CHAPTER 33

PRODUCTION PROCESSES AND EQUIPMENT

Magd E. Zohdi
Industrial Engineering Department
Louisiana State University
Baton Rouge, Louisiana

William E. Biles
Industrial Engineering Department
University of Louisville
Louisville, Kentucky

Dennis B. Webster
Industrial Engineering Department
Louisiana State University
Baton Rouge, Louisiana

Mechanical Engineers' Handbook, 2nd ed., Edited by Myer Kutz.
ISBN 0-471-13007-9

33.1 METAL-CUTTING PRINCIPLES

Material removal by chipping process began as early as 4000 BC, when the Egyptians used a rotating bowstring device to drill holes in stones. Scientific work developed starting about the mid-19th century. The basic chip-type machining operations are shown in Fig. 33.1.

Figure 33.2 shows a two-dimensional type of cutting in which the cutting edge is perpendicular to the cut. This is known as *orthogonal* cutting, as contrasted with the three-dimensional *oblique* cutting shown in Fig. 33.3. The main three cutting velocities are shown in Fig. 33.4. The metal-cutting factors are defined as follows:

α	rake angle
β	friction angle
γ	strain
λ	chip compression ratio, t_2/t_1
μ	coefficient of friction
ψ	tool angle
τ	shear stress
ϕ	shear angle
Ω	relief angle
A_o	cross section, wt_1
e_m	machine efficiency factor
f	feed rate ipr (in./revolution), ips (in./stroke), mm/rev (mm/revolution), or mm/stroke
f_t	feed rate (in./tooth, mm/tooth) for milling and broaching
F	feed rate, in./min (mm/sec)
F_c	cutting force
F_f	friction force
F_n	normal force on shear plane
F_s	shear force
F_t	thrust force
HP_c	cutting horsepower
Hp_g	gross horsepower

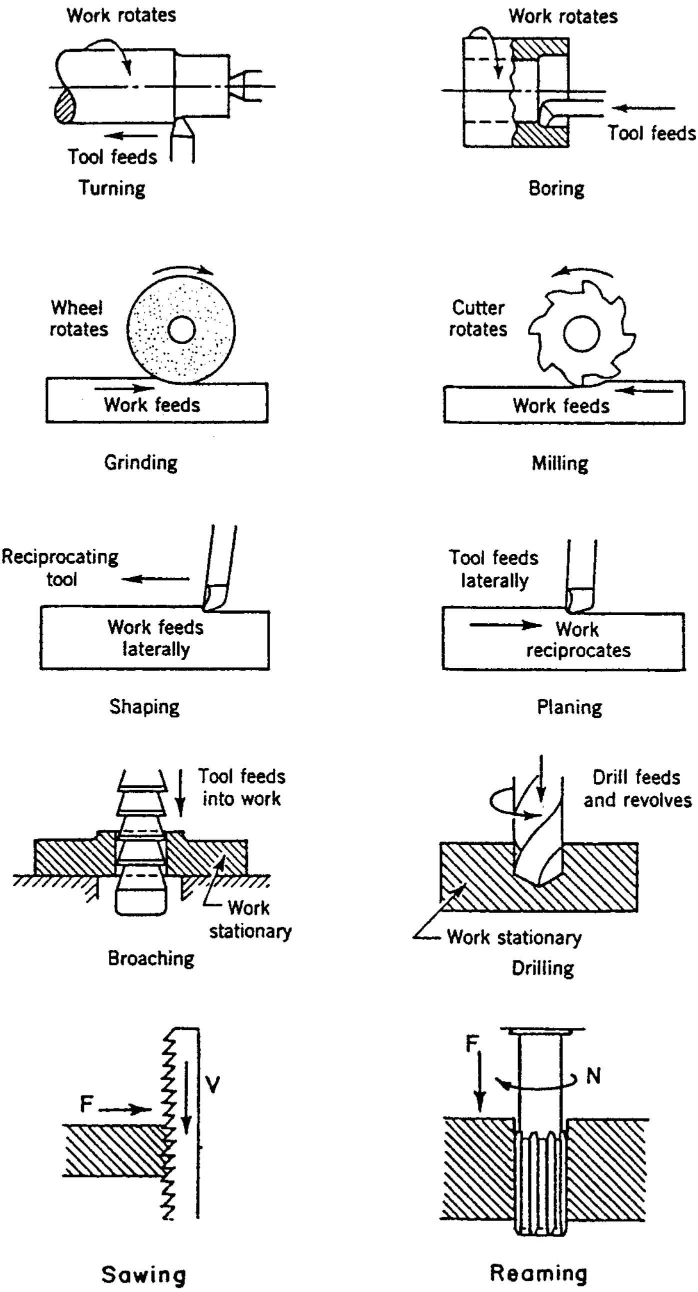


Fig. 33.1 Conventional machining processes.

Hp_{μ}	unit horsepower
N	revolutions per minute
Q	rate of metal removal, in.^{3}/min
R	resultant force
T	tool life in minutes
t_1	depth of cut

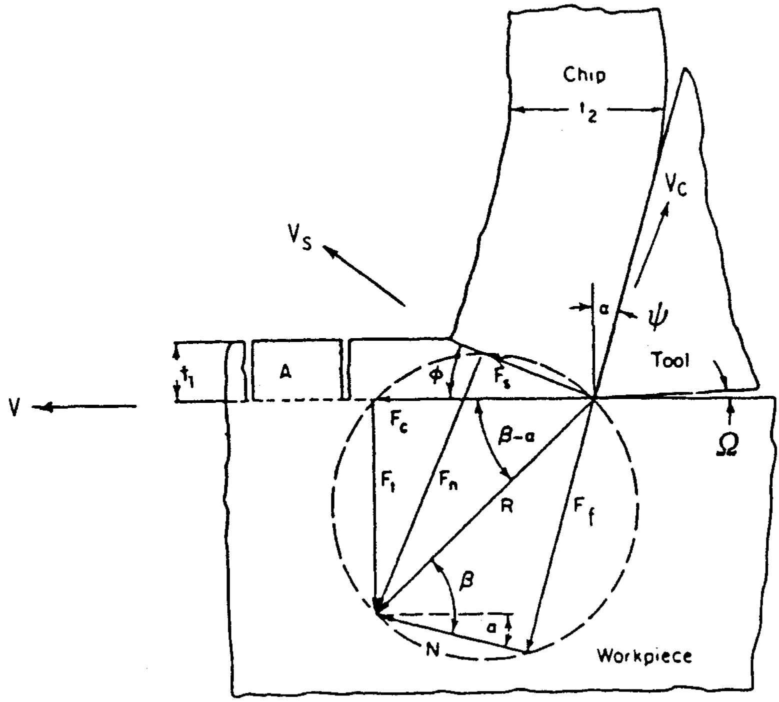


Fig. 33.2 Mechanics of metal-cutting process.

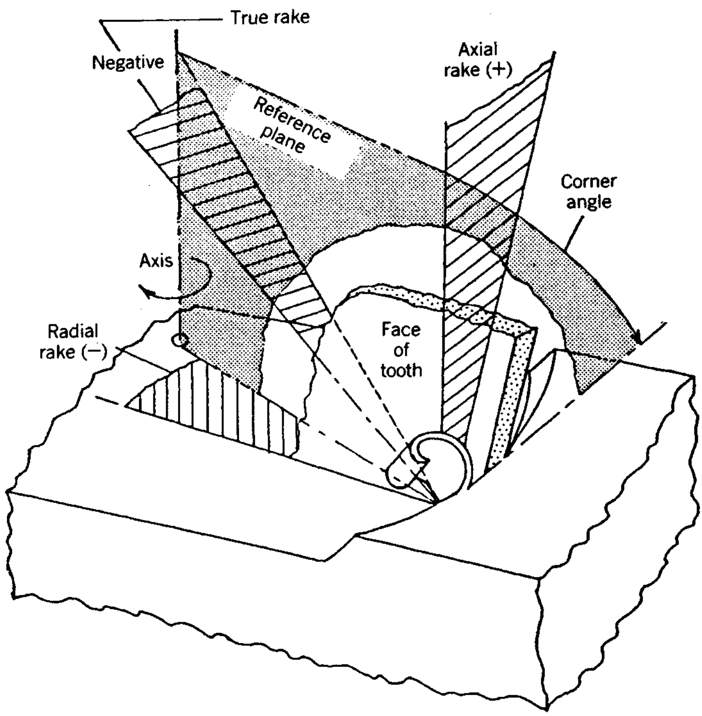


Fig. 33.3 Oblique cutting.

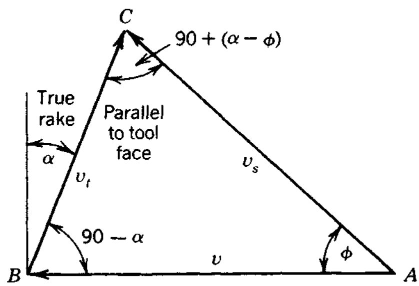


Fig. 33.4 Cutting velocities.

t_2 chip thickness
V cutting speed, ft/min
V_c chip velocity
V_s shear velocity

The *shear angle* ϕ controls the thickness of the chip and is given by

$$\tan \phi = \frac{\cos \alpha}{\lambda - \sin \alpha} \tag{33.1}$$

The *strain* γ that the material undergoes in shearing is given by

$$\gamma = \cot \phi + \tan(\phi - \alpha)$$

The *coefficient of friction* μ on the face of the tool is

$$\mu = \frac{F_t + F_c \tan \alpha}{F_c - F_t \tan \alpha} \tag{33.2}$$

The *friction force* F_t along the tool is given by

$$F_t = F_t \cos \alpha + F_c \sin \alpha$$

Cutting forces are usually measured with dynamometers and/or wattmeters. The shear stress τ in the shear plane is

$$\tau = \frac{F_c \sin \phi \cos \phi - F_t \sin^2 \phi}{A}$$

The speed relationships are

$$\frac{V_c}{V} = \frac{\sin \phi}{\cos(\phi - \alpha)}$$
$$V_c = V/\lambda \tag{33.3}$$

33.2 MACHINING POWER AND CUTTING FORCES

Estimating the power required is useful when planning machining operations, optimizing existing ones, and specifying new machines. The power consumed in cutting is given by

$$\text{power} = F_c V \tag{33.4}$$

$$HP_c = \frac{F_c V}{33{,}000} \tag{33.5}$$

$$= Q\, HP_\mu \tag{33.6}$$

where F_c = cutting force, lb
V = cutting speed, ft per min = $\pi DN/12$ (*rotating operations*)
D = diameter, in.
N = revolutions per min
HP_μ = specific power required to cut a material at a rate of 1 cu in. per min
Q = material removal rate, cu in./min

For SI units,

$$\text{Power} = F_cV \quad \text{watts} \tag{33.7}$$

$$= QW \quad \text{watts} \tag{33.8}$$

where F_c = cutting force, newtons
V = m per sec = $2\pi RN$
W = specific power required to cut a material at a rate of 1 cu mm per sec
Q = material removal rate, cu mm per sec

The specific energies for different materials, using sharp tools, are given in Table 33.1.

