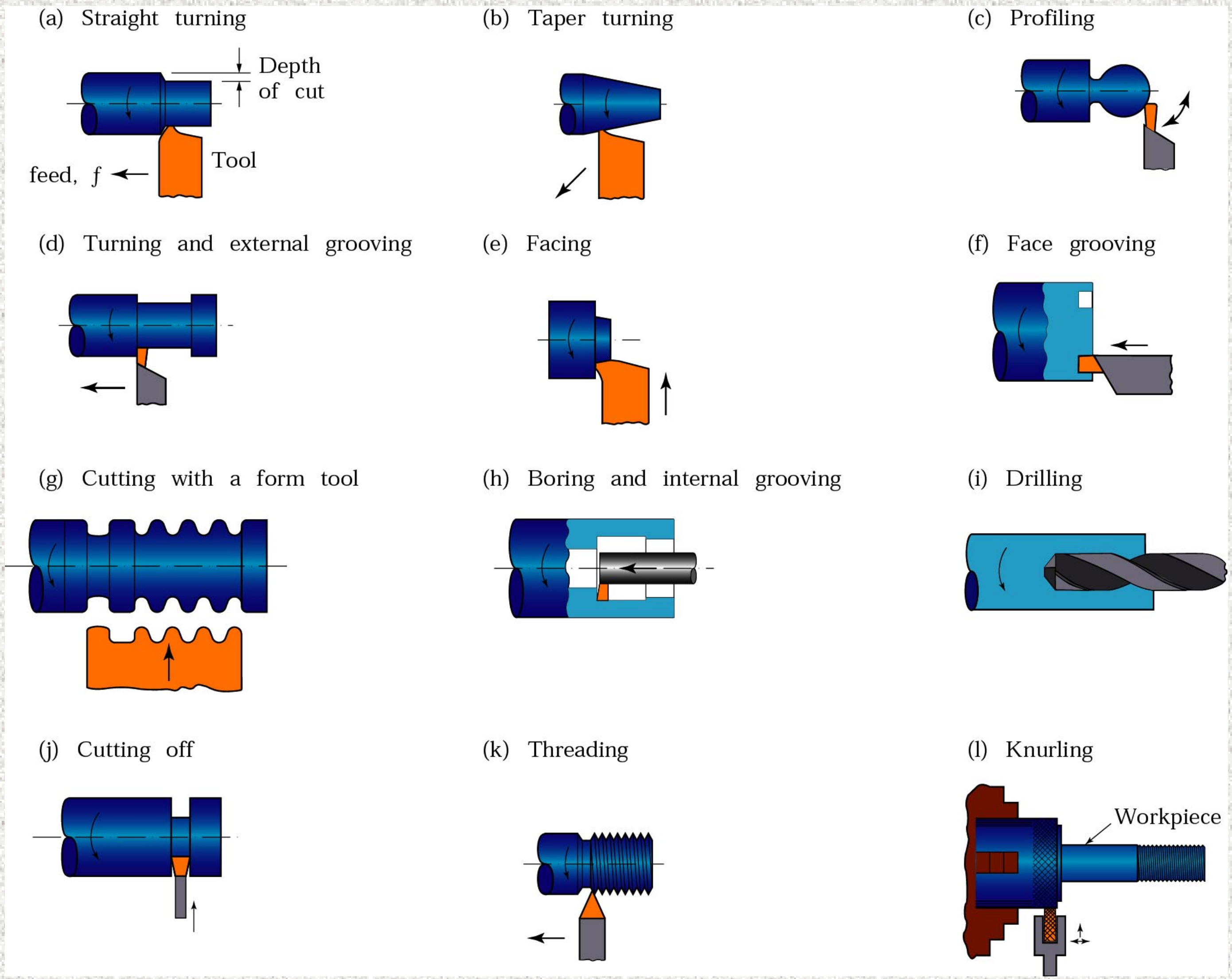
CHAPTER 22

Machining Processes Used to Produce Round Shapes

Cutting Operations

Figure 22.1 Various cutting operations that can be performed on a late. Not that all parts have circular symmetry.



Components of a Lathe

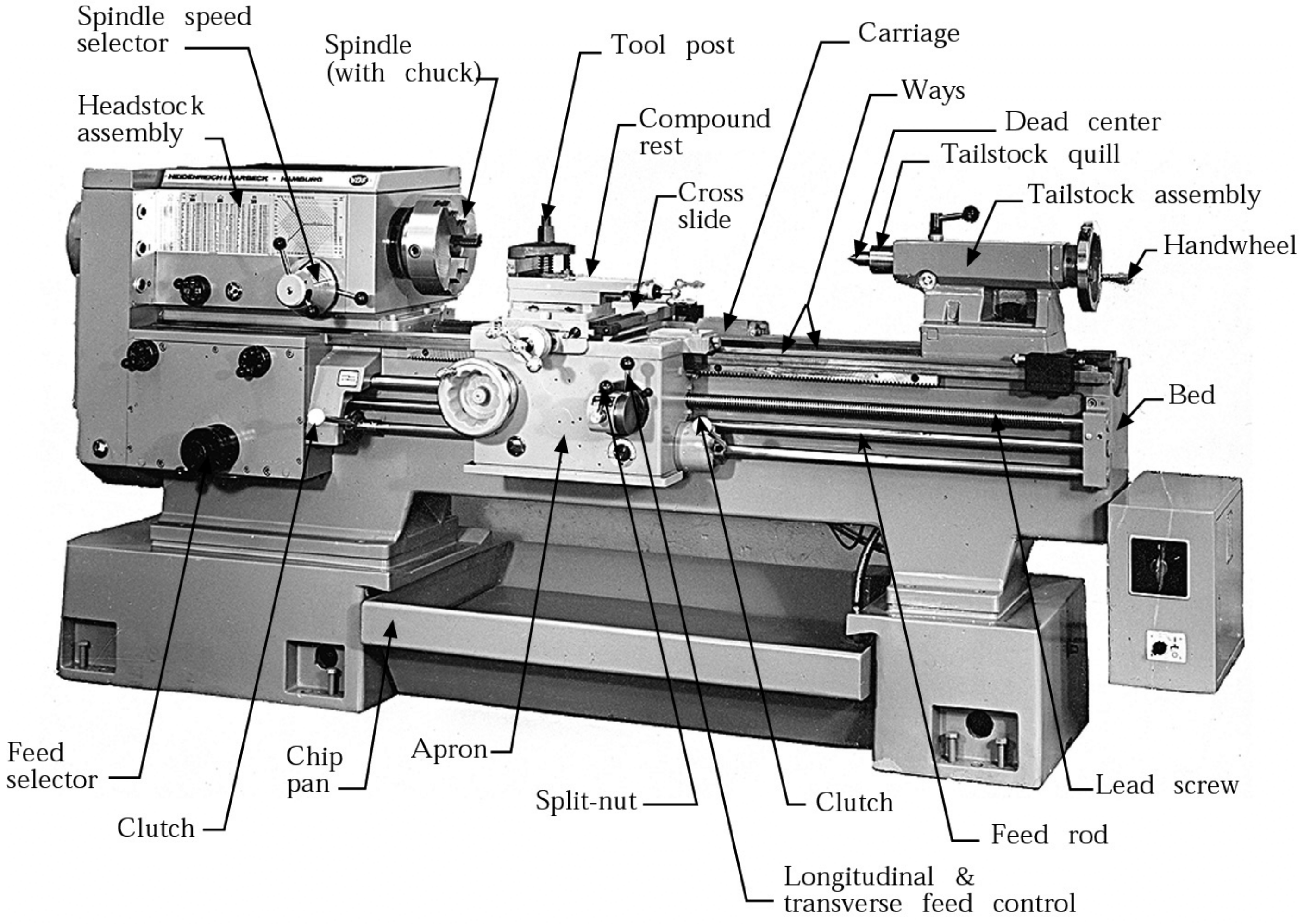


Figure 22.2 Components of a lathe. *Source*: Courtesy of Heidenreich & Harbeck

General Characteristics of Machining Processes

TABLE 22.1 General Characteristics of Machining Processes Described in Chapters 22 and 23

Process	**Characteristics**	**Commercial tolerances(±mm)**
Turning	Turning and facing operations on all types of materials; uses single-point or form tools; requires skilled labor; low production rate, but medium to high with turret lathes and automatic machines, requiring less-skilled labor.	Fine: 0.05–0.13 Rough: 0.13 Skiving: 0.025–0.05
Boring	Internal surfaces or profiles, with characteristics similar to turning ; stiffness of boring bar important to avoid chatter.	0.025
Drilling	Round holes of various sizes and depths; requires boring and reaming for improved accuracy; high production rate; labor skill required depends on hole location and accuracy specified.	0.075
Milling	Variety of shapes involving contours, flat surfaces, and slots; wide variety of tooling; versatile; low to medium production rate; requires skilled labor.	0.13–0.25
Planing	Flat surfaces and straight contour profiles on large surfaces; suitable for low-quantity production; labor skill required depends on part shape.	0.08–0.13
Shaping	Flat surfaces and straight contour profiles on relatively small workpieces; suitable for low-quantity production; labor skill required depends on part shape.	0.05–0.13
Broaching	External and internal flat surfaces, slots, and contours with good surface finish; costly tooling; high production rate; labor skill required depends on part shape.	0.025–0.15
Sawing	Straight and contour cuts on flat or structural shapes; not suitable for hard materials unless saw has carbide teeth or is coated with diamond; low production rate; requires only low labor skill.	0.8

Schematic Illustration of a Turning Operation

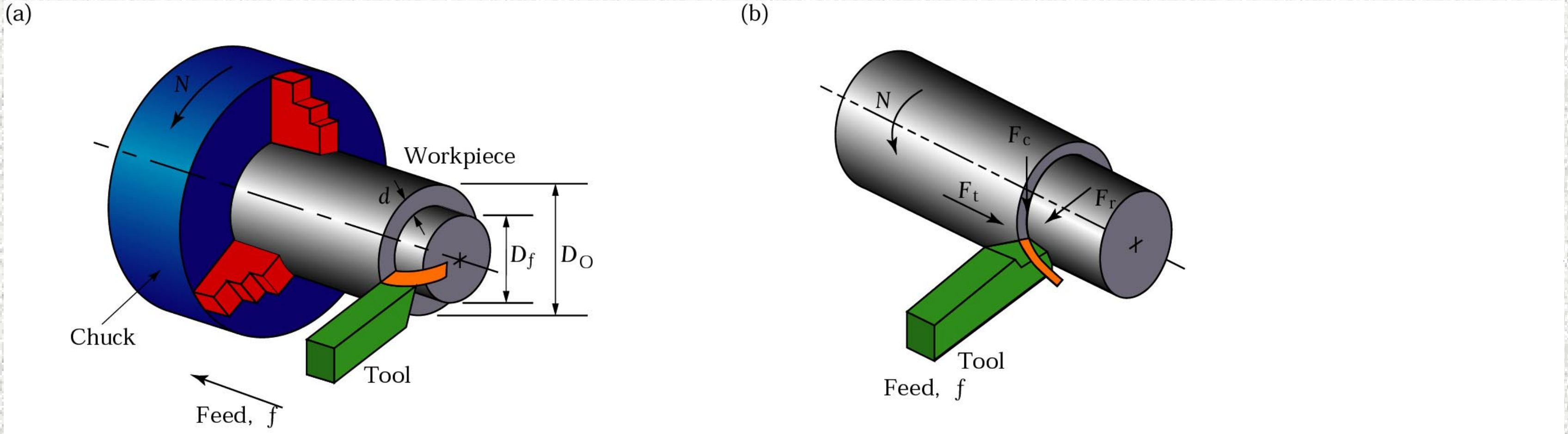
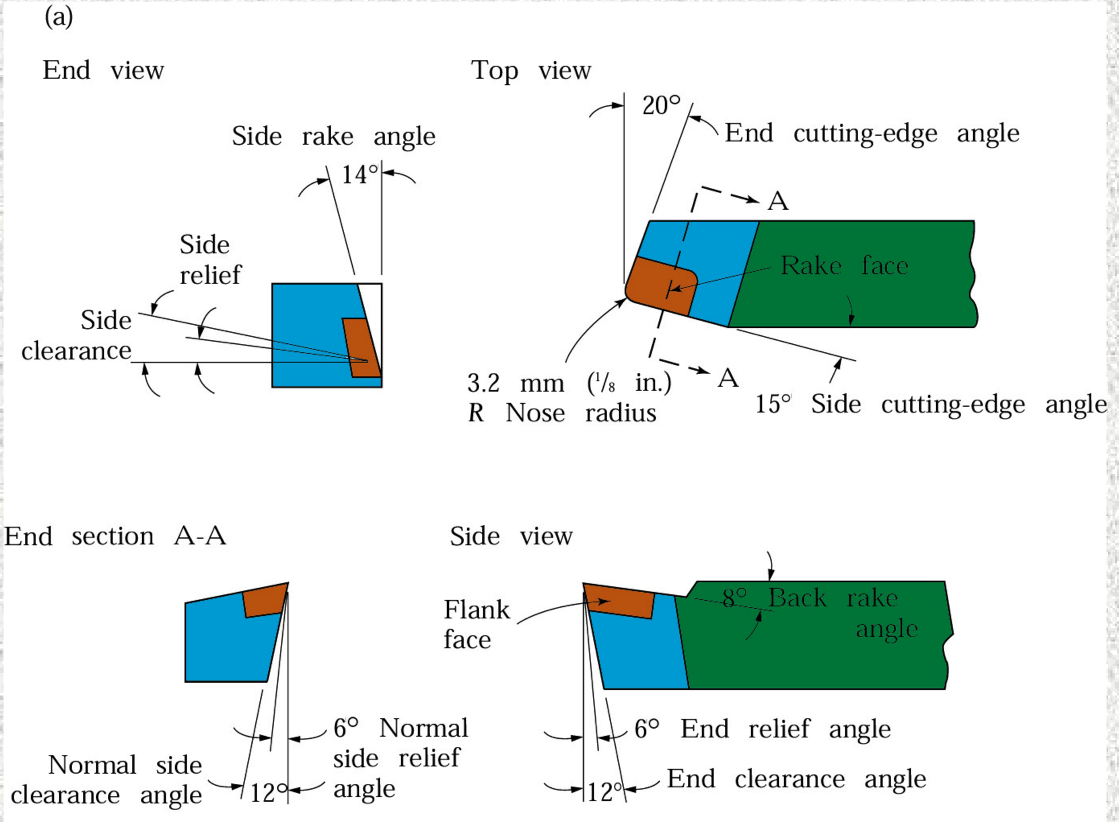


Figure 22.3 (a) Schematic illustration of a turning operation showing depth of cut, *d*, and feed, *f*. Cutting speed is the surface speed of the workpiece at the F_c is the cutting force, F_t is the thrust or feed force (in the direction of feed, F_r is the radial force that tends to push the tool away from the workpiece being machined. Compare this figure with Fig. 20.11 for a two-dimensional cutting operation.

