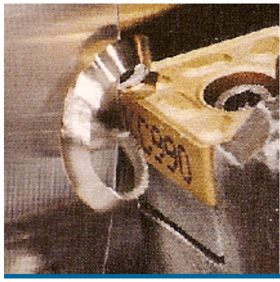
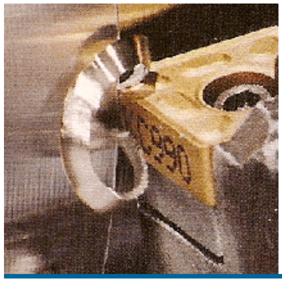


Failure of Cutting Tools and Tool Wear

- *Fracture failure*
  - Cutting force becomes excessive, leading to brittle fracture
- *Temperature failure*
  - Cutting temperature is too high for the tool material
- *Gradual wear*
  - Gradual wearing of the cutting tool



Preferred Mode of Tool Failure: Gradual Wear

- Fracture and temperature failures are premature failures
- Gradual wear is preferred because it leads to the longest possible use of the tool
- Gradual wear occurs at two locations on a tool:
    - *Crater wear* – occurs on top rake face
    - *Flank wear* – occurs on flank (side of tool)

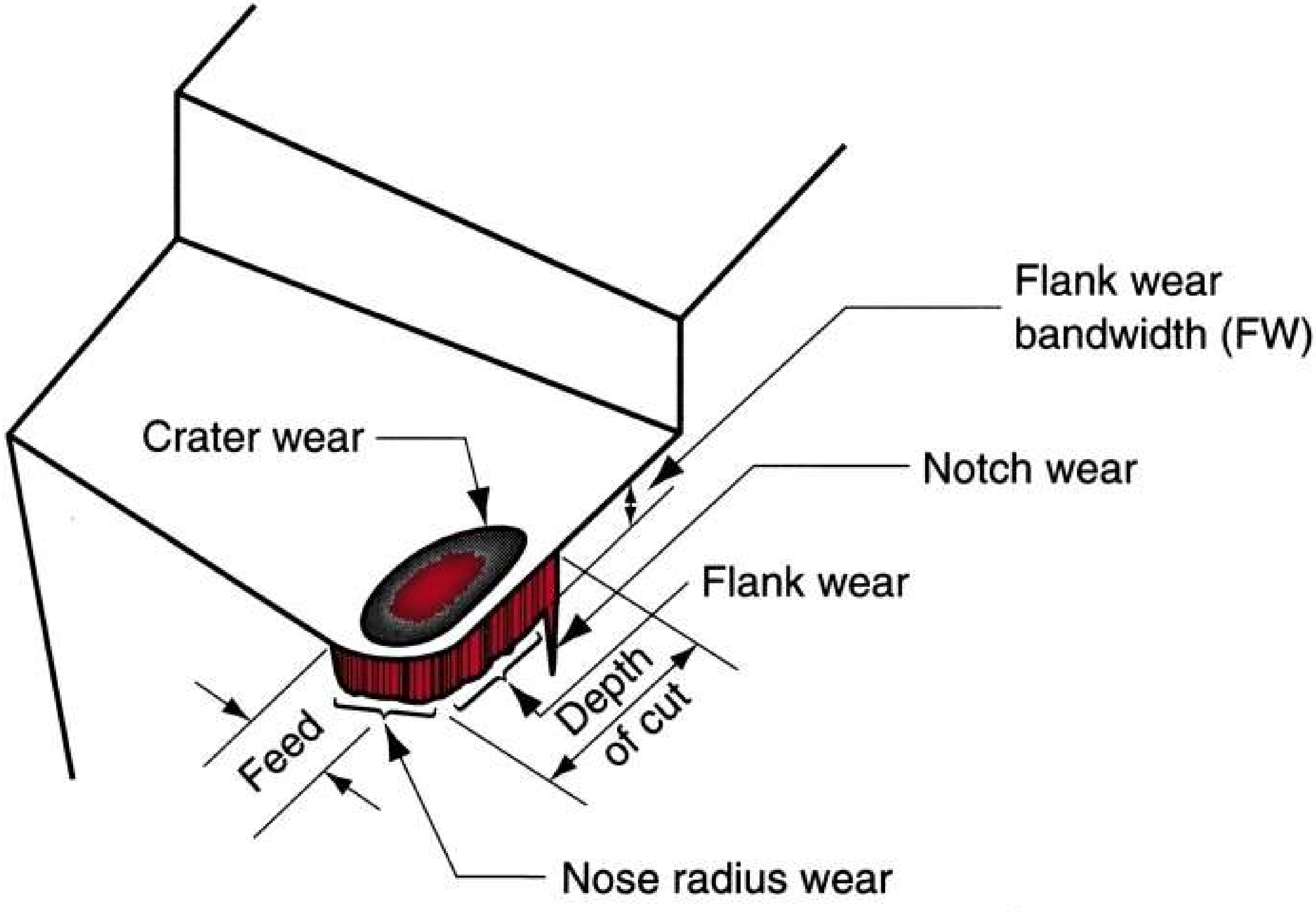
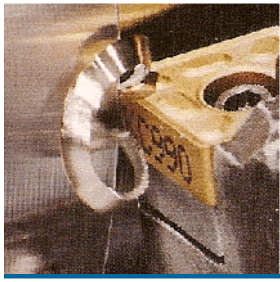


Figure - Diagram of worn cutting tool, showing the principal locations and types of wear that occur



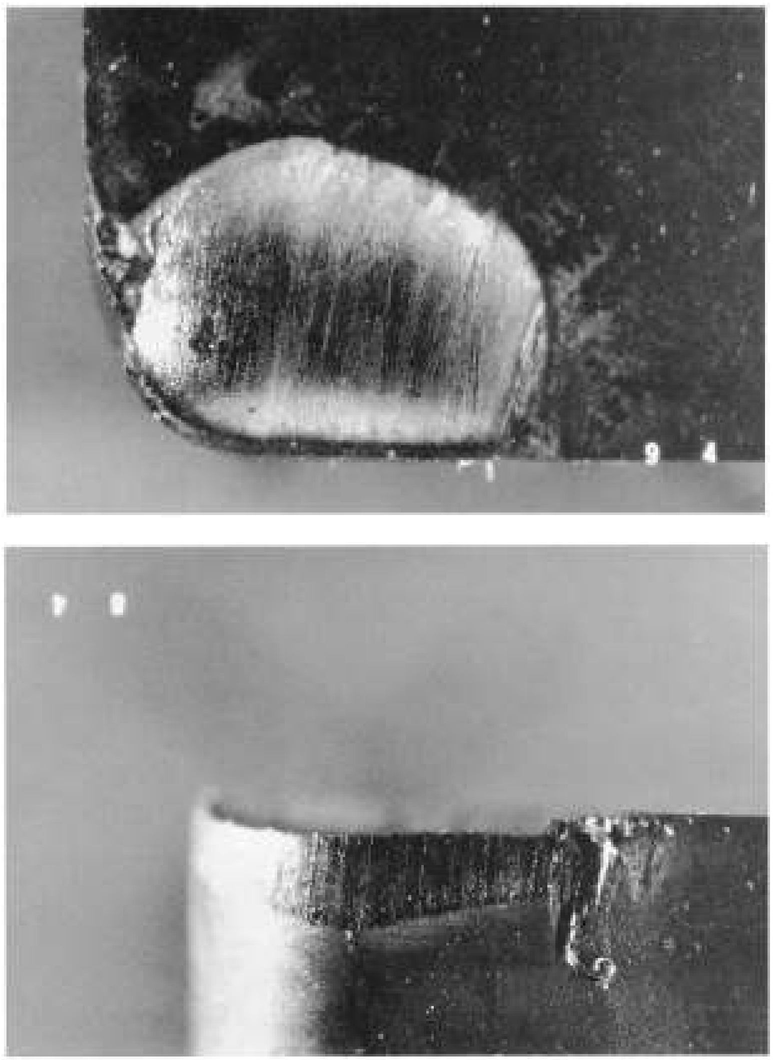


Figure -

(a) Crater wear, and

(b) flank wear on a cemented carbide tool, as seen through a toolmaker's microscope

(Source: Manufacturing Technology Laboratory, Lehigh University, photo by J. C. Keefe)

