UNIT 6 MACHINE AND TOOL SELECTION

Structure

6.1 Introduction
   Objectives

6.2 Factors Influencing the Selection of Tools

6.3 Tool Material
   6.3.1 High Carbon Steels
   6.3.2 High Speed Steel
   6.3.3 Cast Cobalt Tools
   6.3.4 Carbides
   6.3.5 Coated Carbides
   6.3.6 Ceramics
   6.3.7 Diamond
   6.3.8 Cubic Boron Nitride (CBN)
   6.3.9 Polycrystalline Diamond (PCD)

6.4 Properties of Work Material

6.5 Tool Types

6.6 Accuracy and Surface Finish Required

6.7 Geometry of Turning Tools

6.8 Tool Angles

6.9 Classification of Lathe Tools

6.10 Cutting Speed, Feed and Depth of Cut in Turning Operations

6.11 Tools Used in Milling
   6.11.1 Nomenclature of Milling Cutter
   6.11.2 Types of Milling Cutters
   6.11.3 Milling Operations

6.12 Cutting Speed and Feed in Milling Operation

6.13 Tools Used in Drilling
   6.13.1 Classification of Drills
   6.13.2 Twist Drill Nomenclature
   6.13.3 Important Angles of a Drill
   6.13.4 Drill Specification

6.14 Cutting Speed, Feed and Depth of Cut in Drilling Operation

6.15 Summary

6.1 INTRODUCTION

Over the last few decades there has been a rapid development in machining technology and fierce competition right round the world. This has forced the manufacturing engineers to produce goods of highest quality at lowest possible cost. It is important to note the fact that the quality of a product largely depends on the type of machine, the type of process, machining conditions and the right selection of tools etc., this unit shall deal with the aspects of “Selection of Tools and Cutting Conditions”.

Role of Cutting Tools
There has been considerable development in the design of machine tools as well as cutting tools. However, because of the low cost and the small size, development in cutting tools is not fully appreciated and hence are not given their due share in most of the manufacturing organisations. Unlike the machine tools or any capital goods, the decision regarding purchase of tools is often done with very little enquiry or analysis. But, organisations who are aware of the importance of cutting tools have specialists who study and evaluate the performance of cutting tools before a purchase decision is made.

The current trend in manufacturing is to adopt more and more of automation of the fixed and flexible type. Hence, with a machine tool of a given level of automation the scope for improving productivity to a certain extent lies in the selection and application of cutting tools.

Objectives
After studying this unit, you should be able to
• elaborate the factors influencing the selection of tools,
• know the properties of important tool and work material,
• classify the tools, and
• decide the various cutting parameters.

6.2 FACTORS INFLUENCING THE SELECTION OF TOOLS

(a) Properties of tool material :
   (i) Hardness
   (ii) Strength, stiffness and resilience
   (iii) Wear resistance
   (iv) Low coefficient of friction
   (v) Thermal conductivity and specific heat

(b) Properties of work materials

(c) The type of process

(d) The accuracy and surface finish required

(e) The cutting conditions

(f) Tool wear and tool life

(g) Nature of art, e.g. intermittent or continuous.

6.2.1 Properties of Tool Material
The type of tool material and its properties will greatly influence the machining process. The type of machining process and parameters of machining like speed, feed, depth of cut etc., will be chosen for a particular combination of work and tool materials, thus the properties of tool and work materials play an important role in the machining process. The following are the important properties of tool materials.

(a) Hot hardness

(b) Strength, resilience and stiffness

(c) Wear resistance; abrasion resistance

(d) Coefficient of friction
(e) Thermal Conductivity and Specific Heat
(f) Toughness
(g) Ease of fabrication

Hot Hardness

Tool material must be harder than workpiece material. Hardness at room
temperature is not enough. Hardness must be maintained at elevated temperature
prevalent at the cutting zone (1000°C) during machining. The critical temperature
at which the hardness of a tool material starts to fall rapidly signifies the limit of its
capability.

Strength, Resilience and Stiffness

Tool material must have high tensile, compressive and torsional strength to resist
deformation under cutting forces. Strength is required to resist disintegration of
tool material under load. Strength of the tool must be greater than the workpiece
material. Tool material must have sufficient impact resistance to withstand shock
load experienced during the entry and exit of the tool. Greater impact resistance is
required for interrupted cuts. Tool material must have resilience to absorb shock
without permanent deformation and needs to have high stiffness, i.e. high ratio of
load to deflection to suppress chatter and vibration.

Wear Resistance

Tool material must have high wear resistance. Wear resistance is dependent upon
many properties of the material like hot hardness, strength (e.g. crushing),
chemical inertness and propensity for diffusion.

Low Coefficient of Friction

The tool material must have low coefficient of friction to reduce sliding friction
between chip and tool. The friction between chip and tool is more due to welding
between chip and tool and subsequent shearing of the welds. Hence low wettability
by metals is a desirable property of tool material.

Thermal Conductivity and Specific Heat

The tool must absorb heat and transmit it away quickly so that the temperature of
the tool is not increased. In high speed machining, as possible with carbide and
ceramics, the chips carry away most of the heat. Thus this is not an important
property of a tool material. With our understanding of the desirable properties of
tool materials, let us see to what extent these properties are present in the tool
materials of today.

6.3 TOOL MATERIAL

The productivity of engineering industries has increased by continuous development of
cutting tool materials and increased cutting speeds. Right from the conventional Carbon
tool steel and HSS, the development of superior cutting tool materials like coated
carbides, Ceramics, CBN (Cubic Boron Nitride) and polycrystalline diamond etc., have
reduced metal cutting time so low that automation of other elements of operation become
easy and economical. Figure 6.1 shows how drastically the cutting time has reduced, by
the introduction of these new tool materials.

Even though new tool materials are being developed the older tool materials are not
completely wiped out from the scene. They continue to have their own special area of
application and, in fact the older materials also are undergoing further developments and
their capabilities are constantly being extended.
Elements of CAPP

Figure 6.1

The following are some of the important tool materials in use:

(a) High Carbon Steels
(b) High Speed Steels
(c) Cast Cobalt (Stellite) Tools
(d) Carbides
(e) Coated Carbides
(f) Ceramics
(g) Diamonds
(h) Cubic Boron Nitride (CBN)
(i) Polycrystalline Diamond

6.3.1 High Carbon Steels

These are the earliest tool materials used. These are essentially plain carbon steels with carbon percentages between 0.6 and 1.5% and some very small alloy additions such as manganese, silicon, tungsten, molybdenum, chromium and vanadium. The major disadvantage of these cutting tool materials is their inability to withstand high temperatures. Beyond 200°C they lose their hardness and cease to cut. Thus they are useful only for very low cutting speeds (about 0.15 m/s) and to be used with low temperature generating operations such as machining wood, magnesium, brass and aluminium. They are easy to prepare and grind, and as a result they are used for form tool making for low quantity production, and hand tools, e.g. chisels, saw, taps, dies, etc.

6.3.2 High Speed Steels

Taylor and White developed this new generation tool material at the turn of the twentieth century. They were able to significantly improve the cutting speeds by three to five times (about 0.5 m/s) of the cutting speed attainable at that time, using carbon tool steels. Because of this high cutting speed capability they were termed as high speed steels or HSS.

This class of tool materials have significant quantities of tungsten, molybdenum, chromium and vanadium. The complex carbides of tungsten, molybdenum and chromium distributed throughout the metal matrix provides very good hot hardness and abrasion resistance. The major alloying elements which contribute to the hardness is tungsten and
molybdenum. Tungsten is expensive, while molybdenum is cheap but has higher toughness. For the same hardness, less amount of molybdenum needs to be added, however more care needs to be exercised in hardening as decarburizing takes place in molybdenum steels. Also they have a narrow temperature range for heat treatment. Molybdenum tool steels are more popular.

The main advantages of high speed steels is in their high hardness, good wear resistance, high toughness and reasonable cost. Toughness of high speed steels is highest among all the cutting tool materials. Thus they are quite extensively used in interrupted cutting such as in milling. The hardness of HSS falls rapidly beyond 650°C as shown in Figure 6.2 and thus they are limited to lower cutting speeds of the order of 0.5 to 0.75 m/s.

![Figure 6.2](image)

### 6.3.3 Cast Cobalt Tools

Also called white bits, these contain costly materials like cobalt (50%), chromium (28%), tungsten (20%) and carbon (2%). Although very hard and convenient for cutting forgings and castings ‘stellite’ can be shaped only by casting and is very difficult to machine and grind. This limitation of workability has rendered cast cobalt tools uneconomical and obsolete.

### 6.3.4 Carbides

Extremely hard and brittle tungsten carbide powder is mixed with soft and tough cobalt (bond) powder. The mixture is pressed to form blanks. These are pre-sintered at 900°C to impart strength and machinability to the blanks. The blanks can be suitably shaped by turning, drilling, grinding, etc. before a second sintering is carried out between 1400°C and 1600°C. The second sintering melts the cobalt bond and facilitates absorption of carbides. Solidification at 1350°C during cooling imparts excellent toughness and wear resistance to cemented carbides. Finer carbide grains enhance hardness while coarse grain and more cobalt increases toughness. Addition of Tantalum and Niobium carbides improves wear resistance making the tool more suitable for machining of steel and other materials producing long continuous chips.

### 6.3.5 Coated Carbides

Coating carbide tip/inserts with 4-10 microns (0.004-0.01 mm) of titanium carbide coating enhances wear resistance to triple the tool life. Coated carbides also permit operation at 40 per cent higher cutting speed. A mixture of vaporized titanium tetrachloride, hydrogen and methane is passed over a cooling system into the retort. The
titanium carbide layer formed on the surface is bound metallurgically to the cemented carbide tip within. This cycle of vapour deposition coating takes nearly eight hours.

Resharpening of the carbide tips involves lapping on expensive diamond wheels. Consequently, it is much more economical to use clamped carbide tip/insert in a steel holder. The inserts are provided with more than one cutting edges. After wear the insert can be indexed to bring a new cutting edge in the operating position. Consequently, these inserts can be thrown away after wear instead of resorting to expensive lapping on diamond wheels. Figure 6.3 shows how the coating on tools will improve the wear resistance.

![Figure 6.3: Effect of Coating Carbides](image)

**6.3.6 Ceramics**

Ceramics are essentially alumina based high refractory materials introduced specifically for high speed machining of materials and cast iron. These can withstand very high temperatures, are chemically more stable and have higher wear resistance than other cutting tool materials. In view of their ability to withstand high temperatures, they can be used for machining at very high speeds of the order of 10 m/s. It is possible to get a mirror finish on cast iron using ceramic turning. The main problems with ceramic tools are their low strength, poor thermal characteristics and their tendency towards chipping. The machine tools used for ceramic machining have to be extremely rigid to provide vibration free machining conditions. They are not suitable for intermittent cutting or slow cutting speeds.

