9.1 Machining and Part Geometry

Machined parts can be classified as rotational or non-rotational (Figure 9.1). A rotational workpart has a cylindrical or disk-like shape. 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. A non-rotational (also called prismatic) workpart is block-like or plate-like, as in Figure 9.1(b).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.

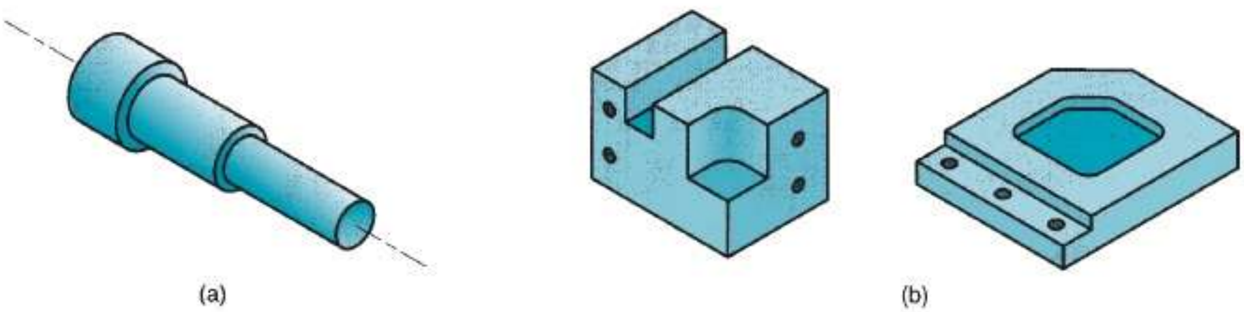

FIGURE 9.1 Machined parts are classified as (a) rotational, or (b) nonrotational, shown here by block and flat parts.

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. Examples of generating the work shape in machining include straight turning, taper turning, contour turning, peripheral milling, and profile milling, all illustrated in Figure 9.2. In each of these operations, material removal is accomplished by the speed motion in the operation, but part shape is determined by the feed motion. The feed trajectory may involve variations in depth or width of cut during the operation. For example, in the contour turning and profile milling operations shown in our figure, the feed motion results in changes in depth and width, respectively, as cutting proceeds.

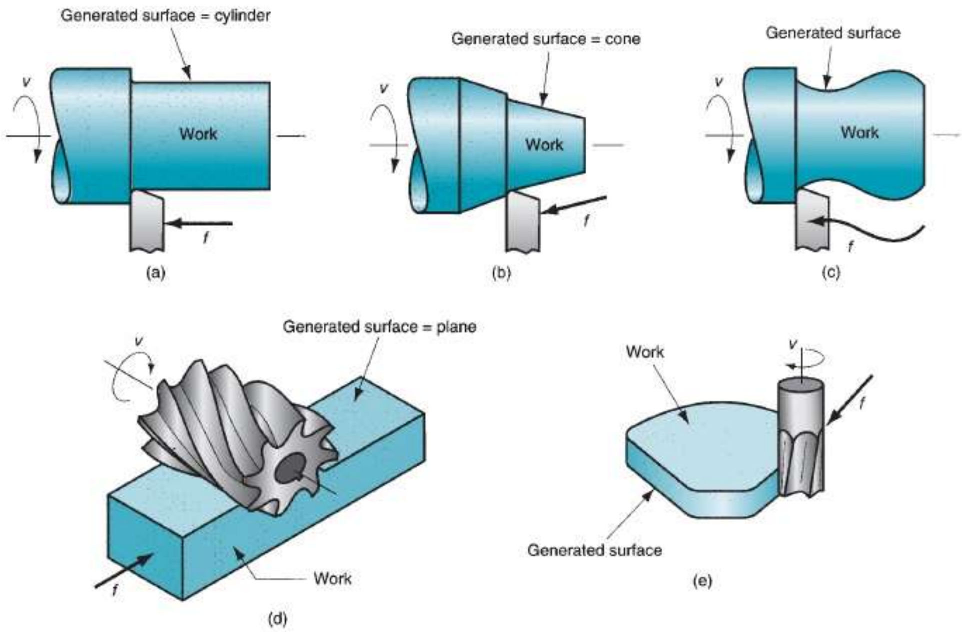

FIGURE 9.2 Generating shape in machining: (a) straight turning, (b) taper turning, (c) contour turning, (d) plain milling, and (e) profile milling.

