Basics of Tolerance 120
Today's businesses face the challenge of maintaining a trained workforce. Companies must locate apprenticeship programs, cover travel and lodging costs, and...
Lesson: 1/12

Objectives
- Identify common tolerances in a manufacturing environment.
- Define tolerance.
- Identify how tolerance is determined.
- Compare tolerances possible in machining operations.
- Describe methods of describing tolerance.
- Identify advantages of different tolerance methods.
- Identify elements of tolerance for holes.
- Describe elements of surface tolerance.
- Identify the relationship between dimensions and tolerance.

Figure 1. Tolerance impacts how pieces fit together.
Lesson: 2/12

The Importance of Measurements
Imagine that one of the headlights of your car burns out. You go to the store, buy a replacement bulb, and install the light bulb when you return home. Of course, you would expect the light bulb to fit. We take for granted that the products we buy fit properly. However, the ability to create precise product sizes and dimensions requires time, energy, and a dedication to product quality.

Depending on the situation, a person may have a different idea of accuracy and precision. If you are planting flowers in a flowerbed and you want to space them one foot apart from each other, you may be content if you are off by an inch. By comparison, many manufactured products require sizes that are within a few thousandths of an inch. This is the same as the width of a typical human hair, which is 0.001 in. (0.025 mm), as shown in Figure 1.

Figure 1. Precision parts are measured within thousandths of an inch or smaller.
Lesson: 3/12

What Is Tolerance?
It is almost impossible to create a size or shape that is exactly the same as the intended dimension. Even if it is possible, the amount of time, money, and energy that goes into achieving that size or shape is often not worth the effort.

In order to strike a balance between the accuracy of a dimension and the required effort, manufacturers determine a small range of measurements that are acceptable. This range is called the tolerance for a given dimension. Tolerance is usually expressed as a range that is greater than and less than the original dimension. Figure 1 shows a drawing that contains a few tolerances.

Measurements are particularly important when parts are designed to fit together. Even the difference of a hair can affect the usefulness of precision parts.

Figure 1. The size of the part and the matching hole both require tolerances.
Lesson: 4/12

Tolerance and Use
Every product starts out as a blueprint drawing or computer image created by a designer. The designer chooses the degree of tolerance that is necessary for the part to function the way it is intended. For some items, such as the decorative object in Figure 1, tolerance is not terribly important. However, a precision part such as the locating pin in Figure 2 will certainly have tight tolerance restrictions.

If you purchase a pair of pants in your size but find they fit too tight, you have experienced improper tolerancing. A pair of pants could be off by an eighth of an inch and you might not notice. But eventually, there will be a measurement that deviates far enough from the original dimension that you will take notice and the pants will not fulfill the intended function.

Figure 1. A ceramic pot will not have strict tolerance restrictions.

Figure 2. This locating pin requires tight tolerances for its applications.
Lesson: 5/12

Tolerance and Cost
Parts that require a greater degree of accuracy cost more to produce. When a designer chooses
tolerance for a particular product, he or she must balance the intended use of the product with its
cost.

In other words, a tolerance is valuable if it improves the quality of a product. If a smaller tolerance
range does not affect the use of a product, then the added time and cost is not justified. The
three-inch nails you buy in the hardware store may vary in length by 0.01 in. (0.254 mm), but this
deviation does not affect the use of the nails. The quality of the nails does not change for
construction purposes.

However, if a special bolt, made with a diameter of 3.050 inches (7.75 cm), is designed to fit into
the paneling of a jet plane, even a difference of 0.001 in. (0.0254 mm) can affect the quality and
safety of that bolt. The time, money, and energy that it takes to produce that part is worth the
effort.

Figure 1. Railroad rails do not require extreme accuracy.

Figure 2. The aerospace and automotive industries require precision even for small parts.
Lesson: 6/12

Tolerances for Various Operations
Different manufacturing processes produce different tolerances. These processes can be ranked according to their typical tolerance limits. Figure 1 shows the tolerance ranges for some of the more common manufacturing processes.

