize any adverse reactions, adhesion, and tool-chip diffusion that would contribute to tool wear.

3.2 Cutting Tool Materials:

The various cutting tool materials which are broadly used in machining of materials are:
High Speed Steels, Carbon Steels, Carbides, Coated tools, Cubic boran Nitrade, Diamond, Aluminium Oxides etc.,

Carbon Steels: Carbon steels are the oldest tool materials and have been used widely for drills, taps, broaches, and reamers since the 1880s. Low-alloy and medium-alloy steels were developed later for similar applications but with longer tool life. Although inexpensive and easily shaped and sharpened, these steels do not have sufficient hot hardness and wear resistance for cutting at high speeds when the temperature rises significantly. Their use is limited to very low speed cutting operations, particularly in woodworking; hence, they are not of any particular significance in modern machining operations.

High Speed Steels: High-speed steel (HSS) tools are so named because they were developed to machine at higher speeds than Carbon Steels. High-speed steel (HSS) is a highly alloyed tool steel capable of maintaining hardness at elevated temperatures better than high carbon and low alloy steels. Its good hot hardness permits tools made of HSS to be used at higher cutting speeds. Compared with the other tool materials at the time of its development, it was truly deserving of its name ‘‘high speed.’’ A wide variety of high-speed steels are available, but they can be divided into two basic types: (1) tungsten-type, designated T-grades by the American Iron and Steel Institute (AISI); and (2) molybdenum-type, designated M-grades by AISI.

Tungsten-type HSS contains tungsten (W) as its principal alloying ingredient. Additional alloying elements are chromium (Cr), and vanadium (V). One of the original and best known HSS grades is T1, or 18-4-1 high-speed steel, containing 18%W,4%Cr, and 1%V. **Molybdenum HSS** grades contain combinations of tungsten and molybdenum(Mo), plus the same additional alloying elements as in the T-grades. Cobalt (Co) is sometimes added to HSS to enhance hot hardness. Of course, high-speed steel contains carbon, the element common to all steels.

Cemented Carbides: Cemented carbides (also called sintered carbides) are a class of hard tool material formulated from tungsten carbide (WC) using powder metallurgy techniques with cobalt (Co) as the binder. There may be other carbide compounds in the mixture, such as titanium carbide (TiC) and/or tantalum carbide (TaC), in addition to WC.

Because of their high hardness over a wide range of temperatures high elastic modulus, high thermal Conductivity, and low thermal expansion, carbides are among the most important, versatile, and cost-effective tool and die materials for a wide range of applications. The two major groups of carbides used for machining are tungsten Carbide and titanium carbide.

Ceramics: ceramic cutting tools are composed primarily of fine-grained aluminium oxide (Al2O3), pressed and sintered at high pressures and temperatures with no binder into insert form (Section 17.2). The aluminum oxide is usually very pure (99% is typical), although some manufacturers add other oxides (such as zirconium oxide) in small amounts. In producing ceramic tools, it is important to use a very fine grain size in the alumina powder, and to maximize density of the mix through high-pressure compaction to improve the material’s low toughness.

Aluminum oxide cutting tools are most successful in high-speed turning of cast iron and steel. Applications also include finish turning of hardened steels using high cutting speeds, low feeds and depths, and a rigid work setup. Many premature fracture failures of ceramic tools are because of non-rigid machine tool setups, which subject the tools to mechanical shock. When properly applied, ceramic cutting tools can be used to obtain very good surface finish. Ceramics are not recommended for heavy interrupted cut operations (e.g., rough milling) because of their low toughness. In addition to its use as inserts in conventional machining operations, Al2O3 is widely used as an abrasive in grinding and other abrasive processes.

Cubic Boron Nitride: Next to diamond, cubic boron nitride (Section 7.3.3) is the hardest material known, and its fabrication into cutting tool inserts is basically the same as Synthetic polycrystalline Diamonds; that is, coatings on WC–Co inserts. Cubic boron nitride (symbolized CBN) does not react chemically with iron and nickel as SPD does; therefore, the applications of CBN-coated tools are for machining steel and nickel-based alloys. Both SPD and CBN tools are expensive, as one might expect, and the applications must justify the additional tooling cost.

Diamonds: Diamond is the hardest material known. By some measures of hardness, diamond is three to four times as hard as tungsten carbide or aluminum oxide. Since high hardness is one of the desirable properties of a cutting tool, it is natural to think of diamonds for machining and grinding applications. Synthetic diamond cutting tools are made of sintered polycrystalline diamond (SPD), which dates from the early 1970s. Sintered polycrystalline diamond is fabricated by sintering fine-grained diamond crystals under high temperatures and pressures into the desired shape. Little or no binder is used. The crystals have a random orientation and this adds considerable toughness to the SPD tools compared with single crystal diamonds. Tool inserts are typically made by depositing a layer of SPD about 0.5mm (0.020 in) thick on the surface of a cemented carbide base. Very small inserts have also been made of 100% SPD.

