COMPUTER NUMERICAL CONTROL OF MACHINE TOOLS

Laboratory for Manufacturing Systems and Automation
Department of Mechanical Engineering and Aeronautics
University of Patras, Greece

Dr. Dimitris Mourtzis
Assistant Professor

Patras 2016
Chapter 7: Tolerancing
# Table of Contents

## Chapter 7: Tolerancing

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.1 Tolerancing for Interchangeability</td>
<td>10</td>
</tr>
<tr>
<td>7.2 Tolerance Types</td>
<td>16</td>
</tr>
<tr>
<td>7.3 General Definitions</td>
<td>22</td>
</tr>
<tr>
<td>7.4 Tolerancing Standards</td>
<td>29</td>
</tr>
<tr>
<td>7.5 Inch Tolerances</td>
<td>32</td>
</tr>
<tr>
<td>7.6 Metric Tolerances</td>
<td>50</td>
</tr>
<tr>
<td>7.7 Selecting Tolerances</td>
<td>67</td>
</tr>
<tr>
<td>7.8 Tolerance Accumulation</td>
<td>72</td>
</tr>
<tr>
<td>7.9 Formatting Tolerances</td>
<td>90</td>
</tr>
</tbody>
</table>
Objectives

- Learn about **tolerancing** and how important this technique is, to mass production

- Learn various **tolerancing methods**

- Learn about **tolerancing standards** and the most common standards agencies

- And, ways of formatting **inch and metric tolerances**
• After completing the construction of a piece, its **dimensions appear to deviate** compared to its nominal values.

• These variations depend on the **accuracy of the machine-tools used** and the **available measuring devices**.

• **Minor deviations**, depending on the intended use of the piece, **may be tolerated** without destroying its functionality.

• **Tolerances** are defined as the **permissible deviations** from the prescribed shape, size or position of an element in one piece with respect to the corresponding ones of the drawing.
Tolerances

- In most of the cases the craftsman who undertakes the construction of a piece is unable to reproduce the exact dimensions as outlined in the drawing.

- For this reason constructional drawings indicate the permissible deviations from the nominal dimension.

Example:

A given dimension in a drawing is 1.50 ± .04 mm, meaning that the piece should have a particular dimension between 1.46 and 1.54 mm, and that the possible permissible tolerance on this dimension is 0.08 mm.
Greater accuracy requires greater cost

- For this reason **not all the parts of a product are constructed with the same accuracy** (same tolerances)

- The selection of the desired **accuracy** of a product’s part **depends on its final use** (assembly with other parts, e.g., shaft and hole, etc.).

- **Tolerances** are **heavily dependent on the machine tool** used for the manufacturing of the piece

- The following figure presents the typical achievable tolerances for various production material removal processes
Tolerances

Figure 1: Typical tolerances for material removal processes

(Φυλλάδιο εργαστηριακών ασκήσεων Μηχανουργείου, Μουρτζής Δ. κ.α., Πανεπιστήμιο Πατρών)
Tolerances

Figure 2: Range of Dimensional Tolerances in Machining as a Function of Workpiece Size

I. Tolerancing for Interchangeability
Tolerancing / Interchangeability

- Tolerancing is dimensioning for interchangeability
  An interchangeable part is simply a mass produced part
  (a replacement part)

- Tolerancing allows a **range specification of accuracy** for every
  feature of a product, so the parts will fit together and function properly
  when assembled

- How is a feature on an interchangeable part dimensioned?
  - The feature **is not dimensioned using a single value**, but a
    **range** of values

\[ \begin{align*}
1.00 & \rightarrow 1.005 \\
.994 & \quad .994
\end{align*} \]
Tolerancing / Interchangeability

- **Tolerance** is the total amount a specific dimension is permitted to vary.

- Usage of **generous tolerances** when possible is preferred because increased precision makes parts more **expensive to manufacture**.

- A tolerance that specifies a large or small variation can be chosen.
Understanding Tolerance

Size limits: 1.005 - 0.994 = 0.011
Tolerancing / Interchangeability

- Why do we want a part’s size to be controlled by two limits?

- It is necessary because it is **impossible to manufacture parts** without some variation

- The stated limits are a form of quality control
Choosing the correct tolerance for a particular application depends on:

- the design intent (end use) of the part
- cost
- how it is manufactured
- experience
II. Tolerance Types
Tolerance Types

The tolerancing methods presented are:

1. Limit dimensions
2. Plus or minus tolerances
3. Page or block tolerances
1. Limit Dimensions

- **Limits** are the **maximum and minimum size** that a part can obtain and still pass inspection.

