Chapter 23.  
Machining Processes Used to Produce Round shapes: Turning and Hole Making

23.1 Introduction

- Typical products (of round shapes) are as small as miniature screws for the hinges of eyeglass frames and as large as turbine shafts for hydroelectric power plants, rolls for rolling mills, cylinders, and gun barrels.
- Turning (one of the most basic machining processes) means that the part is rotated while it is being machined. The starting material is generally a workpiece that has been made by other processes, such as casting, forging, extrusion, drawing, or powder metallurgy.
- Turning processes (typically are carried on a lathe) are outlined in Fig. 23.1 and Table 23.1.

![Figure 23.1 Miscellaneous cutting operations that can be performed on a lathe. Note that all parts are circular – a property known as axisymmetry. The tools used, their shape, and the processing parameters are described throughout this chapter.](image-url)
These machines are very versatile and capable of producing a wide variety of shapes:

- **Turning**: to produce straight, conical, curved, or grooved workpieces (Fig. 23.1a-d), such as shafts, spindles, and pins.
- **Facing**: to produce a flat surface at the end of the part (Fig. 22.1e), useful for parts that are assembled with other components. Face grooving produces grooves for applications such as O-ring seats (Fig. 23.1f).
- **Cutting with form tools**: (Fig. 23.1g) to produce various axisymmetric shapes for functional or aesthetic purposes.
- **Drilling**: to produce a hole (Fig. 23.1i), which may be followed by boring to improve its dimensional accuracy and surface finish.
- **Boring**: to enlarge a hole or cylindrical cavity made by a previous process or to produce circular internal grooves (Fig. 23.1h).
- **Parting**: (also called cutting off) to cut a piece from the end of part, as is done in the production of slugs or blanks for additional processing into discrete products (Fig. 23.1j).
- **Threading**: to produce external or internal threads (Fig. 23.1k).
- **Knurling**: to produce a regularly shaped roughness on cylindrical surfaces, as in making knobs (Fig. 23.1l).

### TABLE 23.1

<table>
<thead>
<tr>
<th>Process</th>
<th>Characteristics</th>
<th>Typical dimensional tolerances, mm (in.)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turning</td>
<td>Turning and facing operations on all types of materials, uses single-point or form tools, engine lathes require skilled labor, low production rate (but medium-to-high rate with turret lathes and automatic machines) requiring less-skilled labor</td>
<td>Fine: 0.025-0.13 (0.001-0.005) Rough: 0.13 (0.005)</td>
</tr>
<tr>
<td>Boring</td>
<td>Internal surfaces or profiles with characteristics similar to turning, stiffness of boring bar important to avoid chatter</td>
<td>0.025 (0.001)</td>
</tr>
<tr>
<td>Drilling</td>
<td>Round holes of various sizes and depths, high production rate, labor skill required depends on hole location and accuracy specified, requires boring and reaming for improved accuracy</td>
<td>0.075 (0.003)</td>
</tr>
<tr>
<td>Milling</td>
<td>Wide variety of shapes involving contours, flat surfaces, and slots; versatile; low-to-medium production rate; requires skilled labor</td>
<td>0.13-0.25 (0.005-0.01)</td>
</tr>
<tr>
<td>Planing</td>
<td>Large flat surfaces and straight contour profiles on long workpieces, low-quantity production, labor skill required depends on part shape</td>
<td>0.08-0.13 (0.003-0.005)</td>
</tr>
<tr>
<td>Shaping</td>
<td>Flat surfaces and straight contour profiles on relatively small workpieces, low-quantity production, labor skill required depends on part shape</td>
<td>0.05-0.13 (0.002-0.003)</td>
</tr>
<tr>
<td>Broaching</td>
<td>External and internal surfaces, slots, and contours; good surface finish; costly tooling; high production rate; labor skill required depends on part shape</td>
<td>0.025-0.15</td>
</tr>
<tr>
<td>Sawing</td>
<td>for hard materials unless saw has carbide teeth or is coated with diamond, low production rate, generally low labor skill</td>
<td>0.8</td>
</tr>
</tbody>
</table>

The cutting operations summarized typically are performed on a lathe (Fig. 23.2) which is available in a variety of designs, sizes, capacities, and computer-controlled features.
Turning parameters: turning (see Fig. 23.3) may be performed at various:

1. rotational speeds (N) of the workpiece clamped in a spindle,
2. depths of cut, d, and
3. feeds, f.

Such parameters depend on the workpiece and tool materials, the surface finish and dimensional accuracy required, and the characteristics of the machine tool.
23.2 The Turning Process

The majority of turning operations involve the use of simple single-point cutting tools, with the geometry of a typical right-hand cutting tool shown in Fig. 23.4 (see Fig. 23.3).

![Diagram of a right-hand cutting tool]

Figure 23.4 Designations for a right-hand cutting tool. Right-hand means the tool travels from right to left, as shown in Fig. 23.3.

23.2.1 Tool geometry:

The various angles in a single-point cutting tool have important functions in machining operations. These angles are measured in a coordinate system consisting of three major axes of the tool shank (see Fig. 23.4):

- **Rake angle** is important in controlling both direction of chips flow and the strength of tool tip. Positive rake angles improve the cutting operation by reducing forces and temperatures. However, positive angles result in a small included angle of the tool tip which may lead to premature tool chipping and failure. Side rake angle is more important than the back rake angle (which usually controls the direction of chip flow).

- **Cutting-edge angle** affects chip formation, tool strength, and cutting forces to various degrees. Typically they are around 15°.

- **Relief angle** controls interference and rubbing at the tool-workpiece interface. If the relief angle is too large, the tool tip may chip off; if it is too small, flank wear may be excessive. Relief angles typically are 5°.

- **Nose radius** affects surface finish and tool-tip strength. The smaller the nose radius (sharp tool), the rougher the surface finish of the workpiece and the lower the strength of the tool. However, large nose radii can lead to tool chatter.
Each group of workpiece materials has an optimum set of tool angles, which have been developed largely through experience; see Table 23.2.

