23 Economic and Product Design Considerations in Machining

Chapter Contents

23.1 Machinability

23.2 Tolerances and Surface Finish
23.2.1 Tolerances in Machining
23.2.2 Surface Finish in Machining

23.3 Selection of Cutting Conditions
23.3.1 Selecting Feed and Depth of Cut
23.3.2 Optimizing Cutting Speed

23.4 Product Design Considerations in Machining

This chapter concludes the coverage of conventional machining with several remaining topics. The first topic is machinability, which is concerned with how work material properties affect machining performance. The second topic is concerned with the tolerances and surface finishes (Chapter 5) that can be expected in machining processes. Third, the selection of cutting conditions (speed, feed, and depth of cut) in a machining operation is explored. This selection determines to a large extent the economic success of a given operation. Finally, some guidelines are provided for product designers to consider when they design parts that are to be produced by machining.

23.1 Machinability

Properties of the work material have a significant influence on the success of the machining operation. These properties and other characteristics of the work are often summarized in the term "machinability." ***Machinability*** denotes the relative ease with which a material (usually a metal) can be machined using appropriate tooling and cutting conditions.

There are various criteria used to evaluate machinability, the most important of which are (1) tool life, (2) forces and power, (3) surface finish, and (4) ease of chip disposal. Although machinability generally refers to the work material, it should be recognized that machining performance depends on more than just material. The type of machining operation, tooling, and cutting conditions are also important factors. In addition, the machinability criterion is a source of variation. One material may yield a longer tool life whereas another material provides a better surface finish. All of these factors make evaluation of machinability difficult.

Machinability testing usually involves a comparison of work materials. The machining performance of a test material is measured relative to that of a base (standard) material. Possible measures of performance in machinability testing include (1) tool life, (2) tool wear, (3) cutting force, (4) power in the operation, (5) cutting temperature, and (6) material removal rate under standard test conditions. The relative performance is expressed as an index number, called the machinability rating (MR). The base material used as the standard is given a machinability rating of 1.00. B1112 steel is often used as the base material in machinability comparisons. Materials that are easier to machine than the base have ratings greater than 1.00, and materials that are more difficult to machine have ratings less than 1.00. Machinability ratings are often expressed as percentages rather than index numbers. The following example illustrates how a machinability rating might be determined using a tool life test as the basis of comparison.

Example 23.1 Machinability

A series of tool life tests are conducted on two work materials under identical cutting conditions, varying only speed in the test procedure. The first material, defined as the base material, yields a Taylor Tool Life equation $vT^{0.28} = 350$, and the other material (test material) yields a Taylor equation $vT^{0.27} = 440$, where speed is in m/min and tool life is in min. Determine the machinability rating of the test material using the cutting speed that provides a 60-min tool life as the basis of comparison. This speed is denoted by v_{60}.

Solution: The base material has a machinability rating $= 1.0$. Its v_{60} value can be determined from the Taylor Tool Life equation as follows:

$$v_{60} = (350/60^{0.28}) = 111 \text{ m/min}$$

The cutting speed at a 60-min tool life for the test material is determined similarly:

$$v_{60} = (440/60^{0.27}) = 146 \text{ m/min}$$

Accordingly, the machinability rating can be calculated as

$$MR(\text{for the test material}) = \frac{146}{111} = 1.31 \ (\mathbf{131\%})$$

Many work material factors affect machining performance. Important mechanical properties include hardness and strength. As hardness increases, abrasive wear of the tool increases so that tool life is reduced. Strength is usually indicated as tensile strength, even though machining involves shear stresses. Of course, shear strength and tensile strength are correlated. As work material strength increases, cutting forces, specific energy, and cutting temperature increase, making the material more difficult to machine. On the other hand, very low hardness can be detrimental to machining performance. For example, low carbon steel, which has relatively low hardness, is often too ductile to machine well. High ductility causes tearing of the metal as the chip is formed, resulting in poor finish, and problems with chip disposal. Cold drawing is often used on low carbon bars to increase surface hardness and promote chip-breaking during cutting.

A metal's chemistry has an important effect on properties; and in some cases, chemistry affects the wear mechanisms that act on the tool material. Through these relationships, chemistry affects machinability. Carbon content has a significant effect on the properties of steel. As carbon is increased, the strength and hardness of the steel increases; this reduces machining performance. Many alloying elements added to steel to enhance properties are detrimental to machinability. Chromium, molybdenum, and tungsten form carbides in steel, which increase tool wear and reduce machinability. Manganese and nickel add strength and toughness to steel, which reduce machinability. Certain elements can be added to steel to improve machining

TABLE • 23.1 Approximate values of Brinell hardness and typical machinability ratings for selected work materials.

Work Material	Brinell Hardness (HB)	Machinability Rating[a]	Work Material	Brinell Hardness (HB)	Machinability Rating[a]
Base steel: B1112	180–220	1.00	Tool steel (unhardened)	200–250	0.30
Low carbon steel: C1008, C1010, C1015	130–170	0.50	Cast iron		
			Soft	60	0.70
Medium carbon steel: C1020, C1025, C1030	140–210	0.65	Medium hardness	200	0.55
			Hard	230	0.40
High carbon steel: C1040, C1045, C1050	180–230	0.55	Super alloys		
Alloy steels24[b]			Inconel	240–260	0.30
			Inconel X	350–370	0.15
1320, 1330, 3130, 3140	170–230	0.55	Waspalloy	250–280	0.12
4130	180–200	0.65	Titanium		
4140	190–210	0.55	Plain	160	0.30
4340	200–230	0.45	Alloys	220–280	0.20
4340 (casting)	250–300	0.25	Aluminum		
6120, 6130, 6140	180–230	0.50	2-S, 11-S, 17-S	soft	5.00[c]
8620, 8630	190–200	0.60	Aluminum alloys (soft)	soft	2.00[d]
B1113	170–220	1.35	Aluminum alloys (hard)	hard	1.25[d]
Free machining steels	160–220	1.50	Copper	soft	0.60
Stainless steel			Brass	soft	2.00[d]
301, 302	170–190	0.50	Bronze	soft	0.65[d]
304	160–170	0.40			
316, 317	190–200	0.35			
403	190–210	0.55			
416	190–210	0.90			

Values are estimated average values based on [1], [4], [5], [7], and other sources. Ratings represent relative cutting speeds for a given tool life (see Example 23.1).