Right-Hand Cutting Tool



Tool Signature	Dimensions	Abbreviation
8	Back. rake . angle	BR
14	Side . rake angle	SR
6	End . relief . angle	ER
12	End . clearance angle	. . .
6	Side . relief . angle	SRF
12	Side . clearance angle	. . .
20	End . cutting-edge angle	ECEA
15	Side . cutting-edge angle	SCEA
$^{1}/_{8}$	Nose. radius .	NR

Figure 22.4 (a) Designations and symbols for a right-hand cutting tool; solid high-speed-steel tools have a similar designation. Right-hand means that the tool travels from right to left as shown in Fig. 22.1a. (continued)

Right-Hand Cutting Tool (cont.)

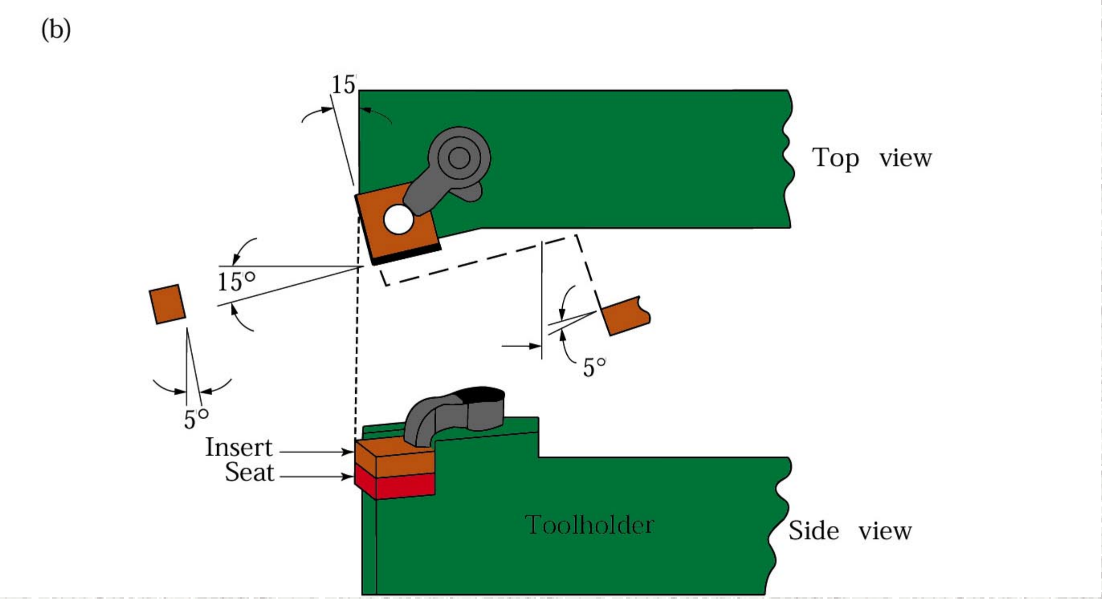


Figure 22.4 (continued) (b) Square insert in a right-hand toolholder for a turning operation. A wide variety of toolholders are available for holding inserts at various angles. *Source*: Kennametal Inc.

General Recommendations for Turning Tool Angles

TABLE 22.2

	High-speed steel					Carbide (inserts)				
Material	**Back rake**	**Side rake**	**End relief**	**Side relief**	**Side and end cutting edge**	**Back rake**	**Side rake**	**End relief**	**Side relief**	**Side and end cutting edge**
Aluminum and magnesium alloys	20	15	12	10	5	0	5	5	5	15
Copper alloys	5	10	8	8	5	0	5	5	5	15
Steels	10	12	5	5	15	–5	–5	5	5	15
Stainless steels	5	8–10	5	5	15	–5–0	–5–5	5	5	15
High-temperature alloys	0	10	5	5	15	5	0	5	5	45
Refractory alloys	0	20	5	5	5	0	0	5	5	15
Titanium alloys	0	5	5	5	15	–5	–5	55	5	
Cast irons	5	10	5	5	15	–5	–5	5515		
Thermoplastics	0	0	20–30	15–20	10	0	0	20–30	15–20	10
Thermosets	0	0	20–30	15–20	10	0	15	5	5	15

Summary of Turning Parameters and Formulas

TABLE 22.3

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
= p $D_o N$ (for maximum speed)
= $p D_{\mathrm{avg}} N$ (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.
= $(D_{\mathrm{o}} + D_f)/2$
d = Depth of cut, mm or in.
= $(D_{\mathrm{o}} + D_{\mathrm{f}})/2$
t = Cutting time, s or min
$=l/f N$
MRR = mm^3/min or in^3/min
= $p D_{\mathrm{avg}} d f N$
Torque = Nm or lb ft
= $(F_{\mathrm{c}})(D_{\mathrm{avg}}/2)$
Power = kW or hp
= (Torque) (w , where w=2p radians/min

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

Cutting Speeds for Various Tool Materials

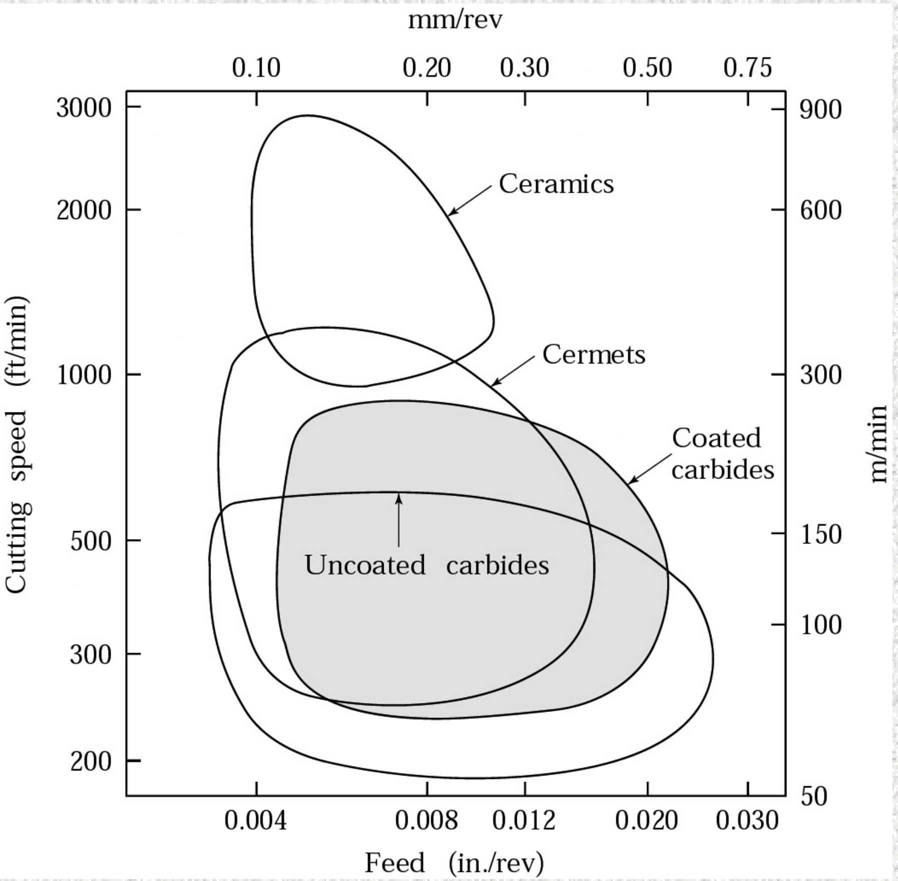


Figure 22.5 The range of applicable cutting speeds and feeds for a variety of tool materials. *Source*: Valenite.

General Recommendations for Turning Operations

TABLE 22.4

		General-purpose starting conditions			Range for roughing and finishing		
Workpiece material	**Cutting tool**	**Depth of cut mm (in.)**	**Feed mm/rev (in./rev)**	**Cutting speed m/min (ft/min)**	**Depth of cut mm (in.)**	**Feed mm/rev (in./rev)**	**Cutting speed m/min (ft/min)**
Low-C and free-machining steels	Uncoated carbide	1.5-6.3 (0.06-0.25)	0.35 (0.014)	90 (300)	0.5-7.6 (0.02-0.30)	0.15-1.1 (0.006-0.045)	60-135 (200-450)
	Ceramic-coated carbide	"	"	245-275 (800-900)	"	"	120-425 (400-1400)
	Triple coated carbide	"	"	185-200 (600-650)	"	"	90-245 (300-800)
	TiN-coated carbide	"	"	105-150 (350-500)	"	"	60-230 (200-750)
	Al_2O_3 ceramic	"	0.25 (0.010)	395-440 (1300-1450)	"	"	365-550 (1200-1800)
	Cermet	"	0.30 (0.012)	215-290 (700-950)	"	"	105-455 (350-1800)
Medium and high-C steels	Uncoated carbide	1.2-4.0 (0.05-0.20)	0.30 (0.012)	75 (250)	2.5-7.6 (0.10-0.30)	0.15-0.75 (0.006-0.03)	45-120 (150-400)
	Ceramic-coated carbide	"	"	185-230 (600-750)	"	"	120-410 (400-1350)
	Triple coated carbide	"	"	120-150 (400-500)	"	"	75-215 (250-700)
	TiN-coated carbide	"	"	90-200 (300-650)	"	"	45-215 (150-700)
	Al_2O_3 ceramic	"	0.25 (0.010)	335 (1100)	"	"	245-455 (800-1500)
	Cermet	"	0.25 (0.010)	170-245 (550-800)	"	"	105-305 (350-1000)

General Recommendations for Turning Operations (cont.)

TABLE 22.4 (continued)

		General-purpose starting conditions			Range for roughing and finishing		
Workpiece material	**Cutting tool**	**Depth of cut mm (in.)**	**Feed mm/rev (in./rev)**	**Cutting speed m/min (ft/min)**	**Depth of cut mm (in.)**	**Feed mm/rev (in./rev)**	**Cutting speed m/min (ft/min)**
Cast iron, gray	Uncoated carbide	1.25-6.3 (0.05-0.25)	0.32 (0.013)	90 (300)	0.4-12.7 (0.015-0.5)	0.1-0.75 (0.004-0.03)	75-185 (250-600)
	Ceramic-coated carbide	"	"	200 (650)	"	"	120-365 (400-1200)
	TiN-coated carbide	"	"	90-135 (300-450)	"	"	60-215 (200-700)
	Al_2O_3 ceramic	"	0.25 (0.010)	455-490 (1500-1600)	"	"	365-855 (1200-2800)
	SiN ceramic	"	0.32 (0.013)	730 (2400)	"	"	200-990 (650-3250)
Stainless steel, austenitic	Triple coated carbide	1.5-4.4 (0.06-0.175)	0.35 (0.014)	150 (500)	0.5-12.7 (0.02-0.5)	0.08-0.75 (0.003-0.03)	75-230 (250-750)
	TiN-coated carbide	"	"	85-160 (275-525)	"	"	55-200 (175-650)
	Cermet	"	0.30 (0.012)	185-215 (600-700)	"	"	105-290 (350-950)
High-temperature alloys, nickel base	Uncoated carbide	2.5 (0.10)	0.15 (0.006)	25-45 (75-150)	0.25-6.3 (0.01-0.25)	0.1-0.3 (0.004-0.012)	15-30 (50-100)
	Ceramic-coated carbide	"	"	45 (150)	"	"	20-60 (65-200)
	TiN-coated carbide	"	"	30-55 (95-175)	"	"	20-85 (60-275)
	Al_2O_3 ceramic	"	"	260 (850)	"	"	185-395 (600-1300)
	SiN ceramic	"	"	215 (700)	"	"	90-215 (300-700)
	Polycrystalline CBN	"	"	150 (500)	"	"	120-185 (400-600)

General Recommendations for Turning Operations (cont.)

TABLE 22.4 (continued)

		General-purpose starting conditions			Range for roughing and finishing		
Workpiece material	**Cutting tool**	**Depth of cut mm (in.)**	**Feed mm/rev (in./rev)**	**Cutting speed m/min (ft/min)**	**Depth of cut mm (in.)**	**Feed mm/rev (in./rev)**	**Cutting speed m/min (ft/min)**
Titanium alloys	Uncoated carbide	1.0-3.8 (0.04-0.15)	0.15 (0.006)	35-60 (120-200)	0.25-6.3 (0.01-0.25)	0.1-0.4 (0.004-0.015)	10-75 (30-250)
	TiN-coated carbide	"	"	30-60 (100-200)	"	"	10-100 (30-325)
Aluminum alloys, free machining	Uncoated carbide	1.5-5.0 (0.06-0.20)	0.45 (0.018)	490 (1600)	0.25-8.8 (0.01-0.35)	0.08-0.62 (0.003-0.025)	200-670 (650-2000)
	TiN-coated carbide	"	"	550 (1800)	"	"	60-915 (200-3000)
	Cermet	"	"	490 (1600)	"	"	215-795 (700-2600)
	Polycrystalline diamond	"	"	760 (2500)	"	"	305-3050 (1000-10,000)
High silicon	Polycrystalline diamond	"	"	530 (1700)	"	"	365-915 (1200-3000)
Copper alloys	Uncoated carbide	1.5-5.0 (0.06-0.20)	0.25 (0.010)	260 (850)	0.4-7.51 (0.015-0.3)	0.15-0.75 (0.006-0.03)	105-535 (350-1750)
	Ceramic-coated carbide	"	"	365 (1200)	"	"	215-670 (700-2200)
	Triple-coated carbide	"	"	215 (700)	"	"	90-305 (300-1000)
	TiN-coated carbide	"	"	90-275 (300-900)	"	"	45-455 (150-1500)
	Cermet	"	"	245-425 (800-1400)	"	"	200-610 (650-2000)
	Polycrystalline diamond	"	"	520 (1700)	"	"	275-915 (900-3000)

General Recommendations for Turning Operations (cont.)

		General-purpose starting conditions			Range for roughing and finishing		
Workpiece material	**Cutting tool**	**Depth of cut mm (in.)**	**Feed mm/rev (in./rev)**	**Cutting speed m/min (ft/min)**	**Depth of cut mm (in.)**	**Feed mm/rev (in./rev)**	**Cutting speed m/min (ft/min)**
Tungsten alloys	Uncoated carbide	2.5 (0.10)	0.2 (0.008)	75 (250)	0.25-5.0 (0.01-0.2)	0.12-0.45 (0.005-0.018)	55-120 (175-400)
	TiN-coated carbide	"	"	85 (275)	"	"	60-150 (200-500)
Thermoplastics and thermosets	TiN-coated carbide	1.2 (0.05)	0.12 (0.005)	170 (550)	0.12-5.0 (0.005-0.20)	0.08-0.35 (0.003-0.015)	90-230 (300-750)
	Polycrystalline diamond	"	"	395 (1300)	"	"	150-730 (500-2400)
Composites, graphite reinforced	TiN-coated carbide	1.9 (0.075)	0.2 (0.008)	200 (650)	0.12-6.3 (0.005-0.25)	0.12-1.5 (0.005-0.06)	105-290 (350-950)
	Polycrystalline diamond	"	"	760 (2500)	"	"	550-1310 (1800-4300)

Source: Based on data from Kennametal, Inc.

