Fundamentals of Machining / Orthogonal Machining

Material Removal Process

- Types of machining processes
- Basic Chip Formation
- Mechanics of Machining

Metal Cutting Machining Processes

Machining: Removal of unwanted material in the form of chips in order to create a geometric shape

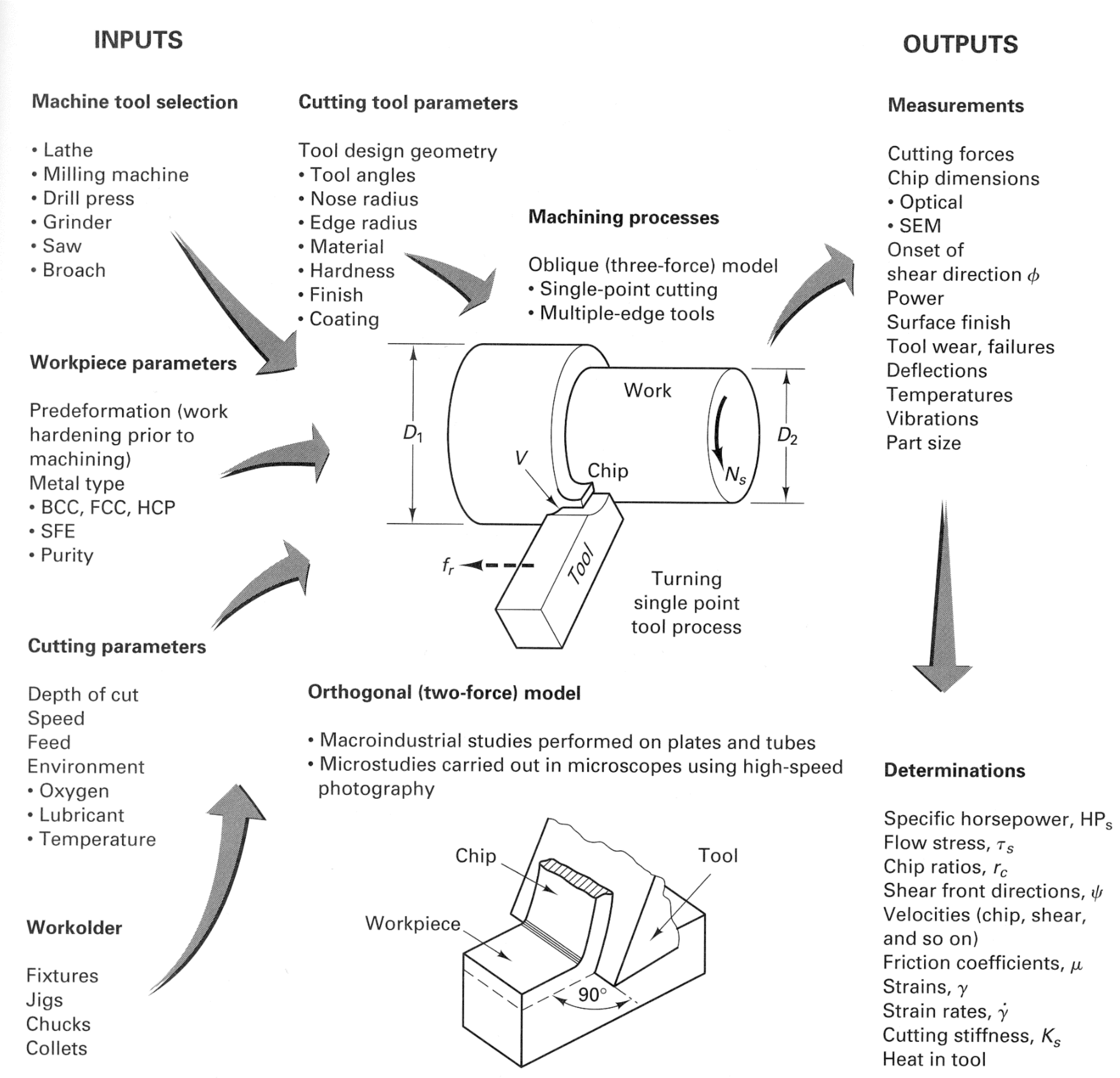
Market: $60b, **Tolerance:** 0.0001” or 2.5um

What makes the process unique and difficult?

The complexity induced by:

- difference between material properties
- process is unconstrained and asymmetrical
- Large stress and strain
- Non-robust process (sensitive to tool geometry, material, temp., cutting fluids, process dynamics, chatter, vibration, etc.)

Fundamentals



INPUTS

Machine tool selection

- Lathe
- Milling machine
- Drill press
- Grinder
- Saw
- Broach

Workpiece parameters

Predeformation (work hardening prior to machining)
Metal type
- BCC, FCC, HCP
- SFE
- Purity

Cutting parameters

Depth of cut
Speed
Feed
Environment
- Oxygen
- Lubricant
- Temperature

Workolder

Fixtures
Jigs
Chucks
Collets

Cutting tool parameters

Tool design geometry
- Tool angles
- Nose radius
- Edge radius
- Material
- Hardness
- Finish
- Coating

Machining processes

Oblique (three-force) model
- Single-point cutting
- Multiple-edge tools

Turning single point tool process

Orthogonal (two-force) model

- Macroindustrial studies performed on plates and tubes
- Microstudies carried out in microscopes using high-speed photography

OUTPUTS

Measurements

Cutting forces
Chip dimensions
- Optical
- SEM

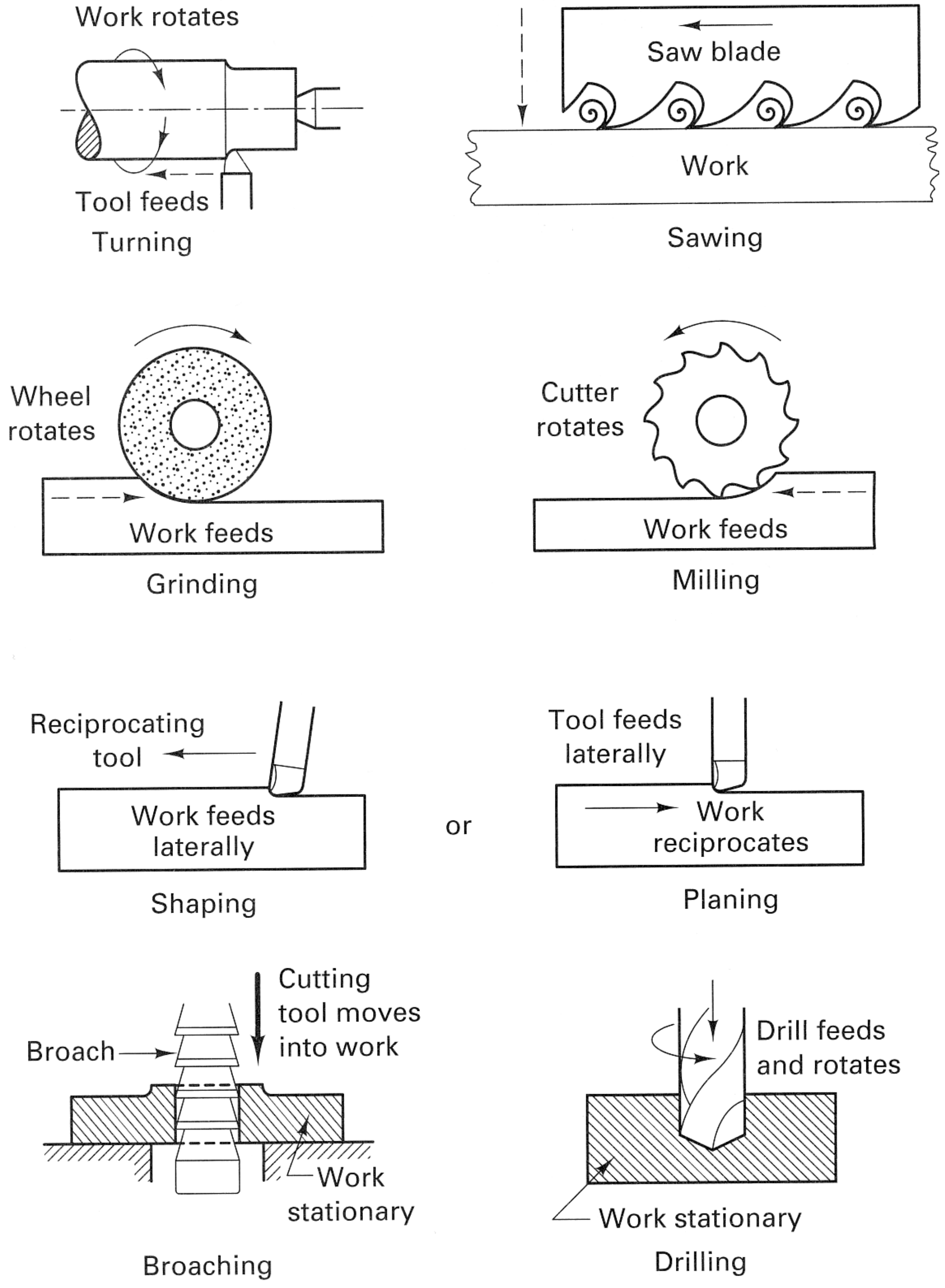
Onset of shear direction ϕ
Power
Surface finish
Tool wear, failures
Deflections
Temperatures
Vibrations
Part size

Determinations

Specific horsepower, HP_s
Flow stress, τ_s
Chip ratios, r_c
Shear front directions, ψ
Velocities (chip, shear, and so on)
Friction coefficients, μ
Strains, γ
Strain rates, $\dot{\gamma}$
Cutting stiffness, K_s
Heat in tool

FIGURE 21-1 The fundamental inputs and outputs to machining processes.

Seven basic chip formation processes



Turning

Sawing

Grinding

Milling

Shaping

Planing

Broaching

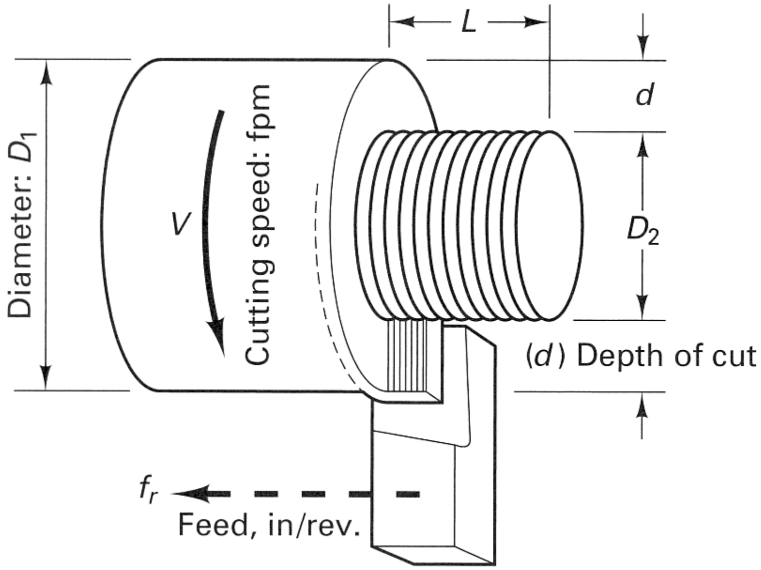
Drilling

FIGURE 21-2 The seven basic machining processes used in chip formation.

Seven type of processes:

1. **Shaping and planing**
2. **Turning**
3. **Milling**
4. **Drilling**
5. **Sawing**
6. **Broaching**
7. **Grinding (abrasive machining)**

Basic machining processes in detail

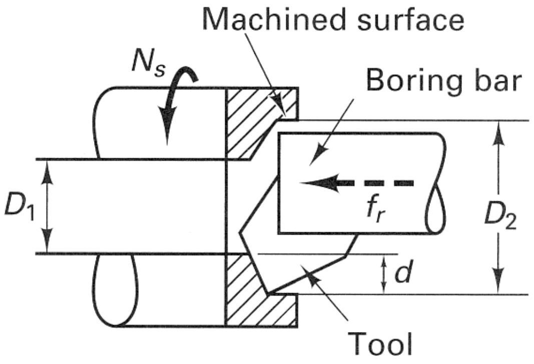


Turning

Speed, stated in surface feet per minute (sfpm), is the peripheral speed at the cutting edge. Feed per revolution in turning is a linear motion of the tool parallel to the rotating axis of the workpiece. The depth of cut reflects the third dimension.

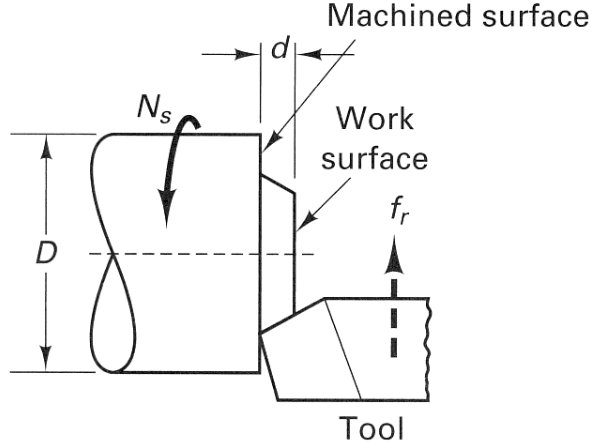
L = length of cut

$$T_m = \frac{L + A}{f_r N_s}$$



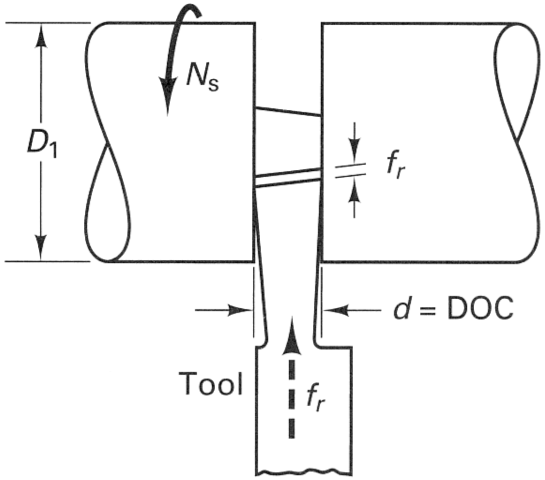
Boring

Enlarging hole of diameter D_1 to diameter D_2. Boring can be done with multiple cutting tools. Feed in inches per revolution, f_r.



Facing

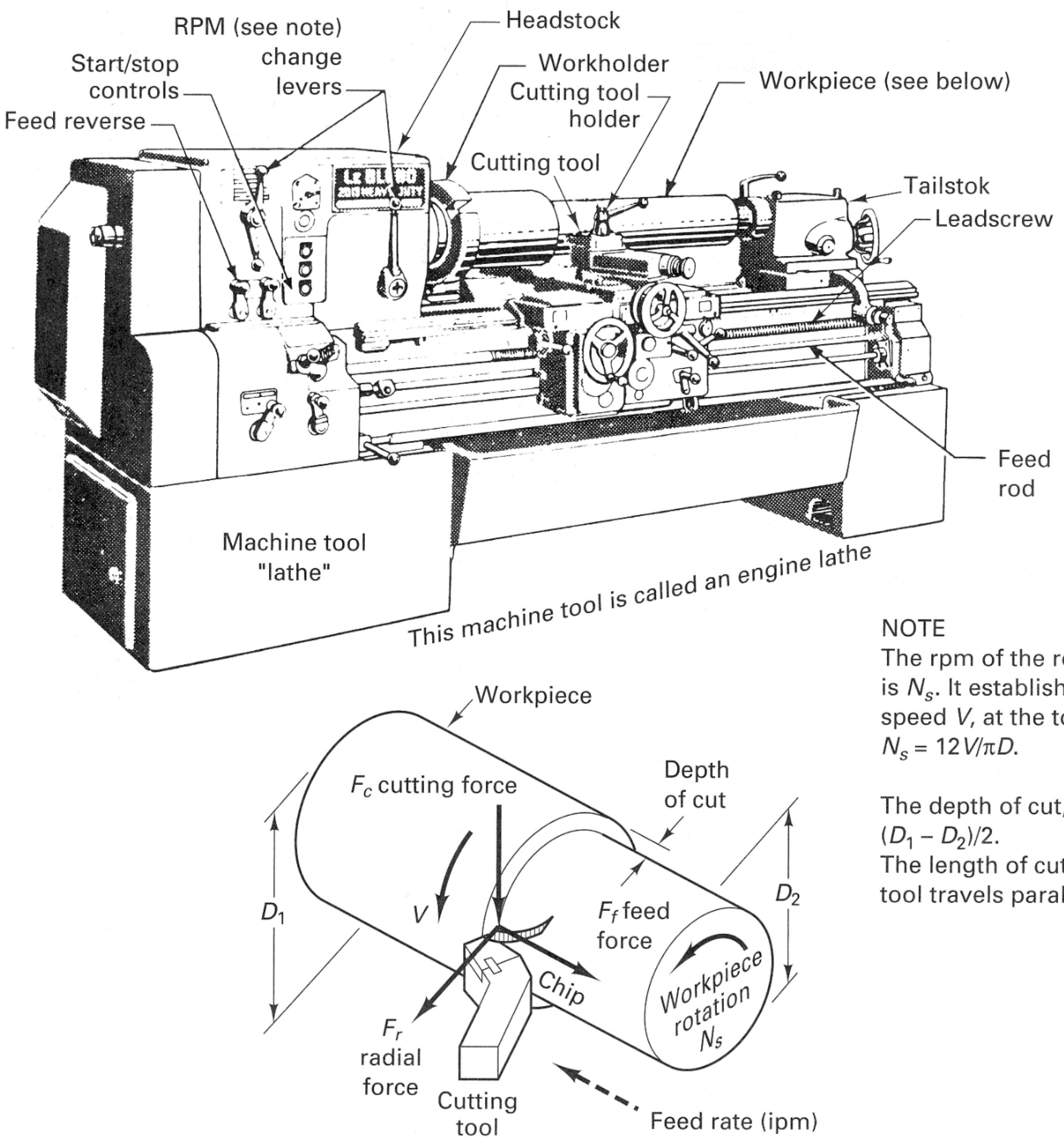
Tool feeds to center of workpiece so $L = D/2$. The cutting speed is decreasing as the tool approaches the center of the workpiece.



Grooving, parting or cutoff

Tool feed perpendicular to the axis of rotation. The width of the tool produces the depth of cut (DOC).

Definitions: Speed, feed, and depth of cut (DOC)



NOTE
The rpm of the rotating workpiece is N_s. It establishes the cutting speed V, at the tool, according to $N_s = 12V/\pi D$.

The depth of cut, d, is equal to $(D_1 - D_2)/2$.
The length of cut is the distance the tool travels parallel to the axis, L.

FIGURE 21-3 Turning a cylindrical workpiece on a lathe requires you select the cutting speed, feed, and depth of cut.

$$\text{DOC} = \frac{D_1 - D_2}{2} = d$$

$$V = \frac{\pi D_1 N_s}{12}$$

Basic Chip formation process:

Chips in metal cutting: **Due to shear of material under compression generated by the cutting motion**

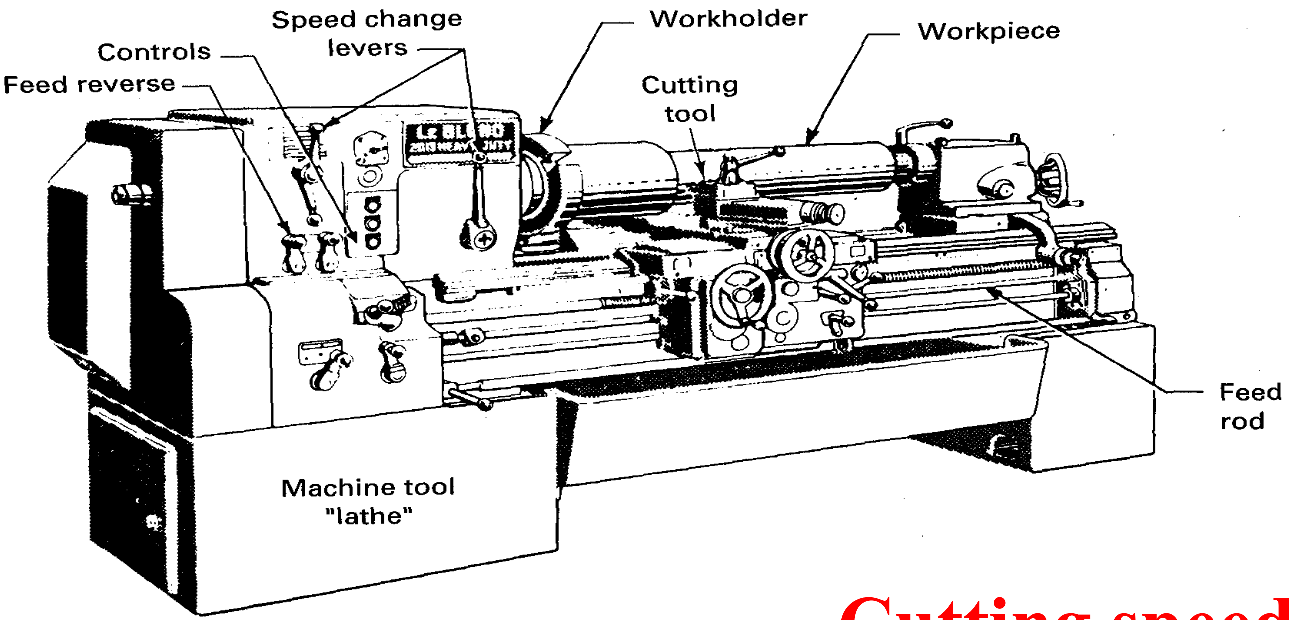
Basic Parameters:

Speed or cutting speed, (V), ***primary cutting motion*****,** f/min, in/min, m/min

Feed (f_r), ***material removed / revolution*****,** in/rev, m/rev, m/min

Chip thickness (depth of cut, DOC), d, ***Tool plunged distance, m, in***

Basic Machine Tools Lathe (Turning)



Recommended Machining Parameters

Cutting speed → rpm

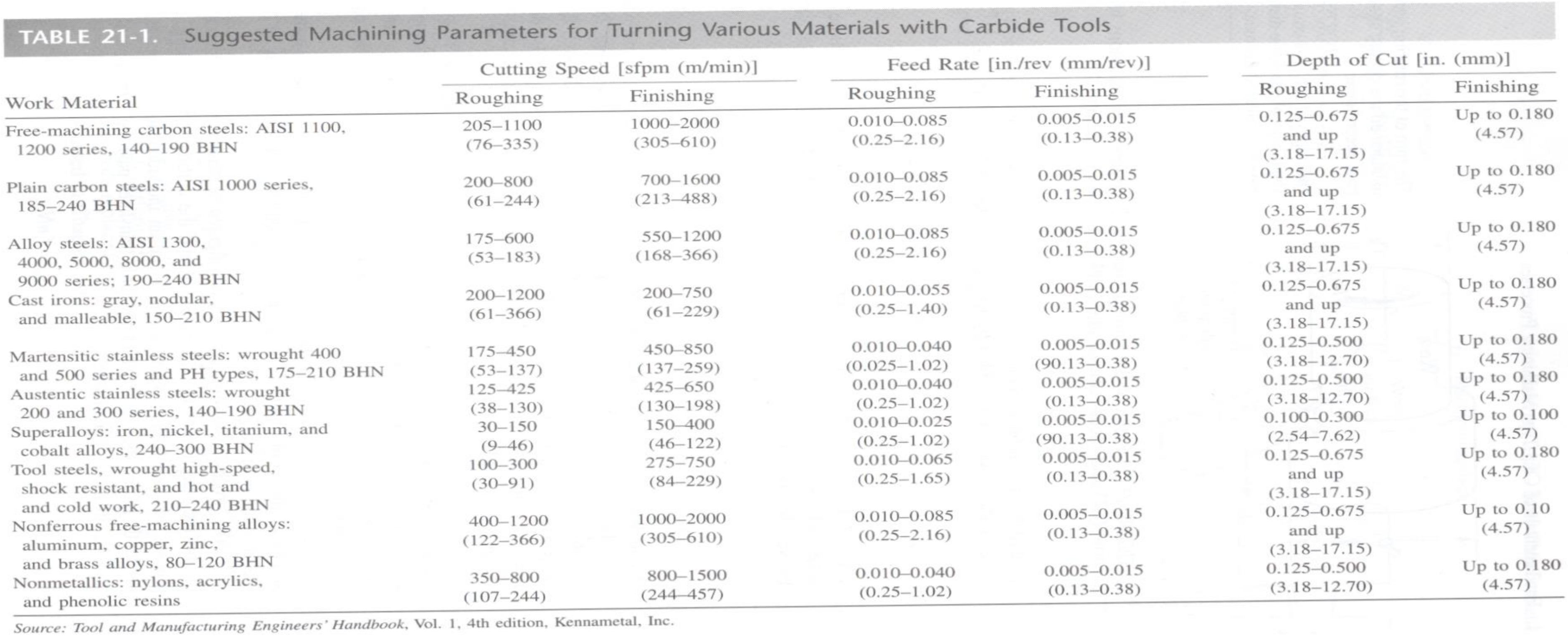


TABLE 21-1. Suggested Machining Parameters for Turning Various Materials with Carbide Tools

Work Material	Cutting Speed [sfpm (m/min)]		Feed Rate [in./rev (mm/rev)]		Depth of Cut [in. (mm)]	
	Roughing	Finishing	Roughing	Finishing	Roughing	Finishing
Free-machining carbon steels: AISI 1100, 1200 series, 140–190 BHN	205–1100 (76–335)	1000–2000 (305–610)	0.010–0.085 (0.25–2.16)	0.005–0.015 (0.13–0.38)	0.125–0.675 and up (3.18–17.15)	Up to 0.180 (4.57)
Plain carbon steels: AISI 1000 series, 185–240 BHN	200–800 (61–244)	700–1600 (213–488)	0.010–0.085 (0.25–2.16)	0.005–0.015 (0.13–0.38)	0.125–0.675 and up (3.18–17.15)	Up to 0.180 (4.57)
Alloy steels: AISI 1300, 4000, 5000, 8000, and 9000 series; 190–240 BHN	175–600 (53–183)	550–1200 (168–366)	0.010–0.085 (0.25–2.16)	0.005–0.015 (0.13–0.38)	0.125–0.675 and up (3.18–17.15)	Up to 0.180 (4.57)
Cast irons: gray, nodular, and malleable, 150–210 BHN	200–1200 (61–366)	200–750 (61–229)	0.010–0.055 (0.25–1.40)	0.005–0.015 (0.13–0.38)	0.125–0.675 and up (3.18–17.15)	Up to 0.180 (4.57)
Martensitic stainless steels: wrought 400 and 500 series and PH types, 175–210 BHN	175–450 (53–137)	450–850 (137–259)	0.010–0.040 (0.025–1.02)	0.005–0.015 (90.13–0.38)	0.125–0.500 (3.18–12.70)	Up to 0.180 (4.57)
Austentic stainless steels: wrought 200 and 300 series, 140–190 BHN	125–425 (38–130)	425–650 (130–198)	0.010–0.040 (0.25–1.02)	0.005–0.015 (0.13–0.38)	0.125–0.500 (3.18–12.70)	Up to 0.180 (4.57)
Superalloys: iron, nickel, titanium, and cobalt alloys, 240–300 BHN	30–150 (9–46)	150–400 (46–122)	0.010–0.025 (0.25–1.02)	0.005–0.015 (90.13–0.38)	0.100–0.300 (2.54–7.62)	Up to 0.100 (4.57)
Tool steels, wrought high-speed, shock resistant, and hot and and cold work, 210–240 BHN	100–300 (30–91)	275–750 (84–229)	0.010–0.065 (0.25–1.65)	0.005–0.015 (0.13–0.38)	0.125–0.675 and up (3.18–17.15)	Up to 0.180 (4.57)
Nonferrous free-machining alloys: aluminum, copper, zinc, and brass alloys, 80–120 BHN	400–1200 (122–366)	1000–2000 (305–610)	0.010–0.085 (0.25–2.16)	0.005–0.015 (0.13–0.38)	0.125–0.675 and up (3.18–17.15)	Up to 0.10 (4.57)
Nonmetallics: nylons, acrylics, and phenolic resins	350–800 (107–244)	800–1500 (244–457)	0.010–0.040 (0.25–1.02)	0.005–0.015 (0.13–0.38)	0.125–0.500 (3.18–12.70)	Up to 0.180 (4.57)

Source: Tool and Manufacturing Engineers' Handbook, Vol. 1, 4th edition, Kennametal, Inc.