[a] Machinability ratings are often expressed as percents (index number $\times$ 100%).

[b] The list of alloy steels is by no means complete. The table includes some of the more common alloys and indicates the range of machinability ratings among these steels.

[c] The machinability of aluminum varies widely. It is expressed here as $MR = 5.00$, but the range is probably from 3.00 to 10.00 or more.

[d] Aluminum alloys, brasses, and bronzes also vary significantly in machining performance. Different grades have different machinability ratings. For each case, we have attempted to reduce the variation to a single average value to indicate relative performance with other work materials.

performance, such as lead, sulfur, and phosphorus. The additives have the effect of reducing the coefficient of friction between the tool and chip, thereby reducing forces, temperature, and built-up edge formation. Better tool life and surface finish result from these effects. Steel alloys formulated to improve machinability are referred to as ***free machining steels*** (Section 6.2.3).

Similar relationships exist for other work materials. Table 23.1 lists selected metals and their approximate machinability ratings. These ratings are intended to summarize the machining performance of the materials.

23.2 Tolerances and Surface Finish

Machining operations are used to produce parts with defined geometries to tolerances and surface finishes specified by the product designer. This section examines these issues of tolerance and surface finish in machining.

23.2.1 TOLERANCES IN MACHINING

There is variability in any manufacturing process, and tolerances are used to set permissible limits on this variability (Section 5.1.1). Machining is often selected when tolerances are close, because it is more accurate than most other shape-making processes. Table 23.2 indicates typical tolerances that can be achieved for most machining operations examined in Chapter 21. It should be mentioned that the values in this tabulation represent ideal conditions, yet conditions that are readily achievable in a modern factory. If the machine tool is old and worn, process variability will likely

TABLE • 23.2 Typical tolerances and surface roughness values (arithmetic average) achievable in machining operations.

Machining Operation	Tolerance Capability—Typical mm	Tolerance Capability—Typical in	Surface Roughness AA—Typical μm	Surface Roughness AA—Typical μ-in	Machining Operation	Tolerance Capability—Typical mm	Tolerance Capability—Typical in	Surface Roughness AA—Typical μm	Surface Roughness AA—Typical μ-in
Turning, boring			0.8	32	Reaming			0.4	16
Diameter $D < 25$ mm	±0.025	±0.001			Diameter $D < 12$ mm	±0.025	±0.001		
25 mm $< D <$ 50 mm	±0.05	±0.002			12 mm $< D <$ 25 mm	±0.05	±0.002		
Diameter $D > 50$ mm	±0.075	±0.003			Diameter $D > 25$ mm	±0.075	±0.003		
Drilling*			0.8	32	Milling			0.4	16
Diameter $D < 2.5$ mm	±0.05	±0.002			Peripheral	±0.025	±0.001		
2.5 mm $< D <$ 6 mm	±0.075	±0.003			Face	±0.025	±0.001		
6 mm $< D <$ 12 mm	±0.10	±0.004			End	±0.05	±0.002		
12 mm $< D <$ 25 mm	±0.125	±0.005			Shaping, slotting	±0.025	±0.001	1.6	63
Diameter $D > 25$ mm	±0.20	±0.008			Planing	±0.075	±0.003	1.6	63
Broaching	±0.025	±0.001	0.2	8	Sawing	±0.50	±0.02	6.0	250

* Drilling tolerances are typically expressed as biased bilateral tolerances (e.g., $+0.010/-0.002$). Values in this table are expressed as closest bilateral tolerance (e.g., ± 0.006).

Compiled from various sources, including [2], [5], [7], [8], [12], and [15].

be greater than the ideal, and these tolerances would be difficult to maintain. On the other hand, newer machine tools can achieve closer tolerances than those listed.

Tighter tolerances usually mean higher costs. For example, if the product designer specifies a tolerance of ± 0.10 mm on a hole diameter of 6.0 mm, this tolerance could be achieved by a drilling operation, according to Table 23.2. However, if the designer specifies a tolerance of ± 0.025 mm, then an additional reaming operation is needed to satisfy this tighter requirement. This is not to suggest that looser tolerances are always good. It often happens that closer tolerances and lower variability in the machining of the individual components will lead to fewer problems in assembly, final product testing, field service, and customer acceptance. Although these costs are not always as easy to quantify as direct manufacturing costs, they can nevertheless be significant. Tighter tolerances that push a factory to achieve better control over its manufacturing processes may lead to lower total operating costs for the company over the long run.

23.2.2 SURFACE FINISH IN MACHINING

Because machining is often the manufacturing process that determines the final geometry and dimensions of the part, it is also the process that determines the part's surface texture (Section 5.3.2). Table 23.2 lists typical surface roughness values that can be achieved in various machining operations. These finishes should be readily achievable by modern, well-maintained machine tools.

The roughness of a machined surface depends on many factors that can be grouped as follows: (1) geometric factors, (2) work material factors, and (3) vibration and machine tool factors. The discussion of surface roughness in this section examines these factors and their effects.

