ME 114 – Engineering Drawing II
FITS, TOLERANCES and SURFACE QUALITY MARKS

Dr. A. Tolga Bozdana
Assistant Professor
Tolerancing

- Tolerances are used **to control the variation** that exists on all manufactured parts.
- Toleranced dimensions control the amount of variation on each part of an assembly.
- **The more accuracy needed in the machined part, the higher the manufacturing cost.** Therefore, tolerances must be specified in such a way that a product is functional as its cost is reasonable.

- Tolerances are assigned to mating parts. For instance, the slot in the part must accommodate another part (Fig. 1). Thus, a system is two or more mating parts.

- A tolerance of **4.650 ± 0.003** means that the final measurement of the machined part can be **between 4.653 and 4.647** and the part would still be acceptable.

- **The lower and upper allowable sizes** are referred to as **the limit dimensions**, and **the tolerance** is **the difference between these limits**. So, the upper limit (largest value) for the part is 4.653, the lower limit (smallest value) is 4.647, and the tolerance is 0.006.
Tolerance Representation

- **Tolerance is the total amount that a dimension may vary (i.e. the difference between the maximum and minimum limits).**

- As it is impossible to make everything to an exact size, tolerances are used on production drawings to control the manufacturing process more accurately and to control the variation between mating parts.

- Tolerances can be expressed in several ways:
  1. Direct limits or tolerance values applied directly to a dimension
  2. Geometric tolerances
  3. Notes referring to specific conditions
  4. A general tolerance note in the title block

**General Tolerances**

- General tolerances are given in a note or as part of the title block. A general tolerance note would be similar to:

  
  **ALL DECIMAL DIMENSIONS TO BE HELD TO ± .002"**

This means that a dimension (such as 0.500) would be assigned a **tolerance of ±.002**, resulting in an **upper limit of 0.502** and a **lower limit of 0.498**.
Tolerance Representation

Limit Dimensions

- Tolerances can be applied directly to dimensioned features: **the maximum and minimum sizes are specified as part of the dimension (Fig. 2A)**. The upper limit is placed above the lower limit and a minimum spacing of 1.5 mm is required between them.

Plus and Minus Dimensions

- With this approach, the basic size is given followed by a **plus/minus sign and the tolerance value (Fig. 2B)**. Tolerances can be unilateral (varying in only one direction) or bilateral (varying in both directions) as shown in **Fig. 3**.

- If **the variation is equal in both directions**, then the variation is preceded by a **± symbol**. Note that this approach can only be used when the two variations are equal.
Fig. 4 shows a system of two parts with toleranced dimensions.

- **Nominal size**: A dimension used to describe the general size, usually expressed in common fractions. The slot has a nominal size of 0.500".
- **Basic size**: The theoretical size used as a starting point for the application of tolerances. The basic size of the slot is 0.500".
- **Actual size**: The measured size of the finished part after machining. The actual size of the slot is 0.501".
- **Limits**: The maximum and minimum sizes shown by the toleranced dimension. The slot has limits of 0.502 and 0.498, and the mating part has limits of 0.495 and 0.497.
**Terminology in Tolerancing**

- **Allowance:** The minimum clearance or the maximum interference between parts (i.e. the tightest fit between two mating parts). The tightest fit occurs when the slot is machined to its smallest allowable size (0.498) and the mating part is machined to its largest allowable size (0.497). The difference between them (0.001) is the allowance.

- **Tolerance:** The total allowable variance in a dimension (i.e. the difference between upper and lower limits). The tolerance of the slot is 0.004" (0.502-0.498) and the tolerance of the mating part is 0.002" (0.497-0.495).

![Figure 4](image-url)
> **Maximum Material Condition (MMC):** The condition of a part when it contains the greatest amount of material. MMC of an external feature (the mating part) is the upper limit. MMC of an internal feature (the slot) is the lower limit.

> **Least Material Condition (LMC):** The condition of a part when it contains the least amount of material possible. LMC of an external feature (the mating part) is the lower limit. LMC of an internal feature (the slot) is the upper limit.