### Table 23.2

General Recommendations for Tool Angles in Turning

<table>
<thead>
<tr>
<th>Material</th>
<th>High-speed steel</th>
<th>Carbid inserts</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Back rake</td>
<td>Side rake</td>
</tr>
<tr>
<td>Aluminum and magnesium alloys</td>
<td>20</td>
<td>15</td>
</tr>
<tr>
<td>Copper alloys</td>
<td>5</td>
<td>10</td>
</tr>
<tr>
<td>Steels</td>
<td>10</td>
<td>12</td>
</tr>
<tr>
<td>Stainless steels</td>
<td>5</td>
<td>8-10</td>
</tr>
<tr>
<td>High-temperature alloys</td>
<td>0</td>
<td>10</td>
</tr>
<tr>
<td>Refractory alloys</td>
<td>0</td>
<td>20</td>
</tr>
<tr>
<td>Titanium alloys</td>
<td>0</td>
<td>5</td>
</tr>
<tr>
<td>Cast irons</td>
<td>5</td>
<td>10</td>
</tr>
<tr>
<td>Thermoplastics</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Thermosets</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

#### 23.2.2 Material-removal rate:

- The material-removal rate (MRR) in turning is the volume of material removed per unit time (units: mm$^3$/min or in$^3$/min).
- Referring to Figure 23.3, note that for each revolution of the workpiece, we remove a ring-shaped layer of material:
  - Cross-sectional area of this ring equals the product of distance the tool travels in one revolution (feed, $f$) and the depth of cut, $d$.
  - The volume of this ring is the product of the cross sectional area ($f$)(d) and the average circumference of the ring, $\pi D_{avg}$ where
    \[
    D_{avg} = \frac{D_o + D_f}{2}
    \]
    For light cuts on large-diameter workpieces, the average diameter may be replaced by $D_o$.
  - The material removal rate per revaluation is: $\pi D_{avg} f d$.
  - With the rotational speed of the workpiece $N$ revolution per minute, then the removal rate is:
    \[
    MRR = \pi D_{avg} f d N
    \]
    Check units of in the Eq.; it indicates volume rate of removal.
- If $V$ is the cutting speed then we may write eq. 923.1a) as
  \[
  MRR = d f V
  \]
The cutting time, $t$, for a workpiece of length $l$ can be calculated by noting that the tool travels at a feed rate of $fN = (\text{mm/rev}) (\text{rev/min}) = \text{mm/min}$. Since the distance traveled is $l$ mm, the cutting time ($t$) is:

$$t = \frac{l}{fN} \quad (23.2)$$

The cutting time does not include the time required for tool approach and retraction.

The foregoing equations and terminology used are summarized in Table 23.3

| TABLE 23.3 |
| Summary of Turning Parameters and Formulas |

- $N$ = Rotational speed of the workpiece, rpm
- $f$ = Feed, mm/rev or in./rev
- $v$ = Feed rate, or linear speed of the tool along workpiece length, mm/min or in./min
  
  = $fN$
- $V$ = Surface speed of workpiece, m/min or ft/min
  
  = $\pi D_o N$ (for maximum speed)
  
  = $\pi D_{avg} N$ (for average speed)
- $l$ = Length of cut, mm or in.

- $D_o$ = Original diameter of workpiece, mm or in.
- $D_f$ = Final diameter of workpiece, mm or in.

- $D_{avg}$ = Average diameter of workpiece, mm or in.
  
  = $(D_o + D_f)/2$
- $d$ = Depth of cut, mm or in.
  
  = $(D_o - D_f)/2$
- $t$ = Cutting time, s or min
  
  = $l/fN$

- $MRR$ = $\text{mm}^3/\text{min}$ or $\text{in.}^3/\text{min}$
  
  = $\pi D_{avg}dfN$

- Torque = $N\cdot\text{m}$ or $\text{lb}\cdot\text{ft}$
  
  = $F_c D_{avg}/2$

- Power = kW or hp
  
  = (Torque)$\omega$, where $\omega = 2\pi N \text{ rad/min}$

*Note:* The units given are those that are used commonly; however, appropriate units must be used and checked in the formulas.
23.2.3 Forces in turning:

The three principal forces acting on a cutting tool are shown in Fig. 23.5:

- **The cutting force**, $F_c$, acts downward on the tool tip, tends to deflect the tool downward and the workpiece upward. This is the force that supplies the energy required for the cutting operation (see section 21.3 to calculate this force).
- The product of the cutting force and its radius from the workpiece center determines the **torque** on the spindle.
  \[
  Torque = F_c \frac{D_{avg}}{2}
  \]
  where \( D_{avg} \) = average diameter of the tool.

- The product of the torque and the spindle speed determines the **power** required in the turning operation.
  \[
  Power = Torque \times \omega
  \]
  where \( \omega \) = angular speed = 2 \( \pi \) \( N \) rad/min.

- **The thrust force**, $F_t$, acts in the longitudinal direction. This force is also called the **feed force** because it is in the feed direction of the tool. It tends to push the tool towards the right, and away from the chuck.

- **The radial force**, $F_r$, acts in the radial direction and tends to push the tool away from the workpiece.

- Because of the many factors involved in the cutting process, forces $F_t$ and $F_r$, are difficult to calculate directly; they usually are determined experimentally.

- The machine tool and its components must be able to withstand these forces without causing significant deflections, vibrations, and chatter in the overall operation.
23.2.4 Roughing and finishing cuts:

V In machining, the usual procedure is to first perform one or more roughing cuts at high feed rates and large depths-of-cut (high MRR) but with little consideration of dimensional tolerance and surface roughness.

V These cuts then are followed by a finishing cut, at a lower feed and depth-of-cut in order to produce a good surface finish.

23.2.5 Tool materials, feeds, and cutting speeds:

V The general characteristics of cutting-tool materials have been described in Chapter 22.

V A broad range of applicable cutting speeds and feeds for these tool materials is given Fig. 23.6 as a general guideline in turning operations.

Figure 23.6 The range of applicable cutting speeds and feeds for a variety of tool materials

V Specific recommendations regarding turning-process parameters for workpiece materials and cutting tools are given in Table 23.4.
### General Recommendations for Turning Operations

<table>
<thead>
<tr>
<th>Workpiece material</th>
<th>Cutting tool</th>
<th>General-purpose starting conditions</th>
<th>Range for roughing and finishing</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Depth of cut, mm (in.)</td>
<td>Feed, mm/rev (in./rev)</td>
</tr>
<tr>
<td>Low-C and free machining steels</td>
<td>Uncoated carbide</td>
<td>1.3–6.3 (0.06–0.25)</td>
<td>0.35 (0.014)</td>
</tr>
<tr>
<td></td>
<td>Ceramic-coated carbide</td>
<td>*</td>
<td>*</td>
</tr>
<tr>
<td></td>
<td>Triple-coated carbide</td>
<td>*</td>
<td>*</td>
</tr>
<tr>
<td></td>
<td>TiN-coated carbide</td>
<td>*</td>
<td>*</td>
</tr>
<tr>
<td></td>
<td>Al₂O₃ ceramic</td>
<td>*</td>
<td>0.25 (0.010)</td>
</tr>
<tr>
<td></td>
<td>Cernet</td>
<td>*</td>
<td>0.30 (0.012)</td>
</tr>
<tr>
<td>Medium and high-C steels</td>
<td>Uncoated carbide</td>
<td>1.2–4.0 (0.05–0.20)</td>
<td>0.30 (0.012)</td>
</tr>
<tr>
<td></td>
<td>Ceramic-coated carbide</td>
<td>*</td>
<td>*</td>
</tr>
<tr>
<td></td>
<td>Triple-coated carbide</td>
<td>*</td>
<td>*</td>
</tr>
<tr>
<td></td>
<td>TiN-coated carbide</td>
<td>*</td>
<td>0.25 (0.010)</td>
</tr>
<tr>
<td></td>
<td>Al₂O₃ ceramic</td>
<td>*</td>
<td>0.25 (0.010)</td>
</tr>
<tr>
<td></td>
<td>Cernet</td>
<td>*</td>
<td>0.32 (0.013)</td>
</tr>
<tr>
<td></td>
<td>SiN ceramic</td>
<td>1.25–6.3 (0.05–0.25)</td>
<td>0.32 (0.013)</td>
</tr>
</tbody>
</table>