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

General Recommendations for Cutting Fluids for Machining

TABLE 22.5

Material	Type of fluid
Aluminum	D, MO, E, MO FO, CSN
Beryllium	MC, E, CSN
Copper	D, E, CSN, MO FO
Magnesium	D, MO, MO FO
Nickel	MC, E, CSN
Refractory	MC, E, EP
Steels (carbon and low alloy)	D, MO, E, CSN, EP
Steels (stainless)	D, MO, E, CSN
Titanium	CSN, EP, MO
Zinc	C, MC, E, CSN
Zirconium	D, E, CSN

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

Typical Capacities and Maximum Workpiece Dimensions for Machine Tools

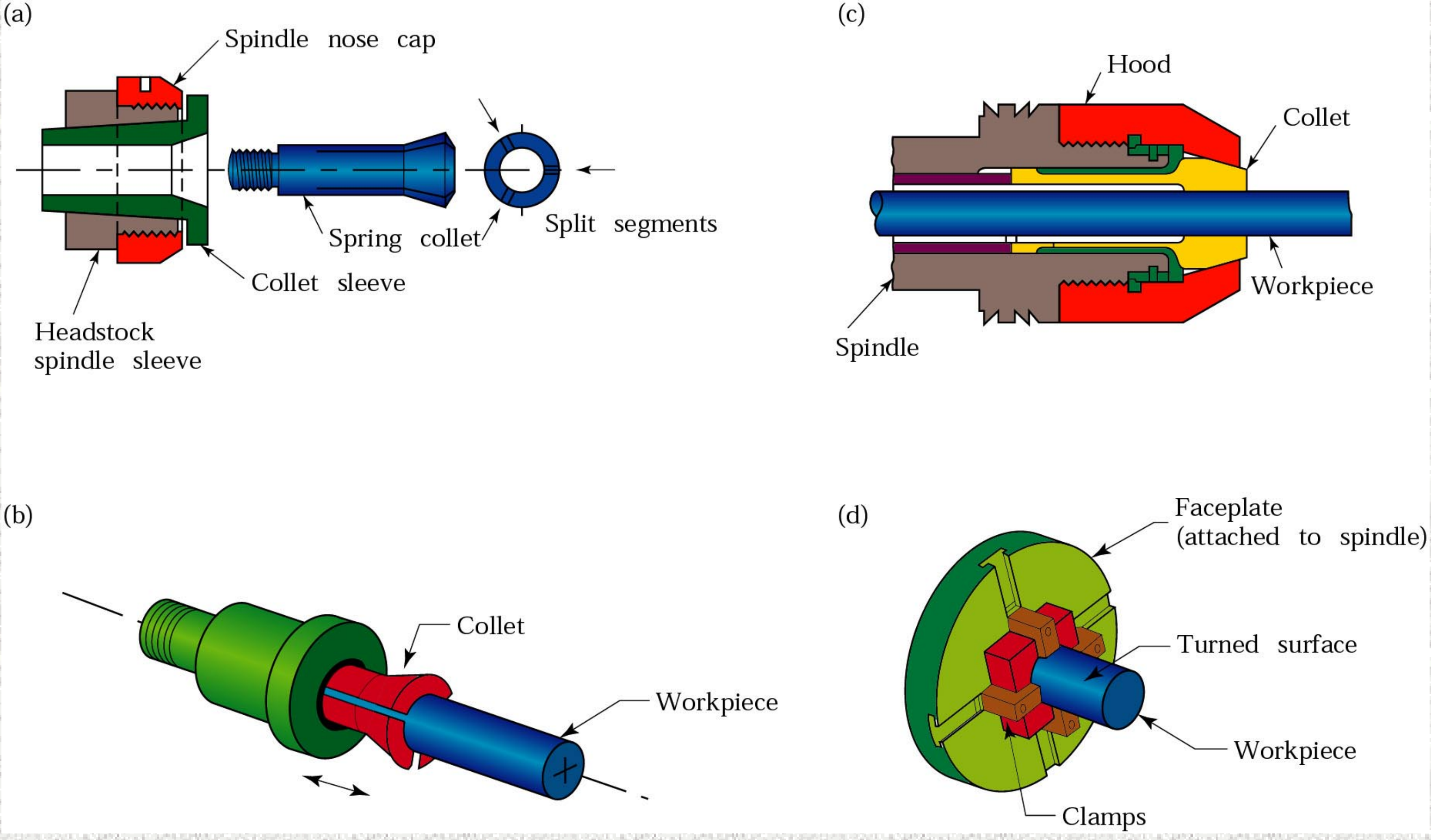
TABLE 22.6

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

Note: Larger capacities are available for special applications.

Collets

Figure 22.6 (a) and (b) Schematic illustrations of a draw-in type collet. The workpiece is placed in the collet hole, and the conical surfaces of the collet are forced inward by pulling it with a draw bar into the sleeve. (c) A push-out type collet. (d) Workholding of a part on a face plate.



Mandrels

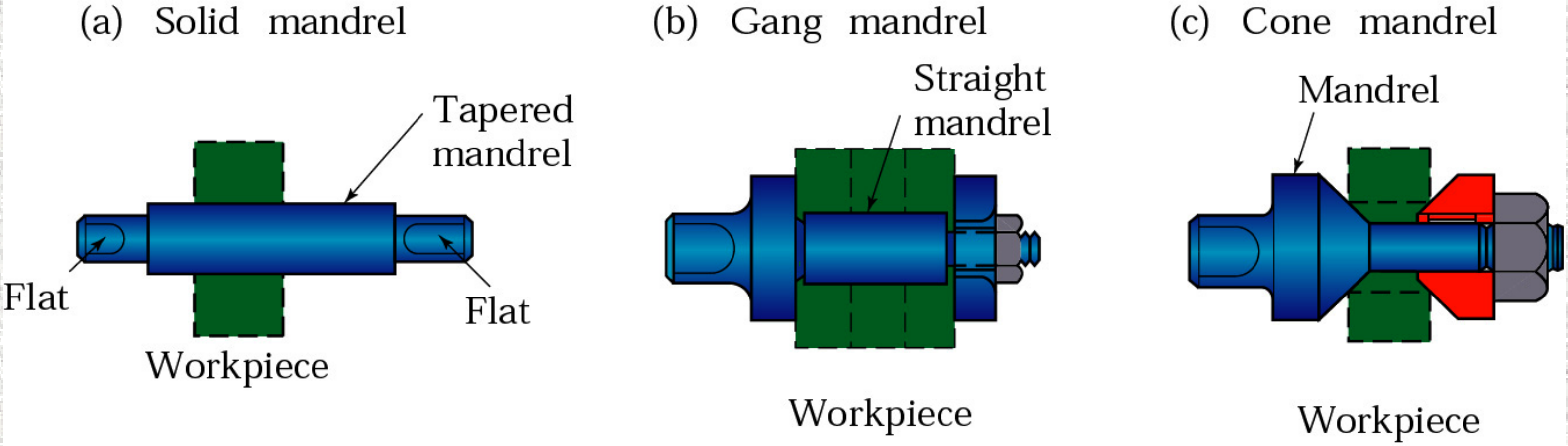


Figure 22.7 Various types of mandrels to hold workpieces for turning. These mandrels are usually mounted between centers on a lathe. Note that in (a), both the cylindrical and the end faces of the workpiece can be machined, whereas in (b) and (c), only the cylindrical surfaces can be machined.

Swiss-Type Automatic Screw Machine

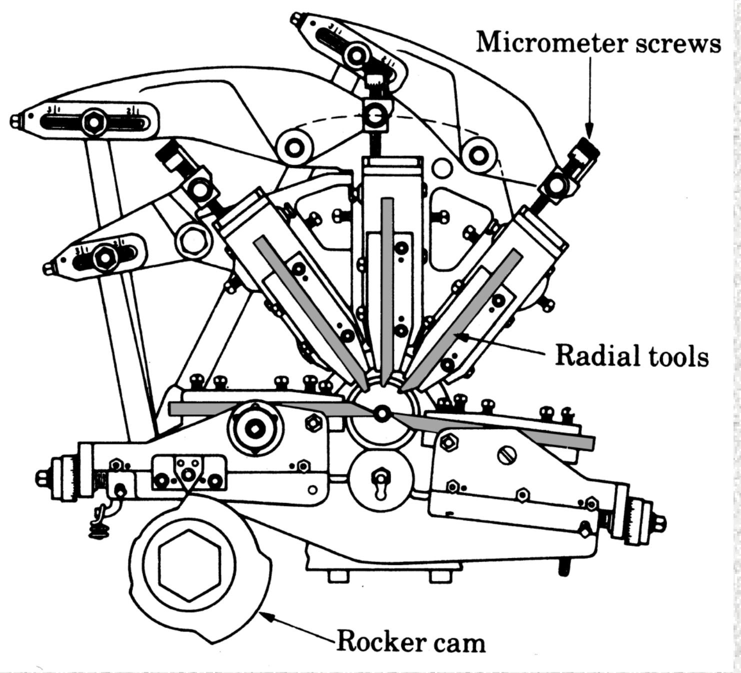


Figure 22.8 Schematic illustration of a Swiss-type automatic screw machine. *Source*: George Gorton Machine Company.

Turret Lathe

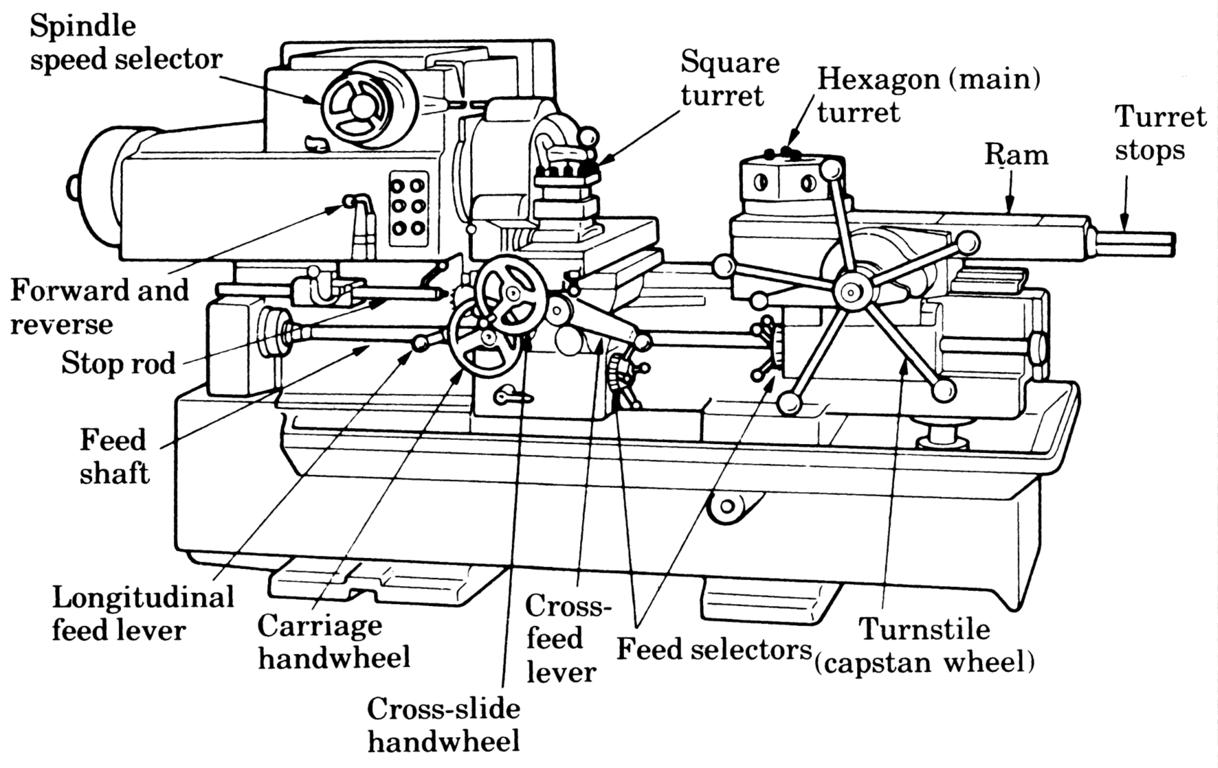
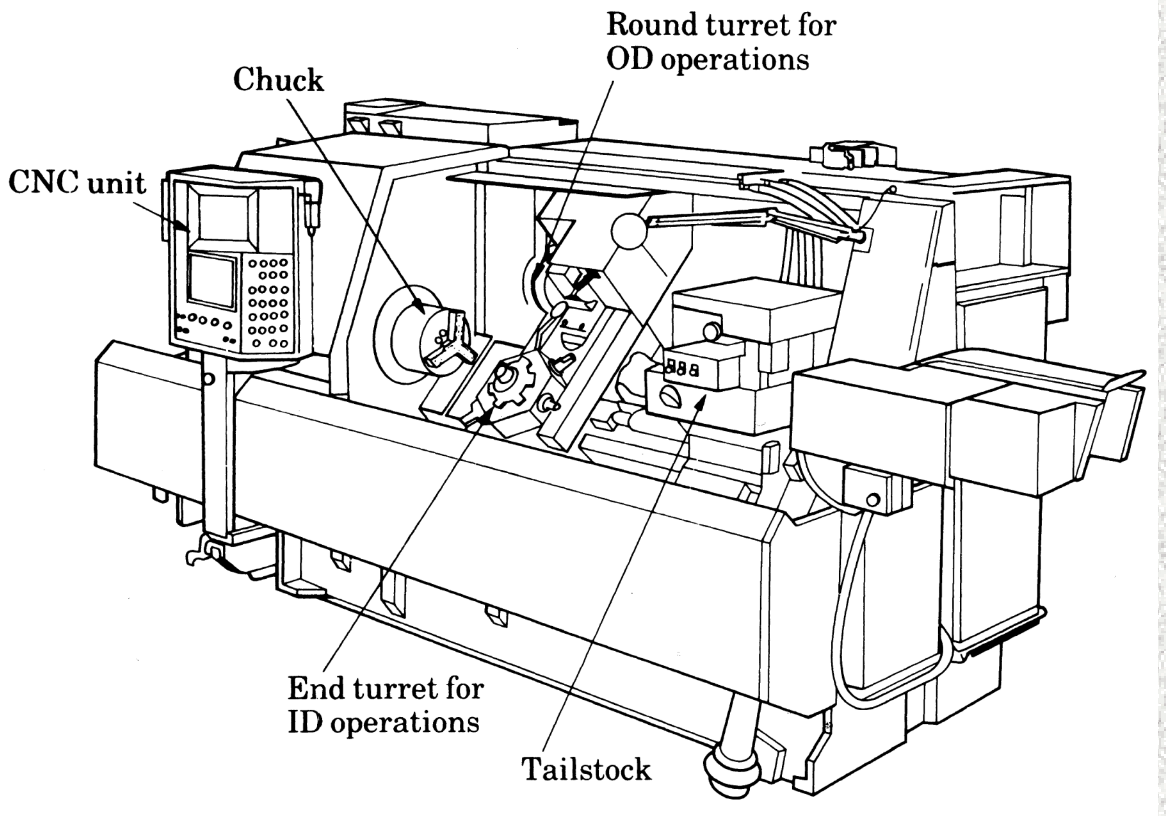


Figure 22.9 Schematic illustration of the components of a turret lathe. Note the two turrets: square and hexagonal (main). *Source*: *American Machinist and Automated Manufacturing.*

Computer Numerical Control Lathe

Figure 22.10 A computer numerical control lathe. Note the two turrets on this machine. *Source*: Jones & Lamson, Textron, Inc.