Geometric Factors These are the machining parameters that determine the surface geometry of a machined part. They include (1) type of machining operation; (2) cutting tool geometry, most importantly nose radius; and (3) feed. The surface geometry that would result from these factors is referred to as the "ideal" or "theoretical" surface roughness, which is the finish that would be obtained in the absence of work material, vibration, and machine tool factors.

Type of operation refers to the machining process used to generate the surface. For example, peripheral milling, facing milling, and shaping all produce a flat surface; however, the surface geometry is different for each operation because of differences in tool shape and the way the tool interacts with the surface. A sense of the differences can be seen in Figure 5.14 showing various possible lays of a surface.

Tool geometry and feed combine to form the surface geometry. The shape of the tool point is the important tool geometry factor. The effects can be seen for a single-point tool in Figure 23.1. With the same feed, a larger nose radius causes the feed marks to be less pronounced, thus leading to a better finish. If two feeds are compared with the same nose radius, the larger feed increases the separation between feed marks, leading to an increase in the value of ideal surface roughness. If feed rate is large enough and the nose radius is small enough so that the end cutting edge participates in creating the new surface, then the end cutting-edge angle will affect surface geometry. In this case, a higher ECEA will result in a higher surface roughness value. In theory, a zero ECEA would yield a perfectly smooth surface; however, imperfections in the tool, work material, and machining process preclude achieving such an ideal finish.

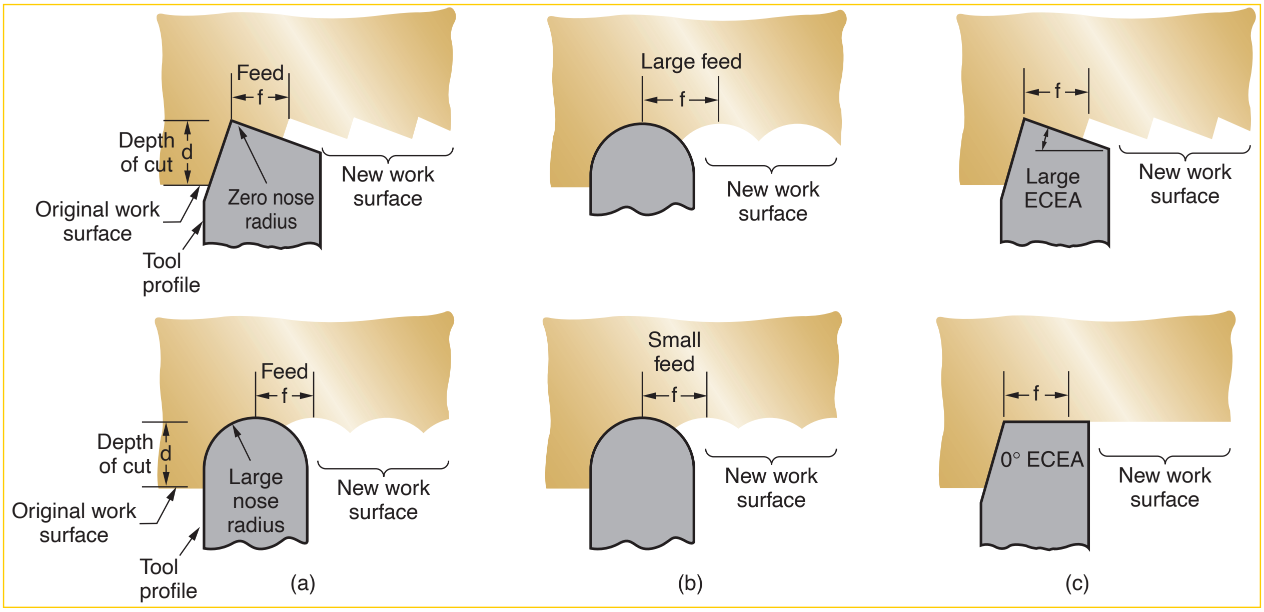

FIGURE 23.1 Effect of geometric factors in determining the theoretical finish on a work surface for single-point tools: (a) effect of nose radius, (b) effect of feed, and (c) effect of end cutting-edge angle.

The effects of nose radius and feed can be combined in an equation to predict the ideal average roughness for a surface produced by a single-point tool. The equation applies to operations such as turning, shaping, and planing:

$$R_i = \frac{f^2}{32NR} \tag{23.1}$$

where $R_i =$ theoretical arithmetic average surface roughness, mm (in); $f =$ feed, mm (in); and $NR =$ nose radius on the tool point, mm (in). The equation assumes that the nose radius is not zero and that feed and nose radius will be the principal factors that determine the geometry of the surface. The values for R_i will be in units of mm (in), which can be converted to μm (μ-in). Equation (23.1) can also be used to estimate the ideal surface roughness in face milling with insert tooling, using f to represent the chip load (feed per tooth).

Equation (23.1) assumes a sharp cutting tool. As the tool wears, the shape of the cutting point changes, which is reflected in the geometry of the work surface. For slight amounts of wear, the effect is not noticeable. However, when tool wear becomes significant, especially nose radius wear, surface roughness deteriorates compared with the ideal values given by Equation (23.1).