Apart from the pure alumina based ceramics, sometimes other materials such as titanium carbide are added to enhance their transverse rupture strength. Ceramic tools are used for machining workpieces which have high hardness such as hard castings, case hardened and hardened steels. Typical products that can be machined are brake discs, brake drums, cylinder liners and flywheels. The correct cutting speed produces good surface finish, optimum productivity and better tool life.

Ceramic tools cannot machine some materials such as aluminum and titanium since they have a strong affinity towards them, as a result of which chemical reactions could take place.

Among other things, some of the vital requirements when machining with ceramics tools are:

(a) Using the highest cutting speed recommended and preferably selecting square or round inserts with large nose radius.

(b) Using rigid machines with high spindle speeds and safe clamping angles.

(c) Machining rigid workpieces.

(d) Ensuring adequate and uninterrupted power supply.

(e) Using negative rake angles so that lower cutting stress is applied directly to the ceramic tip.
Using a large nose radius and side cutting edge angle on the ceramic insert reduces the tendency to chipping.

Avoiding coolants with aluminium oxide based ceramics.

### 6.3.7 Diamond

Diamond is the hardest known (Knoop hardness ~ 8000 kg/mm$^2$) material that can be used as a cutting tool material. It has most of the desirable properties of a cutting tool material such as high hardness, good thermal conductivity, low friction, non-adherence to most materials, and good wear resistance. However, the factors that weigh against its use are the high cost, possibility of oxidation in air, allotropic transformation to graphite above temperatures of 700°C, very high brittleness and difficulties associated in shaping it to a suitable cutting tool form.

Natural diamond tools can be used for relatively light cuts and have an extremely high tool life, which justifies their high cost. However, a natural diamond is unreliable because of the impurities present in that and its easy cleavage. Artificial diamonds which are basically polycrystalline are extensively used in industries as a tool material because they can be formed to any given shape with a substrates of cemented carbide.

They are used with a negative rake angle (−5°) for machining hard materials whereas, positive rake angles (15°) can be used for soft materials such as copper. They cannot be used for machining low carbon steels, titanium, nickel, cobalt or zirconium because of the possible reaction with the work material.

### 6.3.8 Cubic Boron Nitride (CBN)

Cubic Boron Nitride (CBN) is next in hardness only to diamond (Knoop hardness ~ 4700 kg/mm$^2$). Not a natural material, it is produced in the laboratory using a high temperature/high pressure process similar to the making of artificial diamond. CBN is less reactive with materials like hardened steels, hard chill cast iron, and nickel base and cobalt based super alloys and hence it is extensively used for machining these alloys. They are more expensive than cemented carbides but have higher accuracy which makes productivity possible for difficult to-machine materials.

### 6.3.9 Polycrystalline Diamond (PCD)

It is the hardest material we know of and has superior abrasive resistance to any other cutting material. Due to it’s extraordinary hardness and wear resistance, PCD cutting tips retain their sharpness even in demanding applications, running up to cutting speeds of 1500 m/min. and beyond. As a result, PCD inserts provide up to 100 times the tool life of carbide. Excellent for light roughing and high speed finishing on turning, boring and milling operations. However, diamonds are found to be unsuitable for machining of ferrous materials especially at temperatures above 700°C, as during machining, diamond converts to relatively weakly bonded graphite. This leads to diffusion wear during machining as carbon has a strong affinity for iron, leading to formation of iron carbides which wears away the cutting tip.

The PCD can be produced to different grades with varying toughness and hardness. They are available in three grain sizes termed fine, medium and coarse. The fine grain PCD is more wear resistant and less resistant to shock than the coarse grained one. The medium grained PCD has properties in between other two.

<table>
<thead>
<tr>
<th>Tool material</th>
<th>Work materials</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Carbon steels</td>
<td>Low strength, softer materials, non-ferrous alloys, plastics</td>
<td>Low cutting speeds, low strength materials</td>
</tr>
<tr>
<td>Low/medium alloy steels</td>
<td>Low strength, softer materials, non-ferrous alloys, plastics</td>
<td>Low cutting speeds, low strength materials</td>
</tr>
<tr>
<td></td>
<td>All materials of low and medium strength and hardness</td>
<td>Low to medium cutting speeds, low to medium strength materials</td>
</tr>
<tr>
<td>-----------</td>
<td>-------------------------------------------------------</td>
<td>---------------------------------------------------------------</td>
</tr>
<tr>
<td>HSS</td>
<td>All materials up to medium strength and hardness</td>
<td>Not suitable for low speed application</td>
</tr>
<tr>
<td>Cemented carbides</td>
<td>Cast iron, alloy steels, stainless steels, super alloys offer additional benefits over uncoated carbides</td>
<td>Not suitable for Titanium alloys, non-ferrous alloys</td>
</tr>
<tr>
<td>Coated carbides</td>
<td>Cast iron, Ni-base super alloys, non-ferrous alloys, plastics</td>
<td>Not suitable for low speed operation or interrupted cutting, machining Al, Ti alloys.</td>
</tr>
<tr>
<td>Ceramics</td>
<td>Cast iron, Ni-base super alloys, non-ferrous alloys, plastics</td>
<td>Not suitable for low speed operation or interrupted cutting, machining Al, Ti alloys.</td>
</tr>
<tr>
<td>CBN</td>
<td>Hardened alloy steels, HSS, Ni-base super alloys, hardened chill cast iron, commercially pure nickel</td>
<td>Suitable for high strength, hard materials</td>
</tr>
<tr>
<td>Diamond</td>
<td>Pure copper, pure aluminium, Al-Si alloys, cold pressed cemented carbides, rock, cement, plastics, glass-epoxy composites, non-ferrous alloys, hardened high carbon alloy steels (for burnishing only), fibrous composites</td>
<td>Not suitable for machining low carbon steels, Co, Ni, Ti and Zr.</td>
</tr>
</tbody>
</table>

PCD inserts provide premium performance mainly in the machining of
(a) highly abrasive alluminium alloys with higher silicon content,
(b) low tensile strength non-ferrous or non-metallic materials,
(c) magnesium, and
(d) copper, zinc and their alloys.

### 6.4 PROPERTIES OF WORK MATERIAL

The work material properties have great influence on the selection of the right type of cutting tool. Even for the similar operations different grades of work materials demand different cutting tools.

Following are some of the properties of the work material that influence the selection of cutting tools.

(a) Hardness
(b) Tensile strength
(c) Modulus of elasticity
(d) Chemical composition
(e) Micro structure
(f) Strain hardenability
(g) Thermal conductivity.

**Hardness**

Hardness is the ability of a material to resist penetration by another body. Hardness is often measured in the Brinell or Rockwell scale. As higher hardness implies higher strength, greater cutting force and power are required for machining a harder material. Greater hardness means greater tool wear under a given cutting condition. For optimum tool life the cutting speed will have to be lower and hence productivity will be lower. Normally a harder material gives better surface finish than a softer material as the harder materials have less tendency to form built up edge.

**Tensile Strength**
Tensile strength of a material is the ultimate strength or the maximum stress the test specimen in a tensile test could sustain during the test. Increased tensile strength calls for higher cutting forces during machining as the specific cutting pressure increases with increasing tensile strength. Also to withstand the increased impact loads at entry, the cutting edge must be made stronger to achieve reasonable tool life. The surface finish is better with increased tensile strength of work material due to reduced tendency to form built up edge.

**Modulus of Elasticity**

This property reflects the ability of a material to resist deflection when subjected to external loads. Greater the value of modulus of elasticity lesser is the deflection under load.

Materials with moderate or low modulus of elasticity tend to deflect more under cutting force and hence must be machined with a sharp cutting edge, i.e. with positive rake tool. Material with larger modulus of elasticity are more rigid and hence require a tool with high strength cutting edge (negative rake).

**Chemical Composition**

Chemical composition by itself does not have any influence on machinability. However they influence machinability to the extent they affect the hardness and tensile strength of the material. Under certain machining, the chemical constituents of the work material influence the chemical phenomenon taking place at the tool-chip interface and hence affect the tool life.

Macro inclusions occur in steel unavoidably during the process of manufacture. The inclusions could be hard or soft. Hard inclusions like Al₂O₃ and SiO₂ increase the rate of abrasive wear and reduce tool life. They have no influence on cutting force or surface finish. However, their presence can induce self excited vibrations during cutting.

On the other hand soft inclusions like or manganese sulphide in steel (occurring due to the addition of lead or sulphur to render the steel free machining) apparently smear over the rake face and prevent metal to metal contact between the chip and the tool. Thus soft inclusions improve tool life. In addition, the soft inclusions reduce friction between the chip and tool and reduce cutting force and power. They also improve surface finish by reducing the formation of built up edge.

**Microstructure**

Microstructure has considerable influence on the machinability of a material. For example the microstructure of steel consists of basic constituents of ferrite, pearlite and cementite. Cementite is the hardest, ferrite the softest and pearlite in between. The proportion of cementite, pearlite and ferrite in the microstructure will determine the hardness of the material. Similarly the amount of carbides in the microstructure would also influence the hardness of the material. Thus the microstructure of a material affects the cutting force and power to the extent it affects the hardness and tensile strength of the material.

**Strain Hardenability**

Many metals exhibit a dramatic increase in hardness when they are subjected to strain in the form of cold working. During cold working operations like rolling, forging and bending internal stresses are developed which act to harden the material. The degree to which a material gets hardened due to strain is a typical characteristic of the material.

Since the material gets increasingly hardened as the machining process is proceeding, the cutting force and power required for these strain hardening material are high. The tool life is also very low as the strain hardened particles of the work material cause increased abrasive wear.

However for dead soft material strain hardening by cold working improves tool life and surface finish as it helps avoid built up edge formation.

**Thermal Conductivity**
Thermal conductivity of a material is the ability of the material to conduct heat through it which is expressed as K.Cal/m/hr/°C. If a work material has very low thermal conductivity then the heat generated in the machining process is not transmitted away from the cutting zone. Thus the tool and work piece become extremely hot. This excess heat accelerates the wear at the cutting edge.

### 6.5 TOOL TYPES

Basically the tools can be classified as **Generating tools** and **Forming tools**.

**Generating Tools**

These tools are used to generate required profiles by manipulating the relative motions of the workpiece and the tool cutting edge. Thus the obtained contour of the workpiece would not be identical to the shape of the cutting tool edge. This is generally used for a majority of the general profiles required. The surface shown in Figure 6.4 is produced as a result of the primary motion of the workpiece as well as secondary (feed) motion of the cutting tool.

![Generated Surface](image)

**Forming Tools**

Forming is an alternate method of producing the required profiles, in which the shape of the cutting tool is impressed upon the workpiece as shown in Figure 6.5. The accuracy of the shape thus obtained is dependent upon the accuracy of the form of the tool used.

![Forming Tools](image)

However, many machine tool operations employ a combination of the above two processes.

