Machining Processes

IME 240

Material Removal Processes

- Machining is the broad term used to describe removal of material from a workpiece
- Includes Cutting, Abrasive Processes (grinding), Advanced Machining Processes (electrical, chemical, thermal, hydrodynamic, lasers)
- Automation began when lathes were introduced in 1700s
- Now have computer numerical control (CNC) machines
- Machining operations are a *system* consisting of:
 - Workpiece material, properties, design, temperature
 - Cutting tool shape, material, coatings, condition
 - Machine tool design, stiffness & damping, structure
 - Fixture workpiece holding devices
 - Cutting parameters speed, feed, depth of cut

Independent variables

Range of Material Removal Processes

Energy source	Transfer medium	Process	Energy source	Transfer medium	Process
Mechanical	Rigid	tool Machining	Chemical	Liquid	fluid shield Chemical machining
	Granular	vibrations abrasive Ultrasonic machining	Electrical and Chemical	Liquid	tool ECM
	Liquid/ Gaseous (granular)	high pressure jet	Thermal	Gaseous	gas torch Flame cutting Plasma cutting
Electrical	Liquid	(+) EDM	Energy beam	Gaseous or Vacuum	Laser beam machining Electron beam machining Ion beam machining

Material Removal Processes

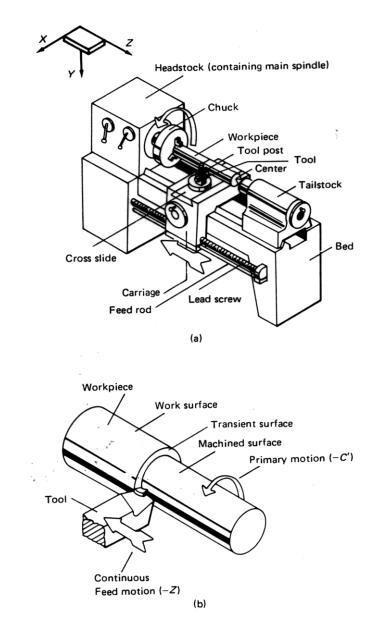
- Material removal processes are often required after casting or forming to:
 - Improve dimensional accuracy
 - Produce external and internal geometric features, sharp corners, or flatness not possible with forming or shaping
 - Obtain final dimensions and surfaces with finishing operations
 - Obtain special surface characteristics or textures
 - Provide the most economical means of producing a particular part
- Limitations, because material removal processes:
 - Inevitably waste material
 - Generally require more energy, capital, and labor than forming or shaping operations
 - Can have adverse effects on the surface quality and properties, unless carried out properly,
 - Generally take longer than shaping a product with other processes

Types of Machining Process

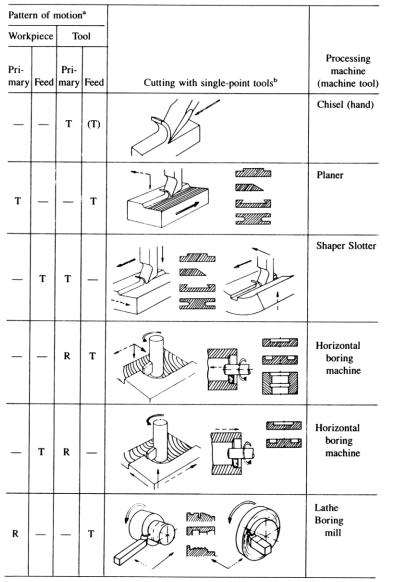
- Single Cutting Edge (Point) Processes
- Multi-Cutting Edge (Point) Processes
- Random Point Cutting Processes Abrasive Machining
- Within each category the basic motions (kinematics) differentiate one process from another

Machine Tool Motions

- Primary motion that causes cutting to take place.
- Feed motion that causes more of the part surface to be machined
- Rotations and/or translations of the workpiece or cutting tool



Single Point Machining Operations

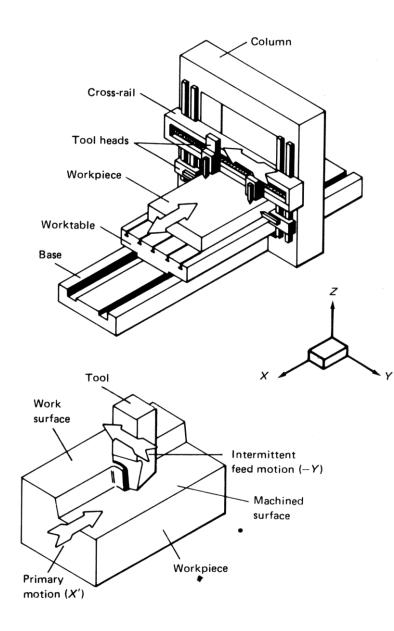


^aR, rotation; T, translation.

^b \Rightarrow , primary motion; -->, feed motion; \rightarrow , adjustment motion.

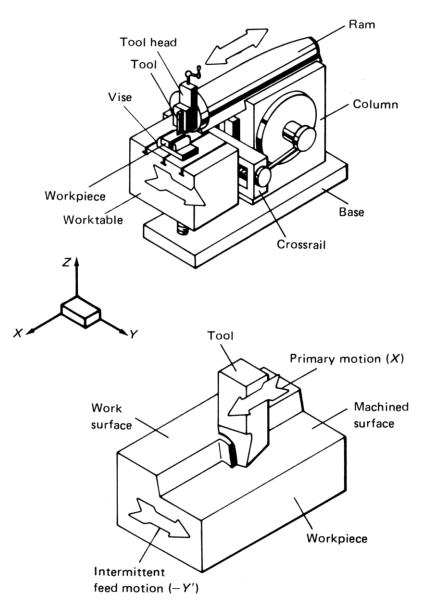
Planing Machine

- Primary motion is translation of the workpiece
- Feed motion is translation of the tool incrementally between cuts



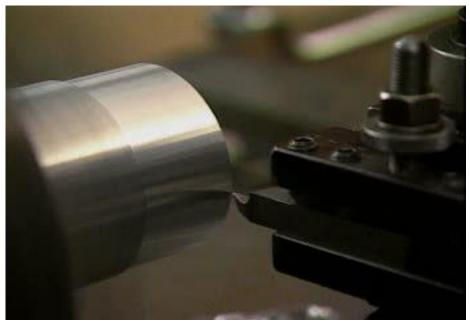
Shaping Machine

- Primary motion is translation of the tool
- Feed motion is translation of the workpiece between cuts



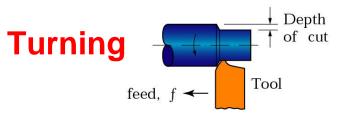
Turning Operations





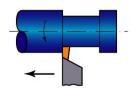
Primary motion is rotation of the workpiece