(Continued)
<table>
<thead>
<tr>
<th>Stainless steel, austenitic</th>
<th>Triple-coated carbide</th>
<th>1.5–4.4 (0.06–0.175)</th>
<th>0.35 (0.014)</th>
<th>150 (500)</th>
<th>0.5–12.7 (0.02–0.5)</th>
<th>0.08–0.75 (0.003–0.03)</th>
<th>75–230 (250–750)</th>
<th>55–200 (175–650)</th>
<th>105–290 (350–950)</th>
</tr>
</thead>
<tbody>
<tr>
<td>TiN-coated carbide</td>
<td>&quot; &quot;</td>
<td>85–160 (275–525)</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>Cermet</td>
<td>&quot; 0.30 (0.012)</td>
<td>185–215 (600–700)</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>High-temperature alloys, nickel based</td>
<td>Uncoated carbide</td>
<td>2.3 (0.10) (0.006)</td>
<td>0.15 (0.006)</td>
<td>25–45 (75–150)</td>
<td>0.25–6.3 (0.01–0.25)</td>
<td>0.1–0.3 (0.004–0.012)</td>
<td>15–30 (50–100)</td>
<td>20–60 (65–200)</td>
<td>185–395 (600–1300)</td>
</tr>
<tr>
<td>Ceramic-coated carbide</td>
<td>&quot; &quot;</td>
<td>45 (150)</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>TiN-coated carbide</td>
<td>&quot; &quot;</td>
<td>30–55 (95–175)</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>Al₂O₃ ceramic</td>
<td>&quot; &quot;</td>
<td>260 (850)</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>SiN ceramic</td>
<td>&quot; &quot;</td>
<td>215 (700)</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>Polycrystalline cBN</td>
<td>&quot; &quot;</td>
<td>150 (500)</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>Titanium alloys</td>
<td>Uncoated carbide</td>
<td>1.0–3.8 (0.04–0.15)</td>
<td>0.15 (0.006)</td>
<td>35–60 (120–200)</td>
<td>0.25–6.3 (0.01–0.25)</td>
<td>0.1–0.4 (0.004–0.015)</td>
<td>10–75 (30–250)</td>
<td>10–100 (30–325)</td>
<td></td>
</tr>
<tr>
<td>TiN-coated carbide</td>
<td>&quot; &quot;</td>
<td>30–60 (100–200)</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>Aluminum alloys</td>
<td>Free machining</td>
<td>1.5–5.0 (0.06–0.20)</td>
<td>0.45 (0.018)</td>
<td>490 (1600)</td>
<td>0.25–8.8 (0.01–0.35)</td>
<td>0.08–0.62 (0.003–0.025)</td>
<td>200–670 (650–2000)</td>
<td>60–915 (200–3000)</td>
<td>215–795 (700–2600)</td>
</tr>
<tr>
<td>Uncoated carbide</td>
<td>&quot; &quot;</td>
<td>&quot;</td>
<td>550 (1800)</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>TiN-coated carbide</td>
<td>&quot; &quot;</td>
<td>&quot;$</td>
<td>490 (1600)</td>
<td>&quot;</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;</td>
<td>&quot;$</td>
<td>&quot;$</td>
</tr>
<tr>
<td>Cermet</td>
<td>&quot; &quot;</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
</tr>
<tr>
<td>Polycrystalline diamond</td>
<td>&quot; &quot;</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
</tr>
<tr>
<td>High silicon</td>
<td>Polycrystalline diamond</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
<td>&quot;$</td>
</tr>
</tbody>
</table>
| 530 (1700)                | (1000–10,000)        | 365–915 (1200–3000)  | (Continued)
<table>
<thead>
<tr>
<th>Workpiece material</th>
<th>Cutting tool</th>
<th>General-purpose starting conditions</th>
<th>Range for roughing and finishing</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Depth of cut, mm (in.)</td>
<td>Feed, mm/rev (in./rev)</td>
</tr>
<tr>
<td>Copper alloys</td>
<td>Uncoated carbide</td>
<td>1.5–5.0 (0.06–0.20)</td>
<td>0.25 (0.010)</td>
</tr>
<tr>
<td></td>
<td>Ceramic-coated carbide</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td></td>
<td>Triple-coated carbide</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td></td>
<td>TiN-coated carbide</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td></td>
<td>Polycrystalline diamond</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>Tungsten alloys</td>
<td>Uncoated carbide</td>
<td>2.5 (0.10)</td>
<td>0.2 (0.008)</td>
</tr>
<tr>
<td></td>
<td>TiN-coated carbide</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>Thermoplastics and thermosets</td>
<td>TiN-coated carbide</td>
<td>1.2 (0.05)</td>
<td>0.12 (0.005)</td>
</tr>
<tr>
<td></td>
<td>Polycrystalline diamond</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>Composites graphite</td>
<td>TiN-coated carbide</td>
<td>1.9 (0.075)</td>
<td>0.2 (0.008)</td>
</tr>
<tr>
<td></td>
<td>Polycrystalline diamond</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
</tbody>
</table>

Source: Based on data from Kennametal, Inc.

Note: Cutting speeds for high-speed-steel tools are about one-half those for uncoated carbides.
23.2.6 Cutting Fluids:

Many metallic and nonmetallic materials can be machined without a cutting fluid, but in most cases, the application of a cutting fluid can improve the operation significantly.

General recommendations for cutting fluids appropriate to various workpiece materials are given in Table 23.5.

However, there is a major current trend towards near-dry and dry machining, as described in section 22.12.1.

### Table 23.5

<table>
<thead>
<tr>
<th>Material</th>
<th>Type of fluid</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminum</td>
<td>D, MO, E, MO+FO, CSN</td>
</tr>
<tr>
<td>Beryllium</td>
<td>MO, E, CSN</td>
</tr>
<tr>
<td>Copper</td>
<td>D, E, CSN, MO+FO</td>
</tr>
<tr>
<td>Magnesium</td>
<td>D, MO, MO+FO</td>
</tr>
<tr>
<td>Nickel</td>
<td>MO, E, CSN</td>
</tr>
<tr>
<td>Refractory metals</td>
<td>MO, E, EP</td>
</tr>
<tr>
<td>Steels</td>
<td></td>
</tr>
<tr>
<td>carbon and low-alloy</td>
<td>D, MO, E, CSN, EP</td>
</tr>
<tr>
<td>stainless</td>
<td>D, MO, E, CSN</td>
</tr>
<tr>
<td>Titanium</td>
<td>CSN, EP, MO</td>
</tr>
<tr>
<td>Zinc</td>
<td>C, MO, E, CSN</td>
</tr>
<tr>
<td>Zirconium</td>
<td>D, E, CSN</td>
</tr>
</tbody>
</table>

*Note: CSN = chemicals and synthetics; D = dry; E = emulsion; EP = extreme pressure; FO = fatty oil; and MO = mineral oil.*

See Example 23.1
23.3 Lathes and Lathe Operations

- Metalworking lathes were first built in late 1970. The most common lathe originally was called an engine lathe. Today, these lathes are all equipped with individual electric motors.