Example

Material: Carbon Steel

Assume the Max. Allowed Cutting Speed, V : 305 m/min
Feed rate, F_r : 0.13 mm/rev
Depth of cut, d : 4.57 mm

What is the maximum rotational speed (N, in RPM) allowed for turning 100mm dia rod.?

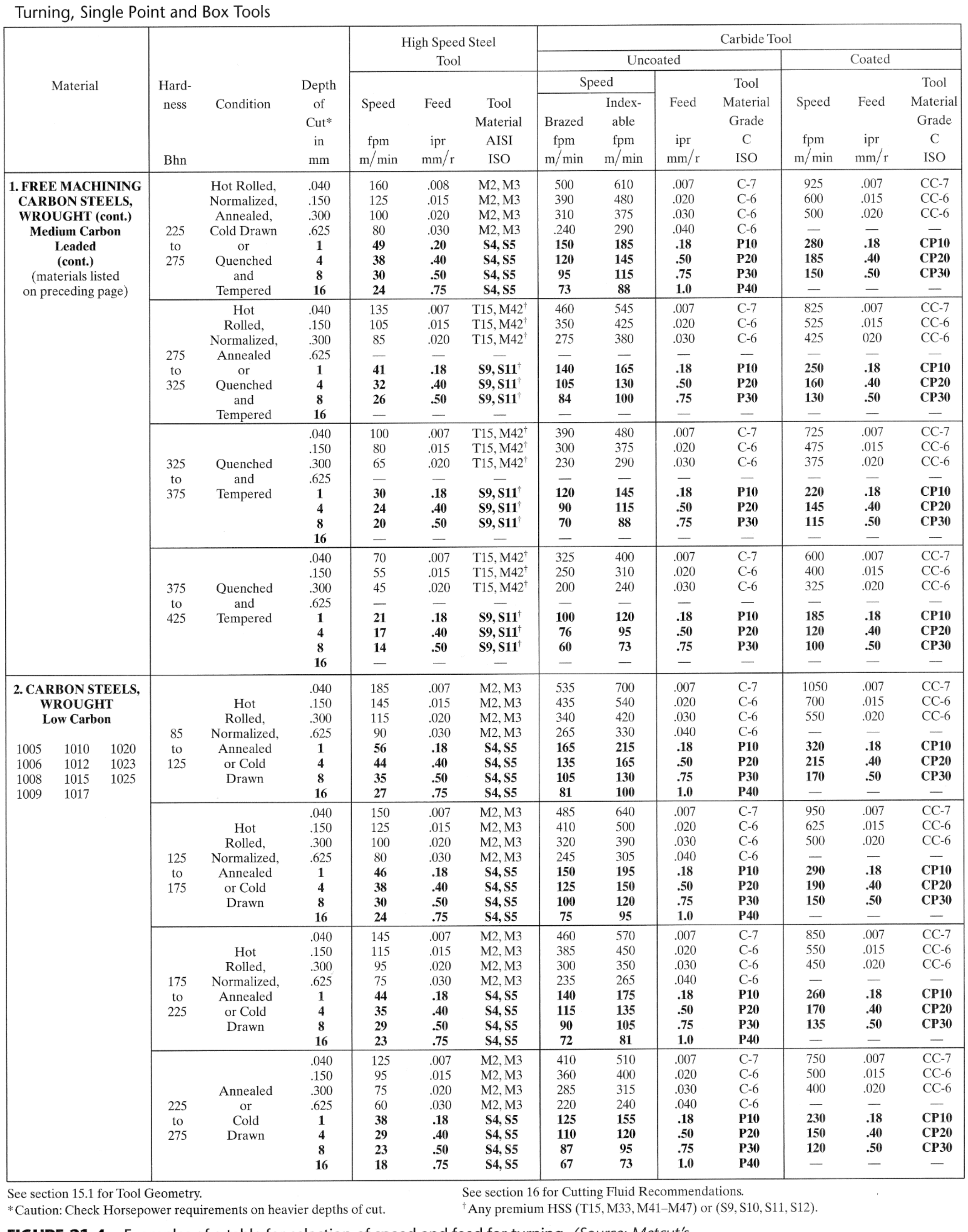
$$v = \pi D_1 N, m/\text{min}, D_1 in\ m$$

$$N = v/\pi D_1, \text{rpm}$$

$$= 305/(\pi * 100\text{x}10^{-3})$$

$$= 970.8\ \text{rpm}$$

Example: A table for selection of speed and feed for turning



Turning, Single Point and Box Tools

Material	Hard-ness Bhn	Condition	Depth of Cut* in mm	High Speed Steel Tool			Carbide Tool						
							Uncoated				Coated		
				Speed fpm m/min	Feed ipr mm/r	Tool Material AISI ISO	Speed Brazed fpm m/min	Speed Index-able fpm m/min	Feed ipr mm/r	Tool Material Grade C ISO	Speed fpm m/min	Feed ipr mm/r	Tool Material Grade C ISO
1. FREE MACHINING CARBON STEELS, WROUGHT (cont.) Medium Carbon Leaded (cont.) (materials listed on preceding page)	225 to 275	Hot Rolled, Normalized, Annealed, Cold Drawn or Quenched and Tempered	.040	160	.008	M2, M3	500	610	.007	C-7	925	.007	CC-7
			.150	125	.015	M2, M3	390	480	.020	C-6	600	.015	CC-6
			.300	100	.020	M2, M3	310	375	.030	C-6	500	.020	CC-6
			.625	80	.030	M2, M3	.240	290	.040	C-6	—	—	—
			1	**49**	**.20**	**S4, S5**	**150**	**185**	**.18**	**P10**	**280**	**.18**	**CP10**
			4	**38**	**.40**	**S4, S5**	**120**	**145**	**.50**	**P20**	**185**	**.40**	**CP20**
			8	**30**	**.50**	**S4, S5**	**95**	**115**	**.75**	**P30**	**150**	**.50**	**CP30**
			16	**24**	**.75**	**S4, S5**	**73**	**88**	**1.0**	**P40**	—	—	—
	275 to 325	Hot Rolled, Normalized, Annealed or Quenched and Tempered	.040	135	.007	T15, M42[†]	460	545	.007	C-7	825	.007	CC-7
			.150	105	.015	T15, M42[†]	350	425	.020	C-6	525	.015	CC-6
			.300	85	.020	T15, M42[†]	275	380	.030	C-6	425	020	CC-6
			.625	—	—	—	—	—	—	—	—	—	—
			1	**41**	**.18**	**S9, S11**[†]	**140**	**165**	**.18**	**P10**	**250**	**.18**	**CP10**
			4	**32**	**.40**	**S9, S11**[†]	**105**	**130**	**.50**	**P20**	**160**	**.40**	**CP20**
			8	**26**	**.50**	**S9, S11**[†]	**84**	**100**	**.75**	**P30**	**130**	**.50**	**CP30**
			16	—	—	—	—	—	—	—	—	—	—
	325 to 375	Quenched and Tempered	.040	100	.007	T15, M42[†]	390	480	.007	C-7	725	.007	CC-7
			.150	80	.015	T15, M42[†]	300	375	.020	C-6	475	.015	CC-6
			.300	65	.020	T15, M42[†]	230	290	.030	C-6	375	.020	CC-6
			.625	—	—	—	—	—	—	—	—	—	—
			1	**30**	**.18**	**S9, S11**[†]	**120**	**145**	**.18**	**P10**	**220**	**.18**	**CP10**
			4	**24**	**.40**	**S9, S11**[†]	**90**	**115**	**.50**	**P20**	**145**	**.40**	**CP20**
			8	**20**	**.50**	**S9, S11**[†]	**70**	**88**	**.75**	**P30**	**115**	**.50**	**CP30**
			16	—	—	—	—	—	—	—	—	—	—
	375 to 425	Quenched and Tempered	.040	70	.007	T15, M42[†]	325	400	.007	C-7	600	.007	CC-7
			.150	55	.015	T15, M42[†]	250	310	.020	C-6	400	.015	CC-6
			.300	45	.020	T15, M42[†]	200	240	.030	C-6	325	.020	CC-6
			.625	—	—	—	—	—	—	—	—	—	—
			1	**21**	**.18**	**S9, S11**[†]	**100**	**120**	**.18**	**P10**	**185**	**.18**	**CP10**
			4	**17**	**.40**	**S9, S11**[†]	**76**	**95**	**.50**	**P20**	**120**	**.40**	**CP20**
			8	**14**	**.50**	**S9, S11**[†]	**60**	**73**	**.75**	**P30**	**100**	**.50**	**CP30**
			16	—	—	—	—	—	—	—	—	—	—
2. CARBON STEELS, WROUGHT Low Carbon 1005 1010 1020 1006 1012 1023 1008 1015 1025 1009 1017	85 to 125	Hot Rolled, Normalized, Annealed or Cold Drawn	.040	185	.007	M2, M3	535	700	.007	C-7	1050	.007	CC-7
			.150	145	.015	M2, M3	435	540	.020	C-6	700	.015	CC-6
			.300	115	.020	M2, M3	340	420	.030	C-6	550	.020	CC-6
			.625	90	.030	M2, M3	265	330	.040	C-6	—	—	—
			1	**56**	**.18**	**S4, S5**	**165**	**215**	**.18**	**P10**	**320**	**.18**	**CP10**
			4	**44**	**.40**	**S4, S5**	**135**	**165**	**.50**	**P20**	**215**	**.40**	**CP20**
			8	**35**	**.50**	**S4, S5**	**105**	**130**	**.75**	**P30**	**170**	**.50**	**CP30**
			16	**27**	**.75**	**S4, S5**	**81**	**100**	**1.0**	**P40**	—	—	—
	125 to 175	Hot Rolled, Normalized, Annealed or Cold Drawn	.040	150	.007	M2, M3	485	640	.007	C-7	950	.007	CC-7
			.150	125	.015	M2, M3	410	500	.020	C-6	625	.015	CC-6
			.300	100	.020	M2, M3	320	390	.030	C-6	500	.020	CC-6
			.625	80	.030	M2, M3	245	305	.040	C-6	—	—	—
			1	**46**	**.18**	**S4, S5**	**150**	**195**	**.18**	**P10**	**290**	**.18**	**CP10**
			4	**38**	**.40**	**S4, S5**	**125**	**150**	**.50**	**P20**	**190**	**.40**	**CP20**
			8	**30**	**.50**	**S4, S5**	**100**	**120**	**.75**	**P30**	**150**	**.50**	**CP30**
			16	**24**	**.75**	**S4, S5**	**75**	**95**	**1.0**	**P40**	—	—	—
	175 to 225	Hot Rolled, Normalized, Annealed or Cold Drawn	.040	145	.007	M2, M3	460	570	.007	C-7	850	.007	CC-7
			.150	115	.015	M2, M3	385	450	.020	C-6	550	.015	CC-6
			.300	95	.020	M2, M3	300	350	.030	C-6	450	.020	CC-6
			.625	75	.030	M2, M3	235	265	.040	C-6	—	—	—
			1	**44**	**.18**	**S4, S5**	**140**	**175**	**.18**	**P10**	**260**	**.18**	**CP10**
			4	**35**	**.40**	**S4, S5**	**115**	**135**	**.50**	**P20**	**170**	**.40**	**CP20**
			8	**29**	**.50**	**S4, S5**	**90**	**105**	**.75**	**P30**	**135**	**.50**	**CP30**
			16	**23**	**.75**	**S4, S5**	**72**	**81**	**1.0**	**P40**	—	—	—
	225 to 275	Annealed or Cold Drawn	.040	125	.007	M2, M3	410	510	.007	C-7	750	.007	CC-7
			.150	95	.015	M2, M3	360	400	.020	C-6	500	.015	CC-6
			.300	75	.020	M2, M3	285	315	.030	C-6	400	.020	CC-6
			.625	60	.030	M2, M3	220	240	.040	C-6	—	—	—
			1	**38**	**.18**	**S4, S5**	**125**	**155**	**.18**	**P10**	**230**	**.18**	**CP10**
			4	**29**	**.40**	**S4, S5**	**110**	**120**	**.50**	**P20**	**150**	**.40**	**CP20**
			8	**23**	**.50**	**S4, S5**	**87**	**95**	**.75**	**P30**	**120**	**.50**	**CP30**
			16	**18**	**.75**	**S4, S5**	**67**	**73**	**1.0**	**P40**	—	—	—

See section 15.1 for Tool Geometry.

*Caution: Check Horsepower requirements on heavier depths of cut.

See section 16 for Cutting Fluid Recommendations.

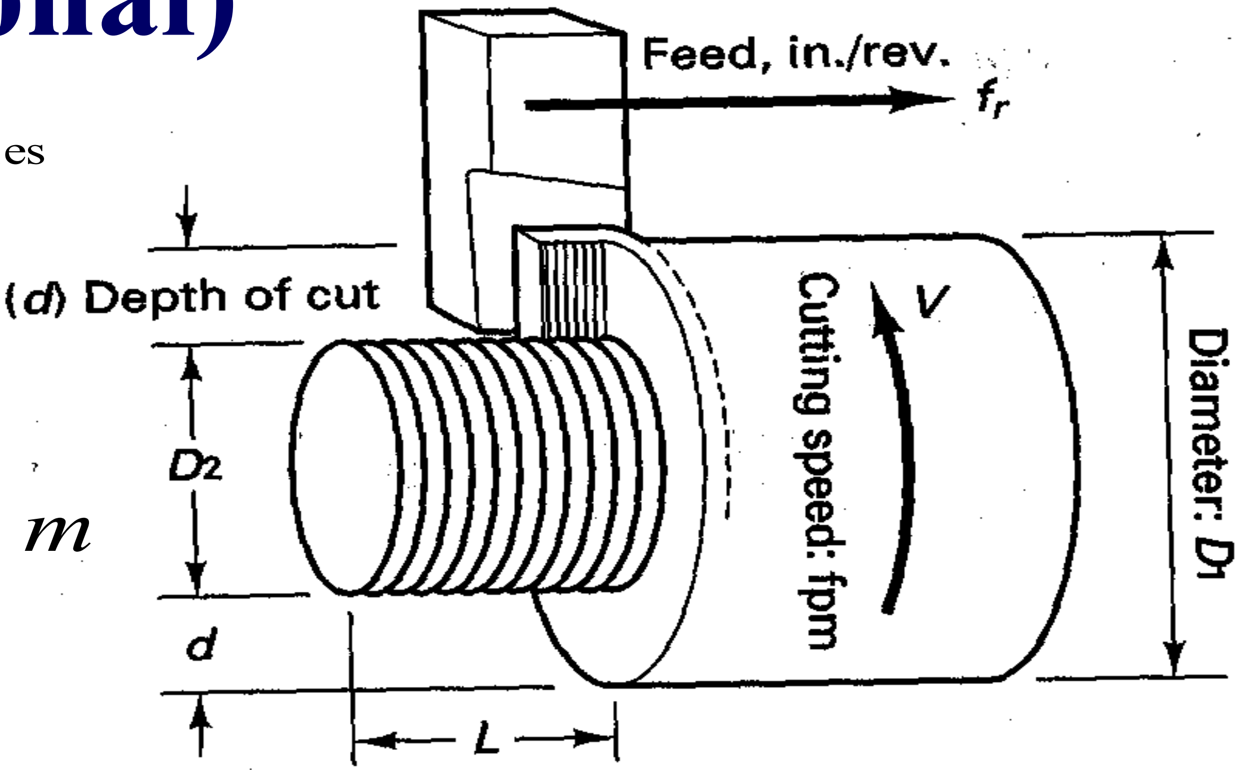
[†] Any premium HSS (T15, M33, M41–M47) or (S9, S10, S11, S12).

Turning (Rotational)

$$v = \frac{\pi D_1 RPM}{12}, f / \text{min}, \text{ D in inches}$$

$$DOC = \frac{D_1 - D_2}{2}, in$$

$$v = \pi D_1 RPM , m / \text{min}, D_1 in\ m$$



Speed, stated in surface feet per minute, (sfpm) is the peripheral speed at the cutting edge. To convert rpm into sfpm, use the following:

$V=(\pi * D1 * N_s)/12$ (converting D to ft.). This applies to milling, drilling turning and all rotary operations.

Feed per revolution in turning (and drilling) is a geared feed driven from the main spindle.

f_r: m/rev, f_t: m/tooth, f_m: m/min

Slab Milling

$$v = \frac{\pi D_1 RPM}{12}, f/\min,$$ D in inches

$$v = \pi D_1 RPM, m/\min, D_1 in\ m$$

DOC, d, ***Tool plunged distance***

Feed (f_r), ***material removed / tooth or***
material removed / revolution

$$f_m = f_t\, n\, N, \quad m/min$$

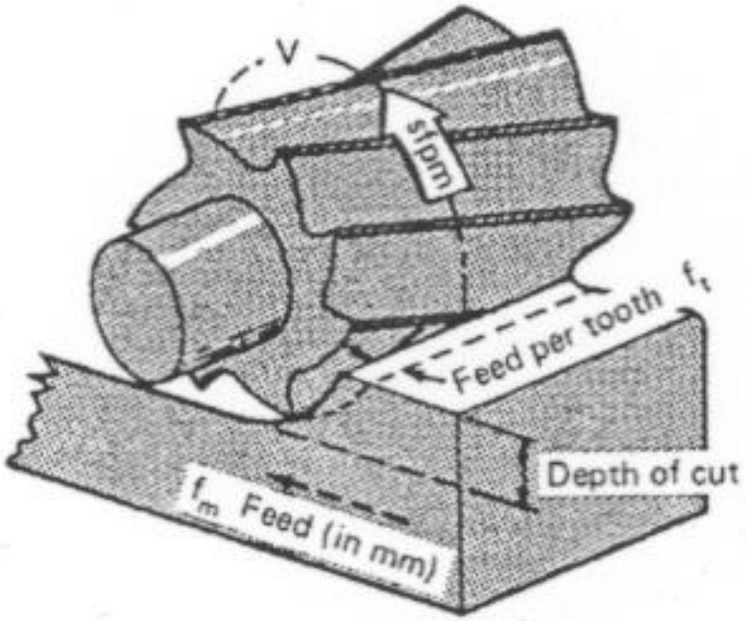
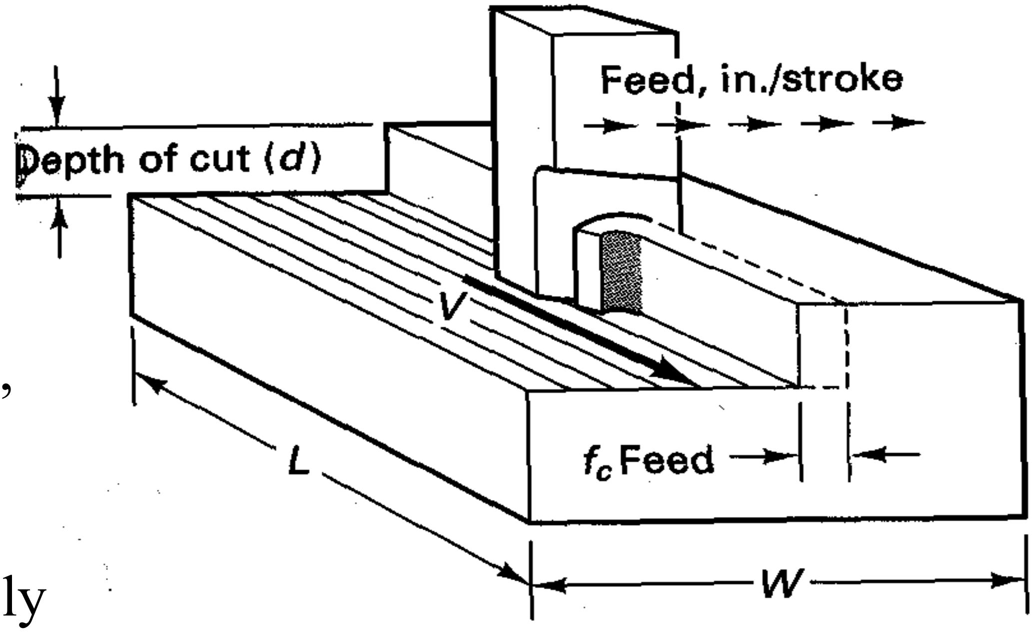


FIGURE Slab milling is an example of a multiple tooth-cutting process which creates a flat surface.

Planing (Linear)

$$d = D_{ini} - D_{fin}$$



Speed : relative motion between tool and work, stated in surface feet per minute (fpm).

Reciprocating motions may be crank-powered, hydraulically, or direct-reversible, electrically driven.

There will always be accelerations for a portion of the stroke. Cutting speed is assumed to be the average speed.

Feed with single-point tool is the amount the tool or work table is indexed.

Shop Eqns.

$$MRR = \frac{volume\_of\_cut}{CT} = CutArea \perp feed * feed / \min$$

For Turning

$$MRR = \pi D \cdot d \cdot f_r \cdot N \quad (\mathrm{m\,m}\,\frac{\mathrm{m}}{\mathrm{rev}}\,\frac{\mathrm{rev}}{\mathrm{min}})$$

$$= \mathrm{V f_r d}, \quad \mathrm{m^3 / min}$$

$$CT = \frac{L_{cut} + allowance}{f_r \times RPM}$$

$$= \frac{m}{\frac{m}{rev} \times \frac{rev}{\min}} = \min$$

Example

Material: **Carbon Steel**

Assume the Max. Allowed Cutting Speed, V : 305 m/min
Feed rate, F_r : 0.13 mm/rev
Depth of cut, d : 4.57 mm

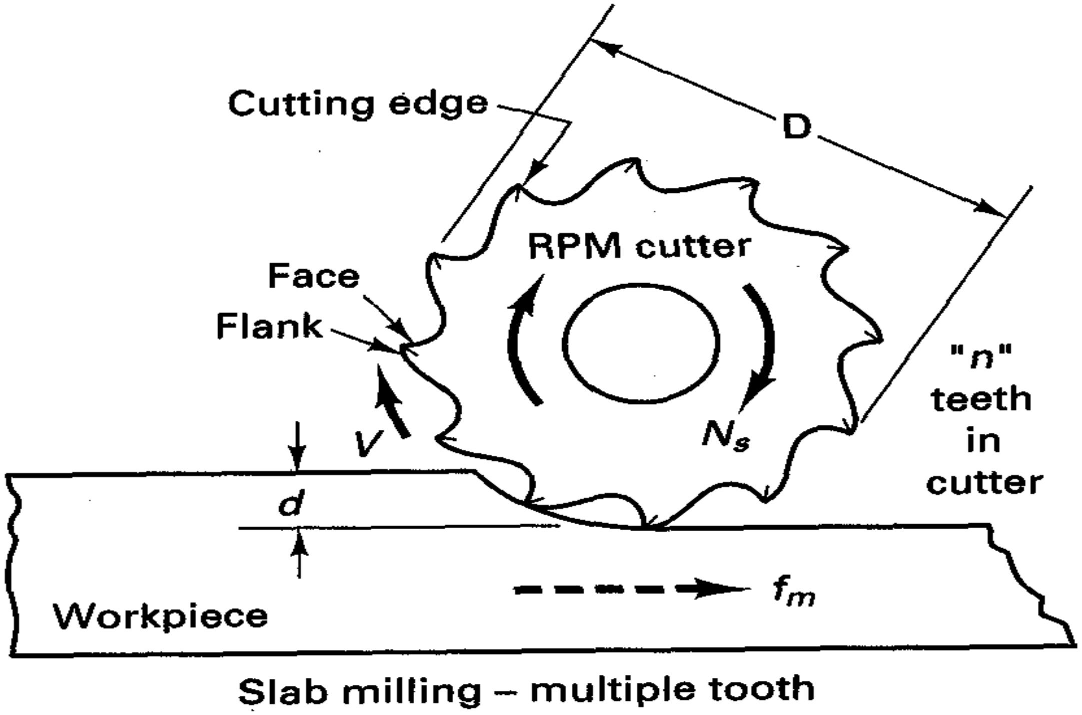
What is the max. MRR?

$$MRR = 305 * 0.00013 * 0.00457$$

$$= 0.0001812 \ \mathrm{m^3 / min}$$

$$= 181.2 \ \mathrm{cm^3 / min}$$

Milling (slab milling or end milling)



Slab milling – multiple tooth

The tool rotates at rpm Ns. The work-piece translates past the cutter at feed rate f_m, the table feed.

The length of cut, L, is the length of work-piece plus allowance, La,

Basics of the milling process (slab milling) as usually performed in a horizontal milling machine.

$$L_A = \sqrt{\frac{D^2}{4} - \left(\frac{D}{2} - d\right)^2} = \sqrt{d(D-d)} \ldots inches$$

$$v = \pi D_1 N, m/\min, D_1 in\ m$$

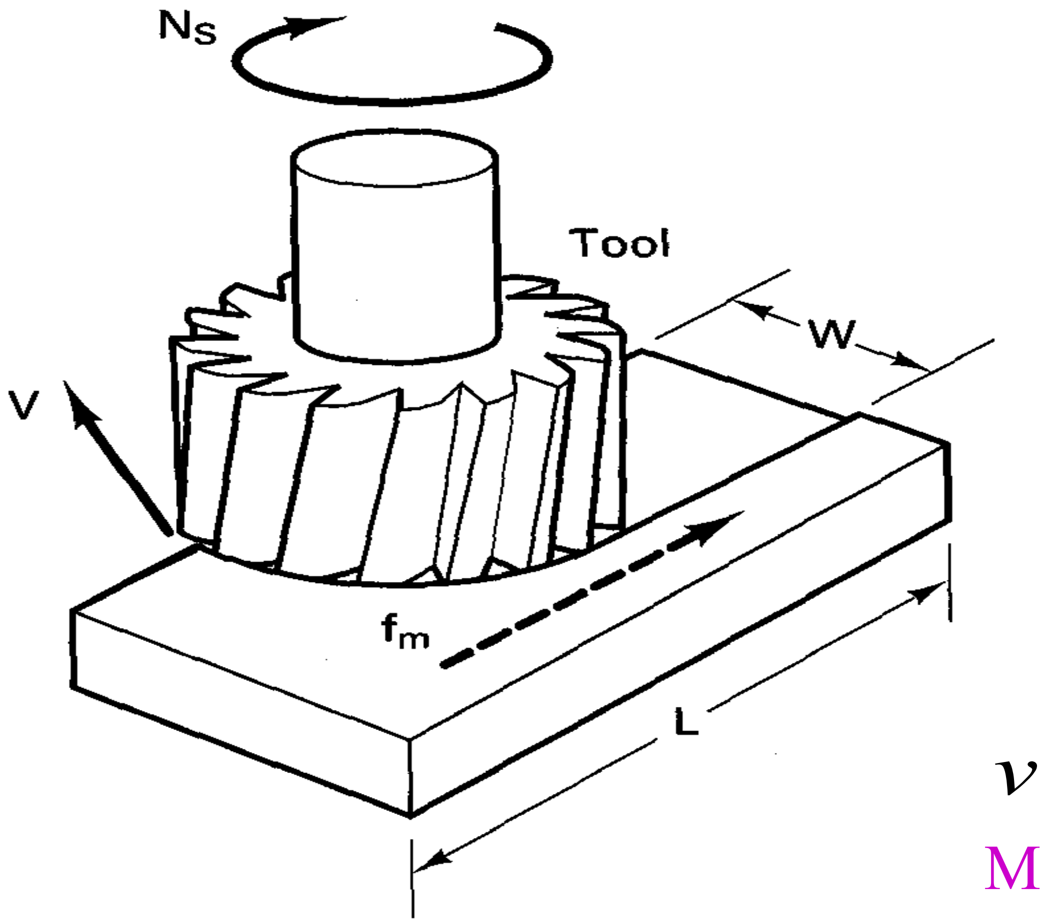
$CT = (L + L_A)/f_m$

$MRR = Wdf_m$, m^3/min where

W = width of the cut, d=depth of cut.