Tools further are selected depending upon the metal removal rate (MRR) required and the type of operation used. The details of which shall be discussed further.

### 6.6 ACCURACY AND SURFACE FINISH REQUIRED

At the time of process planning the tools are selected depending on the accuracy and surface finish required. This is because of the fact that different operations and different tools would produce different accuracy levels and surfaces. The table given below will give you a rough idea of the above statement.

<table>
<thead>
<tr>
<th>Operations</th>
<th>Roughness (RMS) Microns</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>25 12.5  6.25  3.2  1.6  0.8  0.4  0.2  0.1  0.05  0.025</td>
</tr>
<tr>
<td>Flame cutting, Sawing</td>
<td></td>
</tr>
<tr>
<td>Hand grinding</td>
<td></td>
</tr>
<tr>
<td>Filing, disc grinding</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Operations</th>
<th>Roughness (RMS) Microns</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>25 12.5  6.25  3.2  1.6  0.8  0.4  0.2  0.1  0.05  0.025</td>
</tr>
<tr>
<td>Flame cutting, Sawing</td>
<td></td>
</tr>
<tr>
<td>Hand grinding</td>
<td></td>
</tr>
<tr>
<td>Filing, disc grinding</td>
<td></td>
</tr>
</tbody>
</table>
Table 6.2: Accuracy Achievable in Machining Operations

<table>
<thead>
<tr>
<th>Machining Operation</th>
<th>Accuracy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turning</td>
<td>± 25 μm</td>
</tr>
<tr>
<td>Shaping, Slotting</td>
<td>± 25 μm/side</td>
</tr>
<tr>
<td>Planning</td>
<td>± 65 μm/side</td>
</tr>
<tr>
<td>Milling</td>
<td>± 12 to 25 μm</td>
</tr>
<tr>
<td>Drilling in drill press</td>
<td>± 250 μm</td>
</tr>
<tr>
<td>Lathe drilling</td>
<td>± 125 μm</td>
</tr>
<tr>
<td>Boring</td>
<td>± 50 μm</td>
</tr>
<tr>
<td>Internal grinding</td>
<td>± 12 μm</td>
</tr>
<tr>
<td>Reaming</td>
<td>± 2.5 μm</td>
</tr>
<tr>
<td>Reaming with jig</td>
<td>± 2.5 μm</td>
</tr>
<tr>
<td>Jig boring</td>
<td>± 12.5 μm</td>
</tr>
<tr>
<td>Cylindrical and surface grinding</td>
<td>± 5 μm</td>
</tr>
<tr>
<td>Thread cutting</td>
<td>± 2.5 μm</td>
</tr>
<tr>
<td>Broaching</td>
<td>± 12.5 μm</td>
</tr>
<tr>
<td>Lapping</td>
<td>± 5 μm</td>
</tr>
<tr>
<td>Honing</td>
<td>± 12.5 μm</td>
</tr>
</tbody>
</table>

6.6.1 Tool Life Equation

At low cutting speeds, the tools have higher life but the productivity is low, and at higher speeds the productivity is high but tool life is less. This fact has inspired Taylor informulating the relationship of tool life and the cutting speed. After experimentation he proposed the following formula for tool life:

\[ V T^n = C \]

where \( n \) = Constant for tool material,
\( V \) = The cutting speed, m/min,
\( C \) = Machining constant, and
\( T \) = The tool life in min.

Table 6.3: Constant \( C \) for Taylor’s Equation

<table>
<thead>
<tr>
<th>Work material</th>
<th>Tool material</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>HSS</td>
</tr>
<tr>
<td></td>
<td>C</td>
</tr>
</tbody>
</table>
As this formula doesn’t take into account all the effecting parameters, e.g. feed, depth of cut, etc. (as shown in Figures 6.6(a) and (b) for sharp cutting tools and tools having a nose radius respectively) many researchers have modified this formula.

(a) \[ T \theta^b = C \] \hspace{1cm} \ldots (6.1)

(b) \[ VT^{0.5-2x} \left[ \frac{T_c H^{0.5}}{C' u_s A} \right]^{1-2x} \] \hspace{1cm} \ldots (6.2)

where \( H = \) Specific heat x thermal conductivity, 
\( \theta = \) Mean tool chip interface temperature, 
\( A = \) Area of cut, 
\( u_s = \) Specific cutting energy/unit cutting force, and 
\( C \) and \( x \) are constants.

(c) \[ VT^a f^{a_1} d^{a_2} = C \] \hspace{1cm} \ldots (6.3)

where \( f = \) Feed, mm/rev, 
\( d = \) Depth of cut, and 
\( C = \) Constant.

Figure 6.6(c) shows the relationship between feed rate and surface finish.
Example 6.1

Find speed for two hours tool life for dry machining of free cutting mild steel with HSS tool.

Solution

From Table 6.3, a value of \( C = 74 \text{ m/min} \) is chosen whereas for HSS tool a mean value of \( n = 0.125 \) has been assumed.

\[
V t^n = C
\]

\[
V = \frac{C}{t^n} = \frac{74}{120^{0.125}}
\]

\[
= \frac{74}{1.82} = 40.67
\]

\[
= 40.67 \text{ metres/minute}
\]

Example 6.2

A form tool machining at 24 m/minute requires re-sharpening after 30 minutes. What should be the cutting speed for prolonging tool life to 70 minutes \( n = 0.10 \).

Solution

\[
V t^n = V_1 t_1^n
\]

\[
V_1 = V \left[ \frac{t}{t_1} \right]^n = 24 \left[ \frac{30}{70} \right]^{0.1}
\]
6.7 GEOMETRY OF TURNING TOOLS

Turning is the process of producing the required profiles on the cylindrical jobs. In this process the cutting tool is fed against a rotating job.

All cutting tools used in lathes are single point tools, i.e., having only a single cutting edge at one end. The cutting edge is formed at the end of a solid tool, its body (shank) is made long enough so that it can be gripped directly in the tool post. Alternatively, the cutting edge may be formed on the edge of a small bit, which is held in a tool ‘holder’ or ‘brazed’ on a solid shank of a cheaper metal. The shape of the cutting edge varies according to the operation to be performed.

6.7.1 Cutting Tool Geometry

Cutting tool nomenclature comprises the various parts of tool and systematic arrangement of tools angles. The complete nomenclature of single point cutting tool is shown in Figure 6.7.

![Figure 6.7: Nomenclature of Turning Tool](image)

**Shank**

The shank is that part of the tool on one end of which the tool point is formed or the tip or bit is supported.

**Base**

The base is the surface of the shank which bears against the support and takes the tangential pressure of the cut.

**Heel**

The heel is the edge between the base and the flank immediately below the face.

**Point or Tool Tip**

The point is end of the tool, which is shaped to produce the cutting edges and face.

**Cutting Edge**

The portion of the tool edge along which the chip is separated from the work. The cutting edge usually comprises the side cutting edge, the nose radius, and the end cutting edge.

**The Side (Primary) Cutting Edge**

This is the edge formed by intersection of the tool face and side flank. It is mainly responsible for shearing of the work material during cutting.
**The End (Auxiliary) Cutting Edge**

This is the edge formed by joining the side-cutting and end-cutting edges.

**Flank**

The flank is the surface or surfaces below and adjacent to the cutting edge. These are the surfaces of the tool facing the work.

---

### 6.8 TOOL ANGLES

**Back-rake Angle**

The back-rake angle is the angle between the face of a tool and a line parallel to the base of the tool shank measured in a plane parallel to the tool base and passing through the tool point. The angle is positive if the face slopes downwards from the point towards the shank and is negative if the face slopes upwards towards the shank.

**Side-rake Angle**

The side-rack angle is the angle between the face of a tool and a line parallel to the base. It is measured in a plane at right angles to the base and passing through the tool point.

**End-relief Angle**

The angle between a plane perpendicular to the base of a tool and ground flank immediately adjacent to the cutting edge.

**Side-relief Angle**

This is the angle between the portion of the flank immediately below the cutting edge and a line drawn through this cutting edge perpendicular to the base. It is measured in a plane at right angles to the centerline of the point.

**Clearance Angle**

The clearance angle is the angle between a plane perpendicular to the base of a tool and that portion of the flank immediately adjacent to the base.

**End Clearance Angle**

This is the secondary relief angle between the plane perpendicular to the base and end flank (surface below side or main edge) immediately adjacent to the base.

**Nose**

The nose is the curve formed by joining the side cutting and end cutting edges. The angle included between the side cutting edge and end cutting edge is called nose angle and radius will give a better finish, but will also promote chattering in the setup. Therefore the smallest nose radius that will give desired finish should be used.

---

### 6.8.1 Tool Specification

The tool angles are normally specified in the following sequence:

- Tool designation: 8, 14, 6, 6, 20, 15, 0.8

<table>
<thead>
<tr>
<th>Back rake</th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Side rake</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>End relief</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Side relief</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Elements of CAPP

End cutting edge

Side cutting edge

Nose radius

Table 6.4: Recommended Tool Angles in Degrees for High Speed Steel Cutting Tools

<table>
<thead>
<tr>
<th>Work Material</th>
<th>Back Rake Angle</th>
<th>Side Rake Angle</th>
<th>Side Relief Angle</th>
<th>Front Relief Angle</th>
<th>Side Cutting Edge Angle</th>
<th>End Cutting Edge Angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel</td>
<td>8-20</td>
<td>8-20</td>
<td>6</td>
<td>6</td>
<td>10</td>
<td>15</td>
</tr>
<tr>
<td>Cast steel</td>
<td>8</td>
<td>8</td>
<td>6</td>
<td>6</td>
<td>10</td>
<td>15</td>
</tr>
<tr>
<td>Cast iron</td>
<td>0</td>
<td>4</td>
<td>6</td>
<td>6</td>
<td>10</td>
<td>15</td>
</tr>
<tr>
<td>Bronze</td>
<td>4</td>
<td>4</td>
<td>6</td>
<td>6</td>
<td>10</td>
<td>10</td>
</tr>
<tr>
<td>Stainless steel</td>
<td>8-20</td>
<td>8-20</td>
<td>6</td>
<td>6</td>
<td>10</td>
<td>15</td>
</tr>
</tbody>
</table>

Table 6.5: Recommended Tool Angles in Degrees for Cast Alloy Cutting Tools

<table>
<thead>
<tr>
<th>Work Material</th>
<th>Back Rake Angle</th>
<th>Side Rake Angle</th>
<th>Side Relief Angle</th>
<th>Front Relief Angle</th>
<th>Side Cutting Edge Angle</th>
<th>End Cutting Edge Angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel</td>
<td>8-20</td>
<td>8-20</td>
<td>6</td>
<td>6</td>
<td>10</td>
<td>15</td>
</tr>
<tr>
<td>Cast steel</td>
<td>8</td>
<td>8</td>
<td>6</td>
<td>6</td>
<td>10</td>
<td>15</td>
</tr>
<tr>
<td>Cast iron</td>
<td>0</td>
<td>4</td>
<td>6</td>
<td>6</td>
<td>10</td>
<td>15</td>
</tr>
<tr>
<td>Bronze</td>
<td>4</td>
<td>4</td>
<td>6</td>
<td>6</td>
<td>10</td>
<td>10</td>
</tr>
<tr>
<td>Stainless steel</td>
<td>8-20</td>
<td>8-20</td>
<td>6</td>
<td>6</td>
<td>10</td>
<td>15</td>
</tr>
</tbody>
</table>