Work Material Factors Achieving the ideal surface finish is not possible in most machining operations because of factors related to the work material and its interaction with the tool. Work material factors that affect finish include (1) built-up edge effects—as the BUE cyclically forms and breaks away, particles are deposited on the newly created work surface, causing it to have a rough "sandpaper" texture; (2) damage to the surface caused by the chip curling back into the work; (3) tearing of the work surface during chip formation when machining ductile materials; (4) cracks in

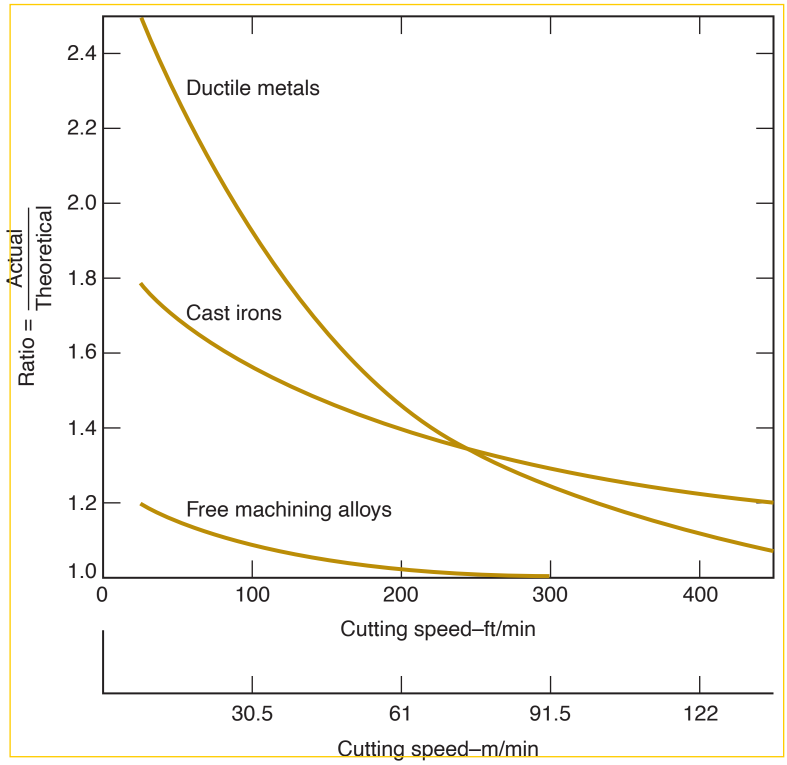

FIGURE 23.2 Ratio of actual surface roughness to ideal surface roughness for several classes of materials. (Source: General Electric Co. data [14].)

the surface caused by discontinuous chip formation when machining brittle materials; and (5) friction between the tool flank and the newly generated work surface. These work material factors are influenced by cutting speed and rake angle, such that an increase in cutting speed or rake angle generally improves surface finish.

The work material factors usually cause the actual surface finish to be worse than the ideal. An empirical ratio can be developed to convert the ideal roughness value into an estimate of the actual surface roughness value. This ratio takes into account BUE formation, tearing, and other factors. The value of the ratio depends on cutting speed as well as work material. Figure 23.2 shows the ratio of actual to ideal surface roughness as a function of speed for several classes of work material.

The procedure for predicting the actual surface roughness in a machining operation is to compute the ideal surface roughness value and then multiply this value by the ratio of actual to ideal roughness for the appropriate class of work material. This can be summarized as

$$R_a = r_{ai} R_i \tag{23.2}$$

where $R_a =$ the estimated value of actual roughness; $r_{ai} =$ ratio of actual to ideal surface finish from Figure 23.2, and $R_i =$ ideal roughness value from Equation (23.1).

Example 23.2 Surface roughness

A turning operation is performed on C1008 steel (a ductile steel) using a tool with a nose radius $= 1.2$ mm. Cutting speed $= 100$ m/min and feed $= 0.25$ mm/rev. Compute an estimate of the surface roughness in this operation.

Solution: The ideal surface roughness can be calculated from Equation (23.1):

$$R_i = (0.25)^2/(32 \times 1.2) = 0.0016 \text{ mm} = 1.6\ \mu\text{m}$$

From the chart in Figure 23.2, the ratio of actual to ideal roughness for ductile metals at 100 m/min is approximately 1.25. Accordingly, the actual surface roughness for the operation would be (approximately)

$$R_a = 1.25 \times 1.6 = \mathbf{2.0\ \mu m}$$

Vibration and Machine Tool Factors These factors are related to the machine tool, tooling, and setup in the operation. They include chatter or vibration in the machine tool or cutting tool; deflections in the fixturing, often resulting in vibration; and backlash in the feed mechanism, particularly on older machine tools. If these machine tool factors can be minimized or eliminated, the surface roughness in machining will be determined primarily by geometric and work material factors described above.

Chatter or vibration in a machining operation can result in pronounced waviness in the work surface. When chatter occurs, a distinctive noise occurs that can be recognized by any experienced machinist. Possible steps to reduce or eliminate vibration include (1) adding stiffness and/or damping to the setup, (2) operating at speeds that do not cause cyclical forces whose frequency approaches the natural frequency of the machine tool system, (3) reducing feeds and depths to reduce forces in cutting, and (4) changing the cutter design to reduce forces. Workpiece geometry can sometimes play a role in chatter. Thin cross sections tend to increase the likelihood of chatter, requiring additional supports to alleviate the condition.

23.3 Selection of Cutting Conditions

One of the practical problems in machining is selecting the proper cutting conditions for a given operation. This is one of the tasks in process planning (Section 39.1). For each operation, decisions must be made about machine tool, cutting tool(s), and cutting conditions. These decisions must give due consideration to work part machinability, part geometry, surface finish, and so forth.

23.3.1 SELECTING FEED AND DEPTH OF CUT

Cutting conditions in a machining operation consist of speed, feed, depth of cut, and cutting fluid (whether a cutting fluid should be used and, if so, what type of cutting fluid). Tooling considerations are usually the dominant factor in decisions about cutting fluids (Section 22.4). Depth of cut is often predetermined by workpiece geometry and operation sequence. Many jobs require a series of roughing operations followed by a final finishing operation. In the roughing operations, depth is made as large as possible within the limitations of available horsepower, machine tool and setup rigidity, strength of the cutting tool, and so on. In the finishing cut, depth is set to achieve the final dimensions for the part.