$$\text{power} = F_cV = F_c2\pi RN$$

$$= F_cR2\pi N$$

$$= M2\pi N \tag{33.9}$$

$$= \frac{MN}{63{,}025} \quad HP \tag{33.10}$$

where M = torque, in.–lbf
N = revolutions per min

In SI units,

$$= \frac{MN}{9549} \quad KW \tag{33.11}$$

Table 33.1 Average Values of Energy per Unit Material Removal Rate

Material	Bhn	HP_c/in.^{3} per min	W/mm^3 per sec
Aluminum alloys	50–100	0.3	0.8
	100–150	0.4	1.1
Cast iron	125–190	0.5	1.6
	190–250	1.6	4.4
Carbon steels	150–200	1.1	3.0
	200–250	1.4	3.8
	250–350	1.6	4.4
Leaded steels	150–175	0.7	1.9
Alloy steels	180–250	1.6	4.4
	250–400	2.4	6.6
Stainless steels	135–275	1.5	4.1
Copper	125–140	1.0	2.7
Copper alloys	100–150	0.8	2.2
Leaded brass	60–120	0.7	1.9
Unleaded brass	50	1.0	2.7
Magnesium alloys	40–70	0.2	0.55
	70–160	0.4	1.1
Nickel alloys	100–350	2.0	5.5
Refractory alloys (Tantalum, Columbium, Molybdenum)	210–230	2.0	5.5
Tungsten	320	3.0	8.0
Titanium alloys	250–375	1.3	3.5

where M = newton–meter
HP/cu in./min 2.73 = ? W/(cu mm/sec)
$M = F_cR$ = power$/2\pi N$
$F_c = M/R$

$$\text{gross power} = \text{cutting power}/e_m \tag{33.13}$$

The cutting horsepowers for different machining operations are given below.
For turning, planing, and shaping,

$$HP_c = (HP_\mu)12CWVfd \tag{33.14}$$

For milling,

$$HP_c = (HP_\mu)CWFwd \tag{33.15}$$

For drilling,

$$HP_c = (HP_\mu)CW(N)f\left(\frac{\pi D^2}{4}\right) \tag{33.16}$$

For broaching,

$$HP_c = (HP_\mu)12CWVn_cwd_t \tag{33.17}$$

where V = cutting speed, fpm
C = feed correction factor
f = feed, ipr (turning and drilling), ips (planing and shaping)
F = feed, ipm = $f \times N$
d = depth of cut, in.
d_t = maximum depth of cut per tooth, in.
n_c = number of teeth engaged in work
w = width of cut, in.
W = tool wear factor

Specific energy is affected by changes in feed rate. Table 33.2 gives feed correction factor (C). Cutting speed and depth of cut have no significant effect on power. Tool wear effect factor (W) is given in Table 33.3.

The gross power is calculated by applying the overall efficiency factor (e_m).

33.3 TOOL LIFE

Tool life is a measure of the length of time a tool will cut satisfactorily, and may be measured in different ways. Tool wear, as in Fig. 33.5, is a measure of tool failure if it reaches a certain limit. These limits are usually 0.062 in. (1.58 mm) for high-speed tools and 0.030 in. (0.76 mm) for carbide tools. In some cases, the life is determined by surface finish deterioration and an increase in cutting

Table 33.2 Feed Correction (C)

Factors for Turning, Milling, Drilling, Planing, and Shaping		
Feed (ipr or ips)	mm/rev or mm/stroke	Factor
0.002	0.05	1.4
0.005	0.12	1.2
0.008	0.20	1.05
0.012	0.30	1.0
0.020	0.50	0.9
0.030	0.75	0.80
0.040	1.00	0.80
0.050	1.25	0.75

Table 33.3 Tool Wear Factors (*W*)

Type of Operations[a]	*W*
Turning	
Finish turning (light cuts)	1.10
Normal rough and semifinish turning	1.30
Extra-heavy-duty rough turning	1.60–2.00
Milling	
Slab milling	1.10
End milling	1.10
Light and medium face milling	1.10–1.25
Extra-heavy-duty face milling	1.30–1.60
Drilling	
Normal drilling	1.30
Drilling hard-to-machine materials and drilling with a very dull drill	1.50
Broaching	
Normal broaching	1.05–1.10
Heavy-duty surface broaching	1.20–1.30

[a]For all operations with sharp cutting tools.

forces. The cutting speed is the variable that has the greatest effect on tool life. The relationship between tool life and cutting speed is given by the Taylor equation.

$$VT^n = C \tag{33.18}$$

where V = cutting speed, fpm (m/sec)
T = tool life, min (sec)
n = exponent depending on cutting condition
C = constant, the cutting speed for a tool life of 1 min

Table 33.4 gives the approximate ranges for the exponent n. Taylor's equation is equivalent to

$$\log V = C - n \log T \tag{33.19}$$

which when plotted on log–log paper gives a straight line, as shown in Fig. 33.6.

Equation (33.20) incorporates the size of cut:

$$K = VT^n f^{n1} d^{n2} \tag{33.20}$$

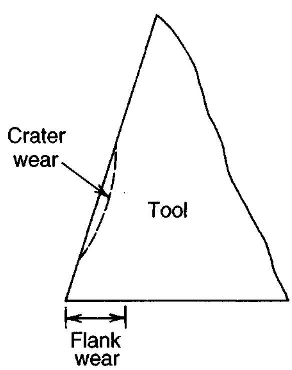


Fig. 33.5 Types of tool wear.

Table 33.4 Average Values of n

Tool Material	Work Material	n
HSS (18-4-1)	Steel	0.15
	C.I.	0.25
	Light metals	0.40
Cemented carbide	Steel	0.30
	C.I.	0.25
Sintered carbide	Steel	0.50
Ceramics	Steel	0.70

Average values for $n_1 = .5\text{–}.8$
$n_2 = .2\text{–}.4$

Equation (33.21) incorporates the hardness of the workpiece:

$$K = VT^n f^{n1} d^{n2} (BHN)^{1.25} \tag{33.21}$$

33.4 METAL-CUTTING ECONOMICS

The efficiency of machine tools increases as cutting speeds increase, but tool life is reduced. The main objective of metal-cutting economics is to achieve the optimum conditions, that is, the minimum cost while considering the principal individual costs: machining cost, tool cost, tool-changing cost, and handling cost. Figure 33.7 shows the relationships among these four factors.

$$\text{machining cost} = C_o t_m \tag{33.22}$$

where C_o = operating cost per minute, which is equal to the machine operator's rate plus appropriate overhead
t_m = machine time in minutes, which is equal to $L/(fN)$, where L is the axial length of cut

$$\text{tool cost per operation} = C_t \frac{t_m}{T} \tag{33.23}$$

where C_t = tool cost per cutting edge
T = tool life, which is equal to $(C/V)^{1/n}$

$$\text{tool changing cost} = C_o t_c (t_m/T) \tag{33.24}$$

where t_c = tool changing time, min

$$\text{handling cost} = C_o t_h$$

where t_h = handling time, min

The average unit cost C_u will be equal to

$$C_u = C_o t_m + \frac{t_m}{T}(C_t + C_o t_c) + C_o t_h \tag{33.25}$$

33.4.1 Cutting Speed for Minimum Cost (V_{min})

Differentiating the costs with respect to cutting speed and setting the results equal to zero will result in V_{min}:

$$V_{min} = \frac{C}{\left(\frac{1}{n} - 1\right)\left(\frac{C_o t + C_t}{C_o}\right)^n} \tag{33.26}$$

33.4.2 Tool Life Minimum Cost (T_m)

Since the constant C is the same in Taylor's equation and Eq. (33.23), and if V corresponds to V_{min}, then the tool life that corresponds to the cutting speed for minimum cost is

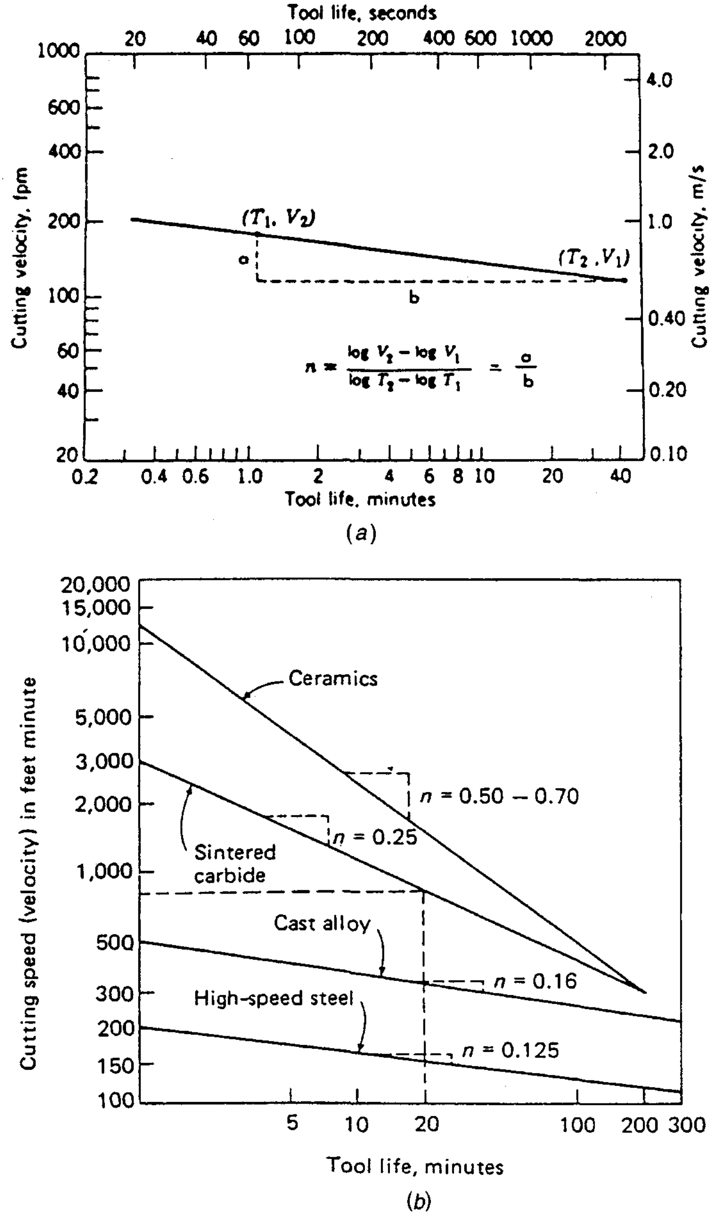


Fig. 33.6 Cutting speed/tool life relationship.

$$T_{\min} = \left(\frac{1}{n} - 1\right)\left(\frac{C_o t_c + C_t}{C_o}\right) \tag{33.27}$$

33.4.3 Cutting Speed for Maximum Production ($V_{\max}$)

This speed can be determined from Eq. (33.26) for the cutting speed for minimum cost by assuming that the tool cost is negligible, that is, by setting $C_1 = 0$:

$$V_{\max} = \frac{C}{\left[\left(\frac{1}{n} - 1\right) t_c\right]^n} \tag{33.28}$$

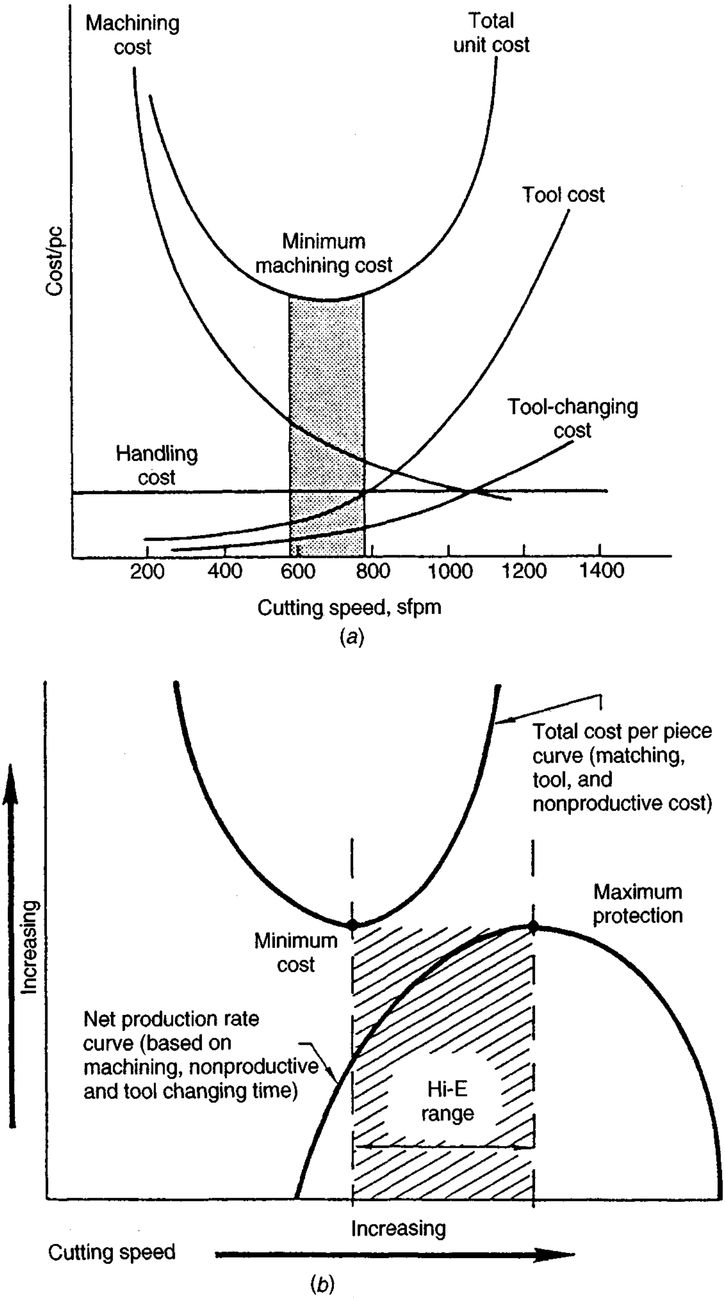


Fig. 33.7 Cost factors.

