IPE 381
Chapter:10
Gauges and Gauge Design

Abdullah-Al-Mamun
Lecturer, Dept. of IPE

Outline

• Basics of Gauges
• Classification of Gauges
• Gauge Design
• Screw Gauges
• Adv & Disadv of Limit Gauges
• Gauge Design problem
Gauges

- Gauges are inspection tool of rigid design, without a scale, which serves to check the dimensions of manufacturing parts.
- Gauge do not indicate the actual value of the inspected part of the component.
- They are used to determine whether the part is made within the specified limit.

Classification of Plain Gauge

- Plain gauges are used for checking plain (unthreaded) holes and shafts. Gauges are classified:
  - According to their type:
    (a) Standard gauges - made as an exact copy of opposed (mating) part
    (b) Limit gauges - made to the limits of the dimensions
  - According to their purposes:
    (a) Workshop Gauge: To Check Dimension after manufacture
    (b) Inspection Gauge: To check part before final acceptance
    (d) Purchase Inspection Gauge: To check part of other factory
    (c) Reference or master gauges: To check the dimension of Gauge
Classification of Plain Gauge

- According to the form of the **tested surface**:
  (a) Plug gauges for checking holes.
  (b) Snap and ring gauges for checking shafts.

Classification of Plain Gauge

- According to their **design**:
  (a) Single limit and double limit gauges
  (b) Single ended and double ended gauges,
  (c) Fixed and adjustable gauges.
Limit Gauge

- Limit gauges are made to the limits of the dimensions of the part to be tested. There are two limit of dimensions, so we need two limit gauge.
- ‘Go gauge’ should pass through or over a part while ‘Not Go gauge’ should not pass through or over the part.

![Image of Limit Gauge](image)

Design of Limit Gauge

- Allocation of Tolerance
  - Manufacturing Tolerance
  - Wear Allowance
- Taylor’s Principle of gauge Design
- Fixing of Gauge elements with handles
  - Taper lock design
  - Trilock Design
- Provision of Guards
- Provision of PilotCorrect Centering
- Materials
- Hardness and Surface finish
- Rigidity
- Alignment of Gauge faces
Allocation of Tolerance

Manufacturing Tolerance
- It is economically impractical to attempt to make “Go” and “Not Go” gauges exactly to the two limits of work tolerance.
- Limit gauges are made 10 times more accurate than the tolerances they are going to control.

Allocation of Mfg Tolerance

(a) Hole (Plug gauge)

(a) Shaft (Snap gauge)

(a) Single Limit

(b) Double Limit

Fig. 10.5. Single Limit and Double Limit Gauge.
Allocation of Mfg Tolerance

Example: let the size of the hole to be tested be 25 ± 0.02 mm. Therefore,

<table>
<thead>
<tr>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>High limit of hole</td>
<td>25.02 mm</td>
</tr>
<tr>
<td>Low limit of hole</td>
<td>24.98 mm</td>
</tr>
<tr>
<td>Work tolerance</td>
<td>0.04</td>
</tr>
<tr>
<td>Gauge tolerance = 10% of tolerance</td>
<td>0.004 mm</td>
</tr>
</tbody>
</table>

- Dimension of ‘Go’ Plug gauge
  - + 0.004
  - 24.98 mm
  - - 0.000

- Dimension of ‘Not Go’ Plug gauge
  - + 0.000
  - 25.02 mm
  - - 0.004

Allocation of Tolerance

Wear Allowance

- The surface of “Go” gauge is constantly rub against the surface of the part in inspection and loose their initial size
- The size of plug gauge is reduced but size of snap gauge is increased.
- 5% wear allowance is provided if working tolerance is greater than 0.09 mm.
Taylor’s Principle

- **Go Gauge** should always be so designed that it will cover the maximum Metal Condition (MMC) of as many dimensions as possible in the same limit gauge, whereas **Not Go gauge** to cover the minimum metal condition of one dimension only.
To Check Circular Holes

Go Plug Gauge
Should Have a Full
Circular Section.

Length should be
equal to the length of
the hole.

Not Go Plug Gauge
should be
In the form of pin or
bar.

Should check the
upper limit of the
hole.
To Check Circular Shaft

• Ring Gauge Should be used as Go Gauge
• Length should be equal to the length of the shaft

To Check Circular Shaft

• Snap Gauge Should be used as Not Go Gauge
• Should check the low limit of shaft
• For non circular shaft, Snap gauge will not pass through
Fixing with Handles

- **Taper Lock Design**
  - Used for diameter upto 63.5 mm.
  - Used for smaller size plain and screw gauges.

- **Trilock Design**
  - Used for diameter from 63.5 mm to 203mm.
  - Gauge is attached with handle with three lock nuts.

Provision of Pilot

- For closely tolerated parts Plug gauge doesn’t easily enter the hole so piloting is used.
Thread or Screw Gauge

- Nuts and internal threads are checked with Plug thread gauge and screws with Ring thread gauge.
- Three types of Fit
  - Close Fit
  - Medium Fit
  - Free Fit

Designing a Gauge

A 25mm H8-f7 fit is to be checked.

The limit of size for H8 Hole are:
  - High Limit - 25.033mm.
  - Low Limit - 25.000mm.

The limits for the f7 shaft are:
  - High limit - 24.980mm.
  - Low limit - 24.959mm.

Taking the gauge makers tolerance to be 10% of the work tolerance, design plug gauge and gap gauge to check the fit.
Designing a Gauge

• Find tolerance for hole an shaft
• Find gauge tolerance
• Basic Size for “Go” Plug Gauge is LL of Hole
• Basic Size for “Go” Plug Gauge is HL of Hole
• For plug gauge:
  – High limit of “Go” Plug gauge
  – Low limit of “Go” Plug gauge
  – High limit of “Not Go” Plug gauge
  – Low limit of “Not Go” Plug gauge
• Repeat for Snap gauge
• Draw the Diagram of “Go”, “Not Go” Plug gauge and “Go”, “Not Go” Snap gauge

Self Study

• 10.7 Advantage of Limit Gauge
• 10.8 Disadvantage of Limit Gauge
• 10.9 Care of Gauges
• 10.10 Other types of Gauge

Thank You