Examples of Turrets

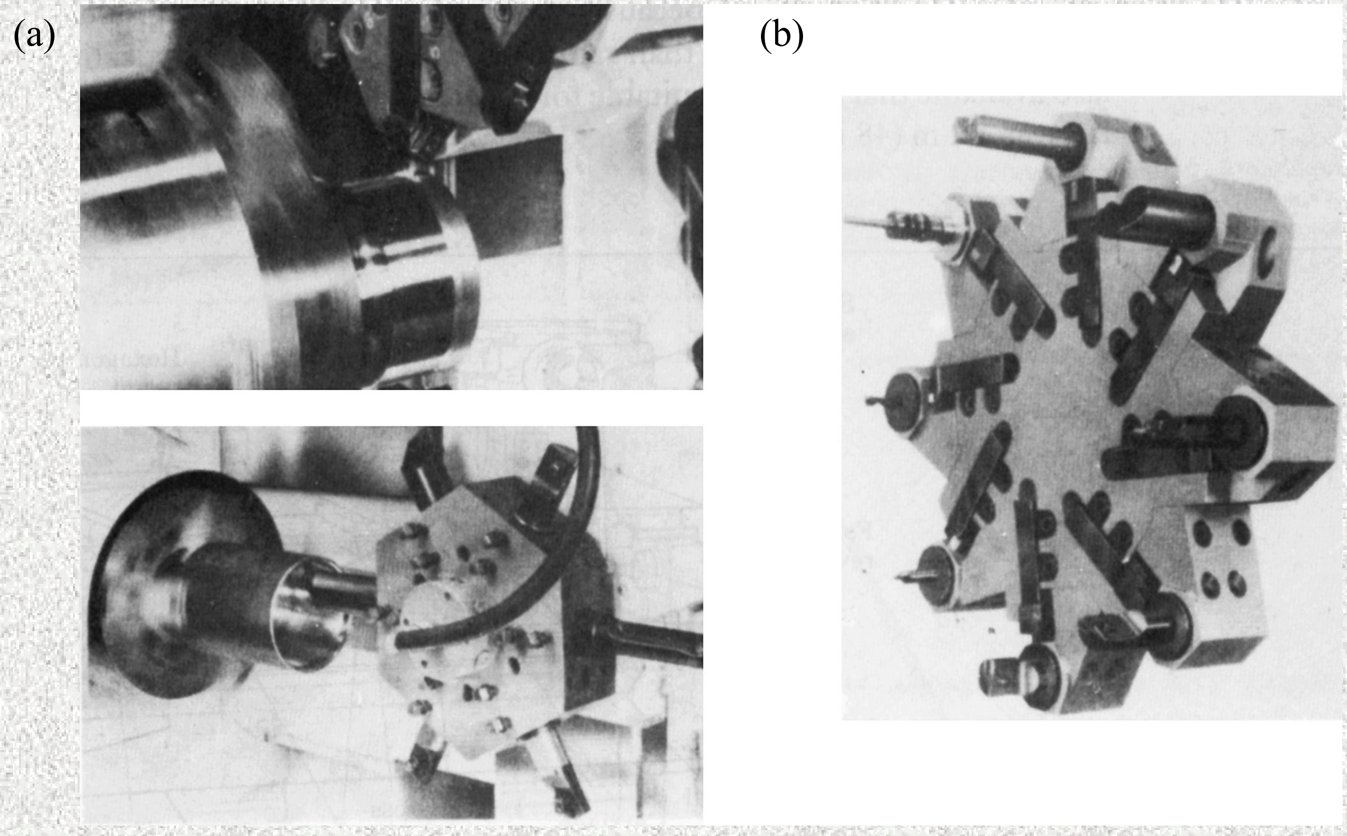
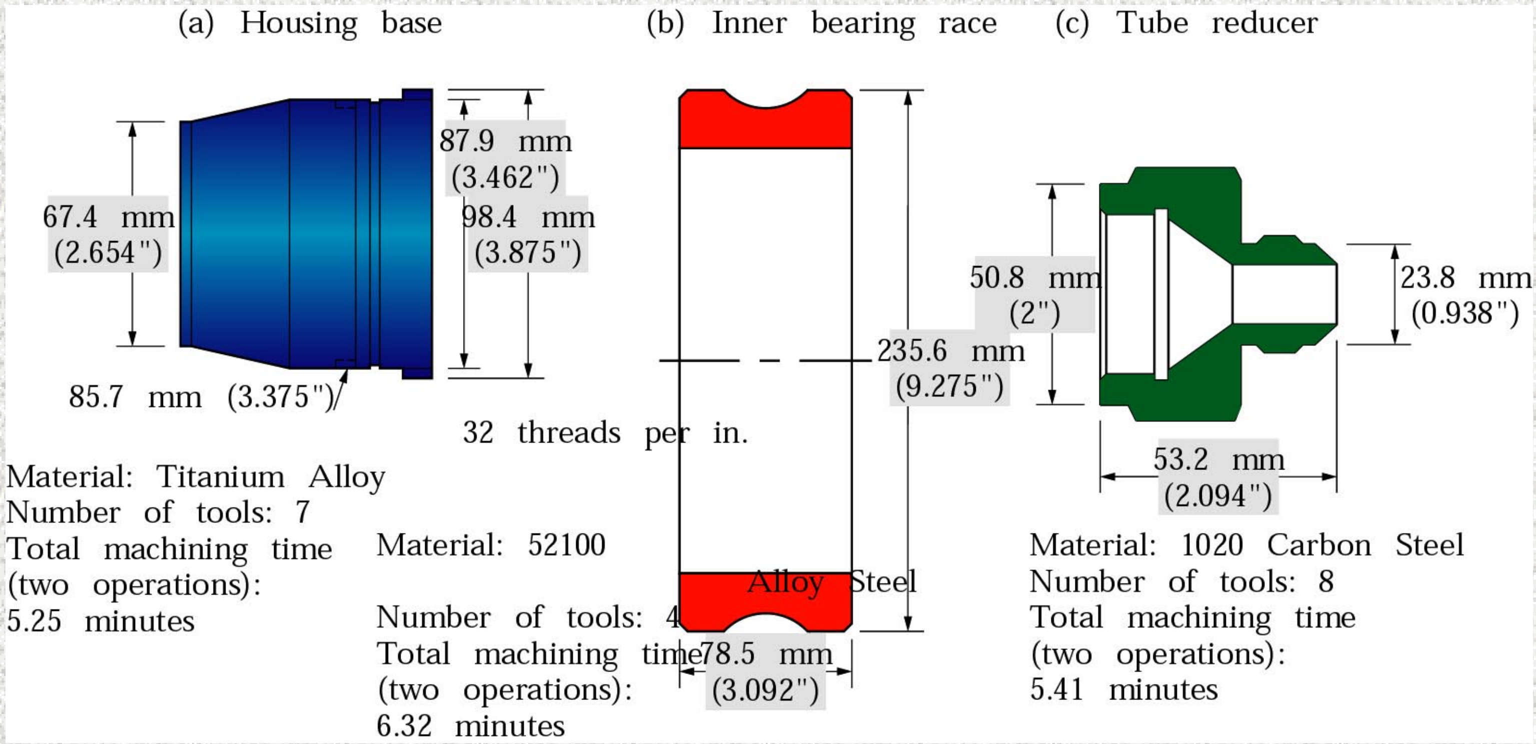


Figure 22.11 (a) A turret with six different tools for inside-diameter and outside-diameter cutting and threading operations. (b) A turret with eight different cutting tools. *Source*: Monarch Machine Tool Company.

Examples of Parts Made on CNC Turning Machine Tools



Material: Titanium Alloy
Number of tools: 7
Total machining time (two operations): 5.25 minutes

Material: 52100 Alloy Steel
Number of tools: 4
Total machining time (two operations): 6.32 minutes

Material: 1020 Carbon Steel
Number of tools: 8
Total machining time (two operations): 5.41 minutes

Figure 22.12

Examples of Machining Complex Shapes

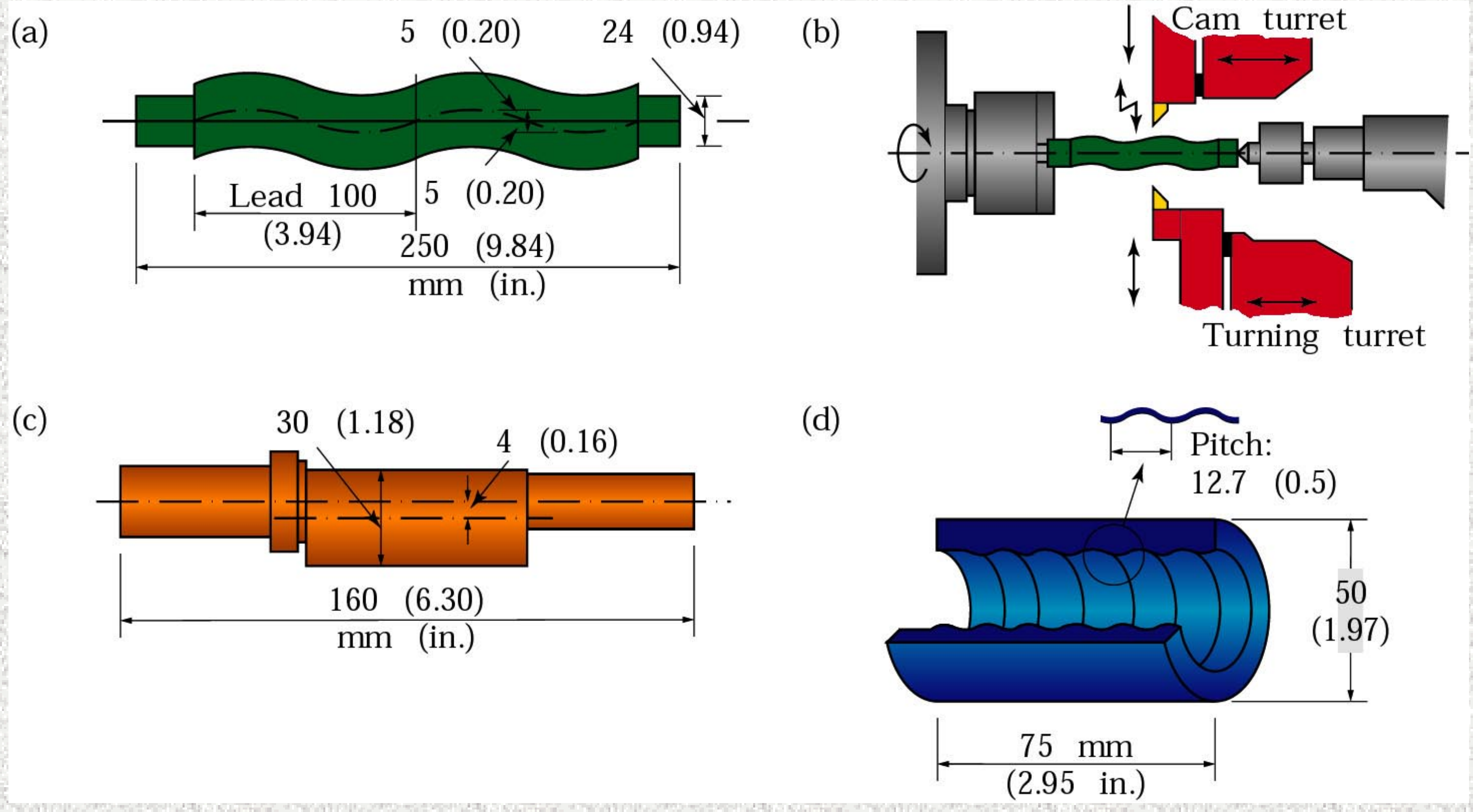


Figure 22.13

Machining of Various Complex Shapes

TABLE 22.7 Example: Machining of Various Complex Shapes

Operation	**Cutting speed**		**Depth of cut**	**Feed**	**Tool**
(a)					
OD roughing	1150 rpm	160 m/min (525 fpm)	3 mm (0.12 in.)	0.3 mm/rev (0.012 ipr)	K10 (C3)
OD finishing	1750	250 (820)	0.2 (0.008)	0.15 (0.0059)	K10 (C3)
Lead roughing	300	45 (148)	3 (0.12)	0.15 (0.0059)	K10 (C3)
Lead finishing	300	45 (148)	0.1 (0.004)	0.15 (0.0059)	Diamond compact
(b)					
Eccentric roughing	200 rpm	5-11 m/min (16-136 fpm)	1.5 mm (0.059 in)	0.2 mm/rev (0.008 ipr)	K10 (C3)
Eccentric finishing	200	5-11 (16-36)	0.1 (0.004)	0.05 (0.0020)	K10 (C3)
(c)					
Thread roughing	800 rpm	70 m/min (230 fpm)	1.6 mm (0.063 in.)	0.15 mm/rev (0.0059 ipr)	Coated carbide
Thread finishing	800	70 (230)	0.1 (0.004)	0.15 (0.0059)	Cermet

Typical Production Rates for Various Cutting Operations

TABLE 22.8

Operation	Rate
Turning	
Engine lathe	Very low to low
Tracer lathe	Low to medium
Turret lathe	Low to medium
Computer-control lathe	Low to medium
Single-spindle chuckers	Medium to high
Multiple-spindle chuckers	High to very high
Boring	Very low
Drilling	Low to medium
Milling	Low to medium
Planing	Very low
Gear cutting	Low to medium
Broaching	Medium to high
Sawing	Very low to low

Note: Production rates indicated are relative: *Very low* is about one or more parts per hour; *medium* is approximately 100 parts per hour; *very high* is 1000 or more parts per hour.