$f_m = f_t\, n\, Ns$, m/min

Face Milling



Face milling
Multiple tooth cutting

$$f_m = f_t \times n \times RPM$$

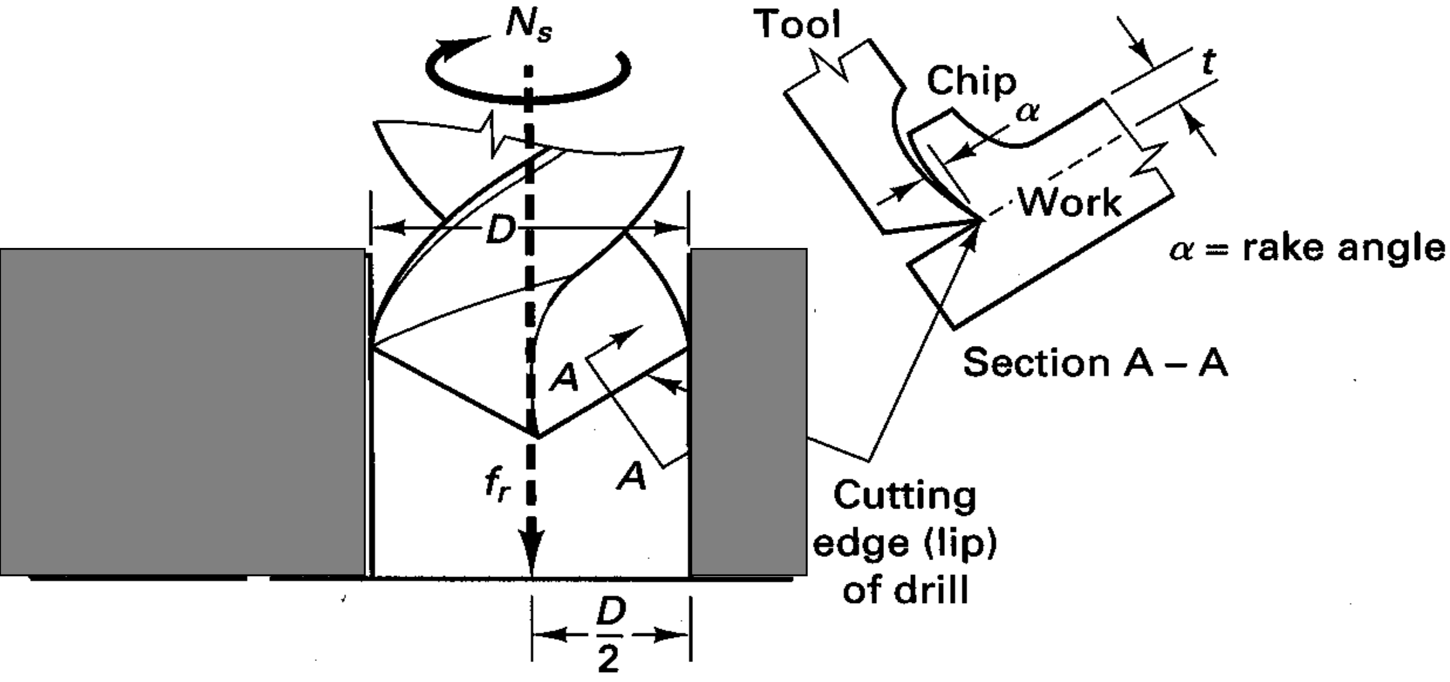
Given a selected cutting speed V and a feed per tooth f_t, the rpm of the cutter is $N_s = 12V/\pi D$ for a cutting of diameter D. The table feed rate is $f_m = f_t n N_s$ for a cutter with n teeth
The cutting time, $CT = (L + L_A + L_O)/f_m$
where $L_O = L_A = \sqrt{W(D - W)}$ for $W < D/2$
or $L_O = L_A = D/2$ for $W \geqslant D/2$.
The MRR = Wdf_m where d = depth of cut.

$$v = \pi D_1 N, m/\min, D_1 in\ m$$

MRR = Wdf_m, m^3/min

Basics of the milling process (face and end milling) as performed on a vertical spindle machine, including equations for cutting time and metal-removal rate.

Drilling



In drill, D = diameter of the drill which rotates 2 cutting edges at rpm N_s. V = velocity of outer edge of the lip of the drill.

$N_s = 12V/\pi D$

$CT = L + A/f_r N_s$ where f_r is the feed rate in in. per rev. The allowance $A = D/2$.

The MRR = $(\pi D^2/4) f_r N_s$ in.^{3} /min which is approximately $3DVf_r$.

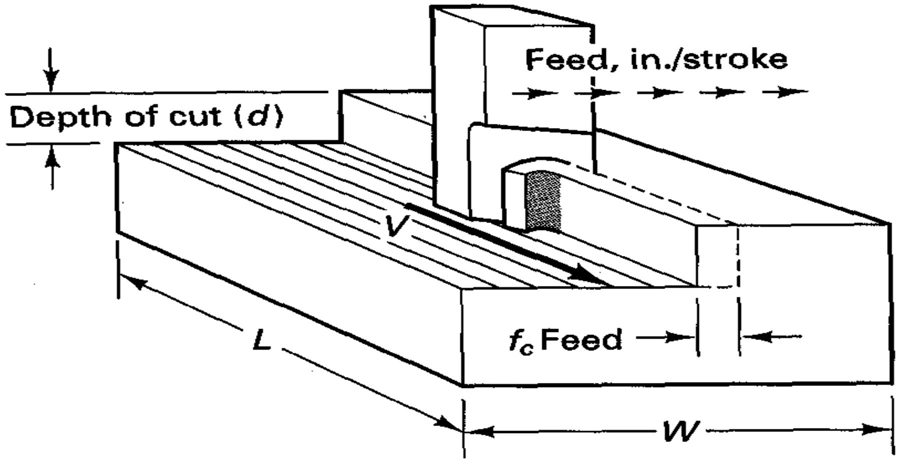
Drilling-multiple edge tool

Basic of the drilling (hole making) process, including equations for cutting time (CT) and metal-removal rate (MRR).

$$v = \pi DN, m/\min, D in\ m$$

$$MRR = (\pi D^2/4) f_r N,\ \ m^3/\min$$

Shaping

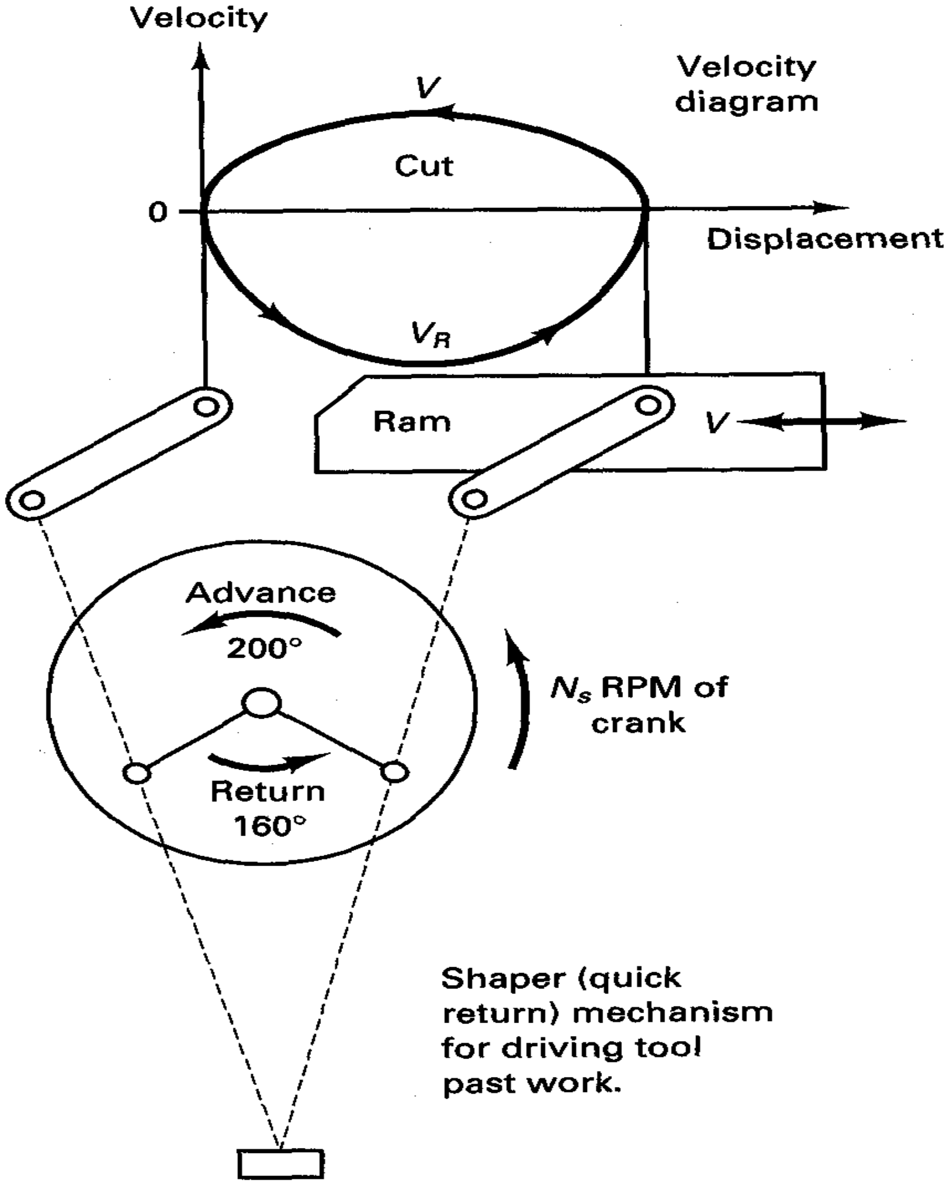


(a)

The tool cuts at velocity V with a return velocity of V_R dictated by the rpm of the crank, N_s. The cutting speed $V = (l + A)N_s/12R_s$ where R_s = stroke ratio = 200°/360° and the length of cut is $l = L$ + ALLOW. The tool feed is f_c inches per stroke

$CT = W/N_s f_c$

MRR = $LdN_s f_c$ in^3/min

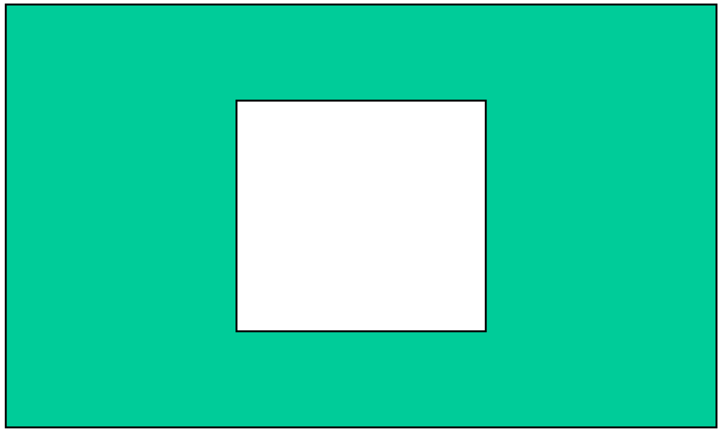


Shaper (quick return) mechanism for driving tool past work.

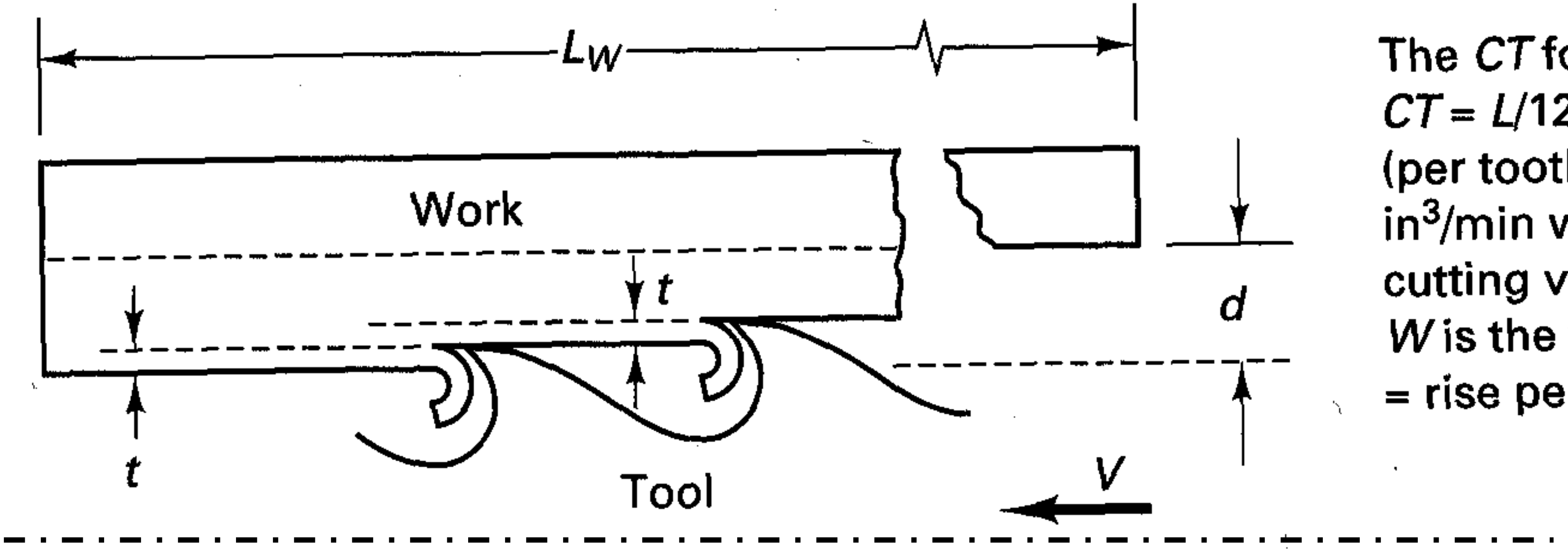
(b)

FIGURE Basics of the shaping process, including equations for cutting time (CT) and metal-removal rate (MRR).

How to make a square hole?



Broaching



The CT for broaching is $CT = L/12V$. The MRR (per tooth) is $12tWV$ in^3/min where V = cutting velocity in fpm, W is the width of cut, t = rise per tooth.

FIGURE . Process basics of broaching. Equations for cutting time and metal-removal rate, developed in Chapter 26.

Cutting Speed, Feed rate, DOC are fixed

OBLIQUE SYSTEM

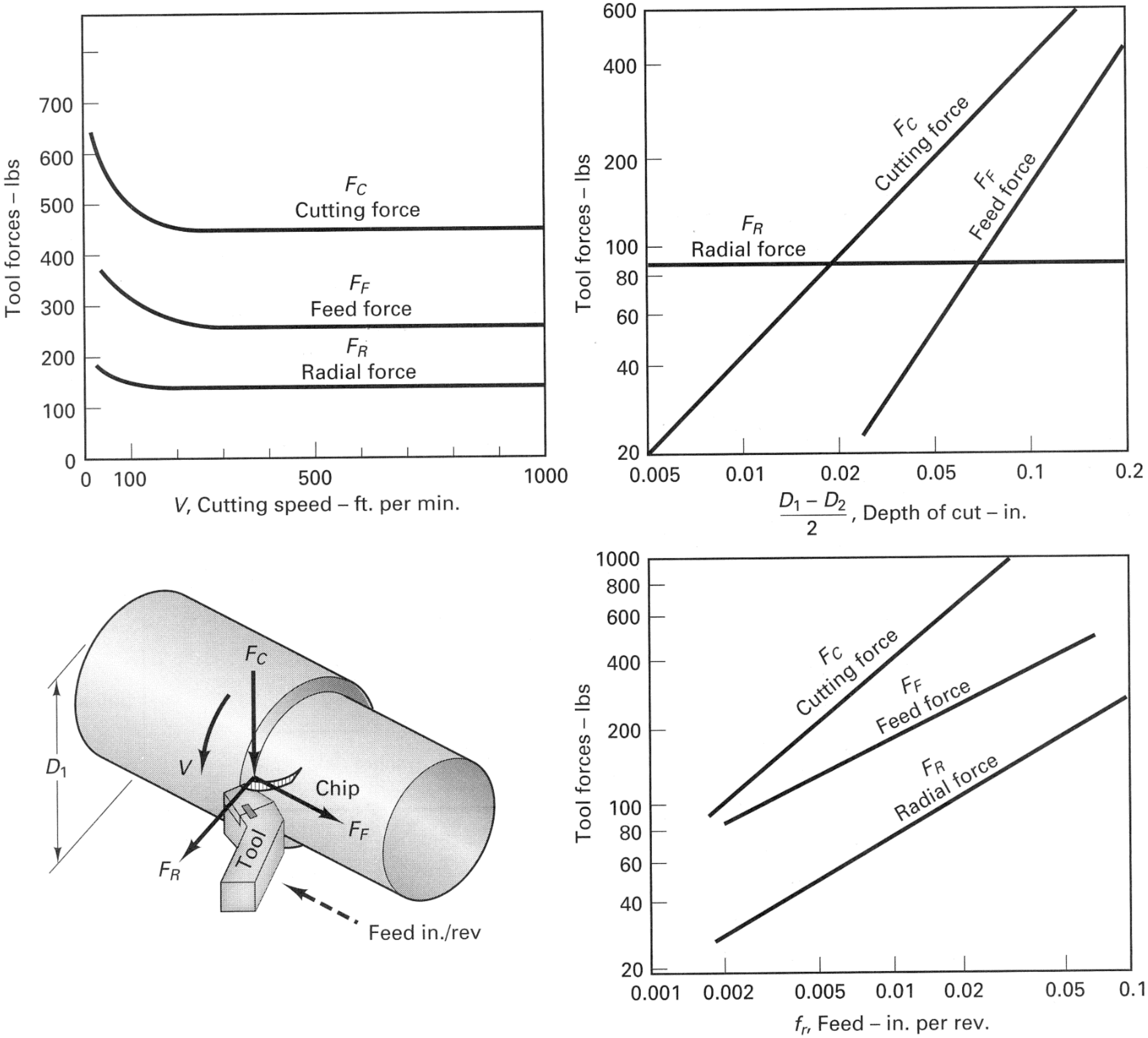


FIGURE 21-11 Oblique machining has three measurable components of forces acting on the tool. The forces vary with speed, depth of cut, and feed.

3 Force

F_C = Cutting force (vertical)

F_R = Radial force (thrust)

F_F = Feed force

Energy & Power in Machining

- Fc: Primary cutting force (largest and accounts for 99% of the required power)
- Ff: Feed force (approximately 50 % of Fc but small since feed rates are smaller than cutting speeds)
- Fr: Radial or thrust force (approximately 50 % of Ff, and contributes very little)

Power requirement

$$P = F_c V \,(\text{ft-lb/min})$$

$$\text{hp} = \frac{F_c V}{33{,}000}$$

$$\text{HP}_\text{s} = \frac{\text{hp}}{\text{MRR}} \,(\text{hp/in}^3\text{/min})$$

$$\text{HP}_\text{m} = \frac{\text{HP}_\text{s} \times \text{MRR} \times CF}{E}$$

CF: Tool wear Correction factor, 1.25

E: Mechanical Efficiency, 0.80

Power requirement – 2

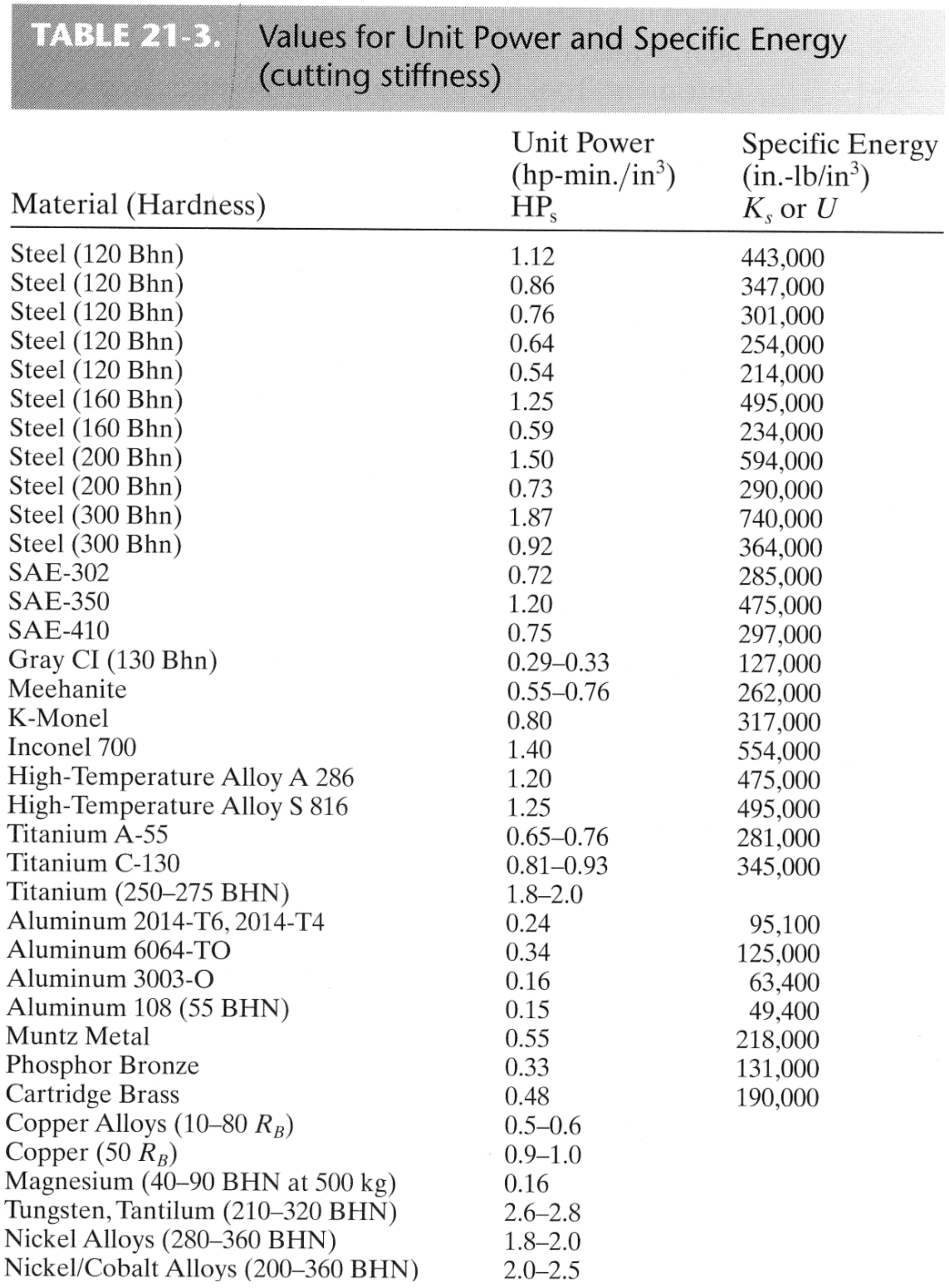


TABLE 21-3. Values for Unit Power and Specific Energy (cutting stiffness)

Material (Hardness)	Unit Power (hp-min./in^3) HP$_s$	Specific Energy (in.-lb/in^3) K_s or U
Steel (120 Bhn)	1.12	443,000
Steel (120 Bhn)	0.86	347,000
Steel (120 Bhn)	0.76	301,000
Steel (120 Bhn)	0.64	254,000
Steel (120 Bhn)	0.54	214,000
Steel (160 Bhn)	1.25	495,000
Steel (160 Bhn)	0.59	234,000
Steel (200 Bhn)	1.50	594,000
Steel (200 Bhn)	0.73	290,000
Steel (300 Bhn)	1.87	740,000
Steel (300 Bhn)	0.92	364,000
SAE-302	0.72	285,000
SAE-350	1.20	475,000
SAE-410	0.75	297,000
Gray CI (130 Bhn)	0.29–0.33	127,000
Meehanite	0.55–0.76	262,000
K-Monel	0.80	317,000
Inconel 700	1.40	554,000
High-Temperature Alloy A 286	1.20	475,000
High-Temperature Alloy S 816	1.25	495,000
Titanium A-55	0.65–0.76	281,000
Titanium C-130	0.81–0.93	345,000
Titanium (250–275 BHN)	1.8–2.0	
Aluminum 2014-T6, 2014-T4	0.24	95,100
Aluminum 6064-TO	0.34	125,000
Aluminum 3003-O	0.16	63,400
Aluminum 108 (55 BHN)	0.15	49,400
Muntz Metal	0.55	218,000
Phosphor Bronze	0.33	131,000
Cartridge Brass	0.48	190,000
Copper Alloys (10–80 R_B)	0.5–0.6	
Copper (50 R_B)	0.9–1.0	
Magnesium (40–90 BHN at 500 kg)	0.16	
Tungsten, Tantilum (210–320 BHN)	2.6–2.8	
Nickel Alloys (280–360 BHN)	1.8–2.0	
Nickel/Cobalt Alloys (200–360 BHN)	2.0–2.5	

Motor Design

- Decide the cutting parameters (f_m, N, doc)
- Calculate MRR (MRR_{max})
- Choose the HPs of the material selected
- Estimate HPm