Most casting operations produce the least precise dimensions. A part that has been cast can vary by several hundredths of an inch, or approximately 0.05 in. (1.3 mm). Machining operations such as turning, milling, and drilling produce tighter tolerances. Finally, abrasive operations such as grinding and lapping are used to produce dimensions that are even more accurate.

Manufacturing frequently involves a series of operations. Consequently, parts may have different tolerances for different features, as shown in Figure 2. For example, a cast part may be milled to produce a flatter surface within a hair’s thickness of 0.001 in. (0.03 mm), as in Figure 3. A machined surface may then be ground afterward to meet tighter tolerance restrictions within 0.0003 in. (0.008 mm). Because costs increase, additional machining should also increase the accuracy of the part.

Table 1: Tolerances for Common Manufacturing Processes

<table>
<thead>
<tr>
<th>Operation</th>
<th>in.</th>
<th>mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel sand casting</td>
<td>±0.060</td>
<td>±1.5</td>
</tr>
<tr>
<td>Cast iron sand casting</td>
<td>±0.050</td>
<td>±1.3</td>
</tr>
<tr>
<td>Drilling 0.250 in. hole</td>
<td>±0.003</td>
<td>±0.08</td>
</tr>
<tr>
<td>Milling</td>
<td>±0.003</td>
<td>±0.08</td>
</tr>
<tr>
<td>Turning</td>
<td>±0.002</td>
<td>±0.05</td>
</tr>
<tr>
<td>Grinding</td>
<td>±0.0003</td>
<td>±0.008</td>
</tr>
<tr>
<td>Lapping</td>
<td>±0.0002</td>
<td>±0.005</td>
</tr>
</tbody>
</table>

Figure 1. Every operation has an expected tolerance range.

Figure 2. Different surfaces may require different tolerances.

Figure 3. The machined outer edge is more accurate than the cast surface.
Lesson: 7/12

Types of Tolerance
There are certain methods for representing tolerance ranges. As you can see in Figure 1, each method has its advantages:

- **Bilateral tolerances** specify the acceptable measurements in two opposite directions from a given dimension.
- **Unilateral tolerances** specify the acceptable measurements in only one direction from a given dimension.
- **Limit dimensions** specify the acceptable measurements within two absolute dimensions.

Generally, a designer will choose a tolerancing method that represents a deviation in the less dangerous direction. If a variation to either side is equally dangerous, bilateral tolerances are preferred. If a variation in one direction is more dangerous than the other, unilateral tolerance is used.

Both bilateral and unilateral tolerancing specify a single dimension that acts as a central target for the operator. Limit dimensioning specifies a range of acceptable measurements instead of a target dimension.

Figure 1. Each tolerance method can be used in the same drawing.
Lesson: 8/12

Advantages of Limit Dimensions
Though bilateral tolerance is the most common method, tolerance limits offer their own advantages for certain operations. With bilateral tolerances, an operator must perform the calculations to determine the acceptable range of measurements. With limit dimensioning, the calculations have already been made for the operator.

Also, bilateral tolerance may occasionally prompt an operator to try to produce a measurement that is more accurate than necessary. This extra effort adds unnecessary cost to production. Instead, limit dimensioning creates a more realistic target for the operator; there is no perfect dimension to distract the operator.

In many ways, limits best suit human personalities. We are used to the idea of staying within an acceptable range. If you drive down the street, you do not follow a central dotted line and operate within a bilateral tolerance of that line. Instead, you stay with the acceptable limits marked by the lines on each side of the street.

Figure 1. Limit tolerances offer flexibility to the machinist.
Lesson: 9/12

Tolerances for a Hole
Tolerancing is extremely important when fitting a shaft and a hole together. The allowance is the space that is intended to exist between the shaft and hole. However, not all holes need to have an allowance.

