24.1 Introduction

- In addition to parts with various external or internal round profiles, machining operations can produce many other parts with more complex shapes.

![Figure 24.1](image)

Figure 24.1 Typical parts and shapes that can be produced with the machining processes described in this chapter.

24.2 Milling and milling machines:

- Milling: a process in which a rotating multi-tooth cutter removes material while traveling along various axes with respect to the workpiece.
- With the use of milling cutter, a multi-tooth tool that produces a number of chips in one revolution (Fig. 24.2).

![Figure 24.2](image)

Figure 24.2 Some basic types of milling cutters and milling operations. (a) Peripheral milling. (b) Face milling. (c) End milling. (d) Ball-end mill with indexable coated-carbide inserts machining a cavity in a die block. (e) Milling a sculptured surface with an end mill, using a five-axis numerical control machine.
24.2.1 Peripheral milling

- Peripheral milling also called plain milling; the axis of cutter rotation is parallel to the workpiece surface, (fig 24.2a).
- The cutter body, which generally is made of HSS, has a number of teeth along its circumferences; each tooth acts like a single-point cutting tool.
- When the cutter is longer than the width of the cut, the process is called slab milling.
- Cutters for peripheral milling may have straight or helical teeth 24.2a, resulting in orthogonal or oblique cutting action, respectively.
- Helical teeth generally are preferred over straight teeth because the tooth is partially engaged with the workpiece as it rotate, consequently, the cutting force and torque on the cutter are lower, resulting in a smoother operation and reduce chatter.

Conventional Milling (up milling)

- Maximum chip thickness is at the end of the cut as the tooth leaves the workpiece surface.
- Advantage: tooth engagement is not a function of workpiece surface characteristics, and contamination or scale on the surface does not affect tool life; cutting process is smooth
- Disadvantages: tendency for the tool to chatter; and the workpiece has a tendency to be pulled upward, necessitating proper clamping.

Climb Milling (Down milling):

- Cutting starts at the surface of the workpiece, where the chip is at its thickest.
- The advantages: downward component of cutting forces hold workpiece in place.
- Because of the resulting high impact forces when the teeth engage the workpiece, this operation must have a rigid setup, and backlash must be eliminated in the table feed mech.
- Not suitable for machining workpiece having surface scale, such as hot-worked metals, forgings, and casting.
- The scale is hard and abrasive and causes excessive wear and damage to the cutter teeth, thus shortening tool life.

![Figure 24.3](image)

Figure 24.3  (a) Schematic illustration of conventional milling and climb milling.  (b) lab-milling operation showing depth-of-cut, \(d\); feed per tooth, \(f\); chip depth-of-cut, \(t_c\); and workpiece speed, \(v\).  (c) Schematic illustration of cutter travel distance, \(l_c\), to reach full depth-of-cut.

Milling Parameters

Cutting speed \(V = \pi DN\)

- \(V\) = cutting speed in peripheral milling is the surface speed of the cutter
- \(D\) = cutter diameter
- \(N\) = rotational speed of the cutter
For a straight-tooth cutter, the approximate undeformed chip thickness, \( t_c \), is given by:

\[
t_c = 2f \sqrt{\frac{d}{D}}
\]

- \( f \) = feed per tooth of the cutter, measured along the workpiece surface, mm/tooth or in./tooth
- \( d \) = depth of cut
- \( D \) = Cutter diameter mm or in.

**Feed per tooth** is given by:

\[
f = \frac{v}{Nn}
\]

- \( v \) = linear speed (feed rate) of workpiece
- \( n \) = Number of teeth on the periphery of cutter
- \( N \) = Rotational speed of the milling cutter, rpm

**Cutting time** is given by:

\[
t = \frac{(l + lc)}{v}
\]

- \( l \) = length of workpiece
- \( lc \) = extent of the cutter’s 1st contact with workpiece

**Material Removal Rate** (based on the assumption that \( lc << l \)) is given by:

\[
MRR = \frac{lwd}{t} = wdv
\]