Feed motion is translation of the cutting tool – continuous driven by spindle rotation (b) Taper turning

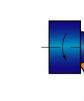


(a) Straight turning

(d) Turning and external grooving

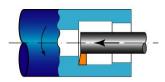


Turning is ^(g) Cutting with a form tool the processs for machining round workpieces on a lathe

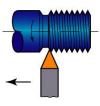


(e) Facing

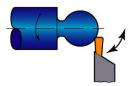
(h) Boring and internal grooving

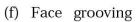


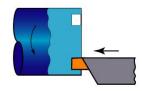
(k) Threading



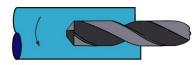
(c) Profiling



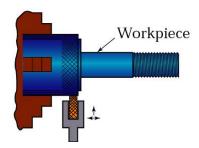




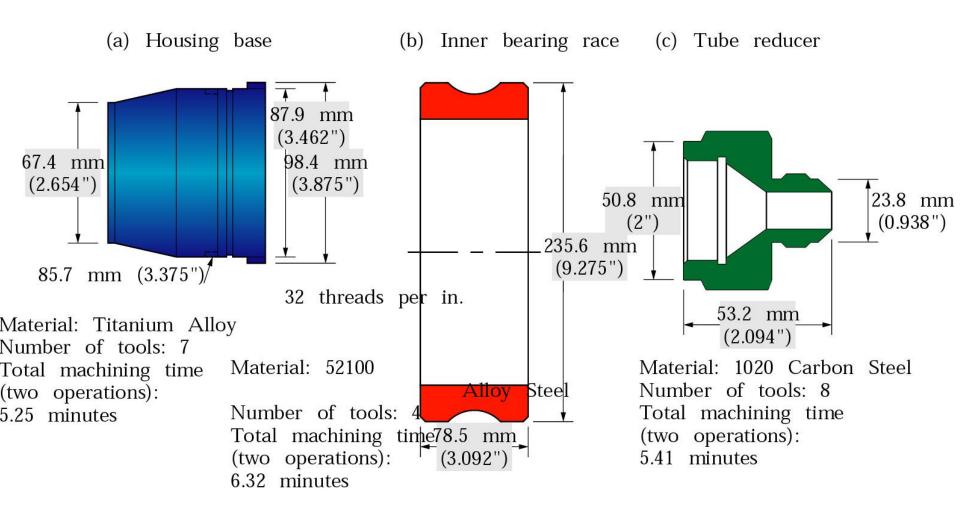
(i) Drilling

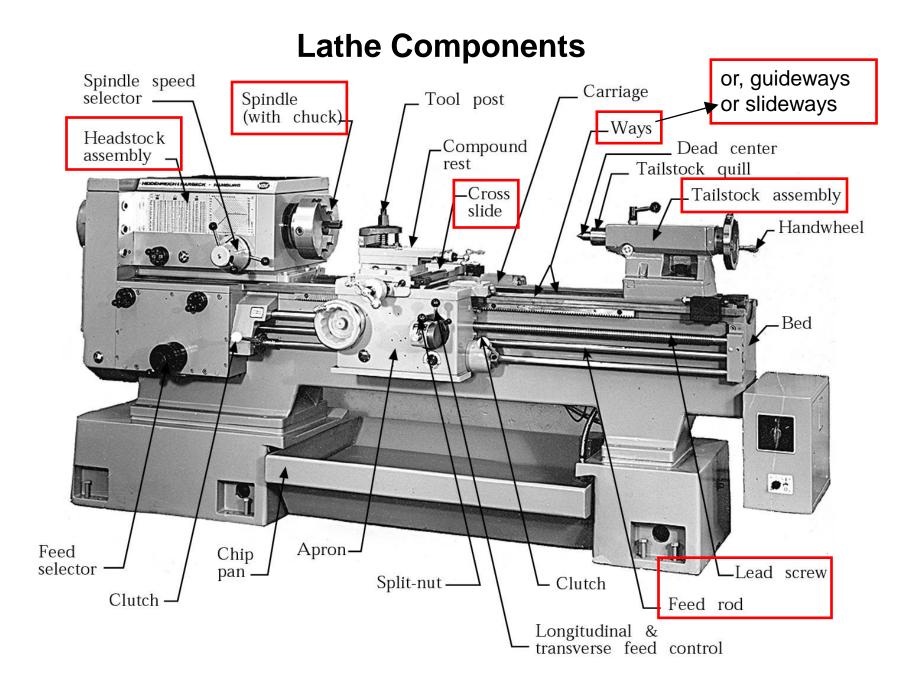


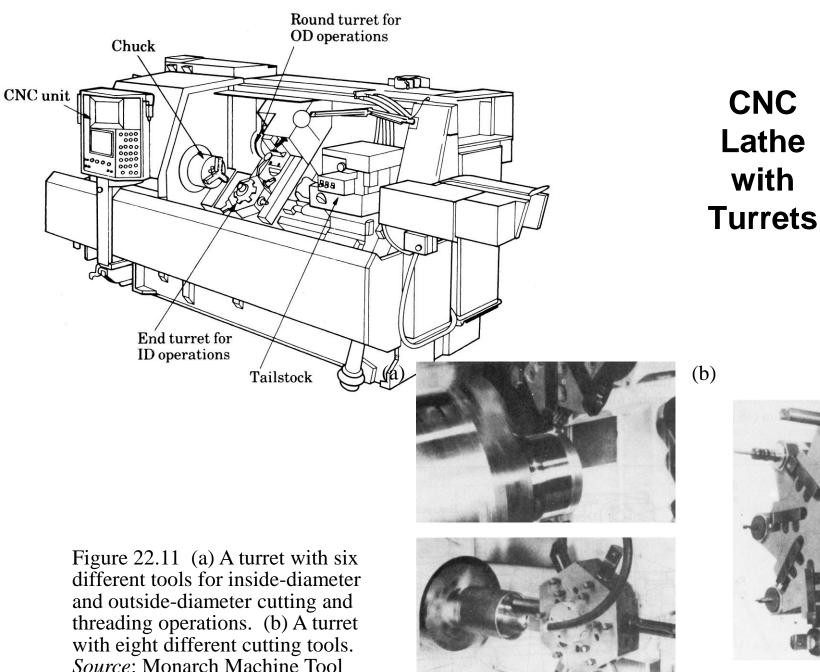
(l) Knurling



Workpieces Made by Turning

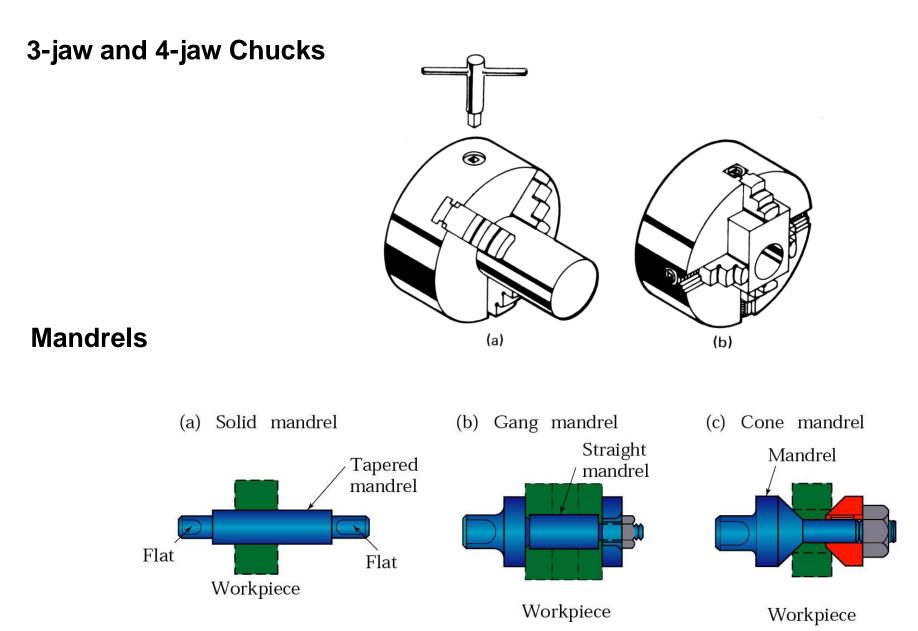




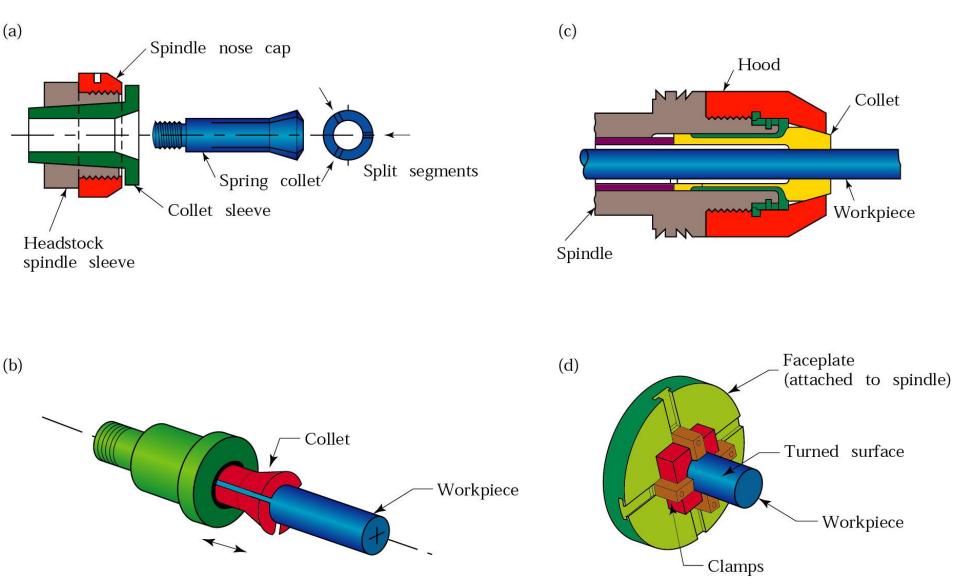


Company.

Workholding Devices



Workholding Devices – Collets and Face Plates



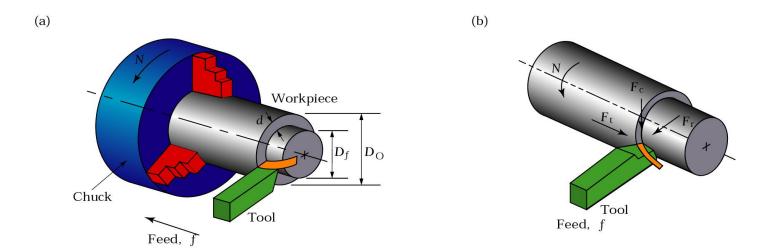
Turning Formulae

 Material removal rate (MRR) is the volume of material removed per unit time (mm³/min or in³/min)

$$MRR = \pi D_{avg} df N = \pi \left(\frac{D_o + D_f}{2}\right) df N \qquad t = \frac{l}{fN}$$

Cutting time (t) not including tool approach and retraction.

Cutting speed,
$$V = \pi D_0 N$$



Power for Machining

- Power required at cutting tool = MRR
 * Specific cutting power
- Power at motor = power at cutting tool/ mechanical efficiency of drive system

TABLE 20.1			
APPROXIMATE ENERGY REC	UIREMENTS IN CUTTING		
OPERATIONS (at drive motor, corrected for 80% efficiency;			