The problem then reduces to selection of feed and speed. In general, values of these parameters should be decided in the order: ***feed first***, ***speed second***. Determining the appropriate feed rate for a given machining operation depends on the following factors:

- ***Tooling***. What type of tooling will be used? Harder tool materials (e.g., cemented carbides, ceramics, etc.) tend to fracture more readily than high-speed steel. These tools are normally used at lower feed rates. HSS can tolerate higher feeds because of its greater toughness.
- ***Roughing or finishing***. Roughing operations involve high feeds, typically 0.5 to 1.25 mm/rev (0.020–0.050 in/rev) for turning; finishing operations involve low feeds, typically 0.125 to 0.4 mm/rev (0.005–0.015 in/rev) for turning.
- ***Constraints on feed in roughing***. If the operation is roughing, how high can the feed rate be set? To maximize metal removal rate, feed should be set as high as possible. Upper limits on feed are imposed by cutting forces, setup rigidity, and sometimes horsepower.
- ***Surface finish requirements in finishing***. If the operation is finishing, what is the desired surface finish? Feed is an important factor in surface finish, and computations like those in Example 23.2 can be used to estimate the feed that will produce a desired surface finish.

23.3.2 OPTIMIZING CUTTING SPEED

Selection of cutting speed is based on making the best use of the cutting tool, which normally means choosing a speed that provides a high metal removal rate yet suitably long tool life. Mathematical formulas have been derived to determine optimal cutting speed for a machining operation, given that the various time and cost components of the operation are known. The original derivation of these ***machining economics*** equations is credited to W. Gilbert [10]. The formulas allow the optimal cutting speed to be calculated for either of two objectives: (1) maximum production rate, or (2) minimum unit cost. Both objectives seek to achieve a balance between material removal rate and tool life. The formulas are based on a known Taylor Tool Life equation for the tool used in the operation. Accordingly, feed, depth of cut, and work material have already been set. The derivation will be illustrated for a turning operation. Similar derivations can be developed for other types of machining operations [3].

Maximizing Production Rate For maximum production rate, the speed that minimizes machining time per workpiece is determined. Minimizing cutting time per unit is equivalent to maximizing production rate. This objective is important in cases when the production order must be completed as quickly as possible.

In turning, there are three time elements that contribute to the total production cycle time for one part:

1. ***Part handling time*** T_h. This is the time the operator spends loading the part into the machine tool at the beginning of the production cycle and unloading the part after machining is completed. Any additional time required to reposition the tool for the start of the next cycle should also be included here.
2. ***Machining time*** T_m. This is the time the tool is actually engaged in cutting metal during the cycle.

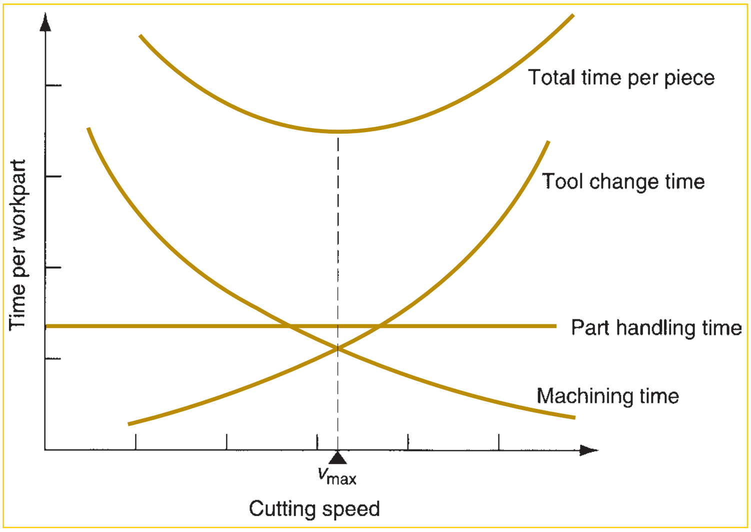

FIGURE 23.3 Time elements in a machining cycle plotted as a function of cutting speed. Total cycle time per piece is minimized at a certain value of cutting speed. This is the speed for maximum production rate.

3. ***Tool change time*** T_t. At the end of the tool life, the tool must be changed, which takes time. This time must be apportioned over the number of parts cut during the tool life. Let n_p = the number of pieces cut in one tool life (the number of pieces cut with one cutting edge until the tool is changed); thus, the tool change time per part $= T_t/n_p$.

The sum of these three time elements gives the total time per unit product for the operation cycle:

$$T_c = T_h + T_m + \frac{T_t}{n_p} \tag{23.3}$$

where T_c = production cycle time per piece, min; and the other terms are defined above.

The cycle time T_c is a function of cutting speed. As cutting speed is increased, T_m decreases and T_t/n_p increases; T_h is unaffected by speed. These relationships are shown in Figure 23.3.

The cycle time per part is minimized at a certain value of cutting speed. This optimal speed can be identified by recasting Equation (23.3) as a function of speed. Machining time in a straight turning operation is given by previous Equation (21.5):

$$T_m = \frac{\pi DL}{vf}$$

where T_m = machining time, min; D = work part diameter, mm (in); L = work part length, mm (in); f = feed, mm/rev (in/rev); and v = cutting speed, mm/min for consistency of units (in/min for consistency of units).