- \( w \) = width of cut

---

**TABLE 24.1**

**Summary of Peripheral Milling Parameters and Formulas**

- \( N \) = Rotational speed of the milling cutter, rpm
- \( F \) = Feed, mm/tooth or in./tooth
- \( D \) = Cutter diameter, mm or in.
- \( n \) = Number of teeth on cutter
- \( v \) = Linear speed of the workpiece or feed rate, mm/min or in./min
- \( V \) = Surface speed of cutter, m/min or ft/min
- \( f \) = Feed per tooth, mm/tooth or in./tooth
- \( l \) = Length of cut, mm or in.
- \( t \) = Cutting time, s or min
- \( MRR \) = \( mm^3/\text{min} \) or \( in.^3/\text{min} \)
- Torque = \( N \times m \) or \( lb \times \text{ft} \)
- Power = kW or hp
24.2.2 Face Milling

- In face milling the cutter is mounted on a spindle having an axis of rotation is perpendicular to the workpiece surface. Fig 24.2b.

- Because of the relative motion between the cutter teeth and the workpiece, face milling leave feed marks on the machined surface (Fig. 24.6).
- Surface roughness of workpiece depends on insert corner geometry and feed per tooth.

Figure 24.5 A face-milling cutter with indexable inserts.

Figure 24.6 Schematic illustration of the effect of insert shape on feed marks on a face-milled surface: (a) small corner radius, (b) corner flat on insert, and (c) wiper, consisting of small radius followed by a large radius which leaves smoother feed marks. (d) Feed marks due to various insert shapes.
Effect of Lead Angle:

- Terminology for a face milling cutter are shown in Fig. 24.7.
- Lead angle of insert in face milling has a direct influence on the undeformed chip thickness.
  - As the lead angle increases, the undeformed chip thickness decreases, and the length of contact increases.
- Lead angles also influences the forces in milling, as the lead angle decreases, there is a smaller vertical force component (axial force on the cutter spindle).
- Lead angles range from 0° to 45°
- Note that the x-sectional area of the undeformed chip remains constant
- Ratio of cutter diameter, D, to the width, w, of cut should be no less than 3:2.

Figure 24.7 Terminology for a face-milling cutter.

Figure 24.8 The effect of the lead angle on the undeformed chip thickness in face milling. Note that as the lead angle increases, the chip thickness decreases, but the length of contact (i.e., chip width) increases. The edges of the insert must be sufficiently large to accommodate the contact length increase.
Cutter and Insert Position in Face Milling:

- The relationships of the cutter diameter to insert angles and their position relative to the surface to be milled is important in that it will determine the angle at which an insert enters and exits the workpiece.
- The same insert may engage the workpiece at different angles, depending on the relative positions of the cutter and the workpiece width.
- Note in fig. 24.9a that the tip of the insert makes the first contact, so there is a possibility for the cutting edge to chip off.
- Note that in fig. 24.9b the first contacts at an angle and away from the tip of the insert, therefore there is lower tendency for the insert to fail, because the forces on the insert vary more slowly.

![Figure 24.9](image)

**Figure 24.9** (a) Relative position of the cutter and insert as it first engages the workpiece in face milling. (b) Insert positions towards the end of cut. (c) Examples of exit angles of insert, showing desirable (positive or negative angle) and undesirable (zero angle) positions. In all figures, the cutter spindle is perpendicular to the page and rotates clockwise.

24.2.3 End Milling

- The cutter usually rotates on an axis perpendicular to the workpiece (Fig. 24.2c), although it can be tilted to machine tapered surface.
- End milling is an important and common machining operation because of its versatility and capability to produce various profiles and curved surfaces.
- End mills are available with ball nose mills for production of curved surfaces, Hollow end mills have internal cutting teeth and are used to machine cylindrical surface or solid round workpiece.
- The machines can be programmed such that the cutter can follow a complex set of paths that optimize the whole machining operation for productivity and minimum cost.
24.2.4 Other milling operations and milling cutters

- **Straddle milling**: two or more cutters are mounted on arbor and are used to machine two parallel surfaces on the workpiece (Fig. 24.11a)
- **Form milling**: produces curved profiles using cutters that have specially shaped teeth (Fig. 24.11b)
- **Slotting / Slitting** operations are done using circular cutters (Fig. 24.11c and d). T-slots such as Fig. 24.12a, a slot first is milled with an end mill; then the cutter machines the complete profile of the T-slot in one pass.

![Figure 24.11](image)

Figure 24.11 Cutters for (a) straddle milling, (b) form milling, (c) slotting, and (d) slitting with a milling cutter.