multiply by 1.25 for dull tools).			

Material	Specific Energy		
	W · s/mm ³	hp min/in. ³	
Aluminum alloys	0.4-1.1	0.15-0.4	
Cast irons	1.6-5.5	0.6-2.0	
Copper alloys	1.4-3.3	0.5-1.2	
High-temperature alloys	3.3-8.5	1.2-3.1	
Magnesium alloys	0.4-0.6	0.15-0.2	
Nickel alloys	4.9-6.8	1.8-2.5	
Refractory alloys	3.8-9.6	1.1-3.5	
Stainless steels	3.0-5.2	1.1–1.9	
Steels	2.7-9.3	1.0-3.4	
Titanium alloys	3.0-4.1	1.1–1.5	

Turning Parameters

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
 - $= \pi \log_{o} N$ (for maximum speed)
 - $= \pi D_{\text{avg}} N$ (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.

$$= (D_{\rm o} + D_f)/2$$

$$d = \text{Depth of cut, mm or in.}$$

= (
$$D_{\rm o}$$
 - $D_{\rm f}$) /2

t =Cutting time, s or min

$$= l/f N_{2}$$

$$MRR = mm^{3}/min \text{ or in}^{3}/min$$

$$=\pi D_{\mathrm{avg}} df N$$

$$= (F_{\rm c})(D_{\rm avg}/2)$$
Power = kW or hp

Power =
$$\kappa W$$
 or np

= (Torque) (ω , where $\omega = 2\pi$ radians/min

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

Turning Considerations

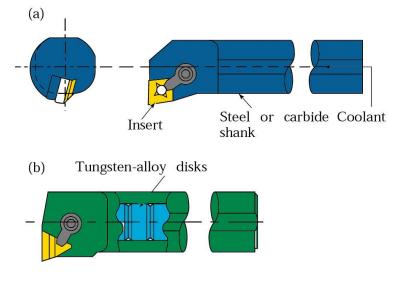
- Design parts for ease of fixturing and clamping (avoid thin, slender parts)
- Specify wide tolerances and surface finishes if possible
- Avoid sharp corners, tapers, and major dimensional variations
- Use near-net-shape forming to reduce machining cycle times
- Design features that only need standard cutting tools, inserts, and toolholders
- Select materials with good machinability
- Provide good support and stiffness in the turning operation
- Adjust parameters if chatter occurs

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.