Surface Roughnesses

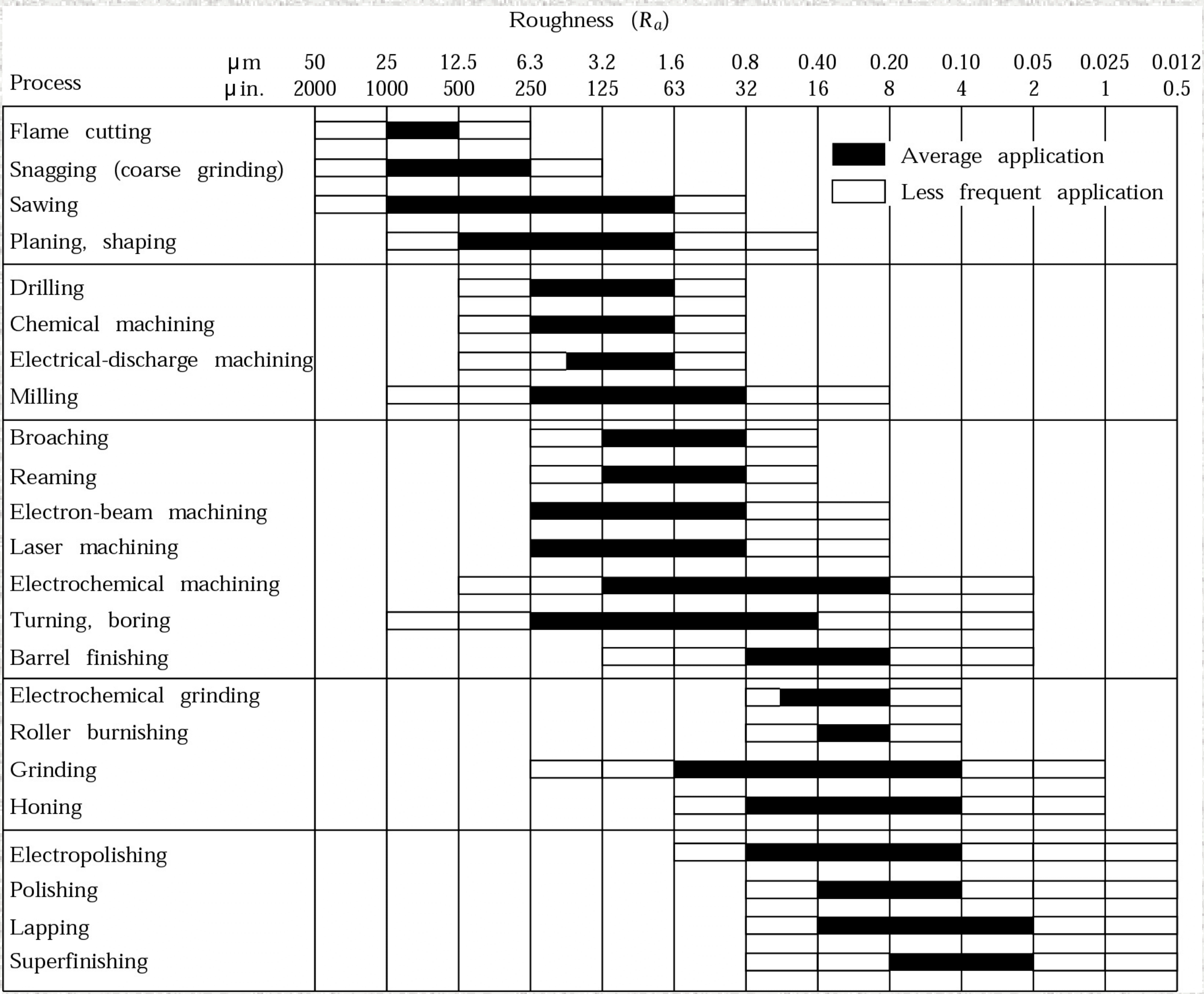
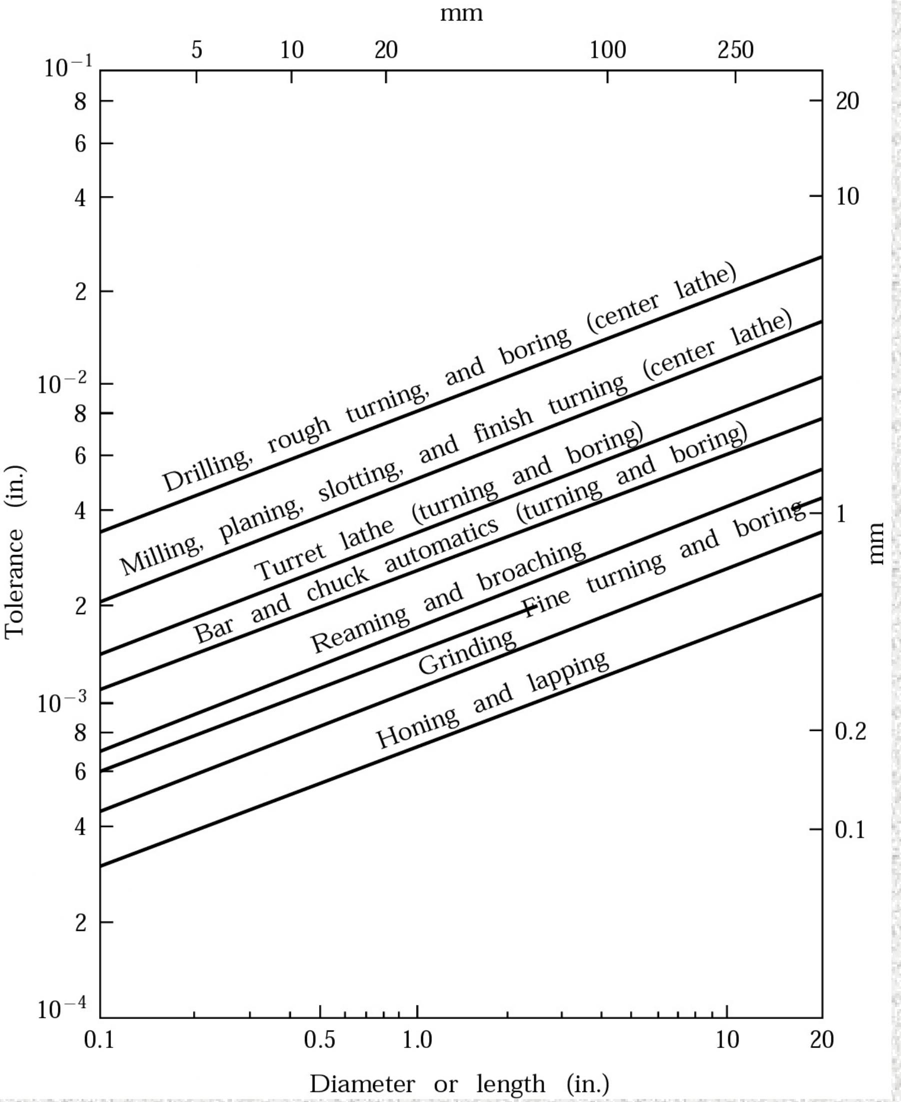


Figure 22.14 The range of surface roughnesses obtained in various machining processes. Note the wide range within each group, especially in turning and boring. See also Fig. 26.4.

Dimensional Tolerances

Figure 22.15 The 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. *Source*: Adapted from *Manufacturing Planning and Estimating Handbook*, McGraw-Hill, 1963.



General Troubleshooting Guide for Turning Operations

TABLE 22.9

Problem	Probable causes
Tool breakage	Tool material lacks toughness; improper tool angles; machine tool lacks stiffness; worn bearings and machine components; cutting parameters too high.
Excessive tool wear	Cutting parameters too high; improper tool material; ineffective cutting fluid ; improper tool angles.
Rough surface finish	Built-up edge on tool; feed too high; tool too sharp, chipped or worn; vibration and chatter.
Dimensional variability	Lack of stiffness; excessive temperature rise; tool wear.
Tool chatter	Lack of stiffness; workpiece not supported rigidly; excessive tool overhang.

Examples of Threads

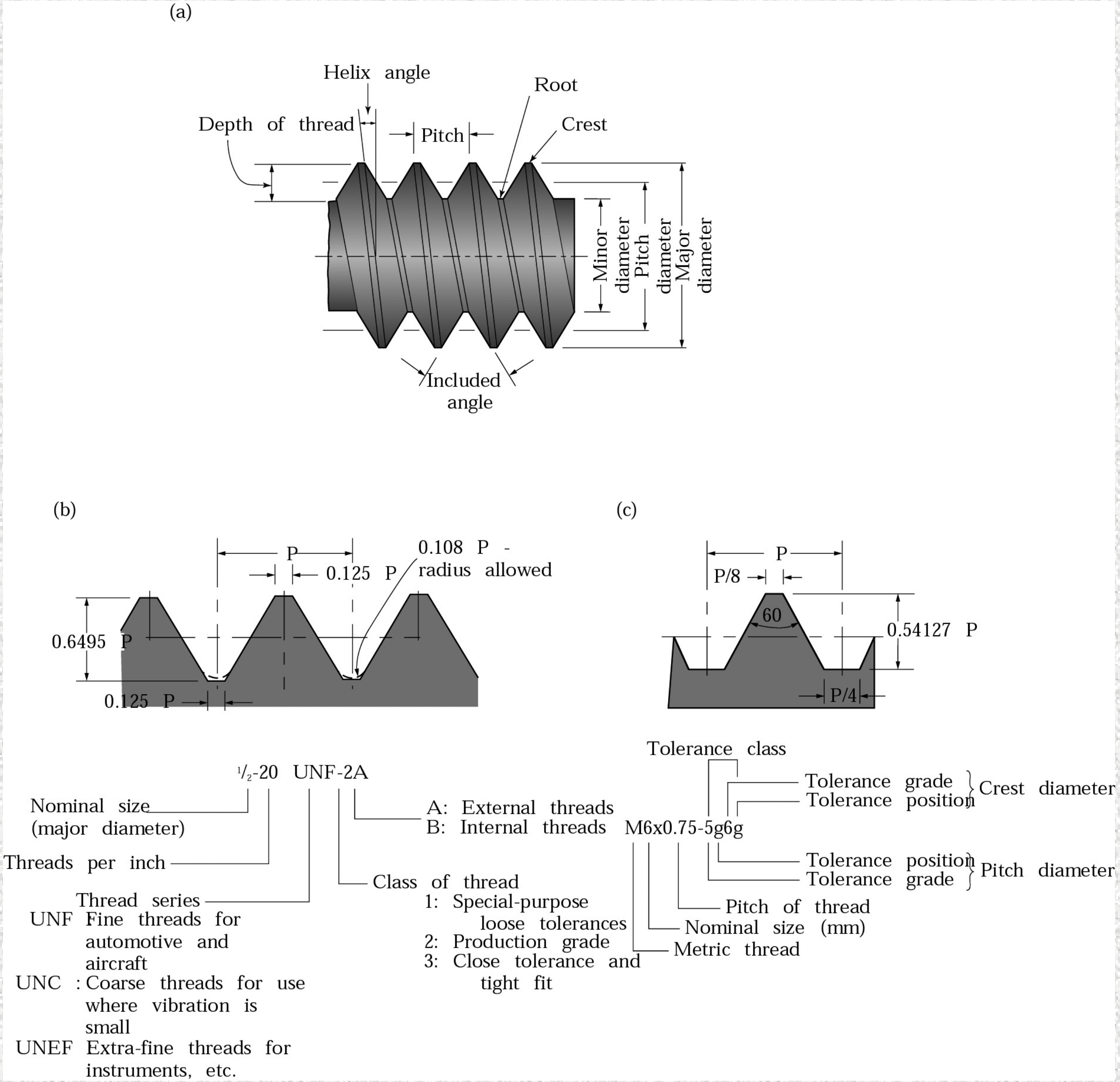


Figure 22.16 (a) Standard nomenclature for screw threads. (b) Unified National thread and identification of threads. (c) ISO metric thread and identification of threads.

Types of Screw Threads

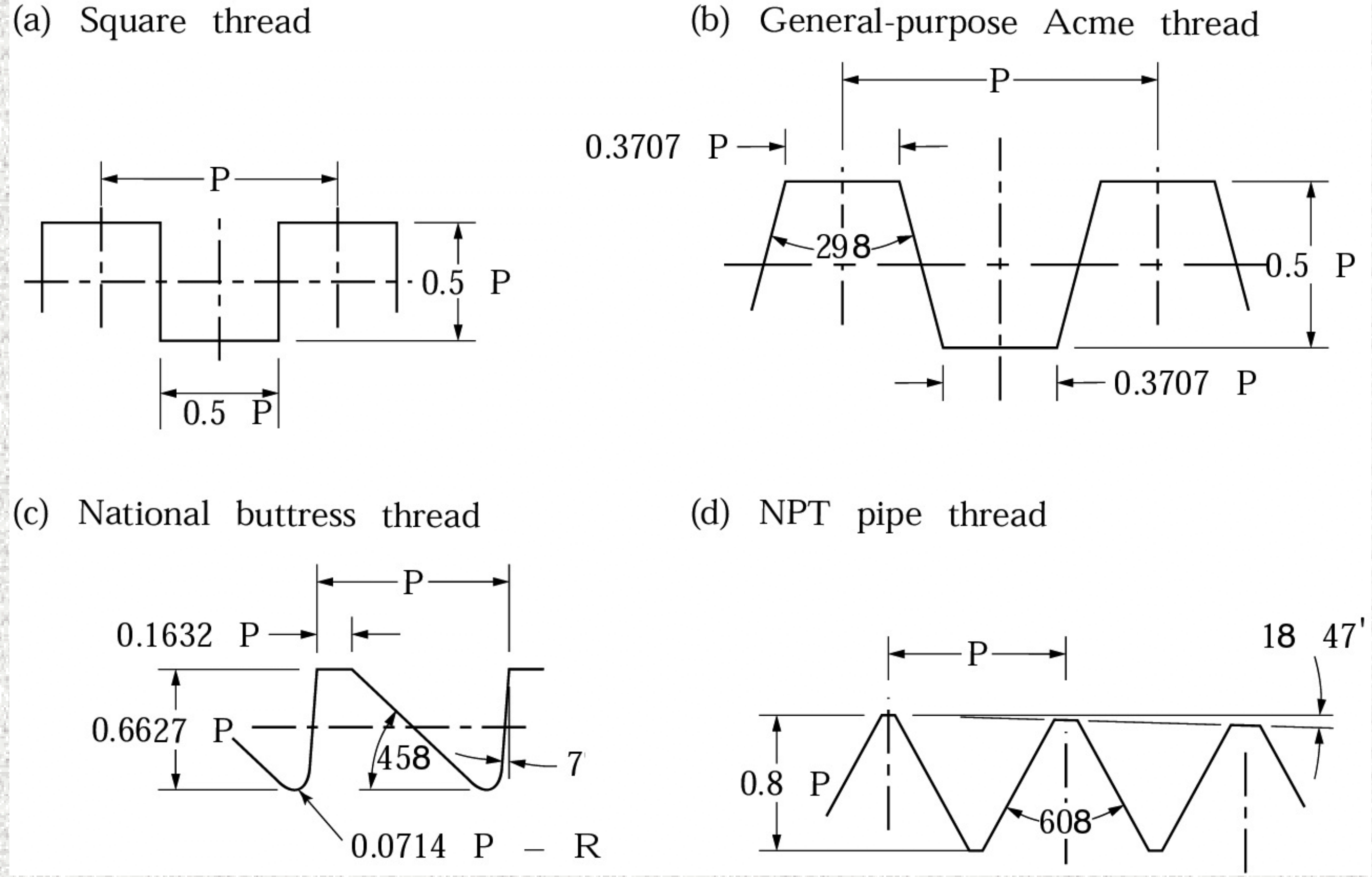


Figure 22.17 Various types of screw threads.