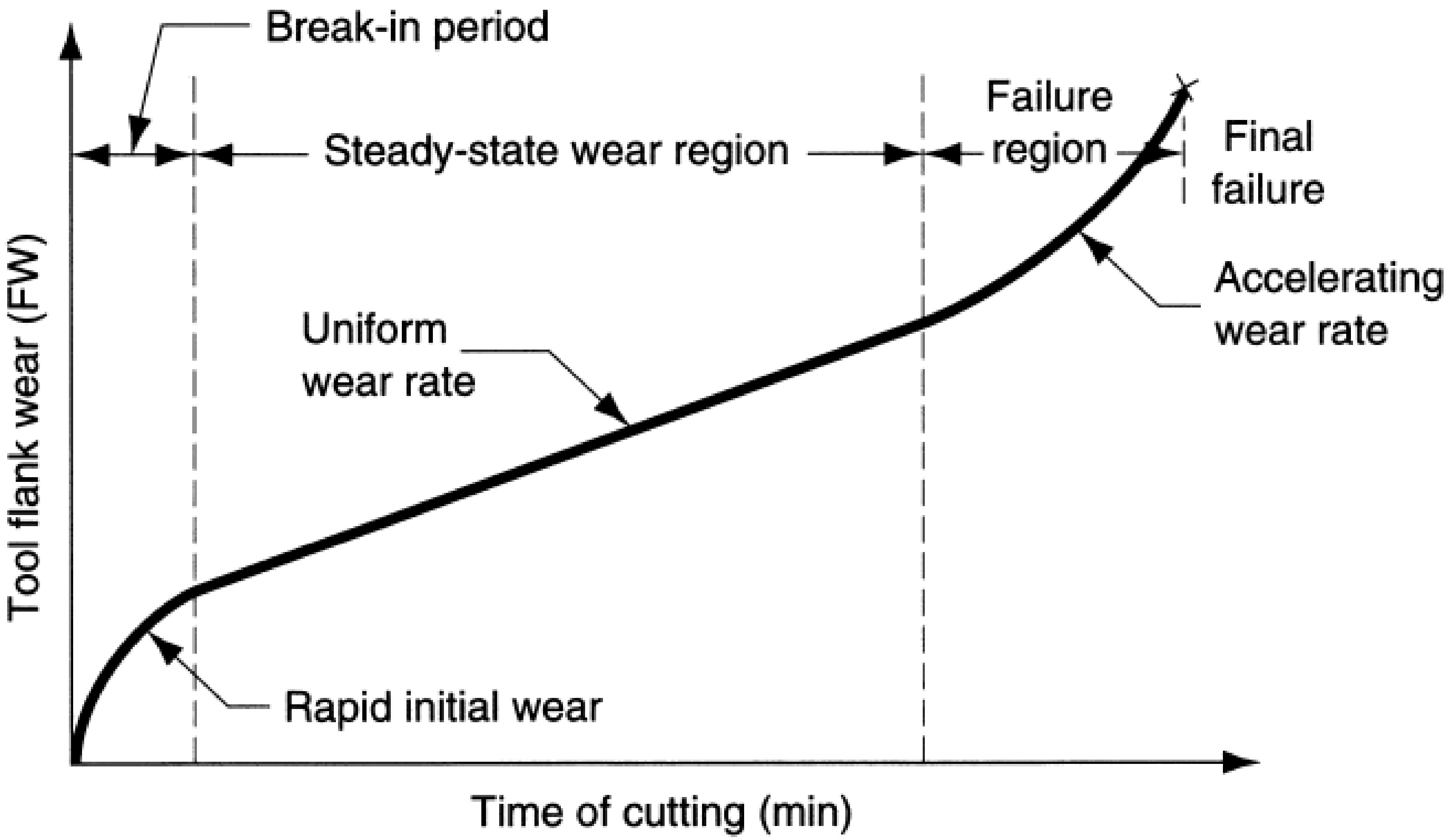


Figure - Tool wear as a function of cutting time

Flank wear (FW) is used here as the measure of tool wear

Crater wear follows a similar growth curve

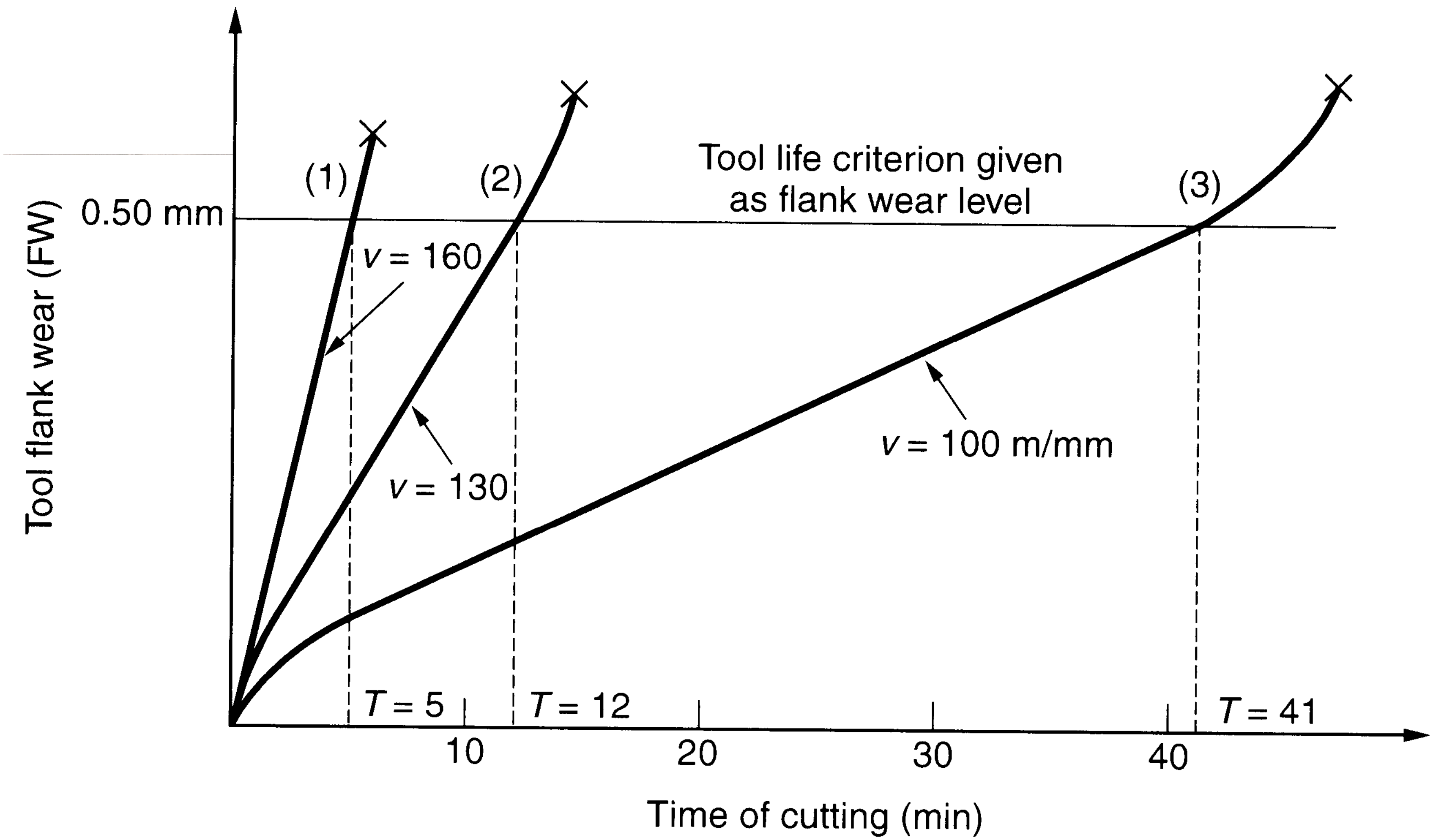
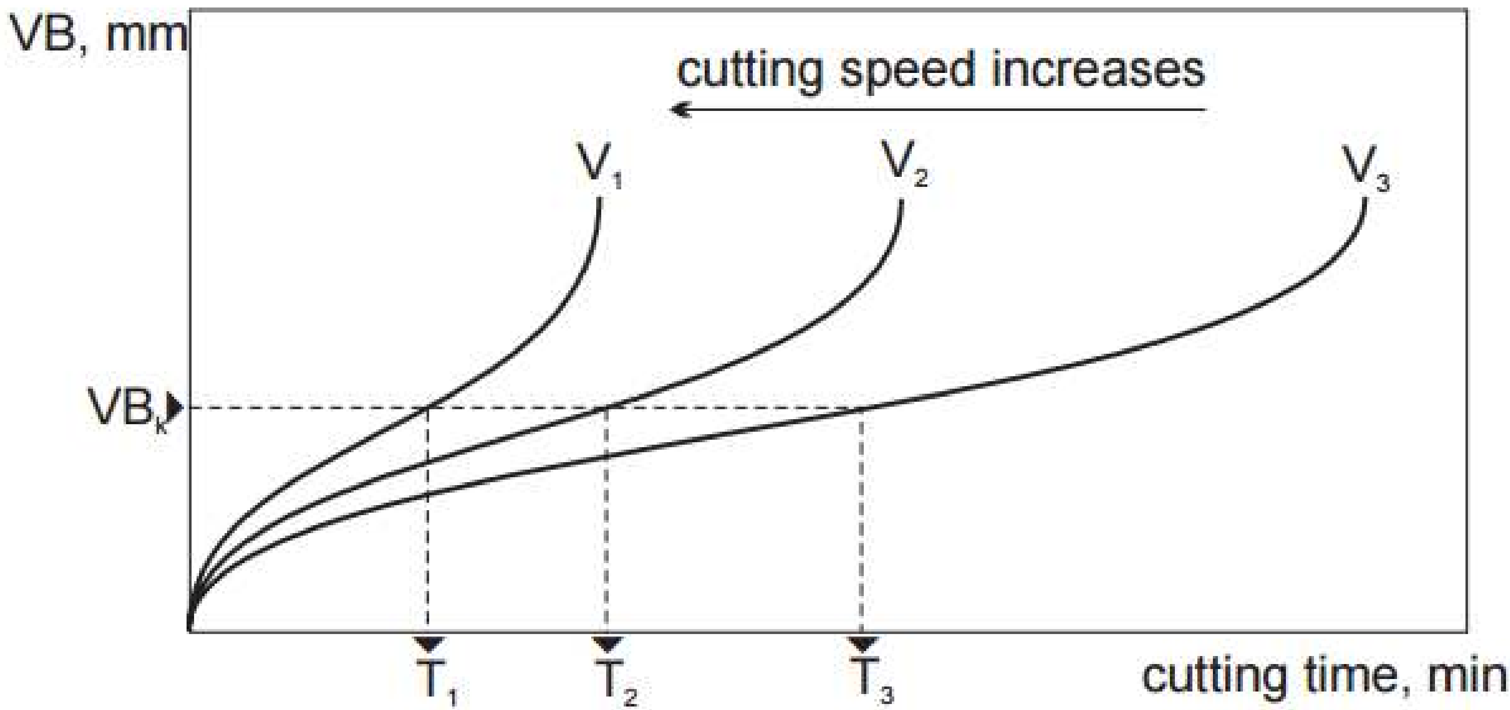
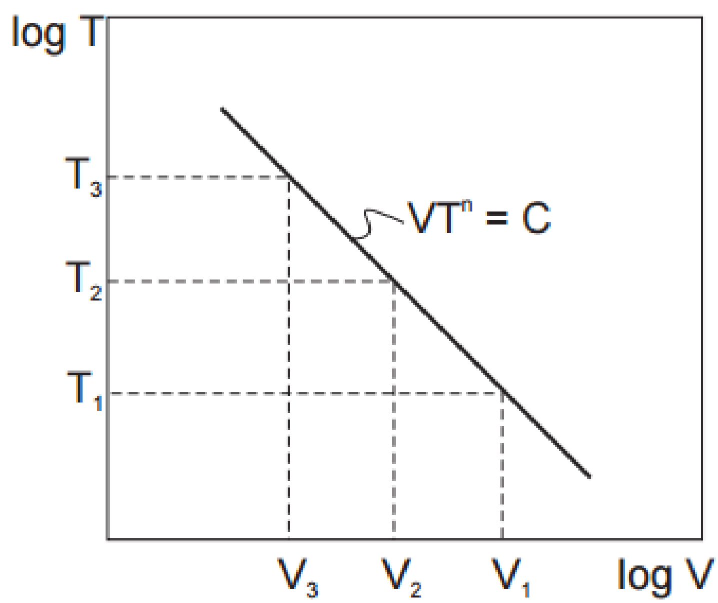
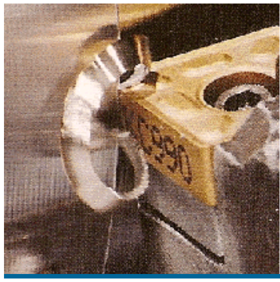


Figure - Effect of cutting speed on tool flank wear (FW) for three cutting speeds, using a tool life criterion of 0.50 mm flank wear


VB, mm
cutting speed increases
V1
V2
V3
VBk
T1
T2
T3
cutting time, min


log T
T3
T2
T1
VT^n = C
V3
V2
V1
log V



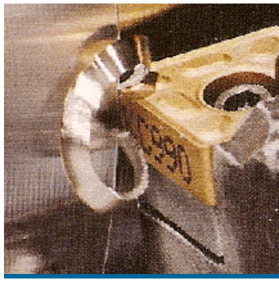
Taylor Tool Life Equation

This relationship is credited to F. W. Taylor (~1900)

$$vT^n = C$$

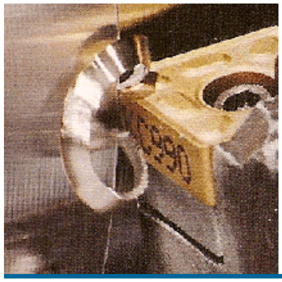
where v = cutting speed; T = tool life; and n and C are parameters that depend on feed, depth of cut, work material, tooling material, and the tool life criterion used

- n is the slope of the plot
- C is the intercept on the speed axis



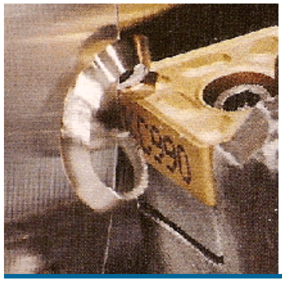
Typical Values of *n* and *C* in Taylor Tool Life Equation

Tool material	n	C (m/min)	C (ft/min)
High speed steel:			
Non-steel work	0.125	120	350
Steel work	0.125	70	200
Cemented carbide			
Non-steel work	0.25	900	2700
Steel work	0.25	500	1500
Ceramic			
Steel work	0.6	3000	10,000



Tool Life Criteria in Production

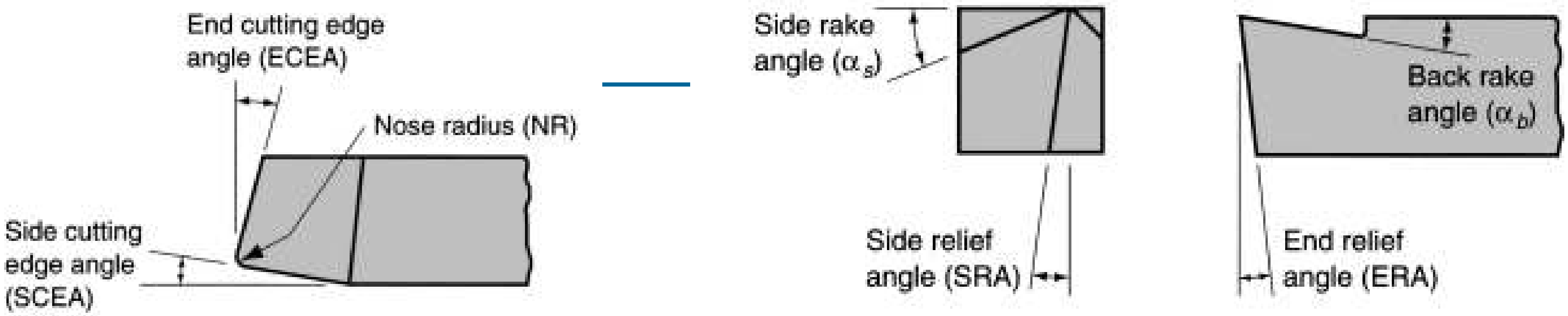
1. Complete failure of cutting edge
2. Visual inspection of flank wear (or crater wear) by the machine operator
3. Fingernail test across cutting edge
4. Changes in sound emitted from operation
5. Chips become ribbony, stringy, and difficult to dispose of
6. Degradation of surface finish
7. Increased power
8. Workpiece count
9. Cumulative cutting time



Variables Affecting Tool Life

- Cutting conditions.
- Tool geometry.
- Tool material.
- Work material.
- Cutting fluid.
- Vibration behavior of the machine-tool work system.
- Built-up edge.

Single-Point Tool Geometry

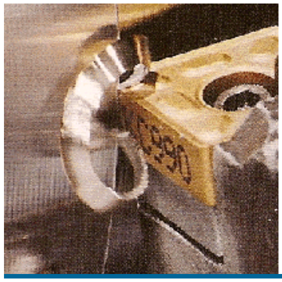


(a) (b) Tool signature: α_b, α_s, ERA, SRA, ECEA, SCEA, NR

Tool specifications (all six angles, and nose radius) :7-8-5-6-9-4-1mm

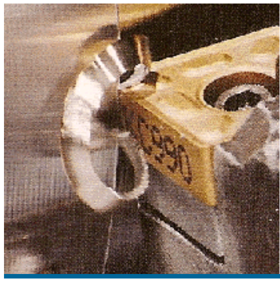
This specification indicates the following:

- Back rake angle (7^0)
- Side rake angle (8^0)
- End clearance (relief) angle (5^0)
- Side clearance (relief) angle (6^0)
- End cutting edge angle (9^0)
- Side cutting edge angle (4^0)
- Nose radius (1 mm)



Tool Geometry: Rake Angle

- Increasing the Rake Angle reduces the cutting force and the cutting temperature resulting in increased tool life.
- However, for large rake angle, tool edge is weakened resulting in increased wear due to chipping of the cutting edge.
- These conditions give an optimum rake angle which gives the maximum tool life.
- Higher is the strength of workpiece material, lower is the value of optimum rake angle.

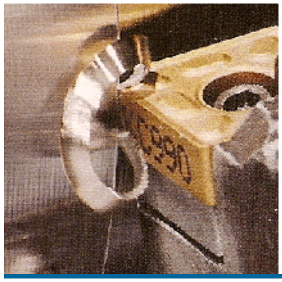


Tool Geometry: Flank Angle

- Increasing the Flank Angle reduces rubbing between tool and the workpiece and hence improves the tool life.
- However, too high a value of flank angle weakens the tool and reduces its life.
- Optimum value of flank angles is also affected by the feed rates. Higher is the feed rate, lower is the optimum value. The flank angle, therefore, should be low if higher feed values are to be used.

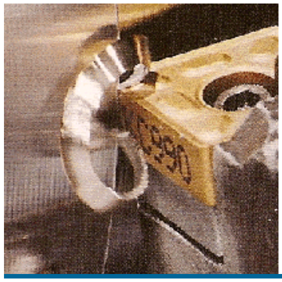
Why?

This is necessary for providing increased strength and better heat dissipation when the feed is increased.



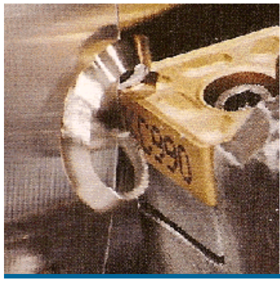
Selection of Cutting Conditions

- One of the tasks in process planning
- For each operation, decisions must be made about machine tool, cutting tool(s), and cutting conditions
- These decisions must give due consideration to workpart machinability, part geometry, surface finish, and so forth
- Cutting conditions: speed, feed, depth of cut, and cutting fluid



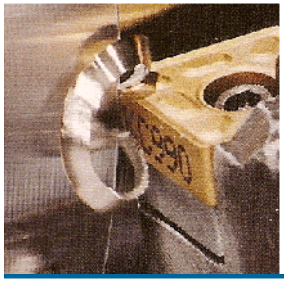
Selecting Depth of Cut

- Depth of cut is often predetermined by workpiece geometry and operation sequence
  - In roughing, depth is made as large as possible to maximize material removal rate, subject to limitations of horsepower, machine tool and setup rigidity, and strength of cutting tool
  - In finishing, depth is set to achieve final part dimensions



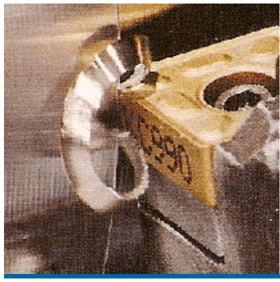
Determining Feed

- In general: *feed first*, *speed second*
- Determining feed rate depends on:
  - *Tooling* – harder tool materials require lower feeds
  - *Roughing or finishing* - Roughing means high feeds, finishing means low feeds
  - *Constraints on feed in roughing* - Limits imposed by cutting forces, setup rigidity, and sometimes horsepower
  - *Surface finish requirements in finishing* – select feed to produce desired finish



Optimizing Cutting Speed

- Select speed to achieve a balance between high metal removal rate and suitably long tool life
- Mathematical formulas are available to determine optimal speed
- Two alternative objectives in these formulas:
    1. Maximum production rate
    2. Minimum unit cost