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. Form turning, drilling, and broaching are examples of this case. In these operations, illustrated in Figure 9.3, the shape of the cutting tool is imparted to the work in order to create part geometry. The cutting conditions in forming usually include the primary speed motion combined with a feeding motion that is directed into the work.

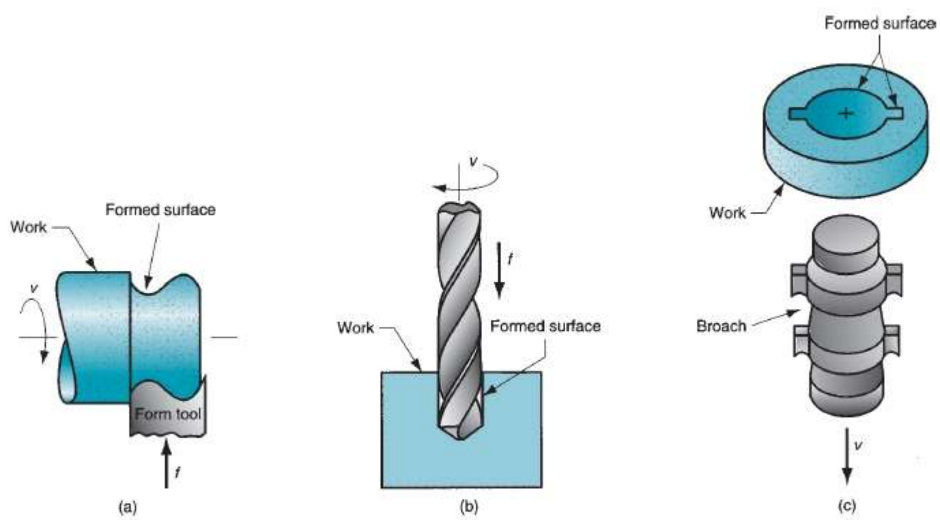

FIGURE 9.3 Forming to create shape in machining: (a) form turning, (b) drilling, and (c) broaching.

Depth of cut in this category of machining usually refers to the final penetration into the work after the feed motion has been completed.

Forming and generating are sometimes combined in one operation, as illustrated in Figure 9.4 for thread cutting on a lathe and slotting on a milling machine. 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 slot milling), the width of the cutter determines the width of the slot, but the feed motion creates the slot.

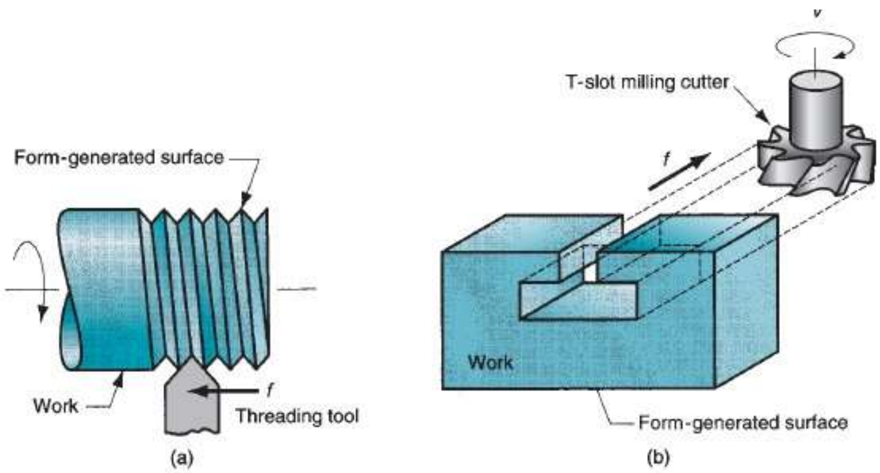

FIGURE 9.4 Combination of forming and generating to create shape: (a) thread cutting on a lathe, and (b) slot milling.

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.

9.2 Turning and Related Operations

Turning is a machining process in which a single-point tool removes material from the surface of a rotating workpiece. The tool is fed linearly in a direction parallel to the axis of rotation to generate a cylindrical geometry, as illustrated in Figures 9.2(a) and 9.5. 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.