- The maximum spindle speed of lathes typically is around 4000 rpm but only about 200 rpm for large lathes. For special applications, speeds may range to 10,000 rpm, 40,000 rpm, or higher.

- The cost of lathes ranges from about $2000 for bench types to over $100,000 for larger units.

23.3.1 Lathe Components

- Lathes are equipped with a variety of components and accessories, as shown in Fig. 23.2 (repeated next). Their features and functions are described next.

- **Bed:**
  - The bed supports all major components of the lathe. Beds have a large mass and are built rigidly, usually from gray or nodular cast iron.
  - The top portion of the bed has two ways with various cross-sections that are hardened and machined for wear resistance and dimensional accuracy during turning.

- **Carriage:** or carriage assembly slides along the ways and consists of an assembly of the cross-slide, tool post, and apron:
  - The cutting tool is mounted on the tool post, usually with a compound rest that swivels for tool positioning and adjustment.
  - The cross-slide moves radially in and out, controlling the radial position of the cutting tool in operations such as facing (Fig. 23.1e).
  - The apron is equipped with mechanisms for both manual and mechanized movement of the carriage and the cross-slide by means of the lead screw.
**Headstock:** The headstock is fixed to the bed and is equipped with motors, pulleys, and V-belts that supply power to a spindle at various rotational speeds.

- Speeds can be set through manually-controlled selectors or by electrical controls.
- Most headstocks are equipped with a set of gears, and some have various drives to provide a continuously variable speed range to the spindle.
- Headstocks have a hollow spindle to which workholding devices (such as chucks and collets) are mounted and long bars or tubing can be fed through them for various turning operations.

**Tailstock:**

- The tailstock, which can slide along the ways and be clamped at any position, supports the other end of the workpiece.
- It is equipped with a center that may be fixed (dead center), or it may be free to rotate with the workpiece (live center).
- Drills and reamers can be mounted on the tailstock quill (a hollow cylindrical part with a tapered hole) to drill axial holes in the workpiece.

**Feed rod and Lead screw:**

- The feed rod is powered by a set of gears through the headstock. It rotates during the operation of the lathe and provides movement to the carriage and the cross-slide by means of gears, a friction clutch, and a keyway along the length of the rod.
- Closing a split nut around the lead screw engages it with the carriage; it is also used for cutting threads accurately.

**Lathe Specifications:** a lathe is generally specified by:

- **Swing:** the maximum diameter of the workpiece that can be machined; see Table 23.6.
- The maximum distance between the headstock and tailstock centers, and
- The length of the bed.

<table>
<thead>
<tr>
<th>Machine tool</th>
<th>Maximum dimension (m)</th>
<th>Power (kW)</th>
<th>Maximum rpm</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Lathes (swing:length)</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bench</td>
<td>0.3/1</td>
<td>61</td>
<td>3000</td>
</tr>
<tr>
<td>Engine</td>
<td>3/5</td>
<td>70</td>
<td>4000</td>
</tr>
<tr>
<td>Turret</td>
<td>0.5/1.5</td>
<td>60</td>
<td>3000</td>
</tr>
<tr>
<td>Automatic screw</td>
<td>0.1/0.3</td>
<td>20</td>
<td>10,000</td>
</tr>
<tr>
<td><strong>Boring machines (work diameter/length)</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Vertical spindle</td>
<td>4/3</td>
<td>200</td>
<td>300</td>
</tr>
<tr>
<td>Horizontal spindle</td>
<td>1.5/2</td>
<td>70</td>
<td>1000</td>
</tr>
<tr>
<td><strong>Drilling machines</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bench and column (drill diameter)</td>
<td>0.1</td>
<td>10</td>
<td>12,000</td>
</tr>
<tr>
<td>Radial (column to spindle distance)</td>
<td>3</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Numerical control (table travel)</td>
<td>4</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

*Note: Larger capacities are available for special applications.*
23.3.2 Workholding devices and accessories:

- **Workholding** devices are important as they must hold the workpiece securely.

- Figure 23.3 shows how the end of the workpiece is clamped to the spindle of the lathe by a chunk, collet, face plate, or mandrel.

- **A chuck** is usually equipped with three or four jaws:
  - Three-jaws chucks generally have a geared-scroll design that makes the jaws self-centering. They are used for round workpieces (such as bar stock, pipes, tubes), which can be centered to within 0.025 mm.
  - Four-jaw (independent) chucks have jaws that can be moved and adjusted independently of each other. They can be used for square, rectangular, or odd-shaped workpieces. They are used for heavy workpieces or for work requiring multiple chuckings where concentricity is important.
  - Jaws can be reversed to permit clamping of workpieces either on the outside surface or on the inside surfaces of hollow workpieces, such as pipes and tubing.
  - Chucks can be power actuated or manually actuated using a chuck wrench.
  - Chucks are available in various designs and sizes. Their selection depends on:
    - Type and speed of operation,
    - Workpiece size,
    - Production and dimensional accuracy requirements, and
    - Jaw forces required.
  - High spindle speeds can reduce jaw (clamping) forces largely due to the effect of centrifugal forces; this effect is important particularly in precision tube turning.

- **Power chucks**, actuated pneumatically or hydraulically, are used in automated equipment for high production rates, including the loading of parts using industrial robots. These have been introduced to meet the increasing demands for stiffness, precision, versatility, power, and high cutting speeds.

- **A collet** is basically a longitudinally-split tapered bushing.
  - The workpiece (generally with a maximum diameter of 1 in) is placed inside the collet, and the collet is pulled (draw-in collet; Fig. 23.7a & b) or pushed (pushed-out collet; Fig. 23.7c) mechanically into the spindle.
  - Collets are used for round workpieces as well as for other shapes and are available in a wide range of incremental sizes.
  - The tapered surfaces shrink the segments of the collet radially, tightening onto the workpiece.
  - Collets are used for round workpieces as well as for other shapes (square or hexagonal) and are available in a wide range of incremental sizes.
Face plates are used for clamping irregularly shaped workpieces. The plates are round and have several slots and holes through which the workpiece is bolted or clamped (Fig. 23.7d).

Mandrels (Fig. 23.8) are placed inside hollow or tubular workpieces and are used to hold workpieces that require machining on both ends or on their cylindrical surfaces. Some mandrels are mounted between centers on the lathe.
ACCESSORIES: several devices are available as accessories and attachments for lathes. Among these devices are the following:

- **Carriage and cross-slide stops**, with various designs to stop the carriage at a predetermined distance along the bed.
- Devices for turning parts with various tapers.
- Milling, sawing, gear-cutting, and grinding attachments.
- Various attachments for boring, drilling, and thread cutting.

**23.3.3 Lathe Operations:**

In a typical turning operation, the workpiece is clamped by one of the Workholding devices described previously.