  For example, the diameter of a shaft might be specified as follows.

\[
\phi^{1.001}_{0.999} \quad \text{or} \quad \phi^{1.001 - 0.999}
\]
1. Limit Dimension Order

- **External dimensions:**
  - The larger dimension is first or on top and the smaller dimension is last or on the bottom

- **Internal dimensions:**
  - The smaller dimension is first and the larger dimension is last
2. Plus or Minus Tolerances

- **Plus or minus tolerances** give a basic size and the variation that can occur around that basic size.

![Diagram showing plus or minus tolerances](image)

(a) Unilateral tolerance  
(b) Bilateral tolerance

Figure 3: Example of plus or minus tolerances
3. Page or Block Tolerances

- A **page tolerance** is actually a general note that applies to all dimensions not covered by some other tolerancing type.

```
UNLESS OTHERWISE SPECIFIED ALL:
 .XX = ± .010 inch
 .XXX = ± .005 inch
 .XXXX = ± .002 inch
```
III. General Definitions
General Definitions

- What are the limits, tolerance and allowance for the following shaft/hole system? Are they the same or different?
General Definitions

- **Limits**
  Are the maximum and minimum diameters

- **Tolerance**
  Is the difference between two limits

- **Allowance (Minimum Clearance)**
  Is the difference between the largest shaft diameter and the smallest hole diameter

(K. Plantenberg, 2006,)
Example 1

- What are the limits of the shaft and the hole?

  - **Shaft:** \( D_{\text{shaft}} - d_{\text{shaft}} \)

  - **Hole:** \( d_{\text{hole}} - D_{\text{hole}} \)
What is the **tolerance** for the shaft and the hole?

- **Shaft:** \( D_{\text{shaft}} - d_{\text{shaft}} = .. \)
- **Hole:** \( D_{\text{hole}} - d_{\text{hole}} = .. \)
Example 3

- What is the **minimum clearance** (allowance)?

\[ d_{\text{hole}} - D_{\text{shaft}} = \]
Example 4

- What is the **maximum clearance**?

\[ D_{\text{hole}} - d_{\text{shaft}} = \]
IV. Tolerancing Standards
Tolerancing Standards

- **Standards** are needed to:

  ✓ Make it possible to manufacture parts at **different times** and in **different places** that still **assemble properly**

  ✓ Establish **dimensional limits for parts** that are to be **interchangeable**
The two most common standards agencies are:

- American National Standards Institute (ANSI) / (ASME)
- International Standards Organization (ISO)
V. Inch Tolerances
Inch Tolerances Definitions

- **Limits**
The limits are the **maximum and minimum size** that the part is allowed to be.

- **Basic Size**
The **basic size** is the size from which the **limits are calculated**.
  - It is common for both the hole and the shaft and is usually the closest fraction.

- **Tolerance**
The **tolerance** is the total amount a **specific dimension is permitted to vary**.

(K. Plantenberg, 2006)
Inch Tolerances Definitions

- **Maximum Material Condition (MMC):**

  The MMC is the size of the part when it consists of the most material.

- **Least Material Condition (LMC):**

  The LMC is the size of the part when it consists of the least material.
Inch Tolerances Definitions

- **Maximum Clearance:**
The maximum amount of space that can exist between the hole and the shaft

\[
\text{Max. Clearance} = \text{LMC}_{\text{hole}} - \text{LMC}_{\text{shaft}}
\]

- **Minimum Clearance (Allowance):**
The minimum amount of space that can exist between the hole and the shaft

\[
\text{Min. Clearance} = \text{MMC}_{\text{hole}} - \text{MMC}_{\text{shaft}}
\]
Types of Fits

- **Clearance Fit**: There is always a space, $\text{Min. Clearance} > 0$

- **Interference Fit**: There is never a space, $\text{Max. Clearance} \leq 0$

- **Transition Fit**: Depending on the sizes of the shaft and hole, there could be a space or no space, $\text{Max. Clearance} > 0$
  $\text{Min. Clearance} < 0$

- **Line Fit**: There is a space or a contact (hole diameter = shaft diameter),
  $\text{Max. Clearance} > 0$
  $\text{Min. Clearance} = 0$