33.4.4 Tool Life for Maximum Production (T_{max})

By analogy to Taylor's equation, the tool life that corresponds to the maximum production rate is given by

$$T_{max} = \left(\frac{1}{n} - 1\right) t_c \tag{33.29}$$

33.5 CUTTING-TOOL MATERIALS

The desirable properties for any tool material include the ability to resist softening at high temperature, which is known as red hardness; a low coefficient of friction; wear resistance; sufficient toughness and shock resistance to avoid fracture; and inertness with respect to workpiece material.

The principal materials used for cutting tools are carbon steels, cast nonferrous alloys, carbides, ceramic tools or oxides, and diamonds.

1. *High-carbon steels* contain (0.8–1.2%) carbon. These steels have good hardening ability, and with proper heat treatment hold a sharp cutting edge where excessive abrasion and high heat are absent. Because these tools lose hardness at around 600°F (315°C), they are not suitable for high speeds and heavy-duty work.
2. *High-speed steels* (HSS) are high in alloy contents such as tungsten, chromium, vanadium, molybdenum, and cobalt. High-speed steels have excellent hardenability and will retain a keen cutting edge to temperatures around 1200°F (650°C).
3. *Cast nonferrous alloys* contain principally chromium, cobalt, and tungsten, with smaller percentages of one or more carbide-forming elements, such as tantalum, molybdenum, or boron. Cast-alloy tools can maintain good cutting edges at temperatures up to 1700°F (935°C) and can be used at twice the cutting speed as HSS and still maintain the same feed. Cast alloys are not as tough as HSS and have less shock resistance.
4. *Carbides* are made by powder-metallurgy techniques. The metal powders used are tungsten carbide (WC), cobalt (Co), titanium carbide (TiC), and tantalum carbide (TaC) in different ratios. Carbide will maintain a keen cutting edge at temperatures over 2200°F (1210°C) and can be used at speeds two or three times those of cast alloy tools.
5. *Coated tools,* cutting tools, and inserts are coated by titanium nitride (TiN), titanium carbide (TiC), titanium carbonitride (TiCN), aluminum oxide (Al_2O_3), and diamond. Cutting speeds can be increased by 50% due to coating.
6. *Ceramic or oxide tool* inserts are made from aluminum oxide (Al_2O_3) grains with minor additions of titanium, magnesium, or chromium oxide by powder-metallurgy techniques. These inserts have an extremely high abrasion resistance and compressive strength, lack affinity for metals being cut, resistance to cratering and heat conductivity. They are harder than cemented carbides but lack impact toughness. The ceramic tool softening point is above 2000°F (1090°C) and these tools can be used at high speeds (1500–2000 ft/min) with large depth of cut. Ceramic tools have tremendous potential because they are composed of materials that are abundant in the earth's crust. Optimum cutting conditions can be achieved by applying negative rank angles (5–7°), rigid tool mountings, and rigid machine tools.
7. *Cubic boron nitride* (CBN) is the hardest material presently available, next to diamond. CBN is suitable for machining hardened ferrous and high-temperature alloys. Metal removal rates up to 20 times those of carbide cutting tools were achieved.
8. *Single-crystal diamonds* are used for light cuts at high speeds of 1000–5000 fpm to achieve good surface finish and dimensional accuracy. They are used also for hard materials difficult to cut with other tool material.
9. *Polycrystalline diamond* cutting tools consist of fine diamond crystals, natural or synthetic, that are bonded together under high pressure and temperature. They are suitable for machining nonferrous metals and nonmetallic materials.

33.5.1 Cutting-Tool Geometry

The shape and position of the tool relative to the workpiece have a very important effect in metal cutting. There are six single-point tool angles critical to the machining process. These can be divided into three groups.

Rake angles affect the direction of chip flow, the characteristics of chip formation, and tool life. Positive rake angles reduce the cutting forces and direct the chip flow away from the material. Negative rake angles increase cutting forces but provide greater strength, as is recommended for hard materials.

Relief angles avoid excessive friction between the tool and workpiece and allow better access of coolant to tool–work interface.

The *side cutting-edge angle* allows the full load of the cut to be built up gradually. The *end cutting-edge angle* allows sufficient clearance so that the surface of the tool behind the cutting point will not rub over the work surface.

The purpose of the *nose radiuses* is to give a smooth surface finish and to increase the tool life by increasing the strength of the cutting edge. The elements of the single-point tool are written in the following order: back rake angle, side rake angle, end relief angle, side relief angle, end cutting-edge angle, side cutting-edge angle, and nose radius. Figure 33.8 shows the basic tool geometry.

Cutting tools used in various machining operations often appear to be very different from the single-point tool in Figure 33.8. Often they have several cutting edges, as in the case of drills, broaches, saws, and milling cutters. Simple analysis will show that such tools are comprised of a number of single-point cutting edges arranged so as to cut simultaneously or sequentially.

33.5.2 Cutting Fluids

The major roles of the cutting fluids—liquids or gases—are

1. Removal of the heat friction and deformation
2. Reduction of friction among chip, tool, and workpiece
3. Washing away chips
4. Reduction of possible corrosion on both workpiece and machine
5. Prevention of built-up edges

Cutting fluids work as coolants and lubricants. Cutting fluids applied depend primarily on the kind of material being used and the type of operation. The four major types of cutting fluids are

1. Soluble oil emulsions with water-to-oil ratios of 20:1 to 80:1
2. Oils
3. Chemicals and synthetics
4. Air

At low cutting speeds (40 ft/min and below), oils are highly recommended, especially in tapping, reaming, and gear and thread machining. Cutting fluids with the maximum specific heat, such as soluble oil emulsions, are recommended at high speeds.

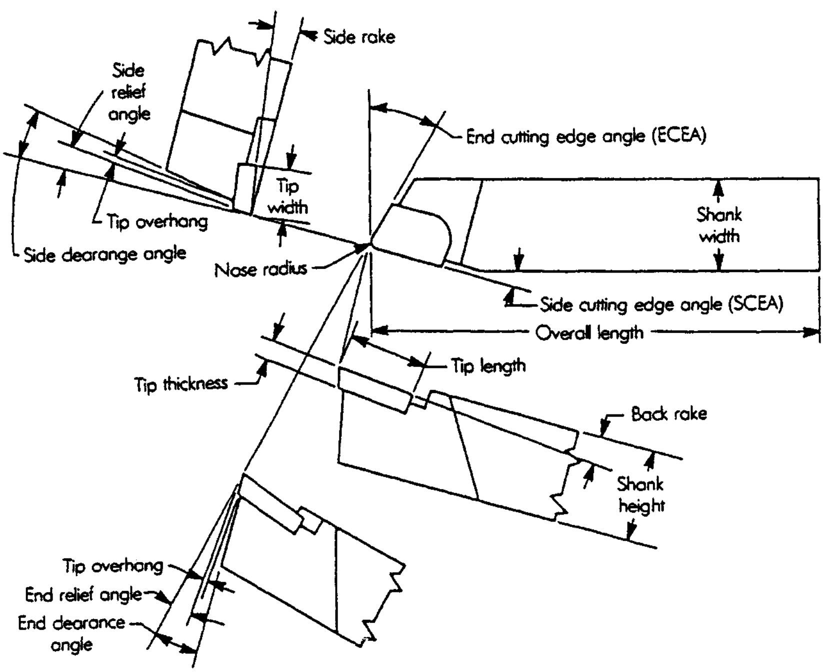


Fig. 33.8 Basic tool geometry.

33.5.3 Machinability

Machinability refers to a system for rating materials on the basis of their relative ability to be machined easily, long tool life, low cutting forces, and acceptable surface finish. Additives such as lead, manganese sulfide, or sodium sulfide with percentages less than 3% can improve the machinability of steel and copper-based alloys, such as brass and bronze. In aluminum alloys, additions up to 1–3% of zinc and magnesium improve their machinability.

33.5.4 Cutting Speeds and Feeds

Cutting speed is expressed in feet per minute (m/sec) and is the relative surface speed between the cutting tool and the workpiece. It may be expressed by the simple formula $CS = \pi DN/12$ fpm in., where D is the diameter of the workpiece in inches in case of turning or the diameter of the cutting tool in case of drilling, reaming, boring, and milling, and N is the revolutions per minute. If D is given in millimeters, the cutting speed is $CS = \pi DN/60{,}000$ m/sec.

Feed refers to the rate at which a cutting tool advances along or into the surface of the workpiece. For machines in which either the workpiece or the tool turns, feed is expressed in inches per revolution (ipr) (mm/rev). For reciprocating tools or workpieces, feed is expressed in inches per stroke (ips) (mm/stroke).

The recommended cutting speeds, and depth of cut that resulted from extensive research, for different combinations of tools and materials under different cutting conditions can be found in many references, including Society of Manufacturing Engineers (SME) publications such as *Tool and Manufacturing Engineers Handbook*;[1] *Machining Data Handbook*;[2] Metcut Research Associates, Inc.; *Journal of Manufacturing Engineers*; *Manufacturing Engineering Transactions*; *American Society for Metals (ASM) Handbook*;[3] *American Machinist's Handbook*;[4] *Machinery's Handbook*;[5] American Society of Mechanical Engineering (ASME) publications; Society of Automotive Engineers (SAE) Publications; and *International Journal of Machine Tool Design and Research.*

33.6 TURNING MACHINES

Turning is a machining process for generating external surfaces of revolution by the action of a cutting tool on a rotating workpiece, usually held in a lathe. Figure 33.9 shows some of the external operations that can be done on a lathe. When the same action is applied to internal surfaces of revolution, the process is termed *boring.* Operations that can be performed on a lathe are turning, facing, drilling, reaming, boring, chamfering, taping, grinding, threading, tapping, and knurling.