Long and slender parts should be supported by a steady rest and follow rest placed on the bed; otherwise the part will deflect under the cutting force. These rests usually are equipped with three adjustable fingers or rollers that support the workpiece while allowing it to rotate freely.

The cutting tool is attached to the tool post, which is driven by the lead screw, and removes material by traveling along the bed.

- A right-hand tool travels toward the headstock.
- A Left-hand tool travels toward the tailstock.
- Facing operations are done by moving the tool radially with the cross-slide and also clamping the carriage for better dimensional accuracy.

**Form tools** are used to produce various shapes on solid, round workpieces (Fig. 23.1g) by turning; moving the tool radially inward while the part is rotating (form cutting):

- Form cutting is not suitable for deep and deep grooves or sharp corners because vibration and chatter may result and cause poor surface finish.
- As a rule:
  - (a) the formed length of the part should not be greater than 2.5 times the minimum diameter of the part,
  - (b) the cutting speed should be set properly, and
  - (c) cutting fluids should be used.

**Boring** (section 23.4) on a lathe is similar to turning. It is performed inside hollow workpieces or in a hole made previously by drilling or other means.

- Out-of-shape holes can be straightened by boring.
- The workpiece is held in a chuck or in some other suitable workholding device.

**Drilling** (section 23.5) can be performed on a lathe by mounting the drill bit in a chuck into the tailstock quill. The workpiece is clamped in a workholder on the headstock, and the drill bit is advanced by rotating the hand wheel of the tailstock.

- Holes drilled in this manner may not be sufficiently concentric because of the tendency for the drill to drift radially.
- The concentricity of the hole can be improved by subsequently boring the drilled hole.

**Reaming:** drilled holes may be reamed on lathes in a manner similar to drilling, thus improving hole dimensional tolerance and surface finish.

**Knurling:** is performed on a lathe with hardened rolls (Fig. 23.1L), in which the surface of the rolls is a replica of the profile to be generated. The rolls are pressed radially against the rotating workpiece, while the tool moves axially along the part.
The tools for **parting, grooving, thread cutting**, and various other operations are specially shaped for their particular purpose or are available as inserts.

### 23.3.4 Types of Lathes

There are a number of other lathe types, brief descriptions are given next.

1. **Bench lathes** are placed on a workbench;
   - **they have low power and are usually operated by hand feed**
   - **used to precision-machine small workpiece.**

2. **Special purpose lathes**
   - used for special applications such as railroad wheels and gun barrels.
   - **workpiece sizes as large as 1.7 m in diameter by 8 m length and capacities of 450 kW.**

3. **Tracer lathes**
   - have special attachments that are capable of turning parts with various contours.
   - **cutting tool follows a path that duplicates the contour of the template which guides the cutting tool along the workpiece without operator intervention (duplicating or contouring lathes).**

4. **Automatic lathes**
   - Manual machine control have been replaced by various mechanisms that enable cutting operation to follow a certain prescribed sequence.
   - **In a fully automatic machine, parts are fed and removed automatically, whereas in semiautomatic machines, these functions are performed by the operator.**
   - **Automatic lathes are suitable for medium to high volume production.**

5. **Automatic bar machines (Automatic screw machines):**
   - **Designed for high-production-rate machining of screws and similar threaded parts.**
   - **After each screw or part is machined to finished dimensions, the bar stock is fed forward automatically and then cut off.**

6. **Turret lathes:**
   - **Are capable of performing multiple cutting operations, such as turning, boring, drilling, thread cutting, and facing; see Fig. 23.9.**
   - **Several cutting tools (as many as 6) are mounted on the hexagonal main turret, which is rotated after each specific cutting operation is completed.**
7. **Computer-controlled lathes**

Movement and control of the machine and its components are actuated by computer numerical controls (CNC); see Fig. 23.10a.

These lathes are usually equipped with one or more turrets, each turret is equipped with a variety of tools and performs several operations on different surfaces of the workpiece; Fig. 23.10b.

Workpiece diameters may be as much as 1 m.

CNC lathes are designed to operate faster and have higher power available as compared to other lathes.

They are equipped with automatic tool changers (ATC). Their operations are reliably repetitive, maintain the desired dimensional accuracy, and require less skilled labor.

See examples 23.2, and 23.3

---

Figure 23.9 Schematic illustration of the components of a turret lathe. Note the two turrets: square and hexagonal (main).

Figure 23.10 (a) A computer numerical-control lathe. Note the two turrets on this machine. These machines have higher power and spindle speed than other lathes in order to take advantage of new cutting tools with enhanced properties. (b) A typical turret equipped with ten tools, some of which are powered.
23.3.5 Turning Process capabilities

- Relative production rates in turning are shown (as well as other operations) in Table 23.8.

![Table 23.8](image)

<table>
<thead>
<tr>
<th>Operation</th>
<th>Rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turning</td>
<td></td>
</tr>
<tr>
<td>Engine lathe</td>
<td>Very low to low</td>
</tr>
<tr>
<td>Tracer lathe</td>
<td>Low to medium</td>
</tr>
<tr>
<td>Turret lathe</td>
<td>Low to medium</td>
</tr>
<tr>
<td>Computer-controlled lathe</td>
<td>Low to medium</td>
</tr>
<tr>
<td>Single-spindle chuckers</td>
<td>Medium to high</td>
</tr>
<tr>
<td>Multiple-spindle chuckers</td>
<td>High to very high</td>
</tr>
<tr>
<td>Boring</td>
<td>Very low</td>
</tr>
<tr>
<td>Drilling</td>
<td>Low to medium</td>
</tr>
<tr>
<td>Milling</td>
<td>Low to medium</td>
</tr>
<tr>
<td>Planing</td>
<td>Very low</td>
</tr>
<tr>
<td>Gear cutting</td>
<td>Low to medium</td>
</tr>
<tr>
<td>Broaching</td>
<td>Medium to high</td>
</tr>
<tr>
<td>Sawing</td>
<td>Very low to low</td>
</tr>
</tbody>
</table>

- Surface finish (Fig. 23.13) and dimensional accuracy (Fig. 23.14) obtained in turning and related operations depend on several factors, such as:
  - The characteristics and condition of the machine tool, stiffness, vibration and chatter, process parameters, tool geometry and wear, the use of cutting fluids, the machinability of workpiece material, and operator skill
  - As a result, a wide range of surface finish can be obtained, as shown in Fig. 23.13.
Figure 23.13  The range of surface roughnesses obtained in various machining processes. Note the wide range within each group, especially in turning and boring.
Figure 23.14 Range of **dimensional tolerances** obtained in various machining processes as a function of workpiece size. Note that there is an order of magnitude difference between small and large workpieces.
23.3.6 Design considerations and guidelines for turning operations:

When turning operations are necessary, the following general design guidelines should be followed:

1. Parts should be designed so that they can be fixtured and clamped in workholding devices with relative ease.
2. Dimensional accuracy and surface finish should be as wide as permissible for the part to still function properly.
3. Avoid sharp corners, tapers, steps, and major dimensional variations.
4. Blanks to be machined should be as close to final dimensions as possible, so to reduce production cycle time.
5. Parts should be designed so that cutting tools can travel across the workpiece without obstruction.
6. Design features should be such that commercially available standard cutting tools, inserts, and toolholders can be used.
7. Workpiece materials should be chosen according to machinability as possible.