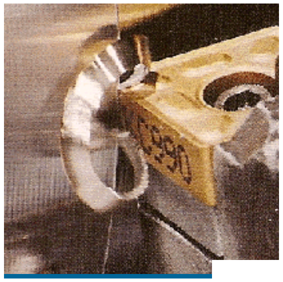
Maximum Production Rate

- Maximizing production rate = minimizing cutting time per unit
- In turning, total production cycle time for one part consists of:
  1. *Part handling time per part* = T_h
  2. *Machining time per part* = T_m
  3. *Tool change time per part* = T_t/n_p , where n_p = number of pieces cut in one tool life

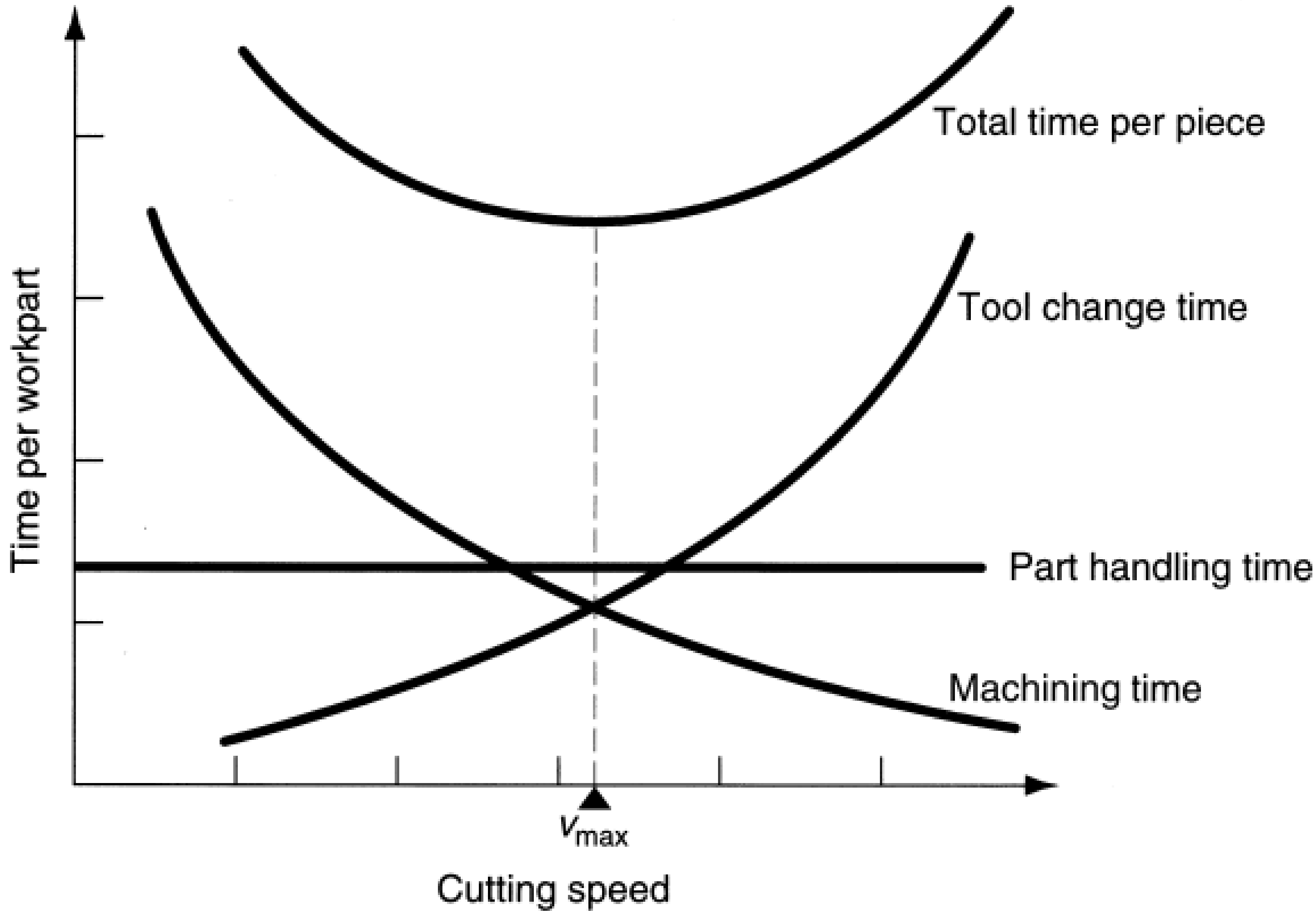
Total time per unit product for operation:

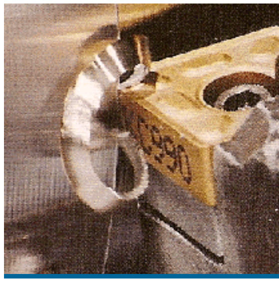
$$T_c = T_h + T_m + T_t/n_p$$

Cycle time T_c is a function of cutting speed



Cycle Time vs. Cutting Speed





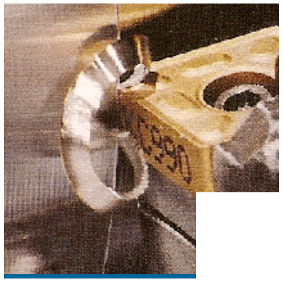
Minimizing Cost per Unit

- In turning, total production cycle cost for one part consists of:
  1. *Cost of part handling time* = $C_o T_h$, where C_o = cost rate for operator and machine
  2. *Cost of machining time* = $C_o T_m$
  3. *Cost of tool change time* = $C_o T_t / n_p$
  4. *Tooling cost* = C_t / n_p , where C_t = cost per cutting edge

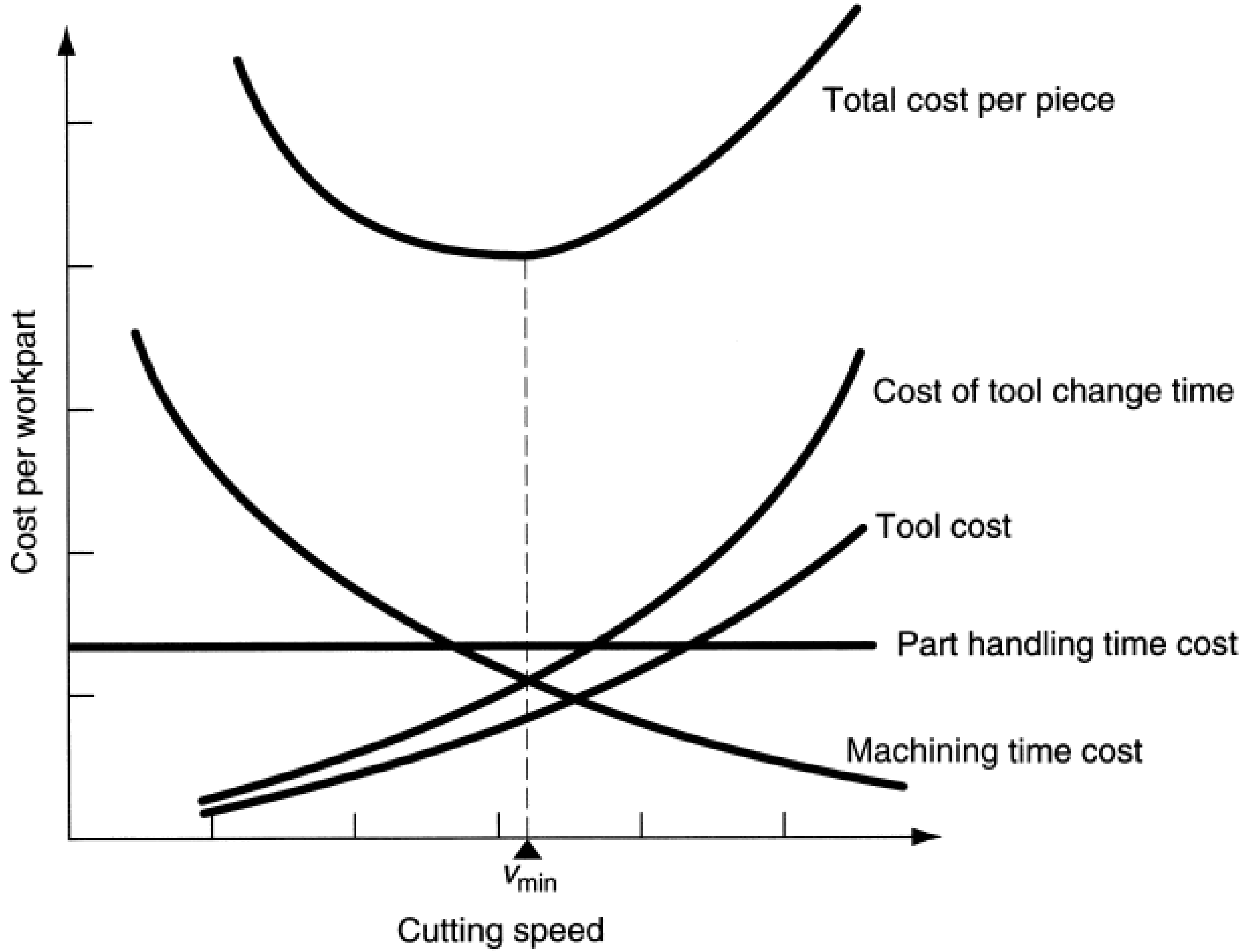
Total cost per unit product for operation:

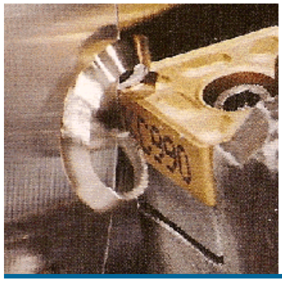
$$C_c = C_o T_h + C_o T_m + C_o T_t / n_p + C_t / n_p$$

Again, unit cost is a function of cutting speed, just as T_c is a function of v



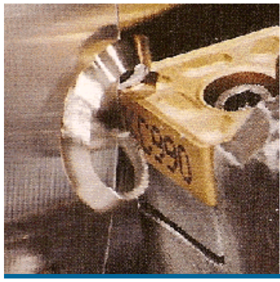
Unit Cost vs. Cutting Speed





Comments on Machining Economics

- As *C* and *n* increase in Taylor tool life equation, optimum cutting speed should be reduced
  - Cemented carbides and ceramic tools should be used at speeds significantly higher than for HSS
- As tool change time T_t and/or tooling cost C_t increase, cutting speed should be reduced
  - Tools should not be changed too often if either tool cost or tool change time is high
  - Disposable inserts have an advantage over regrindable tools because tool change time is lower



Classification of Machined Parts

1. *Rotational* - cylindrical or disk-like shape
2. *Nonrotational* (also called *prismatic*) - block-like or plate-like

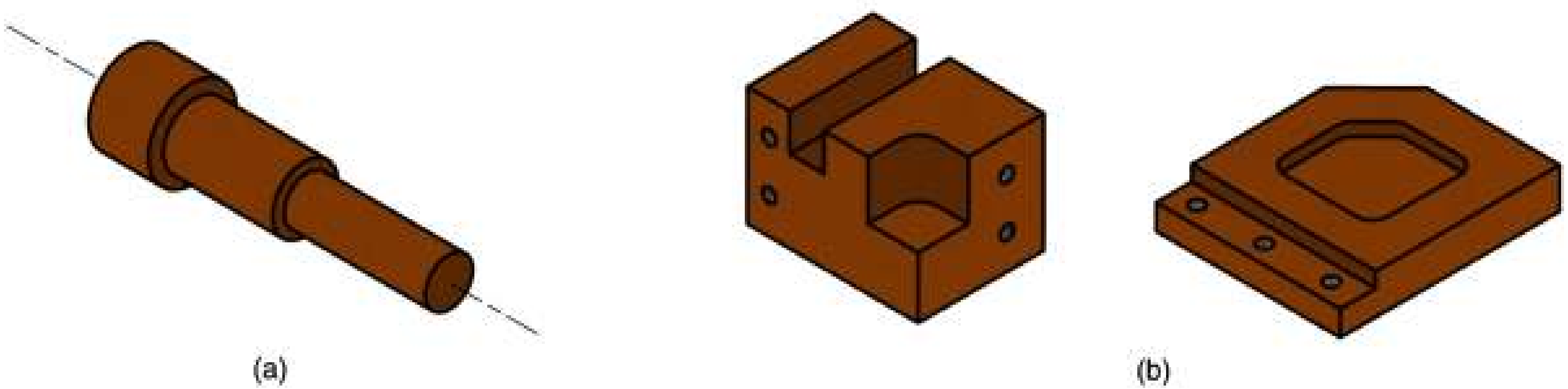
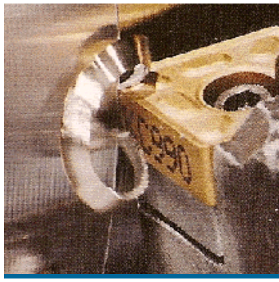


Figure - Machined parts are classified as: (a) rotational, or (b) nonrotational, shown here by block and flat parts



Machining Operations and Part Geometry

Each machining operation produces a characteristic part geometry due to two factors:

1. Relative motions between the tool and the workpart
   - Generating – part geometry is determined by the feed trajectory of the cutting tool
2. Shape of the cutting tool
   - Forming – part geometry is created by the shape of the cutting tool