Cutting Screw Threads

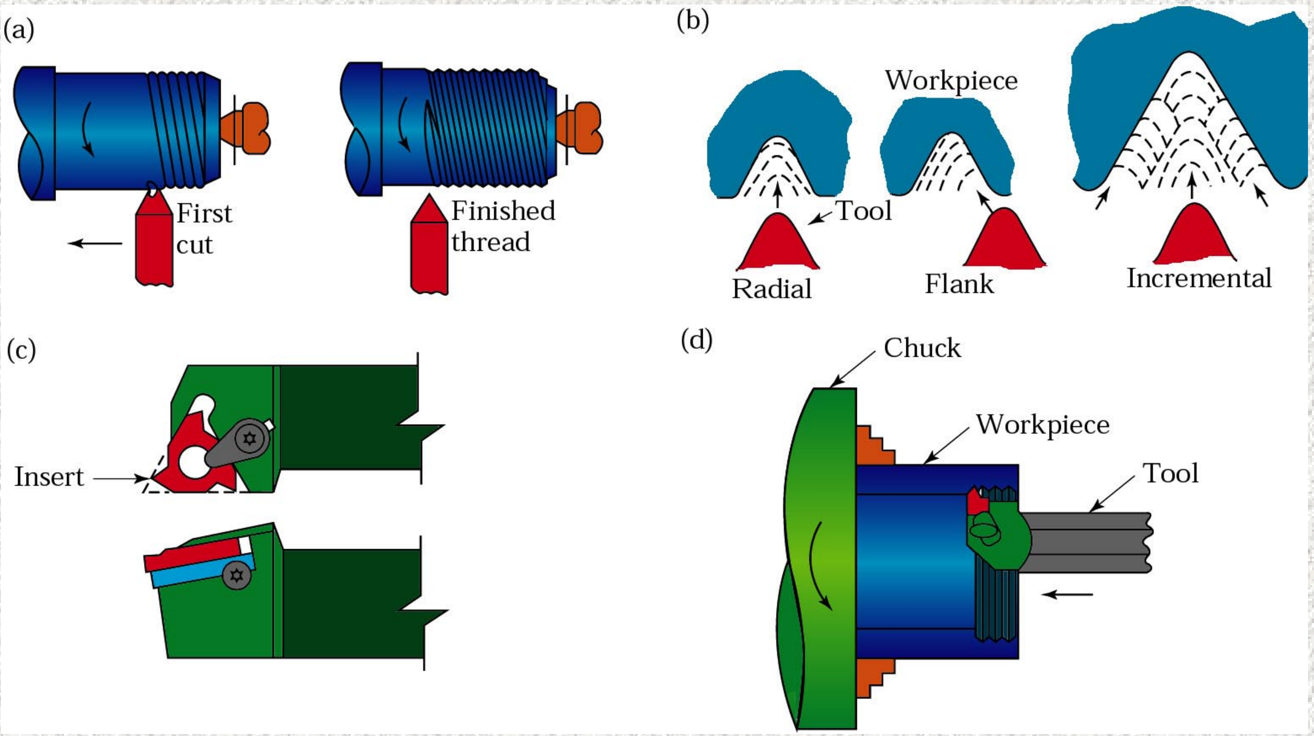


Figure 22.18 (a) Cutting screw threads on a lathe with a single-point cutting tool. (b) Cutting screw threads with a single-point tool in several passes, normally utilized for large threads. The small arrows in the figures show the direction of feed, and the broken lines show the position of the cutting 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 carbide insert and toolholder for cutting screw threads. (d) Cutting internal screw threads with a carbide insert. (See also Figs. 21.2 and 21.3.)

Threading Die

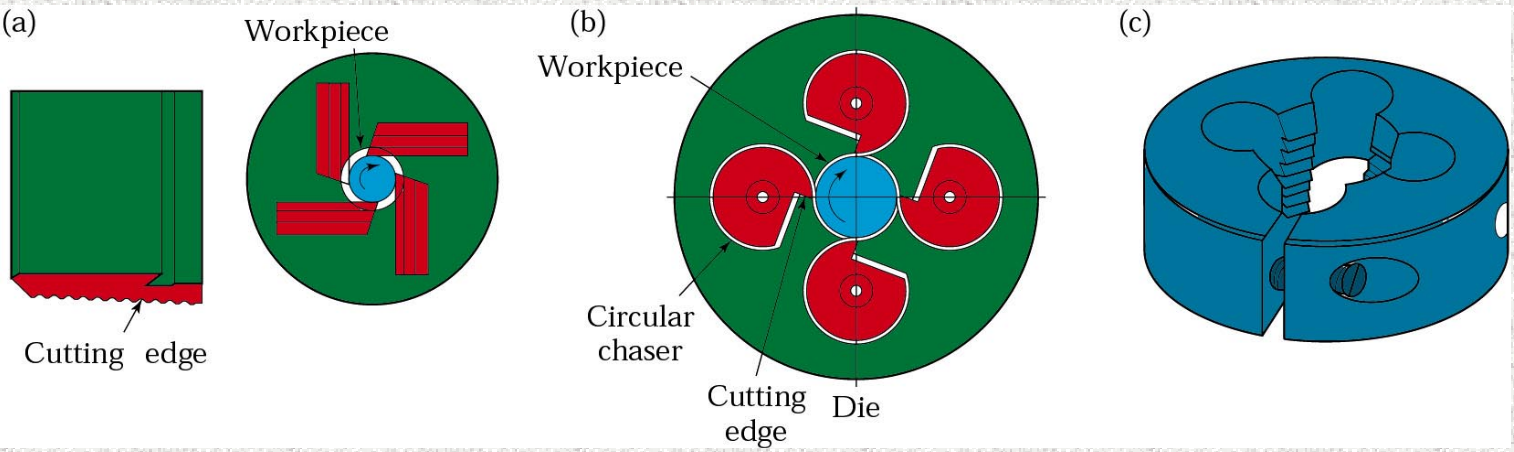
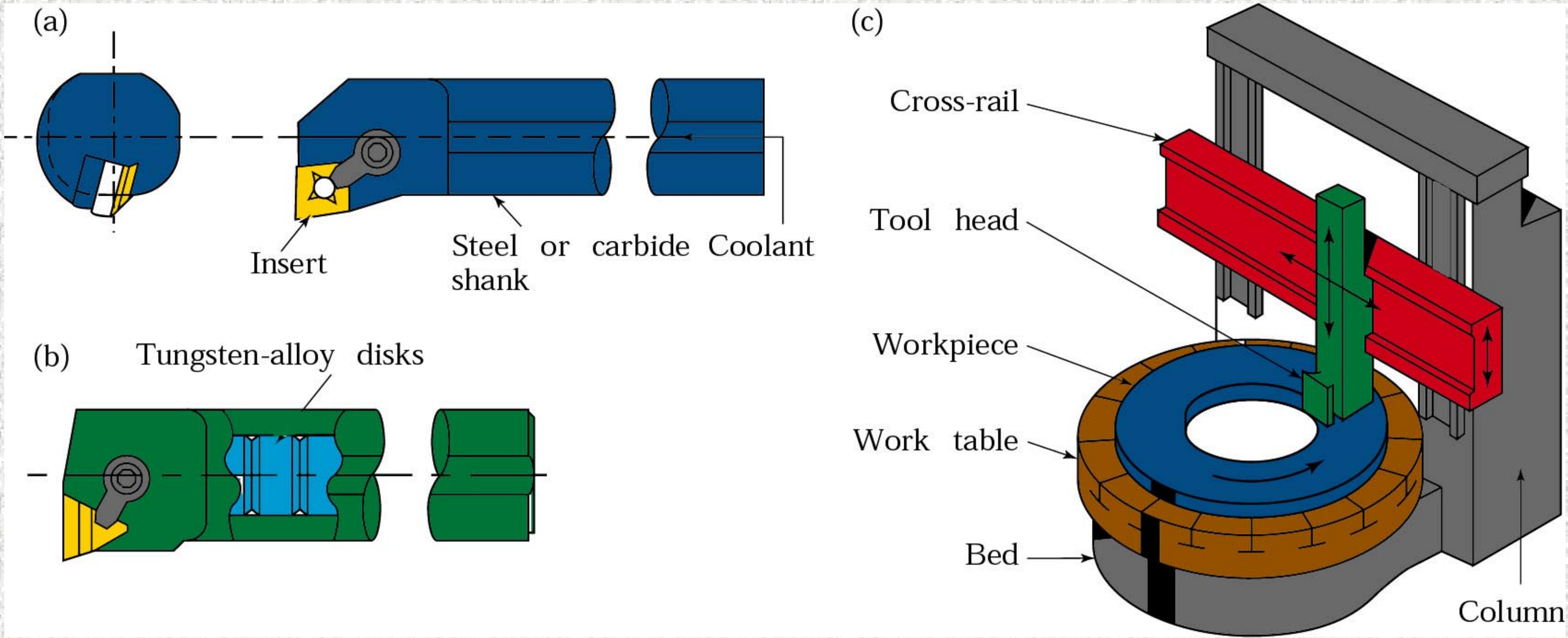


Figure 22.19 (a) Straight chasers for cutting threads on a lathe. (b) Circular chasers. (c) A solid threading die.

Boring

Figure 22.20 (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. (c) Schematic illustration of the components of a vertical boring mill. *Source*: Kennametal Inc.



Horizontal Boring Mill

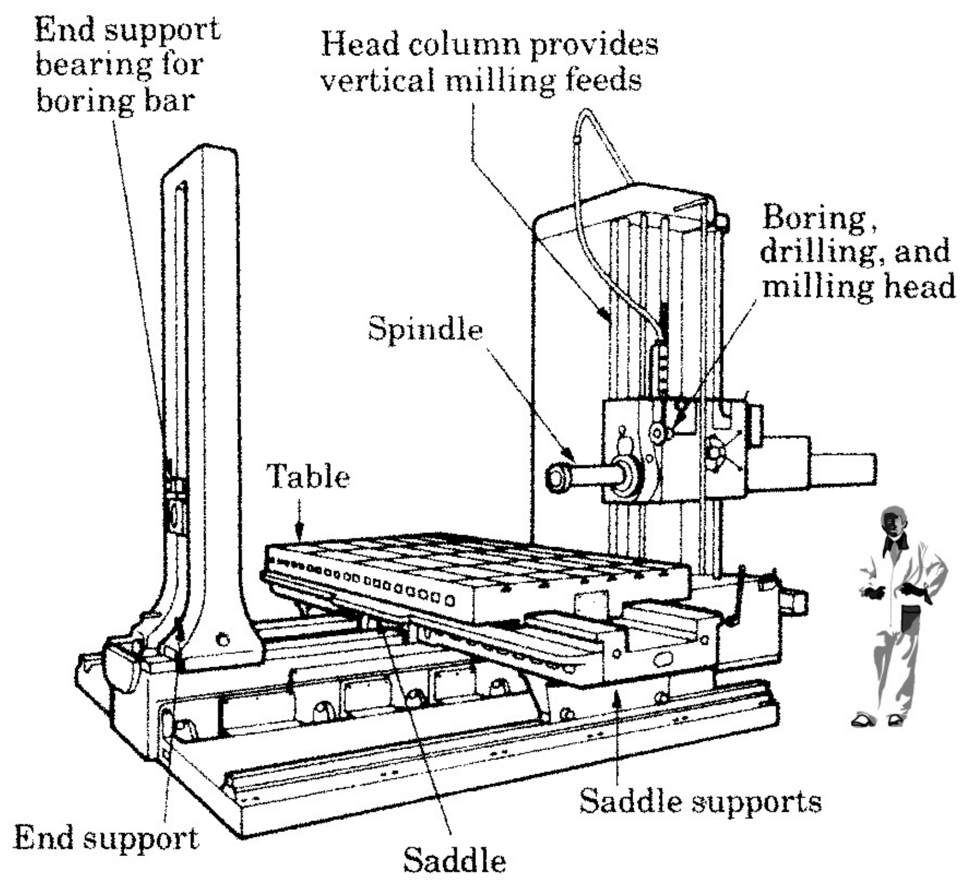


Figure 22.21 Horizontal boring mill. *Source*: Giddings and Lewis, Inc.