Vibration during cutting can cause poor surface finish, poor dimensional accuracy, excessive tool wear, and premature tool failure.

Guidelines for turning operations. Table 23.9 shows the probable causes of problems in turning operations. The following list of some guidelines for turning operations.

1. Minimize tool overhang.
2. Support workpiece rigidly.
3. Use machine tools with high stiffness and high damping capacity.
4. When tools begin to vibrate and chatter, modify one of the process parameters, such as tool geometry, cutting speed, feed rate, depth-of-cut, or use of cutting fluid.

Because of complexity of the problem, some guidelines have to be implemented of a trial-and-error basis.

<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, machine tool lacks stiffness, worn bearings and machine components, machining parameters too high</td>
</tr>
<tr>
<td>Excessive tool wear</td>
<td>Machining parameters too high, improper tool material, ineffective cutting fluid, improper tool angles</td>
</tr>
<tr>
<td>Rough surface finish</td>
<td>Built-up edge on tool; feed too high; tool too sharp, chipped, or worn; vibration and chatter</td>
</tr>
<tr>
<td>Dimensional variability</td>
<td>Lack of stiffness of machine tool and workholding devices, excessive temperature rise, tool wear</td>
</tr>
<tr>
<td>Tool chatter</td>
<td>Lack of stiffness of machine tool and workholding devices, excessive tool overhang, machining parameters not set properly</td>
</tr>
</tbody>
</table>
23.3.7 Chip collection systems:**
V Chips produced during machining must be collected and disposed of properly.
V The volume of chips is very high. For example, the loose bulk volume of chips, when milling a 1
in3 steel, is 30 to 40 in3.
V Also called chip management, the operation involves collecting chips from their source in the
machine tool in an efficient manner and removing them from the work area.
V Chip collection systems, chips can be collected by any of the following methods:
1. Allowing gravity to drop them on steel conveyor belt.
2. Dragging chips from a settling tank.
3. Using augers with feed screws (similar to those in meat grinders).
4. Using magnetic conveyors for ferrous chips.
5. Employing vacuum methods of chip removal.
V Dry chips are more valuable for recycling because of reduced environmental contamination.

23.3.8 Cutting screw threads:**
V A screw thread may be defined as a ridge of uniform cross-section that follow a helical or spiral
path on the outside or inside of cylindrical (straight thread) or tapered conical surface (tapered
thread).
V Machine screws, bolts, and nuts have straight threads, as do threaded rods for applications such as
the lead screw in lathes.
V Threads may be right-handed or left-handed.
V Tapered threads commonly are used for water or gas pipes and plumbing supplies.
V Threads traditionally have been machined, but increasingly, they are formed by thread rolling.
V Threads can be machined externally or internally with a cutting tool with a process called thread
cutting or threading.
V Internal threads also can be produced with a special threaded tool, called a tap, and the process is
called tapping.
V Threads may be produced basically by forming (thread rolling) or by cutting and it is also
possible to cast threaded parts.

A. Screw-threads cutting on a lathe.
Figure 23.15 (a) Cutting screw threads on a lathe with a single-point cutting tool.
(b) Cutting screw threads with a single-
point tool in several passes, normally
utilized for large threads. The small
arrows in the figures show the direction of
the feed, and the broken lines show the
position of the cutting tool as time
progresses. Note that in radial cutting, the
tool is fed directly into the workpiece. In
flank cutting, the tool is fed into the piece
along the right face of the thread. In
incremental cutting, the tool is first fed
directly into the piece at the center of the
thread, then at its sides, and finally into
the root. (c) A typical coated-carbide
insert in the process of cutting screw
threads on a round shaft. (d) Cutting
internal screw threads with a carbide insert. Source: (c): Courtesy of Iscar Metals Inc.
B. Threading Die:

![Image of threading die]

The production rate in cutting screw threads can be increased with tools called die-head chasers. These tools have four cutters with multiple teeth and can be adjusted radially; see Fig. 23.16.

C. Design Considerations for Screw Thread Cutting:

To produce high-quality and economical screw threads, the following design considerations must be taken into account:

- Designs should allow for the termination of threads before they reach a shoulder.
- Internal threads in blind holes (does not go through the thickness of the workpiece) should have unthreaded length at bottom.
- Chamfers should be specified at the ends of threaded section.
- Thread sections should not be interrupted with slots, holes or other discontinuities.
- Use standard threading tooling and inserts.
- Thin-walled parts should have sufficient thickness and strength to resist clamping and cutting force.
- All cutting operations should be completed in one setup.

23.4 Boring and Boring Machines

Boring is performed to enlarge a hole made previously by some other process (like drilling) or to produce circular internal profiles in hollow workpieces (Fig. 23.1h).

The cutting tools are similar to those used in turning and are mounted on a boring bar to reach the full length of the bore (Fig. 23.17a).

Boring bars have been designed and built with capabilities for damping vibration (Fig. 23.17b).

![Image of boring bar with carbide insert and tungsten-alloy disks]

Figure 23.17 (a) Schematic illustration of a steel boring bar with a carbide insert. Note the passageway in the bar for cutting fluid application. (b) Schematic illustration of a boring bar with tungsten-alloy “inertia disks” sealed in the bar to counteract vibration and chatter during boring. This system is effective for boring bar length-to-diameter ratios of up to 6.
Boring operations on relatively small workpieces can be carried out on a lathe. **Boring mills** are used for large workpieces.

Boring mills are either vertical or horizontal, and are capable of performing operations such as turning, facing, grooving and chamfering.

In horizontal boring machines, the workpiece is mounted on a lathe that can move horizontally in both the axial and radial directions. The cutting tool is mounted on a spindle that rotates in the headstock, which is capable of both vertical and longitudinal movements.

A vertical boring mill (Fig. 23.18) is similar to a lathe, has a vertical axis of workpiece rotation, and can accommodate workpiece with diameters as much as 2.5m.

Boring machines are available with a variety of features. Machine capabilities range up to 150 kW and are available with computer numerical controls, allowing all movements of the machine to be programmed.

![Vertical Boring Mill Diagram](image)

**Figure 23.18  Schematic illustration of a vertical boring mill.** Such a machine can accommodate workpiece sizes as large as 2.5m (98 in.) in diameter.

### 23.4.1 Design Considerations for Boring:

- Guidelines for efficient and economical boring operations are similar to those for turning. Additionally, the following factors should be considered:
  - Whenever possible, through holes rather than blind holes should be specified.
  - The greater the length-to-bore-diameter ratio, the more difficult it is to hold dimensions because of the deflections of the boring bar due to cutting forces, as well as the higher tendency for vibration and chatter.
  - Avoid interrupted internal surfaces.