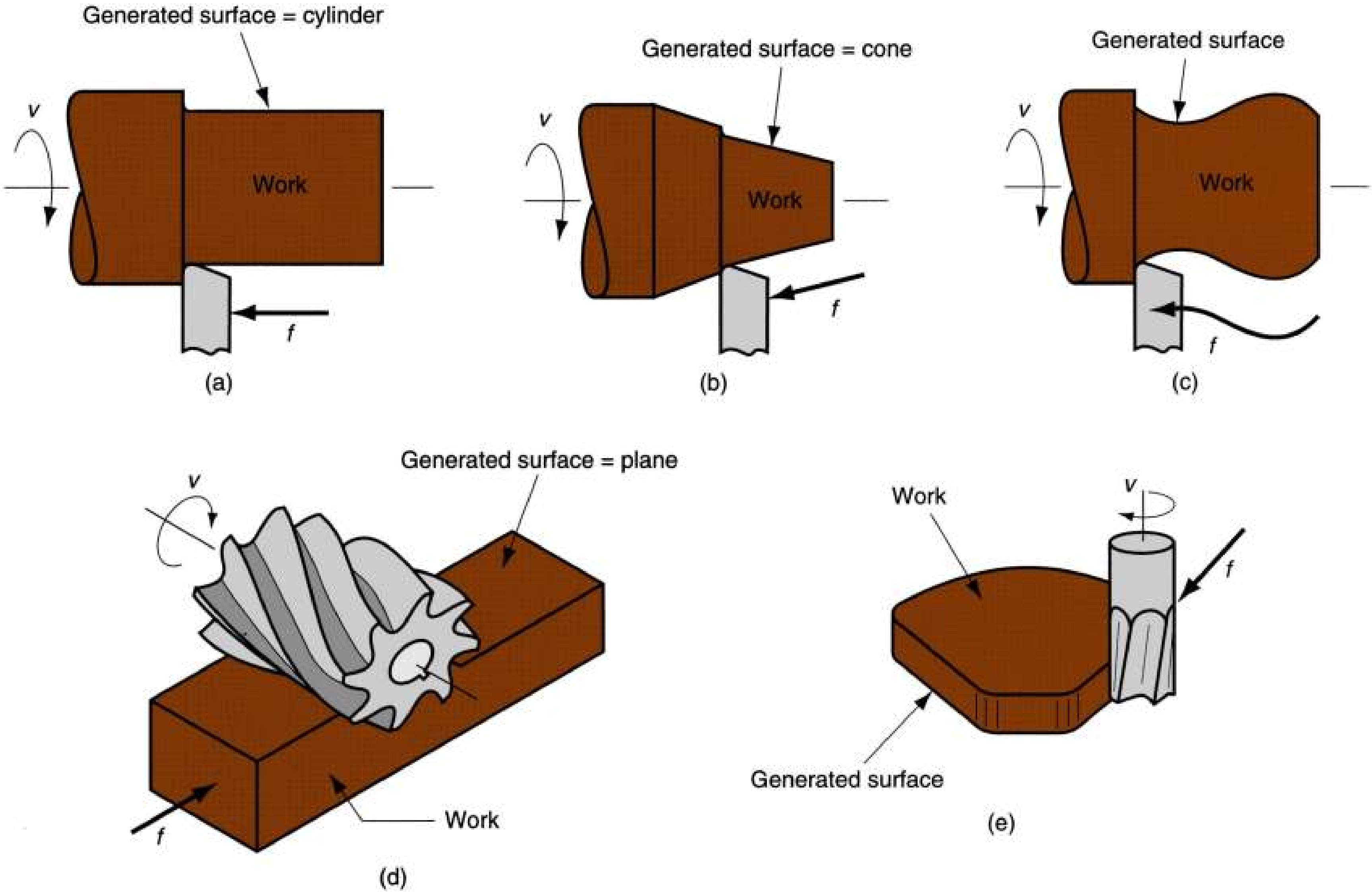


Figure - Generating shape: (a) straight turning, (b) taper turning, (c) contour turning, (d) plain milling, (e) profile milling

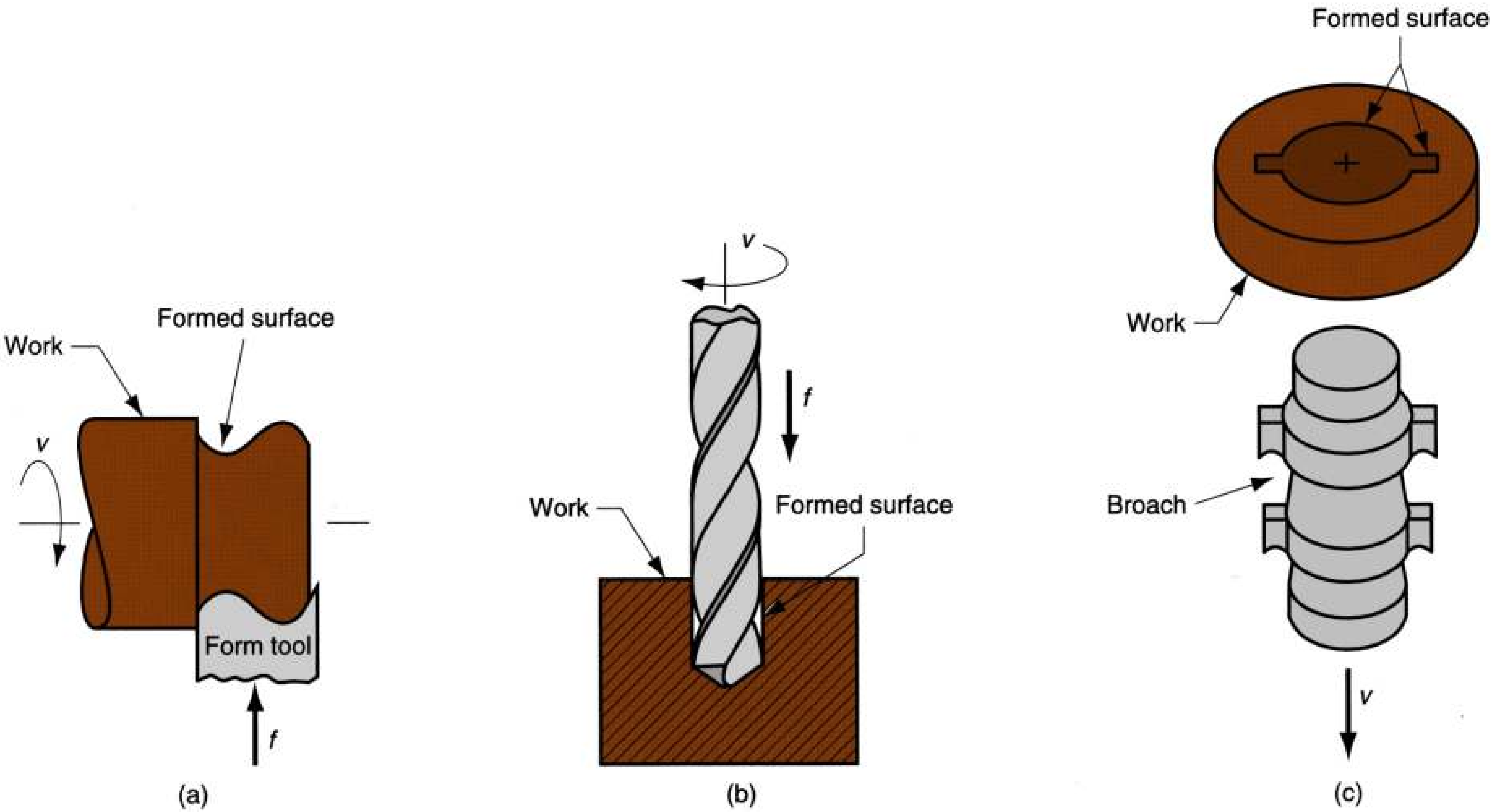


Figure - Forming to create shape: (a) form turning, (b) drilling, and (c) broaching

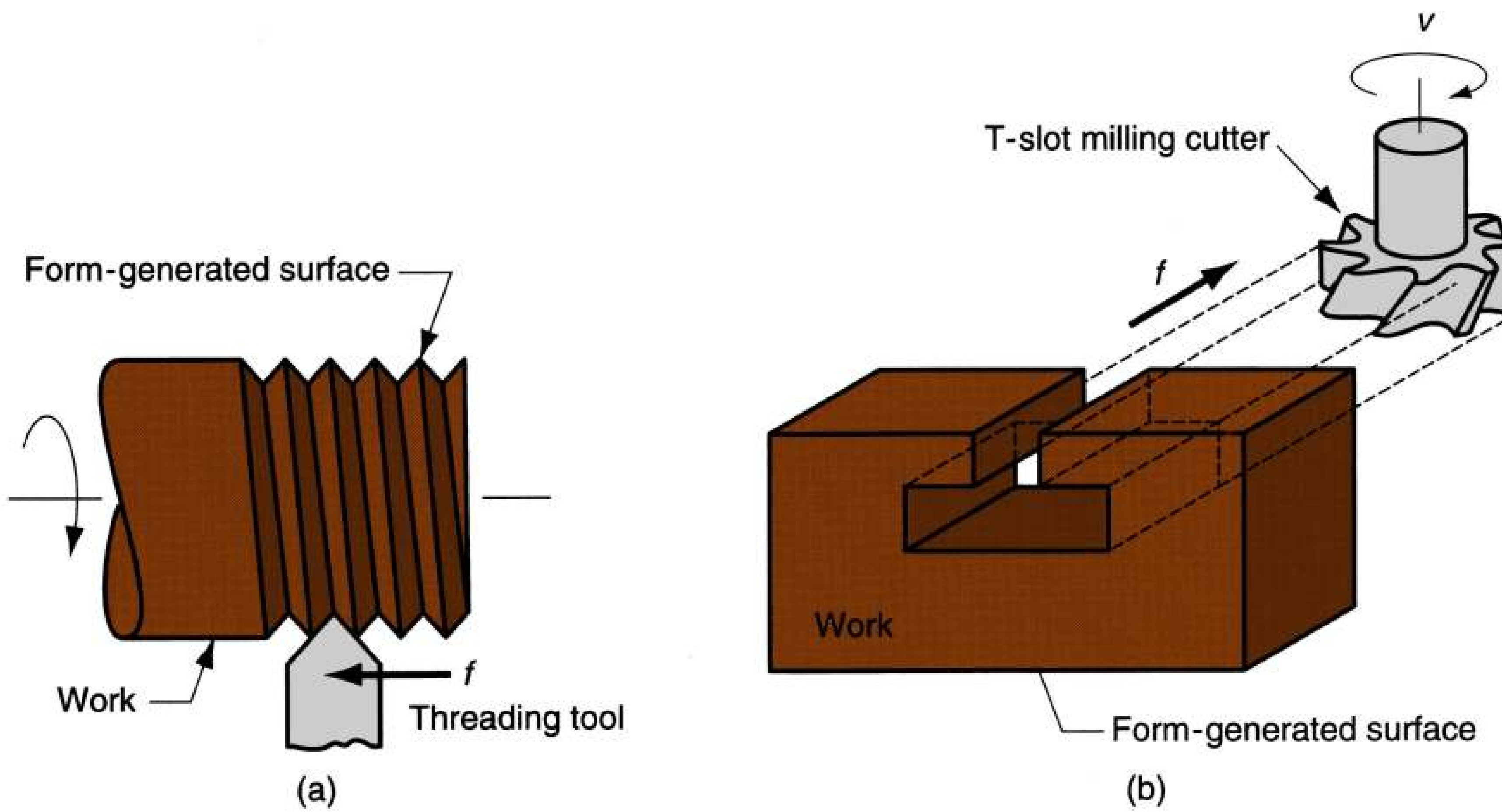
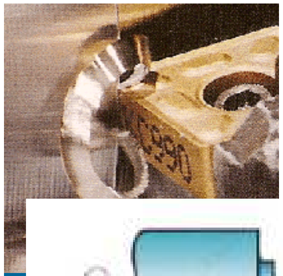
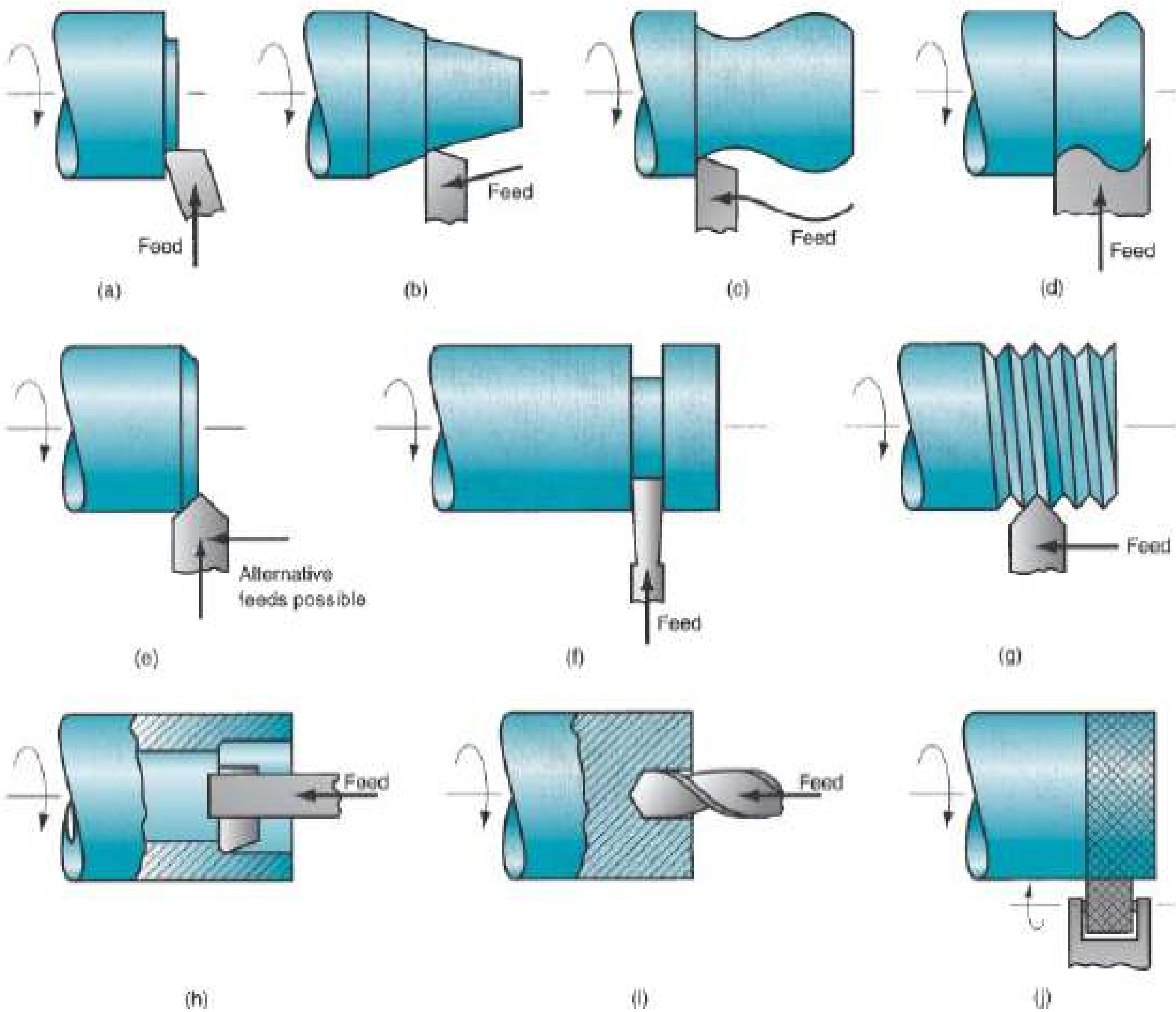


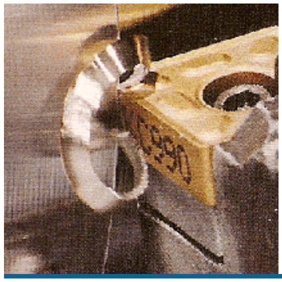
Figure - Combination of forming and generating to create shape: (a) thread cutting on a lathe, and (b) slot milling



Operations Performed on Lathe (Other than Turning)



(a) facing
b) taper turning
(c) contour turning
(d) form turning
(e) chamfering
(f) cutoff
(g) threading
(h) Boring
(i) drilling
(j) knurling



Operations Performed on Lathe (Other than Turning)

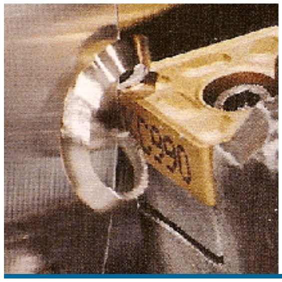
Facing: Tool is fed radially inward to create a flat surface

Taper turning: The tool is fed at an angle instead of feeding parallel to the axis of rotation of work

Contour turning: Instead of feeding the tool parallel to the axis of rotation, tool follows a contour that is other than straight, thus creating a contoured form

Form turning: The tool has a shape that is imparted to the work by plunging the tool radially into work

Chamfering: Cutting edge cuts an angle on the corner of the cylinder, forming a "chamfer"



Operations Performed on Lathe (Other than Turning)

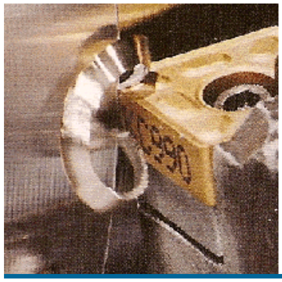
Cutoff: Tool is fed radially into rotating work at some location to cut off end of part

Threading: Pointed form tool is fed linearly across surface of rotating workpart parallel to axis of rotation at a large feed rate, thus creating threads

Boring: The tool is fed parallel to the axis of rotation on the inside diameter of an existing hole

Drilling: Drill is fed into the rotating work along its axis

Knurling: Used to produce a regular cross-hatched pattern in the work surface. Not a machining operation.