The primary factors involved in turning are speed, feed, depth of cut, and tool geometry. Figure 33.10 shows the tool geometry along with the feed (f) and depth of cut (d). The cutting speed (CS) is the surface speed in feet per minute (sfm) or meters per sec (m/s). The feed (f) is expressed in inches of tool advance per revolution of the spindle (ipr) or (mm/rev). The depth of cut (d) is expressed in inches. Table 33.5 gives some of the recommended speeds while using HSS tools and carbides for the case of finishing and rough machining. The cutting speed (fpm) is calculated by

$$CS = \frac{\pi DN}{12} \text{ fpm} \tag{33.30}$$

where D = workpiece diameter, in.
N = spindle revolutions per minute

For SI units,

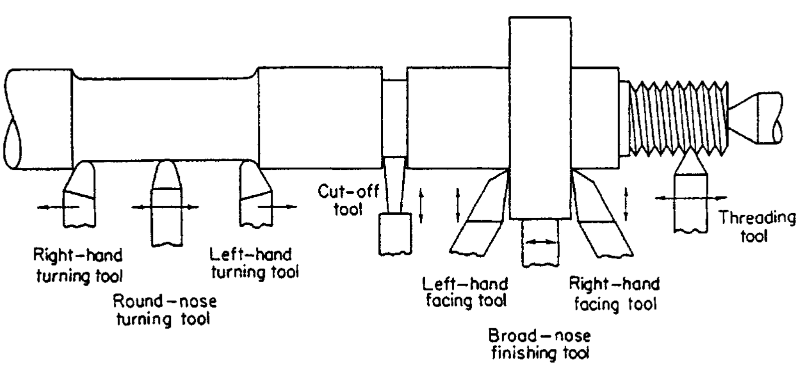


Fig. 33.9 Common lathe operations.

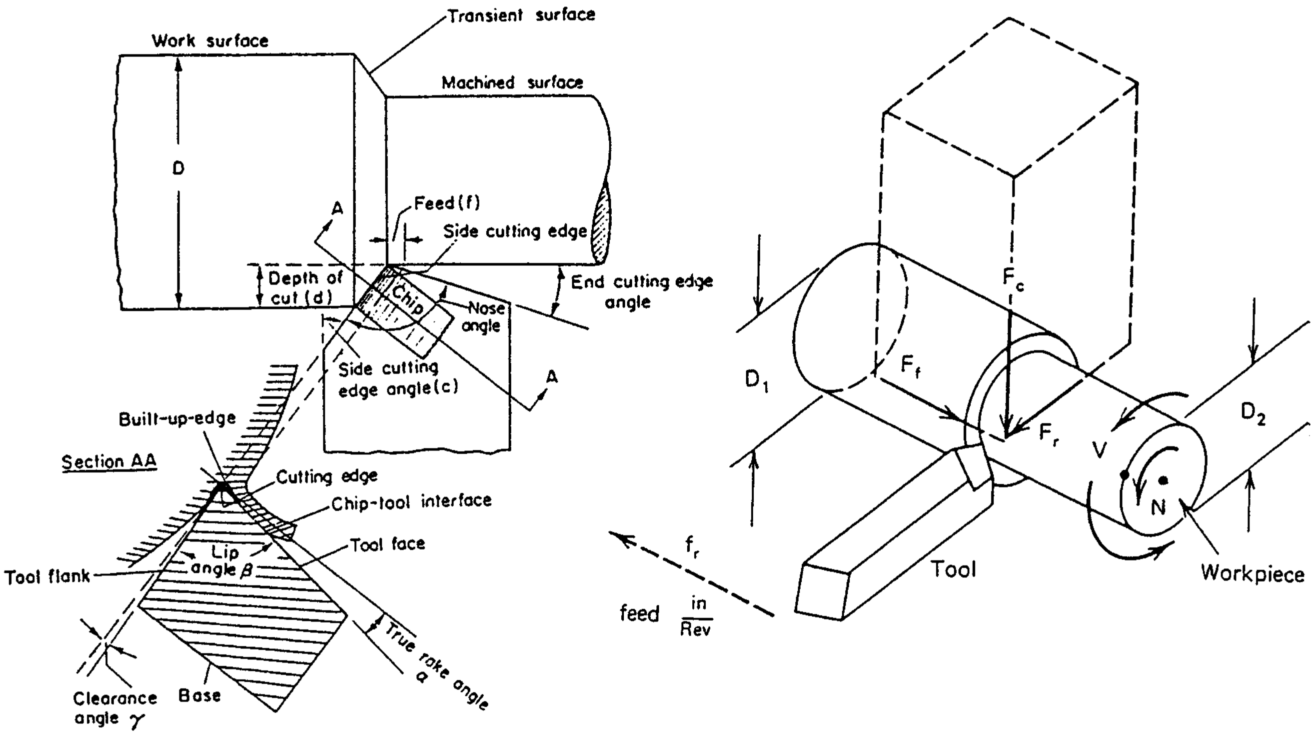


Fig. 33.10 Tool geometry—external turning.

Table 33.5 Typical Cutting Speeds ft/min (m/sec)

	High-Speed Steel		Carbide	
Material	Finish[a]	Rough[b]	Finish[a]	Rough[b]
Free cutting steels, 1112, 1315	250–350 (1.3–1.8)	80–160 (0.4–0.8)	600–750 (3.0–3.8)	350–500 (1.8–2.5)
Carbon steels, 1010, 1025	225–300 (1.1–1.5)	80–130 (0.4–0.6)	550–700 (2.8–3.5)	300–450 (1.5–2.3)
Medium steels, 1030, 1050	200–300 (1.0–1.5)	70–120 (0.4–0.6)	450–600 (2.3–3.0)	250–400 (1.3–2.0)
Nickel steels, 2330	200–300 (1.0–1.5)	70–110 (0.4–0.6)	425–550 (2.1–2.8)	225–350 (1.1–1.8)
Chromium nickel, 3120, 5140	150–200 (0.8–1.0)	60–80 (0.3–0.4)	325–425 (1.7–2.1)	175–300 (0.9–1.5)
Soft gray cast iron	120–150 (0.6–0.8)	80–100 (0.4–0.5)	350–450 (1.8–2.3)	200–300 (1.0–1.5)
Brass, normal	275–350 (1.4–1.8)	150–225 (0.8–1.1)	600–700 (3.0–3.5)	400–600 (2.0–3.0)
Aluminum	225–350 (1.1–1.8)	100–150 (0.5–0.8)	450–700 (2.3–3.5)	200–350 (1.0–1.8)
Plastics	300–500 (1.5–2.5)	100–200 (0.5–1.0)	400–650 (2.0–3.3)	150–300 (0.8–1.5)

[a]Cut depth, 0.015–0.10 in. (0.38–2.54 mm); feed 0.005–0.015 ipr (0.13–0.38 mm/rev).
[b]Cut depth, 0.20–0.40 in. (5.0–10.0 mm); feed, 0.030–0.060 ipr (0.75–1.5 mm/rev).

$$CS = \frac{\pi DN}{1000} \quad \text{m/s} \tag{33.31}$$

where D is in mm
N is in revolutions per second

The tool advancing rate is $F = f \times N$ ipm (mm/sec). The machining time (T_1) required to turn a workpiece of length L in. (mm) is calculated from

$$T_1 = \frac{L}{F} \quad \text{min (sec)} \tag{33.32}$$

The machining time (T_2) required to face a workpiece of diameter D is given by

$$T_2 = \frac{D/2}{F} \quad \text{min (sec)} \tag{33.33}$$

The rate of metal removal (MRR) (Q) is given by

$$Q = 12fdCS \qquad \text{in.}^3/\text{min} \tag{33.34}$$

$$\text{Power} = QHP_{\mu} \qquad HP \tag{33.35}$$

$$\text{Power} = \text{Torque } 2\pi N$$

$$= \frac{\text{Torque} \times N}{63{,}025} \quad HP \tag{33.36}$$

where torque is in in.–lbf

For SI units,

$$\text{Power} = \frac{\text{Torque} \times N}{9549} \quad KW \tag{33.37}$$

where torque is in newton–meter and N in rev/min
torque $= F_c \times R$

$$F_c = \frac{\text{Torque}}{R} \tag{33.38}$$

where R = radius of workpiece

To convert to SI units,

$$\begin{aligned}
\text{HP} \times 746 &= \text{? Watt (W)} \\
\text{f (lb)} \times 4.448 &= \text{? newtons} \\
\text{torque (in.–lb)} \times 0.11298 &= \text{? newton–meter (Nm)} \\
\text{HP/(cu in./min)} \times 2.73 &= \text{? W/(cu mm/sec)} \\
\text{ft/min} \times .00508 &= \text{? m/sec} \\
\text{in.}^3 \times 16{,}390 &= \text{? mm}^3
\end{aligned}$$

Alignment charts were developed for determining metal removal rate and motor power in turning. Figures 33.11 and 33.12 show the method of using these charts either for English or metric units. The unit power (P) is the adjusted unit power with respect to turning conditions and machine efficiency.

33.6.1 Lathe Size

The size of a lathe is specified in terms of the diameter of the work it will swing and the workpiece length it can accommodate. The main types of lathes are engine, turret, single-spindle automatic, automatic screw machine, multispindle automatic, multistation machines, boring, vertical, and tracer. The level of automation can range from semiautomatic to tape-controlled machining centers.

33.6.2 Break-Even (BE) Conditions

The selection of a specific machine for the production of a required quantity q must be done in a way to achieve minimum cost per unit produced. The incremental setup cost is given by ΔC_t, C_1 is the machining cost per unit on the first machine, and C_2 is the machining cost for the second machine, the break-even point will be calculated as follows:

$$BE = \Delta C \sqrt{(C_1 - C_2)}$$

33.7 DRILLING MACHINES

Drills are used as the basic method of producing holes in a wide variety of materials. Figure 33.13 indicates the nomenclature of a standard twist drill and its comparison with a single-point tool. Knowledge of the thrust force and torque developed in the drilling process is important for design consideration. Figure 33.14 shows the forces developed during the drilling process. From the force diagram, the thrust force must be greater than $2P_y + P_y^1$ to include the friction on the sides and to be able to penetrate in the metal. The torque required is equal to P_2X. It is reported in the *Tool and Manufacturing Engineers Handbook*[1] that the following relations reasonably estimate the torque and thrust requirements of sharp twist drills of various sizes and designs.

Torque:

$$M = KF^{0.8}d^{1.8}A \quad \text{in.–lbf} \tag{33.39}$$

Thrust:

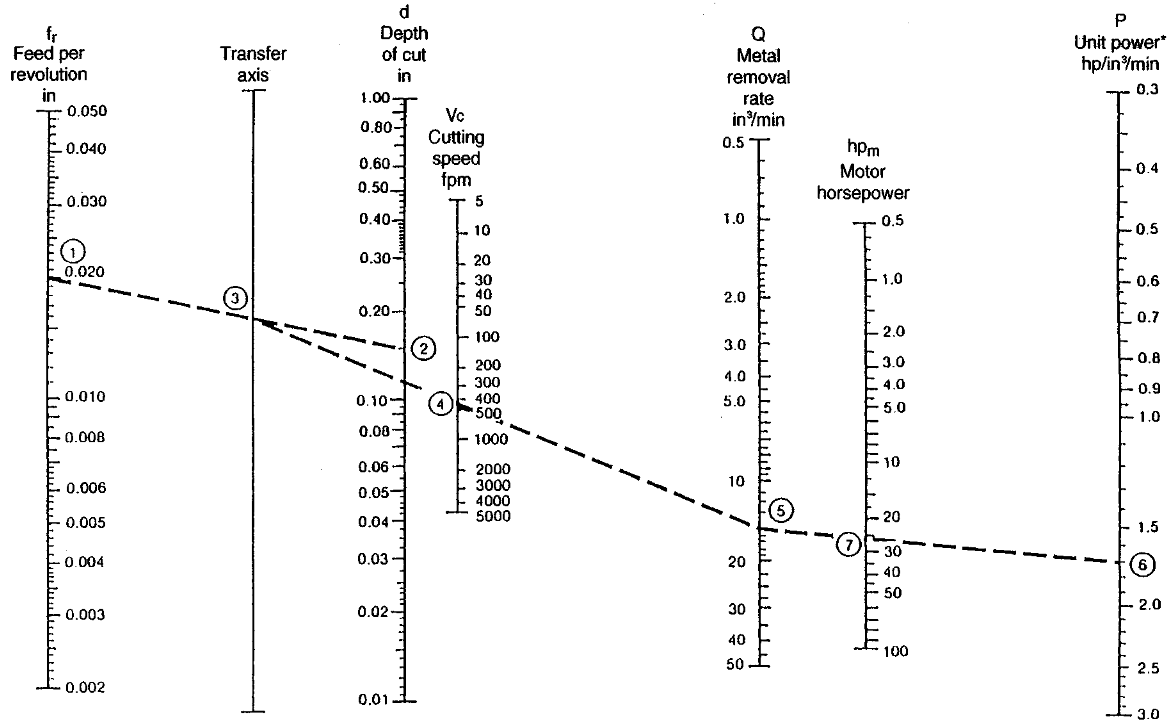
$$T = 2Kf^{0.8}d^{0.8}B + kd^2E \quad \text{lb} \tag{33.40}$$

The thrust force has a large effect upon the required strength, rigidity, and accuracy, but the power required to feed the tool axially is very small.

