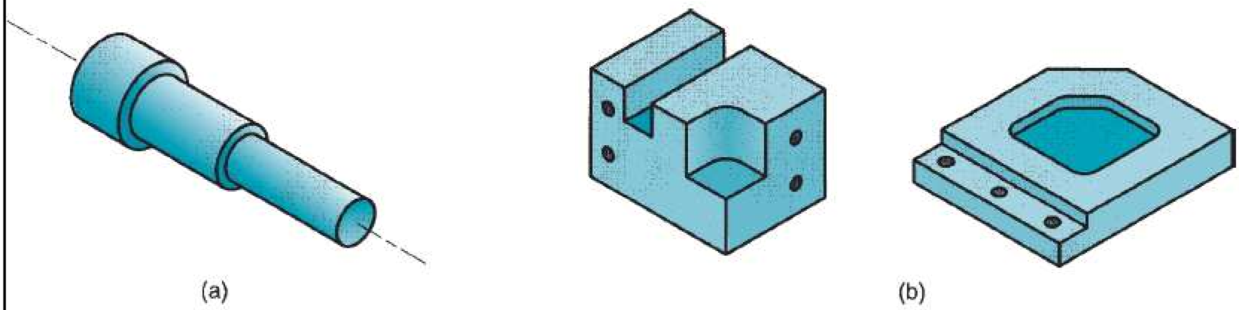
MACHINING OPERATIONS AND MACHINE TOOLS

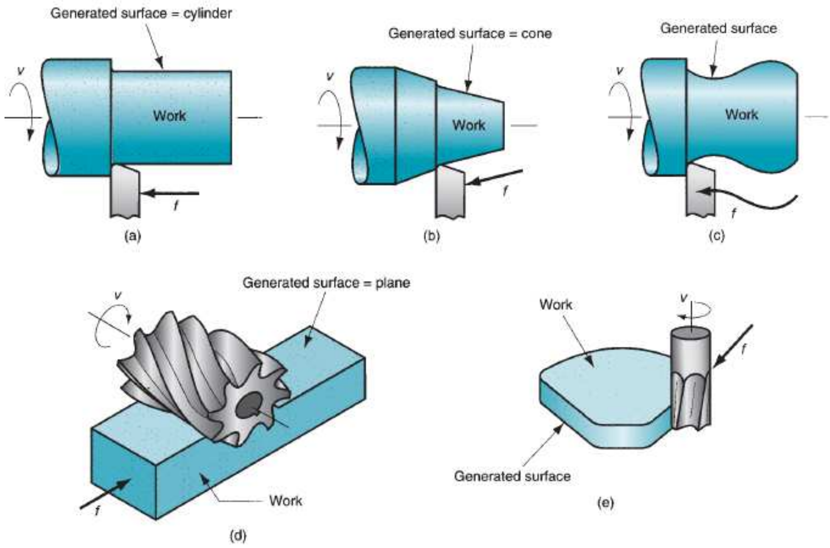


Machined parts are classified as (a) rotational, or (b) nonrotational, shown here by block and flat parts.

- The characteristic operation that produces this geometry is one in which a cutting tool removes material from a rotating workpart.
- Examples include turning and boring. Drilling is closely related except that an internal cylindrical shape is created and the tool rotates (rather than the work) in most drilling operations.

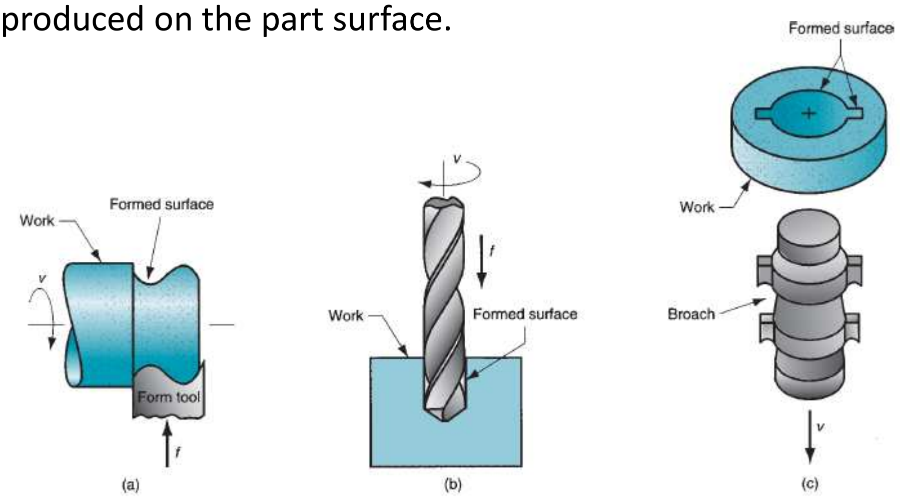
MACHINING AND PART GEOMETRY

- A nonrotational (also called prismatic) workpart is block-like or plate-like.
- This geometry is achieved by linear motions of the workpart, combined with either rotating or linear tool motions.
- Operations in this category include milling, shaping, planing, and sawing.
- Each machining operation produces a characteristic geometry due to two factors: (1) the relative motions between the tool and the workpart and (2) the shape of the cutting tool.
- We classify these operations by which part shape is created as generating and forming.
- **In generating**, the geometry of the workpart is determined by the feed trajectory of the cutting tool.
- The path followed by the tool during its feed motion is imparted to the work surface in order to create shape



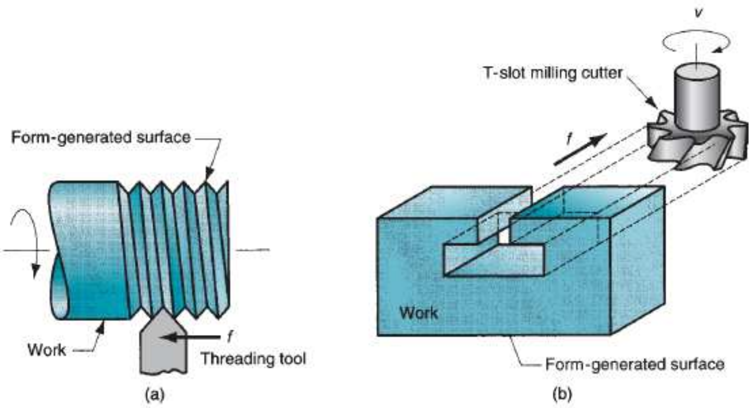
Generating shape in machining: (a) straight turning, (b) taper turning, (c) contour turning, (d) plain milling, and (e) profile milling.