Table 6.6: Recommended Tool Angles in Degrees for Carbide Cutting Tool

<table>
<thead>
<tr>
<th>Work Material</th>
<th>Back Rake Angle</th>
<th>Side Rake Angle</th>
<th>Side Relief Angle</th>
<th>End Relief Angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminium and magnesium alloys</td>
<td>0-10</td>
<td>10-20</td>
<td>6</td>
<td>6</td>
</tr>
<tr>
<td>Copper</td>
<td>0-4</td>
<td>15-20</td>
<td>6-8</td>
<td>6-8</td>
</tr>
<tr>
<td>Brass and bronze</td>
<td>0-5</td>
<td>-5-8</td>
<td>6-8</td>
<td>6-8</td>
</tr>
<tr>
<td>Cast iron</td>
<td>-7-0</td>
<td>-7-6</td>
<td>5-8</td>
<td>5-8</td>
</tr>
<tr>
<td>Plain carbon steels</td>
<td>-7-0</td>
<td>-7-6</td>
<td>5-8</td>
<td>5-8</td>
</tr>
<tr>
<td>Alloy steels</td>
<td>-7-0</td>
<td>-7-6</td>
<td>5-8</td>
<td>5-8</td>
</tr>
<tr>
<td>Stainless steels</td>
<td>-7-0</td>
<td>-7-6</td>
<td>5-8</td>
<td>5-8</td>
</tr>
<tr>
<td>Titanium alloys</td>
<td>-5-6</td>
<td>-5-0</td>
<td>5-8</td>
<td>5-8</td>
</tr>
</tbody>
</table>

6.9 CLASSIFICATION OF LATHE TOOLS

The single point lathe tools are classified as following:

(a) According to the direction of feed:
   (i) Right hand tools – fed from right to left.
   (ii) Left hand tools – fed from left to right.

(b) According to the form of the point:
   (i) Straight shank – when viewed from tool tip with straight axis.
   (ii) Bent shank tools: when viewed from top the tools, carry their axes as bent towards right or left near the face.

(c) According to the purpose:

Depending on purpose of applications the tools can be classified as turning, facing, threading, boring, parting off, grooving, forming, chamfering tool, etc.
According to the method of manufacturing:

(i) **Forged or Solid Tool**: Tools are forged to the required shape in a single piece.

(ii) **Tipped Tool**: These consist of HSS, cemented carbide or ceramic tip, brazed or mechanically held on a carbon steel shank or clamped mechanically onto specially made tool holders.

6.9.1 Lathe Operations

**Centering**

Producing conical holes at the ends of the work process by using center drills.

**Turning**

Operation in which excess material is removed from the work periphery to produce cylindrical or conical surfaces. This may include:

(a) Straight turning

(b) Shoulder turning

(c) Taper turning

**Facing**

Process of producing flat end surface that is normal to the work axis rotation.

**Thread Cutting**

It is the process of cutting helical groove over a cylindrical or conical work piece. For thread cutting, it is necessary to relate the tool travel with rotation of the work piece by using change gears.

**Knurling**

It is the process of indentation of various forms on cylindrical work surface. A knurl tool is held in tool post and pressed against the rotating work with the cross slide, and then it is fed for required length with the carriage.

**Parting Off**
Parting off or cutting off is the process of separating (cutting) the finished work piece from a bar stock. It is done by a parting tool.

**Forming**

It is the process of producing special contours on the work surface by using form tools.

**Drilling**

It is the process of making holes on work piece. The drill is held stationary in the lathe tail stock spindle, and then this is slowly fed into the rotating work piece.

**Boring**

It is the process of enlarging the previously drilled hole with an aid of a single-point cutting tool (Boring tool). It can be referred as internal turning.

**Reaming**

It is the process of making a hole smoothly and accurately to size. Holes may be reamed by a straight shank reamer or taper shank reamer.

### 6.10 CUTTING SPEED, FEED AND DEPTH OF CUT IN TURNING OPERATIONS

**Cutting Speed**

Expressed in terms of mts./min, it is the speed at which the metal is removed by the cutting edge from the work piece.

\[
V_c = \frac{\pi DN}{1000} \text{ m/min}
\]

where

\[
D = \text{Dia. of workpiece, and}
\]

\[
N = \text{Spindle speed (RPM)}.
\]

**Feed**

Feed is defined as the distance the cutting tool advances per one revolution of the work. It is expressed in mm/rev.

**Depth of Cut**

This is the distance between the unfinished surface of the work and the bottom of the cut, measured in a direction normal to the machine a surface of the work.

\[
\text{Depth of cut} = \left( d - \frac{d_c}{2} \right)
\]

where

\[
d = \text{Dia. of unfinished work, and}
\]

\[
d_c = \text{Dia. of finished work.}
\]

**Table 6.7 : Cutting Process Parameters for Turning**

<table>
<thead>
<tr>
<th>Work Material</th>
<th>Hardness BHN</th>
<th>High Speed Steel Tool</th>
<th>Carbide Tool</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Speed m/min</td>
<td>Feed mm/rev</td>
</tr>
<tr>
<td>Grey cast Iron</td>
<td>150-180</td>
<td>30</td>
<td>0.25</td>
</tr>
<tr>
<td>Grey cast Iron</td>
<td>220-260</td>
<td>20</td>
<td>0.25</td>
</tr>
<tr>
<td>Malleable Iron</td>
<td>160-220</td>
<td>33</td>
<td>0.25</td>
</tr>
<tr>
<td>Malleable Iron</td>
<td>240-270</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Cast steel</td>
<td>140-180</td>
<td>40</td>
<td>0.25</td>
</tr>
<tr>
<td>Cast steel</td>
<td>190-240</td>
<td>26</td>
<td>0.25</td>
</tr>
<tr>
<td>C20 steel</td>
<td>110-160</td>
<td>40</td>
<td>0.30</td>
</tr>
</tbody>
</table>
6.10.1 Formulae to be Remembered

(a) \[ V = \frac{\pi DN}{1000} \text{ m/min} \]

where, 
\( V \) = Cutting speed, m/min
\( D \) = Diameter of the workpiece, mm
\( N \) = Rotational speed of workpiece, rpm

(b) Time for single pass
\[ t = \frac{L + L_0}{fN} \text{ min} \]

where, 
\( L \) = Length of the job, mm
\( L_0 \) = Over travel of the tool beyond the length of the job to help in tool setting, mm
\( f \) = Feed rate, mm/rev.
[over travel is usually 2 to 3 mm on either side]

(c) Roughing passes
\[ d_r = A - A_f \]

where, 
\( A \) = Total machining allowance, mm
\( A_f \) = Finish machining allowance, mm
\( d_r \) = Depth of cut in roughing, mm

(d) Finishing passes
\[ d_f = A_f \]

where, 
\( d_f \) = Depth of cut in finishing, mm

(e) Power required
\[ P = \frac{F \times V}{60} = \frac{(k \times d \times f) \times V}{60} \]

where, 
\( P \) = Power, kW
\( F \) = Cutting force, kN
Elements of CAPP

\[ k = \text{Constant depending in work material, N/mm}^2 \quad \text{(specific cutting pressure)} \]

\[ f = \text{Feed rate mm/rev.} \]

\[ d = \text{Depth of cut, mm} \]

\[ V = \text{Cutting speed, m/min} \]

\[ x, y \text{ are constants depending upon the work material.} \]

**Example 6.3**

Calculate the spindle speed (N) if a C20 steel job of 50 mm dia is to be turned by (a) HSS tool and (b) Carbide Tool.

**Solution**

(a) \( D = 50 \text{ mm} \)

From table above

The cutting speed for C20 steel \( V_c = 40 \text{ m/min} \)

We know

\[ N = \frac{V_c \times 1000}{\pi D} = \frac{40 \times 1000}{\pi \times 50} = 254.54 \text{ rpm} \]

∴ The spindle speed should be \( N = 254.54 \text{ rpm} \) when an HSS tool is used.

(b) \( D = 50 \text{ mm}; V_c = 150 \text{ (from table)} \)

\[ V_c = \frac{\pi DN}{1000} \text{ m/min} \]

\[ N = \frac{V_c \times 1000}{\pi D} = \frac{150 \times 1000}{\pi \times 50} = 954 \text{ rpm} \]

∴ The spindle speed should be \( N = 954 \text{ rpm} \) when an carbide tool is used.

**Example 6.4**

Calculate the cutting speed when a workpiece of 100 mm dia is being turned. The spindle speed being 200 rpm.

**Solution**

Dia of the job \( D = 100 \text{ mm} \)

Spindle speed \( N = 200 \text{ rpm} \)

Cutting speed \( V_c = \frac{\pi DN}{1000} \text{ m/min} \)

\[ = \frac{\pi \times 100 \times 200}{1000} = 62.85 \text{ m/min} \]

**Example 6.5**

Estimate the actual machining time required for the component of diameter 42 mm and length 120 mm. The available speeds are 70, 110, 176, 280, 440, 700, 1100, 1760 and 2800. Use a roughing speed of 30 m/min and finish speed of 60 m/min. the feed roughing is 0.24 mm/rev. while that for finishing is 0.10 mm/rev. The maximum depth of cut for roughing is 2 mm. Finish allowance may be taken as 0.75 mm. Blank to be used for machining is 50 mm in diameter.

**Solution**

Stock to be removed \[ = \frac{50 - 42}{2} = 4 \text{ mm} \]
Finish allowance = 0.75 mm

**Roughing**

Stock available = 4 – 0.75 = 3.25 mm

Since maximum depth of cut to be taken is 2 mm, there are two roughing passes.

Given cutting speed \( V = 30 \text{ m/min} \)

Average depth of cut \( \frac{50 + 42}{2} = 46 \text{ mm} \)

Spindle speed \( N = \frac{1000 \times 30}{\pi \times 46} = 207.59 \text{ rpm} \)

The nearest rpm available from the list is 176 rpm as 280 is very high compared to 207 as calculated.

Machining time for one pass \( \frac{(120 + 2)}{0.24 \times 176} = 2.898 \text{ minutes} \)

**Finishing**

Given cutting speed \( V = 60 \text{ m/min} \)

Spindle speed \( N = \frac{1000 \times 30}{\pi \times 42} = 439.05 \text{ rpm} \)

The nearest rpm available from the list is 440 rpm

Machining time for one pass \( \frac{120 \times 2}{0.10 \times 440} = 2.77 \text{ minutes} \)

Total machining time = \( 2 \times 2.888 + 2.77 = 8.546 \text{ minutes} \)

**Example 6.6**

Calculate the power required for roughing and finishing passes in Example 6.5.