Shop Equations

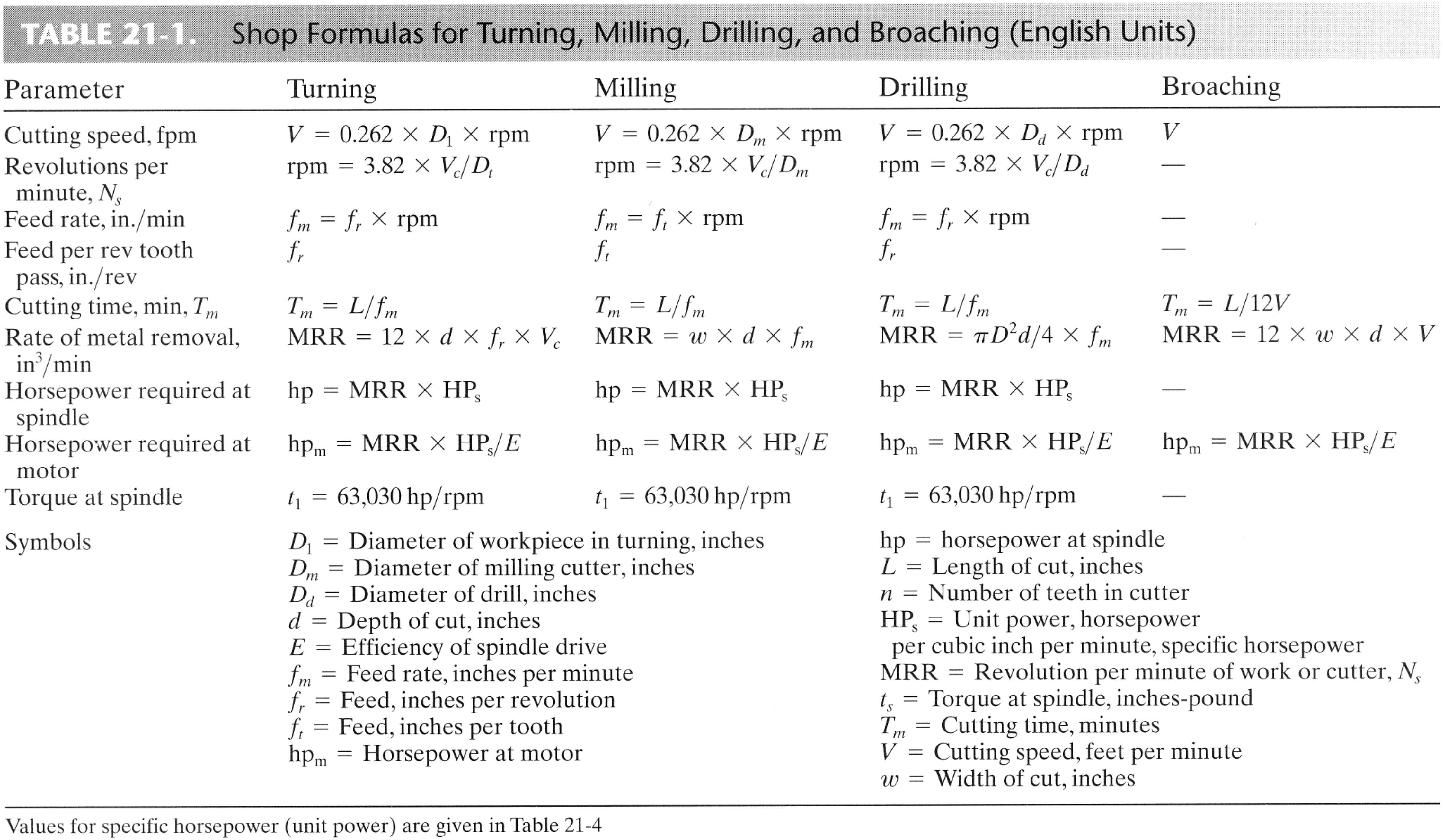


TABLE 21-1. Shop Formulas for Turning, Milling, Drilling, and Broaching (English Units)

Parameter	Turning	Milling	Drilling	Broaching
Cutting speed, fpm	$V = 0.262 \times D_1 \times \text{rpm}$	$V = 0.262 \times D_m \times \text{rpm}$	$V = 0.262 \times D_d \times \text{rpm}$	V
Revolutions per minute, N_s	$\text{rpm} = 3.82 \times V_c/D_t$	$\text{rpm} = 3.82 \times V_c/D_m$	$\text{rpm} = 3.82 \times V_c/D_d$	—
Feed rate, in./min	$f_m = f_r \times \text{rpm}$	$f_m = f_t \times \text{rpm}$	$f_m = f_r \times \text{rpm}$	—
Feed per rev tooth pass, in./rev	f_r	f_t	f_r	—
Cutting time, min, T_m	$T_m = L/f_m$	$T_m = L/f_m$	$T_m = L/f_m$	$T_m = L/12V$
Rate of metal removal, in^3/min	$\text{MRR} = 12 \times d \times f_r \times V_c$	$\text{MRR} = w \times d \times f_m$	$\text{MRR} = \pi D^2 d/4 \times f_m$	$\text{MRR} = 12 \times w \times d \times V$
Horsepower required at spindle	$\text{hp} = \text{MRR} \times \text{HP}_s$	$\text{hp} = \text{MRR} \times \text{HP}_s$	$\text{hp} = \text{MRR} \times \text{HP}_s$	—
Horsepower required at motor	$\text{hp}_m = \text{MRR} \times \text{HP}_s/E$	$\text{hp}_m = \text{MRR} \times \text{HP}_s/E$	$\text{hp}_m = \text{MRR} \times \text{HP}_s/E$	$\text{hp}_m = \text{MRR} \times \text{HP}_s/E$
Torque at spindle	$t_1 = 63{,}030\ \text{hp/rpm}$	$t_1 = 63{,}030\ \text{hp/rpm}$	$t_1 = 63{,}030\ \text{hp/rpm}$	—

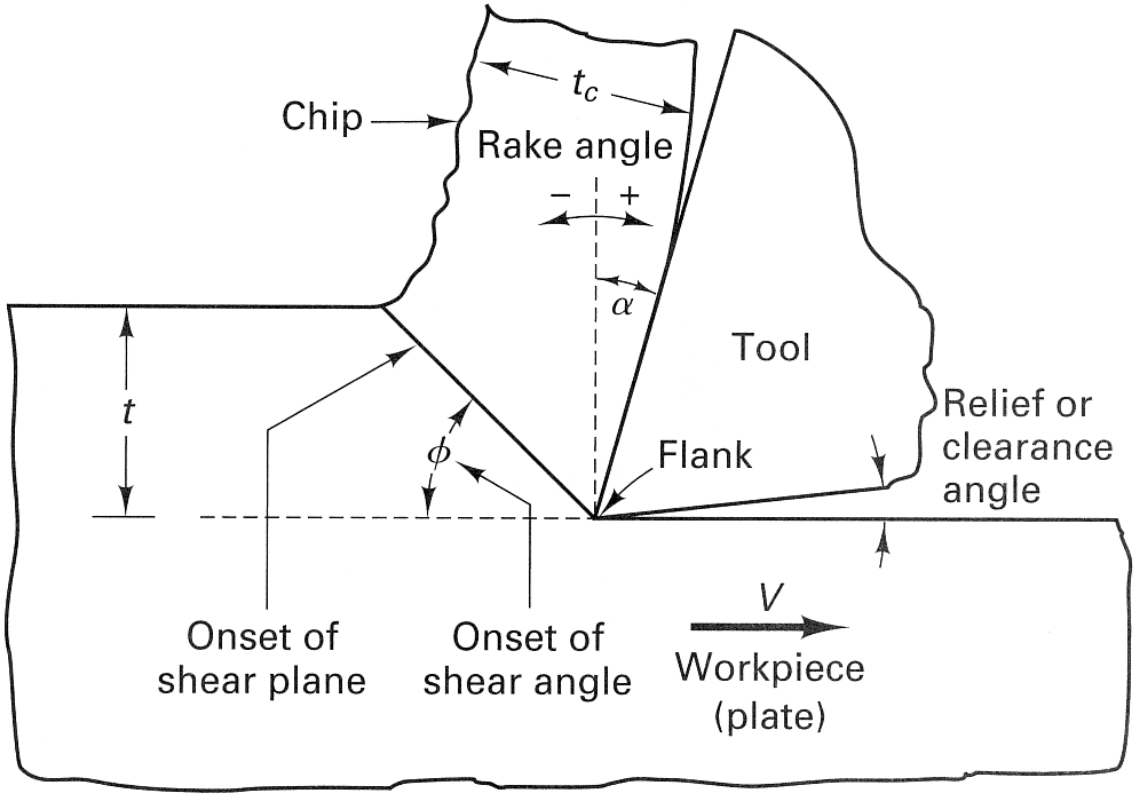
Symbols

D_1 = Diameter of workpiece in turning, inches
D_m = Diameter of milling cutter, inches
D_d = Diameter of drill, inches
d = Depth of cut, inches
E = Efficiency of spindle drive
f_m = Feed rate, inches per minute
f_r = Feed, inches per revolution
f_t = Feed, inches per tooth
hp_m = Horsepower at motor
hp = horsepower at spindle
L = Length of cut, inches
n = Number of teeth in cutter
HP_s = Unit power, horsepower per cubic inch per minute, specific horsepower
MRR = Revolution per minute of work or cutter, N_s
t_s = Torque at spindle, inches-pound
T_m = Cutting time, minutes
V = Cutting speed, feet per minute
w = Width of cut, inches

Values for specific horsepower (unit power) are given in Table 21-4

Orthogonal Machining (Two-Force Model)

- Cutting tool geometry is simplified from the 3-D (oblique) geometry to a 2-D (orthogonal) geometry



(c) Orthogonal plate machining with fixed tool, moving plate.

Chip Formation: Simplified Orthogonal Machining in-plane model of the cutting process

Assumptions

Work-piece: flat plate
Cutting tool: geometry given by the back rake angle (α) only
Shear: in one single plane which is doing ϕ degrees with the direction of speed.

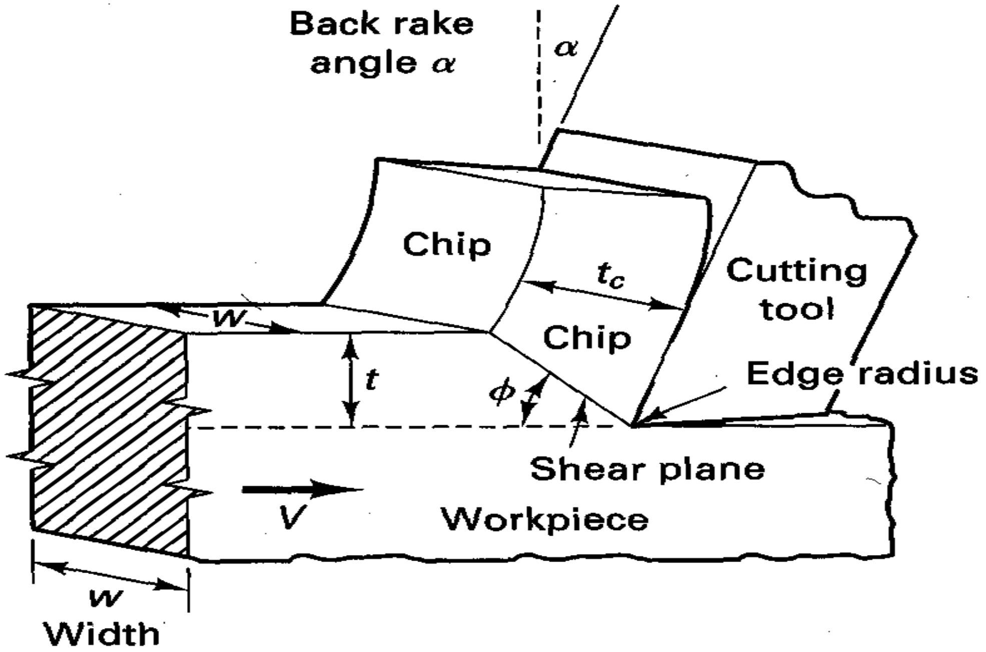


FIGURE Schematic of orthogonal machining. The cutting edge of the tool is perpendicular to the direction of motion (V). The back rake angle is α. The shear angle is ϕ.

Chip is formed by shearing. High shear stress over a narrow region.

Large strain and high strain rate plastic deformation due to radial compression.

Compression zone travels ahead of the tool.

Elastic to plastic deformation when yield stress is exceeded.

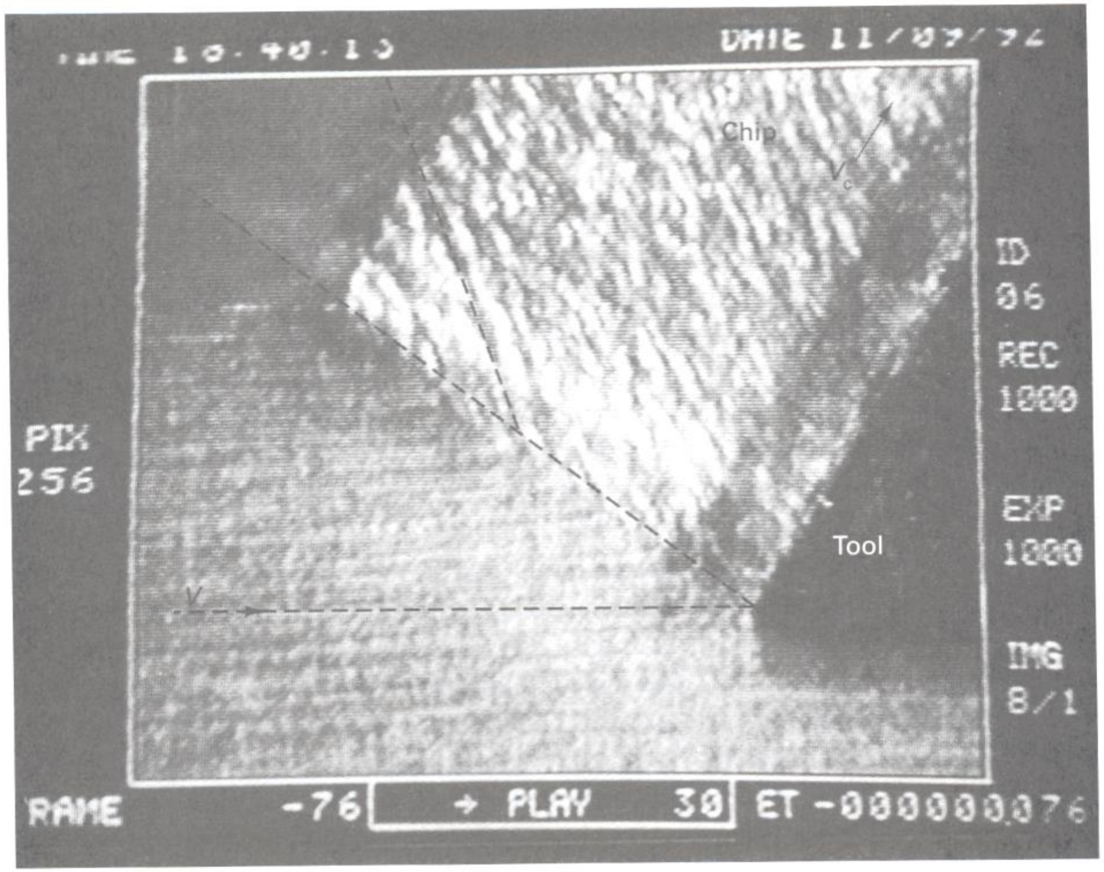


FIGURE Videograph of the orthogonal machining process.

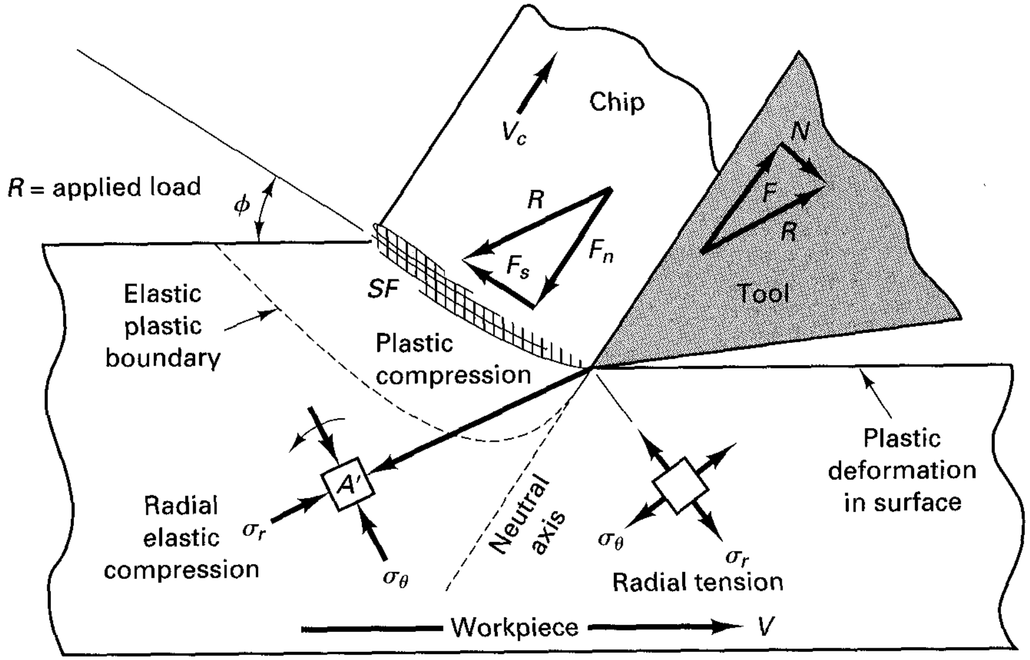


FIGURE The machining process produces a radial compression ahead of the shear process. The stress reverses from compression to tension across the neutral axis (NA).

Mechanics of machining:

Velocity diagram

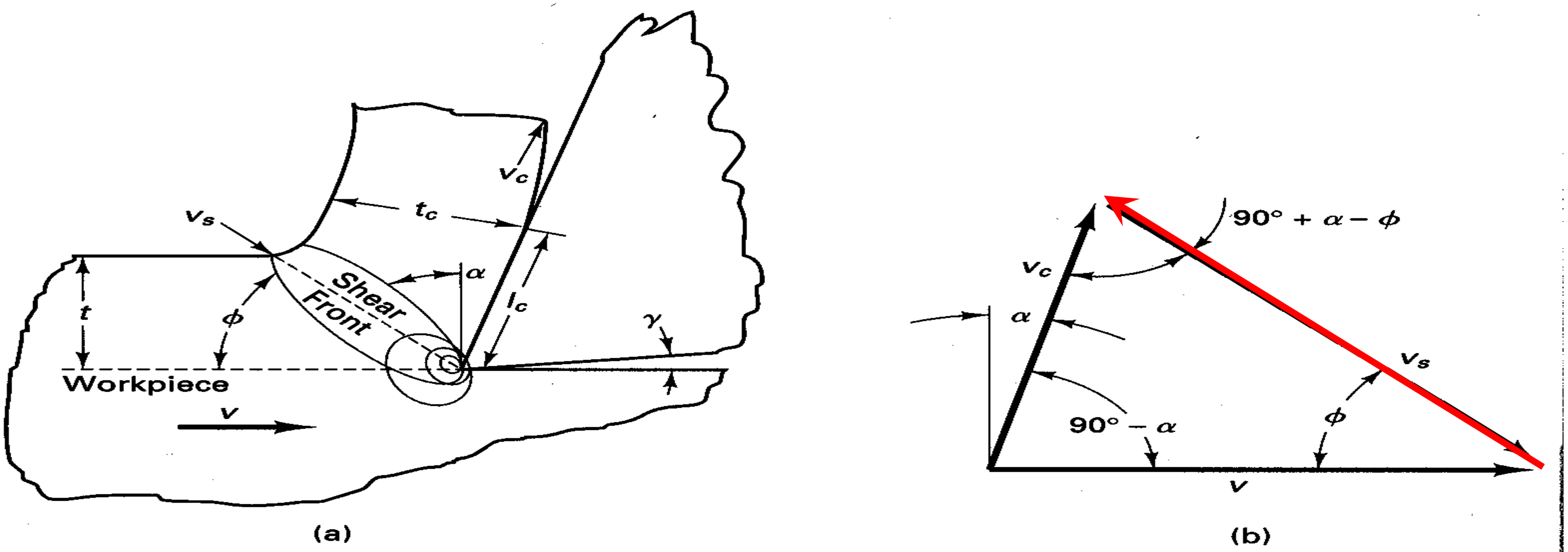


FIGURE (a) Schematic of orthogonal machining process; (b) velocity diagram associated with orthogonal machining.

Chip thickness ratio, $r_c = \dfrac{t}{t_c} = \dfrac{AB\sin\phi}{AB\cos(\phi-\alpha)} = \dfrac{\sin\phi}{\sin(90-(\phi-\alpha))}$ = Vc/V

Shear angle, $\tan\phi = \dfrac{r_c\cos\alpha}{1-r_c\sin\alpha}$

During the cutting, the chip undergoes a shear strain $\varepsilon = \dfrac{2\cos\alpha}{1+\sin\alpha}$

Where, AB is the length of the shear plane.

Chip Compression ratio = 1/rc

Quick stop device : Sudden disengagement of the tool

Types of chips: discontinuous, continuous with built-up edge

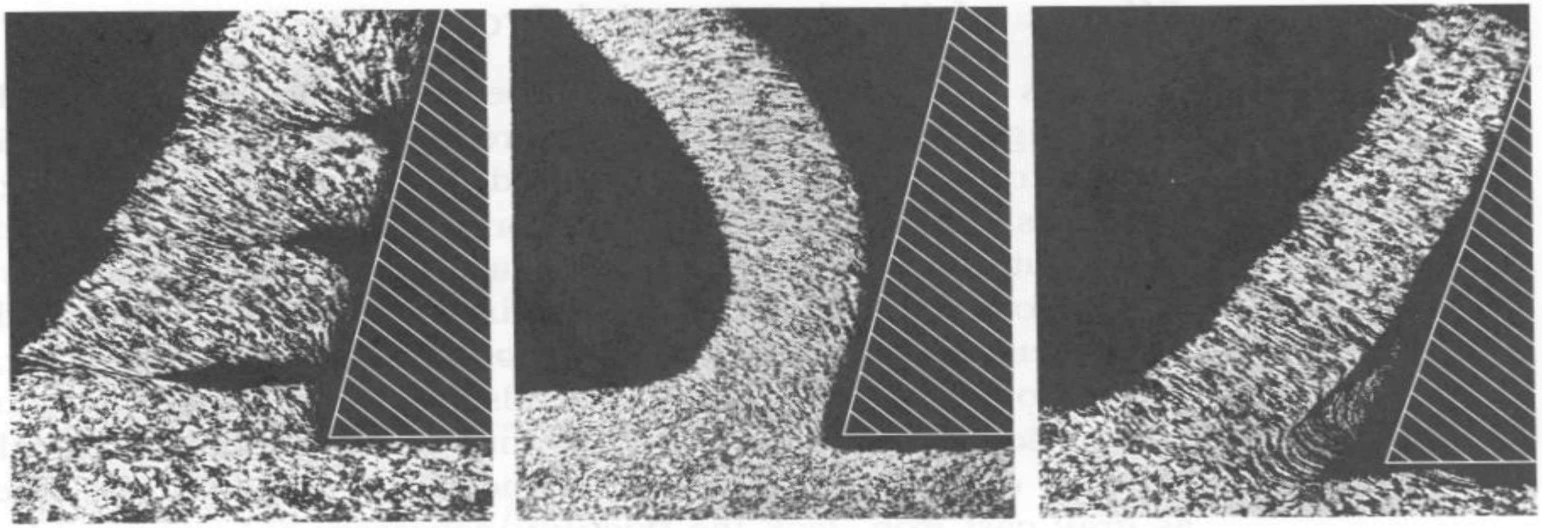


FIGURE Three characteristic types of chips. (Left to right) discontinuous, continuous, and continuous with built-up edge. Chip samples produced by quick-stop techniques. (*Courtesy of Cincinnati Milacron, Inc.*)

Mechanics of machining:

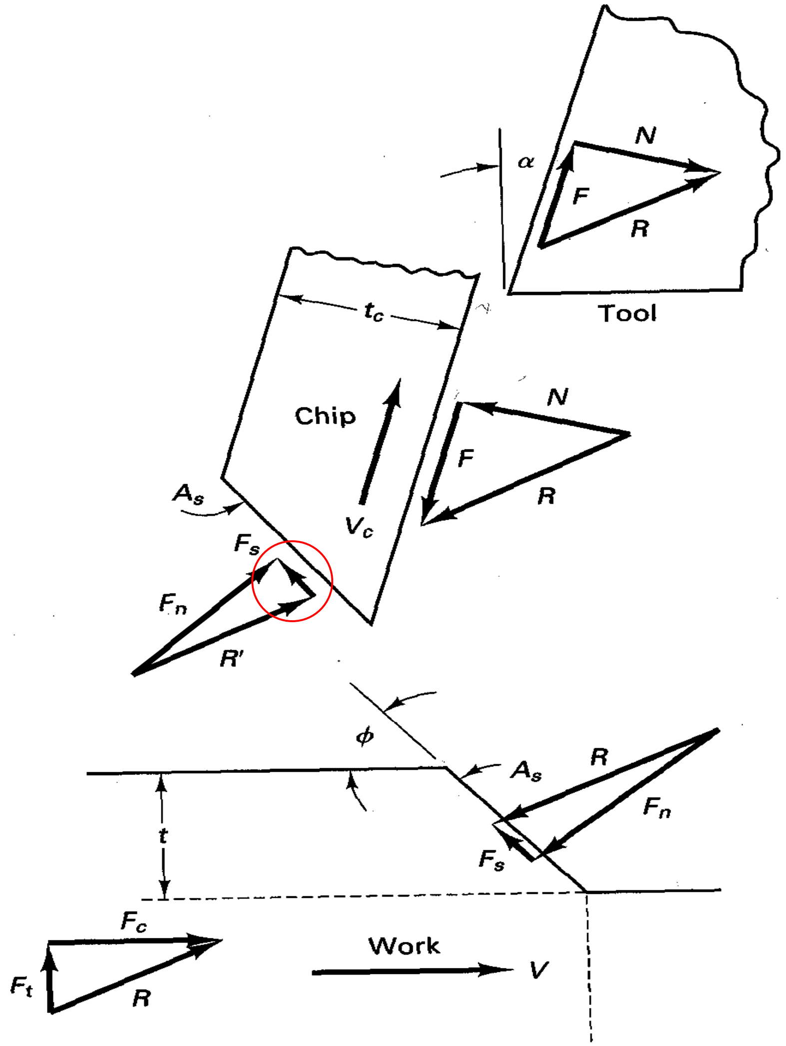


FIGURE Free-body diagram of orthogonal chip formation process, showing equilibrium condition between resultant forces *R* and *R′*.