- In forming, the shape of the part is created by the geometry of the cutting tool.
- In effect, the cutting edge of the tool has the reverse of the shape to be produced on the part surface.



Forming to create shape in machining: (a) form turning, (b) drilling, and (c) broaching.

- Forming and generating are sometimes combined in one operation.
- In thread cutting, the pointed shape of the cutting tool determines the form of the threads, but the large feed rate generates the threads.
- In slotting (also called slotmilling), the width of the cutter determines the width of the slot, but the feed motion creates the slot.



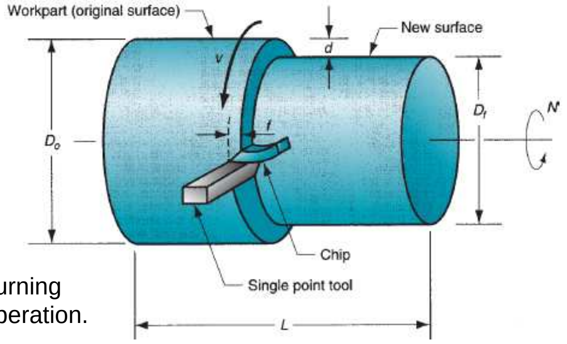
Combination of forming and generating to create shape: (a) thread cutting on a lathe, and (b) slot milling.

MACHINING AND PART GEOMETRY

- Machining is classified as a secondary process. In general, secondary processes follow basic processes, whose purpose is to establish the initial shape of a workpiece.
- Examples of basic processes include casting, forging, and bar rolling (to produce rod and bar stock).
- The shapes produced by these processes usually require refinement by secondary processes.
- Machining operations serve to transform the starting shapes into the final geometries specified by the part designer.
- For example, bar stock is the initial shape, but the final geometry after a series of machining operations is a shaft.

TURNING AND RELATED OPERATIONS

- Turning is a machining process in which a single-point tool removes material from the surface of a rotating workpiece.
- Turning is traditionally carried out on a machine tool called a lathe, which provides power to turn the part at a given rotational speed and to feed the tool at a specified rate and depth of cut.



Turning operation.

The rotational speed in turning is related to the desired cutting speed at the surface of the cylindrical workpiece by the equation

$$N = \frac{v}{\pi D_o}$$

where N = rotational speed, rev/min; v = cutting speed, m/min (ft/min); and D_o = original diameter of the part, m (ft).

The turning operation reduces the diameter of the work from its original diameter D_o to a final diameter D_f, as determined by the depth of cut d:

$$D_f = D_o - 2d$$

The feed in turning is generally expressed in mm/rev (in/rev). This feed can be converted to a linear travel rate in mm/min (in/min) by the formula

$$f_r = Nf$$

where f_r = feed rate, mm/min (in/min); and f = feed, mm/rev (in/rev).

The time to machine from one end of a cylindrical workpart to the other is given by

$$T_m = \frac{L}{f_r}$$

where T_m = machining time, min; and L = length of the cylindrical workpart, mm (in). A more direct computation of the machining time is provided by the following equation:

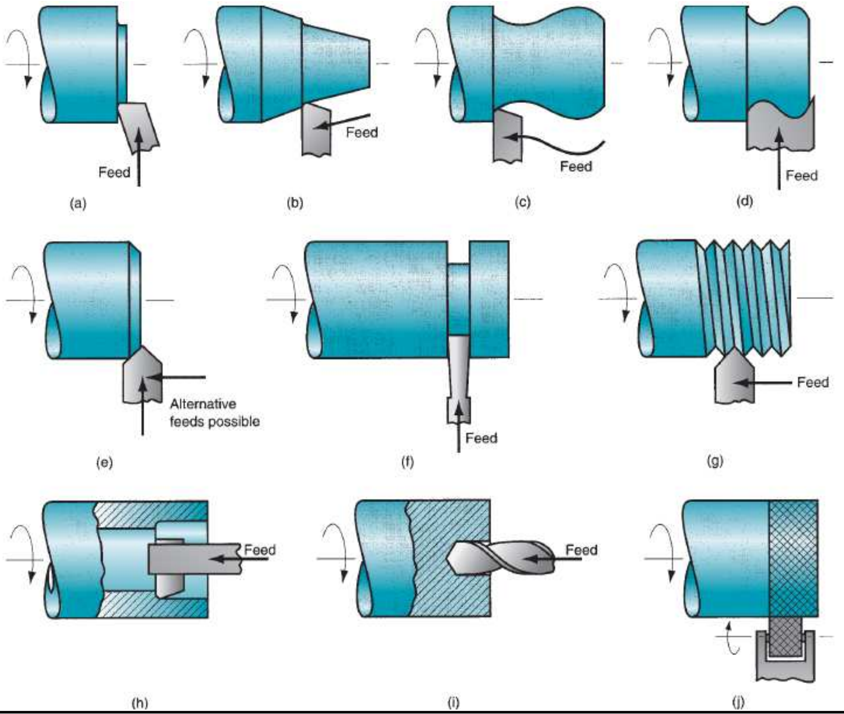
$$T_m = \frac{\pi D_o L}{fv}$$

where D_o = work diameter, mm (in); L = workpart length, mm (in); f = feed, mm/rev (in/rev); and v = cutting speed, mm/min (in/min). As a practical matter, a small distance is usually added to the workpart length at the beginning and end of the piece to allow for approach and overtravel of the tool. Thus, the duration of the feed motion past the work will be longer than T_m.