(K. Plantenberg, 2006, )
## Types of Fits

- From everyday life, list some examples of clearance and interference fits

<table>
<thead>
<tr>
<th>Fit</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clearance</td>
<td>Lock and Key</td>
</tr>
<tr>
<td></td>
<td>Door and Door frame</td>
</tr>
<tr>
<td></td>
<td>Coin and Coin slot</td>
</tr>
<tr>
<td>Interference</td>
<td>Pin in a bicycle chain</td>
</tr>
<tr>
<td></td>
<td>Hinge pin</td>
</tr>
<tr>
<td></td>
<td>Wooden peg and hammer toy</td>
</tr>
</tbody>
</table>
Example 5

- Determine the basic size and type of fit given the limits for the shaft and hole

<table>
<thead>
<tr>
<th>Shaft Limits</th>
<th>Hole Limits</th>
<th>Type of fit</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.500 – 1.498</td>
<td>1.503 – 1.505</td>
<td>1.5 Clearance</td>
</tr>
<tr>
<td>.755 - .751</td>
<td>.747 - .750</td>
<td>.75 Interference</td>
</tr>
<tr>
<td>.378 - .373</td>
<td>.371 - .375</td>
<td>.375 Transition</td>
</tr>
<tr>
<td>.250 - .247</td>
<td>.250 - .255</td>
<td>.25 Line</td>
</tr>
</tbody>
</table>
ANSI Standard Limits and Fits

The following fit types and classes are in accordance with the ANSI B4.1-1967(R1994) standard:

- **RC: Running or Sliding Clearance fit**
  - Intended to provide running performance with suitable lubrication.
  - RC9 (loosest) – RC1 (tightest)

- **FN: Force Fits**
  - Force fits provide a constant bore pressure throughout the range of sizes.
  - FN1 – FN5 (tightest)
ANSI Standard Limits and Fits

- **Locational fits (LC, LT, LN)**

  - **Locational fits** are intended to determine only the location of the mating parts

  - **LC** = Locational clearance fits
  - **LT** = Locational transition fits
  - **LN** = Locational interference fits
Example 6

- Given a basic size of .50 inches and a fit of RC8, calculate the limits for both the hole and the shaft

➢ Use the **ANSI limits and fit tables** given in the following Appendix
**Appendix**

<table>
<thead>
<tr>
<th>Nominal Size Range Inches</th>
<th>Class RC5</th>
<th>Class RC6</th>
<th>Class RC7</th>
<th>Class RC8</th>
<th>Class RC9</th>
</tr>
</thead>
<tbody>
<tr>
<td>Over</td>
<td>Hole</td>
<td>Hole</td>
<td>Hole</td>
<td>Hole</td>
<td>Hole</td>
</tr>
<tr>
<td>To</td>
<td>Shaft</td>
<td>Shaft</td>
<td>Shaft</td>
<td>Shaft</td>
<td>Shaft</td>
</tr>
<tr>
<td>0</td>
<td>0.6</td>
<td>1.0</td>
<td>1.0</td>
<td>1.6</td>
<td>2.5</td>
</tr>
<tr>
<td>-0.12</td>
<td>-0.6</td>
<td>-0.6</td>
<td>-1.0</td>
<td>-2.5</td>
<td>-4.0</td>
</tr>
<tr>
<td>0.71</td>
<td>+1.2</td>
<td>+2.0</td>
<td>+2.0</td>
<td>+3.5</td>
<td>+5.0</td>
</tr>
<tr>
<td>-1.19</td>
<td>-1.6</td>
<td>-1.6</td>
<td>-2.5</td>
<td>-4.5</td>
<td>-7.0</td>
</tr>
<tr>
<td>1.19</td>
<td>+1.6</td>
<td>+2.5</td>
<td>+2.5</td>
<td>+4.0</td>
<td>+6.0</td>
</tr>
<tr>
<td>-1.97</td>
<td>-2.0</td>
<td>-2.0</td>
<td>-3.0</td>
<td>-5.0</td>
<td>-8.0</td>
</tr>
<tr>
<td>1.97</td>
<td>+1.8</td>
<td>+3.0</td>
<td>+3.0</td>
<td>+4.5</td>
<td>+7.0</td>
</tr>
<tr>
<td>-3.15</td>
<td>-2.5</td>
<td>-2.5</td>
<td>-4.0</td>
<td>-9.0</td>
<td>-13.5</td>
</tr>
<tr>
<td>3.15</td>
<td>+2.2</td>
<td>+3.5</td>
<td>+3.5</td>
<td>+5.0</td>
<td>+9.0</td>
</tr>
<tr>
<td>-4.73</td>
<td>-3.0</td>
<td>-3.0</td>
<td>-5.0</td>
<td>-10.0</td>
<td>-15.0</td>
</tr>
<tr>
<td>4.73</td>
<td>+2.5</td>
<td>+4.0</td>
<td>+4.0</td>
<td>+6.0</td>
<td>+10.0</td>
</tr>
<tr>
<td>-7.09</td>
<td>-3.5</td>
<td>-3.5</td>
<td>-8.0</td>
<td>-12.0</td>
<td>-18.0</td>
</tr>
<tr>
<td>7.09</td>
<td>+2.8</td>
<td>+4.5</td>
<td>+4.5</td>
<td>+7.0</td>
<td>+12.0</td>
</tr>
<tr>
<td>-9.85</td>
<td>-4.0</td>
<td>-4.0</td>
<td>-7.0</td>
<td>-15.0</td>
<td>-22.0</td>
</tr>
<tr>
<td>9.85</td>
<td>+3.0</td>
<td>+5.0</td>
<td>+5.0</td>
<td>+8.0</td>
<td>+12.0</td>
</tr>
<tr>
<td>-12.41</td>
<td>-5.0</td>
<td>-5.0</td>
<td>-8.0</td>
<td>-17.0</td>
<td>-26.0</td>
</tr>
<tr>
<td>12.41</td>
<td>+3.5</td>
<td>+6.0</td>
<td>+6.0</td>
<td>+9.0</td>
<td>+14.0</td>
</tr>
<tr>
<td>-15.75</td>
<td>-6.0</td>
<td>-6.0</td>
<td>-10.0</td>
<td>-20.0</td>
<td>-31.0</td>
</tr>
<tr>
<td>15.75</td>
<td>+4.0</td>
<td>+6.0</td>
<td>+6.0</td>
<td>+10.0</td>
<td>+16.0</td>
</tr>
<tr>
<td>-19.69</td>
<td>-8.0</td>
<td>-8.0</td>
<td>-12.0</td>
<td>-16.0</td>
<td>-25.0</td>
</tr>
</tbody>
</table>