[Hint : \( K = 1600 \text{ N/mm}^2 \)].

**Solution**

**Roughing**

Given feed rate \( f = 0.24 \text{ mm/rev.} \)

Depth of cut \( d = 2 \text{ mm} \)

Cutting speed \( \frac{\pi \times 176 \times 46}{1000} = 25.43 \text{ m/min} \)

The value of \( K = 1600 \)

Power \( = \frac{1600 \times 25.43 \times 0.24 \times 2}{60} = 325.5 \text{ W} = 0.326 \text{ kW} \)

**Finishing**

Given feed rate \( f = 0.10 \text{ mm/rev.} \)

Depth of cut \( d = 0.75 \text{ mm} \)

Cutting speed \( \frac{\pi \times 440 \times 43.5}{1000} = 60.13 \text{ m/min} \)

Power \( = \frac{1600 \times 60.13 \times 0.10 \times 0.75}{60} = 120.26 \text{ W} = 0.120 \text{ kW} \).
6.11 TOOLS USED IN MILLING

Milling is a machining process in which metal is removed by a rotating multiple edged cutting tool called ‘milling cutter’ while the workpiece is fed against it. The work can be fed in vertical, longitudinal or cross direction.

![Figure 6.9 : Nomenclature of Milling Cutter](image)

6.11.1 Nomenclature of Milling Cutter

The nomenclature of milling cutter includes various elements and angle of cutters.

**Body**

The part of the cutter to which the teeth are formed or attached at its periphery.

**Cutting Edge**

Edge formed by the face and the circular land or the surface which is forming the primary clearance.

**Face**

The surface adjacent to the cutting edge on which the chip impinges as it is cut from the work.

**Gash**

Gash or flute is the chip space between the back of one tooth and the face of the next tooth.

**Fillet**

The curved surface which joins the face of one tooth to the back of the tooth immediately ahead.

**Land**

The part of the back of the tooth which is adjacent to cutting edge.

**Lip Angle**

Included angle between the land and the face of tooth is called lip angle.

**Primary Clearance**

It is the angle between land surface (or a line passing through land) and a tangent to the periphery at the cutting edge. For the most of the cutters the clearance of 5° is provided.

**Secondary Clearance (Relief) Angle**

To control the land width, a secondary clearance is ground on the tooth. It is the angle between back of teeth and a line passing through land. It is usually 3° greater than primary clearance angle.
Radial Rake Angle

The angle between face of the cutter and a radial line passing through the tooth of cutting edge. It facilitates removal of chips. The radial rake angle usually ranges from 10° to 20°. Larger angles are adopted for milling soft materials and smaller angles for harder material. Carbide tipped cutters are provided with a negative rake angle which varies from 10° to 15°.

Helix Angle

The angle between the tangent to helical cutting edge and the axis of cylindrical cutter (or line parallel to axis) is called helix angle. Standard helical cutters have a helix angle of 20° to 30°.

Axial Rake Angle

It is the angle between the face of the tooth and axis of the cutter.

Nomenclature of side milling cutter is shown in Figure 6.9. It has cutting edges on periphery as well as on face. It has relief angles, clearance angle and rake angles on periphery cutting edges as well as on the face cutting edges.

Table 6.8: Recommended Angle for Milling Cutters

<table>
<thead>
<tr>
<th>Material</th>
<th>Rake Angles (Degrees)</th>
<th>Relief Angles (Degrees)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>HSS Tools</td>
<td>Stellite Tools</td>
</tr>
<tr>
<td>Cast iron (Soft)</td>
<td>10-15</td>
<td>6-8</td>
</tr>
<tr>
<td>Cast iron (Hard)</td>
<td>10</td>
<td>3-6</td>
</tr>
<tr>
<td>Mild steel</td>
<td>10-15</td>
<td>3-6</td>
</tr>
<tr>
<td>Al. Alloys</td>
<td>20-30</td>
<td>10-15</td>
</tr>
<tr>
<td>Brasses &amp; Bronzes</td>
<td>10-12</td>
<td>5</td>
</tr>
<tr>
<td>Mg. Alloys</td>
<td>20-30</td>
<td>15-20</td>
</tr>
</tbody>
</table>

6.11.2 Types of Milling Cutters

Milling cutters are rotating tools with number of cutting edges. The milling cutters may have either straight teeth, i.e., parallel to the axis of rotation or helical in shape. The helix may be right handed or left handed and this will decide the direction of rotation of the cutter for performing the cutting operation. Further, a milling cutter may be made of single piece of steel or having the cutting portion welded to a tough shank or having removable cutting teeth (bits) inserted in a solid body. The broad classification of milling cutters is according to the shape of teeth they carry, such as plain, inserted, formed or saw teeth, etc. This classification covers a large number of milling cutters.

Various milling cutters and their applications are tabulated in Table 6.9.

Table 6.9: Various Milling Cutters and Their Applications

<table>
<thead>
<tr>
<th>Sl. No.</th>
<th>Type of Cutter</th>
<th>Features</th>
<th>Applications</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Plain and face milling cutter (MILL)</td>
<td>Straight or helical teeth</td>
<td>For milling flat surface parallel to cutter axis</td>
</tr>
<tr>
<td>2</td>
<td>Side and face milling cutter</td>
<td>Similar to plain cutter, but has teeth on periphery as well as on one or both sides of the tool. Teeth may be straight, spiral or staggered.</td>
<td>Used for slotting, face milling and straddle milling</td>
</tr>
</tbody>
</table>
Elements of CAPP

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>End mill cutter</td>
<td>It has integral shaft for driving purpose and teeth on both periphery and ends. Flutes on cutter may be straight or helical. Shank may either be straight or tapered.</td>
</tr>
<tr>
<td>4</td>
<td>Fly cutter</td>
<td>It has a cylindrical body with a provision to mount one or more tool bits.</td>
</tr>
<tr>
<td>5</td>
<td>Slitting or saw cutter</td>
<td>Thin plain milling cutter with straight or staggered teeth</td>
</tr>
<tr>
<td>6</td>
<td>Hobs</td>
<td>Have cylindrical body with cutting teeth</td>
</tr>
</tbody>
</table>

Besides the above types of cutters, there are other cutters used for special purpose such as,

- Cam-relieve cutters
- T-slot cutters
- Angle milling cutters
- Form cutters etc.

**Plain Milling Cutters**

These are also called as cylindrical cutters and they have straight or helical teeth only on the periphery. The cutter with more width are called as “slab cutters”. They are used to produce flat surfaces parallel to the cutter axis. They can be further classified as:

(a) **Light Duty Plain Milling Cutter** has face width less than 20 mm. These straight teeth cutters are used for less metal removal.

(b) **Heavy Duty Plain Milling Cutter** has wider face with helical teeth (25 to 45° helix).

(c) **Helical Plain Milling Cutters** has a helix angle ranging from 45 to 60°. These are capable of providing better surface finish because of better shearing action.

**Side Milling Cutters**

These are similar to plain milling cutters but has teeth on the periphery as well as on the sides. They are used for slotting, straddle milling and face milling. The diameter of the cutter ranges from 50 to 200 mm, width ranging from 5 to 32 mm. They are classified as:

(a) **Plain milling cutters** have straight teeth on periphery and both sides of the tool.

(b) **Half Side Milling Cutters** have straight helical teeth on its circumference and on one of its sides only. It is mainly used for straddle milling operations.

(c) **Staggered Tooth Side Milling Cutters** have alternate teeth with opposite helix angle. They are used for deep slots and heavy duty milling.

**Metal Slitting Saw**

Metal slitting saw is similar to plain milling cutter; but have very smaller width. The slitting saws are available in the following types.
Machine and Tool Selection

Staggered teeth metal slitting saw have its teeth staggered at the periphery in a way similar to staggered teeth side milling cutter. The width of saw ranges from 5 mm to 8 mm.

Side teeth slitting cutter is used for making a deep slot or parting off wider materials.

Angle Milling Cutters

The angle milling cutters are used for cutting of chamfers, dovetails, V-notches, helical flutes and serrations.

They are made in the following types:

(a) **Single-angle Milling Cutters** have teeth on both the angled periphery and the vertical side face. These are made with angles of 30°, 45° and 60° to 85° in steps of 5°, with one side of the angle at 90° to the axis of the cutter. It is particularly suitable for producing straight tooth milling cutters and reamers.

(b) **Double-angle Cutters** have teeth on both angled surfaces, forming V-shaped cutting edge. Double-angle cutters usually have an included angles of 45°, 60° and 90°. They are particularly adapted for cutting spiral grooves on a blank.

End-milling Cutter

End milling cutter has teeth on the end face and periphery. It is used to machine both horizontal and vertical surfaces. Its diameter ranges from 3 to 50 mm and adapted for light cutting such as machinery profiles and narrow surfaces. The various types of end milling cutters are described below.

(a) **Shank Type Milling Cutters** are available with taper shank or straight shank. Taper Shanks confirm to the Morse taper and is directly secured in the spindle nose. The diameter of taper shank ranges from 10 to 50 mm. It is used for machining profiles and preparation of dies for press work and die-casting.

The straight Shanks are held in a spring collet. The diameter of straight shank end mill ranges from 2 to 50 mm and are extensively used in die-sinking and similar operations.

(b) **Shell End Milling Cutters** are made without shank and are provided with central hole for mounting on the machine arbor. They have teeth on end face and periphery. The shell end mills are larger and heavier. The diameter of cutter ranges from 50 to 160 mm and width from 32 to 63 mm. These cutters are used for wider slots and for facing up of flat surfaces.

(c) **T-Slot Milling Cutter** has teeth on the periphery and on both sides of the cutter. It is used for machining the base groove or portion of T-slot. The shank of the cutters may be reduced for part of its length so as to fit into the groove or slot already machined.

(d) **Slot Drill** or two-lipped end mill has two cutting edges and is similar in action to twist drill. It is used for machining a sunk keyway in a shaft and similar operations.

(e) **Woodruff key Slot Milling Cutter** is used for cutting Woodruff keyway and is provided with a shank. It has straight or staggered teeth over the periphery.

(f) **Fly Cutter** is a single point cutting tool and is attached to the tool arbor. It is used to reproduce contoured surface. It is adapted when the standard cutters are not available.
Form Milling Cutters have cutting edges of different profiles in order to generate required contours on the work piece. The use of different form cutters is described below.

(i) **Convex milling cutter** has teeth curved outwards on the circumference. It is used to produce concave bottom grooves or for making radius in an inside corner. Diameter of cutters ranges from 50 to 125 mm and width varies from 3.2 to 40 mm.