The volumetric rate of material removal can be most conveniently determined by the following equation:

$$R_{MR} = vfd$$

where R_{MR} = material removal rate, mm^3/min (in^3/min). In using this equation, the units for f are expressed simply as mm (in), in effect neglecting the rotational character of turning. Also, care must be exercised to ensure that the units for speed are consistent with those for f and d.



Machining operations other than turning that are performed on a lathe: (a) facing, (b) taper turning, (c) contour turning, (d) form turning, (e) chamfering, (f) cutoff, (g) threading, (h) boring, (i) drilling, and (j) knurling.

- **(a) Facing.** The tool is fed radially into the rotating work on one end to create a flat surface on the end.
- **(b) Taper turning.** Instead of feeding the tool parallel to the axis of rotation of the work, the tool is fed at an angle, thus creating a tapered cylinder or conical shape.
- **(c) Contour turning.** Instead of feeding the tool along a straight line parallel to the axis of rotation as in turning, the tool follows a contour that is other than straight, thus creating a contoured form in the turned part.
- **(d) Form turning.** In this operation, sometimes called forming, the tool has a shape that is imparted to the work by plunging the tool radially into the work.
- **(e) Chamfering.** The cutting edge of the tool is used to cut an angle on the corner of the cylinder, forming what is called a ''chamfer.''
- **(f) Cutoff.** The tool is fed radially into the rotating work at some location along its length to cut off the end of the part. This operation is sometimes referred to as parting.
- **(g) Threading.** A pointed tool is fed linearly across the outside surface of the rotating workpart in a direction parallel to the axis of rotation at a large effective feed rate, thus creating threads in the cylinder. **(h) Boring.** Asingle-point tool is fed linearly, parallel to the axis of rotation, on the inside diameter of an existing hole in the part.
- **(i) Drilling.** Drilling can be performed on a lathe by feeding the drill into the rotating work along its axis. Reaming can be performed in a similar way.
- **(j) Knurling.** This is not a machining operation because it does not involve cutting of material. Instead, it is a metal forming operation used to produce a regular crosshatched pattern in the work surface.

Problem:

A cylindrical workpart 200 mm in diameter and 700 mm long is to be turned in an engine lathe. Cutting speed= 2.30 m/s, feed= 0.32 mm/rev, and depth of cut= 1.80 mm. Determine (a) cutting time, and (b) metal removal rate.

THE ENGINE LATHE

- The basic lathe used for turning and related operations is an engine lathe.
- It is a versatile machine tool, manually operated, and widely used in low and medium production.

- The headstock contains the drive unit to rotate the spindle, which rotates the work.
- Opposite the headstock is the tailstock, in which a center is mounted to support the other end of the workpiece.
- The cutting tool is held in a tool post fastened to the cross-slide,which is assembled to the carriage.
- The carriage is designed to slide along the ways of the lathe in order to feed the tool parallel to the axis of rotation.

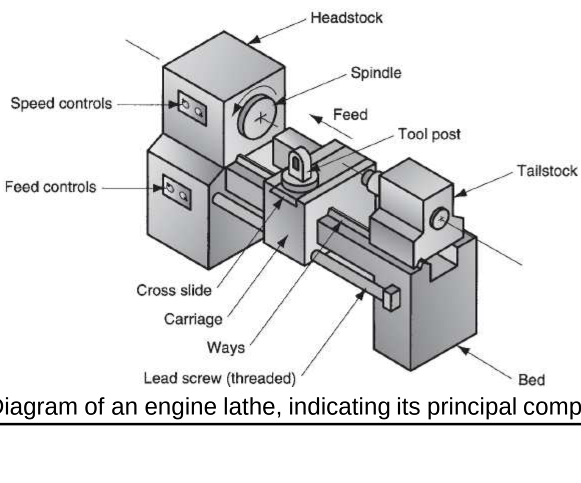
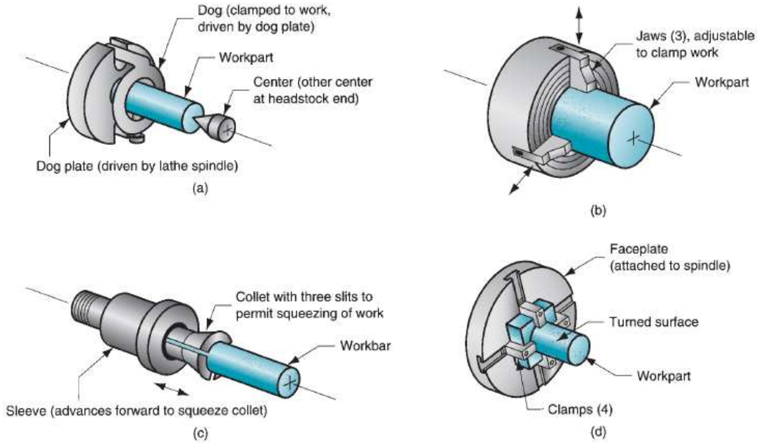


Diagram of an engine lathe, indicating its principal components.