Applications of diamond cutting tools include high-speed machining of nonferrous metals and abrasive non-metals such as fiberglass, graphite, and wood. Machining of steel, other ferrous metals, and nickel-based alloys with SPD tools is not practical because of the chemical affinity that exists between these metals and carbon (a diamond, after all, is carbon.)

3.3 Cutting Tool Geometry:

A cutting tool must possess a shape that is suited to the machining operation. One important way to classify cutting tools is according to the machining process. cutting tools can be divided into single-point tools and multiple-cutting-edge tools. Single-point tools are used in turning, boring, shaping, and planing. Multiple-cutting-edge tools are used in drilling, reaming, tapping, milling, broaching, and sawing. Many of the principles that apply to single-point tools also apply to the other cutting-tool types, simply because the mechanism of chip formation is basically the same for all machining operations.

3.3.1 Single point Tool Geometry:

The general shape of a single-point cutting tool is illustrated in Figure 3.1.

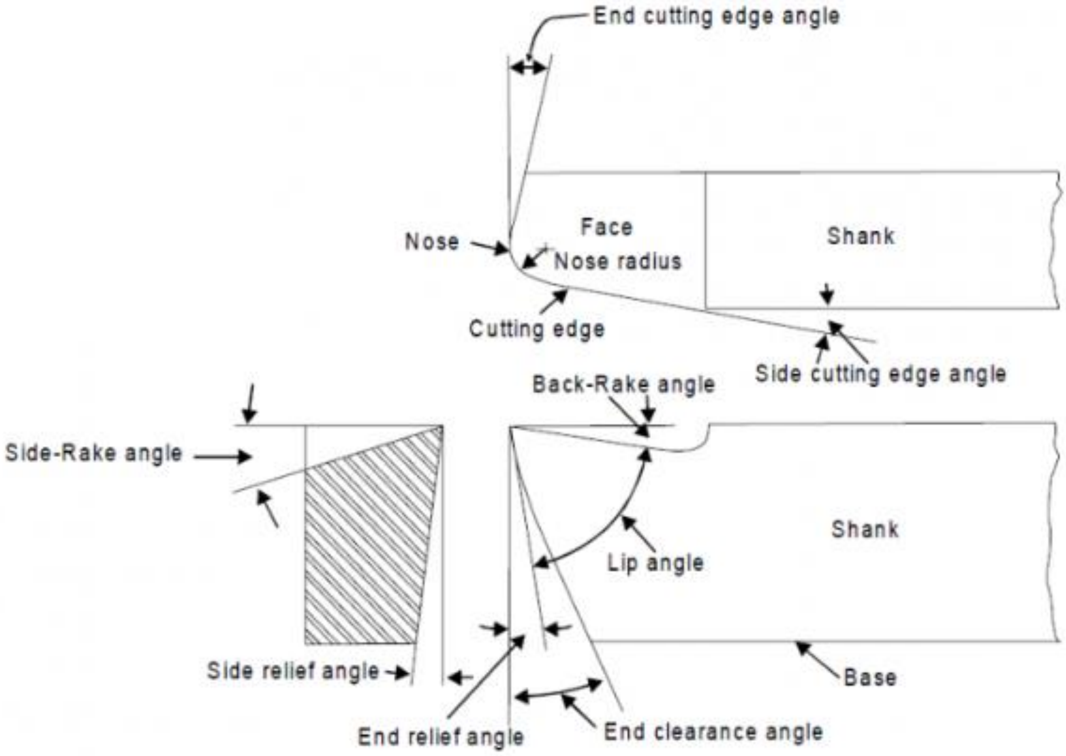

Figure 3.1: Single point Cutting Tool nomenclature

i) **Shank:** It is that portion of the tool which will be hold on the tool post.

ii) **Back Rake angle:** Back rake angle is the angle between the face of the single point cutting tool and a line parallel with base of the tool measured in a perpendicular plane through the side cutting edge. If the slope face is downward toward the nose, it is negative back rake angle and if it is upward toward nose, it is positive back rake angle. Back rake angle helps in removing the chips away from the workpiece.