Drills

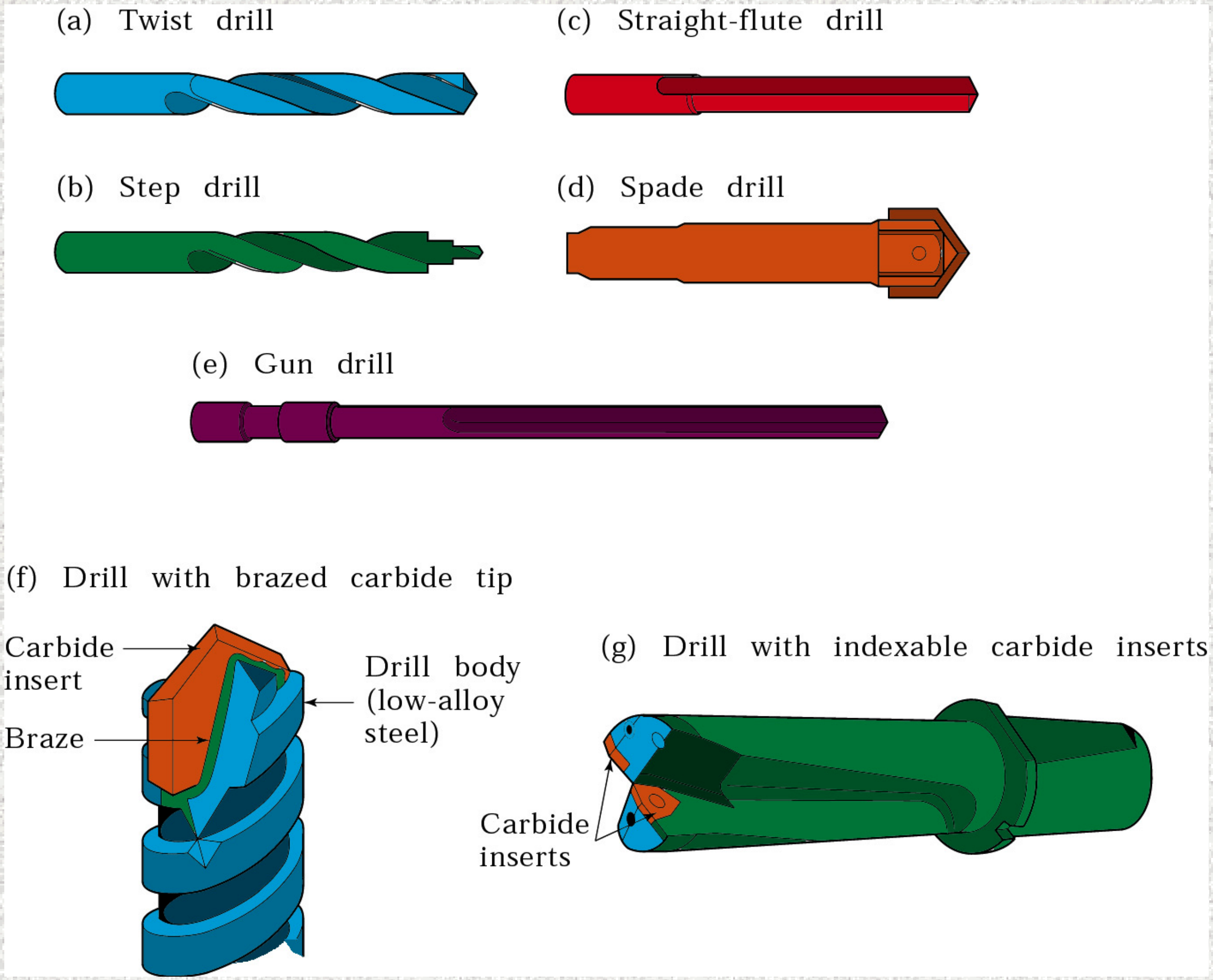


Figure 22.2
Various types
of drills

Drill Point Geometries

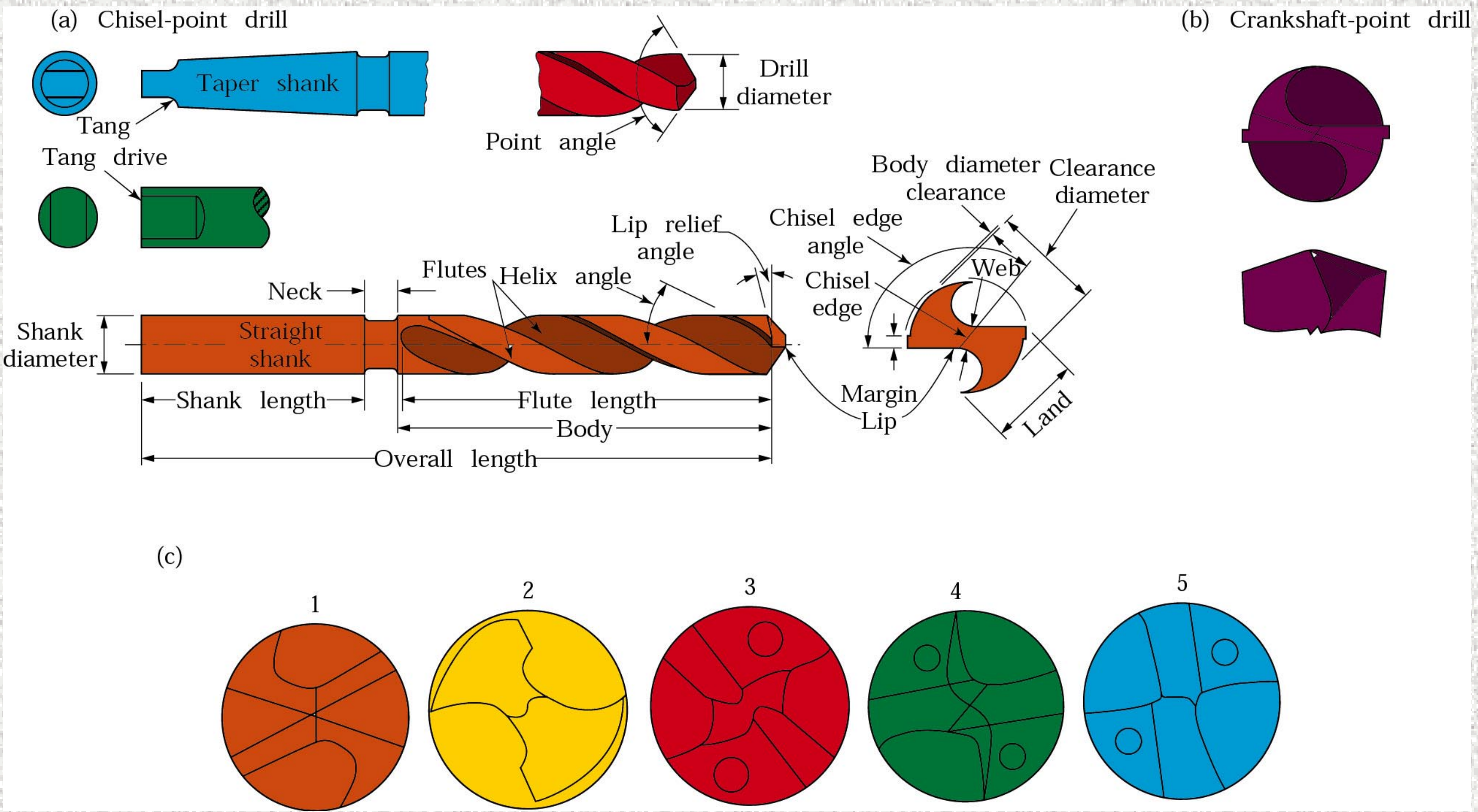


Figure 22.23 (a) Standard chisel-point drill indicating various features. The function of the pair of margins is to provide a bearing surface for the drill against walls of the hole as it penetrates into the workpiece; drills with four margins (double-margin) are available for improved drill guidance and accuracy. Drills with chip-breaker features are also available. (b) Crankshaft-point drill. (c) Various drill points and their manufacturers: 1. Four-facet split point, by Komet of America. 2. SE point, by Hertel. 3. New point, by Mitsubishi Materials. 4. Hosoi point, by OSG Tap and Die. 5. Helical point.

General Recommendations for Drill Geometry

TABLE 22.10 General Recommendations for Drill Geometry for High-Speed Twist Drills

Workpiece material	**Point angle**	**Lip-relief angle**	**Chisel-edge angle**	**Helix angle**	**Point**
Aluminum alloys	90–118	12–15	125–135	24–48	Standard
Magnesium alloys	70–118	12–15	120–135	30–45	Standard
Copper alloys	118	12–15	125–135	10–30	Standard
Steels	118	10–15	125–135	24–32	Standard
High-strength steels	118–135	7–10	125–135	24–32	Crankshaft
Stainless steels, low strength	118	10–12	125–135	24–32	Standard
Stainless steels, high strength	118–135	7–10	120–130	24–32	Crankshaft
High-temp. alloys	118–135	9–12	125–135	15–30	Crankshaft
Refractory alloys	118	7–10	125–135	24–32	Standard
Titanium alloys	118–135	7–10	125–135	15–32	Crankshaft
Cast irons	118	8–12	125–135	24–32	Standard
Plastics	60–90	7	120–135	29	Standard

Drilling and Reaming Operations

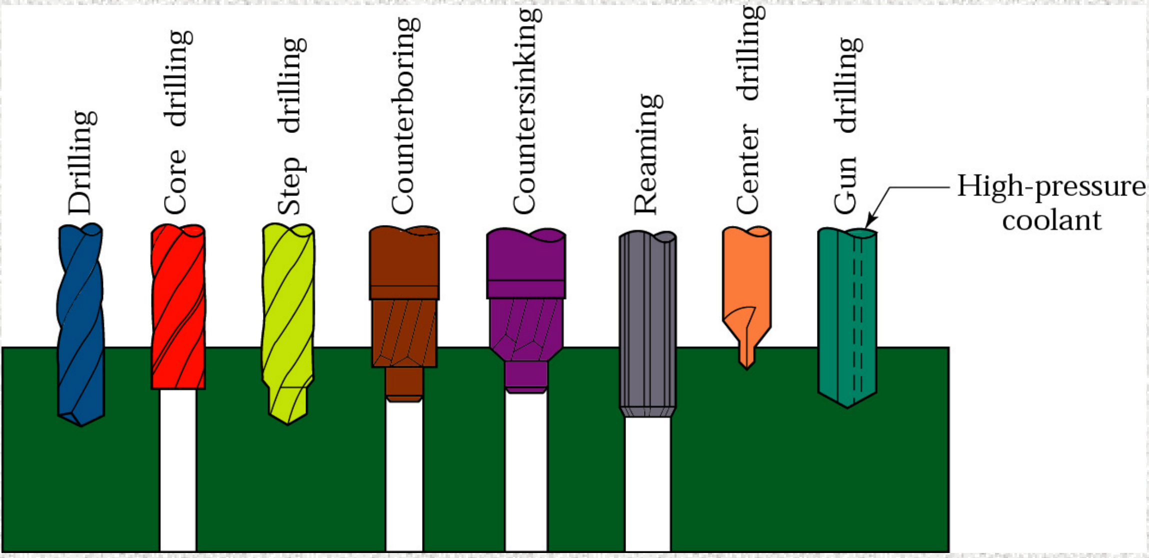
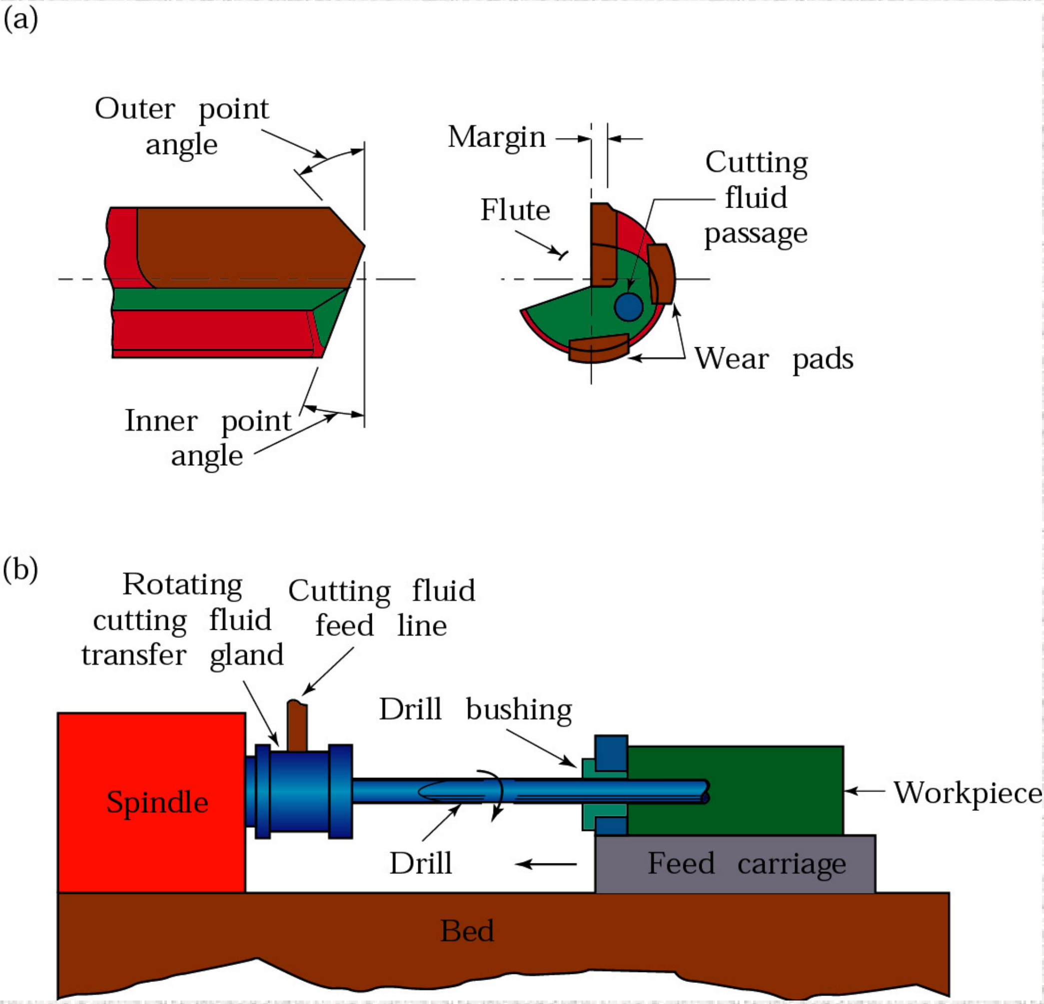


Figure 22.24 Various types of drilling and reaming operations.

Gun Drilling

Figure 22.25 (a) A gun drill showing various features. (b) Method of gun drilling. *Source*: Eldorado Tool and Manufacturing Corporation.