THE ENGINE LATHE

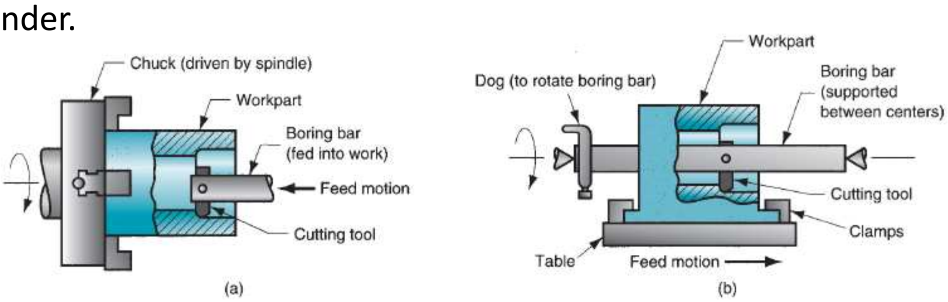
- Methods of Holding the Work in a Lathe
- There are four common methods used to hold work parts in turning.

Four work holding methods used in lathes: (a) mounting the work between centers using a dog, (b) three-jaw chuck, (c) collet, and (d) faceplate for noncylindrical workparts.



BORING MACHINES

- Boring is similar to turning. It uses a single-point tool against a rotating workpart.
- The difference is that boring is performed on the inside diameter of an existing hole rather than the outside diameter of an existing cylinder.

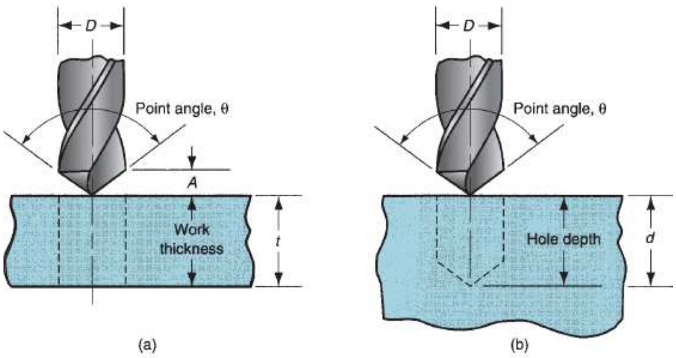


Two forms of horizontal boring: boring bar is fed into a rotating workpart, and (b) work is fed past a rotating boring bar.

Boring bar made of cemented carbide (WC–Co) that uses indexable cemented carbide inserts.

DRILLING AND RELATED OPERATIONS

- Drilling, is a machining operation used to create a round hole in a workpart.
- This contrasts with boring, which can only be used to enlarge an existing hole.
- Drilling is usually performed with a rotating cylindrical tool that has two cutting edges on its working end.
- The tool is called a drill or drill bit.



Two hole types: (a) through hole and (b) blind hole.

The cutting speed in a drilling operation is the surface speed at the outside diameter of the drill. It is specified in this way for convenience, even though nearly all of the cutting is actually performed at lower speeds closer to the axis of rotation. To set the desired cutting speed in drilling, it is necessary to determine the rotational speed of the drill. Letting N represent the spindle rev/min,

$$N = \frac{v}{\pi D} \tag{22.7}$$

where v = cutting speed, mm/min (in/min); and D = the drill diameter, mm (in). In some drilling operations, the workpiece is rotated about a stationary tool, but the same formula applies.

Feed f in drilling is specified in mm/rev (in/rev). Recommended feeds are roughly proportional to drill diameter; higher feeds are used with larger diameter drills. Since there are (usually) two cutting edges at the drill point, the uncut chip thickness (chip load) taken by each cutting edge is half the feed. Feed can be converted to feed rate using the same equation as for turning:

$$f_r = Nf \tag{22.8}$$

where f_r = feed rate, mm/min (in/min).

Drilled holes are either through holes or blind holes, Figure 22.13. In ***through holes,*** the drill exits the opposite side of the work; in ***blind holes,*** it does not. The machining time required to drill a through hole can be determined by the following formula:

$$T_m = \frac{t + A}{f_r} \tag{22.9}$$

where T_m = machining (drilling) time, min; t = work thickness, mm (in); f_r = feed rate, mm/min (in/min); and A = an approach allowance that accounts for the drill point angle, representing the distance the drill must feed into the work before reaching full diameter, Figure 22.10(a). This allowance is given by

$$A = 0.5\,D \tan\left(90 - \frac{\theta}{2}\right) \tag{22.10}$$

where A = approach allowance, mm (in); and θ = drill point angle. In drilling a through hole, the feed motion usually proceeds slightly beyond the opposite side of the work, thus making the actual duration of the cut greater than T_m in Eq. (22.9) by a small amount.

In a blind-hole, hole depth d is defined as the distance from the work surface to the depth of the full diameter, Figure 22.13(b). Thus, for a blind hole, machining time is given by

$$T_m = \frac{d + A}{f_r} \tag{22.11}$$

where A = the approach allowance by Eq. (22.10).

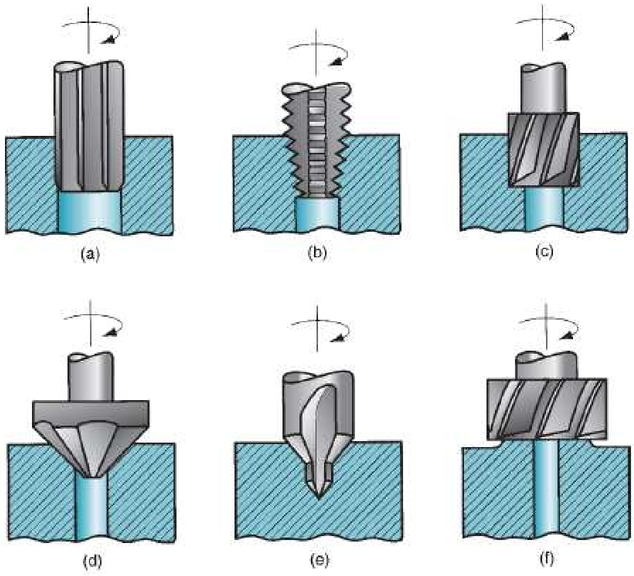
The rate of metal removal in drilling is determined as the product of the drill cross-sectional area and the feed rate:

$$R_{MR} = \frac{\pi D^2 f_r}{4} \tag{22.12}$$

This equation is valid only after the drill reaches full diameter and excludes the initial approach of the drill into the work.