iii) **Side rake angle:** Side rake angle is the angle by which the face of tool is inclined sideways. Side rake angle is the angle between the surface the flank immediately below the point and the line down from the point perpendicular to the base. Side rake angle of cutting tool determines the thickness of the tool behind the cutting edge. It is provided on tool to provide clearance between workpiece and tool so as to prevent the rubbing of workpiece with end flake of tool.

iv) **End relief angle:** End relief angle is defined as the angle between the portion of the end flank immediately below the cutting edge and a line perpendicular to the base of the tool, measured at right angles to the flank. End relief angle allows the tool to cut without rubbing on the workpiece.

v) **Side relief angle:** Side rake angle is the angle between the portion of the side flank immediately below the side edge and a line perpendicular to the base of the tool measured at right angles to the side. Side relief angle is the angle that prevents the interference as the tool enters the material. It is incorporated on the tool to provide relief between its flank and the workpiece surface.

vi) **End cutting edge angle:** End cutting edge angle is the angle between the end cutting edge and a line perpendicular to the shank of the tool. It provides clearance between tool cutting edge and workpiece

vii) **Side cutting edge angle:** Side cutting edge angle is the angle between straight cutting edge on the side of tool and the side of the shank. It is responsible for turning the chip away from the finished surface.

viii) **Nose Radius:** It is the fillet ground on the edge of the cutting point. This is done in order improve the surface finish on the workpiece while machining.

ix) **Shank:** It is that portion of the tool which will be hold on the tool post.

3.3.2 Twist Drill Tool Geometry:

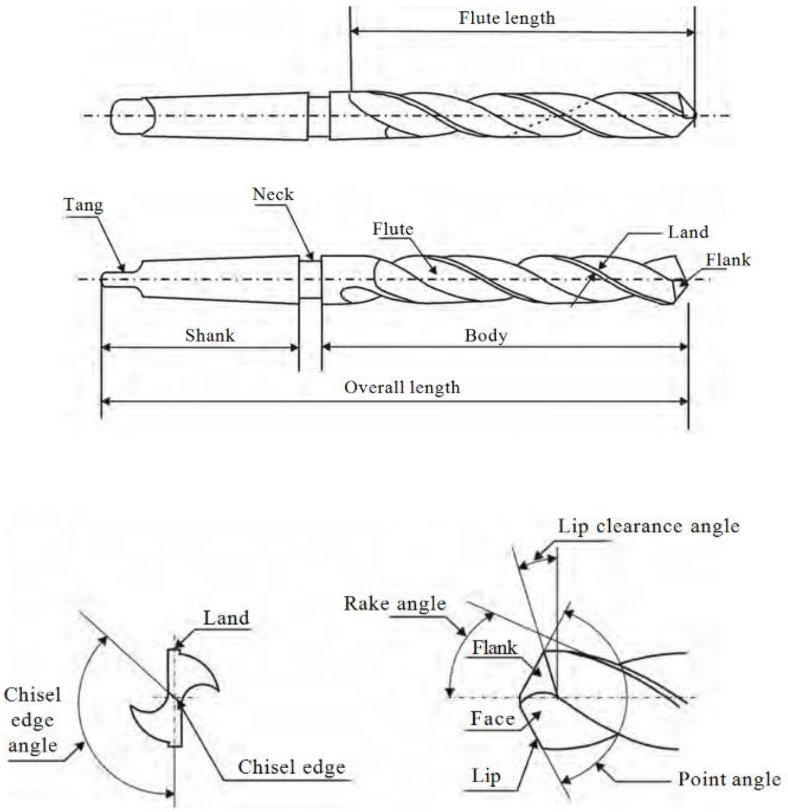

Figure 3.2: Twist Drill Tool nomenclature

Axis: It is the longitudinal center line of the drill running through the centres of the tang and the chisel edge.

Body: It is the part of the drill from its extreme point to the commencement of the neck, if present. Otherwise, it is the part extending up to the commencement of the shank. Helical grooves are cut on the body of the drill.

Shank: It is the part of the drill by which it is held and driven. It is found just above the body of the drill. The shank may be straight or taper. The shank of the drill can be fitted directly into the spindle or by a tool holding device.

Tang: The flattened end of the taper shank is known as tang. It is meant to fit into a slot in the spindle or socket. It ensures a positive drive of the drill.

Neck: It is the part of the drill, which is diametrically undercut between the body and the shank of the drill. The size of the drill is marked on the neck.

Point: It is the sharpened end of the drill. It is shaped to produce lips, faces, flanks and chisel edge.

Lip: It is the edge formed by the intersection of flank and face. There are two lips and both of them should be of equal length. Both lips should be at the same angle of inclination with the axis (59°).

Land: It is the cylindrically ground surface on the leading edges of the drill flutes adjacent to the body clearance surface. The alignment of the drill is maintained by the land. The hole is maintained straight and to the right size.