9.2.1 Cutting Conditions in Turning

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 Do = original diameter of the part, m (ft).

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

D_f = Do - 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 fr = 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:

$T_m = \frac{\pi D_o L}{f v}$

Where Do = 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 Tm.

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.

9.2.2 Operations Related To Turning

A variety of other machining operations can be performed on a lathe in addition to turning; these include the following, illustrated in Figure 9.6:

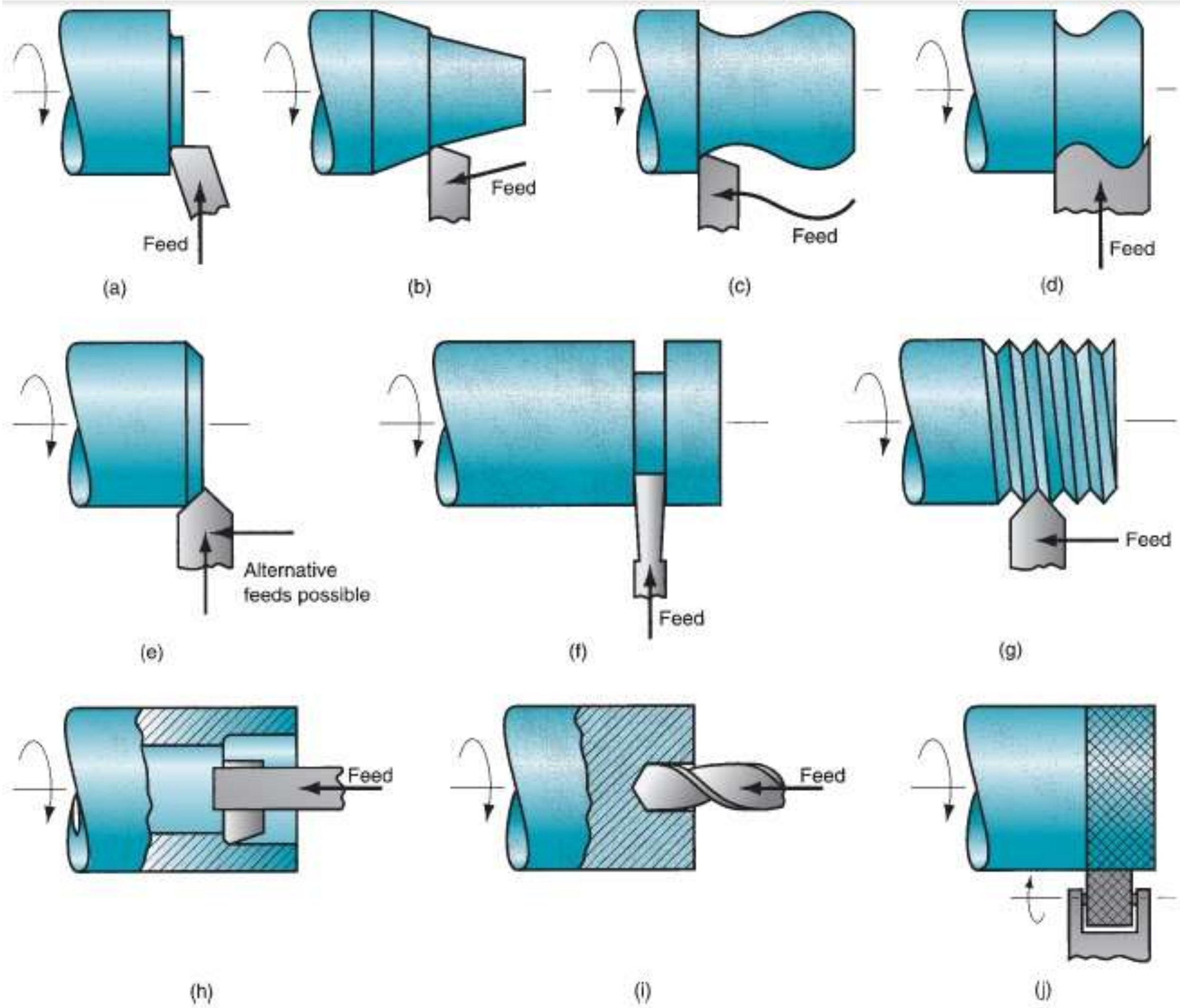

FIGURE 9.6 Machining operations other than turning that are performed on a lathe: (a) facing, (b) taper turning, (c) contour turning, (d) form turning, (e) chamfering, (f) cutoff, (g) threading, (h) boring, (i) drilling, and (j) knurling.