> **Part tolerance:** The difference between upper and lower limits of a single part.

> **System tolerance:** The sum of tolerances of all parts.
Fit Types

- The degree of tightness between mating parts is called the fit. There are three most common types of fit based on basic hole and shaft system found in industry.

Clearance Fit: two tolerated mating parts will always leave a space or clearance

- In Fig. 5, the largest dimension of shaft A can be 0.999, and the smallest hole is 1.000.
- The shaft will always be smaller than the hole with the minimum clearance (allowance) of +0.001. The maximum clearance occurs when the smallest shaft (0.998) is mated with the largest hole (1.001) which results in a difference of +0.003.

![Diagram of clearance fit](image)
Fit Types

**Interference Fit:** two tolerated mating parts will always interfere

- An interference fit fixes or anchors one part into the other, as if the parts were one. In Fig. 5, the smallest dimension of shaft B can be 1.002 and the largest hole is 1.001.

- The shaft will always be larger than the hole with **the minimum interference of -0.001**. When the smallest hole (1.000) is mated with the largest shaft (1.003), **the maximum interference of -0.003** would occur.

- To assemble the parts, it would be necessary to **stretch the hole** or **shrink the shaft** or to **use force to press the shaft into the hole**. This fit is desirable for some design applications (such as fastening two parts together without the use of mechanical fasteners or adhesive).

---

![Diagram of Clearance and Interference Fits](image-url)
**Fit Types**

**Transition Fit:** two tolerated mating parts are sometimes an interference fit and sometimes a clearance fit when assembled

- In *Fig. 6*, the smallest shaft dimension can be 0.998 and the largest hole is 1.001, resulting in a clearance fit with a clearance of +0.003.

- The largest shaft dimension can be 1.002 and the smallest hole is 1.000, resulting in an interference fit with an interference of -0.002.

![Diagram](image-url)
Geometrical Tolerances

- The specification of **positional and form (geometrical) tolerances** is a complex procedure, as in many cases the geometric characteristics of a part must be given in great detail.

- The characteristics and features of drawing objects (such as flatness, straightness, roundness, cylindricity, parallelism, perpendicularity, and so on) may be tolerated on a drawing by symbols, which can be found in **TS 1304 standards suggested by Turkish Standardization Institution (TSE)**.

- **Form tolerances** can be applied to a **single geometric shape** (for instance, to specify flatness, only one surface has to be tolerated). Form may include the **relationship between two or more features** on the same part (for example, parallelism involves the relationship of one plane to another on the same part and the latter being used as a datum plane).

- **A datum** is a feature of a part that **acts as a master reference** used to locate other features of the part. A datum can be a point, a line or a plane which is chosen for its functional nature. Any datum should be readily available feature like a finished surface on a bench vise.

- **Positional tolerances**: are applied to such dimensions that are used **to locate or position geometric shapes** with respect to specified datum planes.
# Geometrical Tolerances

<table>
<thead>
<tr>
<th>TYPE</th>
<th>SYMBOL</th>
<th>AS IN DRAWING</th>
<th>TOLERANCE ZONE</th>
<th>EXPLANATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>FORM</td>
<td>STRAIGHTNESS</td>
<td>[Diagram]</td>
<td>TWO PARALLEL LINES .004 APART</td>
<td>A surface is straight when all its elements are the straight lines. The symbol is shown in the first compartment of the box and the tolerance in the next.</td>
</tr>
<tr>
<td></td>
<td>FLATNESS</td>
<td>[Diagram]</td>
<td>TWO PARALLEL PLANES .004 APART</td>
<td>A surface is flat when all its elements are in one plane. Flatness tolerance specifies a tolerance zone between two parallel planes. Note that the symbol is followed by the tolerance.</td>
</tr>
<tr>
<td></td>
<td>CIRCULARITY (ROUNDNESS)</td>
<td>[Diagram]</td>
<td>TWO CONCENTRIC CIRCLES .004 APART</td>
<td>Circularity tolerance controls only points on a surface intersected by any plane perpendicular to the part’s axis. This tolerance is required on cylinders, cones and spheres. The amount of ovality must lie within the tolerance zone.</td>
</tr>
<tr>
<td></td>
<td>CYLINDRICITY</td>
<td>[Diagram]</td>
<td>TWO CONCENTRIC CYLINDERS .004 APART</td>
<td>Cylindricity involves 2D area wrapping around a surface. Cylindricity tolerance defines a tolerance zone consisting of two concentric cylinders. It is used for cylinders only, and it controls taper, roundness and straightness.</td>
</tr>
</tbody>
</table>
### Geometrical Tolerances