Flutes: The grooves in the body of the drill are known as flutes. Flutes form the cutting edges on the point. It allows the chips to escape and make them curl. It permits the cutting fluid to reach the cutting edges.

Chisel edge angle: The obtuse angle included between the chisel edge and the lip as viewed from the end of the drill. It usually ranges from 120° to 135°.

Helix angle or rake angle: The helix or rake angle is the angle formed by the leading edge of the land with a plane having the axis of the drill. If the flute is straight, parallel to the drill axis, then there would be no rake. If the flute is right handed, then it is positive rake and the rake is negative if it is left handed. The usual value of rake angle is 30° or 45°.

Point angle: This is the angle included between the two lips projected upon a plane parallel to the drill axis and parallel to the two cutting lips. The usual point angle is 118°. When hard alloys are drilled the value increases.

Lip clearance angle: The angle formed by the flank and a plane at right angles to the drill axis. The angle is normally measured at the periphery of the drill. The lip clearance angle ranges from 12° to 15°.

3.3.3 Milling Cutter Nomenclature:

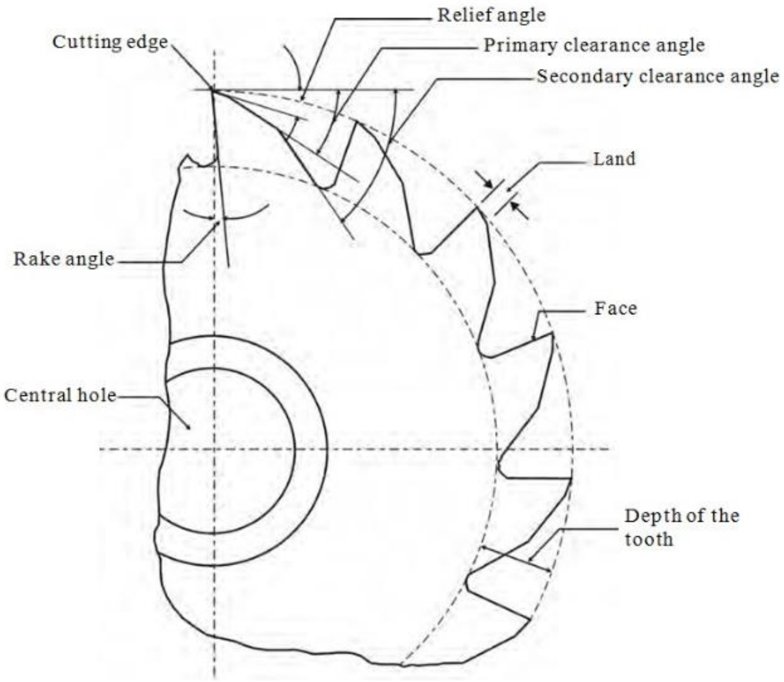

Figure 3.3: Milling cutter Nomenclature

Body of cutter: It is the part of the cutter left after exclusion of the teeth.

Face: The portion of the teeth next to the cutting edge is known as face.

Land: The relieved back portion of the tooth adjacent to the cutting edge. It is relieved to avoid interference between the surface being machined and the cutter.

Outside diameter: The diameter of the circle passing through the peripheral cutting edges.

Relief angle: It is angle the between the land of the tooth and the tangent to the outside diameter of the cutter at the cutting edge of the particular tooth. (approx. 7.5 °)

Primary clearance angle: It is the angle between the back of the tooth and the tangent drawn to the outside diameter of the cutter at the cutting edge. (approx. 15 °)

Secondary clearance angle: It is the angle formed by the secondary clearance surface and the tangent to the periphery of the cutter at the cutting edge.

Rake angle: The angle measured in the diametral plane between the face of the tooth and a radial line passing through the cutting edge of the tooth. The rake angles may be positive, negative or zero.

3.4 Cutting fluids & its applications:

A cutting fluid is any liquid or gas that is applied directly to the machining operation to improve cutting performance. Cutting fluids address two main problems: (1) heat generation at the shear zone and friction zone, and (2) friction at the tool–chip and tool–work interfaces.

In addition to removing heat and reducing friction, cutting fluids provide additional benefits, such as washing away chips (especially in grinding and milling), reducing the temperature of the work part for easier handling, reducing cutting forces and power requirements, improving dimensional stability of the work part, and improving surface finish.

3.4.1 Types of Cutting fluids:

There are four categories of cutting fluids
according to chemical formulation: (1) cutting oils, (2) emulsified oils, (3) semi chemical fluids, and (4) chemical fluids.