(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. A single-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.

Turning, facing, taper turning, contour turning, chamfering, and boring are all performed with single-point tools. A threading operation is accomplished using a single-point tool designed with a geometry that shapes the thread. Certain operations require tools other than single-point. Form turning is performed with a specially designed tool called a form tool. The profile shape ground into the tool establishes the shape of the workpart. A cutoff tool is basically a form tool. Drilling is accomplished by a drill bit.

Knurling is performed by a knurling tool, consisting of two hardened forming rolls, each mounted between centers. The forming rolls have the desired knurling pattern on their surfaces. To perform knurling, the tool is pressed against the rotating workpart with sufficient pressure to impress the pattern onto the work surface.

9.2.3 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 term engine dates from the time when these machines were driven by steam engines.

Engine Lathe Technology Figure 9.7 is a sketch of an engine lathe showing its principal components. 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. The ways are like tracks along which the carriage rides, and they are made with great precision to achieve a high degree of parallelism relative to the spindle axis.

The ways are built into the bed of the lathe, providing a rigid frame for the machine tool.

The carriage is driven by a leadscrew that rotates at the proper speed to obtain the desired feed rate. The cross-slide is designed to feed in a direction perpendicular to the carriage movement. Thus, by moving the carriage, the tool can be fed parallel to the work axis to perform straight turning; or by moving the cross-slide, the tool can be fed radially into the work to perform facing, form turning, or cutoff operations.

The conventional engine lathe and most other machines described in this section are horizontal turning machines; that is, the spindle axis is horizontal. This is appropriate for the majority of turning jobs, in which the length is greater than the diameter. For jobs in which the diameter is large relative to length and the work is heavy, it is more convenient to orient the work so that it rotates about a vertical axis; these are vertical turning machines.

The size of a lathe is designated by swing and maximum distance between centers.

The swing is the maximum workpart diameter that can be rotated in the spindle, determined as twice the distance between the centerline of the spindle and the ways of the machine. The actual maximum size of a cylindrical workpiece that can be accommodated on the lathe is smaller than the swing because the carriage and cross-slide assembly are in the way. The maximum distance between centers indicates the maximum length of a workpiece that can be mounted between headstock and tailstock centers. For example, a 350 mm x 1.2 m (14 in x 48 in) lathe designates that the swing is 350 mm (14 in) and the maximum distance between centers is 1.2 m (48 in).

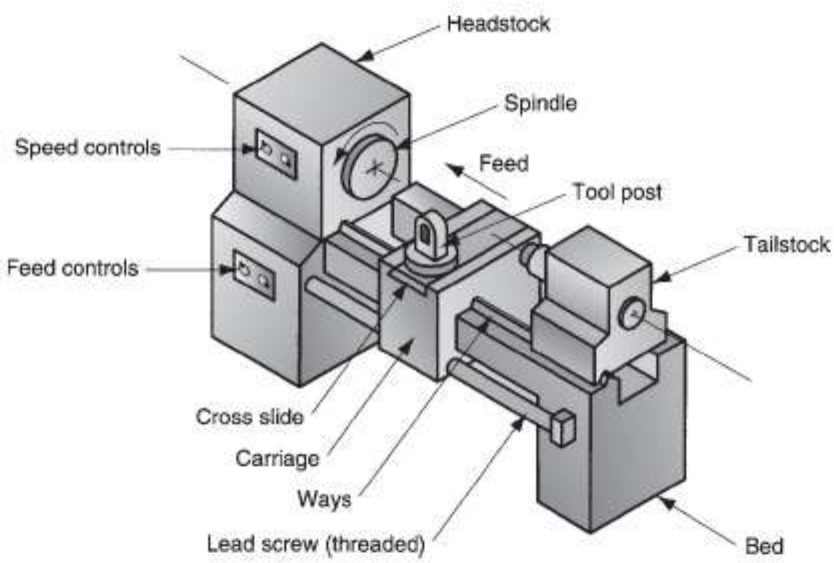

FIGURE 9.7 Diagram of an engine lathe, indicating its principal components.