Cutting power:

$$HP = \frac{MN}{63{,}025} \tag{33.41}$$

where K = work–material constant
f = drill feed, ipr
d = drill diameter, in.
A, B, E = design constants
N = drill speed, rpm



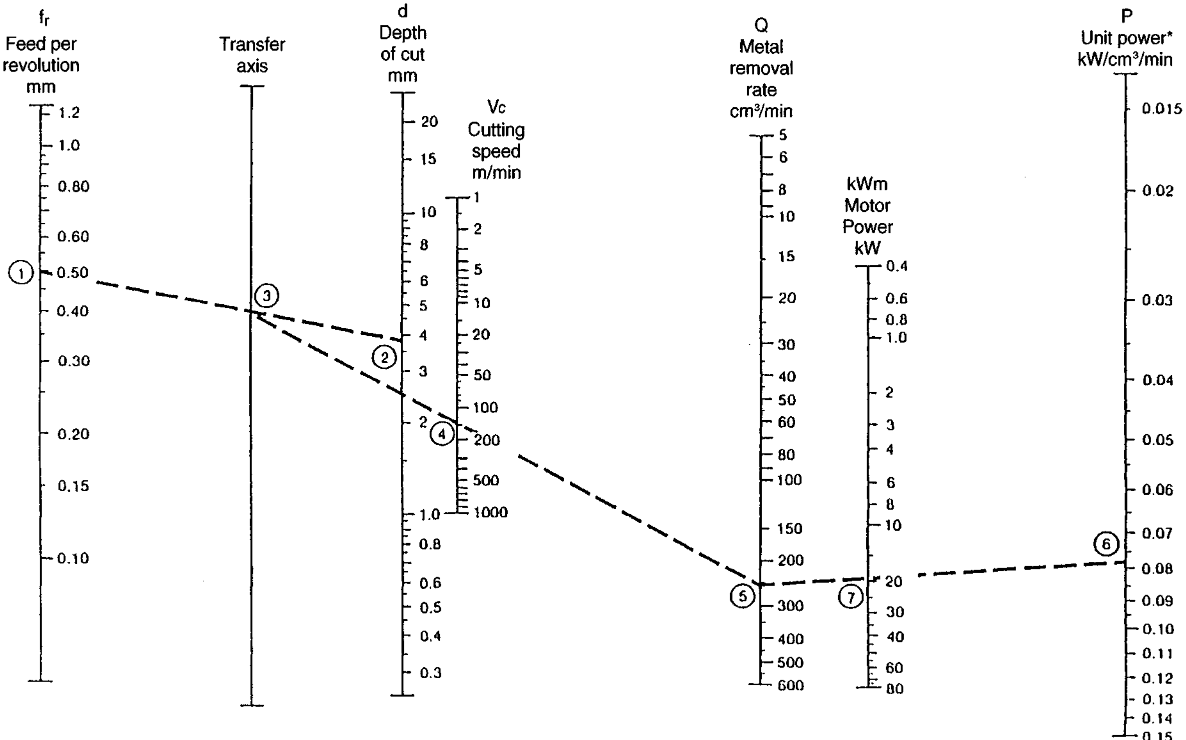
Horsepower = 12 x Feed x Depth X Speed X Unit Power

Example:
f_r = 0.020 in d = 0.150 in
V_c = 425 fpm P = 1.70 hp/in³/min

To Determine Motor Horsepower:

Connect Feed ① with Depth of cut ② to obtain point ③ on Transfer axis.
Connect ③ with Cutting speed ④ to obtain point ⑤ on Metal removal rate scale.
Connect point ⑤ with Unit power ⑥ to obtain 26 Motor horsepower at point ⑦.

Fig. 33.11 Alignment chart for determining metal removal rate and motor horsepower in turning—English units.



Power = Feed x Depth x Speed x Unit power

Example:
f_r = 0.5 mm d = 3.8 mm
V_c = 130 m/min P = 0.077 kW/cm³/min

To Determine Motor Horsepower:

Connect Feed ① with Depth of cut ② to obtain point ③ on Transfer axis.
Connect ③ with Cutting speed ④ to obtain point ⑤ on Metal removal rate scale.
Connect point ⑤ with Unit power ⑥ to obtain 19.02 kW at Motor, point ⑦.

Fig. 33.12 Alignment chart for determining metal removal rate and motor power in turning—metric units.

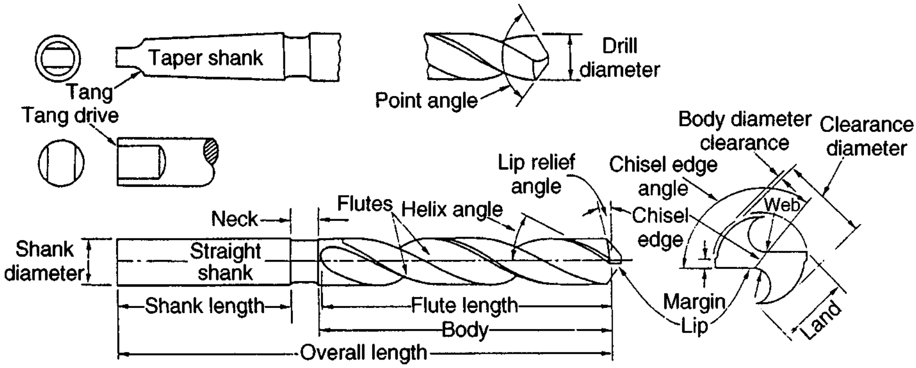


Fig. 33.13 Drill geometry.

Tables 33.6 and 33.7 give the constants used with the previous equations. Cutting speed at the surface is usually taken as 80% of turning speeds and is given by

$$CS = \frac{\pi d N}{12} \quad \text{fpm}$$

Force in cutting direction:

$$F_c = \frac{33{,}000\ HP}{CS} \quad \text{lb} \tag{33.42}$$

For SI units,

$$CS = \frac{\pi d_1 N}{60{,}000} \quad \text{m/sec} \tag{33.43}$$

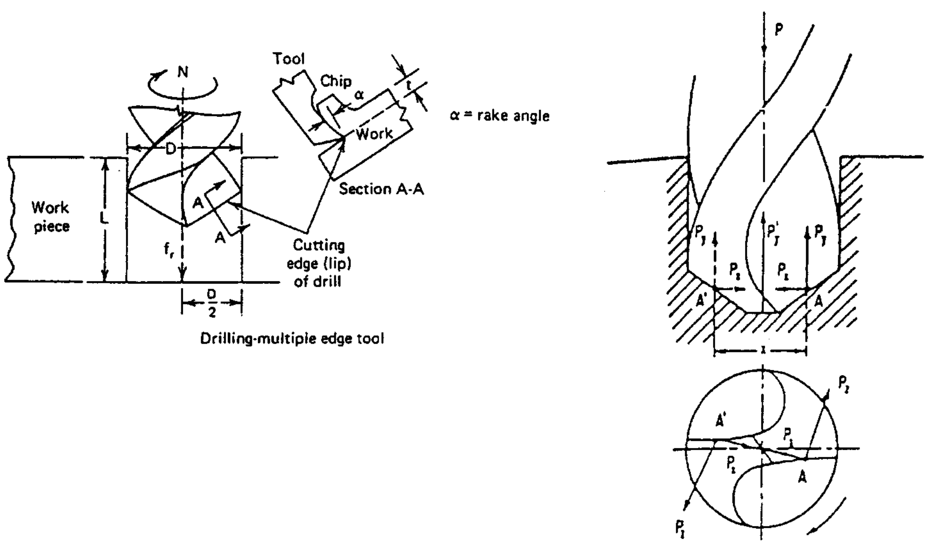


Fig. 33.14 Thrust forces and torque in drilling operation.

Table 33.6 Work-Material Constants for Calculating Torque and Thrust (National Twist Drill)

Work Material	K
Steel, 200 Bhn	24,000
Steel, 300 Bhn	31,000
Steel, 400 Bhn	34,000
Most aluminum alloys	7,000
Most magnesium alloys	4,000
Most brasses	14,000
Leaded brass	7,000
Cast iron, 65 Bhn	15,000
Free-machining mild steel, resulfurized	18,000
Austenitic stainless steel (type 316)	34,000

c = chisel-edge length, in.
d = drill diameter, in.
w = web thickness, in.
d_1 = drill diameter, in mm
Unit HP (hp/in.^{3}/min) $\times$ 2.73 = ? unit power (kW/cm^3/s)

$$\mathrm{kW} = \frac{MN}{9549} \tag{33.44}$$

$$M = \text{torque} \quad \text{Nm}$$

For drills of regular proportion the ratio c/d is = 0.18 and c = 1.15 w, approximately.

It is a common practice to feed drills at a rate that is proportional to the drill diameter in accordance with

$$f = \frac{d}{65} \tag{33.45}$$

For holes that are longer than 3d, feed should be reduced. Also feeds and speeds should be adjusted due to differences in relative chip volume, material structure, cutting fluid effectiveness, depth of hole, and conditions of drill and machine. The advancing rate is

Table 33.7 Torque and Thrust Constants Based on Ratios c/d or w/d (National Twist Drill)

c/d	w/d	Torque Constant A	Thrust Constant B	Thrust Constant E
0.03	0.025	1.000	1.100	0.001
0.05	0.045	1.005	1.140	0.003
0.08	0.070	1.015	1.200	0.006
0.10	0.085	1.020	1.235	0.010
0.13	0.110	1.040	1.270	0.017
0.15	0.130	1.080	1.310	0.022
0.18	0.155	1.085	1.355	0.030
0.20	0.175	1.105	1.380	0.040
0.25	0.220	1.155	1.445	0.065
0.30	0.260	1.235	1.500	0.090
0.35	0.300	1.310	1.575	0.120
0.40	0.350	1.395	1.620	0.160

$$F = f \times N \quad \text{ipm} \tag{33.46}$$

The recommended feeds are given in Table 33.8.