Trepanning

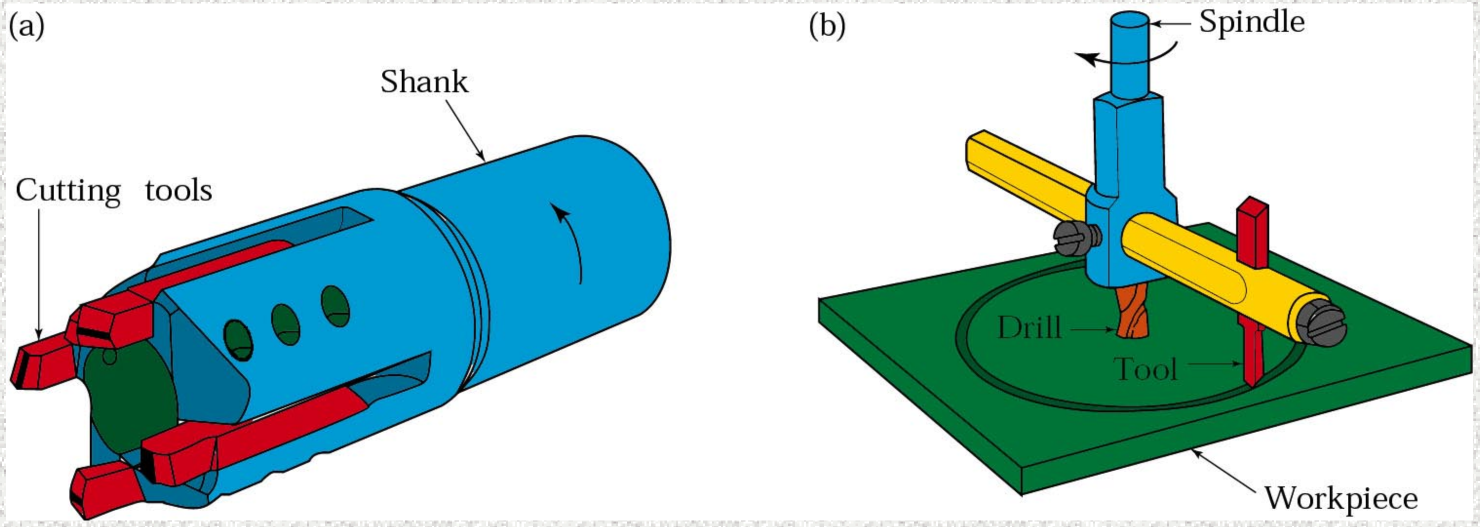


Figure 22.26 (a) Trepanning tool. (b) Trepanning with a drill-mounted single cutter.

Capabilities of Drilling and Boring Operations

TABLE 22.11

Tool type	Diameter range (mm)	Hole depth/diameter	
		Typical	**Maximum**
Twist	0.5–150	8	50
Spade	25–150	30	100
Gun	2–50	100	300
Trepanning	40–250	10	100
Boring	3–1200	5	8

General Recommendations for Speeds and Feeds in Drilling

TABLE 22.12

Workpiece material	Surface speed		Feed, mm/rev (in/rev) Drill Diameter		RPM	
	m/min	**ft/min**	**1.5 mm (0.060 in.)**	**12.5 mm (0.5 in.)**	**1.5 mm**	**12.5 mm**
Aluminum alloys	30–120	100–400	0.025 (0.001)	0.30 (0.012)	6400–25,000	800–3000
Magnesium alloys	45–120	150–400	0.025 (0.001)	0.30 (0.012)	9600–25,000	1100–3000
Copper alloys	15–60	50–200	0.025 (0.001)	0.25 (0.010)	3200–12,000	400–1500
Steels	20–30	60–100	0.025 (0.001)	0.30 (0.012)	4300–6400	500–800
Stainless steels	10–20	40–60	0.025 (0.001)	0.18 (0.007)	2100–4300	250–500
Titanium alloys	6–20	20–60	0.010 (0.0004)	0.15 (0.006)	1300–4300	150–500
Cast irons	20–60	60–200	0.025 (0.001)	0.30 (0.012)	4300–12,000	500–1500
Thermoplastics	30–60	100–200	0.025 (0.001)	0.13 (0.005)	6400–12,000	800–1500
Thermosets	20–60	60–200	0.025 (0.001)	0.10 (0.004)	4300–12,000	500–1500

Note: As hole depth increases, speeds and feeds should be reduced. Selection of speeds and feeds also depends on the specific surface finish required.

General Troubleshooting and Drill Life

TABLE 22.12 General Troubleshooting Guide for Drilling Operations

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

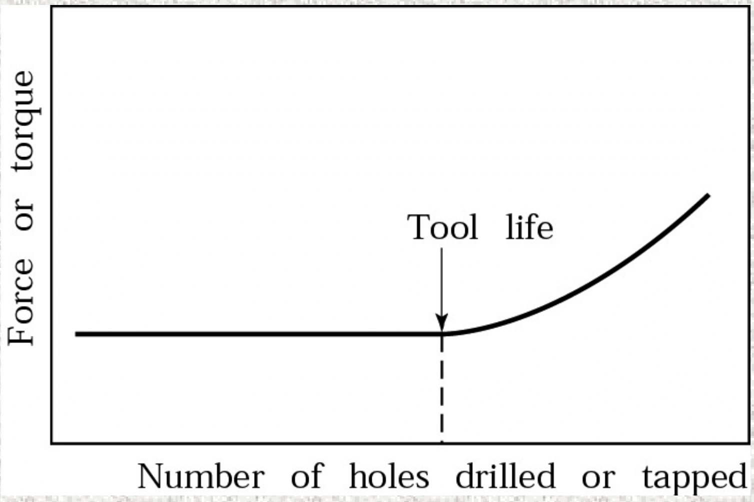


Figure 22.27 The determination of drill life by monitoring the rise in force or torque as a function of the number of holes drilled. This test is also used for determining tap life.

Drilling Machines

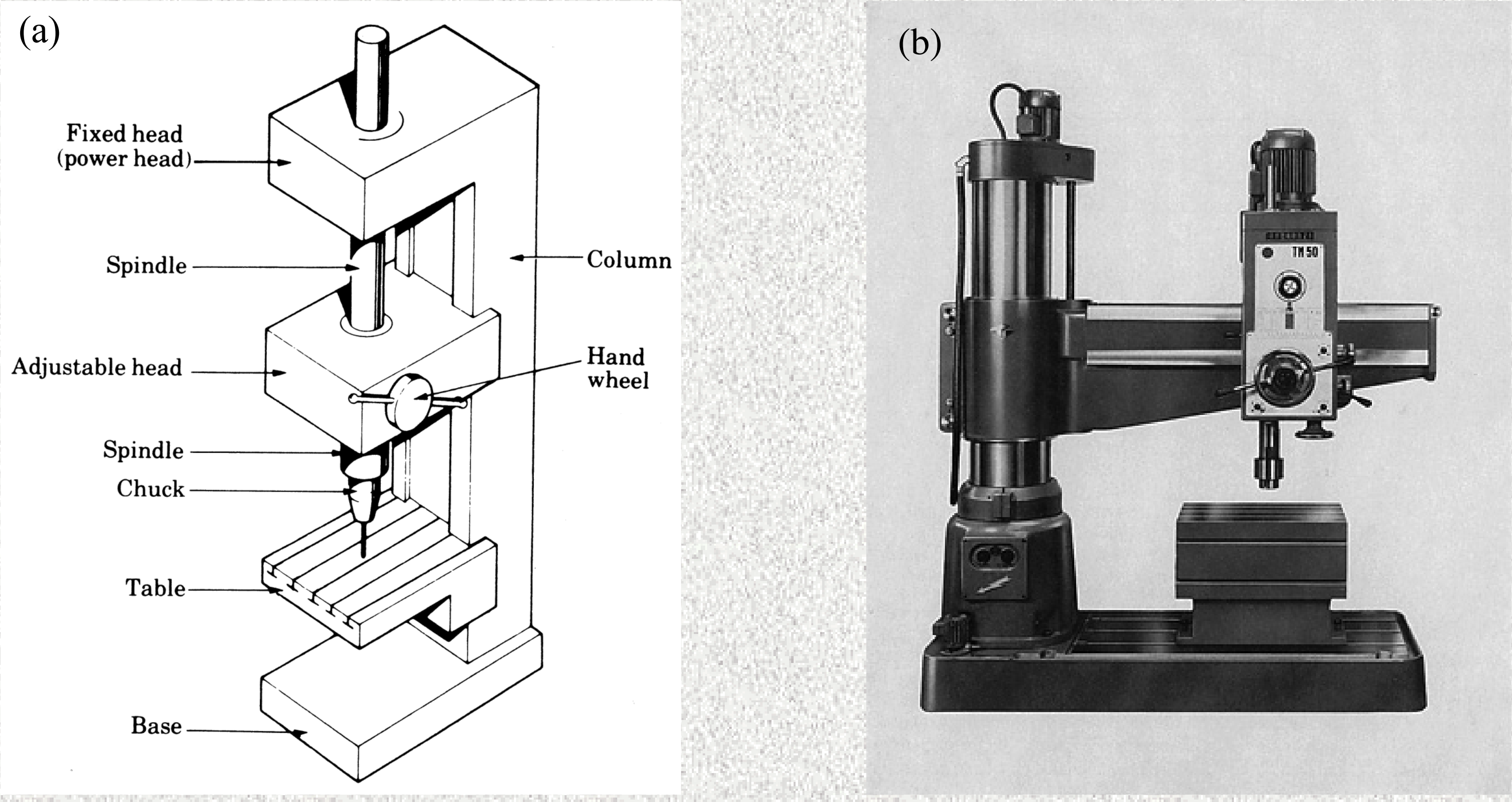


Figure 22.28 Schematic illustration of components of (a) a vertical drill press and (b) a radial drilling machine.

CNC Drilling Machine

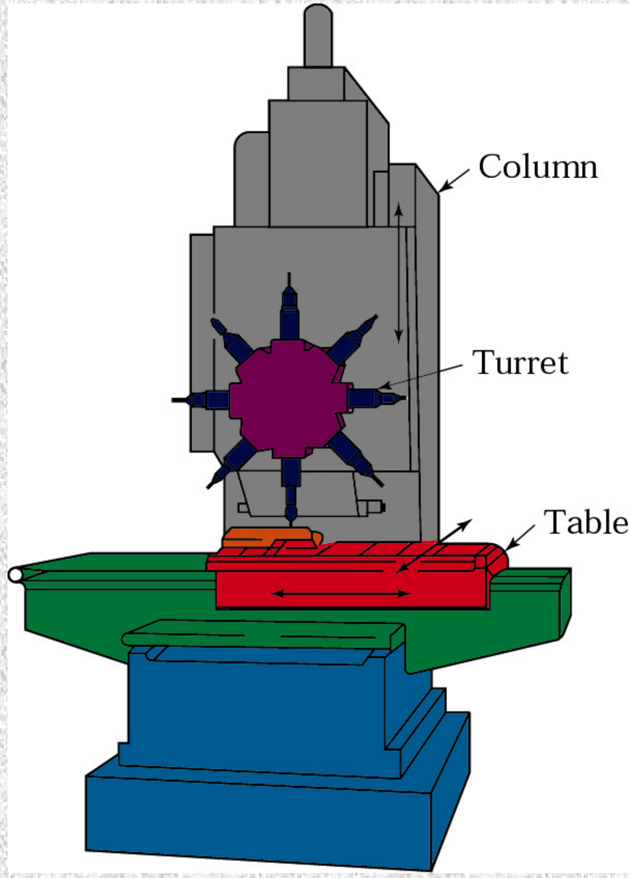


Figure 22.29 A three-axis computer numerical control drilling machine. The turret holds as much as eight different tools, such as drills, taps, and reamers.

Reamers

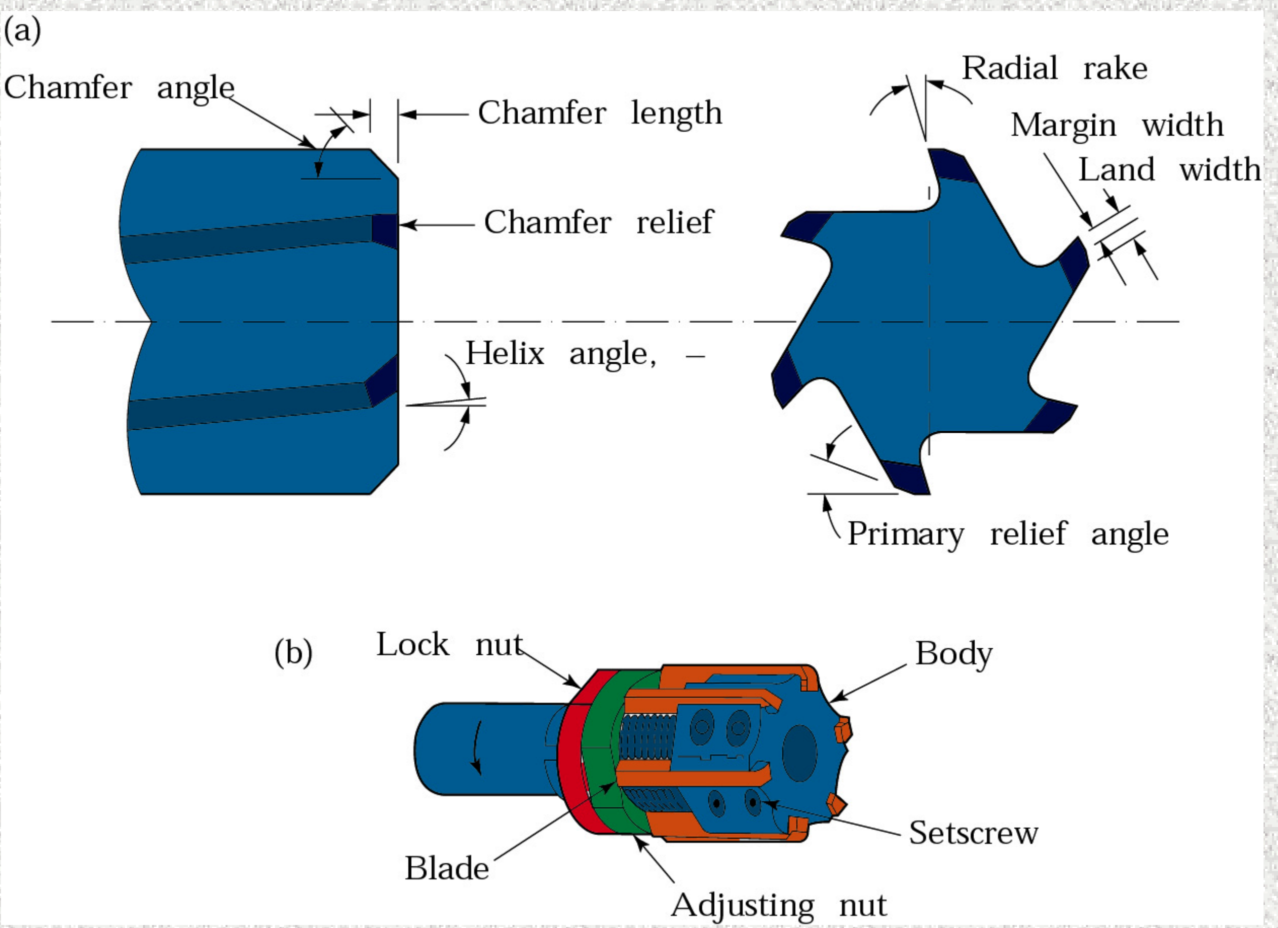


Figure 22.30 (a) Terminology for a helical reamer. (b) Inserted-blade adjustable reamer.

Tapping and Taps

Figure 22.31 (a) Terminology for a tap. (b) Tapping of steel nuts in production.