R resultant force: F_c cutting force

F_t normal (tangential) force:

F_s shear force (N) F_n -normal force

Friction angle β: $$\beta = \tan^{-1}\frac{F}{N}$$

Circular Force Diagram

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

$$N = F_c \cos\alpha - F_t \sin\alpha$$

$$F_s = F_c \cos\Phi - F_t \sin\phi$$

$$F_n = F \sin\Phi - F_t \cos\phi$$

$$R = \sqrt{F_c^{\,2} + F_t^{\,2}} = \sqrt{F^2 + N^2}$$

$$\tau_s = \frac{F_s}{A_s} = \frac{F_c \sin\phi\cos\phi - F_t \sin^2\phi}{t \times w}$$

$$A_s = \frac{t \times w}{\sin\phi}$$

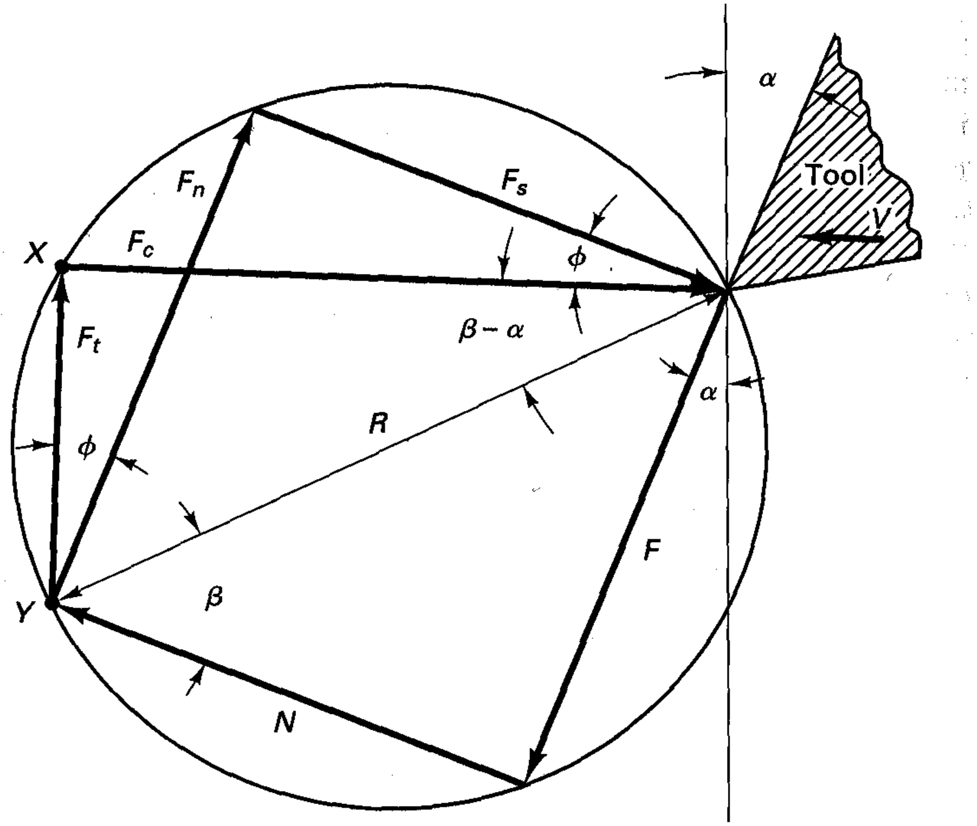
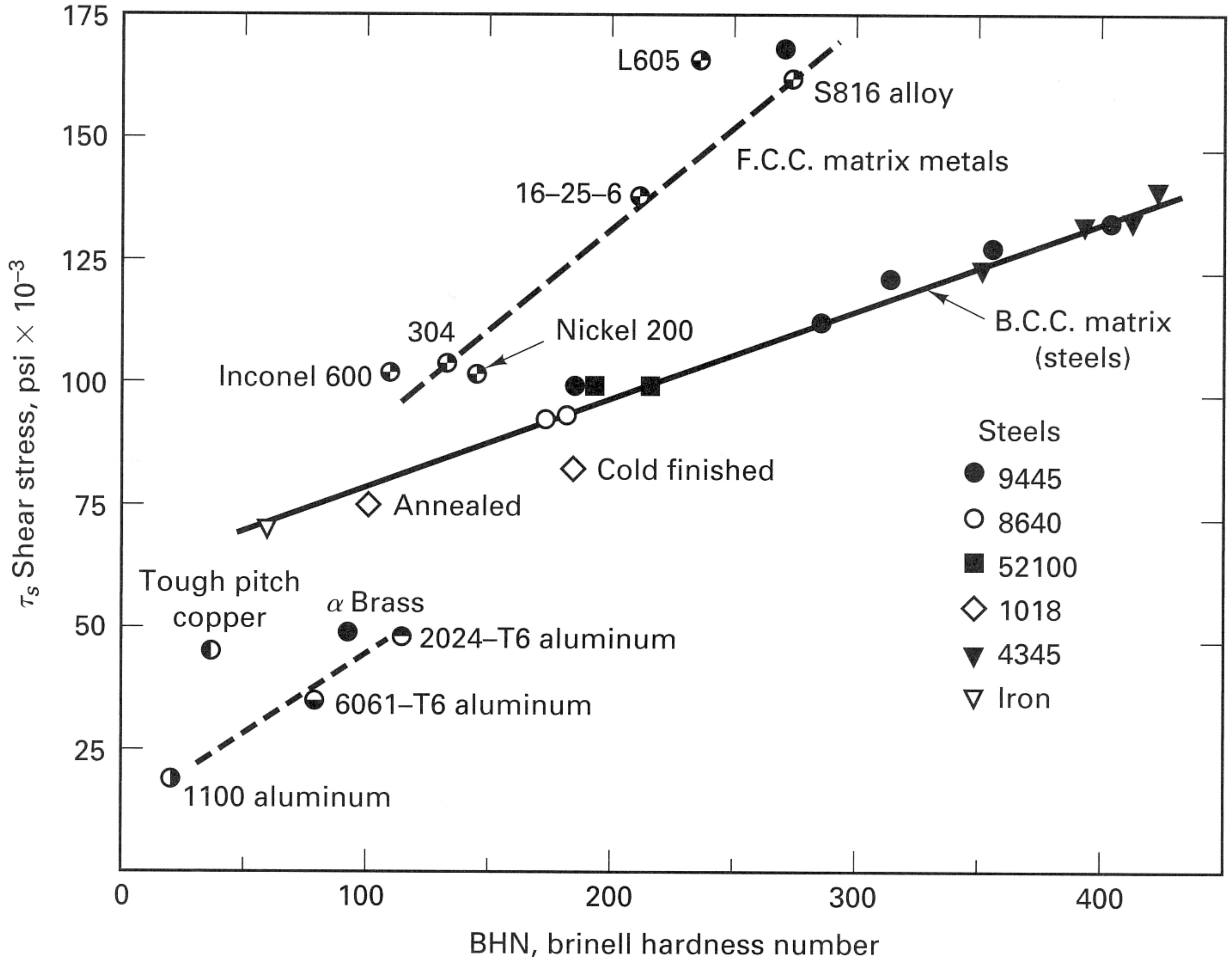


FIGURE Circular force diagram used to derive equations
A for F_s, F_n, F, and N as functions of F_c, F_t, ϕ, α, and β.

Typical shear stress values



- **Necessary power of the machine** $$Power = F_c v / 60, \quad Watts$$

- **Calculation of the specific power** $$Specific \quad Power = \frac{Power}{MRR} = \frac{function(V, F_c)}{function(V, f_r, d)}$$

- **The primary cutting force** $$F_c \approx \frac{Spec.Power \times MRR \times 60}{v}$$

The motor Power = Spec. Power*MRR*CF/η

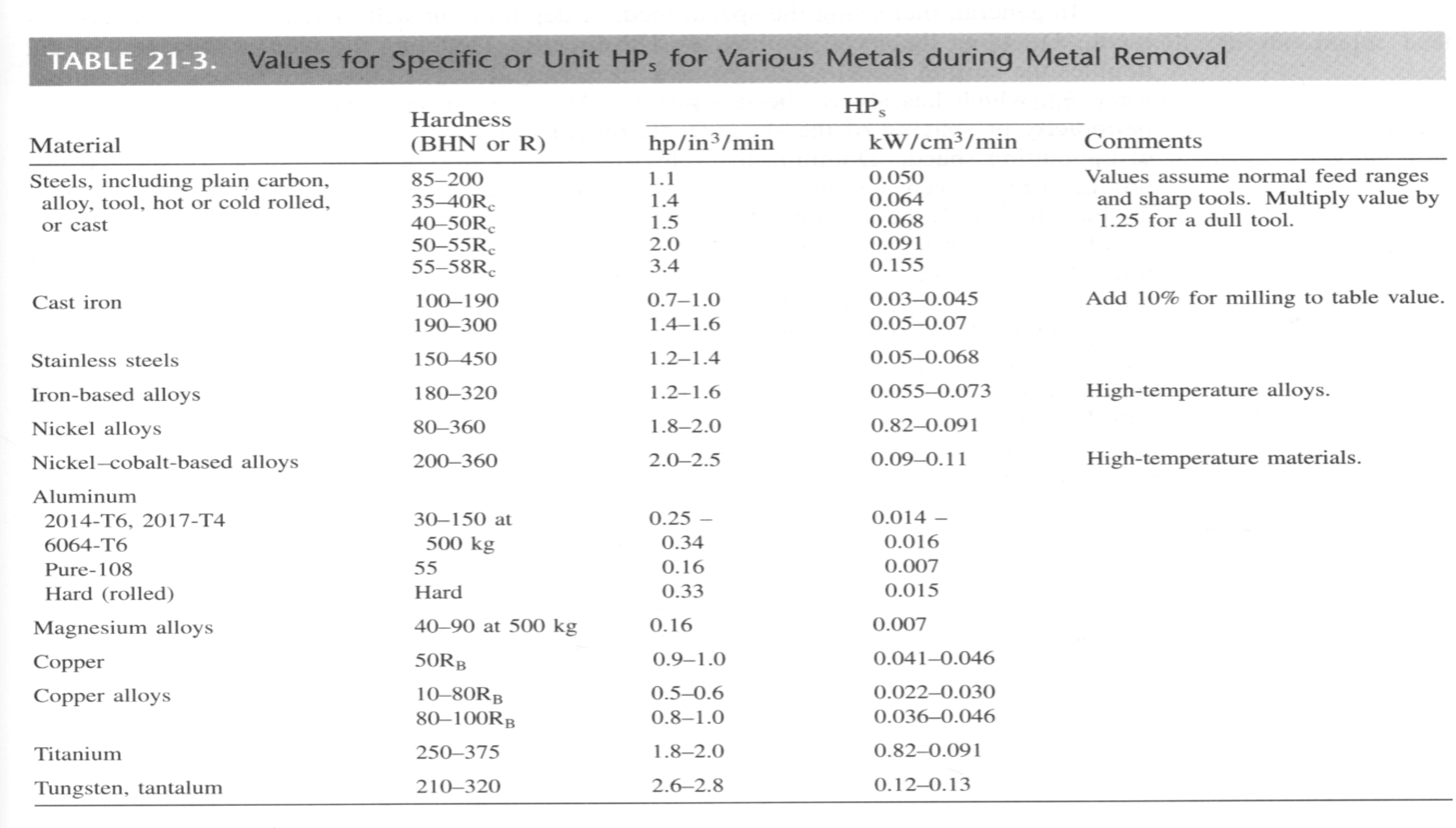


TABLE 21-3. Values for Specific or Unit HP_s for Various Metals during Metal Removal

Material	Hardness (BHN or R)	HP_s		Comments
		hp/in³/min	kW/cm³/min	
Steels, including plain carbon, alloy, tool, hot or cold rolled, or cast	85–200	1.1	0.050	Values assume normal feed ranges and sharp tools. Multiply value by 1.25 for a dull tool.
	35–40R_c	1.4	0.064	
	40–50R_c	1.5	0.068	
	50–55R_c	2.0	0.091	
	55–58R_c	3.4	0.155	
Cast iron	100–190	0.7–1.0	0.03–0.045	Add 10% for milling to table value.
	190–300	1.4–1.6	0.05–0.07	
Stainless steels	150–450	1.2–1.4	0.05–0.068	
Iron-based alloys	180–320	1.2–1.6	0.055–0.073	High-temperature alloys.
Nickel alloys	80–360	1.8–2.0	0.82–0.091	
Nickel–cobalt-based alloys	200–360	2.0–2.5	0.09–0.11	High-temperature materials.
Aluminum				
2014-T6, 2017-T4	30–150 at	0.25 –	0.014 –	
6064-T6	500 kg	0.34	0.016	
Pure-108	55	0.16	0.007	
Hard (rolled)	Hard	0.33	0.015	
Magnesium alloys	40–90 at 500 kg	0.16	0.007	
Copper	50R_B	0.9–1.0	0.041–0.046	
Copper alloys	10–80R_B	0.5–0.6	0.022–0.030	
	80–100R_B	0.8–1.0	0.036–0.046	
Titanium	250–375	1.8–2.0	0.82–0.091	
Tungsten, tantalum	210–320	2.6–2.8	0.12–0.13	

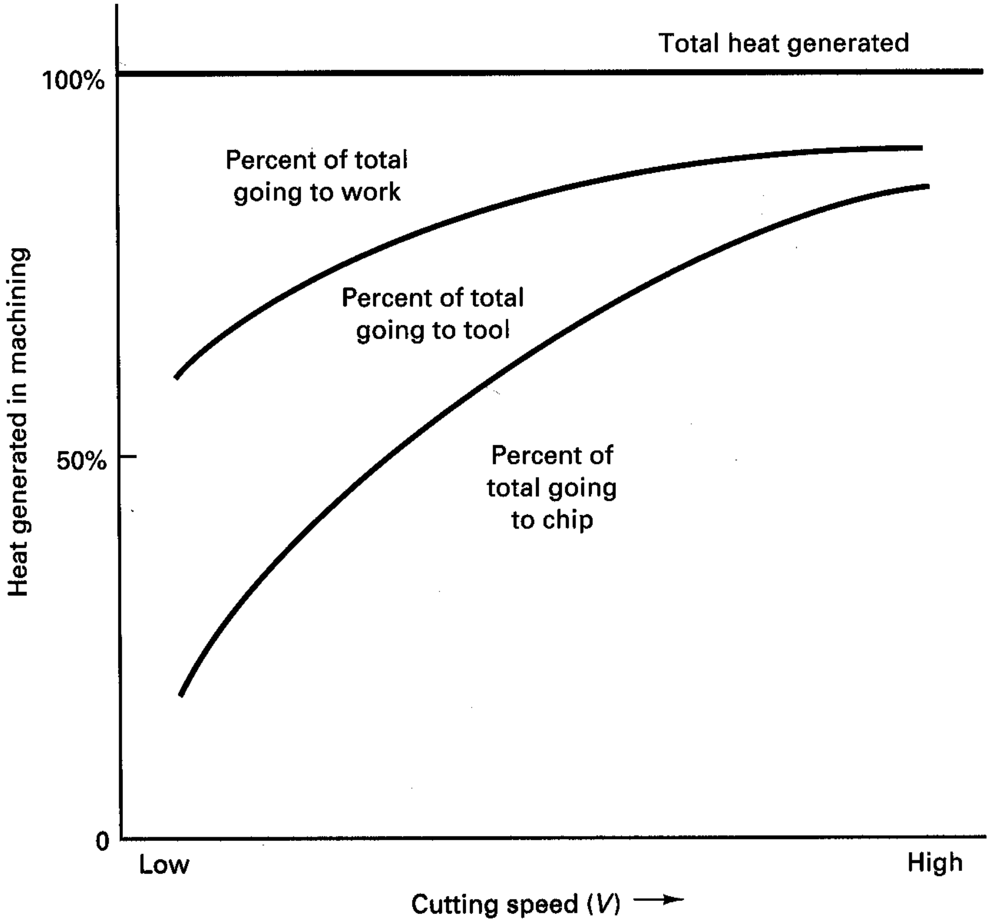
Heat and temperature in metal cutting

Sources: Shear process

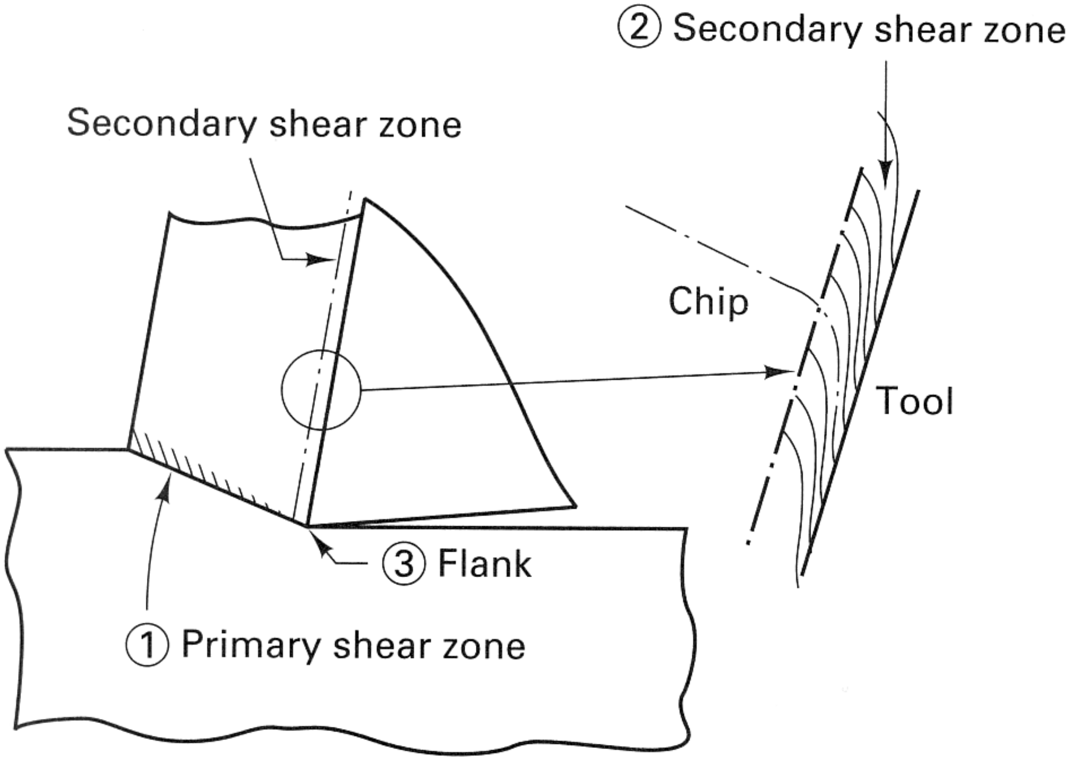
Tool-chip interface (friction)

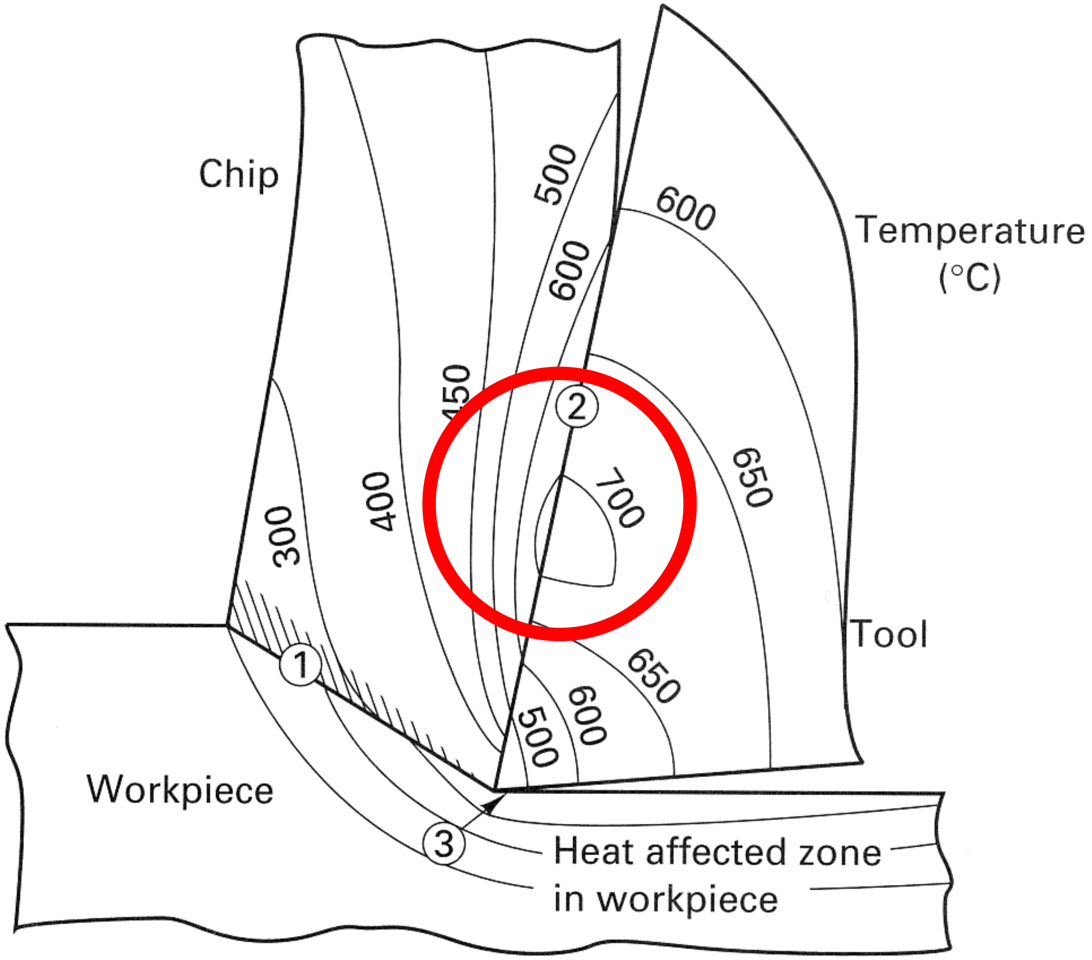
Tool and work piece interface (friction).

Sinks:



Heat and Temperature in Metal Cutting





Chatter: Unwanted vibration of the tool.

Can be reduced by cutting parameters and Active vibration Control.

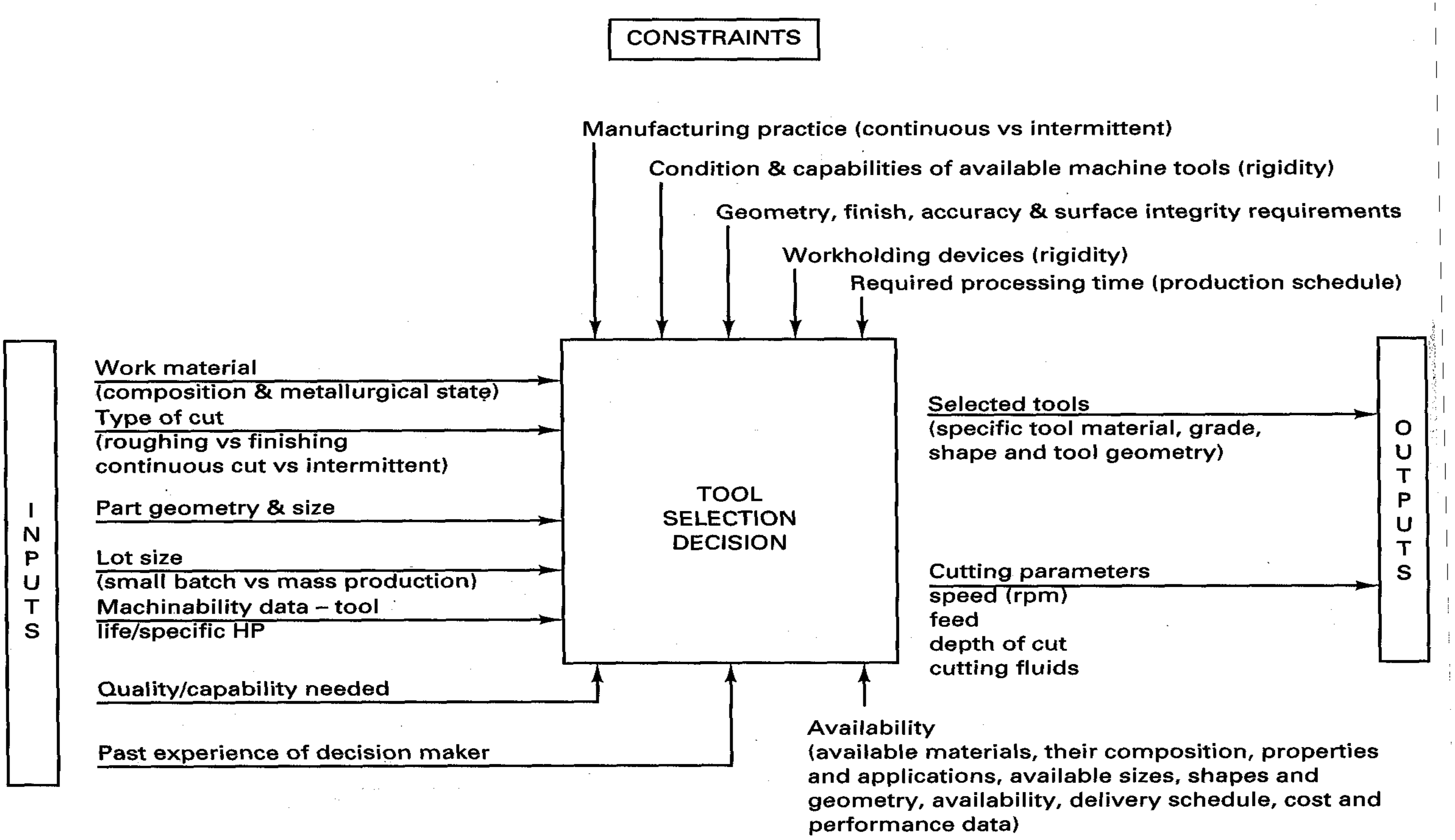
Cutting Tools for Machining

Introduction

- Workpiece material → Selection of the proper cutting tool (material&geometry)
  - High carbon steels
  - Low/medium alloy steels
  - High-speed steels
  - Cast cobalt alloys
  - Cemented/cast/coated carbides
  - Coated high speed steels
  - Ceramics
  - Sintered polycrystalline cubic boron nitride (CBN)
  - Sintered polycrystalline diamond
  - Single-crystal natural diamond

Properties
Performance
Cost

Selection of a cutting tool:



CONSTRAINTS

Manufacturing practice (continuous vs intermittent)

Condition & capabilities of available machine tools (rigidity)

Geometry, finish, accuracy & surface integrity requirements

Workholding devices (rigidity)

Required processing time (production schedule)

INPUTS

Work material (composition & metallurgical state)

Type of cut (roughing vs finishing continuous cut vs intermittent)

Part geometry & size

Lot size (small batch vs mass production)

Machinability data – tool life/specific HP

Quality/capability needed

Past experience of decision maker

TOOL SELECTION DECISION

Availability (available materials, their composition, properties and applications, available sizes, shapes and geometry, availability, delivery schedule, cost and performance data)

OUTPUTS

Selected tools (specific tool material, grade, shape and tool geometry)

Cutting parameters
speed (rpm)
feed
depth of cut
cutting fluids

FIGURE The selection of the cutting tool material and geometry and the cutting conditions for a given application depends on many variables.

Cutting tools for machining

Proper Tool: geometry and material

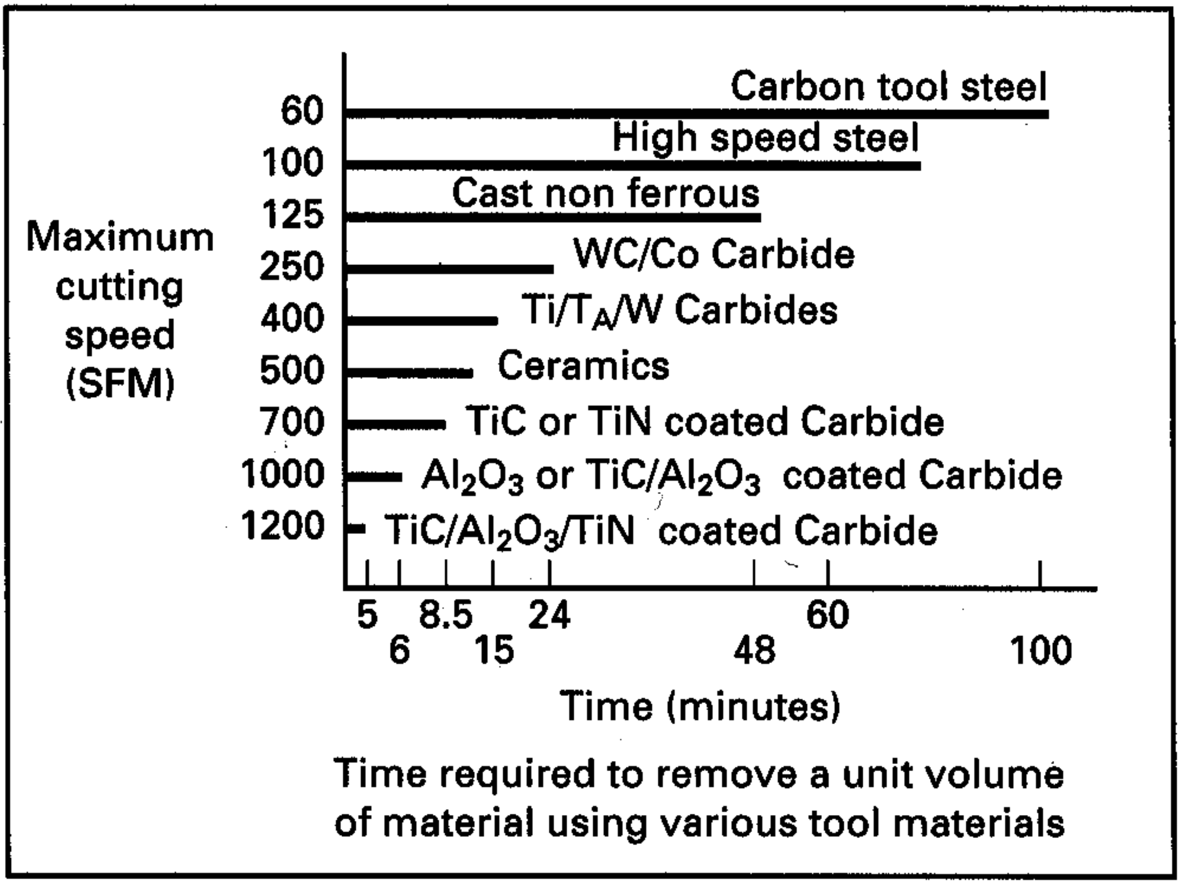
Selection: Work material

Part characteristics (geometry, accuracy, finish, etc.)