(ii) **Concave milling cutter** makes convex surfaces and rounded corners. Its diameter ranges from 56 to 110 mm and the radius of semi circle that can be milled varies from 1.5 to 20 mm.

(iii) **Gear cutters** have a profile which cuts the space between adjacent, involute gear teeth. Gear cutters are commonly available in a range of eight sizes for cutting gear teeth from 12 to 135.

Cutters for milling the flutes of drills, reamers and taps are called fluting cutters. The thread milling cutters are available to mill threads of different forms and sizes on work piece.

### 6.11.3 Milling Operations

#### Plain or Slab Milling

It is the process of producing flat surfaces on a horizontal milling machine. The surface milled is parallel to the axis of the cutter. As usual work is fed past the revolving cutter.

#### Face Milling

It is a process of producing flat surfaces on a vertical milling machine. The surface machined is at right angles to the axis of the cutter. Depth of cut is adjusted by rotating the cross-feed screw.

#### Side Milling

It is the process of producing a flat vertical surface on the side of workpiece. It makes use of side milling cutter.

#### End Milling

It is the process of producing flat surfaces which may be horizontal, vertical or at an angle with respect to table position. This operation makes use of end mill on a vertical milling operation to cut slots, grooves keyways, etc. However, plain and side milling cutters can also be employed.

#### Gang Milling

This is an operation of producing (machining) several surfaces at a time. Two or more cutters of same or different sizes are mounted on the same arbor and work is fed against them. Gang milling is widely used in repetitive work. Cutting speed is calculated on the basis of largest cutter. Profile of different shapes can be produced.
Straddle Milling
In side milling cutters mounted on the same arbor, can machine two vertical flat sides of a workpiece at the same time, the operation is called straddle milling. The pair of cutters are spaced accurately using spacing collars. This operation is very common in industries to produce parallel surfaces.

Form Milling
It is the process of milling irregular surfaces. The operation is carried out on a horizontal milling machine by using a form milling cutter. This cutter will have teeth corresponding to the profile of the surface to be produced.

6.12 CUTTING SPEED AND FEED IN MILLING OPERATION

Cutting Speed
It is the distance traveled per minute by the cutting edge of the cutter. It is measured at the circumference of the cutter and is expressed in mts/min or feet per minute.

\[ \text{Cutting Speed} = \frac{\pi DN}{1000} \text{ m/min} \]

Table 6.10 : Recommended, Average Values in Metres per Minute, of Cutting Speeds in Milling Operation

<table>
<thead>
<tr>
<th>Material to be Machined</th>
<th>Tool Material</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>HSS</td>
</tr>
<tr>
<td>Alluminium</td>
<td>180-240</td>
</tr>
<tr>
<td>Hard Al. Alloys</td>
<td>90-120</td>
</tr>
</tbody>
</table>
Elements of CAPP

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>45-55</td>
<td>70</td>
<td>140</td>
<td>45</td>
<td>30-36</td>
<td>45</td>
<td>90</td>
<td>18-24</td>
<td>10-12</td>
<td>15-20</td>
<td>18-24</td>
<td>10-12</td>
</tr>
</tbody>
</table>

Feed

It represents the table travel in any direction, measured in millimeters. It can be expressed in three ways.

(a) Feed per minute → mm/min

(b) Feed per tooth → The table travel in mm during the period when the cutter revolves through an angle corresponding to the distance between the cutting edges of two adjacent teeth.

(c) Feed per revolution → mm/rev.

6.12.1 Formulae to be Remembered

(a) Cutting speed \( V = \frac{\pi DN}{1000} \) m/min.

(b) Approach distance \( A = \sqrt{\left(\frac{D}{2}\right)^2 - \left(\frac{D - d}{2}\right)^2} = \sqrt{d(D - d)} \)

\( D = \) Diameter of milling cutter, mm

\( d = \) Depth of cut, mm

(c) Time for one pass \( (t) = \frac{1 + 2 \times A}{fn} \) minutes

\( l = \) Length of the milling cut, mm

\( A = \) Approach distance

\( f = \) Feed per tooth, mm

\( z = \) Number of teeth, in the milling cutter.

Example 6.7

Calculate the spindle speed \( (N) \) if a copper job is to be milled by

(a) stellite cutter

(b) cemented carbide cutter.

The diameter of the cutter being limited to 150 mm.

Solution

\( D = 150 \) mm

(a) For stellite cutter

\( V_c = 65 \) m/min (from Table 6.10)
We know \( V_c = \frac{\pi DN}{1000} \text{ m/min} \)

\[
N = \frac{V_c \times 1000}{\pi D} = \frac{65 \times 1000}{\pi \times 150} = 138 \text{ rpm}
\]

(b) For cemented carbide cutter

\[
D = 150 \text{ mm}
\]

\[
V_c = 140 \text{ (from Table 6.10)}
\]

\[
N = \frac{V_c \times 1000}{\pi D} = \frac{140 \times 1000}{\pi \times 150} = 297 \text{ rpm}
\]

Example 6.8

Find the maximum allowable size of the milling cutter if it is to rotate at 65 rpm while the cutting speed is 20 m/min.

Solution

\[
V_c = 20 \text{ m/min}
\]

\[
N = 65 \text{ rpm}
\]

We know \( V_c = \frac{\pi DN}{1000} \text{ m/min} \)

\[
D = \frac{V_c \times 1000}{\pi N} = \frac{20 \times 1000}{\pi \times 65} = 98 \text{ mm}
\]

Example 6.9

An HSS slab mill of 100 mm diameter and 150 mm width is used on a Horizontal milling machine to mill C50 steel. The milling cutter has 8 teeth. Calculate the machining time assuming that the entire stock can be removed in one depth of 2 mm. The cutting speed \( V_c = 20 \text{ m/min} \) and feed rate \( f = 0.13 \text{ mm/tooth} \). The cut length = 150 mm.

Solution

Given

\[
Z = 8
\]

\[
D = 100 \text{ mm}
\]

\[
d = 2 \text{ mm}
\]

\[
V_c = 20 \text{ m/min}
\]

\[
f = 0.13 \text{ mm/tooth}
\]

Approach distance \( A = \sqrt{d(D - d)} = \sqrt{2(100 - 2)} = 14 \text{ mm} \)

Spindle speed \( N = \frac{1000 \times 20}{\pi \times 100} = 63.66 \pm 65 \text{ rev/min} \)

Time for machining \( t = \frac{1 + 2A}{fnz} = \frac{150 + 2 \times 14}{0.13 \times 8 \times 65} = 2.633 \text{ min} \).

### 6.13 TOOLS USED IN DRILLING

Drilling is an operation through which holes are produced in a solid metal by means of a revolving tool called drill.

#### 6.13.1 Classification of Drills

Drills are manufactured in several different forms and sizes. The commonly used drills can be classified in many ways, as follows:
According to the Type of Shank They Carry
(a) Parallel shank
(b) Taper shank

According to the Type of Flutes
(a) Flat or spade drills (parallel longitudinal flutes)
(b) Twist drills (spiral/helical flutes)

According to Length
(a) Short series drills
(b) Stub series drills
(c) Long series drills

According to Applications
(a) Core drills
(b) Drills for long hole drilling
(c) Centre drills
(d) Masonary drills

According to the Tool Material
(a) High speed steel drills
(b) Carbide Tipped Drills

6.13.2 Twist Drill Nomenclature
Drill nomenclature comprises the various parts and important geometric parameters of cutting point. They are shown in Figure 6.11 and defined below:

Shank
The shank is the part of drill which is held in machine spindle and driven by it.

Tang
Flattened end of shank, intended to fit into a slot in the drill holder.

Neck
Reduced p
Then, there are large number of special drills manufactured against order to meet specific requirements. Such drills can be classed as special drills.

**Body**

The fluted portion of a drill.

**Flutes**

Flutes are helical grooves formed in the body of drill.

**Web**

The central portion of the body which separates the flutes and runs through entire length of drill.

**Cutting Lip or Edge**

The edge formed by intersection of flank and face, and correspond to the cutting edge of a single point tool.

**Land**

The cylindrical ground surface on the leading edges of drill flutes.

**Body Clearance**

The diameter over the surface of the body which is situated behind the land.

**Margin**

Narrow surface along the groove which keeps the drill aligned.

**Heel**

The edges formed by the intersection of flute surface and body clearance.

### 6.13.3 Important Angles of a Drill

Many different angles, as shown in Figure 6.11, are provided on a drill so as to ensure an efficient metal cutting. The main angles are the following.

**Rake Angle**

It is also known as the helix angle. It is the angle formed between a plane containing the drill axis and the leading edge of the land. It can have a positive, negative or zero value. For a right hand flute its value is positive, for a left hand flutes negative and for parallel fluted drill its value is 48°. However, 16° to 32° range is quite common for normal materials. Higher values of the helix angle are suitable for softer materials and lower values for harder materials. The power or
Elements of CAPP

the torque required to rotate the drill is greatly influenced by this angle. Larger the value of this angle, lesser will be the torque required and vice-versa.

Point Angle

It is also known as cutting angle. Its most commonly used value for a large variety of materials is 118°. However, it varies from 80° to 140°. Smaller point angle is favoured for brittle materials and the larger one for harder and tougher materials. It is the angle included between the two opposite lips of a drill, measured in a plane containing the axis of the drill and both the lips.

Lip Clearance Angle

The angle formed between the flank and a plane normal to the drill axis, measured at the periphery of the drill, is called lip clearance angle. Its value varies from 8° to 15° for most of the drills, but 12° angle is the most common. This angle is formed as a result of grinding the relief adjacent to the cutting edges to enable easy entry of the drill.

Chisel Edge Angle

When a drill is viewed from its end, there appears to be an obtuse angle formed between the lip and the chisel edge. This angle is called the chisel edge angle. It determines the clearance on the cutting lip near the chisel edge. The greater this angle the larger will be the clearance. Normally this angle varies between 120° and 135°, although on some smaller drills it may be as large as 145°.

The common range of values of these four principal drill angles are given in Table 6.11.