Basic size = .5
Fit = RC8
Example 7

- Given a basic size of .50 inches and a fit of RC8, calculate the limits for both the hole and the shaft.

  - Standard Limits Hole = +2.8  0
  - Standard Limits Shaft = -3.5  -5.1

- These are the values that we add/subtract from the basic size to obtain the limits.
Example 8

- Given a basic size of .50 inches and a fit of RC8, calculate the limits for both the hole and the shaft.

- Hole Limits = \(0.50 - 0 = 0.5000\)
  \[0.50 + 0.0028 = 0.5028\]

- Shaft Limits = \(0.50 - 0.0035 = 0.4965\)
  \[0.50 - 0.0051 = 0.4949\]
Example 9

- Consider the **Milling Jack assembly** shown

  - Notice that there are **many parts that fit into or around other parts**

  - Each of these parts is **toleranced** to ensure **proper fit** and **function**

(K. Plantenberg, 2006, )
Example 9

- The V-Anvil fits into the Sliding Screw with a RC4 fit

- The basic size is .375 (3/8). Determine the limits for both parts
Example 9

3/16 x 3/32 KEYWAY
USE #606 WOODRUFF CUTTER

Ø .375 RC4 - 2 DEEP
5/8 - 18 UNF - 2A

.3750 - .3759

.3745 - .3739

.625 RC5
Example 10

- The Sliding Screw fits into the Base with a RC5 fit

- The basic size is .625 (5/8). Determine the limits for both parts
Example 10

Keyway Cutter

Dimensions:
- Diameter: 0.625 RC5
- Length: 3.56
- Depth: 1.00
- Diameter: 0.375 RC4

Drill:
- Diameter: 0.625 RC5
- Depth: 1.56

Example:
- Diameter Range: 0.6238 - 0.6231
VI. Metric Tolerances
Metric Tolerances Definitions

- Limits, Basic Size, Tolerance, MMC and LMC have the same definition as in the inch tolerance section

- **Upper deviation**
The upper deviation is the difference between the basic size and the permitted maximum size of the part

\[
UD = |\text{basic size} - D_{\text{max}}|
\]

- **Lower deviation**
The lower deviation is the difference between the basic size and the minimum permitted size of the part

\[
LD = |\text{basic size} - D_{\text{min}}|
\]
Metric Tolerances Definitions

● **Fundamental deviation**

The fundamental deviation is the closest deviation to the basic size

➢ The fundamental deviation is the smaller of the UD and the LD

➢ A letter in the fit specification represents the fundamental deviation

Example: Metric Fit = H11/c11
Exercise 1

- Fill in the following table

<table>
<thead>
<tr>
<th></th>
<th>Shaft</th>
<th>Hole</th>
</tr>
</thead>
<tbody>
<tr>
<td>UD</td>
<td>0.2</td>
<td>0</td>
</tr>
<tr>
<td>LD</td>
<td>0.1</td>
<td>0.2</td>
</tr>
<tr>
<td>FD</td>
<td>0.1</td>
<td>0</td>
</tr>
</tbody>
</table>
Exercise 2