![Figure 24.12](image)

Figure 24.12 (a) T-slot cutting with a milling cutter. (b) A shell mill.
24.3 Milling Process Capabilities:

- Milling process capabilities include parameters such as surface finish, dimensional tolerances, production rate, and cost considerations are presented in table 23.1 and 23.8, Fig 23.13 and 23.14.
- The conventional ranges of cutting speeds and feeds for milling are given in table 24.2 depending on the workpiece material and its condition, cutting-tool material, and process parameters.

<table>
<thead>
<tr>
<th>TABLE 24.2</th>
<th>General Recommendations for Milling Operations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material</td>
<td>Cutting tool</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Low-carbon and-free machining steels</td>
<td>Uncoated carbide, coated carbide, cermet</td>
</tr>
<tr>
<td>Alloy steels</td>
<td>Soft Uncoated, coated, cermet</td>
</tr>
<tr>
<td></td>
<td>Hard Cermet, SiC</td>
</tr>
<tr>
<td>Cast iron, gray</td>
<td>Soft Uncoated, coated, cermet, SiC</td>
</tr>
<tr>
<td>Stainless steel, austenitic</td>
<td>Uncoated, coated, cermet</td>
</tr>
<tr>
<td>High-temperature alloys, nickel based</td>
<td>Uncoated, coated, cermet, SiC, PBN</td>
</tr>
<tr>
<td>Titanium alloys</td>
<td>Uncoated, coated,</td>
</tr>
</tbody>
</table>

| Aluminum alloys | Feed (mm/tooth) | Speed (m/min)  | Range of conditions |
|                 | mm/rev          | m/min          |
| Free machining  | Uncoated, coated, | 0.13–0.23 (0.005–0.009) | 60–900 (2000–3000) | 0.08–0.30 (0.003–0.015) | 30–3000 (1200–3000) |
| High silicon    | PCD | 0.13 (0.005) | 610 (2000) | 0.08–0.30 (0.003–0.015) | 370–910 (1200–3000) |
| Copper alloys   | Uncoated, coated, | 0.13–0.23 (0.005–0.009) | 300–750 (1000–2500) | 0.08–0.46 (0.003–0.010) | 90–1070 (300–3500) |
| Plastics        | Uncoated, coated, | 0.13–0.23 (0.005–0.009) | 270–460 (900–1500) | 0.08–0.46 (0.003–0.010) | 90–1370 (300–4500) |

Source: Based on data from Kennametal, Inc.
Note: Depths-of-cut, d, usually are in the range of 1 to 8 mm (0.04 to 0.3 in.). PBN: polycrystalline cubic boron nitride. PCD: polycrystalline diamond. See also Table 23.4 for range of cutting speeds within tool material groups.

24.2.7 Design And Operating Guidelines For Milling:

The guidelines for turning and boring given in the previous chapter also generally are applicable to milling operations. Additional factors relevant to milling operations include the following:

- Standard milling should be used and costly special cutters should be avoided.
- Chamfers should be used instead of radii.
- Avoid internal cavities and pockets with sharp corners due to difficulty of milling them.
- Workpieces should be sufficiently rigid to minimize any deflections resulting from clamping and cutting forces.

Guidelines for avoiding vibration and chatter in milling are similar to those for turning. In addition the following practices should be considered:

- Cutter should be mounted as close to the spindle base as possible in order to reduce tool deflection.
- Tool holder and fixturing devices should be as rigid as possible.
- In case of vibration and chatter tool shape and process conditions should be modified, and cutters with fewer cutting teeth or with random tooth spacing should be used.
24.2.8 Milling machines:

- **Column-and-Knee Type Milling Machines.** Used for general purpose milling operations, column-and-knee type machines are the most common milling machines. The basic components of these machines are as follows:
  - **A worktable,** on which the workpiece is clamped using T-slots. The table moves longitudinally relative to the saddle.
  - **A saddle,** which supports the table and can move in transverse direction.
  - **A knee,** which supports the saddle and gives the table vertical movement so that the depth of cut can be adjusted.
  - **An overarm** in horizontal machines, which is adjustable to accommodate different arbor lengths.
  - **A head,** which contains the spindle and cutter holders. In vertical machines, the head may be fixed or it can be vertically adjustable, and can be swiveled in a vertical plane on the column for cutting tapered surfaces.