The time T required to drill a hole of depth h is given by

$$T = \frac{h + 0.3d}{F} \quad \text{min} \tag{33.47}$$

The extra distance of $0.3d$ is approximately equal to the distance from the tip to the effective diameter of the tool. The rate of metal removal in case of blind holes is given by

$$Q = \left(\frac{\pi d^2}{4}\right) F \quad \text{in.}^3/\text{min} \tag{33.48}$$

When torque is unknown, the horsepower requirement can be calculated by

$$HP_c = Q \times C \times W \times (HP_\mu) \quad \text{hp}$$

C, W, HP_μ are given in previous sections.

$$\begin{aligned} \text{Power} &= HP_c \times 396{,}000 \quad && \text{in.–lb/min} \\ \text{Torque} &= \frac{\text{Power}}{2\pi N} && \text{in–lbf} \\ F_c &= \frac{\text{Torque}}{R} && \text{lb} \end{aligned} \tag{33.49}$$

Along the cutting edge of the drill, the cutting speed is reduced toward the center as the diameter is reduced. The cutting speed is actually zero at the center. To avoid the region of very low speed and to reduce high thrust forces that might affect the alignment of the finished hole, a pilot hole is usually drilled before drilling holes of medium and large sizes. For the case of drilling with a pilot hole

$$\begin{aligned} Q &= \frac{\pi}{4}(d^2 - d_p^2)F \\ &= \frac{\pi}{4}(d + d_p)(d - d_p)F \quad \text{in.}^3/\text{min} \end{aligned} \tag{33.50}$$

Due to the elimination of the effects of the chisel-edge region, the equations for torque and thrust can be estimated as follows:

Table 33.8 Recommended Feeds for Drills

Diameter		Feed	
(in.)	(mm)	(ipr)	(mm/rev)
Under ⅛	3.2	0.001–0.002	0.03–0.05
⅓–¼	3.2–6.4	0.002–0.004	0.05–0.10
¼–½	6.4–12.7	0.004–0.007	0.10–0.18
½–1	12.7–25.4	0.007–0.015	0.18–0.38
Over 1	25.4	0.015–0.025	0.38–0.64

$$M_p = M \left[\frac{1 - \left(\frac{d_p}{d}\right)^2}{\left(1 + \frac{d_1}{d}\right)^{0.2}} \right] \tag{33.51}$$

$$T_p = T \left[\frac{1 - \frac{d_1}{d}}{\left(1 + \frac{d_1}{d}\right)^{0.2}} \right] \tag{33.52}$$

where d_p = pilot hole diameter

Alignment charts were developed for determining motor power in drilling. Figures 33.15 and 33.16 show the use of these charts either for English or metric units. The unit power* (P) is the adjusted unit power with respect to drilling conditions and machine efficiency.

For English units,

$$HP_m = \frac{\pi D^2}{4} \times f \times N \times P^*$$

$$N = \frac{12V}{\pi D}$$

As

$$HP_m = \frac{\pi D^2}{4} \times f \times \frac{12V}{\pi D} \times P^*$$

$$= 3\,D \times f \times V \times P^*$$

For metric units,

$$HP_m = \frac{\pi D^2}{4 \times 100} \times \frac{f}{10} \times N \times P$$

P^* in kW/cm³/min

$$N = \frac{1000V}{\pi D}$$

$$HP_m = \frac{\pi D^2}{4 \times 100} \times \frac{f}{10} \times \frac{100V}{\pi D} \times P^*$$

$$= 0.25\,D \times f \times V \times P^*$$

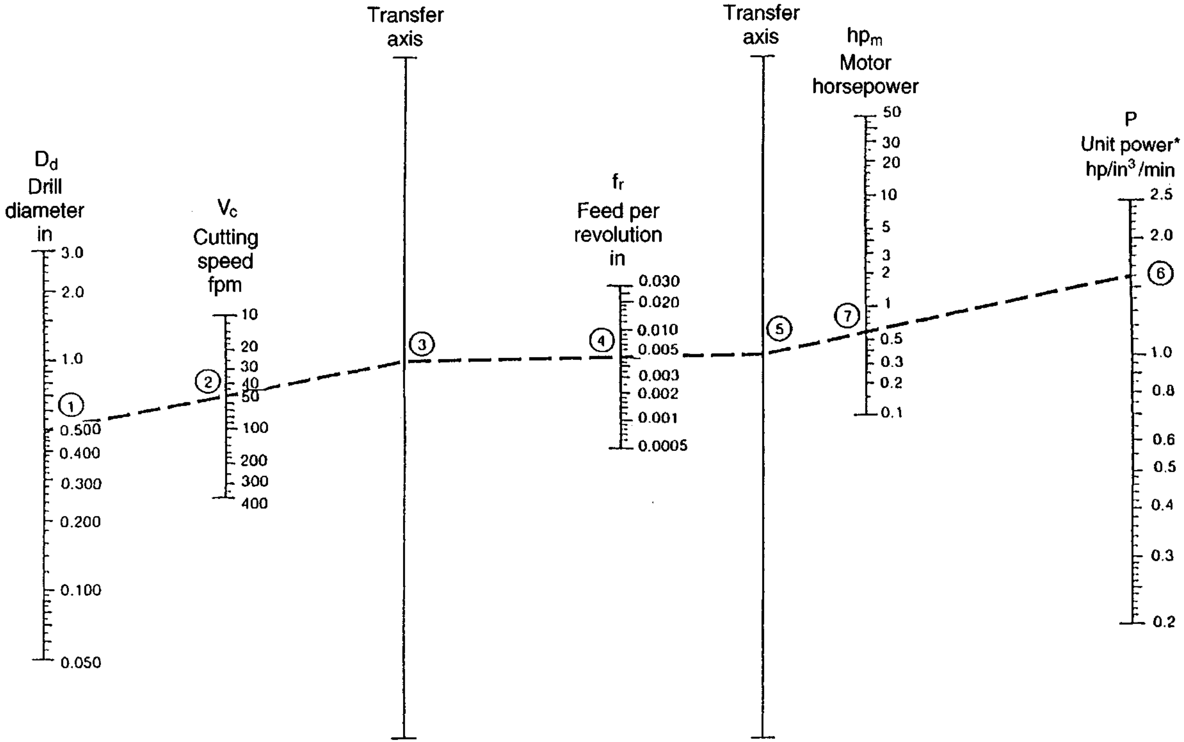
33.7.1 Accuracy of Drills

The accuracy of holes drilled with a two-fluted twist drill is influenced by many factors, including the accuracy of the drill point; the size of the drill, the chisel edge, and the jigs used; the workpiece material; the cutting fluid used; the rigidity and accuracy of the machine used; and the cutting speed. Usually, when drilling most materials, the diameter of the drilled holes will be oversize. Table 33.9 provided the results of tests reported by The Metal Cutting Tool Institute for holes drilled in steel and cast iron.

Gun drills differ from conventional drills in that they are usually made with a single flute. A hole provides a passageway for pressurized coolant, which serves as a means of both keeping the cutting edge cool and flushing out the chips, especially in deep cuts.

Spade drills (Fig. 33.17) are made by inserting a spade-shaped blade into a shank. Some advantages of spade drills are (1) efficiency in making holes up to 15 in. in diameter; (2) low cost, since only the insert is replaced; (3) deep hole drilling; and (4) easiness of chip breaking on removal.

Trepanning is a machining process for producing a circular hole, groove, disk, cylinder, or tube from solid stock. The process is accomplished by a tool containing one or more cutters, usually single-point, revolving around a center. The advantages of trepanning are (1) the central core left is solid material, not chips, which can be used in later work; and (2) the power required to produce a given hole diameter is highly reduced because only the annulus is actually cut.



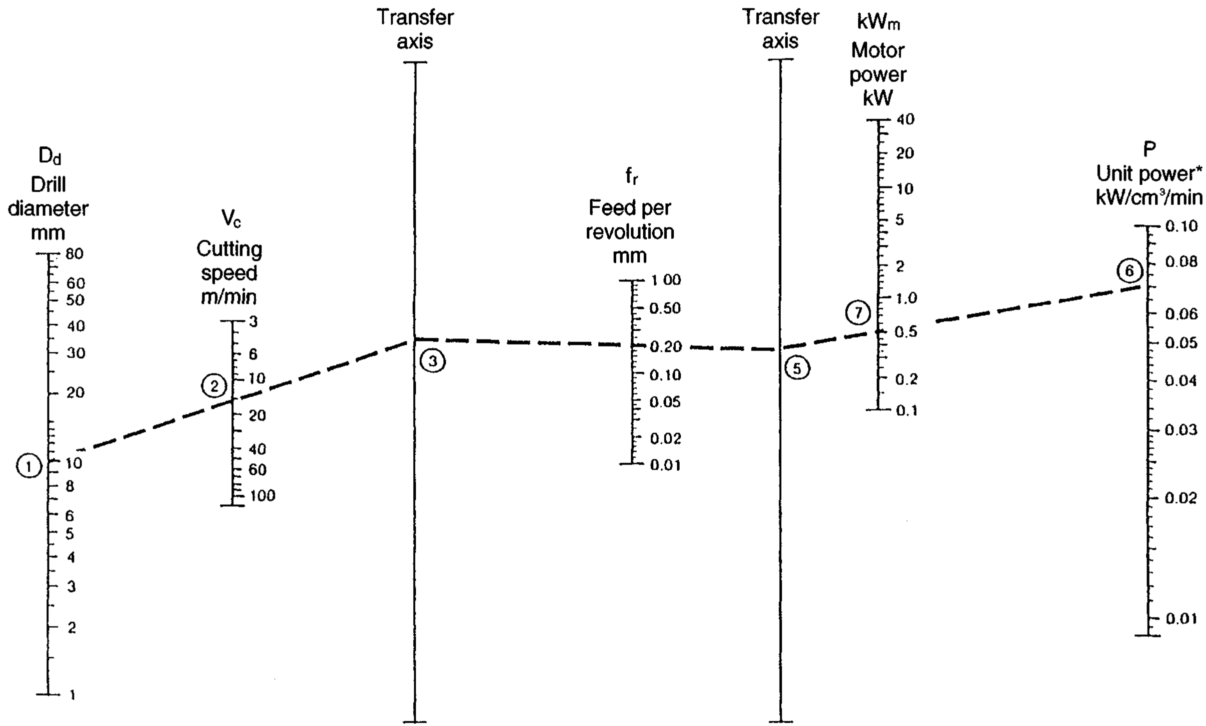
Horsepower = 3 x Drill diameter x Speed x Feed x Unit power

Example:
D_d = 0.500 in f_r = 0.005 in
V_c = 50 fpm P = 1.6 hp/in^3/min

To Determine Motor Horsepower:

Connect Drill diameter ① with Cutting speed ② to obtain point ③ on Transfer axis. Connect ③ with Feed ④ to obtain point ⑤ on Transfer axis. Connect ⑤ with Unit power ⑥ to obtain 0.60 Motor horsepower, point ⑦.

Fig. 33.15 Alignment chart for determining motor horsepower in drilling—English units.



Motor Power = 0.25 x Drill diameter x Speed x Feed x Unit Power

Example:

D_d = 10 mm f_r = 0.2 mm

V_c = 15 m/min P = 0.07 kW/cm³/min

To Determine Motor Horsepower:

Connect Drill diameter ① with Cutting speed ② to obtain point ③ on Transfer axis. Connect ③ with Feed ④ to obtain point ⑤ on Transfer axis. Connect ⑤ with Unit power ⑥ to obtain 0.53 kW at motor power, point ⑦.

Fig. 33.16 Alignment chart for determining motor power in drilling—metric units.

Table 33.9 Oversize Diameters in Drilling

	Amount Oversize (in.)		
Drill Diameter (in.)	Average Max.	Mean	Average Min.
1/16	0.002	0.0015	0.001
1/8	0.0045	0.003	0.001
1/4	0.0065	0.004	0.0025
1/2	0.008	0.005	0.003
3/4	0.008	0.005	0.003
1	0.009	0.007	0.004

Reaming, boring, counterboring, centering and countersinking, spotfacing, tapping, and chamfering processes can be done on drills. Microdrilling and submicrodrilling achieve holes in the range of 0.000025–0.20 in. in diameter.

Drilling machines are usually classified in the following manner:

1. Bench: plain or sensitive
2. Upright: single-spindle or turret
3. Radial
4. Gang
5. Multispindle
6. Deep-hole: vertical or horizontal
7. Transfer

33.8 MILLING PROCESSES

The milling machines use a rotary multitooth cutter that can be designed to mill flat or irregularly shaped surfaces, cut gears, generate helical shapes, drill, bore, or do slotting work. Milling machines are classified broadly as vertical or horizontal. Figure 33.18 shows some of the operations that are done on both types.