Machine tools characteristics (work holders)

Fixture

support systems (Operator, sensors, lubrication, etc.)



Time required to remove a unit volume of material using various tool materials

FIGURE 22-1
Improvements in cutting tool materials have reduced machining time.

Tool is subjected to 1000C, severe friction, high local stresses.

Cutting Tool Requirements :

High hardness
High toughness(impact strength)
Strength to resist bulk deformation
Good thermal properties
Consistent tool life
Resistance to abrasion,wear, chipping
High hot hardness
Chemical stability with temperature
High elastic modulus (stiffness)
Correct geometry and surface finish

Hardness of materials for cutting tools:

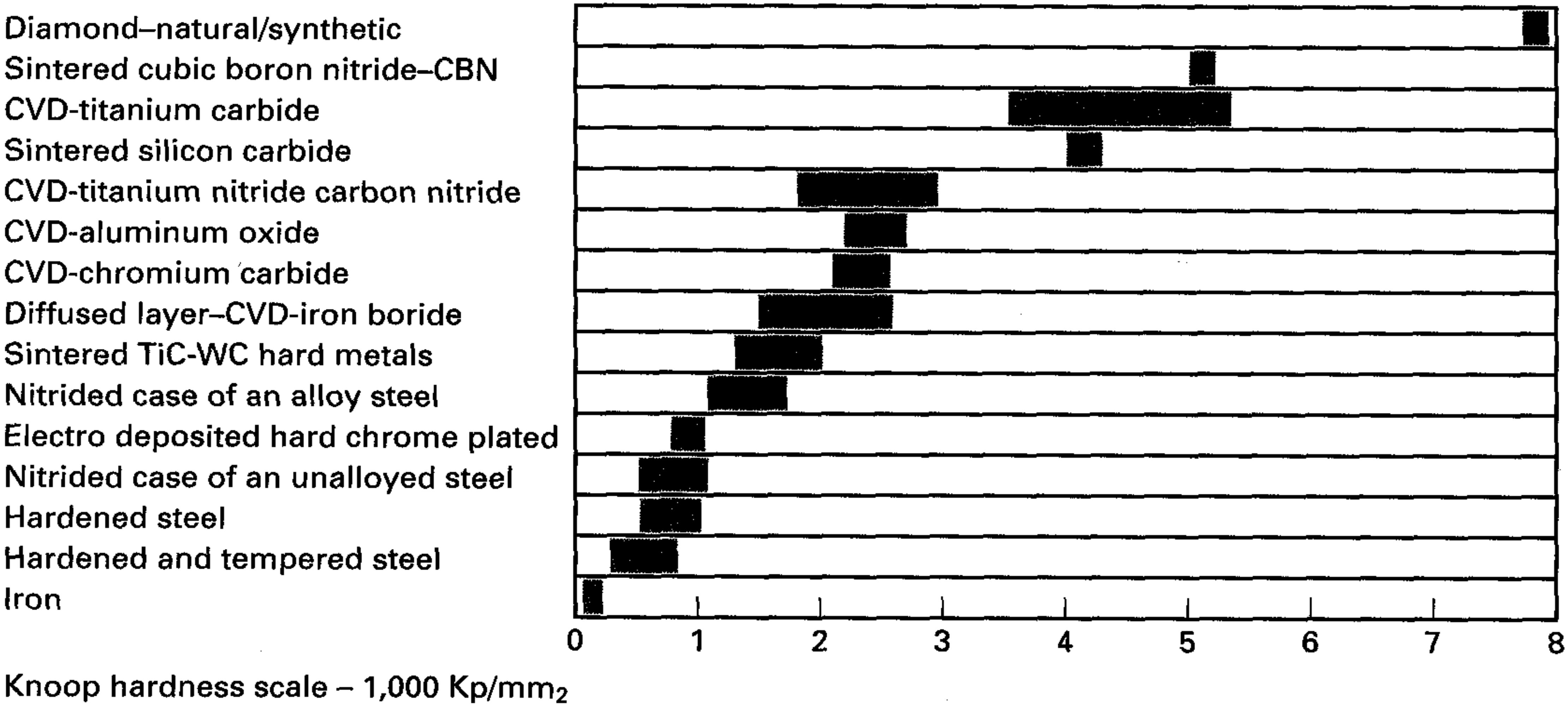


FIGURE Vickers hardness ranges for various cutting tool materials.

The cutting tool should be

1. Hard to resist wear
2. Tough to resist cracking and chipping

→ **High Hot Hardness**

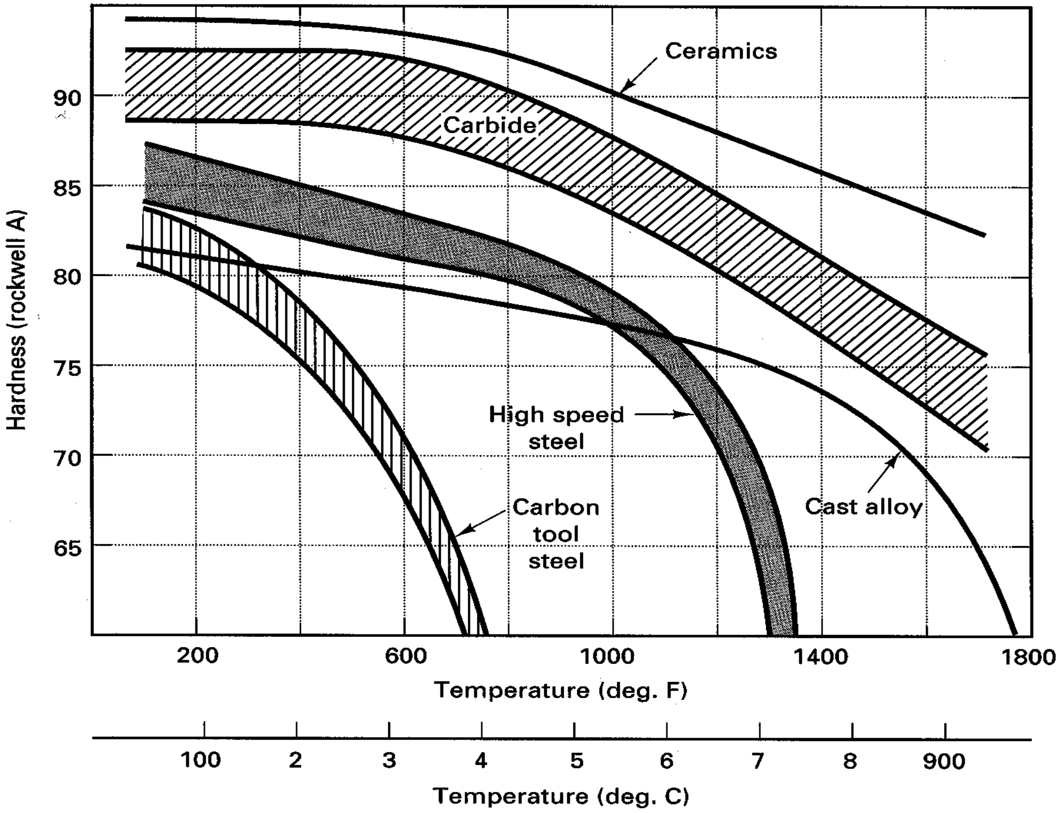


FIGURE Hardness of tool materials decreasing with increasing temperature. Some material display a more rapid drop in hardness above some temperatures. (From Metal Cutting Principles, 2nd ed.; *courtesy of Ingersoll Cutting Tool Company.)*

Life of the tools: depends on the regime the tool is subjected to

TABLE 22-1. Salient Properties of Cutting Tool Materials[a]

	Carbon and Low/Medium-Alloy Steels	High-Speed Steels	Sintered (Demented) Carbides	Coated HSS	Coated Carbides	Ceramics	Polycrystalline CBN	Diamond
Toughness	▶		Decreasing					▶
Hot hardness	▶			Increasing				▶
Impact strength	◀	Decreasing						◀
Wear resistance	▶					Increasing		▶
Chipping resistance	◀	Decreasing						◀
Cutting speed	▶					Increasing		▶
Depth of cut	Light to medium	Light to heavy	Light to heavy	Light to heavy	Light to heavy	Light to heavy	Light to heavy	Very light for single-crystal diamond
Finish obtainable	Rough	Rough	Good	Good	Good	Very good	Very good	Excellent
Method of manufacture	Wrought	Wrought cast, HIP sintering	Cold pressing and sintering, PM	PVD[b] after forming	CVD[c]	Cold pressing and sintering or HIP sintering	High-pressure–high-temperature sintering	High-pressure–high-temperature sintering
Fabrication	Machining and grinding	Machining and grinding	Grinding	Machining and grinding, coating	Grinding before coating	Grinding	Grinding and polishing	Grinding and polishing
Thermal shock resistance	▶						increasing	▶
Tool material cost	▶						increasing	▶

---

[a] Overlapping characteristics exist in many cases. Exceptions to the rule are very common. In many classes of tool materials a wide range of composition and properties is obtainable.

[b] Physical vapor deposition.

[c] Chemical vapor disposition.

Tool Coating Processes

- TiN & TiC
  - CVD : carbide inserts and steel → 950-1050 °C
    - Tooling with loose tolerances
    - Punches, trim dies
    - Solid carbide tooling
  - PVD : TiN coatings on HSS and carbide-tipped cutting tools → 200-485 °C
    - HSS, solid carbide and carbide-tipped tools
    - Fine punches, dies (fine tolerances)
    - Composition independent process; virtually all tooling materials

Cutting Tool Materials

- Tool steels: Carbon steels (0.90-1.30%) and low/medium alloy steels → loose their hardness above 400 °F through tempering
  - Low/medium alloy steels also lose their hardness above 300-650 °F, limited abrasion resistance → relatively inexpensive cutting tools (drills, taps, reamers, broaches) → low-temperature machining
- High-speed steels (HSS): High-alloy steels → up to 1100 °F : good red hardness
- Cutting speeds are higher by three or two-fold than those of tool steels → HSS
- Contain: Fe & C + W, Mo, Co, V, Cr (strength, beyond the tempering temperature [hot hardness], hardness, and wear resistance)
    - Toughness – superior rupture strength
    - Easy to fabricate
    - Suitable for forming complex tool geometry (gear cutters, taps, drills, reamers)

Cutting Tool Materials – 2

- TiN-coated HSS:
- Gear-shaper cutters, drills, reamers, taps, broaches, insert tooling, bandsaw, circular saw, end mills and other milling cutters
    - Improved tool wear
    - Higher hardness, higher abrasion resistance, longer tool life
    - Relative inertness (increased adhesion-related tool life)
    - Low coefficient of friction (→increased shear angle→ reduced cutting forces, spindle power, and heat generation)

Cutting Tool Materials – 3

- Cast Cobalt alloys: *stellite tools*
  - Cobalt-rich, Cr, W and WC and CrC cast alloys
  - Intermediate range between HSS & cemented carbides
  - Higher hot hardness than HSS → higher cutting speeds (25% higher) than those of HSS
  - Cannot be softened or heat treated (hard as cast)
  - Usually cast to shape and finished to size by grinding
  - Come in simple shapes only (single-point tools, saw blades - due to high hardness as cast condition)
  - Currently being phased out mostly because cost

Cutting Tool Materials – 4

- Carbide or Sintered Carbides: Inserts
  - Straight tungsten grades (for machining cast irons, austenitic stainless steel, nonferrous and nonmetallic materials)
  - Grades with high amount of Ti, Ta (for machining ferritic workpieces)
  - Carbides are nonferrous alloys → also called *sintered* or *cemented* carbides → powder metallurgy techniques (Cobalt being the binder)
  - Come as either straight WC or multicarbides of W-Ti or W-Ti-Ta depending of the workpiece material
  - Much harder, chemically more stable, better hot hardness, higher stiffness, lower friction, higher cutting speeds than HSS
  - More brittle & more expensive (due to use of strategic metals – W, Ta, Co)

Cutting Tool Materials – 5

- Carbide or Sintered Carbides: Inserts – Cont’d
  - They come in insert form: squares, triangles, diamonds & rounds
  - Can be either brazed or mechanically clamped (*more popular*) onto the tool shank
  - Can be purchased in the as pressed state or they can be ground to finish tolerances
  - A chip groove with a positive rake angle (reduce cutting forces)
  - Carbide inserts are recycled after use to reclaim the Ta, WC, and Co (strategic materials)

Type tool materials: Nutshell

- Plain Carbon Steel (0.9% to 1.3% C), hardened and tempered, 400F

- Low-Medium Alloy Steel, (Mo, Cr:Hardness, W,Mo:Wear resistance,), hardened and tempered, e.g. drills, taps, etc.

High speed steels (HSS) – Co, W, Mo, Cr (Increased hardness,wear resistance), Rc65-70, Greater toughness, easily complicated geometries can be fabricated

TiN Coated HSS: TiN PVD Coated, Rc 80-85,

Cast cobalt alloys (stellite tools) – Co, Cr, W, C Cast ally + V,B, Ni, Ta,
Simple shapes
Between HSS and Carbide tool
25% higher cutting speeds than HSS

Carbides or sintered carbides WC, TiC, TaC, v>300m/min Used as inserts

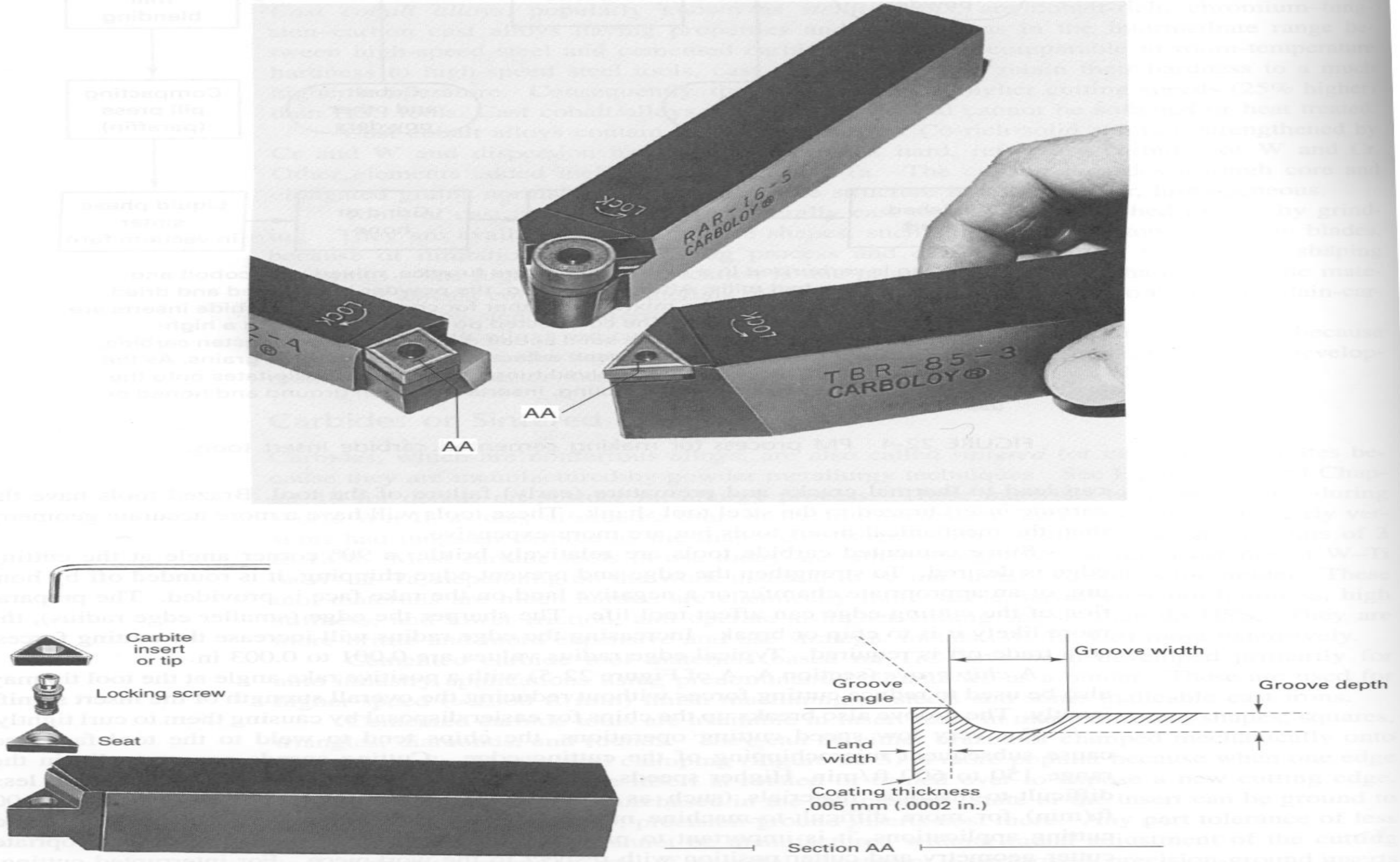
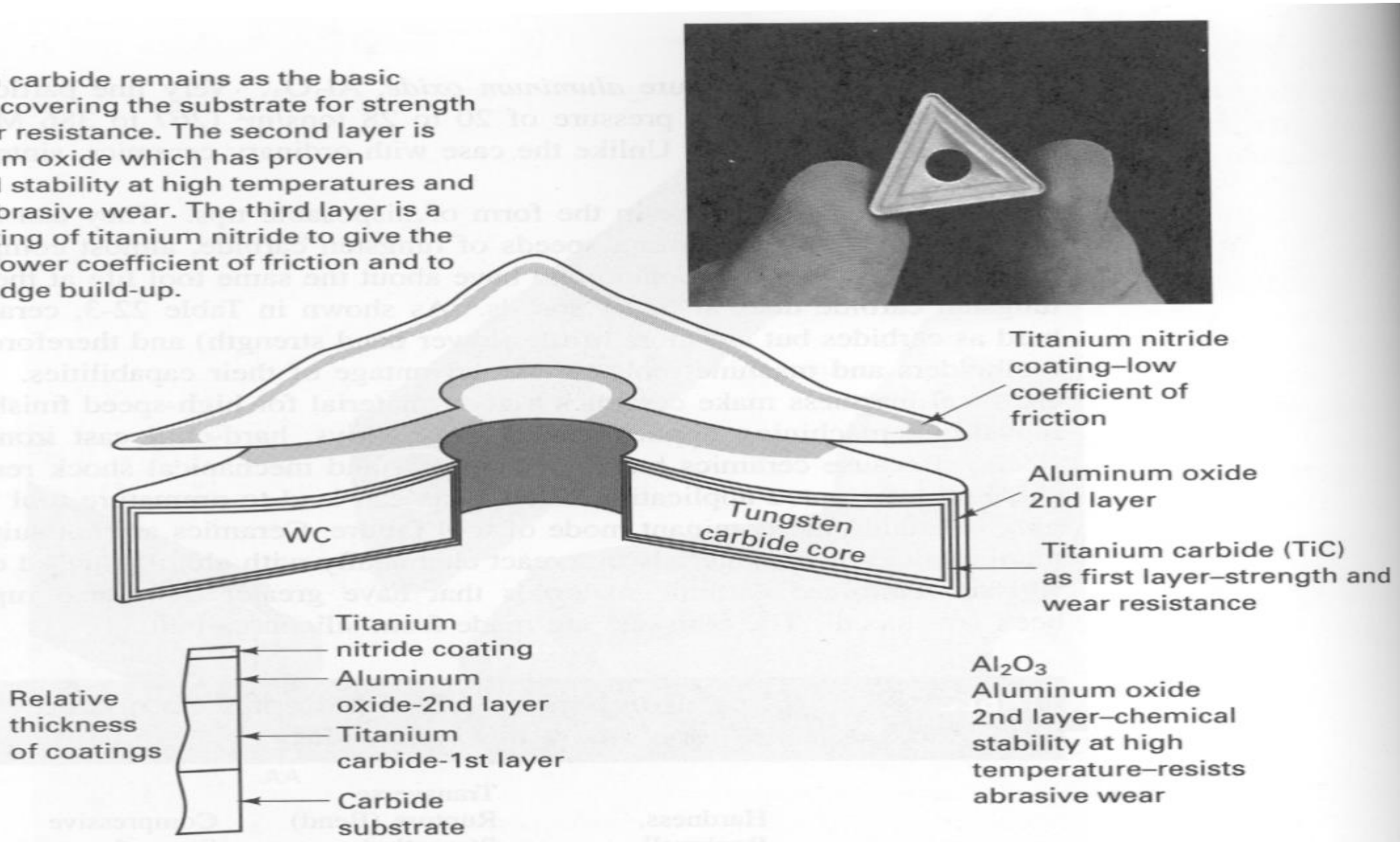


FIGURE (Top) Examples of throwaway carbide cutting tool tip with chipbreaker grooves on rake face; (Left) components of a typical mounting holder; (Section AA) groove design on coated tool to reduce forces and breakup chips. *(Courtesy of General Electric.)*

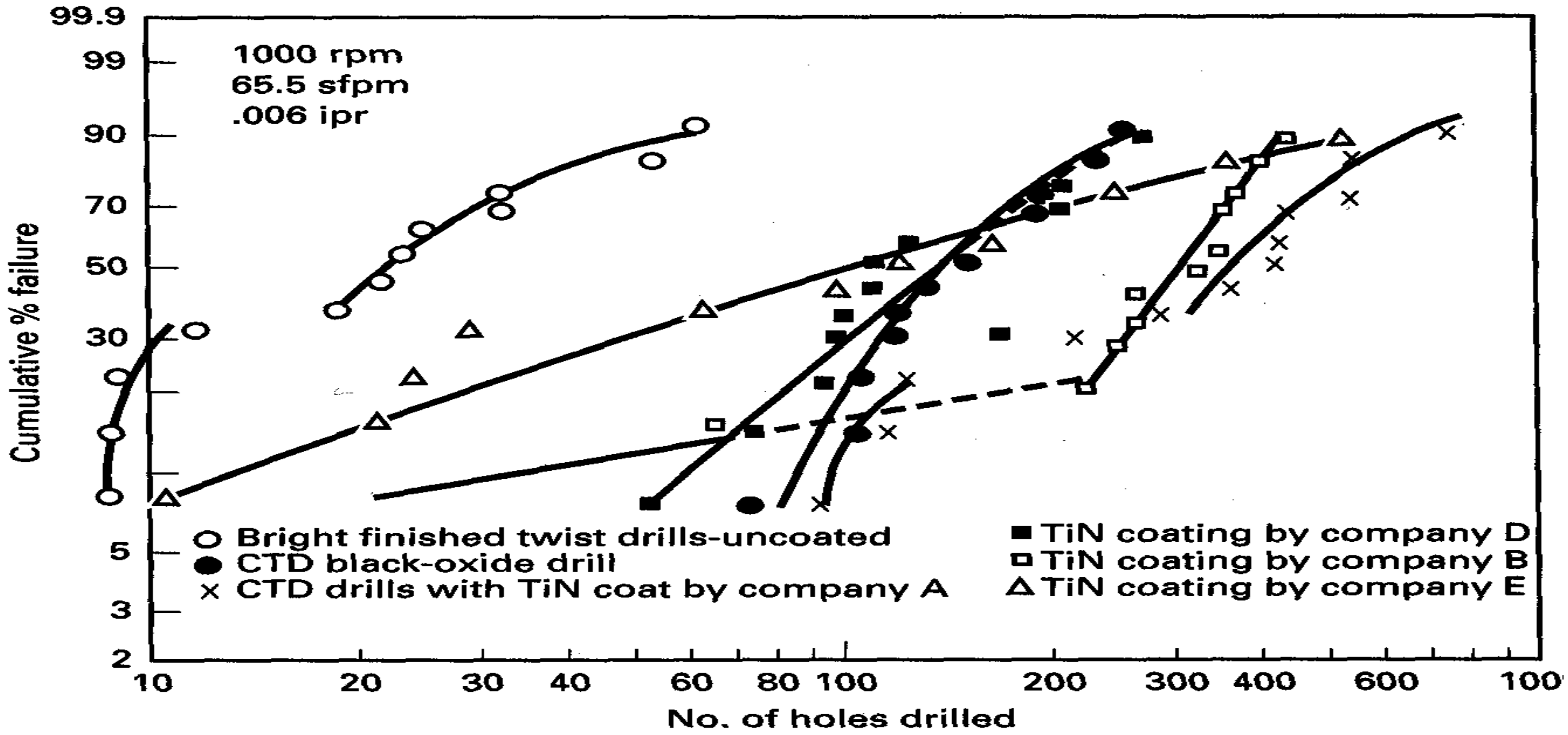
Ceramics – Al_2O_3 –Sintering- brittle

Carbide coated tools WC +TiC+ $Al2O_3$+ TiN (Abrasion resistant, hard, chemically inactive)



Titanium carbide remains as the basic material covering the substrate for strength and wear resistance. The second layer is aluminium oxide which has proven chemical stability at high temperatures and resists abrasive wear. The third layer is a thin coating of titanium nitride to give the insert a lower coefficient of friction and to reduce edge build-up.

FIGURE 22-6 Triple-coated carbide tools provide resistance to wear and plastic deformation in machining of steel, abrasive wear in cast iron, and built-up edge formation.



Drill performance based on the number of holes drilled with 1/4 inch diam. drill in T–1 structural steel.

FIGURE Tool life data for various coated drills. Notice how TiN-coated HSS drills outperform uncoated drills.