Cutting oils are based on oil derived from petroleum, animal, marine, or vegetable origin. Mineral oils (petroleum based) are the principal type because of their abundance and generally desirable lubricating characteristics. To achieve maximum lubricity, several types of oils are often combined in the same fluid. Chemical additives are also mixed with the oils to increase lubricating qualities. These additives contain compounds of sulphur, chlorine, and phosphorus, and are designed to react chemically with the chip and tool surfaces to form solid Films (extreme pressure lubrication) that help to avoid metal-to-metal contact between the two.

Emulsified oils consist of oil droplets suspended in water. The fluid is made by blending oil (usually mineral oil) in water using an emulsifying agent to promote blending and stability of the emulsion. A typical ratio of water to oil is 30:1. Chemical additives based on sulphur, chlorine, and phosphorus are often used to promote extreme pressure lubrication. Because they contain both oil and water, the emulsified oils combine cooling and lubricating qualities in one cutting fluid.

Chemical fluids are chemicals in a water solution rather than oils in emulsion. The dissolved chemicals include compounds of sulphur, chlorine, and phosphorus, plus wetting agents. The chemicals are intended to provide some degree of lubrication to the solution. Chemical fluids provide good coolant qualities but their lubricating qualities are less than the other cutting fluid types. Semi-chemical fluids have small amounts of emulsified oil added to increase the lubricating characteristics of the cutting fluid. In effect, they are a hybrid class between chemical fluids and emulsified oils.

3.4.2 Applications/ functions/ purpose of Cutting fluids:

The primary function of cutting fluid is temperature control through cooling and lubrication. Application of cutting fluid also improves the quality of the workpiece by continually removing metal fines and cuttings from the tool and cutting zone.

1. **To cool the tool:** Cooling the tool is necessary to prevent metallurgical damage and to assist in decreasing friction at the tool chip interface and at the tool workpiece interface.
2. **To cool the work piece:** The role of the cutting fluid in cooling the workpiece is to prevent its excessive thermal distortion.
3. **To lubricate and reduce the friction:** The energy or power consumption in removing metal is reduced. Absorption or wear on the cutting tool is reduced thereby increasing the life of the tool.
4. **To improve the surface finish.**
5. **To protect the finished surface from corrosion.**
6. **To cause the chips break up into small parts.**
7. **To wash away the chips away from the tool.**

3.4.3 Method of application of cutting Fluids:

Cutting fluid may be applied to a cutting tool/workpiece interface through manual, flood or mist application.

Manual application: Simply consists of an operator using a container, such as an oil can, to apply cutting fluid to the cutting tool/workpiece. Although this is the easiest and least costly method of fluid application, it has limited use in machining operations and is often hampered by inconsistencies in application.

Flood application: Delivers fluid to the cutting tool/workpiece interface by means of a pipe, hose or nozzle system. Fluid is directed under pressure to the tool/workpiece interface in a manner that produces maximum results. Pressure, direction and shape of the fluid stream must be regulated in order to achieve optimum performance.

Mist Application: Cutting fluids may also be atomized and blown onto the tool/workpiece interface via mist application. This application method requires adequate ventilation to protect the machine tool operator. The pressure and direction of the mist stream are also crucial to the success of the application.

3.5 Parameters affecting the surface finish:

The major machining parameters affecting the surface finish are Speed, feed and depth of cut. Other parameters are coolants, machining conditions, cutting tool geometry, tool materials etc.,

1. **Speed:** Speed at which the work piece is moved against the cutting tool or the tool against the workpiece plays a vital role. As the cutting speed is more at lower feed rate the machined surface will be smoother. But at higher feed rates the surface finish will be rough as there is a huge friction and heat generation.
2. **Feed:** In order to obtain better surface finish, the feed rate should be keep low. Also it is important to keep the cutting speed high at lower feed rates.
3. **Depth of cut:** Lower depth of cut is always preferred in order to obtain good surface finish. Higher the depth of cut larger will be the friction between the tool and work interface.
4. **Cutting tool Geometry**: The tool should have maintained the desired shape and angels. Wear is unavoidable in machining process. However, the change in the tool geometry results in excessive wear of the tool and also affecting poor surface finish.
5. **Cutting tool materials:** Selection of appropriate cutting tool materials is essential in machining process. Finishing operations requires softer cutting tools than machining rough surfaces. Also based the work piece to be machined, cutting tool is selected.

3.6 Machining equations for cutting operations:

3.6.1 Turning: Turning is a machining process in which a single-point tool removes material from the surface of a rotating workpiece. Turning is traditionally carried out on a machine tool called a lathe, which provides power to turn the part at a given rotational speed and to feed the tool at a specified rate and depth of cut.