Feed in milling (F) is specified in inches per minute, but it is determined from the amount each tooth can remove or feed per tooth (f_t). The feed in./min is calculated from

$$F = f_t \times n \times N \quad \text{in./min} \tag{33.53}$$

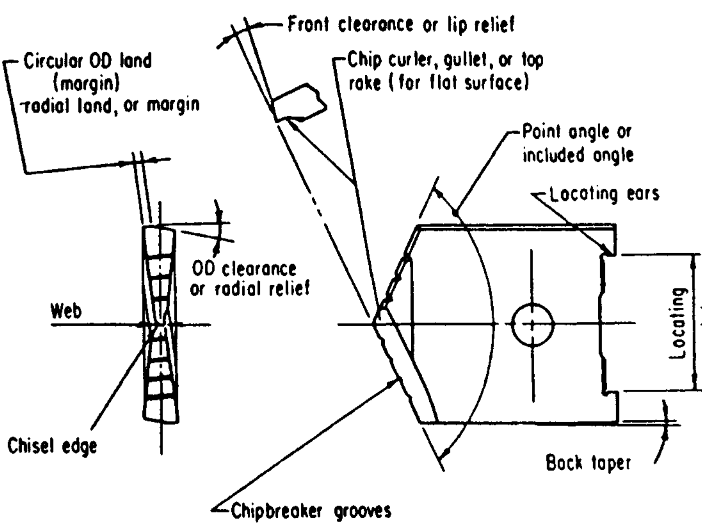


Fig. 33.17 Spade-drill blade elements.

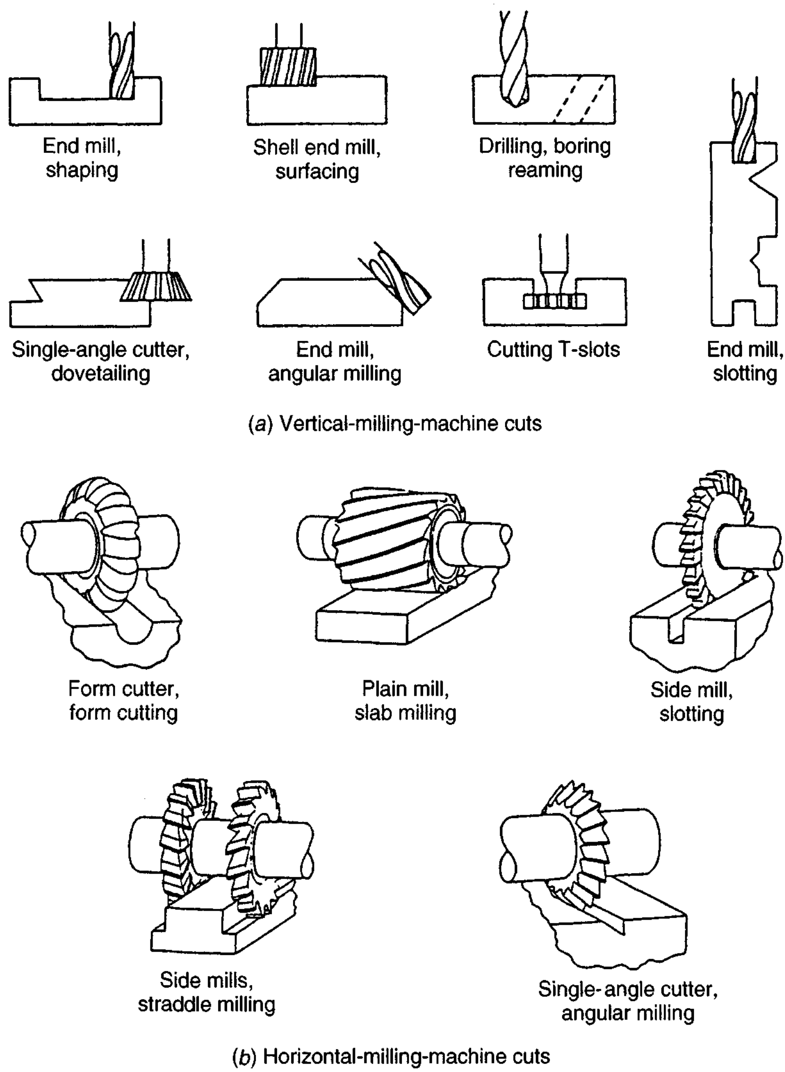


Fig. 33.18 Applications of (*a*) vertical; (*b*) horizontal milling machines.

where n = number of teeth in cutter
N = rpm

Table 33.10 gives the recommended f_t for carbides and HSS tools. The cutting speed CS is calculated as follows:

$$CS = \frac{\pi DN}{12} \quad \text{fpm}$$

where D = tool diameter, in.

Table 33.11 gives the recommended cutting speeds while using HSS and carbide-tipped tools. The relationship between cutter rotation and feed direction is shown in Fig. 33.19. In climb milling or down milling, the chips are cut to maximum thickness at initial engagement and decrease to zero thickness at the end of engagement. In conventional or up milling, the reverse occurs. Because of the initial impact, climb milling requires rigid machines with backlash eliminators.

Table 33.10 Recommended Feed per Tooth for Milling Steel with Carbide and HSS Cutters

	Feed per Tooth	
Type of Milling	Carbides	HSS
Face	0.008–0.015	0.010
Side or straddle	0.008–0.012	0.006
Slab	0.008–0.012	0008
Slotting	0.006–0.010	0.006
Slitting saw	0.003–0.006	0.003

The material removal rate (MRR) is $Q = F \times w \times d$, where w is the width of cut and d is the depth of cut. The horsepower required for milling is given by

$$HP_c = HP_\mu \times Q$$

Machine horsepower is determined by

$$HP_m = \frac{HP_c}{Eff.} + HP_i \tag{33.54}$$

where Hp_i = idle horsepower

Alignment charts were developed for determining metal removal rate (MRR) and motor power in face milling. Figures 33.21 and 33.22 show the method of using these charts either for English or metric units.

The time required for milling is equal to distance required to be traveled by the cutter to complete the cut (L_1) divided by the feed rate F. L_1 is equal to the length of cut (L) plus cutter approach A and the overtravel OT. The machining time T is calculated from

Table 33.11 Table of Cutting Speeds (sfpm)–Milling

	HSS Tools		Carbide-Tipped Tools	
Work Material	Rough Mill	Finish Mill	Rough Mill	Finish Mill
Cast iron	50–60	80–110	180–200	350–400
Semisteel	40–50	65–90	140–160	250–300
Malleable iron	80–100	110–130	250–300	400–500
Cast steel	45–60	70–90	150–180	200–250
Copper	100–150	150–200	600	1000
Brass	200–300	200–300	600–1000	600–1000
Bronze	100–150	150–180	600	1000
Aluminum	400	700	800	1000
Magnesium	600–800	1000–1500	1000–1500	1000–5000
SAE steels				
1020 (coarse feed)	60–80	60–80	300	300
1020 (fine feed)	100–120	100–120	450	450
1035	75–90	90–120	250	250
X-1315	175–200	175–200	400–500	400–500
1050	60–80	100	200	200
2315	90–110	90–110	300	300
3150	50–60	70–90	200	200
4150	40–50	70–90	200	200
4340	40–50	60–70	200	200
Stainless steel	60–80	100–120	240–300	240–300
Titanium	30–70		200–350	

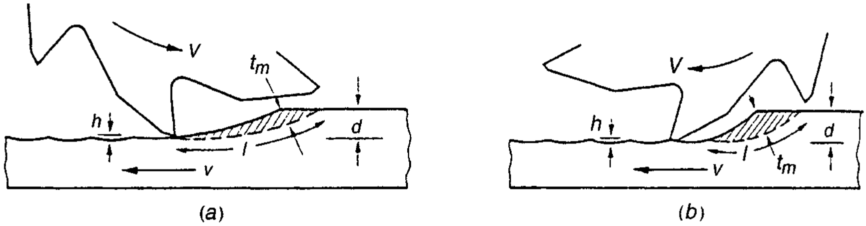


Fig. 33.19 Cutting action in up-and-down milling.

$$T = \frac{L + A + OT}{F} \text{ min} \tag{33.55}$$

OT depends on the specific milling operation.

The milling machines are designed according to the longitudinal table travel. Milling machines are built in different types, including:

1. Column-and-knee: vertical, horizontal, universal, and ram
2. Bed-type, multispindle
3. Planer
4. Special, turret, profilers, and duplicators
5. Numerically controlled

33.9 GEAR MANUFACTURING

Gears are made by various methods, such as machining, rolling, extrusion, blanking, powder metallurgy, casting, or forging. Machining still is the unsurpassed method of producing gears of all types and sizes with high accuracy. Roll forming can be used only on ductile materials; however, it has been highly developed and widely adopted in recent years. Casting, powder metallurgy, extruding, rolling, grinding, molding, and stamping techniques are used commercially in gear production.

33.9.1 Machining Methods

There are three basic methods for machining gears.

Form cutting uses the principle illustrated in Fig. 33.23. The equipment and cutters required are relatively simple, and standard machines, usually milling, are often used. Theoretically, there should be different-shaped cutters for each size of gear for a given pitch, as there is a slight change in the curvature of the involute. However, one cutter can be used for several gears having different numbers of teeth without much sacrifice in their operating action. The eight standard involute cutters are listed in Table 33.12. On the milling machine, the index or dividing head is used to rotate the gear blank through a certain number of degrees after each cut. The rule to use is: turns of index handle = $40/N$, where N is the number of teeth. Form cutting is usually slow.

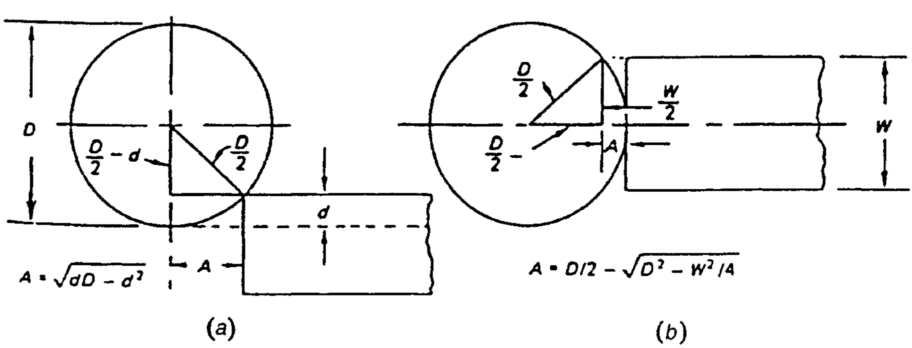
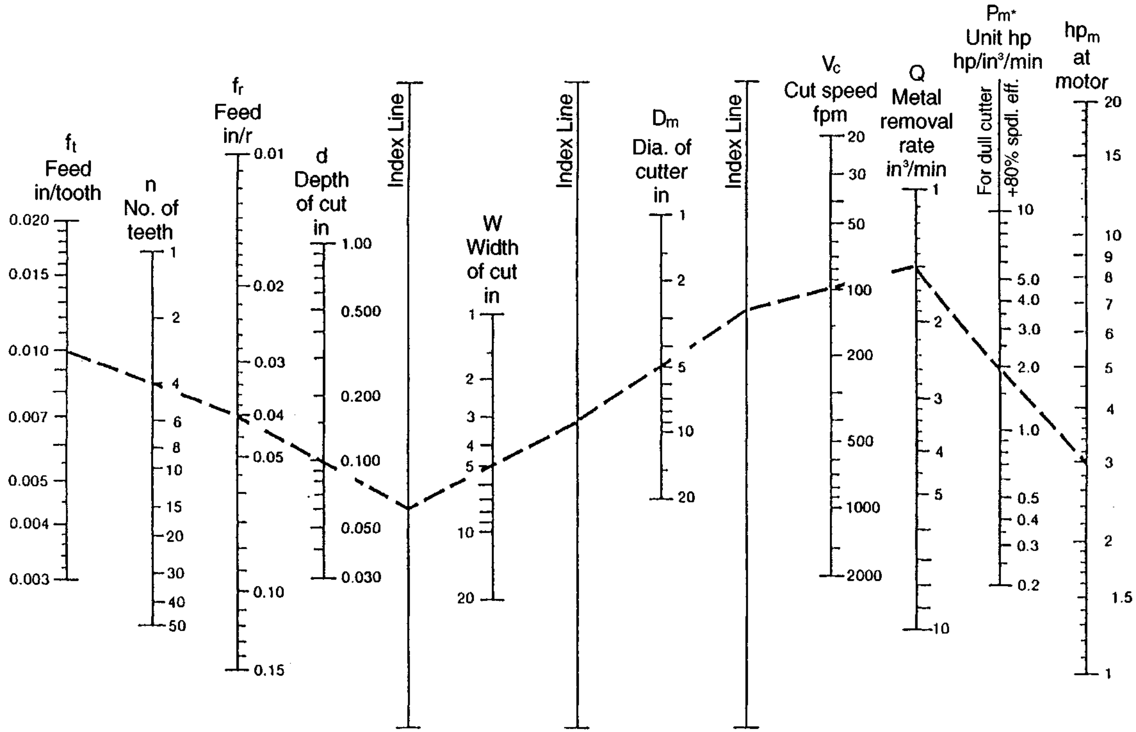


Fig. 33.20 Allowance for approach in (*a*) plain or slot milling; (*b*) face milling.