<table>
<thead>
<tr>
<th>Material to be Drilled</th>
<th>Included Cutting Angle or Point Angle</th>
<th>Lip Clearance Angle</th>
<th>Helix Angle or Rake Angle</th>
<th>Chisel Edge Angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alluminium (pure)</td>
<td>80°-120°</td>
<td>8°-12°</td>
<td>24°-32°</td>
<td>120°-135°</td>
</tr>
<tr>
<td>Cast iron (soft)</td>
<td>118°</td>
<td>8°-12°</td>
<td>24°-32°</td>
<td>120°-135°</td>
</tr>
<tr>
<td>Cast iron (hard)</td>
<td>118°</td>
<td>8°-15°</td>
<td>24°-32°</td>
<td>120°-135°</td>
</tr>
<tr>
<td>Brass</td>
<td>118°</td>
<td>8°-15°</td>
<td>0°-18°</td>
<td>120°-135°</td>
</tr>
<tr>
<td>Copper</td>
<td>120-140°</td>
<td>8°-15°</td>
<td>24°-32°</td>
<td>120°-135°</td>
</tr>
<tr>
<td>Steel</td>
<td>118°</td>
<td>8°-12°</td>
<td>24°-32°</td>
<td>120°-135°</td>
</tr>
<tr>
<td>Stainless steel</td>
<td>120-140°</td>
<td>10°-12°</td>
<td>24°-32°</td>
<td>120°-135°</td>
</tr>
<tr>
<td>Al. Alloys</td>
<td>140°</td>
<td>8°-12°</td>
<td>24°-32°</td>
<td>120°-135°</td>
</tr>
<tr>
<td>Plastics and Hard rubber</td>
<td>80°</td>
<td>8°-12°</td>
<td>24°-32°</td>
<td>120°-135°</td>
</tr>
<tr>
<td>Pure nickel</td>
<td>118°</td>
<td>10°-12°</td>
<td>24°-32°</td>
<td>120°-135°</td>
</tr>
</tbody>
</table>

6.13.4 Drill Specifications

According to Indian standards the drills are specified by their diameters, series they belong to, the material they are made of and the I.S. number. These data are mainly based on the material for which the drill is to be normally used. They are made in three types:

(a) Type N – for normal low carbon steel,
(b) Type H – for hard materials, and
(c) Type S – for soft and tough materials.

Example 6.10

A twist drill specified as ‘9.50 IS : 5101 HS’, means a twist drill of 9.50 mm dia, confirming to IS : 5101, made from high speed steel.
It should be noted that unless otherwise mentioned in the tool designation the type should be taken as ‘N’ and the point angle as 118°.

6.14 CUTTING SPEED, FEED AND DEPTH OF CUT IN DRILLING OPERATION

The cutting speed and feed in drilling, as in case of other machines, depends upon many factors like material to be cut, material of the tool, type of finish required, type of coolant used, capacity of machine and the tool life, etc. The amount of feed per revolution, usually varies between 0.05 mm to 0.38 mm for drills up to 25 mm dia.

\[
\text{Cutting Speed} = \frac{\pi DN}{1000} \text{ m/min}
\]

where \( d \) = Drill dia in mm, and \( N \) = Spindle speed in rpm.

Feed

It is the distance a drill moves, parallel to its axis, into the work in each revolution of the spindle. It is expressed in mm per revolution. If the total distance moved by the drill into the work, parallel to its axis, in one minute is considered, it can be expressed as feed in mm per minute. Now, if \( N \) be the No. of revolutions made per minute by the drill, then:

\[
\text{Feed in mm/min} = \frac{\text{feed in mm}}{\text{rev}} \times N
\]

The following factors govern the amount of feed to be provided:

(a) Workpiece and drill material
(b) Depth of drilling
(c) Range of available feeds
(d) Rigidity of the machine
(e) Degree of surface finish required
(f) Horse power of the motor
(g) Drill size and speed

Depth of Cut

In drilling operation the depth of cut is measured at right angles to the axis of the drill, i.e. the direction of feed, and is numerically equal to one-half of the diameter of the drill. It can be expressed as:

\[
\text{Depth of cut} = \frac{\text{Drill dia}}{2} \text{ mm}
\]

Table 6.12 : Average Cutting Speed for HSS Drills

<table>
<thead>
<tr>
<th>Material</th>
<th>Cutting Speed in mpm</th>
<th>Coolant</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cast Iron (soft)</td>
<td>24-40</td>
<td>Dry</td>
</tr>
<tr>
<td>Cast Iron (hard)</td>
<td>16-27</td>
<td>– do –</td>
</tr>
<tr>
<td>Mild Steel</td>
<td>24-45</td>
<td>Soluble oil</td>
</tr>
<tr>
<td>Medium Carbon Steel</td>
<td>12-30</td>
<td>– do –</td>
</tr>
<tr>
<td>Stainless Steel</td>
<td>12-23</td>
<td>– do –</td>
</tr>
<tr>
<td>Brass and Bronze</td>
<td>45-90</td>
<td>Soluble oil or dry</td>
</tr>
<tr>
<td>Copper</td>
<td>30-45</td>
<td>Soluble oil</td>
</tr>
<tr>
<td>Aluminium</td>
<td>90 and up</td>
<td>– do –</td>
</tr>
</tbody>
</table>

Table 6.13 : Recommended Cutting Parameters for Carbide Tipped Twist Drills
### Elements of CAPP

<table>
<thead>
<tr>
<th>Workpiece Material</th>
<th>U.T.S. Hardness of Work Material</th>
<th>Drill Dia. and Lip Angle</th>
<th>Cutting Speed metres/min</th>
<th>Feed mm/rev</th>
</tr>
</thead>
<tbody>
<tr>
<td>Al. Alloys</td>
<td>Hardness HB &gt; 80</td>
<td>3 to 40 mm (115°/120°)</td>
<td>100-140</td>
<td>0.06-0.2</td>
</tr>
<tr>
<td>Brass</td>
<td>-</td>
<td>3 to 40 mm (115°/120°)</td>
<td>80-120</td>
<td>0.05-0.3</td>
</tr>
<tr>
<td>Bronze (cast)</td>
<td>-</td>
<td>3 to 40 mm (115°/120°)</td>
<td>50-80</td>
<td>0.06-0.15</td>
</tr>
<tr>
<td>Cast iron (Grey)</td>
<td>Hardness HB upto 250</td>
<td>3 to 40 mm (115°/120°)</td>
<td>40-80</td>
<td>0.04-0.25</td>
</tr>
<tr>
<td>Cast iron (chilled)</td>
<td>Hardness Sh. 65-85</td>
<td>5 to 12 mm (125°/140°)</td>
<td>5-12</td>
<td>0.01-0.05</td>
</tr>
<tr>
<td>Hardened steel</td>
<td>Hardness &gt; 5 HRC</td>
<td>3 to 20 mm (125°/140°)</td>
<td>8-12</td>
<td>0.01-0.03</td>
</tr>
<tr>
<td>Steel castings</td>
<td>U.T.S. &gt; 70 kg/mm²</td>
<td>3 to 40 mm (115°/120°)</td>
<td>25-40</td>
<td>0.02-0.15</td>
</tr>
<tr>
<td>Carbon steel</td>
<td>U.T.S. 85-120 kg/mm²</td>
<td>3 to 40 mm (115°/120°)</td>
<td>25-60</td>
<td>0.02-0.15</td>
</tr>
<tr>
<td>(Annealed)</td>
<td>-</td>
<td>3 to 20 mm (125°/140°)</td>
<td>10-28</td>
<td>0.02-0.04</td>
</tr>
<tr>
<td>- do -</td>
<td>120-180 kg/mm²</td>
<td>3 to 40 mm (115°/120°)</td>
<td>50-80</td>
<td>0.03-0.12</td>
</tr>
</tbody>
</table>

### 6.14.1 Formulae to be Remembered

(a) \[ V = \frac{\pi DN}{1000} \]

(b) Total length of tool travel \( L = l + A + 2 \)

\( l \) = Length of the hole, mm

\( A \) = Breakthrough distance

\[ A = \frac{D}{2 \tan \alpha} \] [usually \( \alpha = 59° \)]

(c) Time for drilling the hole \( t = \frac{L}{fN} \)

(d) \[ \text{MRR} = \frac{\pi D^2 f N}{4} \] mm³/min; \( D \) = Dia of the hole.

### Example 6.11

Calculate the cutting speed if a drill of 0.04 m dia is operated at a speed of 250 rpm.

**Solution**

\( D = 0.04 \) m

\( N = 250 \) rpm
We know \[ V_c = \pi DN \]
\[ = \pi \times 0.040 \times 250 = 31.42 \text{ m/min} \]

**Example 6.12**

Calculate the feed in m/min if a drill makes 200 revolutions per minute and penetrates to a distance of 3 mm per revolution.

**Solution**

\[ N = 200 \text{ rpm} \]

Tool feed/rev. = 3 m/rev.

\[ \therefore \text{Feed in m/min} = \frac{\text{feed/rev} \times N}{1000} \]
\[ = \frac{3 \times 200}{1000} = 0.6 \text{ m/min} \]

**Example 6.13**

A hole of 40 mm diameter and 50 mm depth is to be drilled in a mild steel component. The cutting speed can be taken as 65 m/min and the feed rate as 0.25 mm/rev. Calculate the machining time and the material removal rate.

**Solution**

Given \[ V = 65 \text{ m/min} \]

\[ f = 0.25 \text{ mm/rev} \]

\[ D = 40 \text{ mm} \]

\[ L = 50 \text{ mm} \]

Spindle speed, \[ N = \frac{1000 \times 65}{\pi \times 40} = 517.25 \text{ rev./min} \]

Breakthrough distance, \[ A = \frac{40}{2 \tan 59} = 12.02 \text{ mm} \]

Total length of drill travel, \[ L = 50 + 12 + 3 = 65 \text{ mm} \]

Time for drilling the hole \[ = \frac{65}{0.25 \times 520} = 0.50 \text{ minutes} \]

The material removal rate, \[ MRR = \frac{\pi \times 40^2 \times 0.25 \times 520}{4} \]
\[ = 163362.82 \text{ mm}^3/\text{min} \]
\[ = 163.363 \text{ cm}^3/\text{min} \]

**Example 6.14**

Calculate the drilling speed if a 50 mm dia hole of 60 mm depth is to be drilled in a M.S. component. The feed rate is 0.2652 and MRR is 200 cm$^3$/min. Also calculate the machining time.