- Fill in the following table

<table>
<thead>
<tr>
<th>Type of fit</th>
<th>Interference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\phi \frac{2.2}{2.1}$</td>
<td>$\phi \frac{1.8}{2.0}$</td>
</tr>
</tbody>
</table>
Metric Tolerances Definitions

- **International tolerance grade number (IT#)**

  The IT#’s are a set of tolerances that vary according to the basic size and provide the same relative level of accuracy within a given grade.

  - The number in the fit specification represents the IT#
  - A smaller number provides a smaller tolerance
Metric Tolerances Definitions

- **Tolerance zone**

The fundamental deviation in combination with the IT# defines the tolerance zone

- The IT# establishes the magnitude of the tolerance zone or the amount that the dimension can vary

- The fundamental deviation establishes the position of the tolerance zone with respect to the basic size

(K. Plantenberg, 2006, )
## Available Metric Fits

<table>
<thead>
<tr>
<th>Hole Basis</th>
<th>Shaft Basis</th>
<th>Fit</th>
</tr>
</thead>
<tbody>
<tr>
<td>H11/c11</td>
<td>C11/h11</td>
<td>Loose running</td>
</tr>
<tr>
<td>H9/d9</td>
<td>D9/h9</td>
<td>Free running</td>
</tr>
<tr>
<td>H8/f7</td>
<td>F8/h7</td>
<td>Close running</td>
</tr>
<tr>
<td>H7/g6</td>
<td>G7/h6</td>
<td>Sliding</td>
</tr>
<tr>
<td>H7/h6</td>
<td>H7/h6</td>
<td>Locational clearance</td>
</tr>
<tr>
<td>H7/k6 or H7/n6</td>
<td>K7/h6 or N7/h6</td>
<td>Locational transition</td>
</tr>
<tr>
<td>H7/p6</td>
<td>P7/h6</td>
<td>Locational interference</td>
</tr>
<tr>
<td>H7/s6</td>
<td>S7/h6</td>
<td>Medium drive</td>
</tr>
<tr>
<td>H7/u6</td>
<td>U7/h6</td>
<td>Force</td>
</tr>
</tbody>
</table>
Tolerance Designation

- A Metric fit is specified by stating the fundamental deviation and the IT#
  
- IT# = the amount that the dimension can vary (tolerance zone size)
  
- Fundamental deviation (letter) = establishes the position of the tolerance zone with respect to the basic size
  
- Hole = upper case
  
- Shaft = lower case
Tolerance Designation

- Fits are specified by using the:
  - fundamental deviation (letter)
  - IT# (International Tolerance Grade #)

- When specifying the fit:
  - The hole = upper case letter
  - The shaft = lower case letter
Example 11

- Fill in the appropriate name for the fit component.

Basic size

Hole Tolerance Zone
Shaft Tolerance Zone

Fundamental Deviation

IT #
Basic Hole / Basic Shaft Systems

- Metric limits and fits are divided into two different systems; the basic hole system and the basic shaft system

- **Basic hole system**: The basic hole system is used when you want the basic size to be attached to the hole dimension.
  
  ➢ For example, if you want to tolerance a shaft based on a hole produced by a standard drill, reamer, broach, or another standard tool

- **Basic shaft system**: The basic shaft system is used when you want the basic size to be attached to the shaft dimension
  
  ➢ For example, if you want to tolerance a hole based on the size of a purchased a standard drill rod
Example 12

- Identify the type of fit and the system used to determine the limits of the following shaft and hole pairs

<table>
<thead>
<tr>
<th>Shaft</th>
<th>Hole</th>
<th>Type of Fit</th>
<th>System</th>
</tr>
</thead>
<tbody>
<tr>
<td>9.987 – 9.972</td>
<td>10.000 – 10.022</td>
<td>Clearance</td>
<td>Hole</td>
</tr>
<tr>
<td>60.021 – 60.002</td>
<td>60.000 – 60.030</td>
<td>Transition</td>
<td>Hole</td>
</tr>
<tr>
<td>40.000 – 39.984</td>
<td>39.924 – 39.949</td>
<td>Interference</td>
<td>Shaft</td>
</tr>
</tbody>
</table>