The number of pieces per tool n_p is also a function of speed. It can be shown that

$$n_p = \frac{T}{T_m} \tag{23.4}$$

where T = tool life, min/tool; and T_m = machining time per part, min/pc. Both T and T_m are functions of speed; hence, the ratio is a function of speed:

$$n_p = \frac{fC^{1/n}}{\pi DLv^{1/n-1}} \tag{23.5}$$

The effect of this relation is to cause the T_t/n_p term in Equation (23.3) to increase as cutting speed increases. Substituting Equations (21.5) and (23.5) into Equation (23.3) for T_c,

$$T_c = T_h + \frac{\pi DL}{fv} + \frac{T_t(\pi DLv^{1/n-1})}{fC^{1/n}} \tag{23.6}$$

The cycle time per piece is a minimum at the cutting speed at which the derivative of Equation (23.6) is zero:

$$\frac{dT_c}{dv} = 0$$

Solving this equation yields the cutting speed for maximum production rate in the operation:

$$v_{\max} = \frac{C}{\left[\left(\frac{1}{n} - 1\right) T_t\right]^n} \tag{23.7}$$

where $v_{\max}$ is expressed in m/min (ft/min). The corresponding tool life for maximum production rate is

$$T_{\max} = \left(\frac{1}{n} - 1\right) T_t \tag{23.8}$$

Minimizing Cost per Unit For minimum cost per unit, the speed that minimizes production cost per piece for the operation is determined. To derive the equations for this case, the four cost components that determine total cost of producing one part during a turning operation are identified:

1. ***Cost of part handling time***. This is the cost of the time the operator spends loading and unloading the part. Let $C_o =$ the cost rate (e.g., \$/min) for the operator and machine. Thus the cost of part handling time $= C_oT_h$.
2. ***Cost of machining time***. This is the cost of the time the tool is engaged in machining. Using C_o again to represent the cost per minute of the operator and machine tool, the cutting time cost $= C_oT_m$.
3. ***Cost of tool change time***. The cost of tool change time $= C_oT_t/n_p$.
4. ***Tooling cost***. In addition to the tool change time, the tool itself has a cost that must be added to the total operation cost. This cost is the cost per cutting edge C_t, divided by the number of pieces machined with that cutting edge n_p. Thus, tool cost per workpiece is given by C_t/n_p.

Tooling cost requires an explanation, because it is affected by different tooling situations. For disposable inserts (e.g., cemented carbide inserts), tool cost is determined as

$$C_t = \frac{P_t}{n_e} \tag{23.9}$$

where $C_t =$ cost per cutting edge, \$/tool life; $P_t =$ price of the insert, \$/insert; and $n_e =$ number of cutting edges per insert. This depends on the insert type; for example, triangular inserts that can be used only one side (positive rake tooling) have three edges/insert; if both sides of the insert can be used (negative rake tooling), there are six edges/insert; and so forth.

For regrindable tooling (e.g., high-speed steel solid shank tools, brazed carbide tools), the tool cost includes purchase price plus cost to regrind:

$$C_t = \frac{P_t}{n_g} + T_g C_g \tag{23.10}$$

where $C_t =$ cost per tool life, \$/tool life; $P_t =$ purchase price of the solid shank tool or brazed insert, \$/tool; $n_g =$ number of tool lives per tool, which is the number of times the tool can be ground before it can no longer be used (5–10 times for roughing tools and 10–20 times for finishing tools); $T_g =$ time to grind or regrind the tool, min/tool life; and $C_g =$ grinder's rate, \$/min.

The sum of the four cost components gives the total cost per unit product C_c for the machining cycle:

$$C_c = C_o T_h + C_o T_m + \frac{C_o T_t}{n_p} + \frac{C_t}{n_p} \tag{23.11}$$

C_c is a function of cutting speed, just as T_c is a function of v. The relationships for the individual terms and total cost as a function of cutting speed are shown in Figure 23.4. Equation (23.11) can be rewritten in terms of v to yield:

$$C_c = C_o T_h + \frac{C_o \pi D L}{fv} + \frac{(C_o T_t + C_t)(\pi D L v^{1/n - 1})}{fC^{1/n}} \tag{23.12}$$

The cutting speed that obtains minimum cost per piece for the operation can be determined by taking the derivative of Equation (23.12) with respect to v, setting it to zero, and solving for v_{min}:

$$v_{\text{min}} = C\left(\frac{n}{1-n} \cdot \frac{C_o}{C_o T_t + C_t}\right)^n \tag{23.13}$$

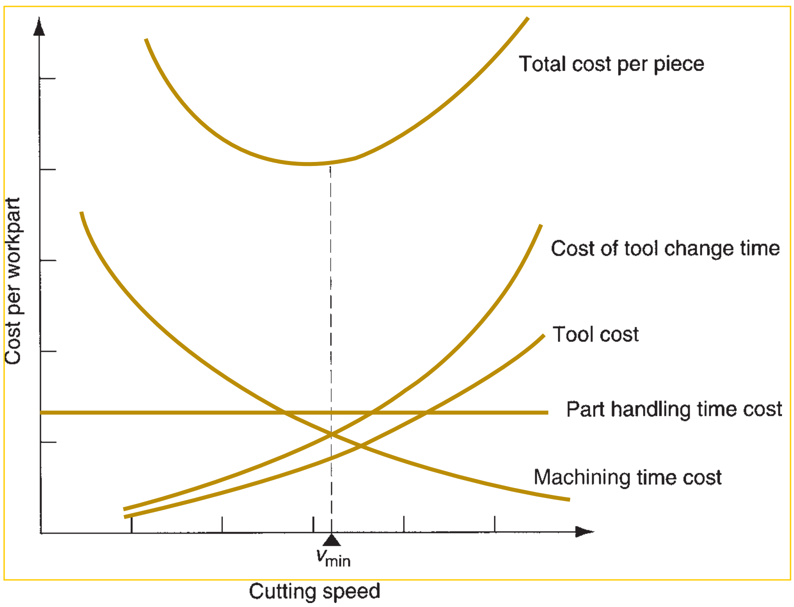

FIGURE 23.4 Cost components in a machining operation plotted as a function of cutting speed. Total cost per piece is minimized at a certain value of cutting speed. This is the speed for minimum cost per piece.