### 23.5 Drilling, Drills, and Drilling Machines

- Holes typically are used for assembly with fasteners (such as bolts, screws, and rivets) or for design purposes (such as weight reduction, ventilation, access into inside parts, or for appearance).
- Hole making is among the most important operations in manufacturing.
- Drilling is a major and common hole making process.

### 23.5.1 Drills

- Drills typically have high length-to-diameter ratios (Fig. 23.19), they are capable of producing relatively deep holes.
- The chips produced within the hole move in a direction opposite to the forward movement of the drill.
- Drills generally leave a burr on the bottom surface upon breakthrough, necessitating deburring operations.
Figure 23.19  Two common types of drills:  (a)  Chisel-point drill.  The function of the pair of margins is to provide a bearing surface for the drill against walls of the hole as it penetrates into the workpiece.  Drills with four margins (double-margin) are available for improved drill guidance and accuracy.  Drills with chip-breaker features also are available.  (b)  Crankshaft drills.  These drills have good centering ability, and because chips tend to break up easily, these drills are suitable for producing deep holes.

- Generally, the hole diameter produced by drilling are slightly larger than the drill diameter (oversize); drill can be removed easily from the hole.
- For better surface finish and dimensional accuracy, drilled holes may be subjected to subsequent operations, such as reaming and honing. The capabilities of drilling and boring are shown in Table 23.10.

### TABLE 23.10

<table>
<thead>
<tr>
<th>Cutting tool type</th>
<th>Diameter range (mm)</th>
<th>Hole depth/diameter</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Typical</td>
</tr>
<tr>
<td>Twist</td>
<td>0.5-150</td>
<td>8</td>
</tr>
<tr>
<td>Spade</td>
<td>25-150</td>
<td>30</td>
</tr>
<tr>
<td>Gun</td>
<td>2-50</td>
<td>100</td>
</tr>
<tr>
<td>Trepansning</td>
<td>40-250</td>
<td>10</td>
</tr>
<tr>
<td>Boring</td>
<td>3-1200</td>
<td>5</td>
</tr>
</tbody>
</table>
A. Twist Drill.

- The most common drill is the conventional standard-point twist drill (23.19a).
- The geometry of the drill point is such that the normal rake angle and the velocity of the cutting edge vary with the distance from the center of the drill.
- The main features of this drill are:
  a) point angle (118 – 135),
  b) lip-relief angle (7 - 15),
  c) chisel-edge angle (125 – 135), and
  d) helix angle (15 – 30).
- Two spiral grooves (flutes) run the length of the drill, and the chips produced are guided upward through these grooves. The grooves also serve as passageways to enable the cutting fluid to reach the cutting edges.
- Drills are available with a chip-breaker feature ground along the cutting edges.
- Small changes in the drill geometry can have a significant effects on a drill’s performance, particularly, in the chisel-edge region, which accounts for 50% of the thrust force in drilling. For example:
  - Too small a lip relief angle (Fig. 23.19a), increases the thrust force, generates excessive heat, and increases wear.
  - Conversely, too large a lip relief angle causes chipping or breaking of the cutting edge.

B. Other Types of Drills:

- Several types of drill are shown in Fig. 23.20.

Figure 23.20 Various types of drills and drilling and reaming operations.

- **Step drill**: produces holes with two or more different diameters.
- **Core drill**: is used to make an existing hole larger.
- **Counterboring and countersinking drills**: produce depressions on the surface to accommodate the heads of screws and bolts below the workpiece surface.
- **Center drill**: is short and is used to produce the hole at the end of a piece of stock so that it may be mounted between centers of the headstock and the tailstock of a lathe.
- **Spot drill**: is used to spot (to start) a hole at the desired location on a surface.
Spade drill: (Fig. 23.21a) have removable tips or bits and are used to produce large diameter and deep holes. A similar drill is the straight-flute drill (Fig. 23.21b).

Solid carbide and carbide-tipped drills (Fig. 23.21c & d) are available for drilling hard materials (such as cast irons), high-temperature metals, abrasive materials (such as concrete and brick- masonry drills), and composite materials with abrasive fiber reinforcements (such as glass and graphite).

**Gun drilling:** is originally developed for drilling gun barrels:
- Used for drilling deep holes.
- The depth-to-diameter ratios of holes produced can be 300:1 or even higher
- Cutting speeds are usually high and feeds are low.
- The cutting fluid is forced under high pressure through a longitudinal hole in the body of the drill.

Figure 23.21 Various types of drills.

Figure 23.22 (a) A gun drill showing various features. (b) Schematic illustration of the gun-drilling operation.
In trepanning, the cutting tool (23.23a) produces a hole by removing a disk-shaped piece (core), usually from flat plates. The trepanning process can be used to make disks up to 250 mm in diameter from flat sheet or plate. It can be used to make circular grooves in which O-rings are placed. Trepanning can be carried out on lathes, drill presses, or other machine tools using single-point or multipoint tools; see Fig. 23.23b.

23.5.2 Material-removal rate in drilling

MRR in drilling is the volume of material removed by the drill per unit time (mm$^3$/min). For a drill with a diameter $D$, the cross-sectional area of the drilled hole is $\pi D^2 / 4$. The velocity of the drill perpendicular to the workpiece is the product of the feed $f$, and the rotational speed $N$ (where $N = V / \pi D$). Thus,

$$MRR = \left(\frac{\pi D^2}{4}\right) f \ N$$

(23.3)

23.5.3 Thrust force and torque

Thrust force in drilling acts perpendicular to the hole axis; if this force is excessive, it can cause the drill to bend or break. The thrust force, which is difficult to calculate, depends on:

a) Strength of workpiece material,

b) Feed,

c) Rotational speed,

d) Drill diameter,

e) Drill geometry, and

f) Cutting fluid.

Torque in drilling. A knowledge of the magnitude of torque in drilling is essential for estimating the power requirement. It is difficult to calculate. It can be estimated from the data given in Table 21.2.

Recall that

$$\text{power} = \text{torque} \times \text{rotational speed}.$$
23.5.4 Drill Materials and Sizes:
- Drills are usually made of HSS (M1, M7, and M10) and solid carbides or with carbide tips (K20-C2 carbide); see Fig. 23.21c and d.
- Drills now are coated with titanium nitride or titanium carbide for increased wear resistance.
- Polycrystalline-diamond coated drills for fastener holes in fiber-reinforced plastics.
- Standard twist-drill sizes:
  - Numerical: No. 97 (0.0059 in) to No. 1 (0.228 in)
  - Letter: A (0.234 in) to Z (0.413 in)
  - Millimeter: from 0.05 mm in increments of 0.01 mm.
  - Fractional:
    - Straight shank: from 1/64 to 1 1/4 in (in 1/64 in. increments) to 1 1/2 in (in 1/32 incr)
    - Taper shank: from 1/8 – 13/4 in (in 1/64 in. increments) to 3.5 (in 1/16 increments)

23.5.5 Drilling practice
- Drills and similar hole-making tools usually are held in drill chucks, which may be tightened with or without keys.
- Because it does not have a centering action, a drill tends to “wake” on the workpiece surface at the beginning of the operation (specially with small-diameter long drills).
- To prevent the drill from walking on the workpiece surface at the beginning of the operation:
  - The drill should be guided using fixtures (such as a bushing) to keep it from deflecting laterally.
  - A small starting hole can be made with a center drill (usually with a point angle of 60°),
  - Use a drill with S shape point (helical or spiral point). This shape has a self centering characteristic, or
  - Use a centering punch to produce an initial impression in which drilling starts or else to incorporate dimples into the cast.