<table>
<thead>
<tr>
<th>Problem</th>
<th>Probable causes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool breakage</td>
<td>Tool material lacks toughness, improper tool angles, machining parameters too high</td>
</tr>
<tr>
<td>Excessive tool wear</td>
<td>Machining parameters too high, improper tool material, improper tool angles, improper cutting fluid</td>
</tr>
<tr>
<td>Rough surface finish</td>
<td>Feed per tooth too high, too few teeth on cutter, tool chipped or worn, built-up edge, vibration and chatter</td>
</tr>
<tr>
<td>Tolerances too broad</td>
<td>Lack of spindle and workholding stiffness, excessive temperature rise, dull tool, chips clogging cutter</td>
</tr>
<tr>
<td>Workpiece surface burnished</td>
<td>Dull tool, depth-of-cut too low, radial relief angle too small</td>
</tr>
<tr>
<td>Back striking</td>
<td>Dull cutting tools, tilt in cutter spindle, negative tool angles</td>
</tr>
<tr>
<td>Chatter marks</td>
<td>Insufficient stiffness of system; external vibrations; feed, depth of cut, and width of cut too large</td>
</tr>
<tr>
<td>Burr formation</td>
<td>Dull cutting edges or too much honing, incorrect angle of entry or exit, feed and depth of cut too high, incorrect insert shape</td>
</tr>
<tr>
<td>Breakout</td>
<td>Lead angle too low, incorrect cutting edge geometry, incorrect angle of entry or exit, feed and depth of cut too high</td>
</tr>
</tbody>
</table>

Figure 24.15 Schematic illustration of (a) a horizontal-spindle column-and-knee type milling machine and (b) vertical-spindle column-and-knee type milling machine.
Plain milling machines have three axes of movement, which usually are imparted manually or by power. In universal column-and-knee milling machines, the table can be swiveled on horizontal plane. In this way complex shapes can be machined to produce parts such as gears, drills, tapes and cutters.

Bed-type milling machines. In bed-type milling machines, the work table is mounted directly on the bed, which replaces the knee and can move only longitudinally. These machines are not as versatile as other type, but they have high stiffness and typically are used for high-production work.

Figure 24.16 Schematic illustration of a bed-type milling machine.

Figure 24.18 Schematic illustration of a five-axis profile milling machine. Note that there are three principal linear and two angular movements of machine components.
24.3 Planning and shaping

Planning
- Cutting operation by which flat surfaces, as well as cross-sections with grooves, notches are produced along the length of workpiece. Usually done on large workpiece (25m x 15m).
- The workpiece is mounted on a table that travels back and forth along straight path.
- Because of the reciprocating motion of workpiece, elapsed time in the return non-cutting stroke is significant. Hence, these operations are suitable only for low quantity production.
- Efficiency can be improved by equipping planners with tool holders and tools that cut in both directions of table travel.
- To prevent chipping of tool cutting edges on the return stroke due to rubbing along workpiece, tools either tilted or lifted mechanically or hydraulically.
- Cutting speeds: up to 120 m/min, power up to 110kW.
- For cast and irons and steels recommended speed 3-6 m/min.
- For Al and Mg recommended speeds up to 90 m/min.
- Feeds in the range of 0.5 to 3 mm/stroke.
- Tool material: HSS (M2 and M3) and (C2 and C6) carbides.

Shaping
- It is much like planning except that (a) it is the tool and not the workpiece that travels (b) workpieces are smaller.
- In horizontal shapers, the tool travels back and forth along a straight path, while the workpiece is stationary.
- In most machines, cutting is done during the forward movement of the ram (push cut); in others, it is done during the return stroke of the ram (draw cut).
- Vertical shapers are used to machine notches, keyways and dies.
24.4 Broaching and Broaching Machines:

- The broaching operation is similar to shaping with multiple teeth and used to machine internal and external surfaces, such as holes of circular, square, or irregular section, keyways, the teeth of internal gears, and flat surfaces.
- A broach: is a long multi-tooth cutting tool; the total depth of material removed in one stroke is the sum of the depths-of-cut of each tooth of the broach. A large broach can remove material up to 38mm in one stroke.
- Broaching process can produce parts with good surface finish and dimensional accuracy, although broaches can be expensive, the cost is justified with high-quantity production run.
Broaches terminology for a typical broach is given in Fig.24.21b.