The corresponding tool life is given by

$$T_{\min} = \left(\frac{1}{n} - 1\right)\left(\frac{C_o T_t + C_t}{C_o}\right) \tag{23.14}$$

Example 23.3 Determining cutting speeds in machining economics

Suppose a turning operation is to be performed with HSS tooling on mild steel, with Taylor tool life parameters $n = 0.125$, $C = 70$ m/min (Table 23.2). Work part length = 500 mm and diameter = 100 mm. Feed = 0.25 mm/rev. Handling time per piece = 5.0 min, and tool change time = 2.0 min. Cost of machine and operator = \$30/hr, and tooling cost = \$3 per cutting edge. Find: (a) cutting speed for maximum production rate, and (b) cutting speed for minimum cost.

Solution: (a) Cutting speed for maximum production rate is given by Equation (23.7):

$$v_{\max} = 70\left(\frac{0.125}{0.875} \cdot \frac{1}{2}\right)^{0.125} = \mathbf{50\ m/min}$$

(b) Converting $C_o = \$30/\text{hr}$ to \$0.50/min, minimum cost cutting speed is given by Equation (23.13):

$$v_{\min} = 70\left(\frac{0.125}{0.875} \cdot \frac{0.50}{0.5(2) + 3.00}\right)^{0.125} = \mathbf{42\ m/min}$$

Example 23.4 Production rate and cost in machining economics

Determine the hourly production rate and cost per piece for the two cutting speeds computed in Example 23.3.

Solution: (a) For the cutting speed for maximum production, $v_{\max} = 50$ m/min, machining time per piece and tool life are calculated as follows:

$$\text{Machining time } T_m = \frac{\pi(0.5)(0.1)}{(0.25)(10^{-3})(50)} = 12.57 \text{ min/pc}$$

$$\text{Tool life } T = \left(\frac{70}{50}\right)^8 = 14.76 \text{ min/cutting edge}$$

The number of pieces per tool $n_p = 14.76/12.57 = 1.17$. Use $n_p = 1$. From Equation (23.3), average production cycle time for the operation is

$$T_c = 5.0 + 12.57 + 2.0/1 = 19.57 \text{ min/pc}$$

Corresponding hourly production rate $R_p = 60/19.57 = \mathbf{3.1\ pc/hr}$. From Equation (23.11), average cost per piece for the operation is

$$C_c = 0.50(5.0) + 0.50(12.57) + 0.50(2.0)/1 + 3.00/1 = \mathbf{\$12.79/pc}$$

(b) For the cutting speed for minimum production cost per piece, $v_{min} = 42$ m/min, the machining time per piece and tool life are calculated as follows:

$$\text{Machining time } T_m = \frac{\pi(0.5)(0.1)}{(0.25)(10^{-3})(42)} = 14.96 \text{ min/pc}$$

$$\text{Tool life } T = \left(\frac{70}{42}\right)^8 = 59.54 \text{ min/cutting edge}$$

The number of pieces per tool $n_p = 59.54/14.96 = 3.98 \rightarrow \text{Use } n_p = 3$ to avoid failure during the fourth workpiece. Average production cycle time for the operation is

$$T_c = 5.0 + 14.96 + 2.0/3 = 20.63 \text{ min/pc.}$$

Corresponding hourly production rate $R_p = 60/20.63 = \mathbf{2.9}$ **pc/hr**. Average cost per piece for the operation is

$$C_c = 0.50(5.0) + 0.50(14.96) + 0.50(2.0)/3 + 3.00/3 = \mathbf{\$11.32/pc}$$

Note that production rate is greater for v_{max} and cost per piece is minimum for v_{min}.

Some Comments on Machining Economics Some practical observations can be made relative to these optimum cutting speed equations. First, as the values of C and n increase in the Taylor tool life equation, the optimum cutting speed increases by either Equation (23.7) or Equation (23.13). Cemented carbides and ceramic cutting tools should be used at speeds that are significantly higher than for high-speed steel tools.

Second, as the tool change time and/or tooling cost (T_t and C_t) increase, the cutting speed equations yield lower values. Lower speeds allow the tools to last longer, and it is wasteful to change tools too frequently if either the cost of tools or the time to change them is high. An important effect of this tool cost factor is that disposable inserts usually possess a substantial economic advantage over regrindable tooling. Even though the cost per insert is significant, the number of edges per insert is large enough and the time required to change the cutting edge is low enough that disposable tooling generally achieves higher production rates and lower costs per unit product.

Third, v_{max} is always greater than v_{min}. The C_t/n_p term in Equation (23.13) has the effect of pushing the optimum speed value to the left in Figure 23.4, resulting in a lower value than in Figure 23.3. Rather than taking the risk of cutting at a speed above v_{max} or below v_{min}, some machine shops strive to operate in the interval between v_{min} and v_{max}—an interval sometimes referred to as the "high-efficiency range."

The procedures outlined for selecting feeds and speeds in machining are often difficult to apply in practice. The best feed rate is difficult to determine because the relationships between feed and surface finish, force, horsepower, and other constraints are not readily available for each machine tool. Experience, judgment, and experimentation are required to select the proper feed. The optimum cutting speed is difficult to calculate because the Taylor equation parameters C and n are not usually known without prior testing. Testing of this kind in a production environment is expensive.