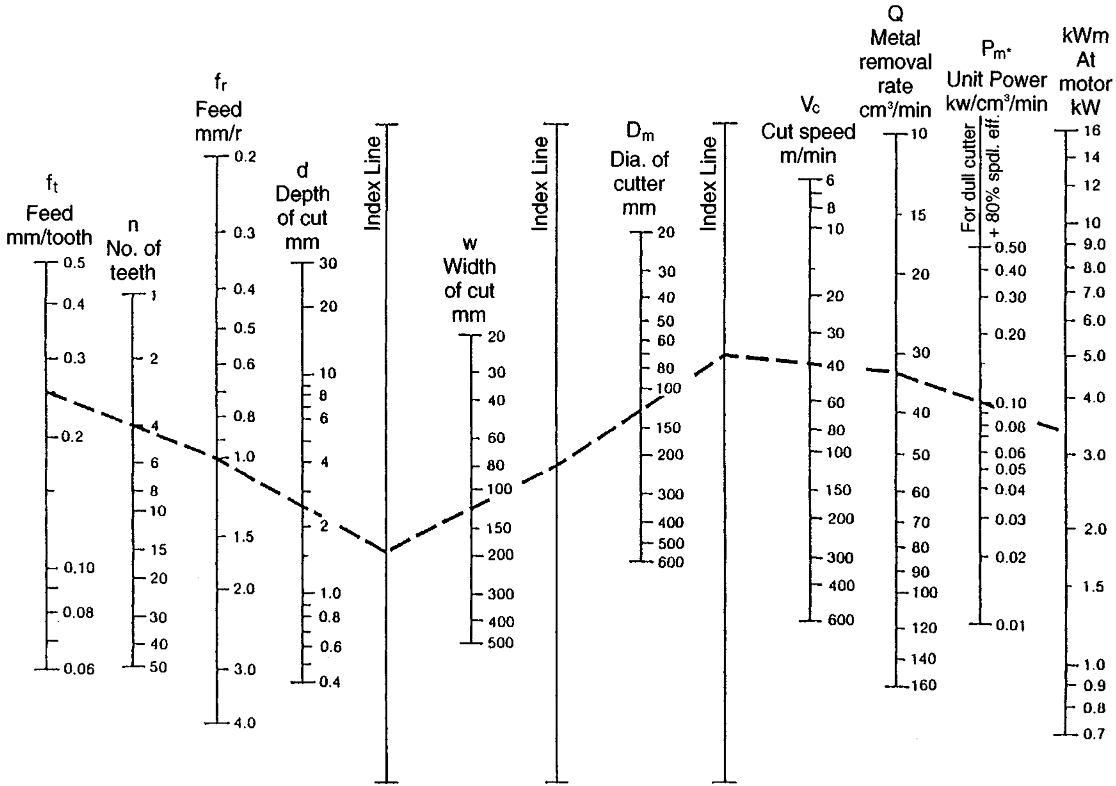


Example:

f_t = 0.010 in/tooth	d = 0.100 in	V_c = 100 fpm
n = 4 teeth	w = 5 in	Q = 1.53 in³/min
f_r = 0.04 in/r	D_m = 5 in	P = 2.0 hp/in³/min
		hp_m = 3.0 hp

$$hp_m = \frac{P_m \times 3.82 \times f_t \times n \times d \times w \times V_c}{D_m}$$

Fig. 33.21 Alignment chart for determining metal removal rate and motor horsepower in face milling—English units.



Example:

f_t = 0.25 mm/tooth
n = 4 teeth
f_r = 1.0 mm/r

d = 2.5 mm
w = 125 mm
D_m = 125 mm

V_c = 40 m/min
Q = 31.8 cm³/min
P = 0.1 kW/cm³/min
kW_m = 3.18 kW

$$hp_m = \frac{P_m \times 318.3 \times f_t \times n \times d \times w \times V_c}{1000\, D_m}$$

Fig. 33.22 Alignment chart for determining metal removal rate and motor power in face milling—metric units.

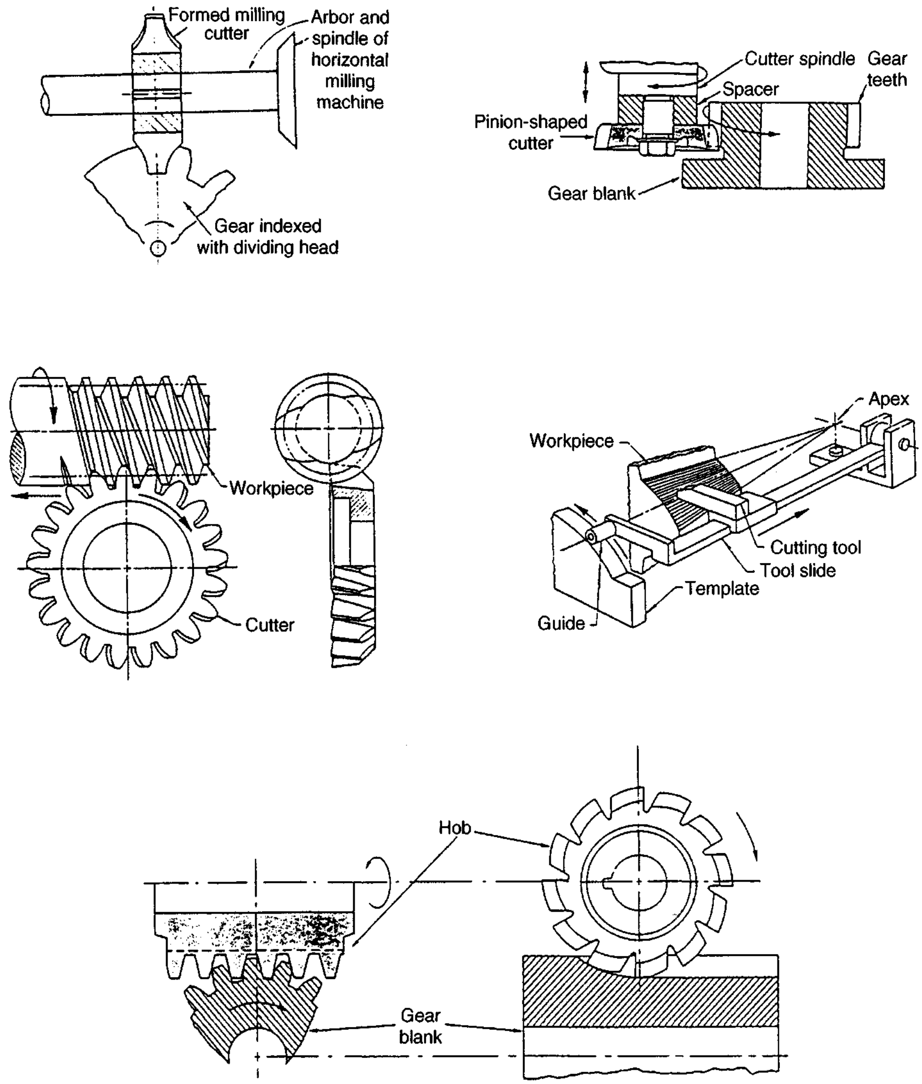


Fig. 33.23 Machining gears.

Table 33.12 Standard Gear Cutters

Cutter Number	Gear Tooth Range
1	135 teeth to rack
2	55–34
3	35–54
4	26–34
5	21–25
6	17–20
7	14–16
8	12–13

Template machining utilizes a simple, single-point cutting tool that is guided by a template. However, the equipment is specialized, and the method is seldom used except for making large-bevel gears.

The *generating process* is used to produce most high-quality gears. This process is based on the principle that any two involute gears, or any gear and a rack, of the same diametral pitch will mesh together. Applying this principle, one of the gears (or the rack) is made into a cutter by proper sharpening and is used to cut into a mating gear blank and thus generate teeth on the blank. Gear shapers (pinion or rack), gear-hobbing machines, and bevel-gear generating machines are good examples of the gear generating machines.

33.9.2 Gear Finishing

To operate efficiently and have satisfactory life, gears must have accurate tooth profile and smooth and hard faces. Gears are usually produced from relatively soft blanks and are subsequently heat-treated to obtain greater hardness, if it is required. Such heat treatment usually results in some slight distortion and surface roughness. *Grinding and lapping* are used to obtain very accurate teeth on hardened gears. Gear-shaving and burnishing methods are used in gear finishing. Burnishing is limited to unhardened gears.

33.10 THREAD CUTTING AND FORMING

Three basic methods are used for the manufacturing of threads; *cutting, rolling,* and *casting.* Die casting and molding of plastics are good examples of casting. The largest number of threads are made by rolling, even though it is restricted to standardized and simple parts, and ductile materials. Large numbers of threads are cut by the following methods:

1. Turning
2. Dies: manual or automatic (external)
3. Milling
4. Grinding (external)
5. Threading machines (external)
6. Taps (internal)

33.10.1 Internal Threads

In most cases, the hole that must be made before an internal thread is tapped is produced by drilling. The hole size determines the depth of the thread, the forces required for tapping, and the tap life. In most applications, a drill size is selected that will result in a thread having about 75% of full thread depth. This practice makes tapping much easier, increases the tap's life, and only slightly reduces the resulting strength. Table 33.13 gives the drill sizes used to produce 75% thread depth for several sizes of UNC threads. The feed of a tap depends on the lead of the screw and is equal to 1/lead ipr.

Cutting speeds depend on many factors, such as

1. Material hardness
2. Depth of cut
3. Thread profile

Table 33.13 Recommended Tap-Drill Sizes for Standard Screw-Thread Pitches (American National Coarse-Thread Series)

Number or Diameter	Threads per Inch	Outside Diameter of Screw	Tap Drill Sizes	Decimal Equivalent of Drill
6	32	0.138	36	0.1065
8	32	0.164	29	0.1360
10	24	0.190	25	0.1495
12	24	0.216	16	0.1770
1/4	20	0.250	7	0.2010
3/8	16	0.375	5/16	0.3125
1/2	13	0.500	27/64	0.4219
3/4	10	0.750	21/32	0.6562
1	8	1.000	7/8	0.875