TABLE 22.9

Boring

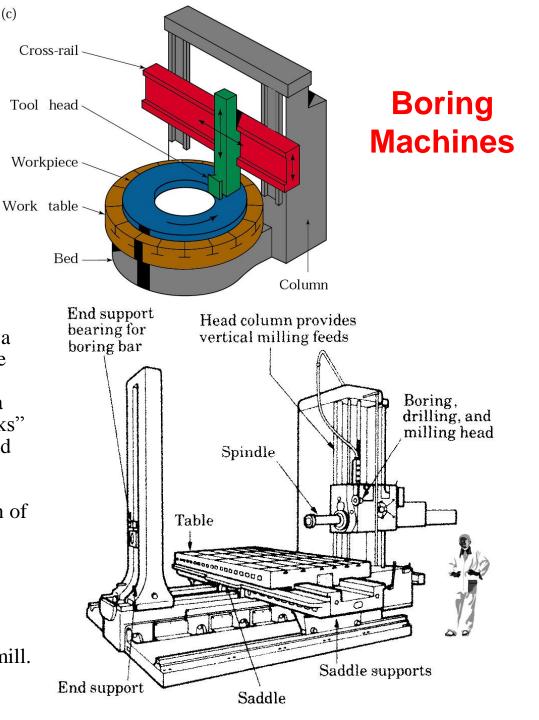
- Machining operation performed on the inside of a hollow workpiece of in a hole made previously by drilling or other processes
- Deflection of the boring bar can cause dimensional inaccuracy
- High stiffness of the boring bar minimizes deflection, vibration, and chatter (such as tungsten carbide material or built-in damping devices)
- Design considerations:
 - Use through holes instead of blind holes if possible
 - The greater the length-to-bore diameter ratio, the more difficult it is to hold dimensions because of deflections of the boring bar due to cutting forces
 - Interrupted internal surfaces should be avoided



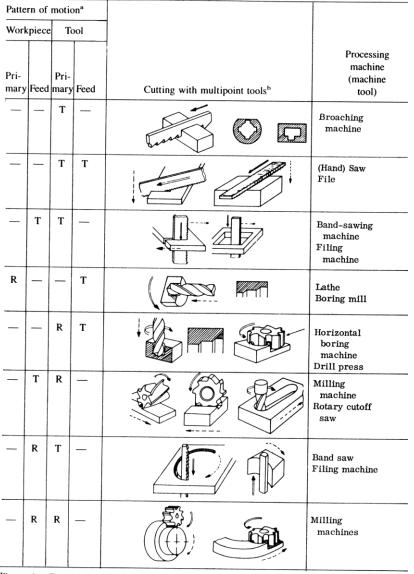
(c)

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.

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



Multi-Point Machining Operations

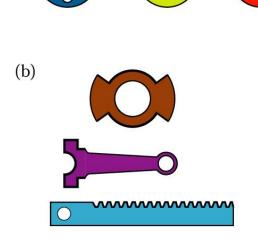


^aR, rotation; T, translation. ^b \Rightarrow , cutting motion; -->, feeding motion; \rightarrow , adjustment motion.

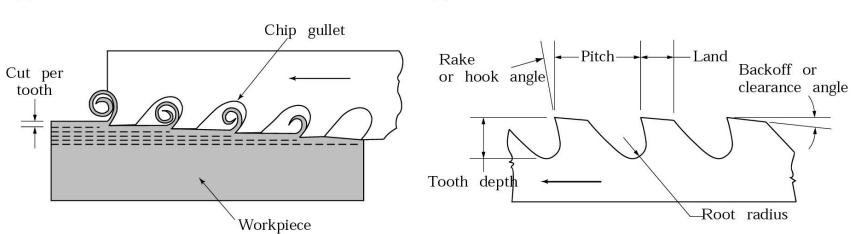
Broaching

- In broaching, multiple teeth machine internal or external surfaces such as holes, keyways, gear teeth, or flat surfaces
- Total depth of cut per stroke is the sum of the per tooth depths, up to 38 mm (1.5 in)
- Expensive, but good productivity, surface finish, and dimensional accuracy

(a)



• Broach tool is pushed or pulled (preferred) through the workpiece



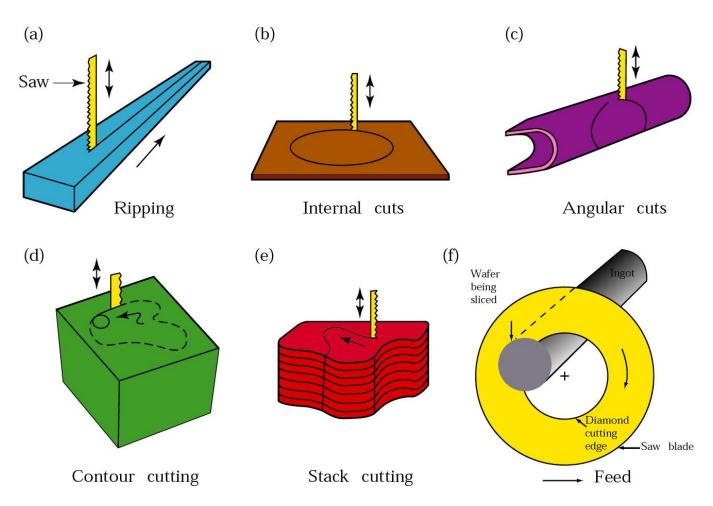
(a) Typical parts made by internal broaching. (b) Parts made by surface broaching. Heavy lines indicate broached surfaces. *Source*: General Broach and Engineering Company.