23.4 Product Design Considerations in Machining

Several important aspects of product design have already been considered in the discussion of tolerance and surface finish (Section 23.2). In this section, some design guidelines for machining are presented, compiled from sources [1], [5], and [15]:

- If possible, parts should be designed that do not need machining. If this is not possible, then minimize the amount of machining required on the parts. In general, a lower-cost product is achieved through the use of net shape processes such as precision casting, closed die forging, or (plastic) molding; or near net shape processes such as impression die forging. Reasons why machining may be required include close tolerances; good surface finish; and special geometric features such as threads, precision holes, cylindrical sections with high degree of roundness, and similar shapes that cannot be achieved except by machining.
- Tolerances should be specified to satisfy functional requirements, but process capabilities should also be considered. See Table 23.2 for tolerance capabilities in machining. Excessively close tolerances add cost but may not add value to the part. As tolerances become tighter (smaller), product costs generally increase because of additional processing, fixturing, inspection, sortation, rework, and scrap.
- Surface finish should be specified to meet functional and/or aesthetic requirements, but better finishes generally increase processing costs by requiring additional operations such as grinding or lapping.
- Machined features such as sharp corners, edges, and points should be avoided; they are often difficult to accomplish by machining. Sharp internal corners require pointed cutting tools that tend to break during machining. Sharp external corners and edges tend to create burrs and are dangerous to handle.
- Deep holes that must be bored should be avoided. Deep hole boring requires a long boring bar. Boring bars must be stiff, and this often requires use of high modulus materials such as cemented carbide, which is expensive.
- Machined parts should be designed so they can be produced from standard available stock. Choose exterior dimensions equal to or close to the standard stock size to minimize machining; for example, rotational parts with outside diameters that are equal to standard bar stock diameters.
- Parts should be designed to be rigid enough to withstand forces of cutting and workholder clamping. Machining of long narrow parts, large flat parts, parts with thin walls, and similar shapes should be avoided if possible.
- Undercuts as in Figure 23.5 should be avoided because they often require additional setups and operations and/or special tooling; they can also lead to stress concentrations in service.
- Materials with good machinability should be selected by the designer (Section 23.1). As a rough guide, the machinability rating of a material correlates with the allowable cutting speed and production rate that can be used. Thus, parts made of materials with low machinability cost more to produce. Parts that are hardened by heat treatment must usually be finish ground or machined with higher cost tools after hardening to achieve final size and tolerance.

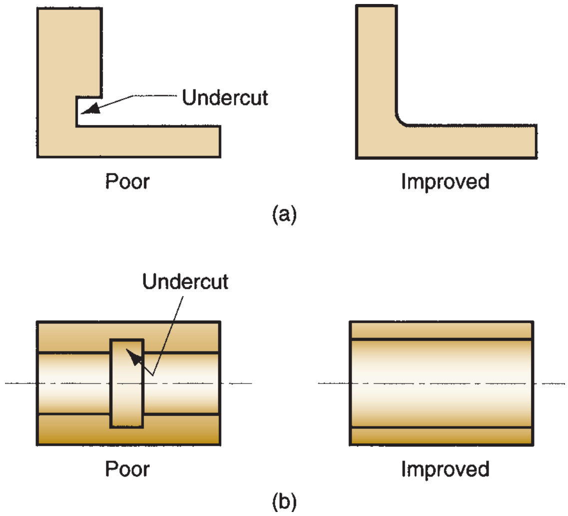

FIGURE 23.5 Two machined parts with undercuts: cross sections of (a) bracket and (b) rotational part. Also shown is how the part design might be improved.

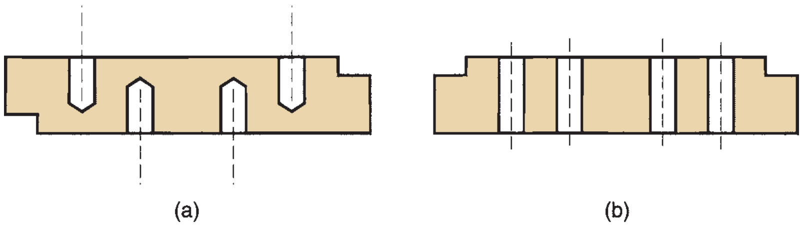

FIGURE 23.6 Two parts with similar hole features: (a) holes that must be machined from two sides, requiring two setups, and (b) holes that can all be machined from one side.

➢ Machined parts should be designed with features that can be produced in a minimum number of setups—one setup if possible. This usually means geometric features that can be accessed from one side of the part (see Figure 23.6).

➢ Machined parts should be designed with features that can be achieved with standard cutting tools. This means avoiding unusual hole sizes, threads, and features with unusual shapes requiring special form tools. In addition, it is helpful to design parts such that the number of individual cutting tools needed in machining is minimized; this often allows the part to be completed in one setup on a machine such as a machining center (Section 21.5).

References

[1] Bakerjian, R. (ed.). ***Tool and Manufacturing Engineers Handbook***. 4th ed. Vol VI, ***Design for Manufacturability***. Society of Manufacturing Engineers, Dearborn, Mich., 1992.

[2] Black, J, and Kohser, R. ***DeGarmo's Materials and Processes in Manufacturing***, 11th ed., John Wiley & Sons, Hoboken, New Jersey, 2012.

[3] Boothroyd, G., and Knight, W. A. ***Fundamentals of Metal Machining and Machine Tools***, 3rd ed. CRC Taylor & Francis, Boca Raton, Florida, 2006.

[4] Boston, O. W. ***Metal Processing***, 2nd ed. John Wiley & Sons, New York, 1951.

[5] Bralla, J. G. (ed.). ***Design for Manufacturability Handbook***, 2nd ed. McGraw-Hill, New York, 1998.

[6] Brierley, R. G., and Siekman, H. J. ***Machining Principles and Cost Control***. McGraw-Hill, New York, 1964.

[7] Drozda, T. J., and Wick, C. (eds.). ***Tool and Manufacturing Engineers Handbook***. 4th ed. Vol I, ***Machining***. Society of Manufacturing Engineers, Dearborn, Michigan, 1983.