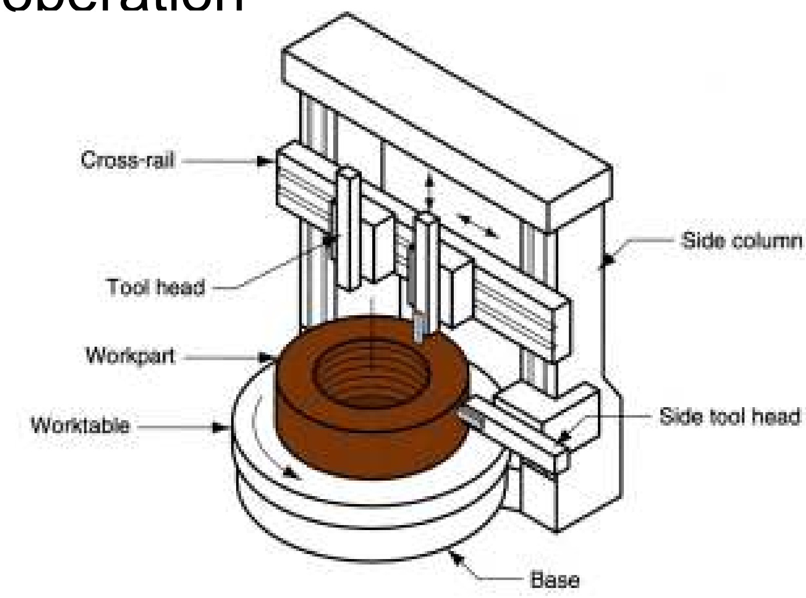


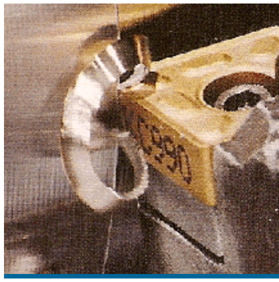
Boring

- Difference between boring and turning:
  - *Boring* is performed on the inside diameter of an existing hole
  - *Turning* is performed on the outside diameter of an existing cylinder
- In effect, boring is an internal turning operation
- Boring machines
  - Horizontal or vertical –

    refers to the orientation of the axis

    of rotation of machine spindle

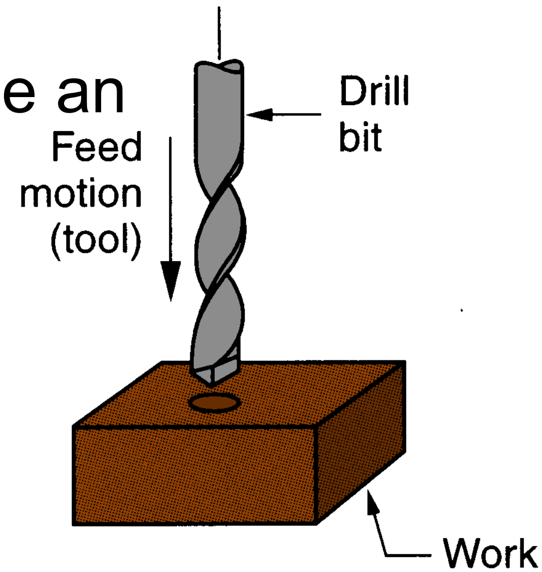




Drilling

Creates a round hole in a workpart

- Contrasts with boring which can only enlarge an existing hole
- Cutting tool called a *drill* or *drill bit*
- Customarily performed on a *drill press*

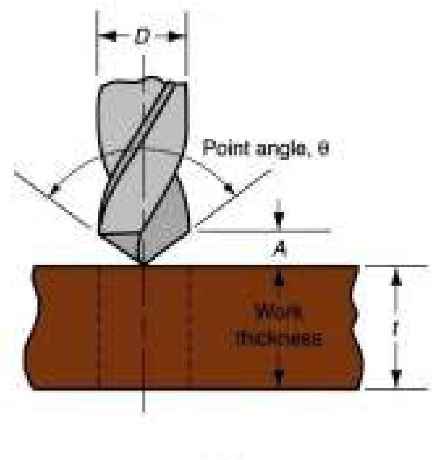


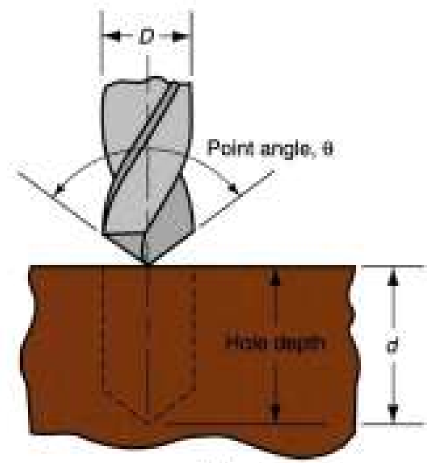
Through-holes

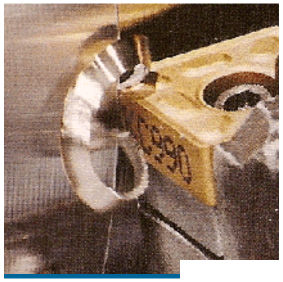
drill exits the opposite side of work

Blind-holes

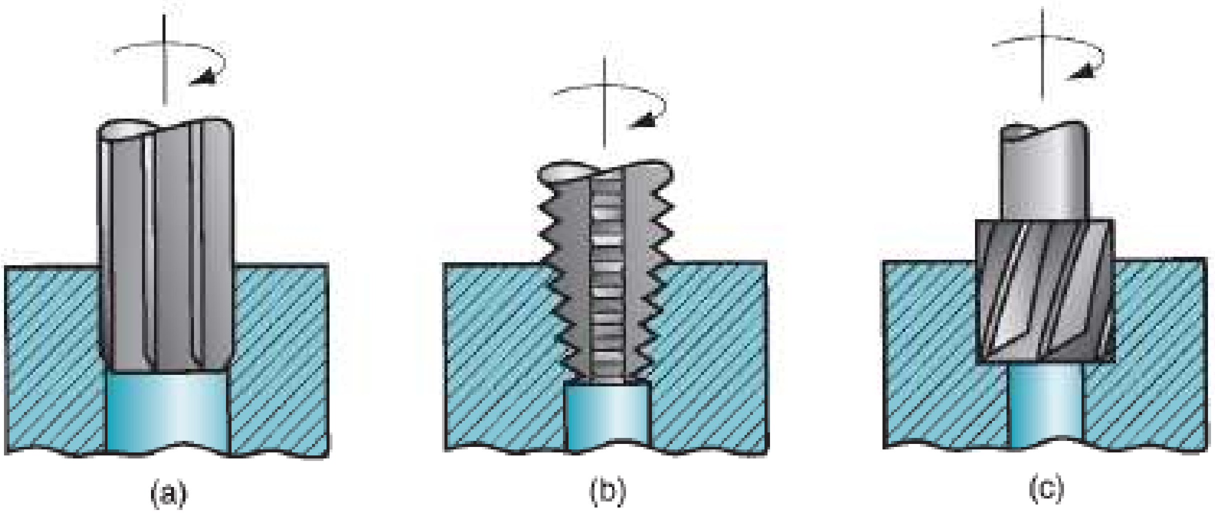
drill does not exit work on opposite side







Machining Operations Related to Drilling

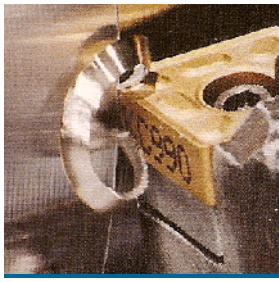


(a) Reaming (b) Tapping (c) Counterboring

Reaming: Used to slightly enlarge a hole, provide better tolerance on diameter, and improve surface finish

Tapping: Used to provide internal screw threads on an existing hole. Tool called a *tap*

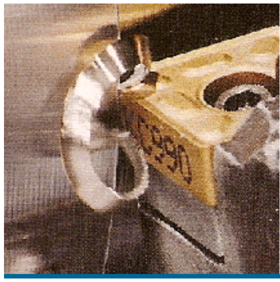
Countreboring: Provides a stepped hole, in which a larger diameter follows a smaller diameter partially into the hole



Milling

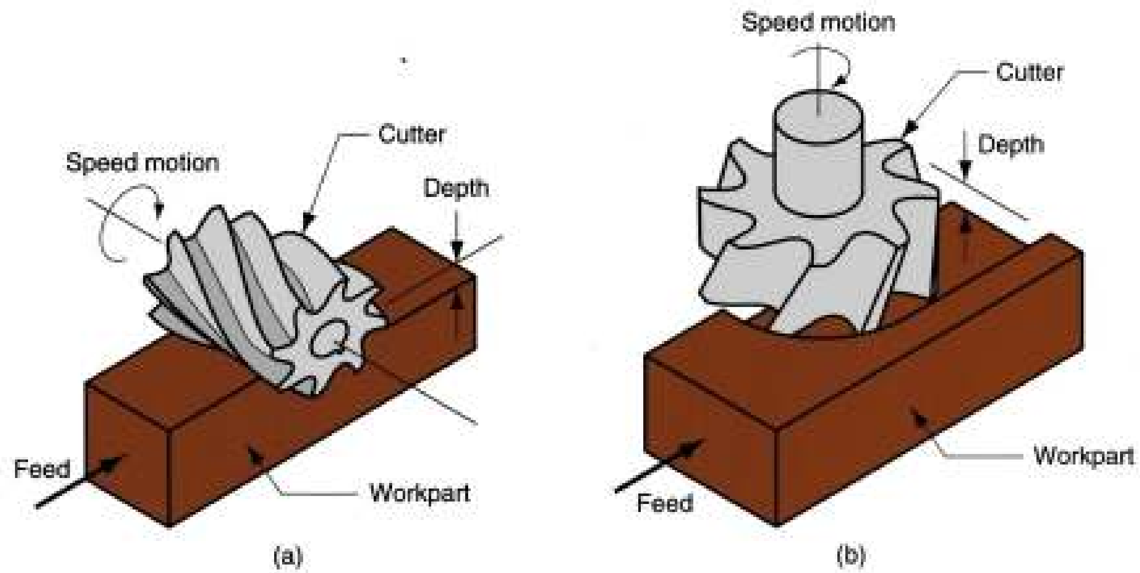
Machining operation in which work is fed past a rotating tool with multiple cutting edges

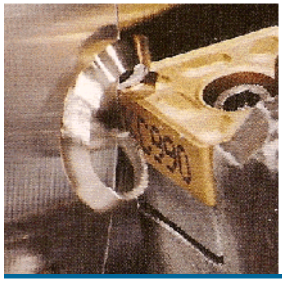
- Axis of tool rotation is perpendicular to feed direction
- Creates a planar surface; other geometries possible either by cutter path or shape
- Other factors and terms:
  - Milling is an *interrupted cutting* operation
  - Cutting tool called a *milling cutter,* cutting edges called "teeth"
  - Machine tool called a *milling machine*



Peripheral Milling vs. Face Milling

- Peripheral milling
  - Cutter axis is parallel to surface being machined
  - Cutting edges on outside periphery of cutter
- Face milling
  - Cutter axis is perpendicular to surface being milled
  - Cutting edges on both the end and outside periphery of the cutter





Peripheral Milling

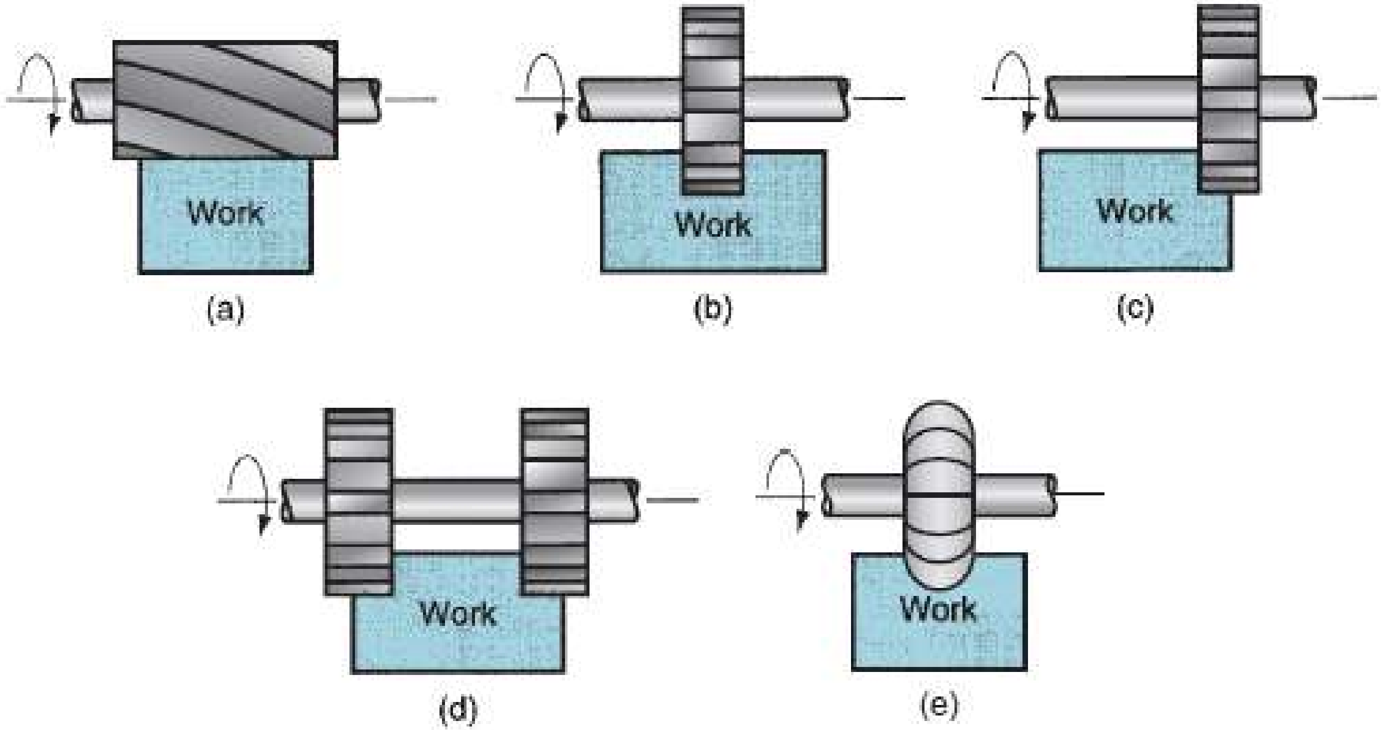
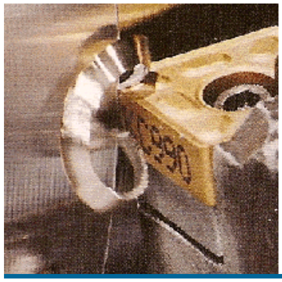


FIGURE 22.18
Peripheral milling: (a) slab milling, (b) slotting, (c) side milling, (d) straddle milling, and (e) form milling.