OPERATIONS RELATED TO DRILLING

Most of the operations follow drilling; a hole must be made first by drilling, and then the hole is modified by one of the other operations



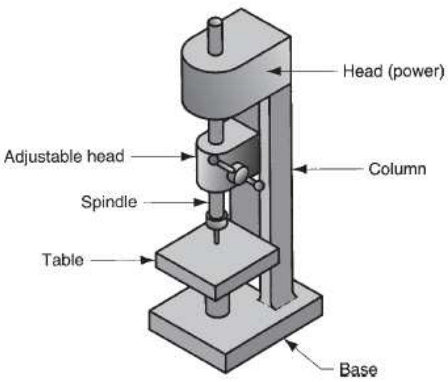
Machining operations related to drilling:
(a) reaming, (b) tapping,
(c) counterboring,
(d) countersinking,
(e) center drilling, and
(f) spot facing.

OPERATIONS RELATED TO DRILLING

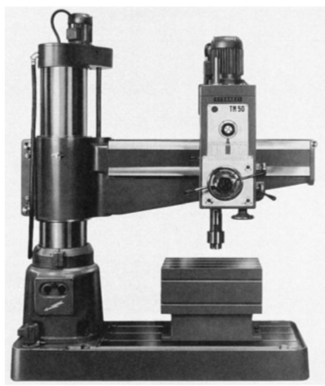
- **(a) Reaming**. Reaming is used to slightly enlarge a hole, to provide a better tolerance on its diameter, and to improve its surface finish. The tool is called a reamer, and it usually has straight flutes.
- **(b) Tapping**. This operation is performed by a tap and is used to provide internal screw threads on an existing hole.
- **(c) Counterboring**. Counterboring provides a stepped hole, in which a larger diameter follows a smaller diameter partially into the hole. A counterbored hole is used to seat bolt heads into a hole so the heads do not protrude above the surface.
- **(d) Countersinking**. This is similar to counterboring, except that the step in the hole is cone-shaped for flat head screws and bolts.
- **(e) Centering**. Also called center drilling, this operation drills a starting hole to accurately establish its location for subsequent drilling. The tool is called a center drill.
- **(f) Spot facing**. Spot facing is similar to milling. It is used to provide a flat machined surface on the workpart in a localized area.

DRILL PRESSES

- The standard machine tool for drilling is the drill press. There are various types of drill press, the most basic of which is the upright drill.



Upright drill press.



Radial drill press

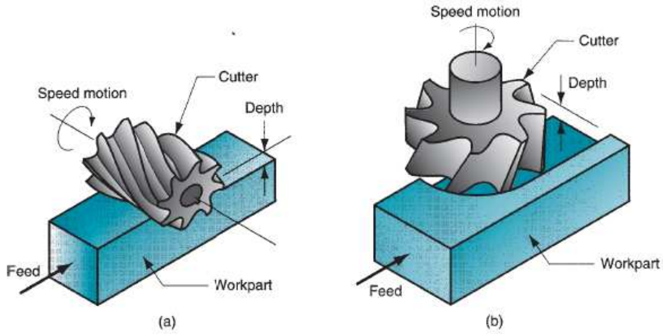
Problem:

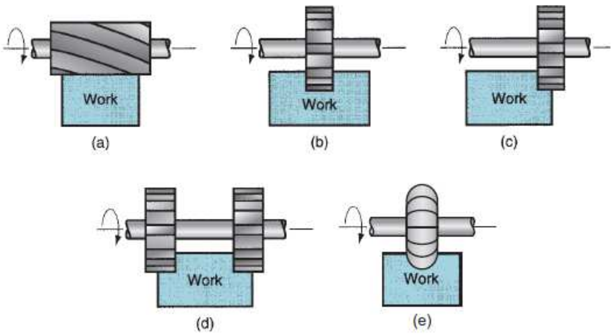
A cylindrical workpart 200 mm in diameter and 700 mm long is to be turned in an engine lathe. Cutting speed= 2.30 m/s, feed= 0.32 mm/rev, and depth of cut= 1.80 mm. Determine (a) cutting time, and (b) metal removal rate.

MILLING

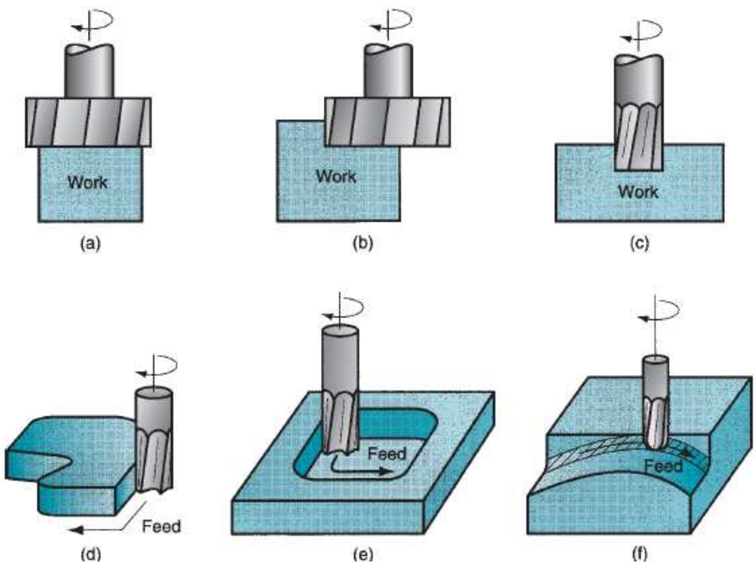
- Milling is a machining operation in which a workpart is fed past a rotating cylindrical tool with multiple cutting edges.
- The axis of rotation of the cutting tool is perpendicular to the direction of feed.
- This orientation between the tool axis and the feed direction is one of the features that distinguishes milling from drilling.
- In drilling, the cutting tool is fed in a direction parallel to its axis of rotation.
- The cutting tool in milling is called a milling cutter and the cutting edges are called teeth.