<table>
<thead>
<tr>
<th>TYPE</th>
<th>SYMBOL</th>
<th>AS IN DRAWING</th>
<th>TOLERANCE ZONE</th>
<th>EXPLANATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>ORIENTATION</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PARALLELISM</td>
<td><img src="image1" alt="Parallelism Symbol" /></td>
<td><img src="image2" alt="Parallelism Drawing" /></td>
<td><img src="image3" alt="Parallelism Tolerance Zone" /></td>
<td>Parallelism exists when a feature is at constant distance from a datum. The feature could be an axis or a plane, as could be datum. The symbol, the tolerance and the datum letter must be shown in the box.</td>
</tr>
<tr>
<td>PERPENDICULARITY</td>
<td><img src="image4" alt="Perpendicularity Symbol" /></td>
<td><img src="image5" alt="Perpendicularity Drawing" /></td>
<td><img src="image6" alt="Perpendicularity Tolerance Zone" /></td>
<td>This can control perpendicularity of a plane with a plane, an axis to a plane or an axis to an axis.</td>
</tr>
<tr>
<td>ANGULARITY</td>
<td><img src="image7" alt="Angularity Symbol" /></td>
<td><img src="image8" alt="Angularity Drawing" /></td>
<td><img src="image9" alt="Angularity Tolerance Zone" /></td>
<td>Angularity relates to an axis or surface at some specified angle to a datum. The angle may not be 90°, where such a case is covered by the perpendicularity.</td>
</tr>
</tbody>
</table>
# Geometrical Tolerances

<table>
<thead>
<tr>
<th>TYPE</th>
<th>SYMBOL</th>
<th>AS IN DRAWING</th>
<th>TOLERANCE ZONE</th>
<th>EXPLANATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>PROFILE</td>
<td><img src="image1.png" alt="Profile of a Line" /></td>
<td><img src="image2.png" alt="Profile of a Line" /></td>
<td><img src="image3.png" alt="Profile of a Line" /></td>
<td>The profile of any part is outline as seen in any 2D view. Profiles contain combinations of lines and arcs. Feature control frame contains the tolerance zone to be applied to the basic dimension.</td>
</tr>
<tr>
<td>PROFILE</td>
<td><img src="image4.png" alt="Profile of a Surface" /></td>
<td><img src="image5.png" alt="Profile of a Surface" /></td>
<td><img src="image6.png" alt="Profile of a Surface" /></td>
<td>This type is applied to a surface, rather than a line. It is actually more common than the line profile tolerancing.</td>
</tr>
<tr>
<td>RUNOUT</td>
<td><img src="image7.png" alt="Circular Runout" /></td>
<td><img src="image8.png" alt="Circular Runout" /></td>
<td><img src="image9.png" alt="Circular Runout" /></td>
<td>Runout is a measure of deviation from perfect form, determined as a part revolved around its axis. Runout tolerance is a combined tolerance that incorporates variations in straightness, roundness and parallelism. Surface features are related to a datum axis. Features controlled may be those wrapped around the datum axis or may be those perpendicular to the datum axis.</td>
</tr>
<tr>
<td>TOTAL RUNOUT</td>
<td><img src="image10.png" alt="Total Runout" /></td>
<td><img src="image11.png" alt="Total Runout" /></td>
<td><img src="image12.png" alt="Total Runout" /></td>
<td></td>
</tr>
</tbody>
</table>
# Geometrical Tolerances