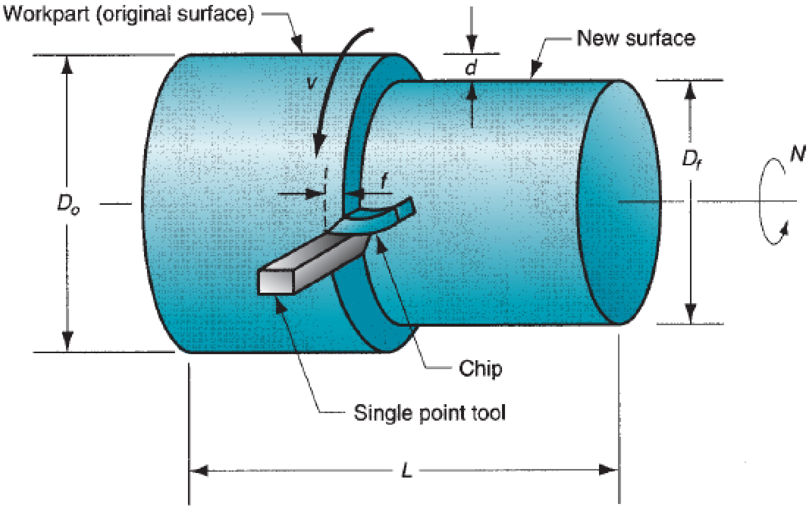

Figure 3.4: Turning operations

The rotational speed in turning is related to the desired cutting speed at the surface of the cylindrical workpiece by the equation

$$N = \frac{v}{\pi D_o}$$

where N=rotational speed, rev/min; v= cutting speed, m/min (ft/min); and Do = original diameter of the part, m (ft).

The turning operation reduces the diameter of the work from its original diameter Do to a final diameter Df, as determined by the depth of cut d:

$$D_f = D_o - 2d$$

The feed in turning is generally expressed in mm/rev (in/rev). This feed can be converted to a linear travel rate in mm/min (in/min) by the formula,

$$f_r = Nf$$

where fr = feed rate, mm/min (in/min); and f = feed, mm/rev (in/rev).

The time to machine from one end of a cylindrical work part to the other is given by,

$$T_m = \frac{L}{f_r}$$

where Tm = machining time, min; and L = length of the cylindrical work part, mm (in). A more direct computation of the machining time is provided by the following equation:

$$T_m = \frac{\pi D_o L}{fv}$$

Where Do = work diameter, mm(in); L = work part length, mm(in); f = feed, mm/rev (in/rev); and v = cutting speed, mm/min (in/min). As a practical matter, a small distance is usually added to the work part length at the beginning and end of the piece to allow for approach and over travel of the tool. Thus, the duration of the feed motion past the work will be longer than Tm. The volumetric rate of material removal can be most conveniently determined by the following equation:

$$R_{MR} = vfd$$

Where, RMR= material removal rate, mm3/min (in3/min). In using this equation, the units for f are expressed simply as mm(in), in effect neglecting the rotational character of turning. Also, care must be exercised to ensure that the units for speed are consistent with those for f and d.

3.6.2 Drilling:

Drilling is a machining operation used to create a round hole in a work part. This contrasts with boring, which can only be used to enlarge an existing hole. Drilling is usually performed with a rotating cylindrical tool that has two cutting edges on its working end. The rotating drill feeds into the stationary work part to form a hole whose diameter is equal to the drill diameter. Drilling is customarily performed on a drill press, although other machine tools also perform this operation.

The cutting speed in a drilling operation is the surface speed at the outside diameter of the drill. It is specified in this way for convenience, even though nearly all of the cutting is actually performed at lower speeds closer to the axis of rotation. To set the desired cutting speed in drilling, it is necessary to determine the rotational speed of the drill. Letting N represent the spindle rev/min,

$$N = \frac{v}{\pi D}$$

where v = cutting speed, mm/min (in/min); and D = the drill diameter, mm (in). In some drilling operations, the workpiece is rotated about a stationary tool, but the same formula applies. Feed f in drilling is specified in mm/rev (in/rev). Recommended feeds are roughly proportional to drill diameter; higher feeds are used with larger diameter drills. Since there are (usually) two cutting edges at the drill point, the uncut chip thickness (chip load) taken by each cutting edge is half the feed. Feed can be converted to feed rate using the same equation as for turning:

$$f_r = Nf$$

where fr = feed rate, mm/min (in/min). Drilled holes are either through holes or blind holes, Figure 3.5. In through holes, the drill exits the opposite side of the work; in blind holes, it does not. The machining time required to drill a through hole can be determined by the following formula:

$$T_m = \frac{t + A}{f_r}$$

where Tm = machining (drilling) time, min; t = work thickness, mm (in); fr = feed rate, mm/min (in/min); and A = an approach allowance that accounts for the drill point angle, representing the distance, the drill must feed into the work before reaching full diameter, Figure 22.10(a). This allowance is given by

$$A = 0.5\, D \tan\left(90 - \frac{\theta}{2}\right)$$

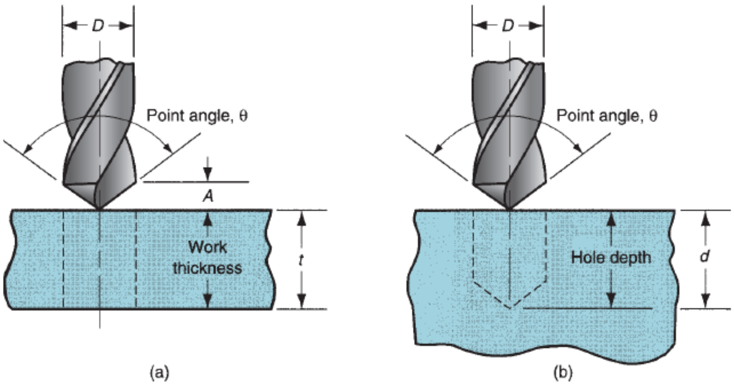

Figure 3.5: Through hole and blind hole

where A = approach allowance, mm (in); and u = drill point angle. In drilling a through hole, the feed motion usually proceeds slightly beyond the opposite side of the work, thus making the actual duration of the cut greater than Tm by a small amount.

In a blind-hole, hole depth d is defined as the distance from the work surface to the depth of the full diameter, Figure 3.5. Thus, for a blind hole, machining time is given by

$$T_m = \frac{d + A}{f_r}$$

where A = the approach allowance by The rate of metal removal in drilling is determined as the product of the drill cross sectional area and the feed rate:

$$R_{MR} = \frac{\pi D^2 f_r}{4}$$

This equation is valid only after the drill reaches full diameter and excludes the initial approach of the drill into the work.

3.6.3: Milling

Milling is a machining operation in which a work part is fed past a rotating cylindrical tool with multiple cutting edges The axis of rotation of the cutting tool is perpendicular to the direction of feed. This orientation between the tool axis and the feed direction is one of the features that distinguishes milling from drilling.

The cutting speed is determined at the outside diameter of a milling cutter. This can be converted to spindle rotation speed using a formula that should now be familiar:

$$N = \frac{v}{\pi D}$$

The feed fin milling is usually given as a feed per cutter tooth; called the chip load, it represents the size of the chip formed by each cutting edge. This can be converted to feed rate by taking into account the spindle speed and the number of teeth on the cutter as follows:

$$f_r = N n_t f$$

where fr=feed rate, mm/min(in/min); N=spindle speed, rev/min; nt=number of teeth on the cutter; and f =chip load in mm/tooth (in/tooth).

Material removal rate in milling is determined using the product of the cross sectional area of the cut and the feed rate. Accordingly, if a slab-milling operation is

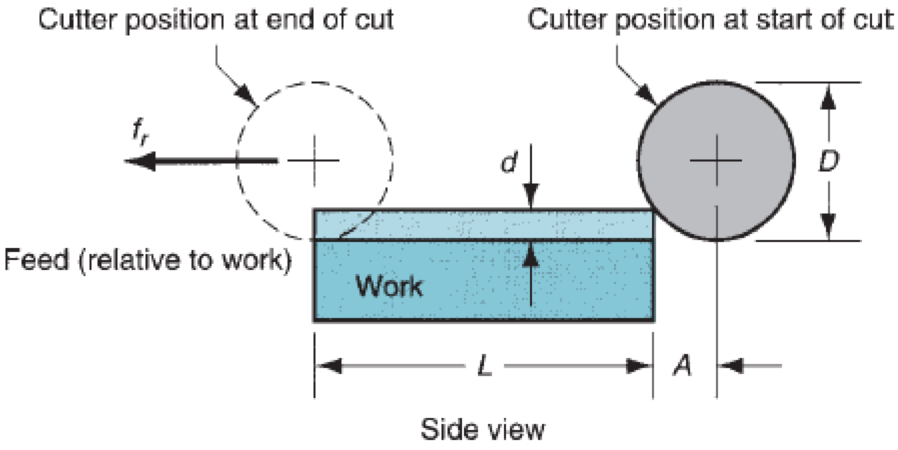

Figure 3.6: Slab Milling operation

cutting a workpiece with width w at a depth d, the material removal rate is

$$R_{MR} = wdf_r$$

This neglects the initial entry of the cutter before full engagement. The Eq can be applied to end milling, side milling, face milling, and other milling operations, making the proper adjustments in the computation of cross-sectional area of cut.