Two basic types of milling operations: (a) peripheral or plain millingand (b) face milling.





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



Face milling: (a) conventional face milling, (b) partial face milling, (c) end milling, (d) profile milling, (e) pocket milling, and (f) surface contouring.

The cutting speed is determined at the outside diameter of a milling cutter. This can be converted to spindle rotation speed using a formula that should now be familiar:

$$N = \frac{v}{\pi D} \tag{22.13}$$

The feed f in milling is usually given as a feed per cutter tooth; called the ***chip load***, it represents the size of the chip formed by each cutting edge. This can be converted to feed rate by taking into account the spindle speed and the number of teeth on the cutter as follows:

$$f_r = N n_t f \tag{22.14}$$

where f_r = feed rate, mm/min (in/min); N = spindle speed, rev/min; n_t = number of teeth on the cutter; and f = chip load in mm/tooth (in/tooth).

Material removal rate in milling is determined using the product of the cross-sectional area of the cut and the feed rate. Accordingly, if a slab-milling operation is

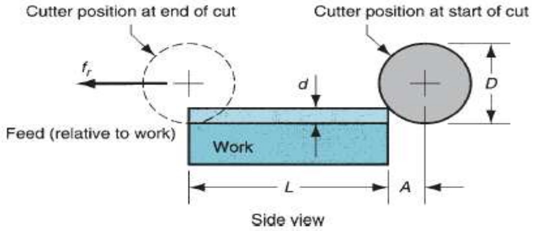


FIGURE 22.21 Slab (peripheral) milling showing entry of cutter into the workpiece.

cutting a workpiece with width w at a depth d, the material removal rate is

$$R_{MR} = w d f_r \tag{22.15}$$

This neglects the initial entry of the cutter before full engagement. Eq. (22.15) can be applied to end milling, side milling, face milling, and other milling operations, making the proper adjustments in the computation of cross-sectional area of cut.

The time required to mill a workpiece of length L must account for the approach distance required to fully engage the cutter. First, consider the case of slab milling, Figure 22.21. To determine the time to perform a slab milling operation, the approach distance A to reach full cutter depth is given by

$$A = \sqrt{d(D-d)} \tag{22.16}$$

where d = depth of cut, mm (in); and D = diameter of the milling cutter, mm (in). The time T_m in which the cutter is engaged milling the workpiece is therefore

$$T_m = \frac{L + A}{f_r} \tag{22.17}$$

For face milling, let us consider the two possible cases pictured in Figure 22.22. The first case is when the cutter is centered over a rectangular workpiece as in Figure 22.22(a). The cutter feeds from right to left across the workpiece. In order for the cutter to reach the full width of the work, it must travel an approach distance given by the following:

$$A = 0.5\left(D - \sqrt{D^2 - w^2}\right) \qquad (22.18)$$

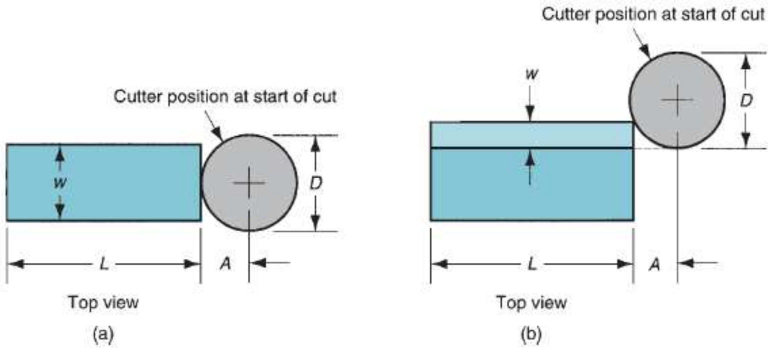


FIGURE 22.22 Face milling showing approach and overtravel distances for two cases: (a) when cutter is centered over the workpiece, and (b) when cutter is offset to one side over the work.

where D = cutter diameter, mm (in) and w = width of the workpiece, mm (in). If $D = w$, then Eq. (22.18) reduces to $A = 0.5D$. And if $D < w$, then a slot is cut into the work and $A = 0.5D$.

The second case is when the cutter is offset to one side of the work, as in Figure 22.22(b). In this case, the approach distance is given by