**Solution**

Given \[ MRR = 200 \text{ cm}^3/\text{min} \]
Elements of CAPP

\[ V = \frac{\pi DN}{1000} = \frac{\pi \times 50 \times 384}{1000} = 60.3 \text{ m/min} \]

Breakthrough distance, \( A = \frac{D}{3.3286} = \frac{50}{3.3286} = 15 \text{ mm} \)

Total length of drill travel, \( L = 1 + A + 2 \)

\[ = 60 + 15 + 2 = 77 \text{ mm} \]

Time for drilling hole \( = \frac{L}{fN} = \frac{77}{0.2652 \times 384} = 0.7561 \text{ min} \)

<table>
<thead>
<tr>
<th>Workpiece Material</th>
<th>Tool Material</th>
<th>Range of Cutting Speed (m/min)</th>
<th>Range of Feed (mm/rev.)</th>
<th>Tool Geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Cast Irons</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Grey C.I.</td>
<td>Cem. Carbide</td>
<td>60 – 150</td>
<td>0.1 – 1.0</td>
<td>– ve rake</td>
</tr>
<tr>
<td></td>
<td>Coated carbide</td>
<td>100 – 300</td>
<td>0.1 – 1.0</td>
<td>(+ ve rake for fine finishing)</td>
</tr>
<tr>
<td></td>
<td>Oxide ceramics</td>
<td>350 – 700</td>
<td>0.1 – 0.4</td>
<td></td>
</tr>
<tr>
<td>Whiteheart C.I.</td>
<td>CBN</td>
<td>100 – 80</td>
<td>0.1 – 0.4</td>
<td>- ve rake</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ductile C.I.</td>
<td>Cem. Carbide</td>
<td>50 – 120</td>
<td>0.1 – 1.0</td>
<td>- ve rake</td>
</tr>
<tr>
<td>(Malleable C.I. &amp; S.G. Iron)</td>
<td>Coated carbide</td>
<td>60 – 250</td>
<td>0.1 – 1.0</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Oxide ceramics</td>
<td>300 – 600</td>
<td>0.1 – 0.4</td>
<td></td>
</tr>
<tr>
<td><strong>Plain Carbon</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Low Carbon</td>
<td>Cem. Carbide</td>
<td>60 – 120</td>
<td>0.1 – 1.2</td>
<td>0 or + ve rake</td>
</tr>
<tr>
<td>Coated carbide</td>
<td>90 – 400</td>
<td>0.1 – 0.8</td>
<td>Chip groove</td>
<td></td>
</tr>
<tr>
<td>Med carbon &amp; High carbon</td>
<td>Coated carbide</td>
<td>80 – 300</td>
<td>0.1 – 0.8</td>
<td>0 or + ve rake</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Chip groove</td>
<td></td>
</tr>
<tr>
<td><strong>Alloy Steels</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Unhardened</td>
<td>Coated carbide</td>
<td>100 – 300</td>
<td>0.1 – 0.8</td>
<td>+ ve rake chip groove/-ve rake depending on strength of the material</td>
</tr>
<tr>
<td>Toughened &lt; 35 HRC</td>
<td>Coated carbide</td>
<td>80 – 250</td>
<td>0.1 – 0.8</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Cermet</td>
<td>80 – 180</td>
<td>0.1 – 0.4</td>
<td></td>
</tr>
<tr>
<td>Hardness &gt; 50 HRC</td>
<td>CBN</td>
<td>~ 90</td>
<td>0.1 – 0.3</td>
<td></td>
</tr>
</tbody>
</table>
### Stainless Steel

<table>
<thead>
<tr>
<th>Type</th>
<th>Material</th>
<th>Range</th>
<th>Rake Angle</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Austenitic</td>
<td>Coated carbide</td>
<td>75 – 220</td>
<td>0.1 – 0.8</td>
<td>+ve rake</td>
</tr>
<tr>
<td></td>
<td>Cermet</td>
<td>80 – 180</td>
<td>0.1 – 0.4</td>
<td></td>
</tr>
<tr>
<td>Ferritic and Martenstic</td>
<td>Coated carbide</td>
<td>45 – 150</td>
<td>0.1 – 0.5</td>
<td>+ve rake</td>
</tr>
<tr>
<td></td>
<td>Cermet</td>
<td>100 – 140</td>
<td>0.1 – 0.4</td>
<td></td>
</tr>
<tr>
<td>Tool Steels</td>
<td>Coated carbide</td>
<td>75 – 220</td>
<td>0.1 – 0.5</td>
<td>-ve rake ( 0° rake for oil and air hardening steel)</td>
</tr>
<tr>
<td>150-250 BHN</td>
<td>CBN</td>
<td>80 – 180</td>
<td>0.1 – 0.4</td>
<td></td>
</tr>
<tr>
<td>50-65 HRC</td>
<td></td>
<td>45 – 150</td>
<td>0.1 – 0.5</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>100 – 140</td>
<td>0.1 – 0.4</td>
<td></td>
</tr>
<tr>
<td>Copper and its alloys</td>
<td>Copper</td>
<td>75 – 220</td>
<td>0.1 – 0.5</td>
<td>+ve rake chip groove</td>
</tr>
<tr>
<td></td>
<td>Brass &amp; Bronze</td>
<td>80 – 180</td>
<td>0.1 – 0.4</td>
<td>+ve rake Chip groove</td>
</tr>
<tr>
<td></td>
<td></td>
<td>45 – 150</td>
<td>0.1 – 0.5</td>
<td></td>
</tr>
<tr>
<td>Aluminium and its alloys</td>
<td>Wrought alloys</td>
<td>75 – 220</td>
<td>0.1 – 0.5</td>
<td>High + ve rake of app. 25° 0° rake</td>
</tr>
<tr>
<td></td>
<td>Cast alloys</td>
<td>80 – 180</td>
<td>0.1 – 0.4</td>
<td></td>
</tr>
<tr>
<td>(Si &gt; 14%)</td>
<td></td>
<td>45 – 150</td>
<td>0.1 – 0.5</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>100 – 140</td>
<td>0.1 – 0.4</td>
<td></td>
</tr>
<tr>
<td>Nickle and its alloys</td>
<td>Inconel and Hestalloy</td>
<td>75 – 220</td>
<td>0.1 – 0.5</td>
<td>+ve rake chip groove</td>
</tr>
<tr>
<td></td>
<td>125-250 BHN</td>
<td>80 – 180</td>
<td>0.1 – 0.4</td>
<td></td>
</tr>
<tr>
<td></td>
<td>200-400 BHN</td>
<td>45 – 150</td>
<td>0.1 – 0.5</td>
<td></td>
</tr>
<tr>
<td>Titanium and its alloys</td>
<td>250-350 BHN</td>
<td>75 – 150</td>
<td>0.1 – 0.5</td>
<td>+ve rake chip groove</td>
</tr>
<tr>
<td></td>
<td>350-400 BHN</td>
<td>100 – 140</td>
<td>0.1 – 0.4</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>100 – 140</td>
<td>0.1 – 0.4</td>
<td></td>
</tr>
</tbody>
</table>

### SAQ 1

(a) Why is it important to select a right tool for a given process?

(b) Name the various factors that influence the selection of a tool?

(c) How are the cast cobalt alloy tools different from the cemented carbide tools?

(d) What are the desirable characteristics of a cutting tool material? Explain how these are satisfied in the case of high speed steel tools.

(e) Explain the advantages of coated carbides tools over the uncoated carbide tools.

(f) What are the requirements to be met when machining with ceramic tools?

(g) Discuss the suitability of polycrystalline diamond (PCD) as a tool material.

(h) Briefly explain how the properties of the work material influence the selection of tools.

(i) Explain functions of various tool angles and the nomenclature of a twist drill.
(j) Describe a standard tool specification in turning.
(k) How are the tools classified in turning?
(l) Discuss the various tool angles of a standard milling cutter.
(m) How are drills classified?
(n) Explain drill specification.
(o) How does the cutting conditions influence the selection of a tool? Explain briefly.

**Exercises**

(a) For a production turning operation, past records have shown that the tool life varies with the cutting speed as follows:

<table>
<thead>
<tr>
<th>Cutting speed, $V$, m/s</th>
<th>Tool life, $T$, min</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.08</td>
<td>110</td>
</tr>
<tr>
<td>2.54</td>
<td>37</td>
</tr>
</tbody>
</table>

Estimate the tool life for this operation at a speed of 2.3 m/s. Outline all the assumptions used to obtain this estimate.

(b) In a normal turning operation the tool life varies with the cutting speed as shown in the following table:

<table>
<thead>
<tr>
<th>Cutting speed, $V$, m/s</th>
<th>Tool life, $T$, min</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>30</td>
</tr>
<tr>
<td>70</td>
<td>2</td>
</tr>
</tbody>
</table>

Estimate the tool life for this operation at a speed of 60 m/min.

(c) A carbide cutting tool has tool life exponent $n = 0.27$. It gives a tool life of 60 minutes while machining a mild steel workpiece at a cutting speed of 120 m/min. Compute the tool life if it is to be cut at a 20% higher cutting speed. Also what is the cutting speed if the tool life is to be doubled?

(d) In a metal cutting experimentation the tool life was found to vary with the cutting speed in the following manner.

<table>
<thead>
<tr>
<th>Cutting speed, $V$, m/min</th>
<th>Tool life, $T$, min</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>120</td>
</tr>
<tr>
<td>130</td>
<td>50</td>
</tr>
</tbody>
</table>

Derive the Taylor’s tool life equation for this operation and estimate the tool life at a speed of 2.5 m/s. Also estimate the cutting speed for a tool life of 80 minutes.

**6.15 SUMMARY**

The quality of a product largely depends on the type of machine, the type of process, machining conditions and the right selection of tools. The current trend in manufacturing is to adopt more and more of automation of the fixed and flexible type. Hence, with a
machine tool of a given level of automation the scope for improving productivity lies more and more in the selection and application of cutting tools.

Some of the factors influencing the selection of cutting tool are properties of tool materials, properties of work materials, the type of process, the accuracy and surface finish required, the cutting conditions, tool wear and tool life. The properties of tool material include Hardness, strength, Stiffness and Resilience, Wear resistance, Low coefficient of friction, Thermal Conductivity and Specific Heat. Some important tool materials used are High Carbon Steels, High Speed Steels, Cast Cobalt (Stellite) Tools, Carbides, Coated Carbides, Ceramics, Diamonds, Cubic Boron Nitride(CBN), Polycrystalline Diamond.

The important properties influencing the selection of cutting tools are : Hardness, Tensile strength, Modulus of elasticity, Chemical composition, Micro structure, Strain hardenability and thermal conductivity. The type of the manufacturing process depends upon the type of the tools used. Basically the tools can be classified into two types generating tools and forming tools. At the time of process planning the tools are selected depending on the accuracy and surface finish required. This is because of the fact that different operations and different tools would produce different accuracy levels and surfaces.

The tool life equation as proposed by Taylor is $VT^n = C$

where $n$ = Constant for tool material,
$V$ = Cutting speed, m/min, and
$T$ = Tool life in min.

As this formula doesn’t take into account all the effecting parameters, many researchers have extended this formula.

$TQ^B = C$

$$VT^{n-f} = C$$

where $H =$ Specific heat $\times$ thermal conductivity,
$\theta =$ Tool temperature,
$A =$ Area of cut,
$u_s =$ Specific cutting energy/unit cutting force, and
$C$ and $x$ are constants.

$VT^n f^ 1, d^2 = C$

where $f =$ Feed, mm/rev.,
$d =$ Depth of cut, and
$C =$ Constant.

All the cutting tools used in lathe are single point tools. The single point cutting tool nomenclature comprises the various parts of tool and systematic arrangement of tools angles. The tool angle are normally specified in the sequence of Back rake, Side rake, End relief, Side relief, End cutting edge, Side cutting edge angles and Nose radius. The various operations performed on a lathe machine are centering, turning, facing, thread cutting, knurling, parting off, forming, drilling, boring, and reaming. Similarly this unit discusses about the other type of tools used in milling and drilling.
**BIBLIOGRAPHY**

Milton C. Shaw, *Metal Cutting Principles*.
Prakash H Joshi, *Cutting Tools*.