Methods of Holding the Work in a Lathe: There are four common methods used to hold workparts in turning. These workholding methods consist of various mechanisms to grasp the work, center and support it in position along the spindle axis, and rotate it. The methods, illustrated in Figure 9.8, are (a) mounting the work between centers, (b) chuck, (c) collet, and (d) face plate.

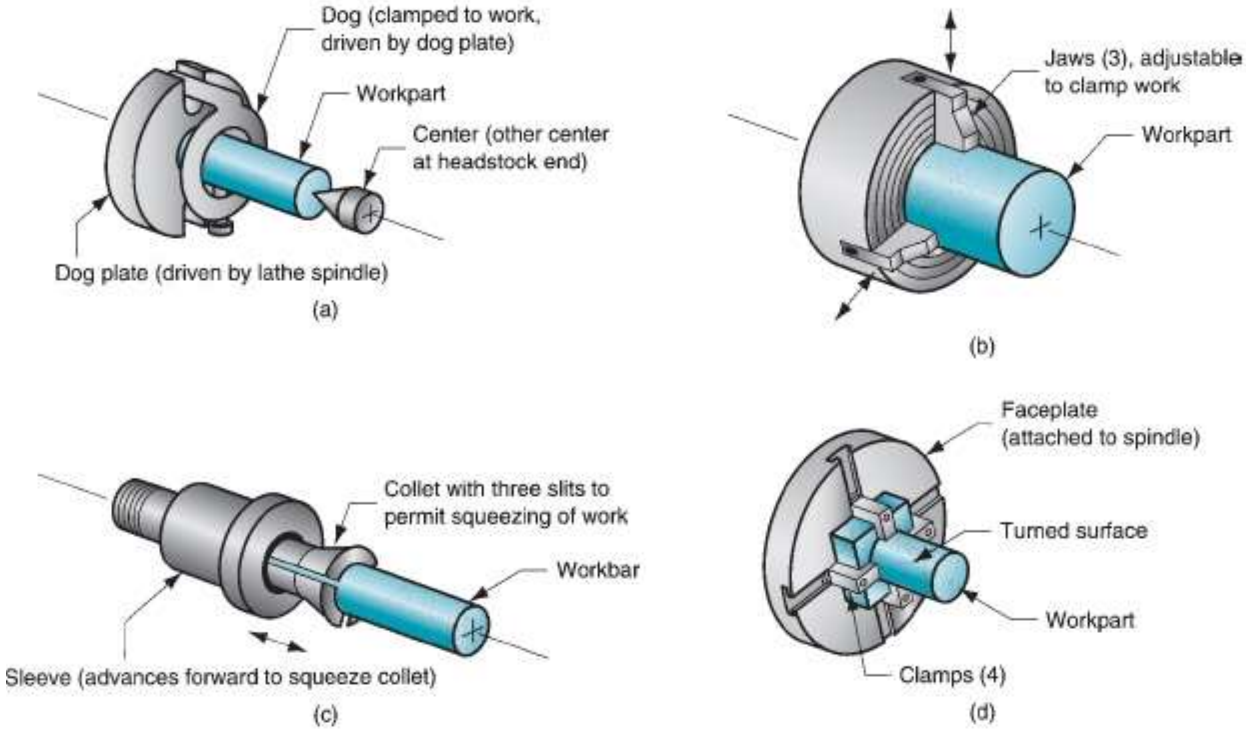

FIGURE 9.8 Four workholding methods used in lathes: (a) mounting the work between centers using a dog, (b) three-jaw chuck, (c) collet, and (d) faceplate for non-cylindrical workparts.