Cermets: ceramics in metal binders(TiC, Ni, Co, TiN)

Hight Hot Hardness, Oxidation resistance, good finish, cold pressed

Diamonds – for precision machining (High Hardness, Low Friction, very good finish)

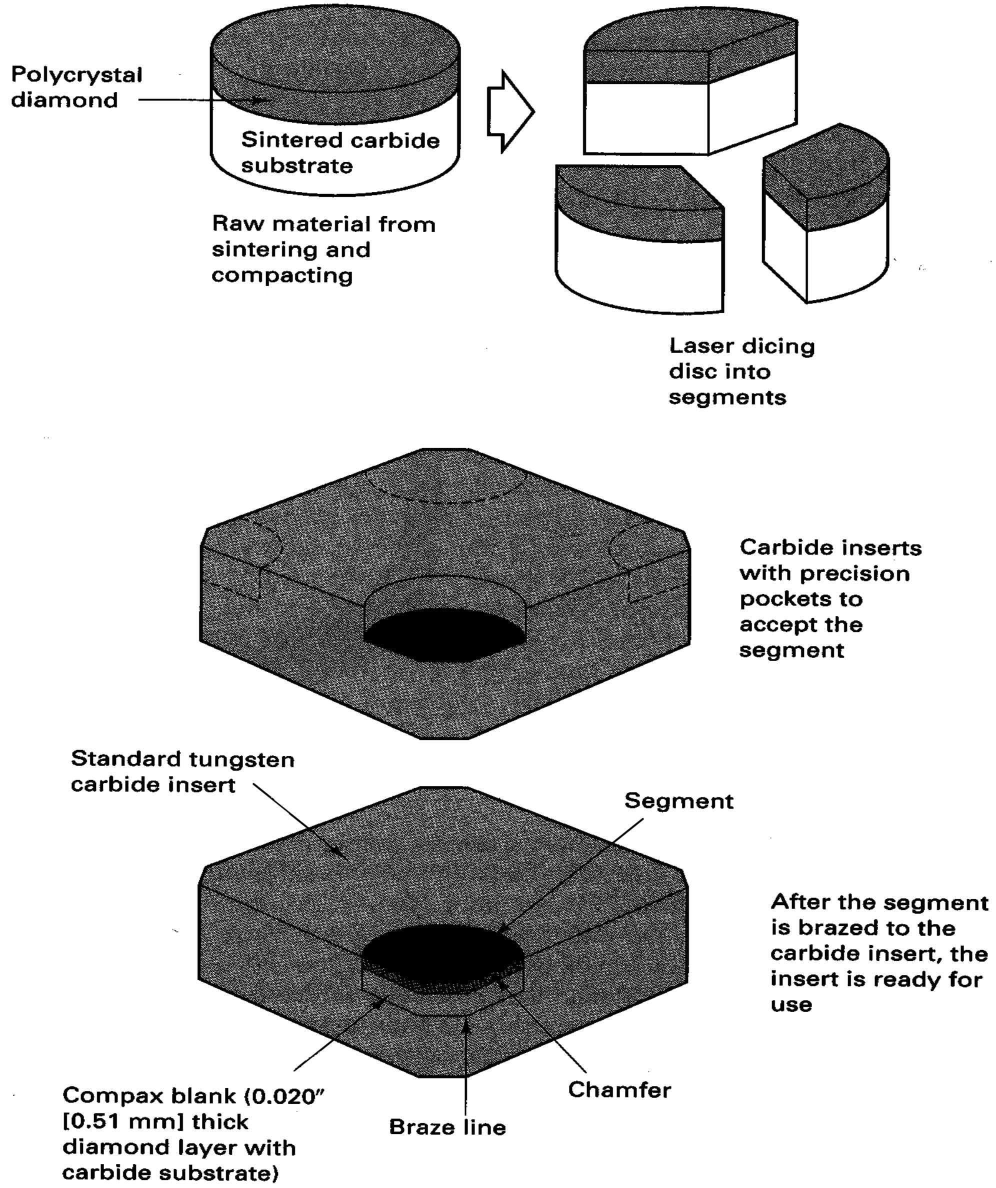


FIGURE Polycrystalline diamond tools are carbides with diamond inserts. They are restricted to simple geometries.

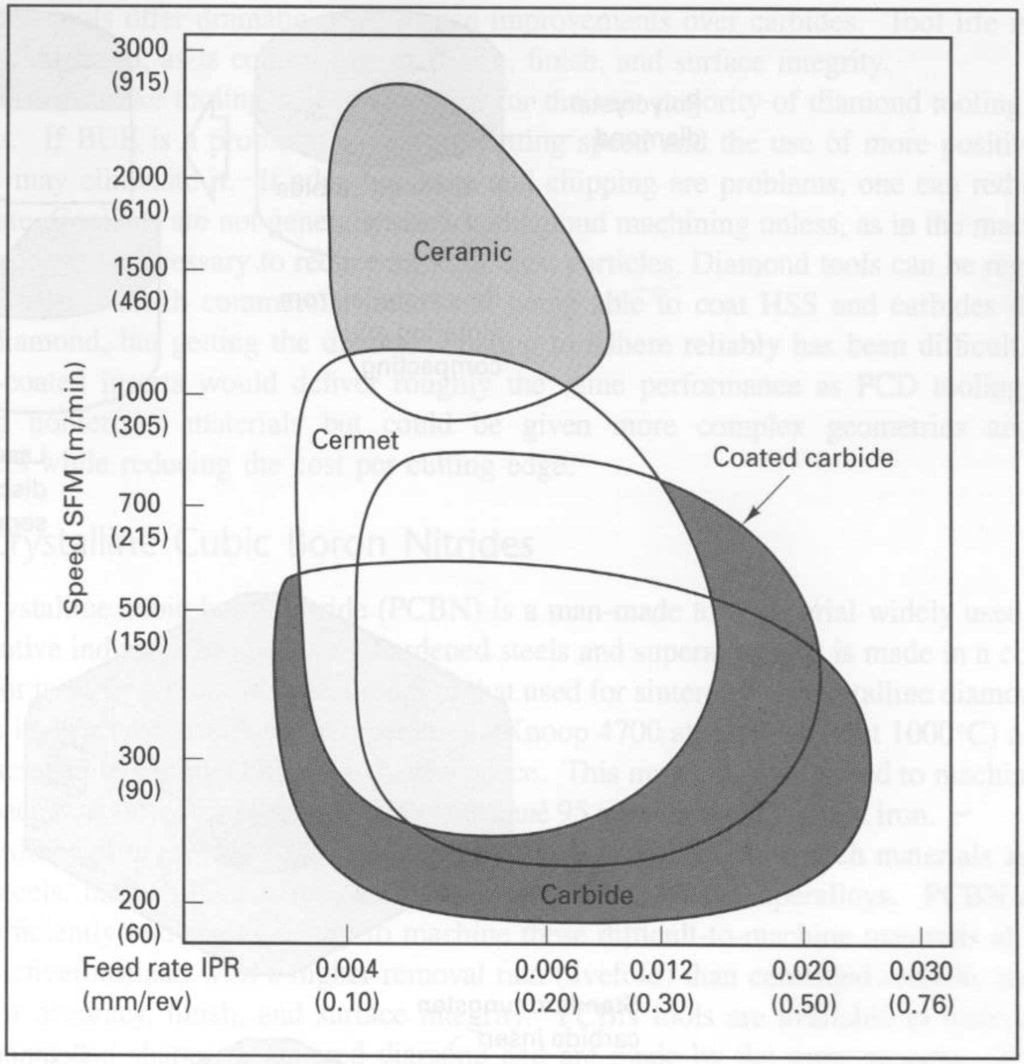
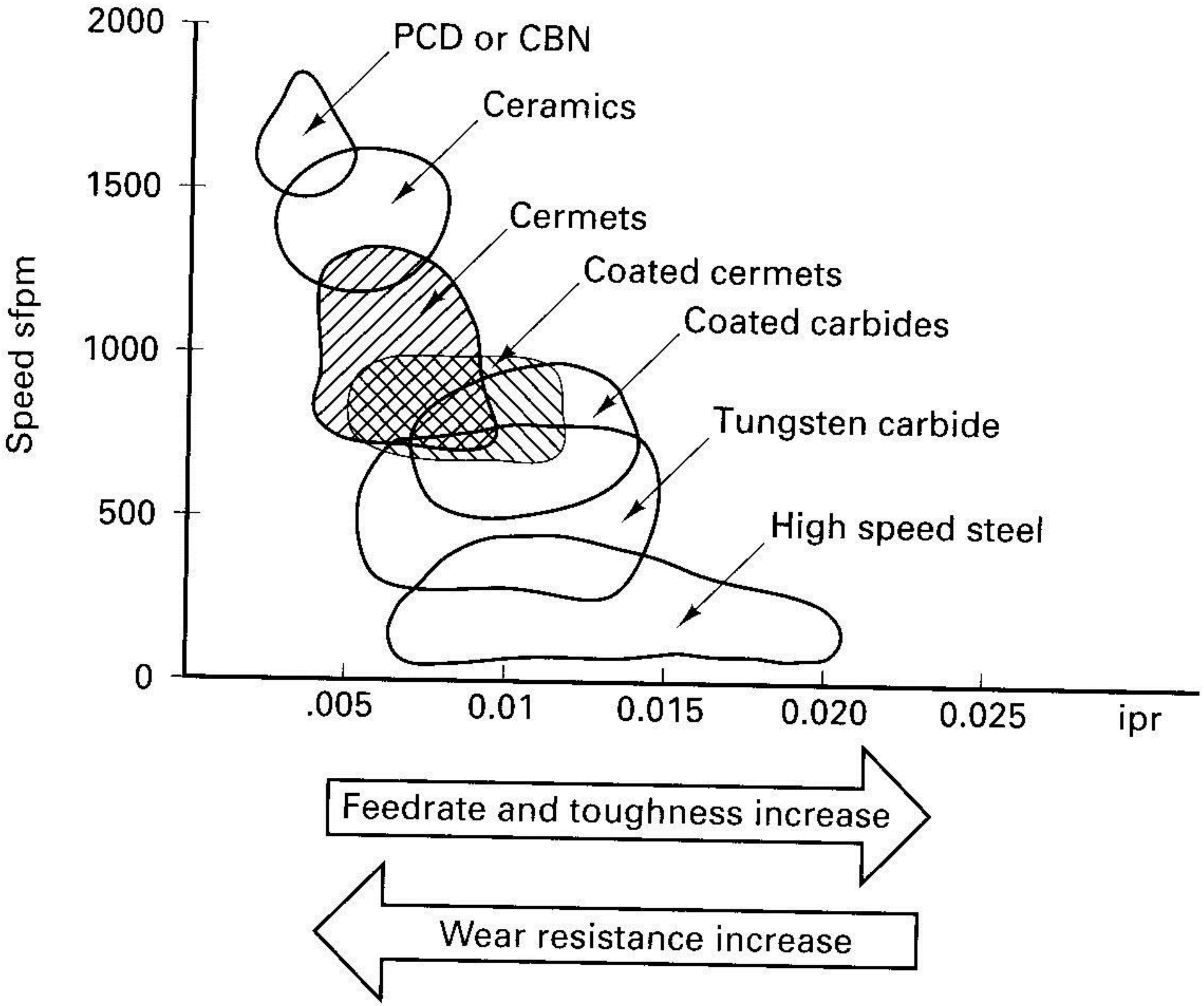


FIGURE Cermets compared to other tool materials in terms of possible cutting speed and feed rates. SFM, surface feet per minute; ipr, inches per revolution.

Comparison of Tool Materials



Tool Geometry

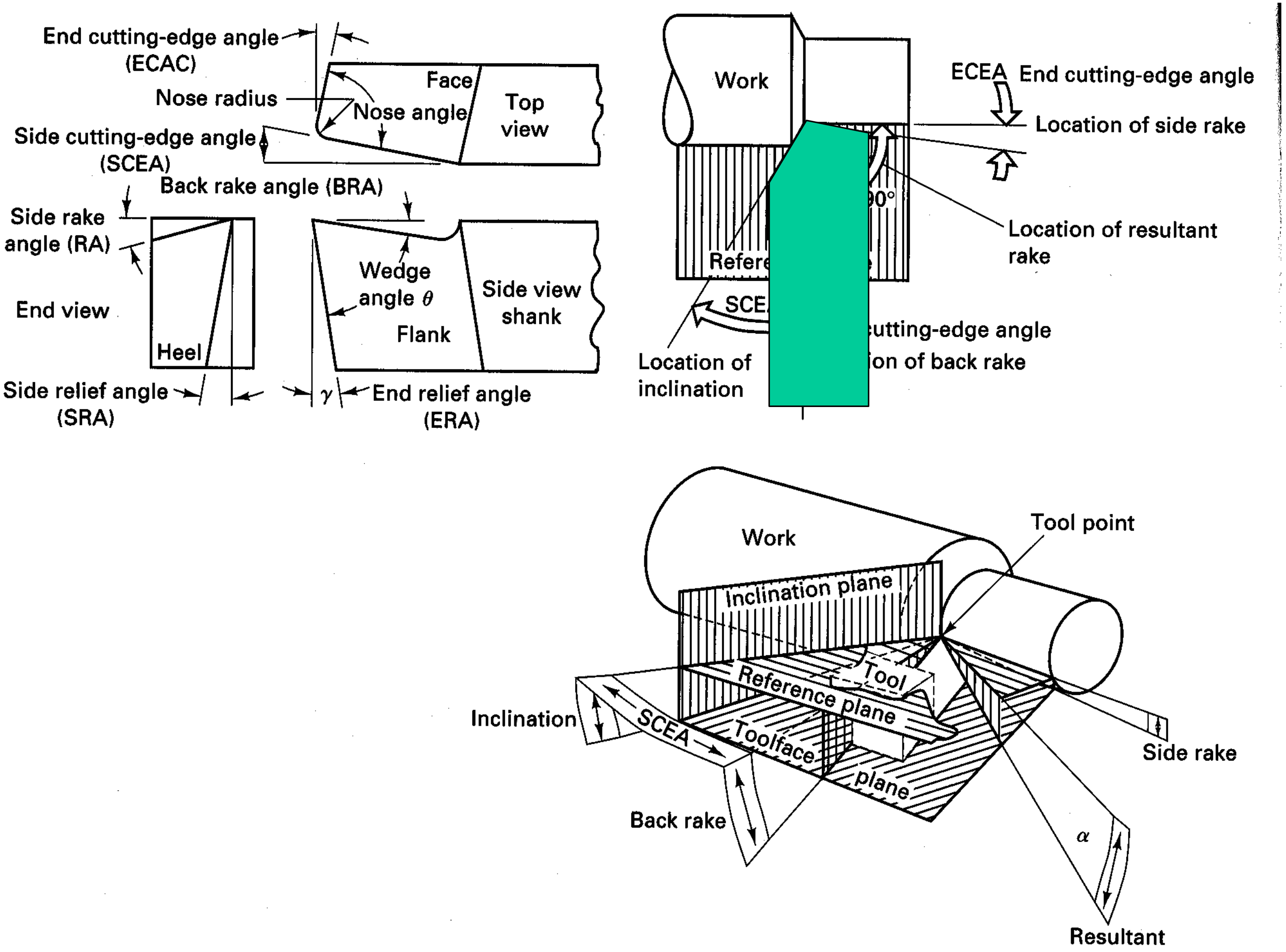


FIGURE Standard terminology to describe the geometry of single-point tools.

+ve Rake angle	-ve Rake angle
Cutting force ↓	
Smaller deflections	
	Useful for hard materials
	-ve for carbide and diamond tools

Hardness:

Higher the hardness → smaller the rake angle

Crabide tools: -6 to +6deg

HSS: +ve rake angle

Cutting force:

Smaller Rake angle → Higher compression force, tool force, friction,
Hot and thick chips, highly deformed chips.

Higher Rake angle → Lower compression force, tool force, friction,
cooler and less deformed chips.

But, Higher Rake angles → Lesser tool strength, Reduced heat transfer.

As wedge angle determines the Tool Strength

Tool Selection: Trade off between force, strength, cooling

Power consumption → 1% decrease for 1deg rake angle

↑ rake angle →↓ power

Wedge angle → increases the tool strength.

Relief angle → Improves surface quality

5-10deg, normal

smaller angles for harder materials.

Nose radius: Determines the cutting force, chipping, surface finish, wear.

↑ radius →↓ wear, ↑life and surface finish,

Tool failure and tool life

Failure mechanism:

Slow-death mechanism – gradual tool wear on flanks and rake

Sudden death mechanism – rapid, unpredictable-plastic deformation, brittle fracture, fatigue fracture

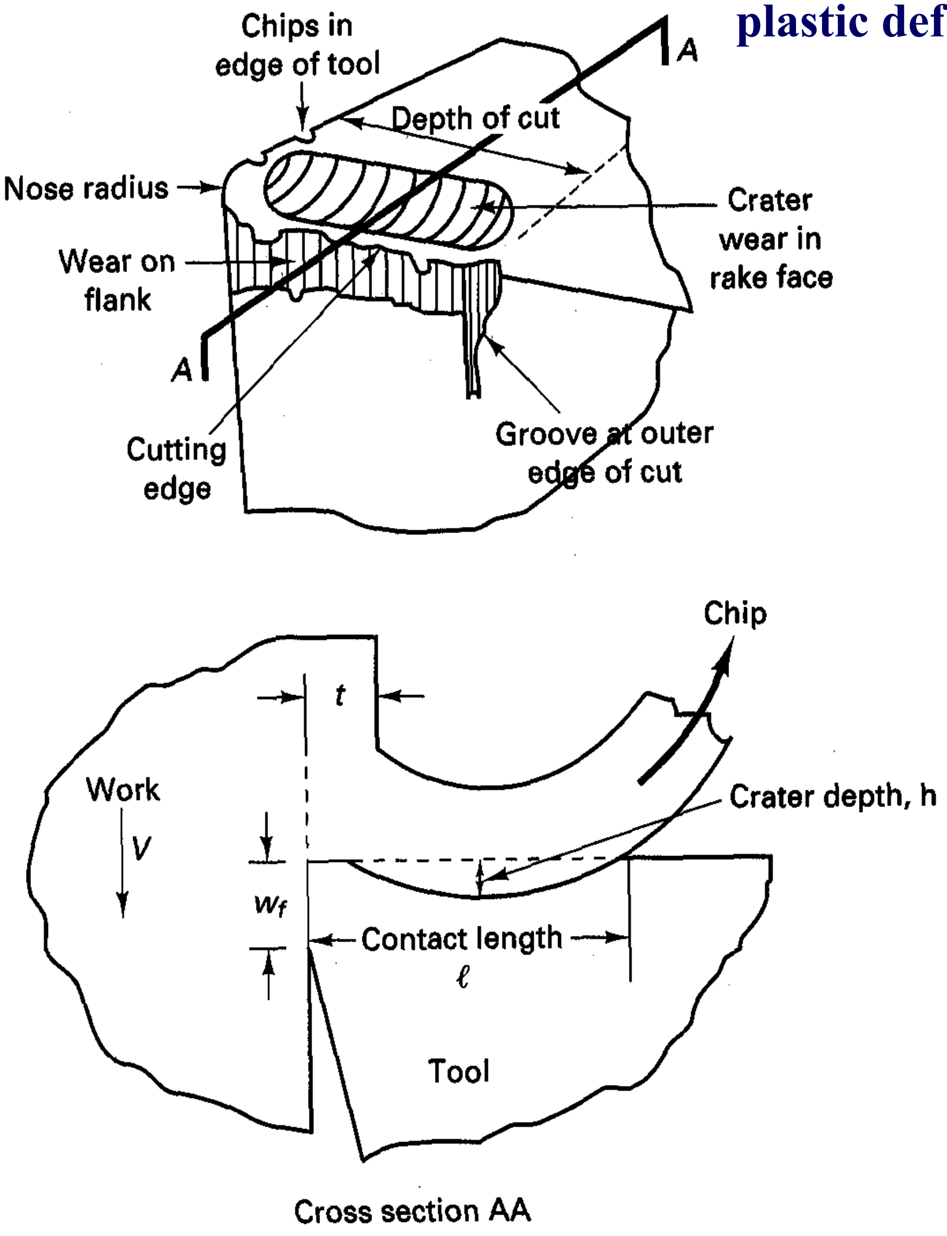


FIGURE Sketch of a worn tool showing various wear elements resulting during oblique cutting: W_f, flank wear land length; t, uncut chip thickness.

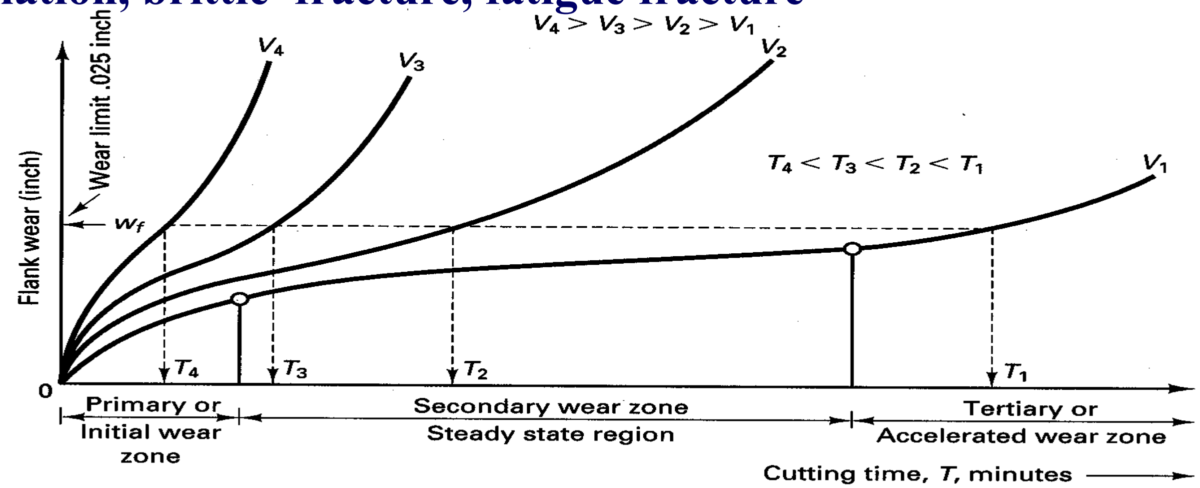


FIGURE . Typical tool wear curves for flank wear at different velocities.

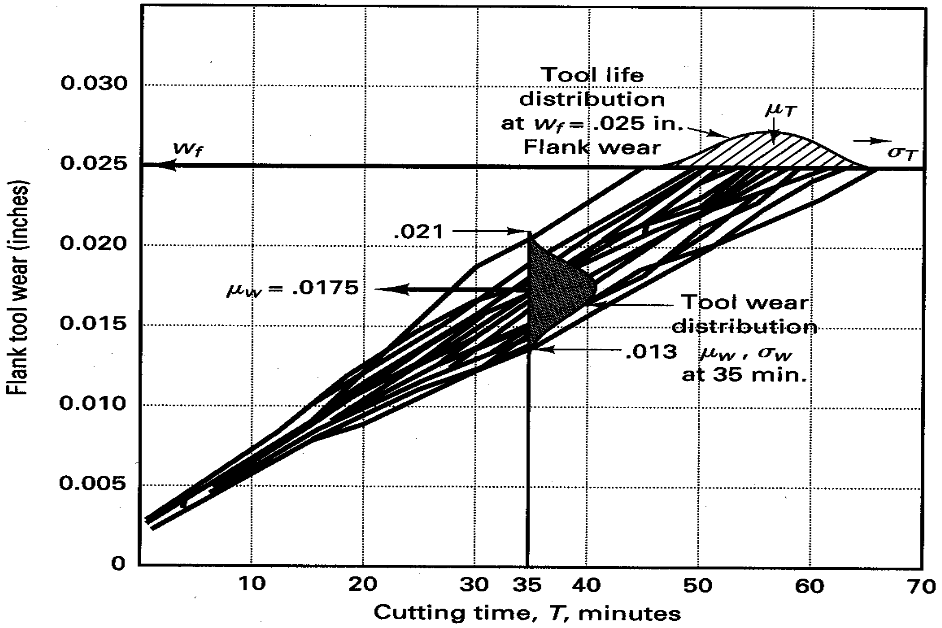


FIGURE Tool wear on the flank displays a random nature, as does tool life.

Taylor Tool Life Model

$$T = \frac{const.}{f_x \times v_y} \quad or \quad VT^n = C$$

n, depends on material

C, depends on all input parameters including feed

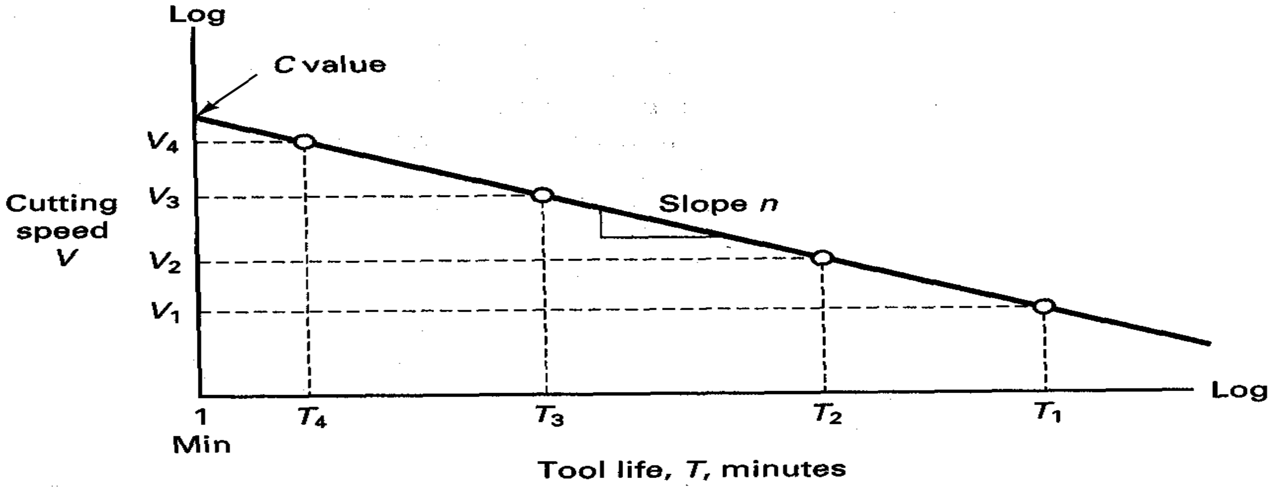


FIGURE Construction of the Taylor tool life curve using data from deterministic tool wear plots like those of Figure 22-13

	n
HSS	0.14-0.16
Carbide	0.21-0.25
TiC insets	0.3
Polydiamonds	0.33
Ceramic coated	0.4

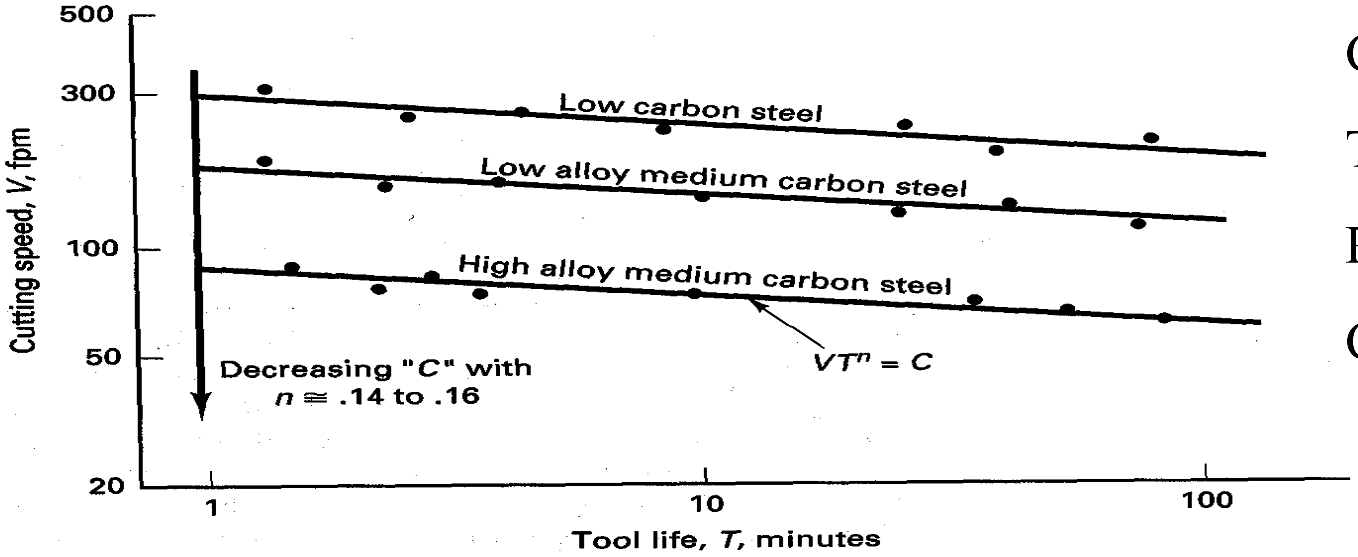


FIGURE : Log-log tool life plots for three steel work materials cut with HSS tool material.