For example, the wheel on a wheelbarrow is designed to spin freely on its axle. This requires a small allowance between the diameter of the shaft and the hole. A designer would set tolerances for both dimensions that would still allow space between the parts. The clearance is the difference between the largest acceptable shaft and the smallest acceptable hole diameter. An example is the part in Figure 1, which is designed to slide freely into the matching bushing.

By comparison, an interference fit is designed so that the shaft diameter is slightly larger than the hole. Consequently, the designer will choose different tolerances for the shaft and hole that do not leave an allowance. The parts must be forced together to assemble this tight fit. Both interference fits in Figure 2 prevent the bushings from sliding out of the plates.

Figure 1. A clearance fit allows the shank to slide freely in the bushing while an interference fit holds the bushing in the plate.

Figure 2. When assembled, interference fits hold the two bushings in their plates.
Lesson: 10/12

Surface Tolerance
Tolerances commonly describe the ranges of linear measurements. However, the surfaces of parts can vary in quality as well. Surface tolerance describes the relative smoothness and texture of a workpiece surface. As you can see in Figure 1, surface texture is commonly divided into these categories:

- **Roughness** describes closely spaced, uniform irregularities created by the cutting tool on the surface during a machining operation. It is the most commonly specified surface characteristic.
- **Waviness** describes uniform, repeating irregularities resulting from machining that are more widely spaced than roughness marks.
- **Lay** is the direction of the irregularities, which can be specified by the designer.
- **Flaws** are infrequent, random defects that appear on the surface. Scratches and cracks are surface flaws.

Keep in mind that surface tolerances should only be used for surfaces that affect the use of a part, such as the pin in Figure 2. Gears, pistons, and ball bearings especially require surface tolerances.

Figure 1. Multiple categories help define the surface tolerance of a part.

Figure 2. The exterior of the pin calls for a specific roughness.
Lesson: 11/12

Dimensioning and Tolerance

Dimensioning and tolerancing go hand in hand. When a designer creates the blueprint for a workpiece, he or she will include both the dimensions and the tolerances for that workpiece. The dimensions are the ideal measurements for the part, and the tolerances describe the realistic, actual parts that will eventually be made.

Manufacturers understand that we do not live in a perfect world. There is no such thing as the perfect part, nor should a manufacturer want to make one. What is important instead is the degree of quality necessary for the part to function properly. Tolerances allow product designers to specify the necessary amount of time and energy during manufacturing, and they provide inspectors with a realistic guideline for distinguishing acceptable and unacceptable parts.

Figure 1. Dimensions and tolerances help manufacturers make actual parts from blueprints.
Lesson: 12/12

Summary
Every part that is manufactured must meet certain specifications. Tolerances describe the range of acceptable measurements that a part can have and still perform its intended function. The dimension of a part represents its ideal measurement, and the tolerance represents the measurements for the parts that will actually be produced. Tolerances attempt to balance the use of a product with the cost required to produce that product.

Tolerance can be measured in a variety of ways. Bilateral tolerances specify the acceptable measurements in two opposite directions from a given dimension. Unilateral tolerances specify the acceptable measurements in only one direction from a given dimension. Limit dimensions specify the acceptable measurements within two absolute dimensions.

Tolerance ranges typically describe a linear measurement. They are especially important when determining the fit between a shaft and a hole. However, the texture of surfaces can also require a certain tolerance as well.

Figure 1. The dimension is the ideal measurement, and the tolerance is the acceptable range of deviation from the dimension.

Figure 2. Tolerances can be specified in multiple ways.
Describe the impact of tolerance on cost. Tolerances attempt to balance the use of quality necessary for the part to function properly. Tolerances allow product designers to specify a part, such as the pin in Figure 2. Gears, pistons, and ball bearings especially require certain tolerance as well. Tolerance ranges typically describe a linear measurement. They are especially important when acceptable measurements in only one direction from a given dimension. Limit dimensions specify the acceptable dimension of a part represents its ideal measurement, and the tolerance represents the realistic guideline for distinguishing acceptable and unacceptable parts.

Waviness

Flaws

Definition

Copyright © 2015 Tooling U, LLC. All Rights Reserved.