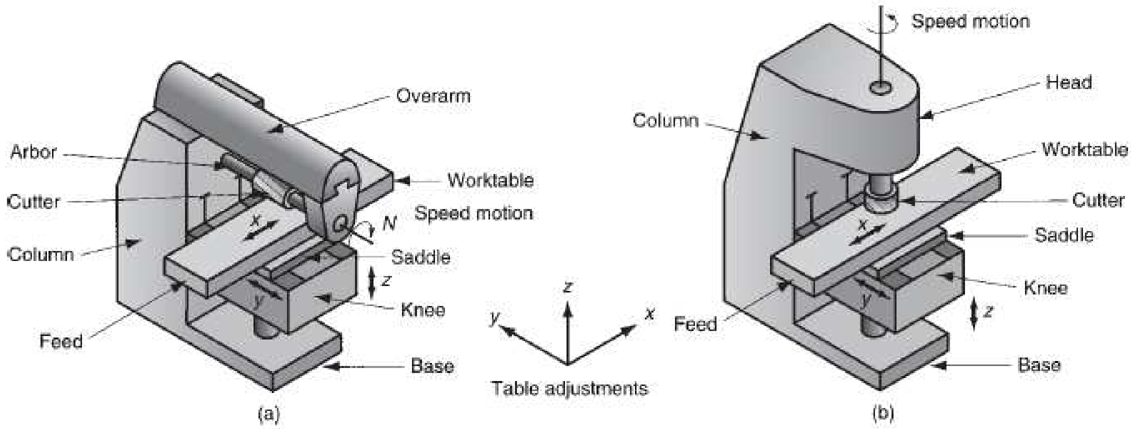
$$A = \sqrt{w(D - w)} \qquad (22.19)$$

where w = width of the cut, mm (in). In either case, the machining time is given by

$$T_m = \frac{L + A}{f_r} \qquad (22.20)$$

It should be emphasized in all of these milling scenarios that T_m represents the time the cutter teeth are engaged in the work, making chips. Approach and overtravel distances are usually added at the beginning and end of each cut to allow access to the work for loading and unloading. Thus the actual duration of the cutter feed motion is likely to be greater than T_m.

MILLING MACHINES



Two basic types of knee-and-column milling machine: (a) horizontal and (b) vertical.

CNC Milling

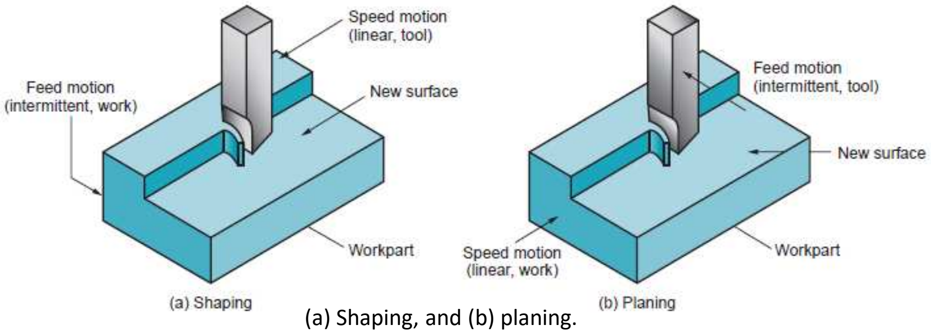
Computer numerical control, four-axis turning center.

Problem:

A face milling operation is used to machine 6.0mm from the top surface of a rectangular piece of aluminum 300 mm long by 125 mm wide in a single pass. The cutter follows a path that is centered over the workpiece. It has four teeth and is 150 mm in diameter. Cutting speed = 2.8 m/s, and chip load = 0.27 mm/tooth. Determine (a) the actual machining time to make the pass across the surface and (b) the maximum metal removal rate during cutting.

OTHER MACHINING OPERATIONS

- Shaping and planing are similar operations, both involving the use of a single-point cutting tool moved linearly relative to the workpart.
- In shaping, the speed motion is accomplished by moving the cutting tool; while in planing, the speed motion is accomplished by moving the workpart.

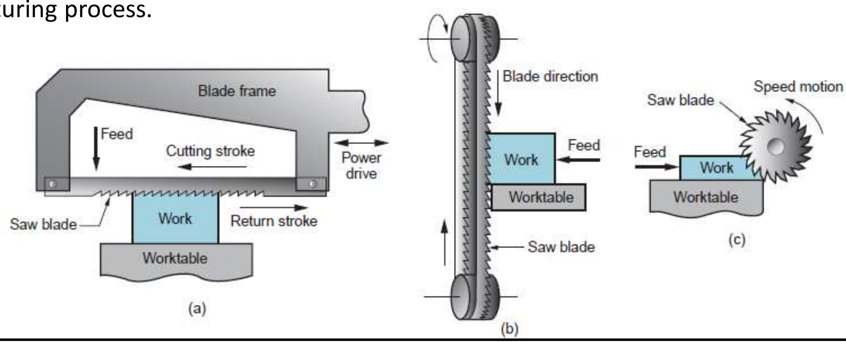


(a) Shaping, and (b) planing.

SAWING

- Sawing is a process in which a narrow slit is cut into the work by a tool consisting of a series of narrowly spaced teeth.
- Sawing is normally used to separate a workpart into two pieces, or to cut off an unwanted portion of a part.
- These operations are often referred to as cutoff operations.
- Since many factories require cutoff operations at some point in the production sequence, sawing is an important manufacturing process.

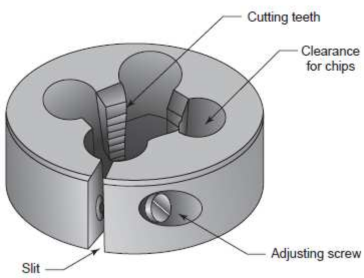
Three types of sawing operations: (a) power hacksaw, (b) bandsaw (vertical), and (c) circular saw.