Holding the work between centers refers to the use of two centers, one in the headstock and the other in the tailstock, as in Figure 9.8(a). This method is appropriate for parts with large length-to-diameter ratios. At the headstock center, a device called a dog is attached to the outside of the work and is used to drive the rotation from the spindle. The tailstock center has a cone-shaped point which is inserted into a tapered hole in the end of the work. The tailstock center is either a ''live'' center or a ''dead'' center. A live center rotates in a bearing in the tailstock, so that there is no relative rotation between the work and the live center, hence, no friction between the center and the workpiece. In contrast, a dead center is fixed to the tailstock, so that it does not rotate; instead, the workpiece rotates about it. Because of friction and the heat buildup that results, this setup is normally used at lower rotational speeds. The live center can be used at higher speeds.

The chuck, Figure 9.8(b), is available in several designs, with three or four jaws to grasp the cylindrical workpart on its outside diameter. The jaws are often designed so they can also grasp the inside diameter of a tubular part. A self-centering chuck has a mechanism to move the jaws in or out simultaneously, thus centering the work at the spindle axis. Other chucks allow independent operation of each jaw. Chucks can be used with or without a tailstock center. For parts with low length-to-diameter ratios, holding the part in the chuck in a cantilever fashion is usually sufficient to withstand the cutting forces. For long workbars, the tailstock center is needed for support.

A collet consists of a tubular bushing with longitudinal slits running over half its length and equally spaced around its circumference, as in Figure 9.8(c). The inside diameter of the collet is used to hold cylindrical work such as barstock. Owing to the slits, one end of the collet can be squeezed to reduce its diameter and provide a secure grasping pressure against the work. Because there is a limit to the reduction obtainable in a collet of any given diameter, these workholding devices must be made in various sizes to match the particular workpart size in the operation.

A face plate, Figure 9.8(d), is a workholding device that fastens to the lathe spindle and is used to grasp parts with irregular shapes. Because of their irregular shape, these parts cannot be held by other workholding methods. The faceplate is therefore equipped with the custom-designed clamps for the particular geometry of the part.

9.2.4 Other Lathes and Turning Machines

In addition to the engine lathe, other turning machines have been developed to satisfy particular functions or to automate the turning process. Among these machines are: (1) toolroom lathe, (2) speed lathe, (3) turret lathe, (4) chucking machine, (5) automatic screw machine, and (6) numerically controlled lathe.

9.3 Drilling and Related Operations

Drilling, Figure 9.3(b), 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. The most common drill bit is the twist drill. The rotating drill feeds into the stationary workpart to form a hole whose diameter is equal to the drill diameter. Drilling is customarily performed on a drill press, although other machine tools also

9.3.1 Operations Related To Drilling

Several operations are related to drilling. These are illustrated in Figure 9.9 and described in this section. 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. Centering and spot facing are exceptions to this rule. All of the operations use rotating tools.

(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.

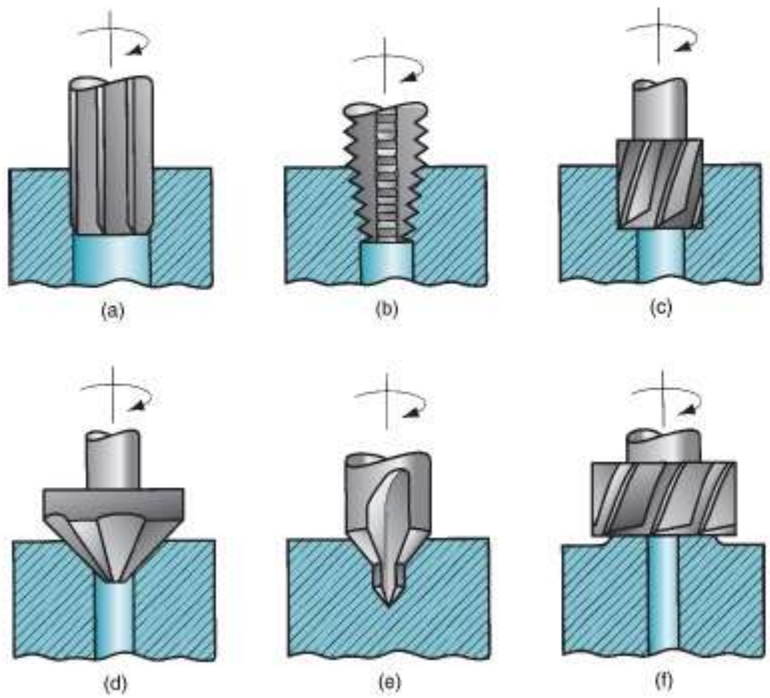

FIGURE 9.9

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

9.3.2 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, Figure 9.10. The upright drill stands on the floor and consists of a table for holding the workpart, a drilling head with powered spindle for the drill bit, and a base and column for support. A similar drill press, but smaller, is the bench drill, which is mounted on a table or bench rather than the floor.