(K. Plantenberg, 2006, )
Example 13

- Find the limits, tolerance, type of fit, and type of system for a n30 H11/c11 fit

➢ Use the tolerance tables given below

<table>
<thead>
<tr>
<th>Basic Size</th>
<th>Loose Running</th>
<th>Free Running</th>
<th>Close Running</th>
<th>Sliding</th>
<th>Locational Clearance</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Hole H11</td>
<td>Shaft c11</td>
<td>Hole H9</td>
<td>Shaft d9</td>
<td>Hole H8</td>
</tr>
<tr>
<td>1 max min</td>
<td>1.060</td>
<td>0.940</td>
<td>1.025</td>
<td>0.980</td>
<td>1.014</td>
</tr>
<tr>
<td></td>
<td>1.000</td>
<td>0.880</td>
<td>1.000</td>
<td>0.955</td>
<td>1.000</td>
</tr>
<tr>
<td>1.2 max min</td>
<td>1.260</td>
<td>1.140</td>
<td>1.225</td>
<td>1.180</td>
<td>1.214</td>
</tr>
<tr>
<td></td>
<td>1.200</td>
<td>1.080</td>
<td>1.200</td>
<td>1.155</td>
<td>1.200</td>
</tr>
<tr>
<td>1.6 max min</td>
<td>1.660</td>
<td>1.540</td>
<td>1.625</td>
<td>1.580</td>
<td>1.614</td>
</tr>
<tr>
<td></td>
<td>1.600</td>
<td>1.480</td>
<td>1.600</td>
<td>1.555</td>
<td>1.600</td>
</tr>
<tr>
<td>2 max min</td>
<td>2.060</td>
<td>1.940</td>
<td>2.025</td>
<td>1.980</td>
<td>2.014</td>
</tr>
<tr>
<td></td>
<td>2.000</td>
<td>1.880</td>
<td>2.000</td>
<td>1.955</td>
<td>2.000</td>
</tr>
<tr>
<td></td>
<td>16.000</td>
<td>15.795</td>
<td>16.000</td>
<td>15.907</td>
<td>16.000</td>
</tr>
<tr>
<td>30 max min</td>
<td>30.130</td>
<td>29.890</td>
<td>30.052</td>
<td>29.935</td>
<td>30.033</td>
</tr>
<tr>
<td></td>
<td>30.000</td>
<td>29.760</td>
<td>30.000</td>
<td>29.883</td>
<td>30.000</td>
</tr>
</tbody>
</table>
Example 14

- Find the limits, tolerance, type of fit, and type of system for an n30 H11/c11 fit.

<table>
<thead>
<tr>
<th>Limits</th>
<th>Shaft Limits</th>
<th>Hole Limits</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>29.890 – 29.760</td>
<td>30.000 – 30.130</td>
</tr>
<tr>
<td>Tolerance</td>
<td>0.13</td>
<td>0.13</td>
</tr>
<tr>
<td>System</td>
<td>Hole</td>
<td></td>
</tr>
<tr>
<td>Fit</td>
<td>Clearance – Loose Running</td>
<td></td>
</tr>
</tbody>
</table>
Example 15

- Find the limits, tolerance, type of fit, and type of system for a n30 P7/h6 fit

➢ Use the tolerance tables given below

<table>
<thead>
<tr>
<th>Basic Size</th>
<th>Locational Transition</th>
<th>Locational Transition</th>
<th>Locational Interference</th>
<th>Medium Drive</th>
<th>Force</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Hole K7</td>
<td>Shaft h6</td>
<td>Hole N7</td>
<td>Shaft h6</td>
<td>Hole P7</td>
</tr>
<tr>
<td>1 max</td>
<td>1.000</td>
<td>1.000</td>
<td>0.996</td>
<td>1.000</td>
<td>0.994</td>
</tr>
<tr>
<td>min</td>
<td>0.990</td>
<td>0.994</td>
<td>0.986</td>
<td>0.994</td>
<td>0.984</td>
</tr>
<tr>
<td>1.2 max</td>
<td>1.200</td>
<td>1.200</td>
<td>1.196</td>
<td>1.200</td>
<td>1.194</td>
</tr>
<tr>
<td>min</td>
<td>1.190</td>
<td>1.194</td>
<td>1.186</td>
<td>1.194</td>
<td>1.184</td>
</tr>
<tr>
<td>1.6 max</td>
<td>1.600</td>
<td>1.600</td>
<td>1.596</td>
<td>1.600</td>
<td>1.594</td>
</tr>
<tr>
<td>min</td>
<td>1.590</td>
<td>1.594</td>
<td>1.586</td>
<td>1.594</td>
<td>1.584</td>
</tr>
<tr>
<td>2 max</td>
<td>2.000</td>
<td>2.000</td>
<td>1.996</td>
<td>2.000</td>
<td>1.994</td>
</tr>
<tr>
<td>min</td>
<td>1.990</td>
<td>1.994</td>
<td>1.986</td>
<td>1.994</td>
<td>1.984</td>
</tr>
<tr>
<td>16 max</td>
<td>16.006</td>
<td>16.000</td>
<td>15.995</td>
<td>16.000</td>
<td>15.989</td>
</tr>
<tr>
<td>min</td>
<td>15.988</td>
<td>15.989</td>
<td>15.977</td>
<td>15.989</td>
<td>15.971</td>
</tr>
<tr>
<td>25 max</td>
<td>25.006</td>
<td>25.000</td>
<td>24.993</td>
<td>25.000</td>
<td>24.986</td>
</tr>
<tr>
<td>30 max</td>
<td>30.006</td>
<td>30.000</td>
<td>29.993</td>
<td>30.000</td>
<td>29.986</td>
</tr>
</tbody>
</table>
Example 16