Slab Milling

The basic form of peripheral milling in which the cutter width extends beyond the workpiece on both sides

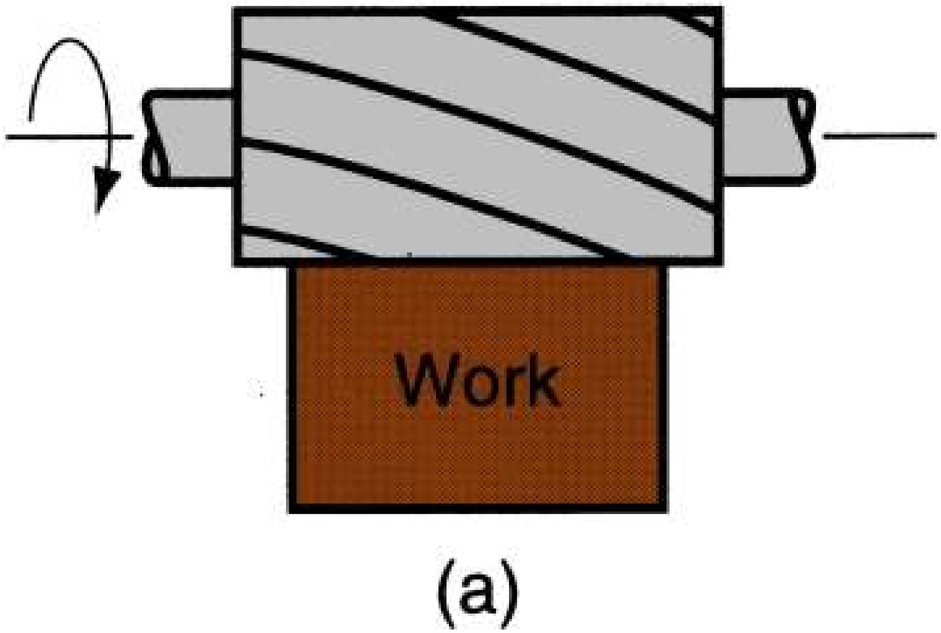
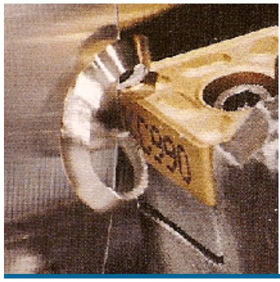


Figure (a) slab milling



Slotting

- Width of cutter is less than workpiece width, creating a slot in the work

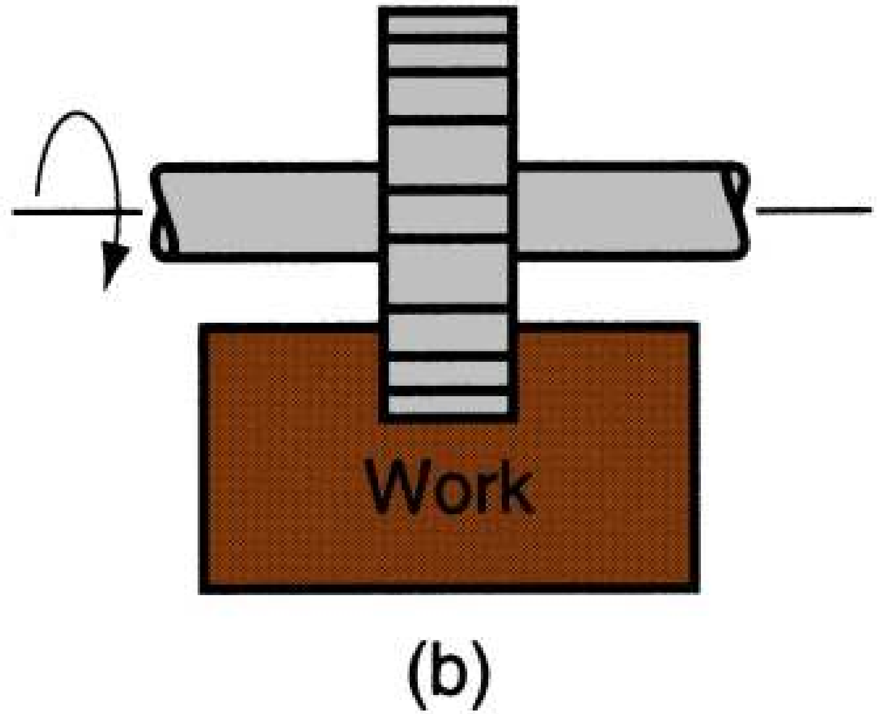
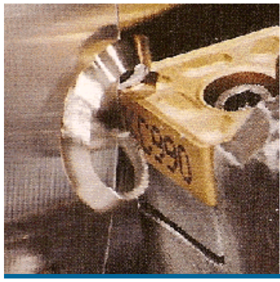
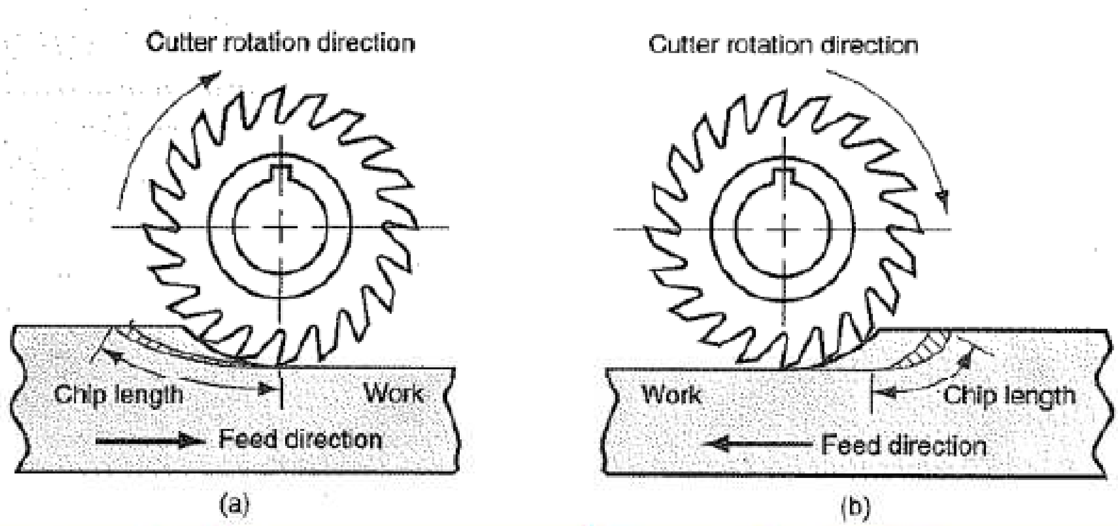


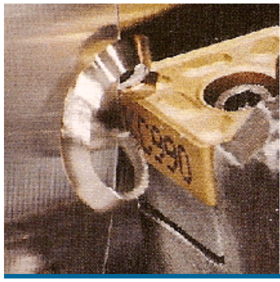
Figure (b) Slotting



Peripheral Milling: Two Forms



Up/conventional milling	Down/climb milling
Cutter teeth is opposite the feed direction	Cutter motion is the same as the feed direction
Chip starts from thin to thick	Chip starts from thick to thin
Chip length is more	Chip length is less
Cutter engaged in the work for longer time/volume of material cut	Cutter engaged for shorted time so better tool life
Lifting of workpart as the teeth exit the w/p	Tends to hold the w/p against the milling m/c table



Conventional Face Milling

Cutter overhangs work on both sides

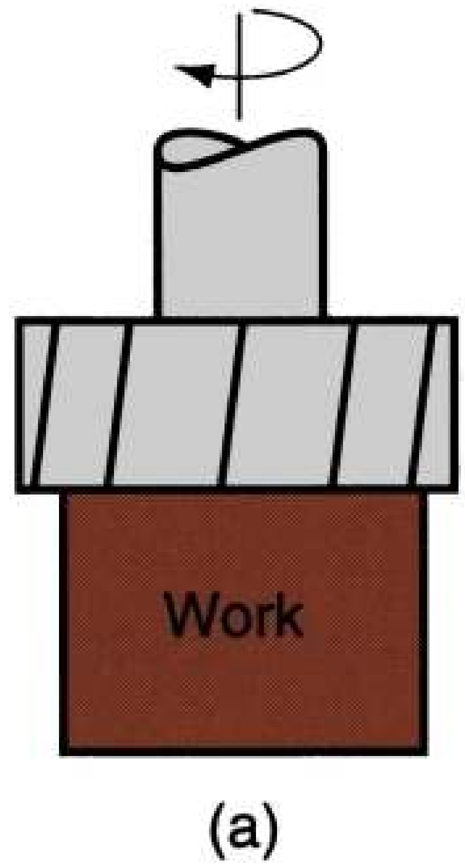
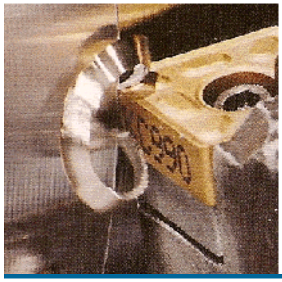


Figure (a) conventional face milling



End Milling

Cutter diameter is less than work width, so a slot is cut into part

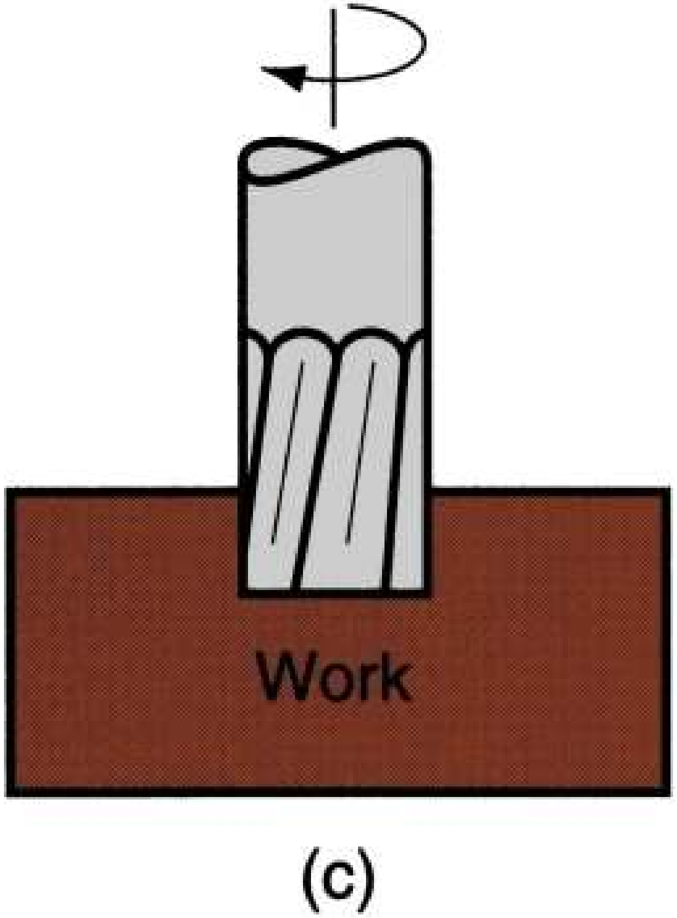
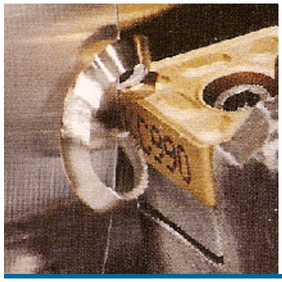


Figure - (c) end milling



Profile Milling

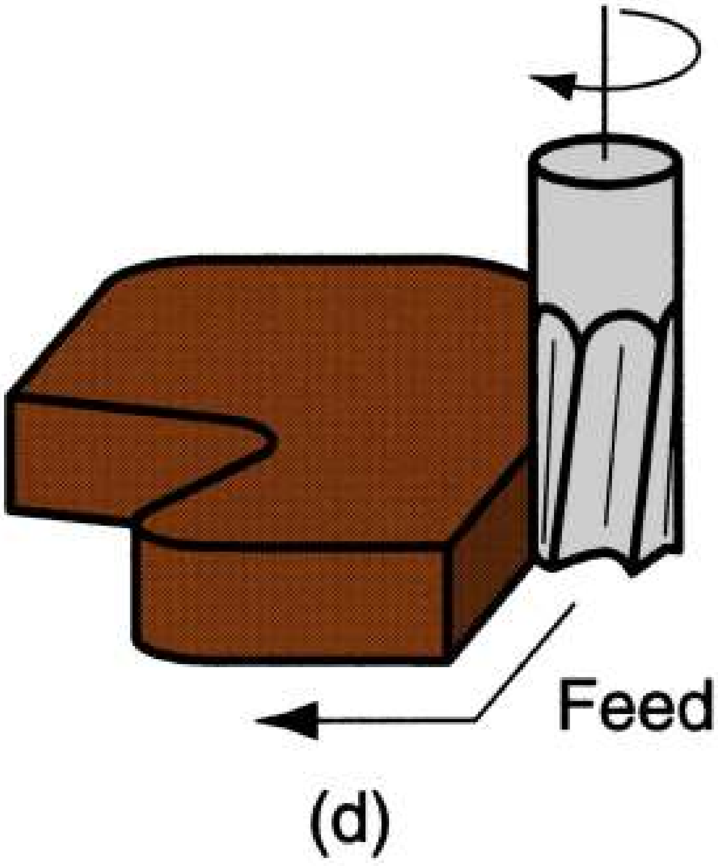
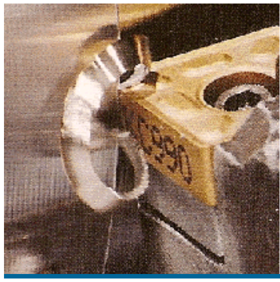