Sources of variability for Tool Life:

Changes in

- •Material (hardness)
- •Geometry of the tool
- •Vibrations
- •Surface Characteristics

Large σ → Life is not predictable

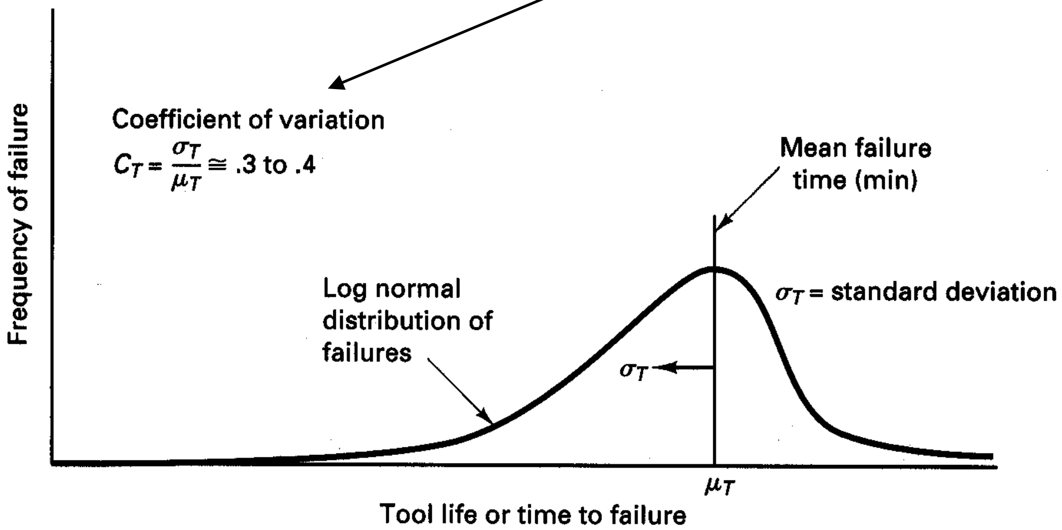


FIGURE Tool life viewed as random variable has a log normal distribution with a large coefficient of variation.

Economics of Machining

50% V↑ → 90%↓ Tool life

50% f ↑ → 60%↓ Tool life

50% d↑ → 15%↓ Tool life

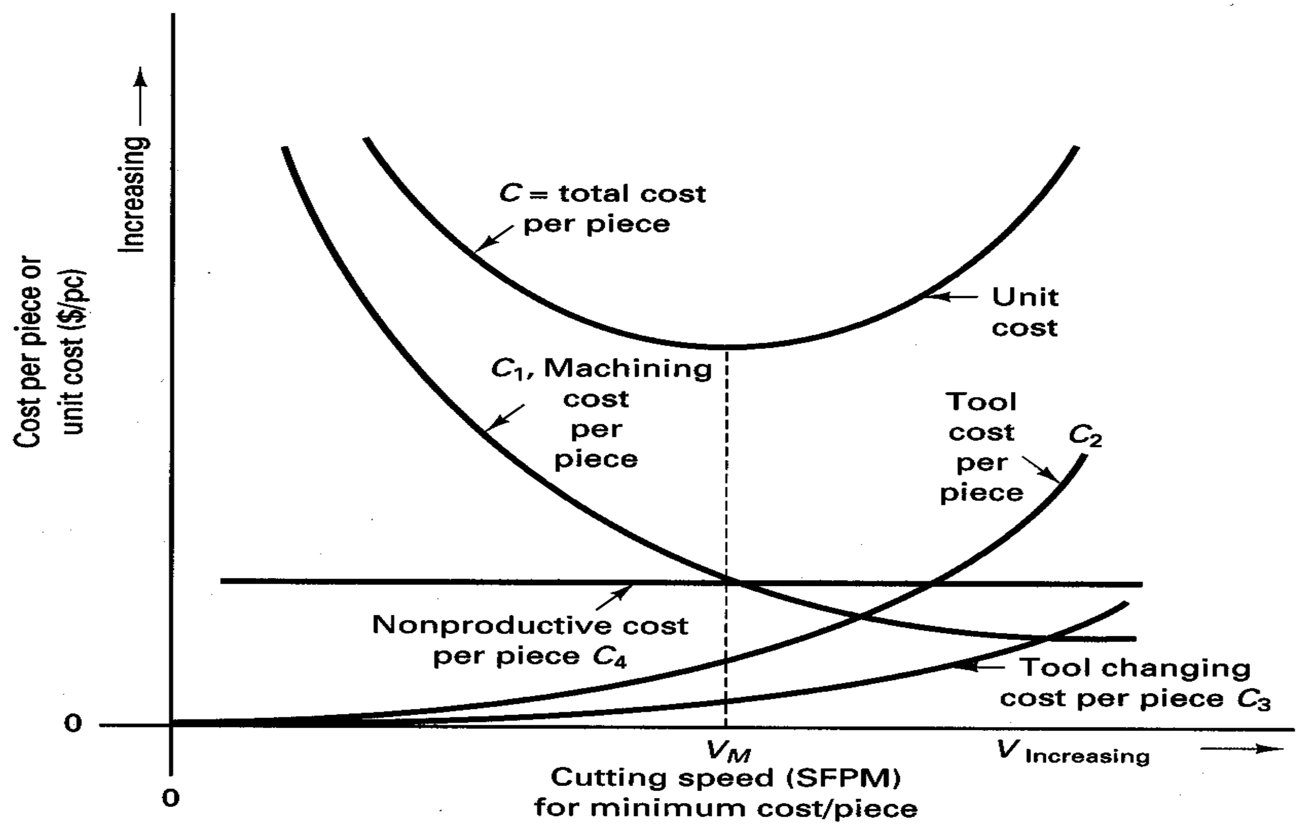


FIGURE Cost per unit for a machining process versus cutting speed.

Reconditioning of the cutting tools.

Whenever possible, re-sharp

Recoating: Cost ~ 1/5 the cost of the tool

Performances of the tool may be reduced

Machinability: ease to perform machining task

Relative evaluations with respect to a standard material cut at a specific speed at the same tool life

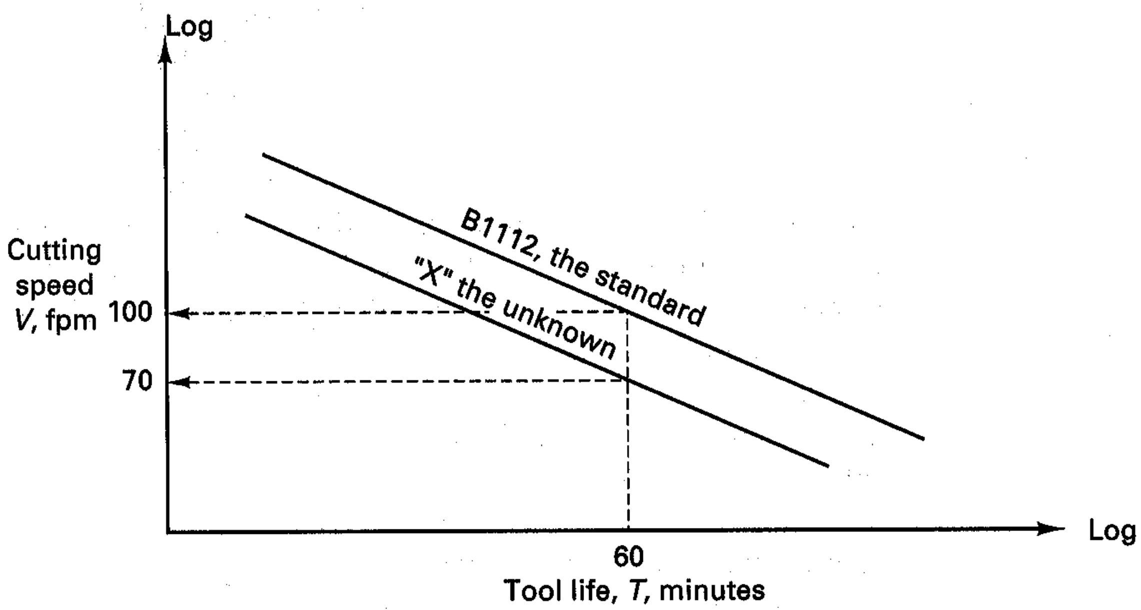


FIGURE 22-19 Machinability ratings defined by deterministic tool life curves.

Reconditioning Cutting Tools

- Resharpen to original tool geometry (CNC grinding machines for tool sharpening)
- Grind cutting edges and surfaces to a fine finish (recoat ground surfaces)
- Remove all burrs
- Avoid overheating, burning or melting the tool

Cutting Fluids

Cooling → increases the cutting speed by two or three-fold → Cutting fluids

Cutting fluids:

- coolant
- lubricant → reducing friction (tool-chip interface & workpiece-tool flank
- helps remove the chip from cutting region
- friction reduction in other regions other than cutting edge-workpiece friction (drilling, sawing, reaming, etc.)
- temperature reduction → hot hardness issue → increased tool life or increased cutting speed for equal tool life
- better dimensional control of the machined workpiece

Water → good coolant(thermal capacity and conductivity), also oxidant, not a good lubricant

Oil → less effective coolant, not an oxidant, some lubrication

Coolants are recycled through restoration

Shaping and Planning:

- single point machining

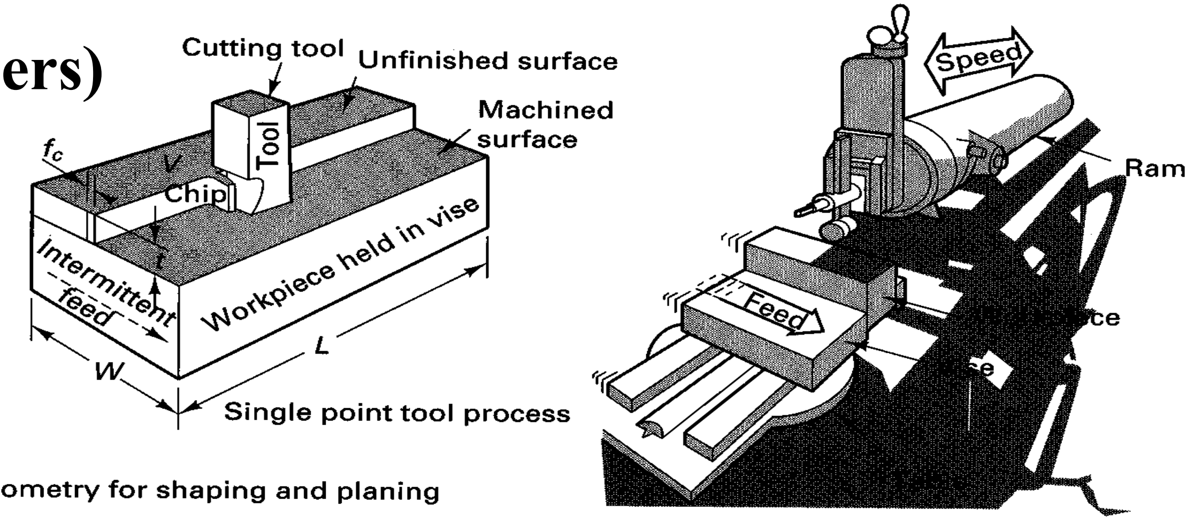
Shapers Horizontal: Push- cut

pull-cut or draw-cut shaper

Vertical: Regular (slotters)

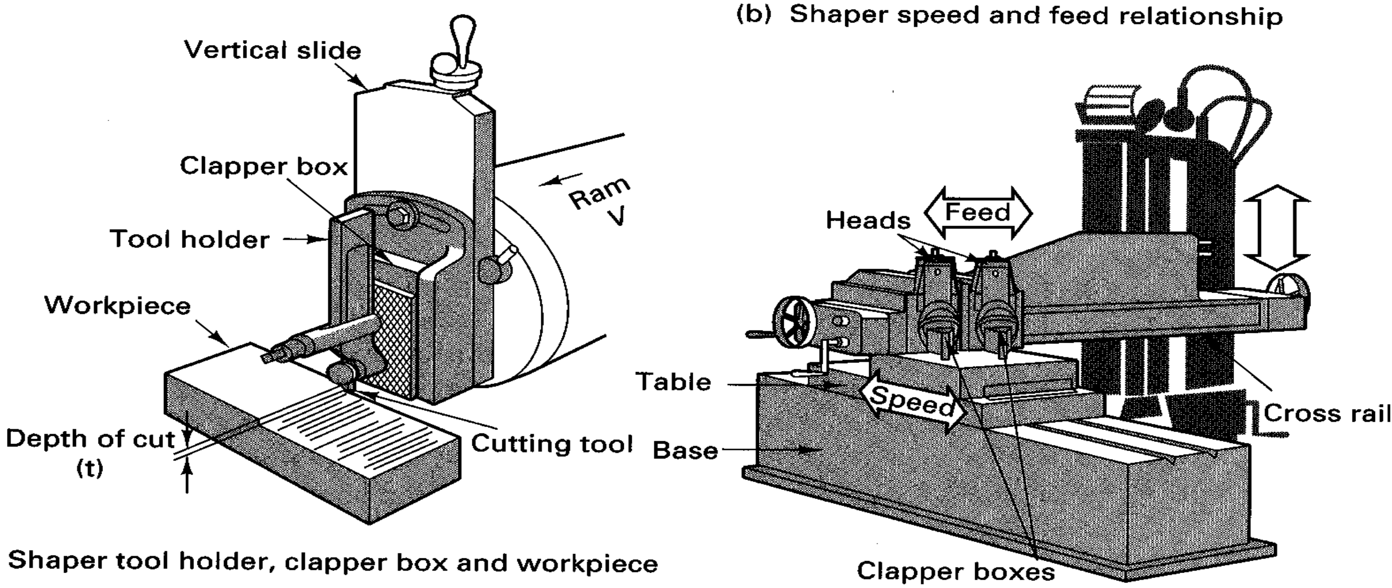
Keyseaters

Special



(a) Basic geometry for shaping and planing

(b) Shaper speed and feed relationship



(c) Shaper tool holder, clapper box and workpiece

(d) Planer machine tool

FIGURE . Basics of shaping and planing.

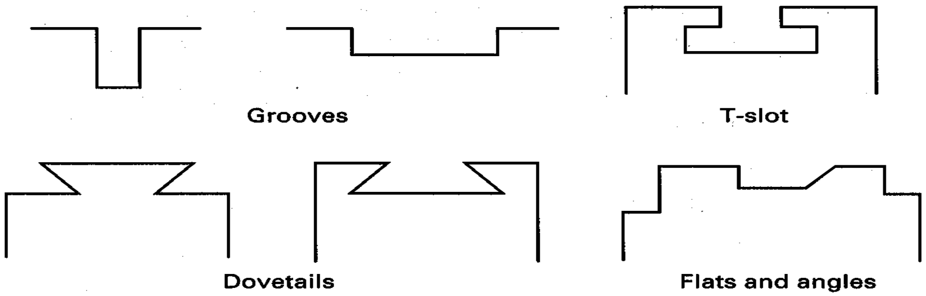


FIGURE Types of surfaces commonly machined by shaping and planing.

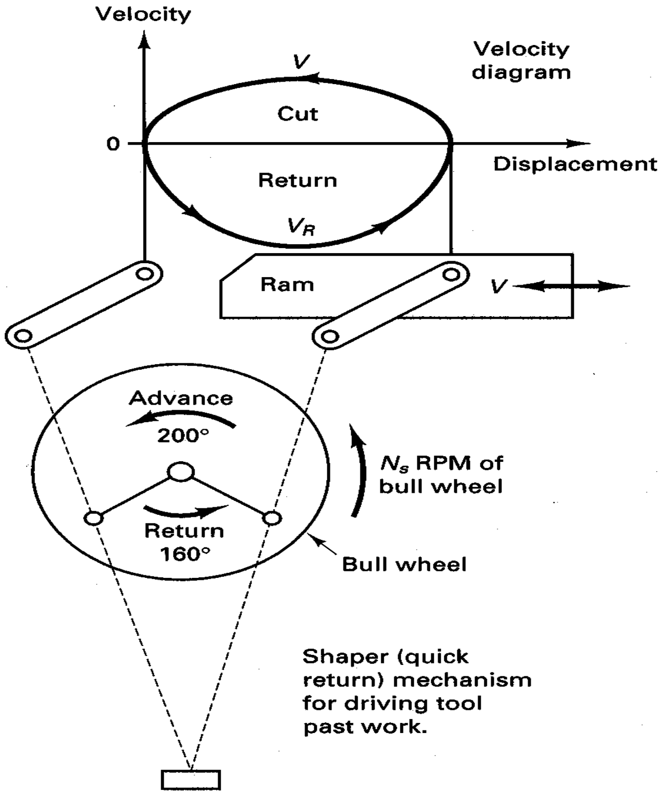


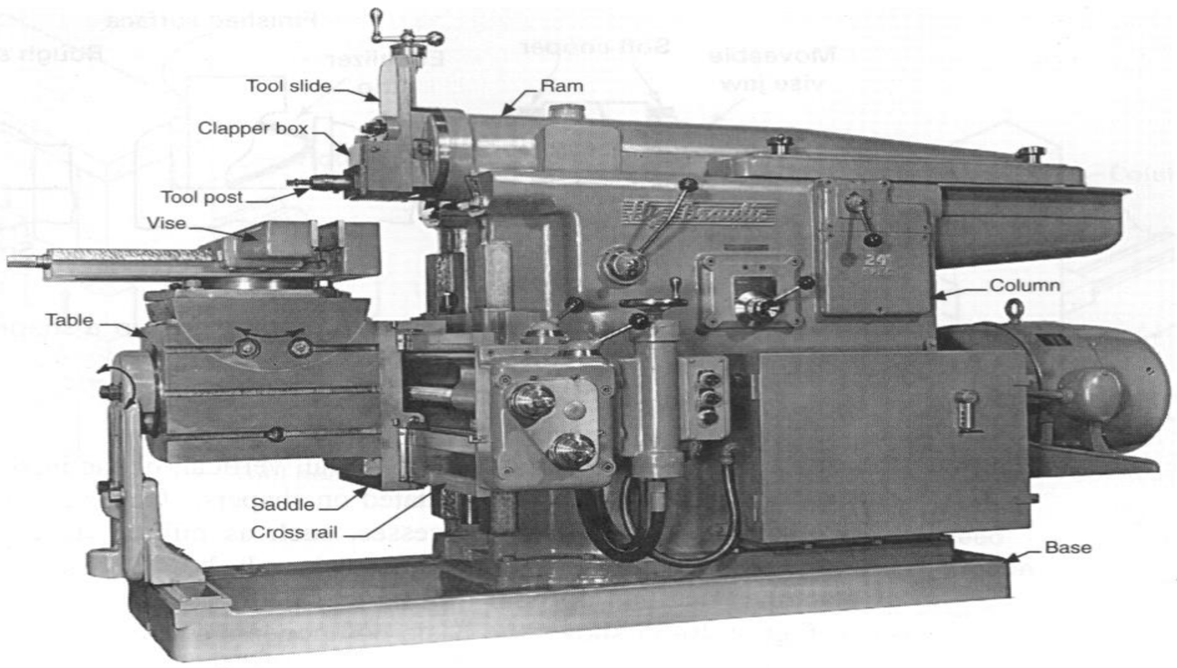
FIGURE The ram of the shaper carries the cutting tool at velocity V and reciprocated at cutting velocity V_R by the rotation of a bull wheel turning at rpm N_s.

Stroke ratio: R_s = cutting stroke angle/360°

$$V = 2\,L\,N_s / R_s$$

$$MRR = (L*W*t)/CT$$

$$CT = W/f_c*N_s$$



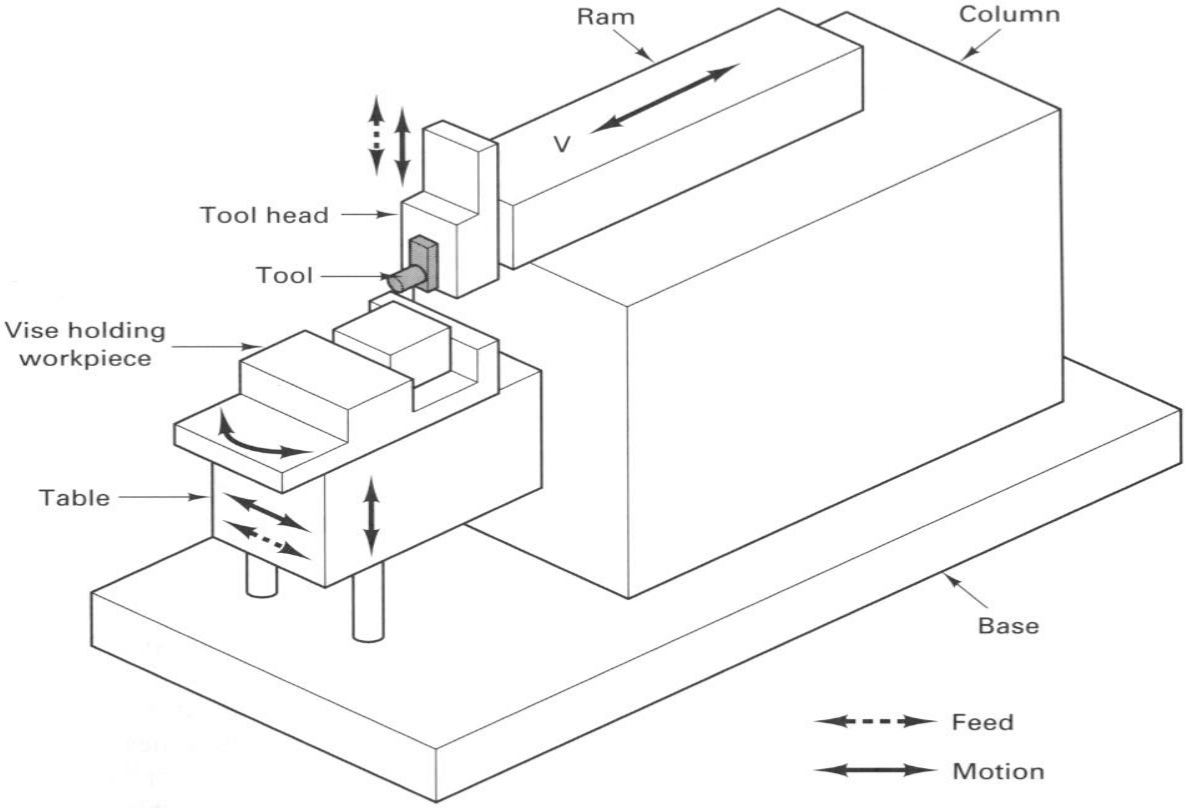


FIGURE Details of a horizontal, push cut shaper.

Work-holding devices

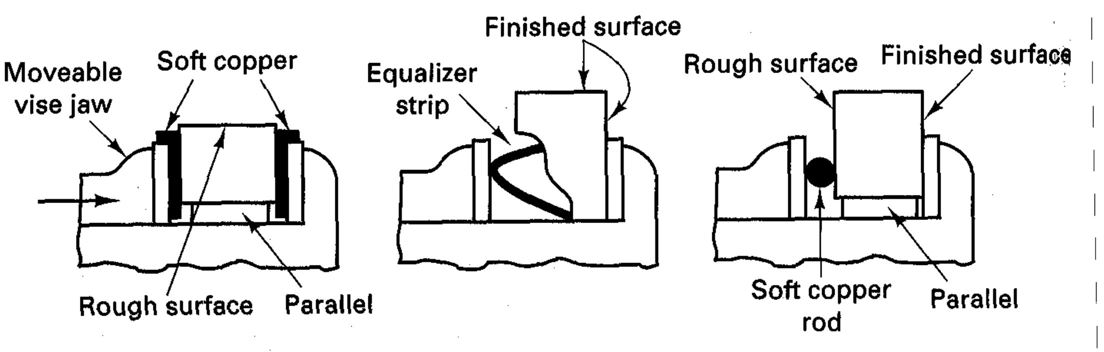
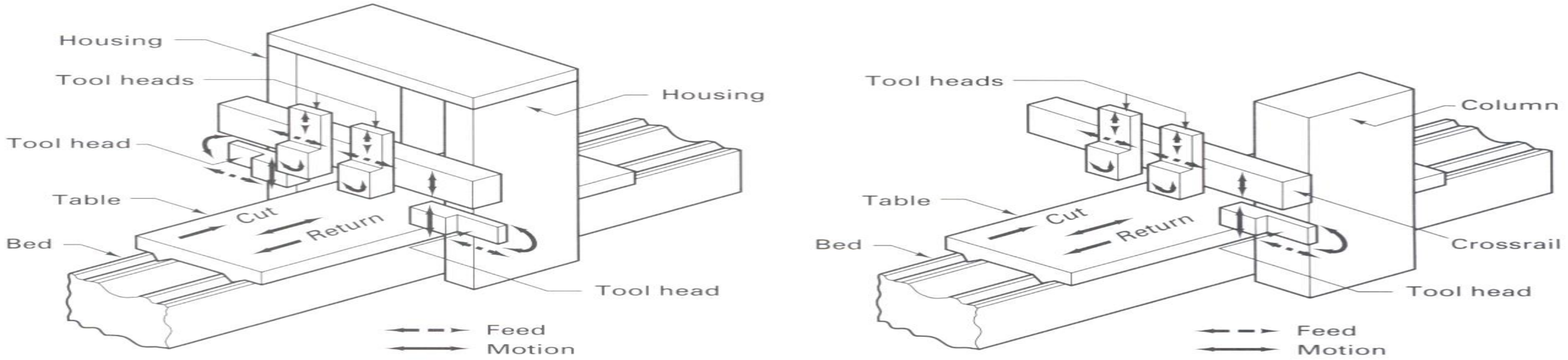


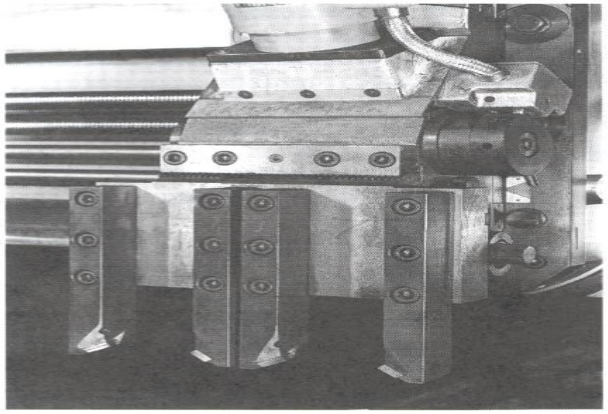
FIGURE Methods of clamping workpieces in a shaper vise.

Planers: Tool is fixed, work is reciprocating



(a) Block diagram showing the basic components of a double-housing planer.

(b) Block diagram of an open-side planer.



(c)

FIGURE Schematic of planers. (a) double-housing planer; (b) open-sided planer; (c) interchangeable multiple tool holder for use in planers. *(Photograph courtesy Gebr. Boehringer GMBH.)*