2 Drilling Recommendations:
- Recommended ranges for drilling speeds and feeds are given in Table 23.11. The speed is the surface speed of the drill at its periphery. Thus a 12.7 mm drill rotating at 300 rpm has a surface speed of

\[
\frac{12.7}{2} \text{ mm } \times \frac{300 \text{ rev } / \text{ min}}{1 \text{ rev } / \text{ rad}} \times \frac{2\pi \text{ rad } / \text{ rev}}{1000 \text{ mm } / \text{ m}} = 11.97 \text{ m } / \text{ min}
\]

- The feed in drilling is the distance the drill travels into the workpiece per revolution.

![Table 23.11](image)

**Note:** As hole depth increases, speeds and feeds should be reduced. Selection of speeds and feeds also depends on the specific surface finish required.
Chip removal during drilling can be difficult, especially for deep holes in soft and ductile workpiece materials. The drill should be retracted periodically (pecking) to remove chips that may have accumulated along the flutes.

A general guide to probable causes of problems in drilling operations is given in Table 23.12.

<table>
<thead>
<tr>
<th>Problem</th>
<th>Probable causes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drill breakage</td>
<td>Dull drill, drill seizing in hole because of chips clogging flutes, feed too high, lip relief angle too small</td>
</tr>
<tr>
<td>Excessive drill wear</td>
<td>Cutting speed too high, ineffective cutting fluid, rake angle too high, drill burned and strength lost when sharpened</td>
</tr>
<tr>
<td>Tapered hole</td>
<td>Drill misaligned or bent, lips not equal, web not central</td>
</tr>
<tr>
<td>Oversize hole</td>
<td>Same as above, machine spindle loose, chisel edge not central, side force on workpiece</td>
</tr>
<tr>
<td>Poor hole surface finish</td>
<td>Drill dull, ineffective cutting fluid, welding of workpiece material on drill margin, improperly ground drill, improper alignment</td>
</tr>
</tbody>
</table>

**Drill Reconditioning:**
- Drills are reconditioned by grinding them either manually or using special fixtures.
- Hand grinding is difficult and requires considerable skill.
- Grinding on fixtures is accurate and is done on special computer controlled grinders.
- Coated drills can be recoated.

**Measuring drill life:**
- Drill life (as well as tap life) is usually measured by the number of holes drilled before they become dull, or until thrust force or torque starts to increase (using force transducer).
- Other technique is to monitor vibration and acoustic emissions.

23.5.6 Drilling machines
- Drilling machines are used for drilling holes, tapping, reaming, and small-diameter boring operations.
- Most common machine is the drill press; see Fig. 23.24a:
  - The workpiece is placed on an adjustable table, either by clamping it directly the slots and hole on the table or by using a vise.
  - The drill is lowered manually by a hand wheel or by power feed at preset rates.
  - Manual feeding requires some skill in judging the appropriate feed rate.
  - Drill presses usually are designated by the largest workpiece diameter that can be accommodated on the table and typically range from 150 mm to 1250 mm.
  - The spindle speed on drilling machines has to be adjustable to accommodate different drill sizes.
- The types of drilling machines range from simple bench-type drills used to drill small-diameter holes to large radial drills (Fig. 23.24b), which can accommodate large workpieces.
Developments in drilling machines include numerically controlled three-axis machines, in which the operations are performed automatically and in the desired sequence which the use of turret (Fig. 23.25). Note that the turret holds several different drilling tools.

Drilling machines with multiple spindles (gang drilling) are used for high-production-rate operations. These machines are capable of drilling, in one cycle, as many as 50 holes of varying sizes, depths, and locations.

Workholding devices keep the workpiece from slipping or rotating during properly. These devices are available in various designs.

Figure 23.25 A three-axis computer numerical-control drilling machine. The turret holds as many as eight different tools, such as drills, taps, and reamers.
23.5.7 Design considerations for drilling

- The basic design guidelines for drilling are as follows:
  1. Designs should allow holes to be drilled on flat surfaces & perpendicular to the drill motion. Exit surfaces for the drill also should be flat.
  2. Interrupted hole surfaces should be avoided or minimized for improved dimensional accuracy, drill life, and to avoid vibrations.
  3. Hole bottoms should match, if possible, standard drill-point angles.
  4. Through holes are preferred over blind holes.
  5. Parts should be designed so that all drilling can be done with minimum of fixturing and without repositioning the workpiece.
  6. Blind holes must be drilled deeper so that reaming or tapping operations may performed.

23.6 Reaming and Reamers

- Reaming is an operation used to:
  a) Make an existing hole dimensionally more accurate than can be obtained by drilling alone.
  b) Improve the hole surface finish.

- The most accurate holes are produced by the following sequence of operations:
  1. Centering,
  2. Drilling,
  3. Boring, and
  4. Reaming.

- A reamer (Fig. 23.26a) is a multiple-cutting-edge tool with straight or helically fluted edges that removes very little material.

- For soft metals, a reamer typically remove a minimum of 0.2 mm on the diameter of a drilled hole, and for harder metal about 0.13 mm.

- In general, reamer speeds are one-half those of the same-sized drill and 3 times the feed rate.

- Reamers typically are made of high-speed steels (M1, M2, and M7) or solid carbides (K20; C2), or have carbide-cutting edges.

![Diagram](image)
23.7 Tapping and Taps

- Internal threads in workpiece can be produced by tapping.
- A tap is a chip-producing threading tool with multiple cutting teeth (Fig. 23.27a).
- Tapes are available with two, three, or four flutes.
- The two-flute tap forces the chips into the hole so that the tap needs to be retracted only at the end of the cut. Three-fluted taps are stronger because more material is available in the flute.
- Tapered taps are designed to reduce the torque required for the tapping of through holes.
- Bottoming taps are used for tapping blind holes to their full depth.
- Collapsible taps are used in large-diameter holes; after tapping has been competed, the tap is collapsed mechanically and is removed from the hole without rotation.
- The use of a cutting fluid and periodic reversal and removal of the tap from the hole are effective means of chip removal and of improving the quality of the tapped hole.
- Drapping, combination of drilling and tapping in a single tool, is used for higher tapping productivity.
- Tapping may be done by hand or with machines such as drilling machines, lathes, automatic screw machines, and vertical CNC milling machines. One system for automatic tapping of nots is shown in Fig. 23.27b.
- With proper lubrication, tap life may be as high as 10,000 holes.
- Tapes are usually made of HSS (M1, M2, M7, and M10) for production work.

![Diagram of tap and nut](image-url)