(b)

(a)

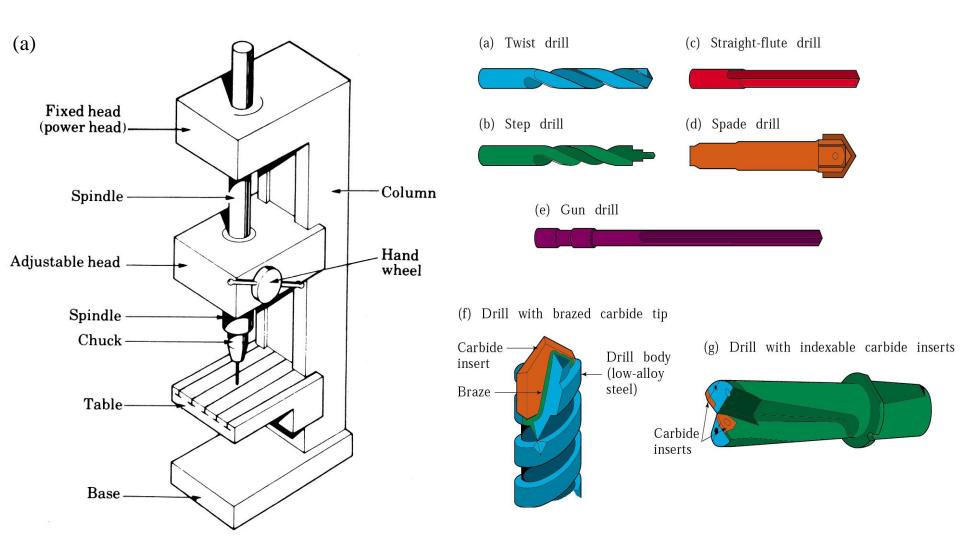
Sawing

- The width of cut, or kerf, in sawing is narrow so less material is wasted
- At least 2 or 3 teeth should always be engaged in the workpiece to avoid snagging
- Hacksaws, Circular saws, Band saws, etc.



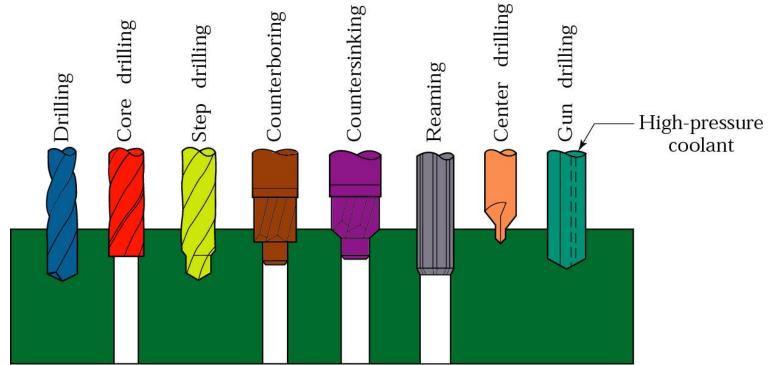
Examples of various sawing operations. Source: DoALL Co.

Drill Presses and Drill Bits



Drilling

- Creating a hole in a workpiece by mounting a drill bit on the tailstock
- The most accurate holes are produced by centering, drilling, boring, and them reaming, as well as possibly honing or grinding to improve internal surface and deburring the tool exit surface of through holes
- Holemaking is a major cost of components such as engines
- Gang drilling produces multiple holes at once



Drilling Parameters

- Drills can have high length-to-diameter ratios allowing them to drill deep holes
- Holes drilled on a lathe are not always concentric due to drift
- Long drill bits can deflect or break due to excessive thrust forces
- Chips must be removed from within the hole being drilled, and coolant may need to be delivered into the hole

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

Max. cutting speed = πDN

 $Torque = \frac{Power}{Rotational\,Speed}$

Cutting Time = hole depth/ fN

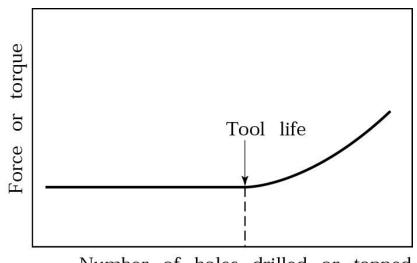
Drilling Considerations

- Designs should allow holes to be drilled on flat surfaces perpendicular to the drill motion to avoid deflection
- Avoid interrupted hole surfaces
- Use standard drill-point angles for hole bottoms if possible
- Through holes are preferred over blind holes
- If holes are large, workpiece should have a preexisting hole from forming or casting
- Design parts to minimize fixturing and repositioning
- It may be difficult to ream blind or intersecting holes due to the possibility of tool breakage so provide extra depth.
- Blind holes should be drilled deeper than subsequent reaming or tapping operations

Troubleshooting Drilling Processes

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.

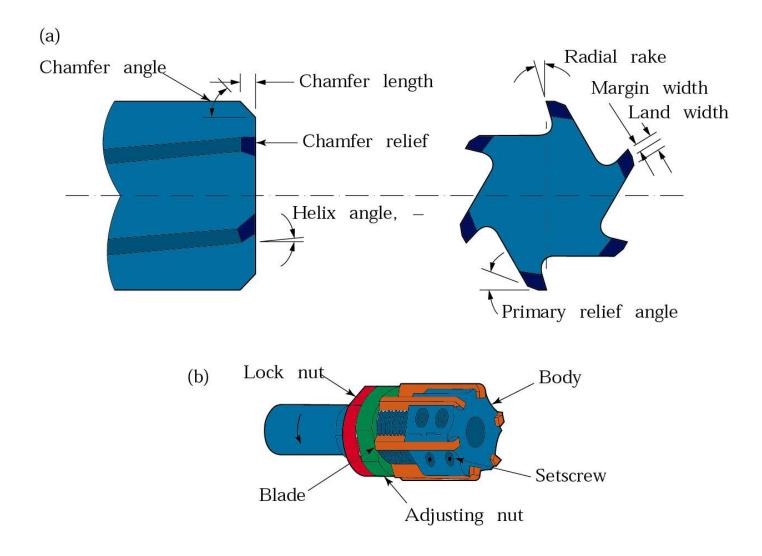
 TABLE 22.12
 General Troubleshooting Guide for Drilling Operations



Number of holes drilled or tapped

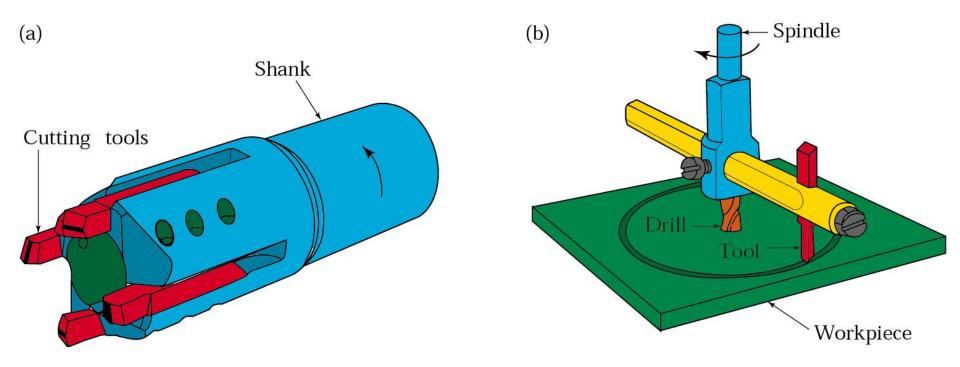
Figure 22.27 The determination of drill life by monitoring the rise in force or torque as a function of the number of holes drilled. This test is also used for determining tap life.