- Find the limits, tolerance, type of fit, and type of system for a n30 P7/h6 fit

<table>
<thead>
<tr>
<th></th>
<th>Shaft</th>
<th>Hole</th>
</tr>
</thead>
<tbody>
<tr>
<td>Limits</td>
<td>30.000 – 29.987</td>
<td>29.965 – 29.986</td>
</tr>
<tr>
<td>Tolerance</td>
<td>0.013</td>
<td>0.021</td>
</tr>
<tr>
<td>System</td>
<td>Shaft</td>
<td></td>
</tr>
<tr>
<td>Fit</td>
<td>Locational Interference</td>
<td></td>
</tr>
</tbody>
</table>
VII. Selecting Tolerances
Selecting Tolerances

- **Tolerances** will govern the method of manufacturing

- When the tolerances are reduced, the cost of manufacturing rises very rapidly

- Specify as generous a tolerance as possible without interfering with the function of the part
Selecting Tolerances

Figure 4: Dependence of manufacturing cost on dimensional tolerances
Selecting Tolerances

● Choosing the most appropriate tolerance depends on many factors such as:
  ➢ Length of engagement
  ➢ Bearing load
  ➢ Speed
  ➢ Lubrication
  ➢ Temperature
  ➢ Humidity
  ➢ Material

● Experience also plays a significant role
# Machining and IT Grades

<table>
<thead>
<tr>
<th>Machining Operation</th>
<th>IT Grades</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>4</td>
</tr>
<tr>
<td>Lapping &amp; Honing</td>
<td></td>
</tr>
<tr>
<td>Cylindrical Grinding</td>
<td></td>
</tr>
<tr>
<td>Surface Grinding</td>
<td></td>
</tr>
<tr>
<td>Diamond Turning</td>
<td></td>
</tr>
<tr>
<td>Diamond Boring</td>
<td></td>
</tr>
<tr>
<td>Broaching</td>
<td></td>
</tr>
<tr>
<td>Reaming</td>
<td></td>
</tr>
<tr>
<td>Turning</td>
<td></td>
</tr>
<tr>
<td>Boring</td>
<td></td>
</tr>
<tr>
<td>Milling</td>
<td></td>
</tr>
<tr>
<td>Planing &amp; Shaping</td>
<td></td>
</tr>
<tr>
<td>Drilling</td>
<td></td>
</tr>
<tr>
<td>Punching</td>
<td></td>
</tr>
<tr>
<td>Die Casting</td>
<td></td>
</tr>
</tbody>
</table>
VIII. Tolerance Accumulation
Tolerance Accumulation

- Figures 5-7 compare the tolerance values resulting from the following **three methods of dimensioning**

![Diagram of Chain Dimensioning]

**Figure 5: Chain Dimensioning-Greatest tolerance accumulation between X and Y**

**(a) Chain Dimensioning:** The maximum variation between two features is equal to the sum of the tolerances on the intermediate distances; this results in the greatest tolerance accumulation. In this figure the tolerance accumulation between surfaces X and Y is ± 0.15

(K. Plantenberg, 2006, )
(b) Base Line Dimensioning: The maximum variation between two features is equal to the sum of the tolerances on the two dimensions from their origin to the features; this results in a reduction of the tolerance accumulation. In this figure, the tolerance accumulation between surfaces X and Y is ±0.1
(c) **Direct Dimensioning**: The maximum variation between two features is controlled by the tolerance on the dimension between the features; this results in the least tolerance. In this figure, the tolerance between surfaces X and Y is ±0.05.