The radial drill, Figure 9.11, is a large drill press designed to cut holes in large parts. It has a radial arm along which the drilling head can be moved and clamped. The head therefore can be positioned along the arm at locations that are a significant distance from the column to accommodate large work. The radial arm can also be swiveled about the column to drill parts on either side of the worktable.

The gang drill is a drill press consisting basically of two to six upright drills connected work in position. A fixture is a workholding device that is usually custom-designed for the particular workpart. The fixture can be designed to achieve higher accuracy in positioning the part relative to the machining operation, faster production rates, and greater operator convenience in use. A jig is a workholding device that is also specially designed for the workpart. The distinguishing feature between a jig and a fixture is that the jig provides a means of guiding the tool during the drilling operation. A fixture does not provide this tool guidance feature. A jig used for drilling is called a drill jig. together in an in-line arrangement. Each spindle is powered and operated independently, and they share a common worktable, so that a series of drilling and related operations can be accomplished in sequence (e.g., centering, drilling, reaming, tapping) simply by sliding the workpart along the worktable from one spindle to the next. A related machine is the multiple-spindle drill, in which several drill spindles are connected together to drill multiple holes simultaneously into the workpart.

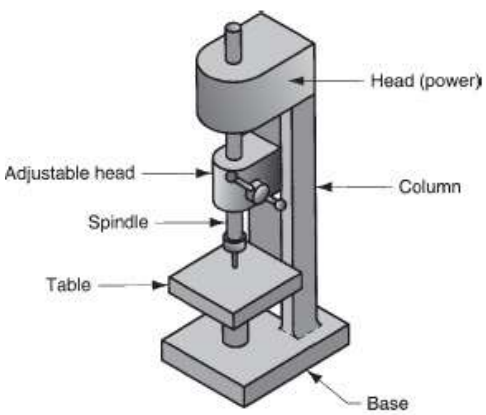

FIGURE 9.10 Upright drill press

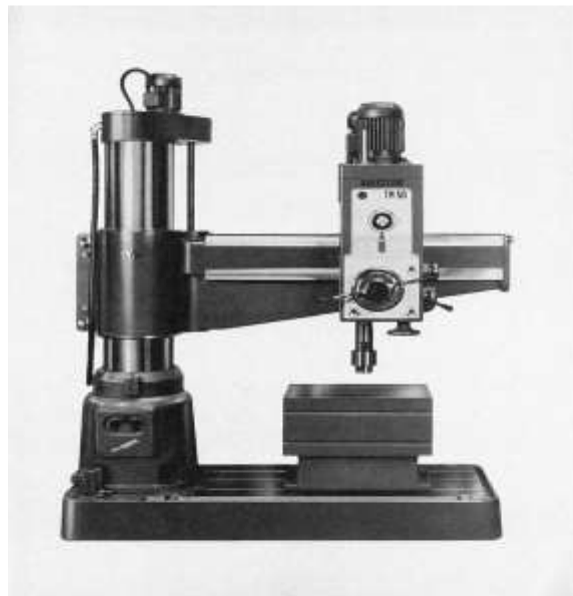

FIGURE 9.11 Radial drill press.

PRACTICE QUESTIONS

9.1. What are the differences between rotational parts and prismatic parts in machining?

9.2. Distinguish between generating and forming when machining workpart geometries.

9.3. Give two examples of machining operations in which generating and forming are combined to create workpart geometry.

9.4. Describe the turning process.

9.5. What is the difference between threading and tapping?

9.6. How does a boring operation differ from a turning operation?

9.7. What is meant by the designation 12 in x 36 in lathe?

9.8. Name the various ways in which a workpart can be held in a lathe.

9.9. 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?

9.10. How does a turret lathe differ from an engine lathe?

9.11. What is a blind hole?

9.12. What is the distinguishing feature of a radial drill press?