Reamers



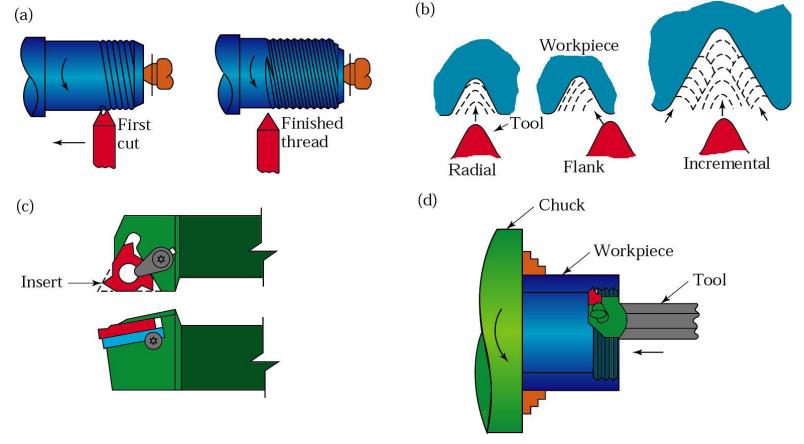
Trepanning

- Trepanning produces a hole without reducing all of the removed material into chips
- Trepanning can produce disks up to 150 mm (6 inches) diameter from flat sheet or plates
- Trepanning can also be used to produce grooves for O-rings



Cutting Screw Threads

- Threads may be right-handed or left-handed, straight or tapered
- Threads can be produced by forming (most), casting (with dimensional inaccuracy), or machining (thread cutting)
- Standard Nomenclature is in Figures 22.16 and 22.17

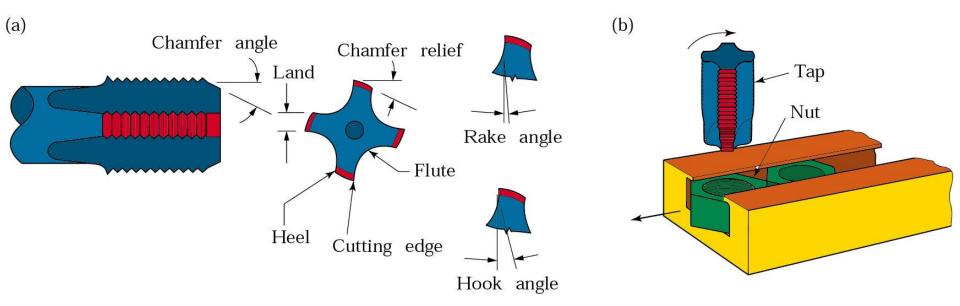


Screw Thread Considerations

- Designs should allow for termination of thread before they reach a shoulder or the bottom of a blind hole
- Eliminate shallow, blind tapped holes
- Use chamfers to minimize finlike threads with burrs
- Do not interrupt threads with slots, holes, or other discontinuities
- Use standard thread tooling and inserts when possible
- Thin-walled parts should have sufficient thickness and strength to resist clamping and cutting forces. A good rule of thumb is that the minimum engagement length of a fastener should be 1.5 times the diameter
- Design parts so that all cutting operations can be completed in one setup

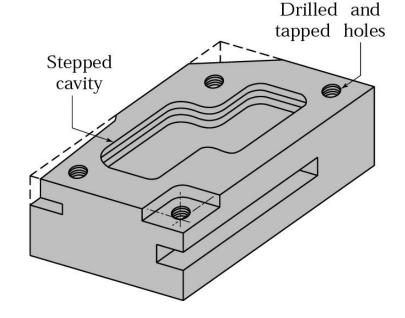
Tapping

- Produces internal screw threads in previously drilled or reamed holes
- A tap has two (most commonly), three, or four cutting teeth (flutes)
- Taps are usually made of carbon steel (light duty) or high-speed steels (heavy production)
- 30-40% of machining operations in automotive manufacturing involves tapping holes
- Chip removal and coolant delivery are important issues
- Drilling and tapping with a single specialized tool is called drapping



Milling

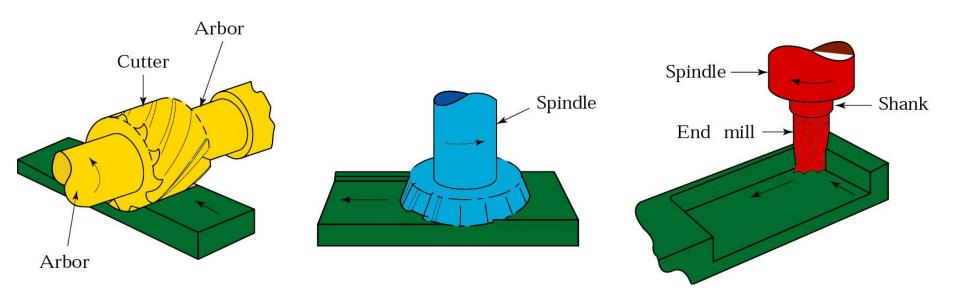
- A rotating, multi-tooth cutter removes material and produces multiple chips in a single revolution
- CNC machining centers can perform multiple operations in a single setup