The time required to mill a workpiece of length L must account for the approach distance required to fully engage the cutter. First, consider the case of slab milling, Figure 3.6. To determine the time to perform a slab milling operation, the approach distance A to reach full cutter depth is given by

$$A = \sqrt{d(D-d)}$$

where d=depth of cut, mm(in); and D=diameter of the milling cutter, mm(in). The time Tm in which the cutter is engaged milling the workpiece is therefore

$$T_m = \frac{L+A}{f_r}$$

For face milling, let us consider the two possible cases pictured in Figure 3.7. The first case is when the cutter is centered over a rectangular workpiece as in Figure 3.7(a). The cutter feeds from right to left across the workpiece. In order for the cutter to reach the full width of the work, it must travel an approach distance given by the following:

$$A = 0.5\left(D - \sqrt{D^2 - w^2}\right)$$

where D=cutter diameter, mm(in) and w=width of the workpiece, mm(in). If D=w, then Eq. reduces to A=0.5D. And if D=w, then a slot is cut into the work and A=0.5D.

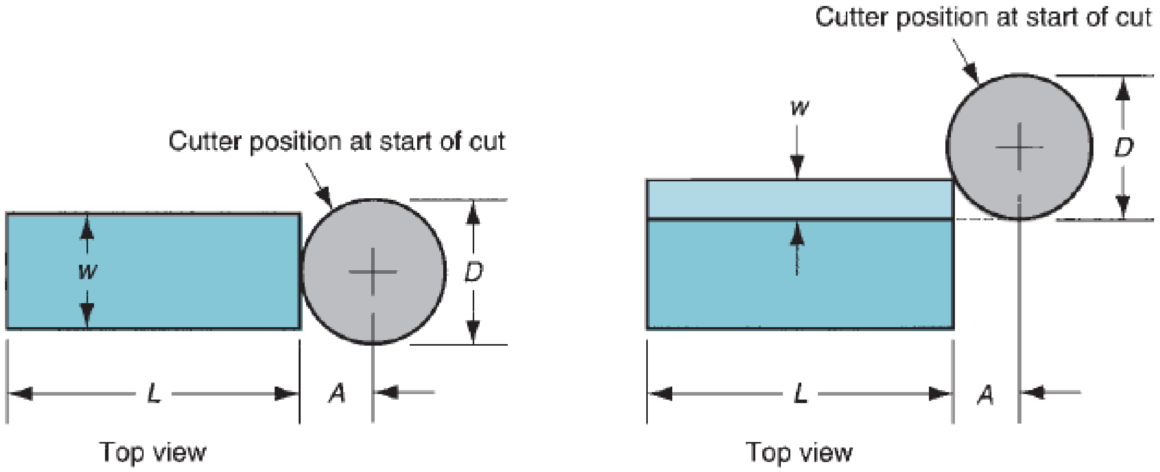

Figure 3.7: Face milling showing approach and over travel distances for two cases: (a) when cutter is centered over the workpiece, and (b) when cutter is offset to one side over the work.

The second case is when the cutter is offset to one side of the work, as in Figure 3.7(b). In this case, the approach distance is given by

$$A = \sqrt{w(D - w)}$$

where w ¼ width of the cut, mm (in). In either case, the machining time is given by

$$T_m = \frac{L + A}{f_r}$$

It should be emphasized in all of these milling scenarios that Tm represents the time the cutter teeth are engaged in the work, making chips. Approach and over travel distances are usually added at the beginning and end of each cut to allow access to the work for loading and unloading. Thus the actual duration of the cutter feed motion is likely to be greater than Tm.

OUTCOMES:

- Student can able understand the various cutting tool materials and the tool signature of single point cutting tool, milling cutter and twist drill.
- Student gather the information about the importance of cutting fluids in machining operations.
- Students can solve the numerical on various aspects of machining of turning operation, milling operation and drilling operations.

QUESTIONS:

1. List and explain the various cutting tool materials used in machining operations.
2. What is tool Signature. With the help of a neat sketch describe the single point cutting tool nomenclature.
3. Describe the nomenclature of milling cutter.
4. Obtain the tool signature of a twist drill with a neat sketch.
5. List the functions of Cutting Fluids.
6. What are the different cutting fluids and method of application of cutting fluid?