- The rake angle depends on the material cut (0 to 20°)
- Clearance angle: 1 – 4°.
- Finishing teeth have smaller angles.
- Too small a clearance angle causes rubbing of the teeth.
- The pitch depends on length of workpiece, tooth strength, and size and shape of chips.
- At least 2 teeth should be in contact with the workpiece at all times.

\[ \text{pitch} = k \sqrt{l} \]

- \( k = 1.76 \) when \( l \) is in mm, and 0.35 when \( l \) is in inch.
- Average pitch range for small broaches: 3.2-6.4mm
- Average pitch range for large broaches: 12.7-25mm

- The cut per tooth depends on workpiece material and desired surface finish:
  - Cut per tooth for medium size broaches: 0.025-0.075mm
  - Cut per tooth for large size broaches: 0.25mm or larger

Figure 24.21  (a) Cutting action of a broach showing various features.  (b) Terminology for a broach.

**Broaching Machines**

- Either pull or push, horizontal or vertical
- Push broaches are usually shorter, 150-350mm
- Pull broaches tend to straighten the whole, where pushing permits the broach to follow any irregularity of the leader hole
- The force required to pull or push the broach depends on the strength of the workpiece material, total depth and width of cut, cutting speed, tooth profile and use of cutting fluids
- Pulling force capacities of broaching machines are as high as 0.9MN (100 tons).

**Broaching Process Parameters:**

- Cutting speed may range from 1.5 m/min for high strength alloys to 15 m/min for Aluminum and Magnesium alloys.
- Broach materials: HSS (M2 and M7), and carbide inserts, or coated with titanium nitride for improving the tool life and surface finish.
- Cutting fluids generally are recommended, especially for internal broaching.

**Design consideration for broaching**

- Part design should allow secure clamping.
- Parts should have sufficient structural strength and stiffness to withstand the cutting forces during broaching
- Avoid blind holes, sharp corners, and large flat surfaces.
- Chamfers are preferable to round corners.
24.5 Sawing:

- Cutting process in which the cutting tool is a blade having a series of small teeth (saw), each tooth removing small amount of material with each stroke of the saw.
- This process can be used for all metallic and nonmetallic materials and is capable of producing various shapes (Fig24.25)
- Near net shape process from raw materials.
- Little waste material as the width of cut in sawing usually narrow.
- Saw blades are made from carbon and HSS (M-2 and M-7)
- At least 2 or 3 teeth should engage with workpiece to prevent snagging (catching of saw tooth on the workpiece).
- Cutting speeds range up to 90 m/min with lower speeds for high-strength metals.
- Cutting fluids generally are used to improve the quality of cut and the life of the saw.

Types of Saws:

- **Hacksaws**
  - Straight blades and reciprocating motions
  - Power hacksaws: 1.2-2.5 mm thick and up to 610 mm long, strokes per minute range from 30 for high-strength alloys to 180 for carbon steel.
- **Circular saws**
  - High production rate sawing.
- **Band saws**
  - Continuous, long, flexible blades and have a continuous cutting action
  - Vertical band saws are used for straight as well as contour cutting of flat sheets and other parts supported on horizontal table.
Friction Sawing

- A process in which a mild steel blade or disk rubs against the workpiece at speeds up to 7600 m/min.
- The frictional energy is converted into heat, which rapidly softens a narrow zone in the workpiece.
- The action of the blade pulls and ejects the softened metal from the cutting zone.
- This process is suitable for hard ferrous metals and reinforced plastics, but not for non-ferrous metals.

Figure 24.25 Examples of various sawing operations.

Figure 23.28 (a) Terminology for saw teeth. (b) Types of tooth set on saw teeth, staggered to provide clearance for the saw blade to prevent binding during sawing.
24.6 Filing and Finishing:

- Small scale removal of material from a surface, corner, or hole, including the removal of burrs.
- Files usually are made of hardened steel and are available in a variety of cross-sections, such as flat, round, half round, square, and triangular.
- Files can have many tooth forms and grades of coarseness.

Rotary files and burs are used for such applications as deburring, scale removal from surfaces, producing chamfers on parts, and removing small amounts of material in die making.

(a) High-speed-steel bur  
(b) Carbide bur  
(c) Rotary File

Figure 24.28 Types of burs used in burring operations.