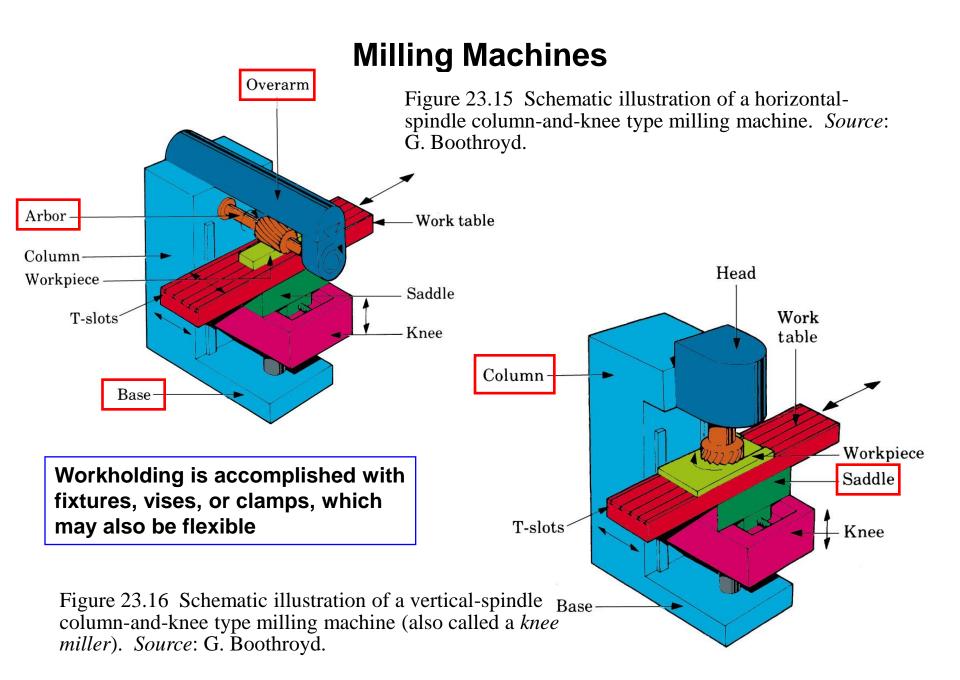


(a) Slab milling

(b) Face milling

(c) End milling





Horizontal Milling Machines

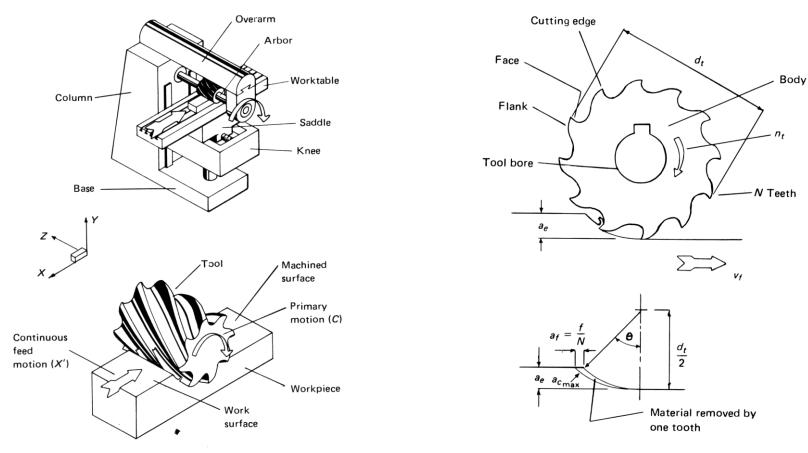
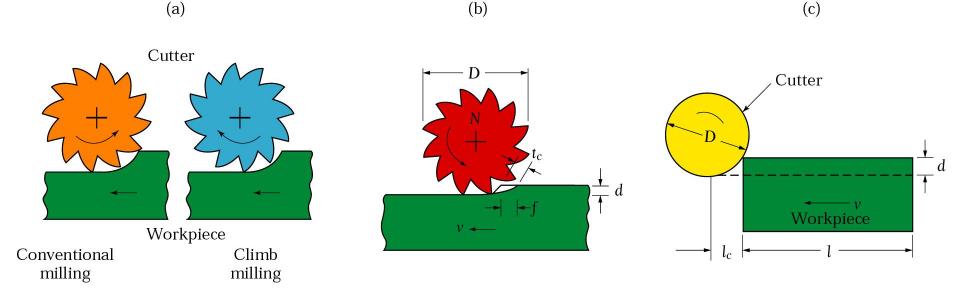


Figure 1.27 Slab milling on a knee-type horizontal-milling machine.

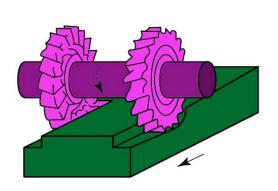
Primary motion is rotation of the cutting tool Feed motion is translation of the workpiece

Slab Milling

- In slab, or peripheral, milling the axis of cutter rotation in parallel to the workpiece surface being machined
- Cutters may have straight or helical teeth, resulting in orthogonal or oblique cutting action, respectively (helical teeth lower tooth load, tool forces, chatter)
- In conventional (up milling) the maximum chip thickness is at the end of the cut, so tooth engagement does not depend upon workpiece surface quality or scaling, but clamping forces must be higher and chatter is harder to avoid
- In climb (down milling) the cut starts with the maximum chip thickness but high impact forces can be a problem

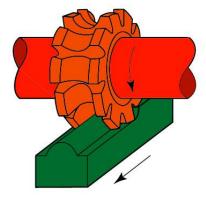


Horizontal Milling Tools

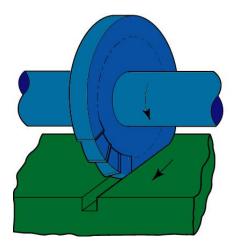


(a) Straddle milling

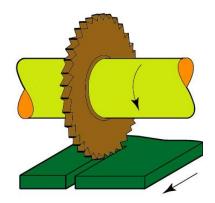
(b) Form milling



(c) Slotting



(d) Slitting



Milling Parameters

TABLE 23.1

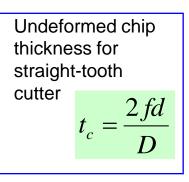
- N = Rotational speed of the milling cutter, rpm
 - f = Feed, mm/tooth or in./tooth
- D =Cutter diameter, mm or in.
- n = Number of teeth on cutter
- v = Linear speed of the workpiece or feed rate, mm/min or in./min
- = Surface speed of cutter, m/min or ft/min $= D N \pi$
- f = Feed per tooth, mm/tooth or in/tooth =v / N n
- l = Length of cut, mm or in.
- t = Cutting time, s or min $= (l+l_c)/v, where l_c = extent of the cutter's first contact with workpiece$
- MRR = mm^{3}/min or in.³/min =w d v, where w is the width of cut

Torque = N-m or lb-ft
$$(F_c) (D/2)$$

Power = kW or hp

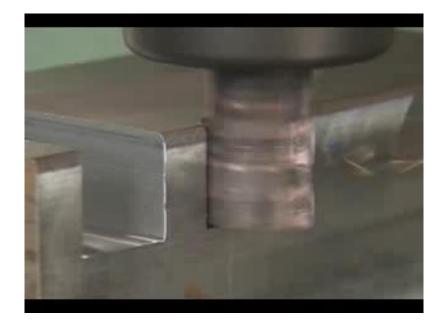
= (Torque) (ω), where $\omega = 2\pi N$ radians/min