Figure (d) profile milling

Form of end milling in which the outside periphery of a flat part is cut



Pocket Milling

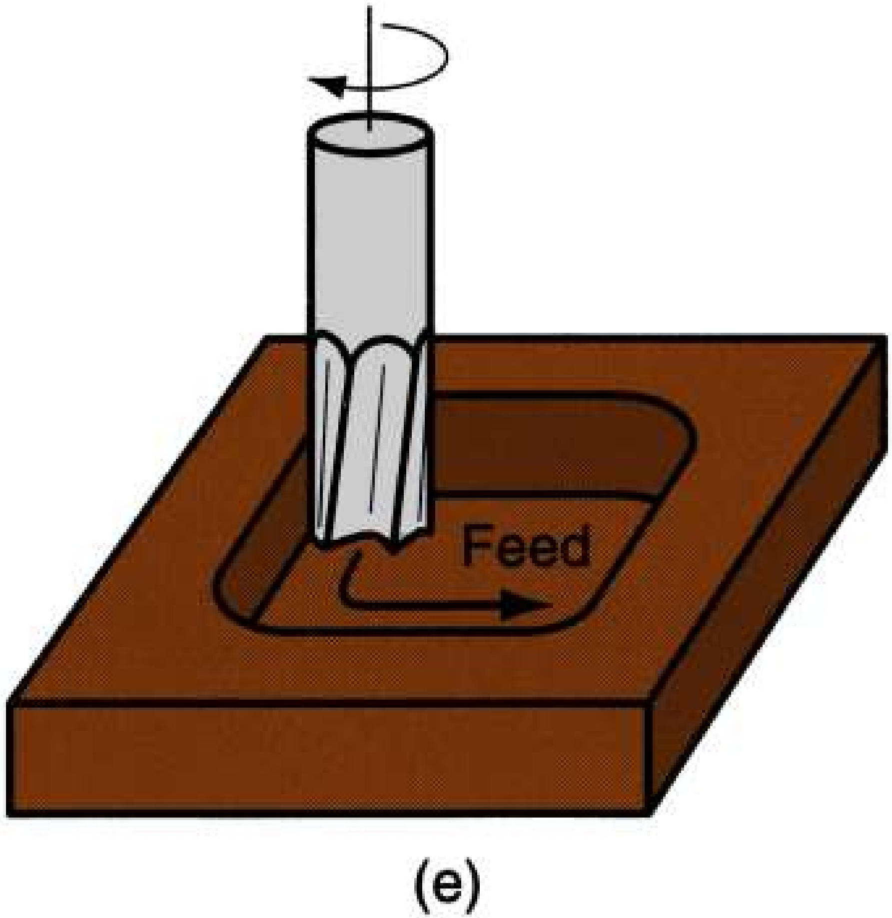
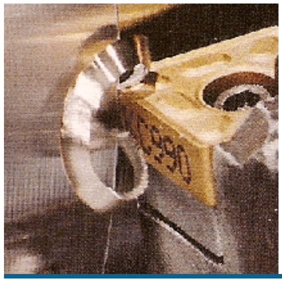
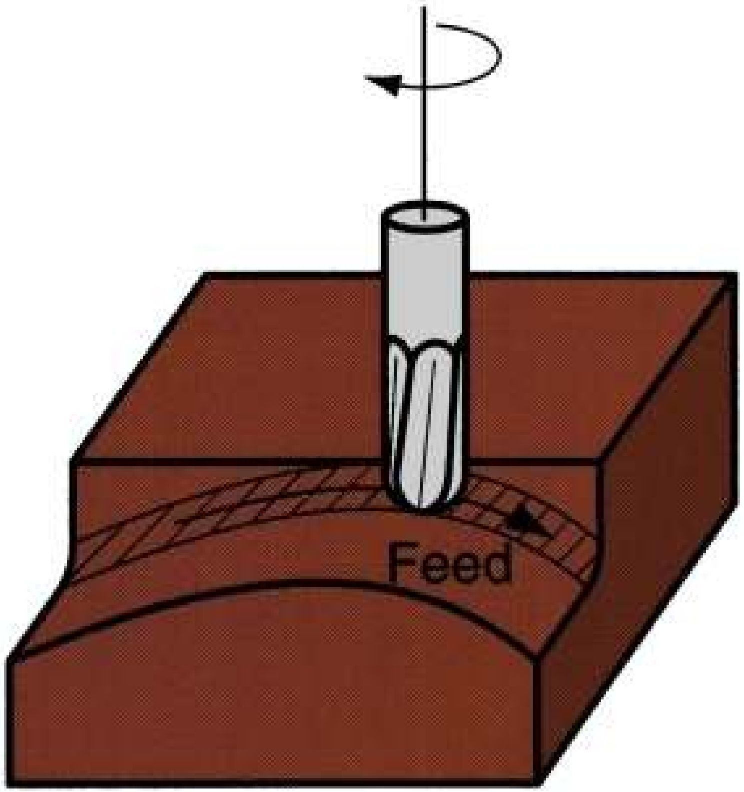


Figure (e) pocket milling

Another form of end milling used to mill shallow pockets into flat parts



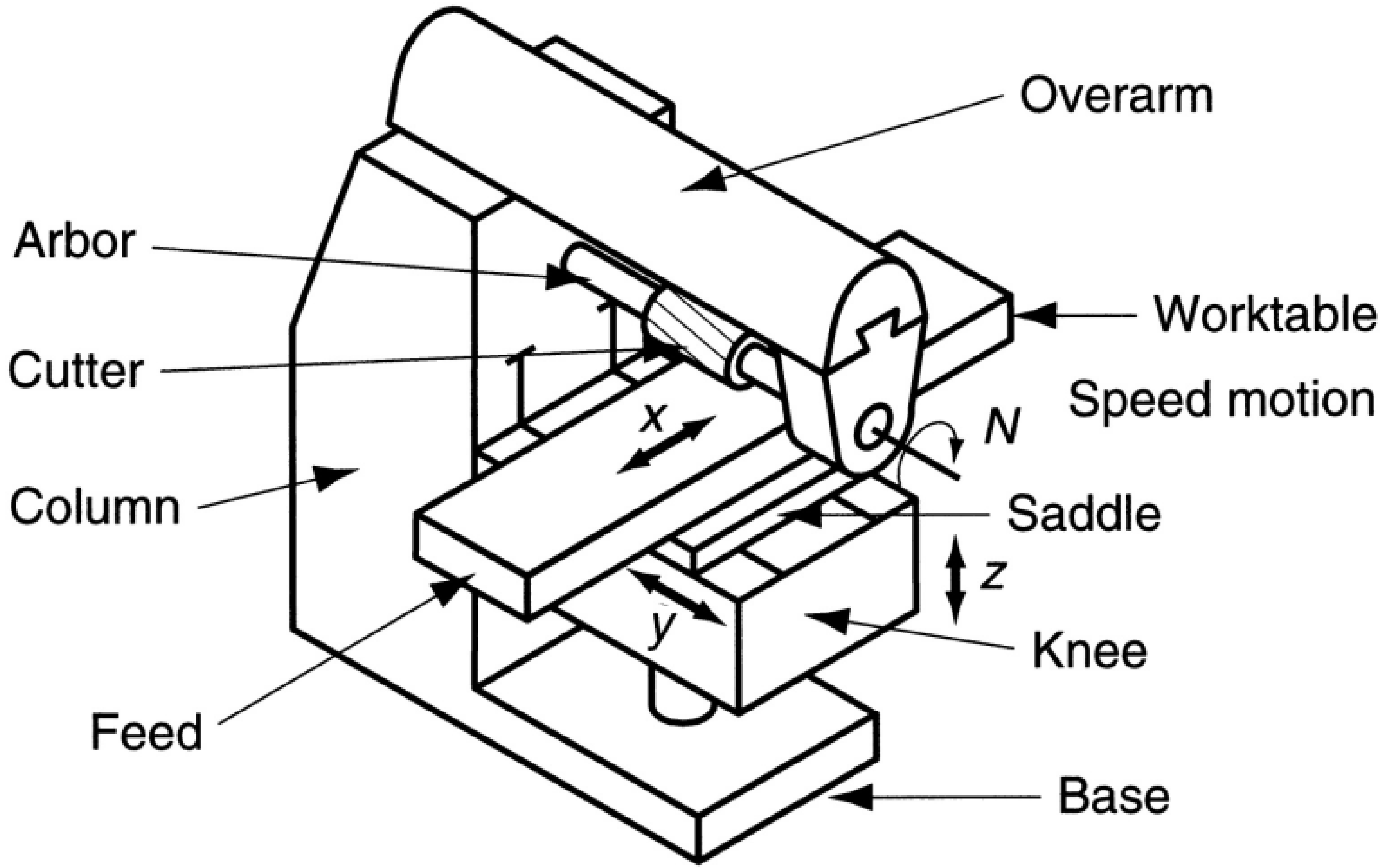
Surface Contouring



(f)

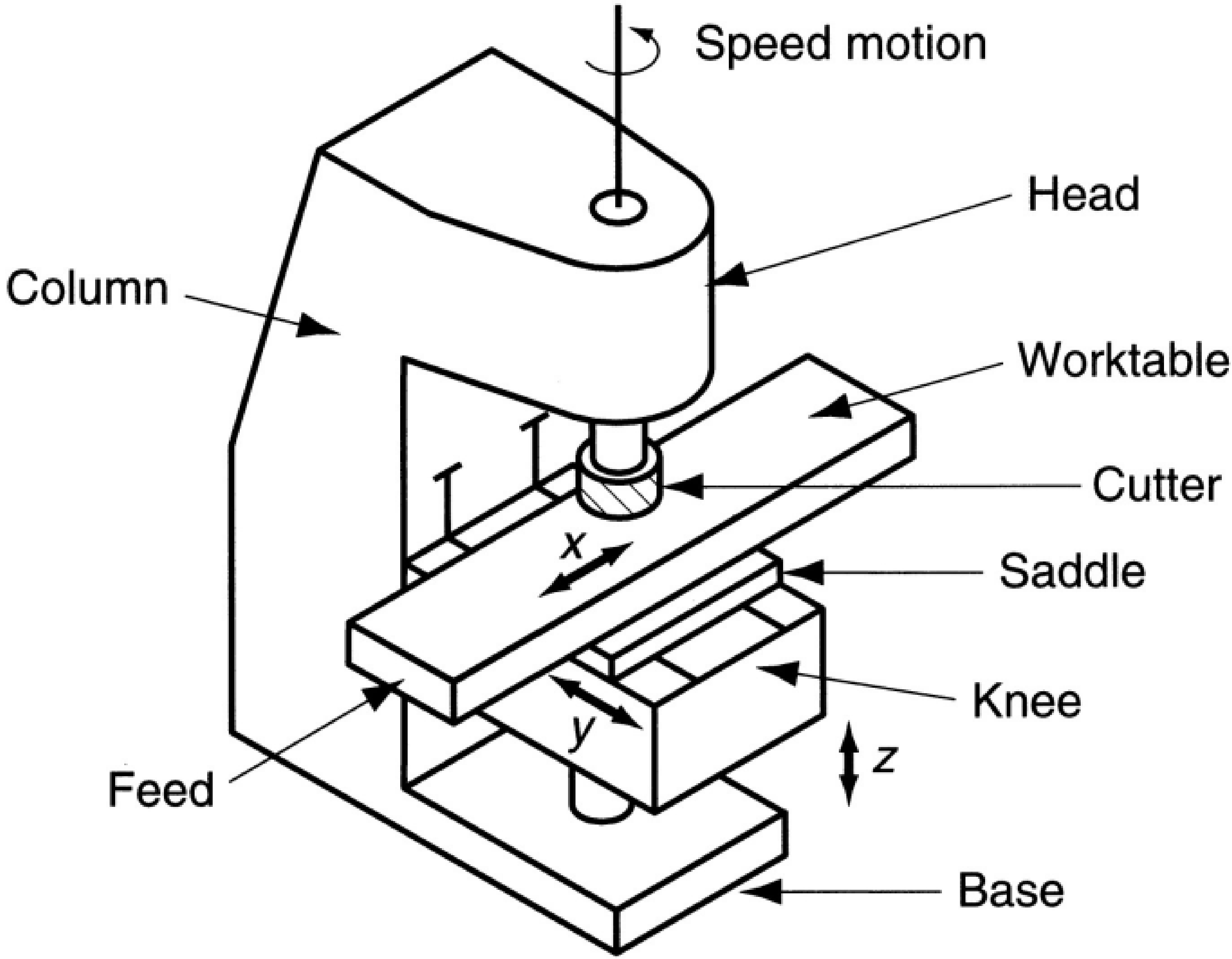
Figure (f) surface contouring

Ball-nose cutter is fed back and forth across the work along a curvilinear path at close intervals to create a three dimensional surface form



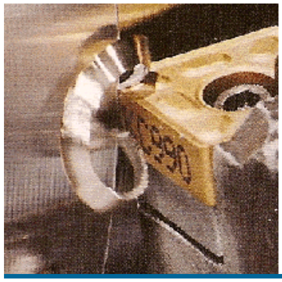
(a)

Figure (a) horizontal knee-and-column milling machine



(b)

Figure (b) vertical knee-and-column milling machine



Shaping and Planing

- Similar operations
- Both use a single point cutting tool moved linearly relative to the workpart

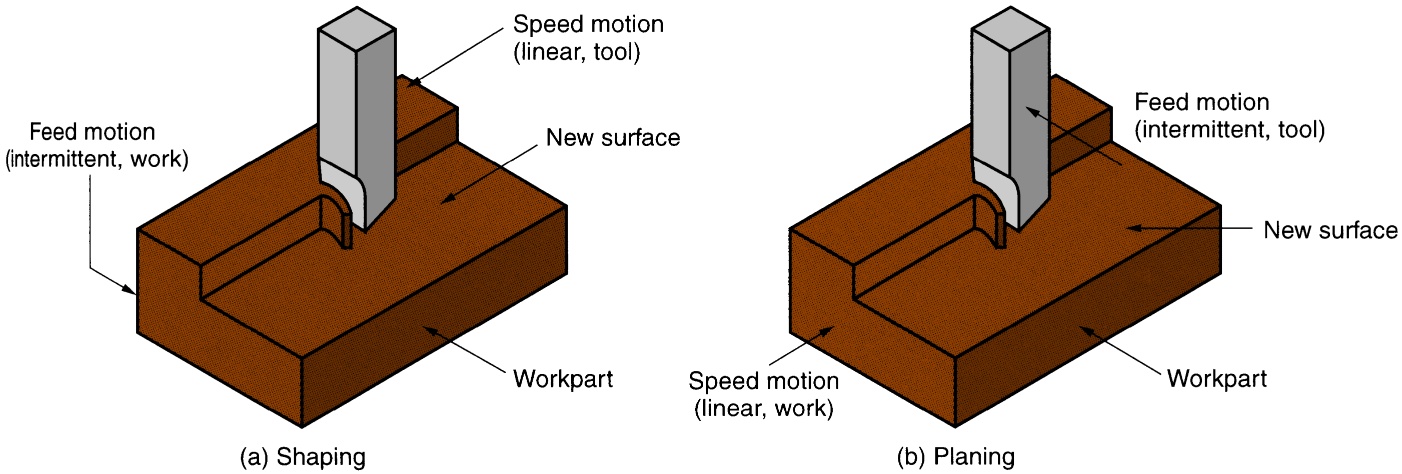
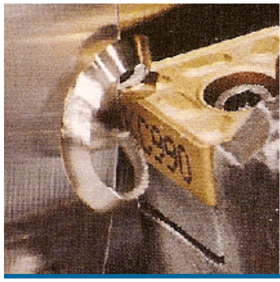