Figure 7: Direct Dimensioning-Least tolerance between X and Y

(K. Plantenberg, 2006, )
The tolerance between two features of a part depends on the number of controlling dimensions.
Tolerance Accumulation

- The **distance** could be controlled by a **single dimension** or **multiple dimensions**
Tolerance Accumulation

- The **maximum variation** between two features is equal to the **sum** of the tolerances placed on the controlling dimensions.

![Diagram showing tolerance accumulation](image-url)
Tolerance Accumulation

- As the number of controlling dimensions increases, the tolerance accumulation increases
Tolerance Accumulation

- **Remember**: even if the dimension does not have a stated tolerance, it has an implied tolerance

- **Example**:

  What is the tolerance accumulation for the distance between surface A and B for the following three dimensioning methods?
Tolerance Accumulation

Tolerance accumulation between surface A and B = 0.3

(K. Plantenberg, 2006)
Tolerance Accumulation

Perfect
Tolerance Accumulation

Worst Case
Tolerance Accumulation

Tolerance accumulation between surface A and B = 0.2
Tolerance Accumulation

Perfect
Tolerance Accumulation

Worst Case

Dimensions:
- 110.1
- 90.1
- 60.1
- 39.9
- 70.2
Tolerance Accumulation

Tolerance accumulation between surface A and B = 0.1
Example 17

- Assuming that the diameter dimensions are correct, explain why this object is dimensioned incorrectly.
Example 17

1. The decimal places don’t match

2. The dimensions are inconsistent

$1.98 + .99 = 2.97$
$2.01 + 1.00 = 3.01$

This part is over dimensioned
IX. Formatting Tolerances
● Tolerances from standardized fit tables are listed on drawings as:

\[
\begin{align*}
\phi_{20.240} &\quad (\phi_{20 \text{ C11}}) \\
\phi_{20 \text{ C11}} &\quad (\phi_{20.240} \\
\phi_{20 \text{ C11}} &\quad \text{The person reading the print must have access to the standard fit tables}
\end{align*}
\]

(K. Plantenberg, 2006)
Formatting Metric Tolerances

- **Unilateral tolerances**
  
  \[
  40.0 \pm 0.02 \quad \text{or} \quad 40.00 \pm 0.02
  \]

  ➢ A single zero without a plus or minus sign

- **Bilateral tolerances**
  
  \[
  10 +0.25 \pm -0.10 \quad \text{not} \quad 10 +0.25 \pm -0.10
  \]

  ➢ Both the plus and minus values have the same number of decimal places
Formatting Metric Tolerances

- **Limit dimensions**

  15.45  
  15.00  not  15.45  
  15     not  15

  ➢ Both values should have the same number of decimal places

- **Using Basic dimensions with the tolerance**

  45±0.15  not  45.00±0.15

  ➢ The number of decimal places in the basic dimension does not have to match the number of decimal places in the tolerance
Formatting Inch Tolerances

- Unilateral and Bilateral tolerances

- The basic dimension and the plus and minus values should have the same number of decimal places
Formatting Inch Tolerances

- Limit dimensions

- Both values should have the same number of decimal places

- Using Basic dimensions with the tolerance

- The number of decimal places in the basic dimension should match the number of decimal places in the tolerance
Formatting Angular Tolerances

- Angular tolerances

- Both the angle and the plus and minus values have the same number of decimal places

(K. Plantenberg, 2006, )
Summary 1/2

- **Tolerance** is the difference between two limits

- If a feature’s size is tolerated, it is allowed to vary within a range of values or limits

- **Tolerancing** enables an engineer to design interchangeable or replacement parts

- **Allowance** is the difference between the largest shaft diameter and the smallest hole diameter

- The two most common standards agencies are; **American National Standards Institute (ANSI) / (ASME)** and **International Standards Organization (ISO)**
The International Tolerance Grade number (IT#) is set of tolerances that vary according to the basic size and provide the same relative level of accuracy within a given grade 00.
References


4. Tolerancing Topics - Exercises presentation ,2006, K.Plantenberg

5. Φυλλάδιο εργαστηριακών ασκήσεων Μηχανουργείου,2012.Δημητρακόπουλος Γ., Μούρτζης Δ., Πανδρεμένος Ι., Παπακώστας Ν., Σταυρόπουλος Π., Φυσικόπουλος Α.