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



End and Face Milling

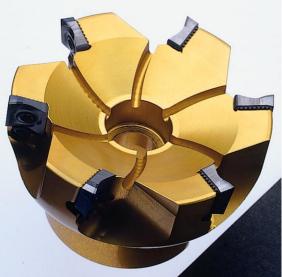






Face Milling Cutters



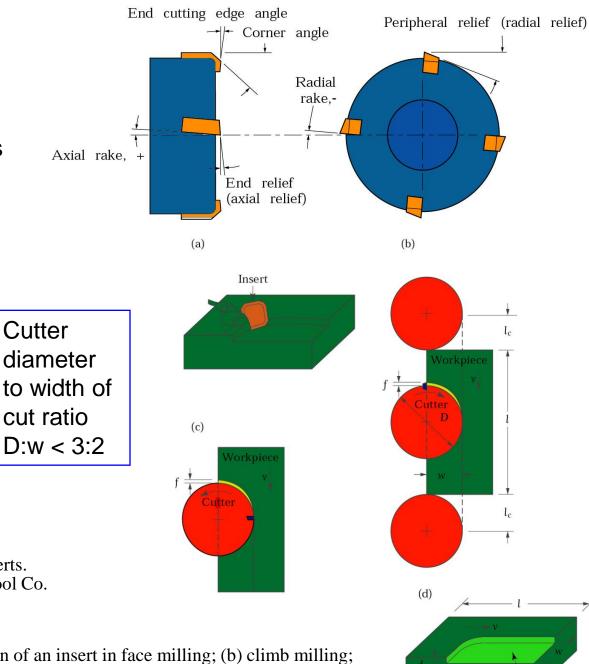






Face Milling

In face milling the axis • of cutter rotation is perpendicular to the workpiece surface





A face-milling cutter with indexable inserts. Source: Courtesy of Ingersoll Cutting Tool Co.

Machined surface

Face-milling operation showing (a) action of an insert in face milling; (b) climb milling; (c) conventional milling; (d) dimensions in face milling. The width of cut, w, is not necessarily the same as the cutter radius. Source: Ingersoll Cutting Tool Co.

Cutter

cut ratio

Face Milling Feed Marks and Chatter

(d)

Cutter

- Face milling leaves feed marks on the surface of the workpiece
- Feed marks can lead to chatter in subsequent cuts
- Chatter is self-excited vibration
- Due to surface variations, cutting forces vary and the tool vibrates in a regenerative manner

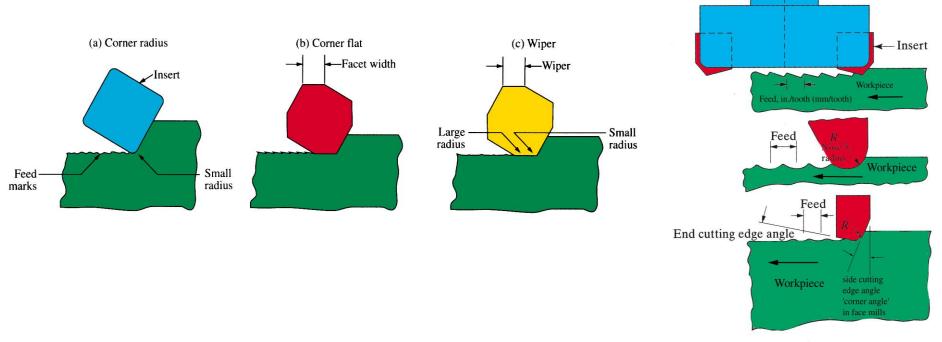


Figure 23.7 Schematic illustration of the effect of insert shape on feed marks on a face-milled surface: (a) small corner radius, (b) corner flat on insert, and (c) wiper, consisting of a small radius followed by a large radius which leaves smoother feed marks. *Source*: Kennametal Inc. (d) Feed marks due to various insert shapes.

Milling Considerations

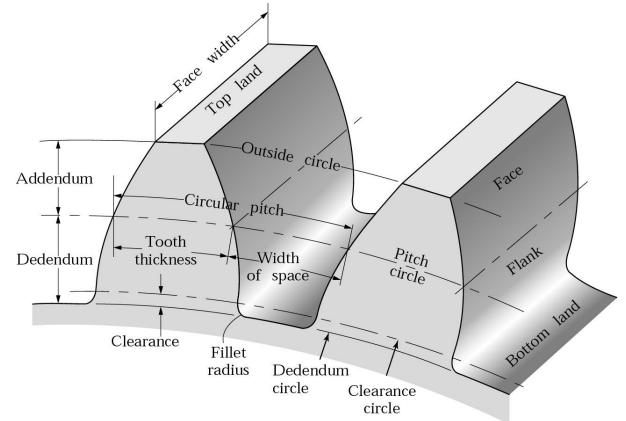
- Use standard milling cutters
- Use chamfers instead of radii
- Avoid internal cavities and pockets because cutters have a finite edge radius
- Stiff workpieces minimize deflections from clamping or cutting forces
- Mount cutters as close to the spindle base as possible to reduce tool deflections and avoid chatter and vibration
- Use rigid tool holders and fixturing
- In case of chatter, change tool shape and process conditions, or use cutters with fewer teeth or random spacing

Troubleshooting Milling Operations

TABLE 23.5	
Problem	Probable causes
Tool breakage	Tool material lacks toughness; improper tool angles; cutting parameters too high.
Tool wear excessive	Cutting parameters too high; improper tool material; improper tool angles; improper cutting fluid.
Rough surface finish	Feed too high; spindle speed too low; too few teeth on cutter; tool chipped or worn; built-up edge; vibration and chatter.
Tolerances too broad	Lack of spindle stiffness; excessive temperature rise; dull tool; chips clogging cutter.
Workpiece surface burnished	Dull tool; depth of cut too low; radial relief angle too small.
Back striking	Dull cutting tools; cutter spindle tilt; negative tool angles.
Chatter marks	Insufficient stiffness of system; external vibrations; feed, depth, and width of cut too large.
Burr formation	Dull cutting edges or too much honing; incorrect angle of entry or exit; feed and depth of cut too high; incorrect insert geometry.
Breakout	Lead angle too low; incorrect cutting edge geometry; incorrect angle of entry or exit; feed and depth of cut too high.

Gear Manufacturing

- Gears can be manufactured by casting, forging, extrusion, drawing, thread rolling, and powder metallurgy
- Blanking sheet metal can be used to make thin gears for watches or clocks
- Plastic gears can be made by injection molding or casting
- Machining gears is accomplished by gear generating or form cutting



Nomenclature for an involute spur gear.

Gear Form Cutting and Generating

- Form cutting is accomplished with a shaped milling cutter or broach
- Generating is done with a hob, or pinion- or rack-shaped cutters