<table>
<thead>
<tr>
<th>TYPE</th>
<th>SYMBOL</th>
<th>AS IN DRAWING</th>
<th>TOLERANCE ZONE</th>
<th>EXPLANATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOCATION</td>
<td><img src="image" alt="Position Symbol" /></td>
<td><img src="image" alt="Position Drawing" /></td>
<td><img src="image" alt="Position Tolerance Zone" /></td>
<td>True position is the location of holes or slots with respect to surfaces. The center of the hole must be on the theoretical interactions of three given axes within the tolerance zone of 0.014 mm. Note that the letter (M) refers to Maximum Material Condition (MMC).</td>
</tr>
<tr>
<td>LOCATION</td>
<td><img src="image" alt="Concentricity Symbol" /></td>
<td><img src="image" alt="Concentricity Drawing" /></td>
<td><img src="image" alt="Concentricity Tolerance Zone" /></td>
<td>Concentricity controls the extent to which the axis of one cylinder is collinear with the axis on an adjoining cylinder.</td>
</tr>
<tr>
<td>LOCATION</td>
<td><img src="image" alt="Symmetry Symbol" /></td>
<td><img src="image" alt="Symmetry Drawing" /></td>
<td><img src="image" alt="Symmetry Tolerance Zone" /></td>
<td>A part or a feature is symmetric when it has the same contour and size on opposite sides of a control plane. Thus, a symmetry tolerance locates or positions the features with respect to a datum plane.</td>
</tr>
</tbody>
</table>
Surface Texture

- Many parts operating within close tolerances require a special surface finish, which needs to be accurately specified in a technical drawing.

- Texture of a part surface is illustrated in Fig. 7. The pattern of surface lines is called lay. When the surface is cut using a perpendicular plane, 2D profile of the surface is obtained. This profile consists of roughness and waviness profiles which can be suppressed as shown in Fig. 7.

- The roughness value, which is denoted by $R_a$ (µm), and the sampling length can be measured on the roughness profile. Thus, a symbol is accompanied by notes in order to mark surface quality required as below (according to ISO 1302-2002 standards).

![Figure 7](image.png)

- **a** Roughness value ($R_a$ value)
- **b** Method of production, treatment, coating
- **c** Sampling length (in µm)
- **d** Direction of lay
- **e** Machining allowance
Surface Quality Marks

Instead of $R_a$ value, standard numbers designating roughness values can also be used:

\[
R_a (\mu m): \quad 50 \quad 25 \quad 12.5 \quad 6.3 \quad 3.2 \quad 1.6 \quad 0.8 \quad 0.4 \quad 0.2 \quad 0.1 \quad 0.05 \quad 0.025
\]

Roughness Number: N12 N11 N10 N9 N8 N7 N6 N5 N4 N3 N2 N1

The information about material removal from a surface is defined by different symbols:

- Material removal by machining is required.

- Material removal is prohibited.

The table shows the symbols used for the lays with various surface patterns and the description of each pattern.

<table>
<thead>
<tr>
<th>Lay symbol</th>
<th>Surface pattern</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>=</td>
<td></td>
<td>Lay is parallel to line representing surface to which symbol is applied.</td>
</tr>
<tr>
<td>(\perp)</td>
<td></td>
<td>Lay is perpendicular to line representing surface to which symbol is applied.</td>
</tr>
<tr>
<td>(\times)</td>
<td></td>
<td>Lay is angular in both directions to line representing surface to which symbol is applied.</td>
</tr>
<tr>
<td>M</td>
<td></td>
<td>Lay is multidirectional.</td>
</tr>
<tr>
<td>C</td>
<td></td>
<td>Lay is circular relative to center of surface to which symbol is applied.</td>
</tr>
<tr>
<td>R</td>
<td></td>
<td>Lay is approximately radial relative to the center of the surface to which symbol is applied.</td>
</tr>
<tr>
<td>P</td>
<td></td>
<td>Lay is particulate, nondirectional, or protuberant.</td>
</tr>
</tbody>
</table>

Common surface finishing methods are:

Machining, Grinding, Plating, Honing, Lapping, Chemical, etc.
Examples
Examples