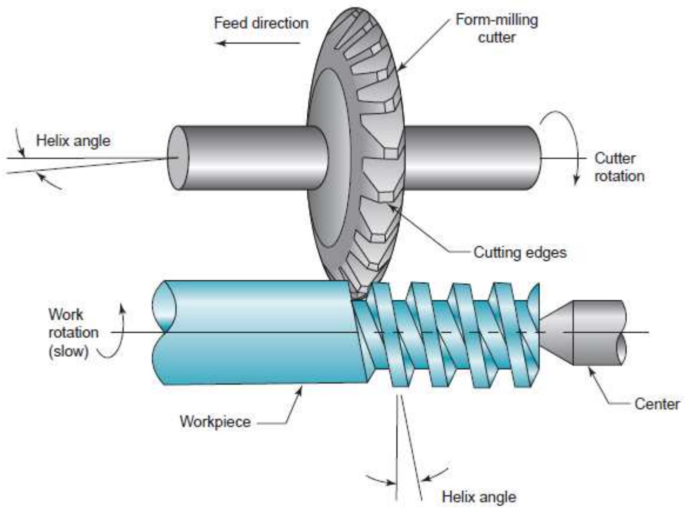
MACHINING OPERATIONS FOR SPECIAL GEOMETRIES

SCREW THREADS

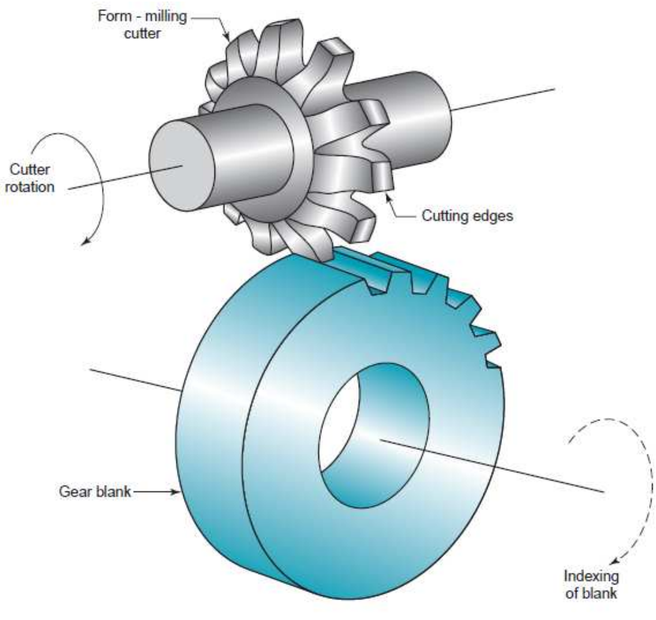
- The simplest and most versatile method of cutting an external thread on a cylindrical workpart is single-point threading, which employs a single-point cutting tool on a lathe.
- An alternative to using a single-point tool is a threading die, shown in Figure.
- To cut an external thread, the die is rotated around the starting cylindrical stock of the proper diameter, beginning at one end and proceeding to the other end.



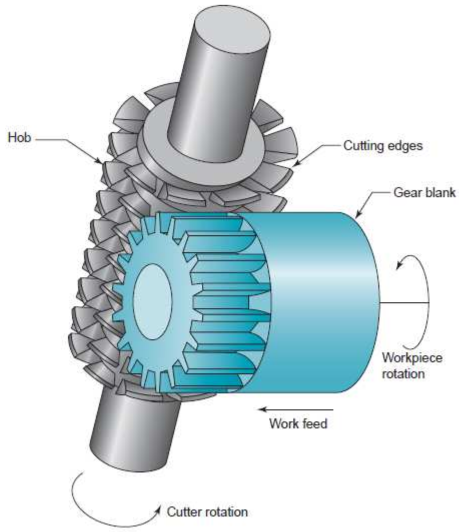
Threading die.



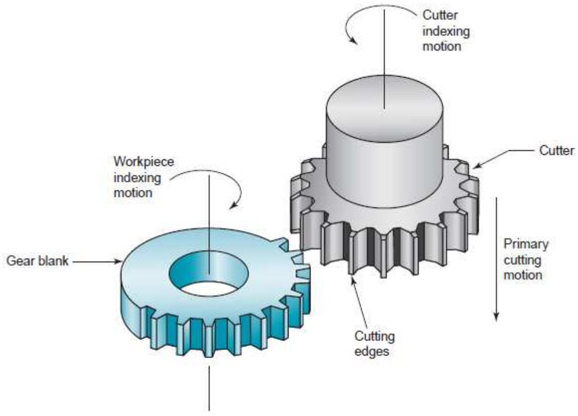
Thread milling using a form milling cutter.



Form milling of **gear** teeth on a starting blank.



Gear hobbing.



Gear shaping.

REVIEW QUESTIONS

1. What are the differences between rotational parts and prismatic parts in machining?
2. Distinguish between generating and forming when machining workpart geometries.
3. Give two examples of machining operations in which generating and forming are combined to create workpart geometry.
4. Describe the turning process.
5. What is the difference between threading and tapping?
6. How does a boring operation differ from a turning operation?
7. Name the various ways in which a workpart can be held in a lathe.
8. What is the difference between a live center and a dead center, when these terms are used in the context of workholding in a lathe?
9. What is a blind hole?
10. What is the distinguishing feature of a radial drill press?
11. What is the difference between peripheral milling and face milling?

REVIEW QUESTIONS (Cont.)

12. Describe profile milling.

13. How do shaping and planing differ?

14. A cylindrical workpart 200 mm in diameter and 700 mm long is to be turned in an engine lathe. Cutting speed = 2.30 m/s, feed = 0.32 mm/rev, and depth of cut = 1.80 mm. Determine (a) cutting time, and (b) metal removal rate.

15. A face milling operation is used to machine 6.0mm from the top surface of a rectangular piece of aluminum 300 mm long by 125 mm wide in a single pass. The cutter follows a path that is centered over the workpiece. It has four teeth and is 150 mm in diameter. Cutting speed = 2.8 m/s, and chip load = 0.27 mm/tooth. Determine (a) the actual machining time to make the pass across the surface and (b) the maximum metal removal rate during cutting.

16. A cylindrical workpart 200 mm in diameter and 700 mm long is to be turned in an engine lathe. Cutting speed= 2.30 m/s, feed= 0.32 mm/rev, and depth of cut= 1.80 mm. Determine (a) cutting time, and (b) metal removal rate.