Figure 29 - (a) Shaping, and (b) planing



Shaping and Planing

- A straight, flat surface is created in both operations
- Interrupted cutting
  - Subjects tool to impact loading when entering work
- Low cutting speeds due to start-and-stop motion
- Usual tooling: single point high speed steel tools

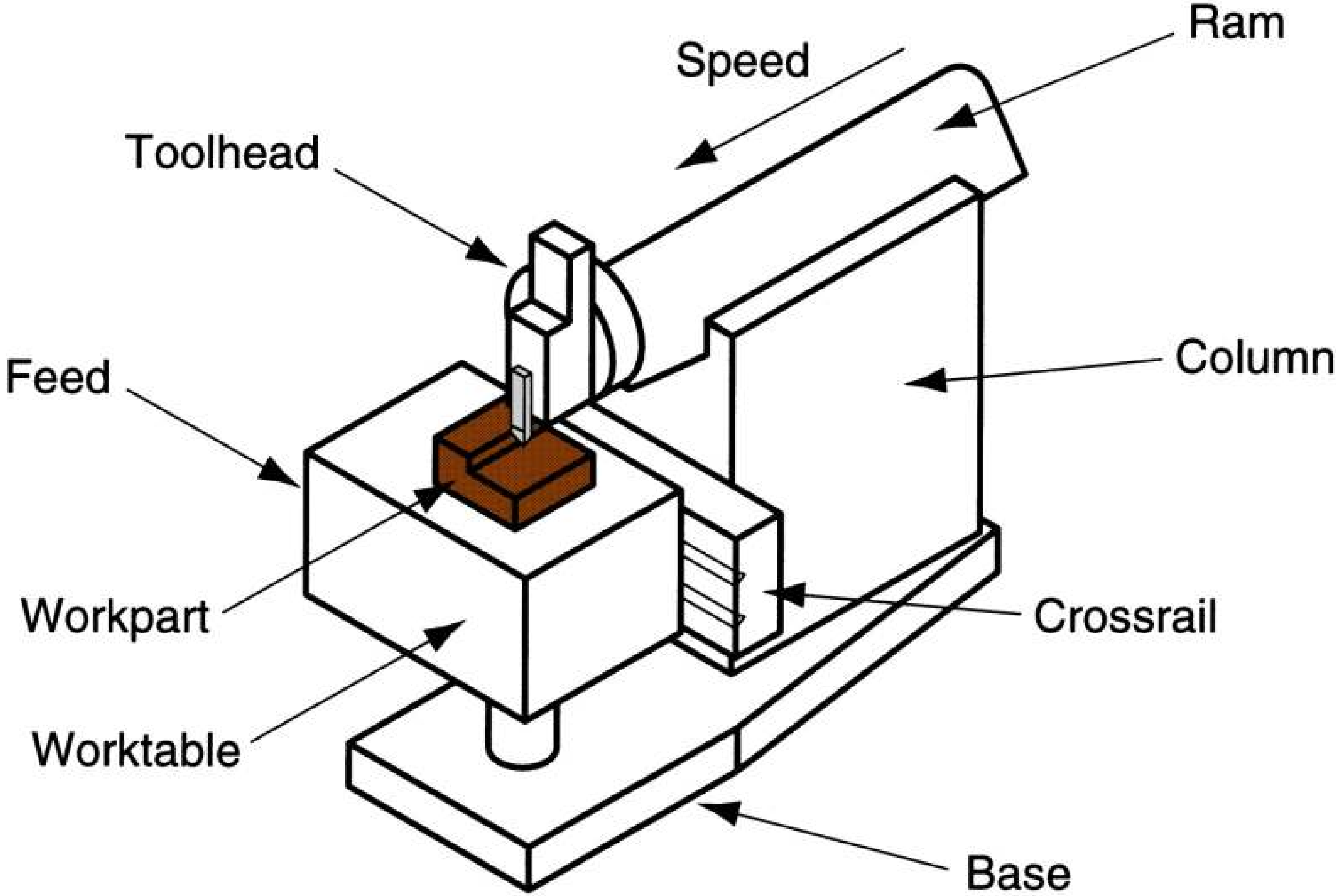


Figure - Components of a shaper

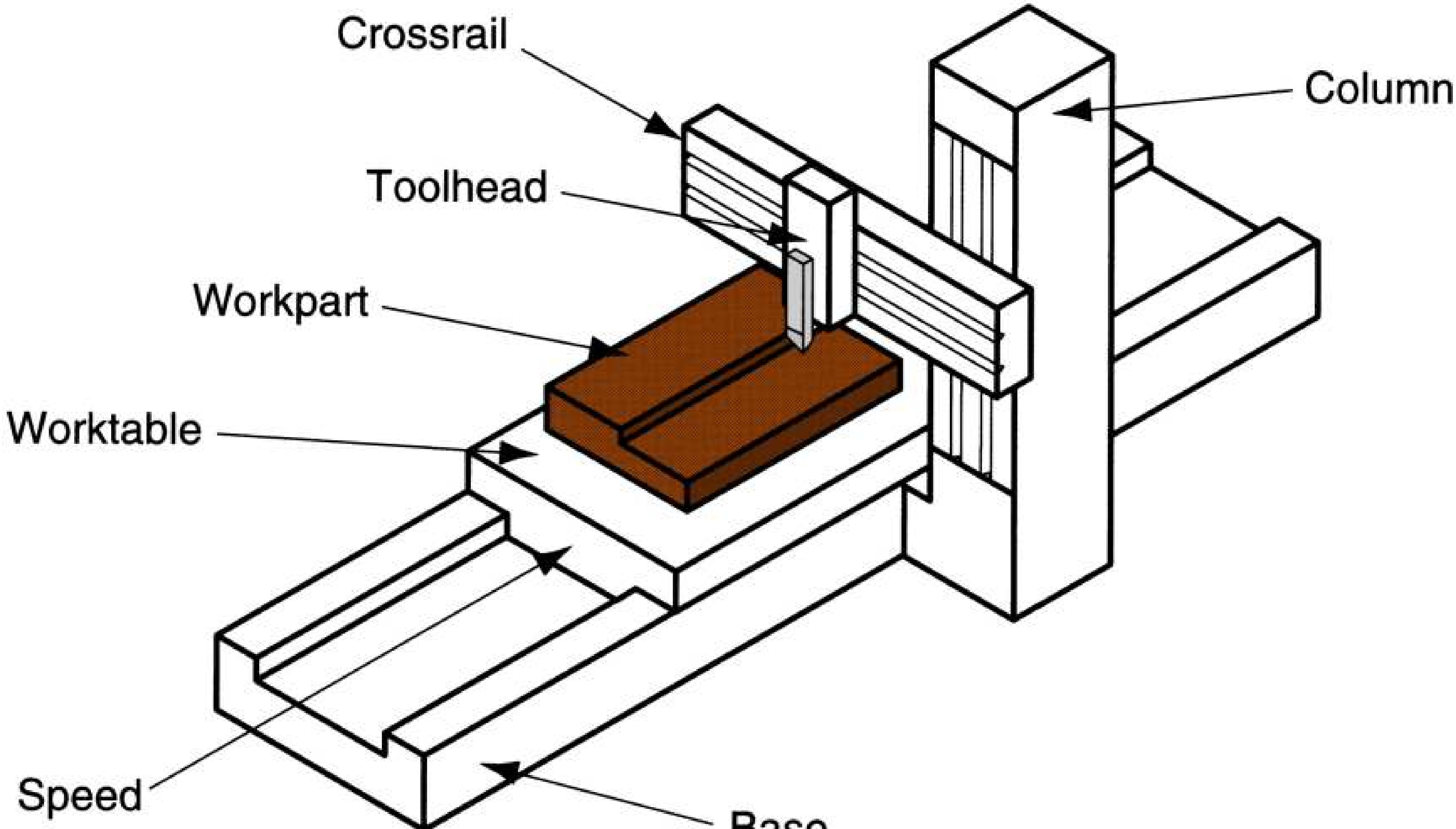
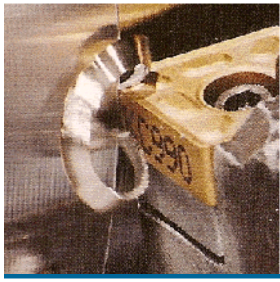


Figure - Open side planer



Broaching

- Moves a multiple tooth cutting tool linearly relative to work in direction of tool axis

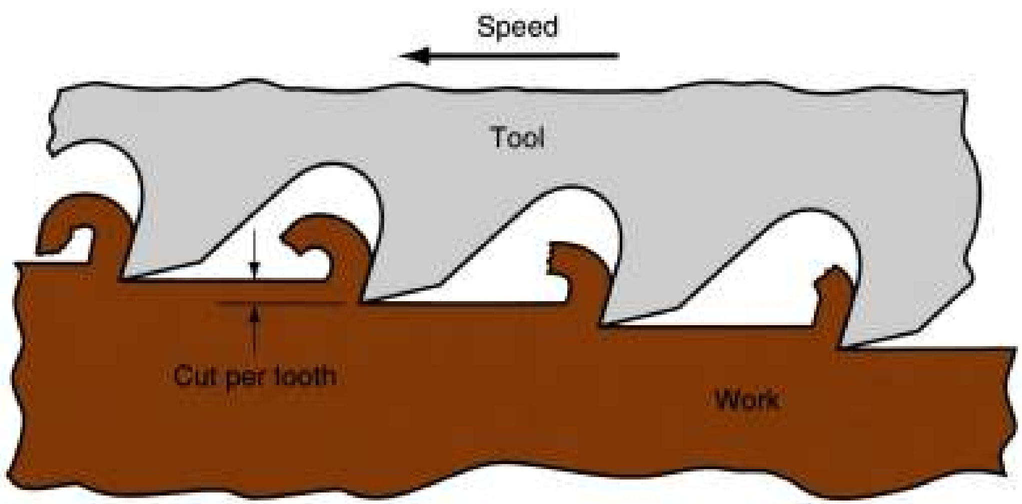
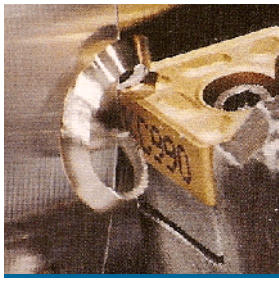


Figure - The broaching operation

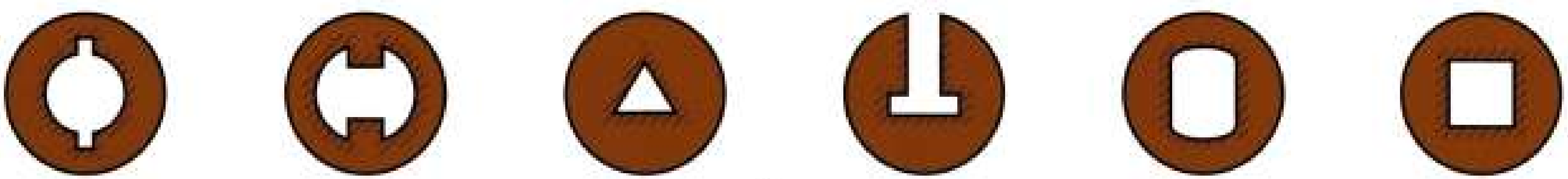
Features:

- Good surface finish
- Close tolerances
- Variety of work shapes possible
- Owing to complicated and often custom-shaped geometry, tooling is expensive



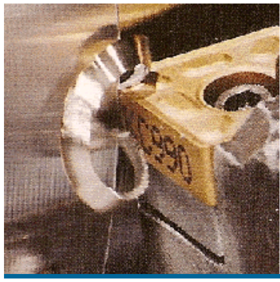
Internal Broaching

- Performed on internal surface of a hole
- A starting hole must be present in the part to insert broach at beginning of stroke



(b)

Figure - Work shapes that can be cut by internal broaching; cross-hatching indicates the surfaces broached



Sawing

- Cuts narrow slit in work by a tool consisting of a series of narrowly spaced teeth
- Tool called a *saw blade*
- Typical functions:
  - Separate a workpart into two pieces
  - Cut off unwanted portions of part

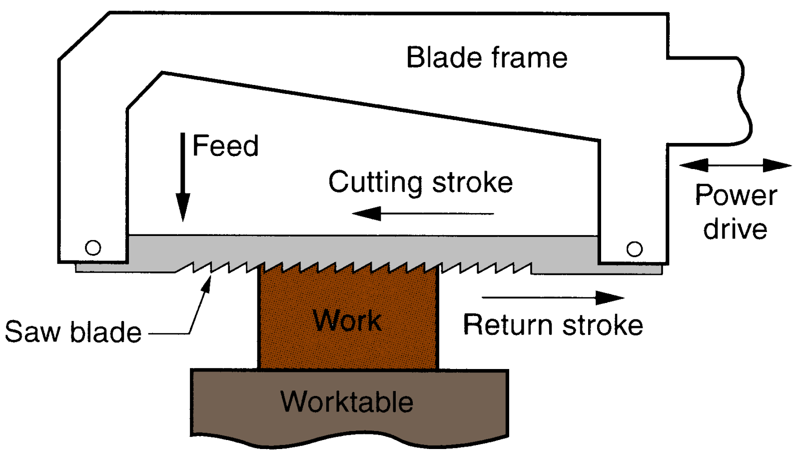
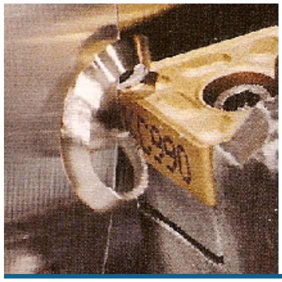
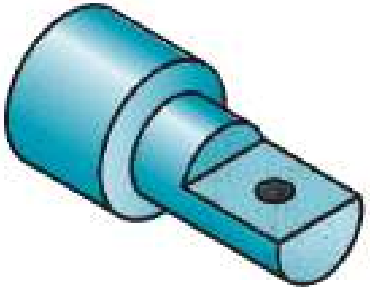


Figure power hacksaw –linear reciprocating motion of hacksaw blade against work



Process Planning For A Component

Example part to be made on a mill-turn center



Sequence of operations