Dimensioning and Tolerancing
Dimensioning

- Before an object can be built, complete information about both the size and shape of the object must be available. The exact shape of an object is communicated through orthographic drawings, which are developed following standard drawing practices. The process of adding size information to a drawing is known as dimensioning the drawing.
Dimensioning

- **Geometrics** is the science of specifying and tolerancing the shapes and locations of features on objects. Once the shape of a part is defined with an orthographic drawings, the size information is added also in the form of **dimensions**.
- Dimensioning a drawing also identifies the tolerance (or accuracy) required for each dimension.
If a part is dimensioned properly, then the intent of the designer is clear to both the person making the part and the inspector checking the part.

A fully defined part has three elements: graphics, dimensions, and words (notes).
A well dimensioned part will communicate the size and location requirements for each feature. Communications is the fundamental purpose of dimensions.

Parts are dimensioned based on two criteria:
- Basic size and locations of the features.
- Details of a part's construction and for manufacturing.
Unit of measure

- On a drawing used in American industry, all dimensions are in inches, unless otherwise stated.
- Most countries outside of the United States use the metric system of measure, or the international system of units (SI), which is based on the meter.
- The SI system is being used more in the United States because of global trade and multinational company affiliations.
Unit of measure

- Occasionally, a company will use dual dimensioning, that is, both metric and English measurements on a drawing.
- Angular dimensions are shown either in decimal degrees or in degrees, minutes, and seconds.
(A) Position Method

(B) Bracket Method
Terminology

- **Dimension** is the numerical value that defines the size or geometric characteristic of a feature.
- **Basic dimension** is the numerical value defining the theoretically exact size of a feature.
- **Reference dimension** is the numerical value enclosed in parentheses provided for information only and is not used in the fabrication of the part.
TEXT HEIGHT .125"

A  2.000
B  .500
C  .375
D  .125

Use guidelines for hand drawings

Decimal dimensioning

TEXT HEIGHT 3mm

A  50.8
B  12.7
C  9.5
D  0.95

Use guidelines for hand drawings

Millimeter dimensioning
Terminology

- **Dimension line** is the thin solid line which shows the extent and direction of a dimension.

- **Arrows** are placed at the ends of dimension lines to show the limits of the dimension.

- **Extension line** is the thin solid line perpendicular to a dimension line indicating which feature is associated with the dimension.
Terminology

- **Leader line** is the thin solid line used to indicate the feature with which a dimension, note, or symbol is associated.

- **Tolerance** is the amount a particular dimension is allowed to vary.

- **Plus and minus dimensioning** is the allowable positive and negative variance from the dimension specified.
Terminology

- **Limits of size** is the largest acceptable size and the minimum acceptable size of a feature.
  - The largest acceptable size is expressed as the maximum material condition (MMC).
  - The smallest acceptable size is expressed as the least material condition (LMC).
**Terminology**

- **Diameter symbol** is the symbol which is placed preceding a numerical value indicating that the associated dimension shows the diameter of a circle. The symbol used is the Greek letter \( \phi \).

- **Radius symbol** is the symbol which is placed preceding a numerical value indicating that the associated dimension shows the radius of a circle. The radius symbol used is the capital letter \( R \).
1. Dimension line
2. Basic dimension
3. Reference dimension
4. Visible gap
5. Arrow
6. Extension line
7. Limits of size
8. Diameter symbol
9. Leader line
10. Plus and minus dimensioning
11. Leader line
12. Radius symbol
13. ALL FORMATS +.05 UNLESS OTHERWISE NOTED
Terminology

- **Datum** is the theoretically exact point used as a reference for tabular dimensioning.
Dimensions are used to describe the size and location of features on parts for manufacture. The basic criterion is, "What information is necessary to make the object?" Dimensions should not be excessive, either through duplication or dimensioning a feature more than one way.
Basic Concepts

- **Size dimension** might be the overall width of the part or the diameter of a drilled hole.
- **Location dimension** might be length from the edge of the object to the center of the drilled hole.
Basic Concepts

- **Size dimensions**
  - Horizontal
  - Vertical
  - Diameter
  - Radius

- **Location and Orientation**
  - Horizontal
  - Vertical
  - Angle
Basic Concepts

- **Rectangular coordinate dimensioning**, a base line (or datum line) is established for each coordinate direction, and all dimensions specified with respect to these baselines. This is also known as *datum dimensioning*, or *baseline dimensioning*. All dimensions are calculated as X and Y distances from an *origin point*, usually placed at the lower left corner of the part.
Datum surface for horizontal dimensions (X)

Datum surface for vertical dimensions (Y)

Origin (0,0)

Symbol
<table>
<thead>
<tr>
<th>A</th>
<th>B</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hole diameter</td>
<td>20</td>
</tr>
</tbody>
</table>
Standard Practices - Placement

- Dimension placement depends on the space available between extension lines. When space permits, dimensions and arrows are placed between the extension lines.
The minimum distance from the object to the first dimension is 10mm (3/8 inch). The minimum spacing between dimensions is 6mm (1/4 inch).

There should be a visible gap between an extension line and the feature to which it refers.

Extension lines should extend about 1mm (1/32 inch) beyond the last dimension line.
Standard Practices - Grouping

- Dimensions should be *grouped* for uniform appearance as shown.
Standard Practices - Staggering

- Where there are several parallel dimensions, the values should be **staggered**.
Extension lines are used to refer a dimension to a particular feature and are usually drawn perpendicular to the associated dimension line. Where space is limited, extension lines may be drawn at an angle.
Extension lines should not cross dimension lines, and should avoid crossing other extension lines whenever possible.

When extension lines cross object lines or other extension lines, they are not broken.

When extension lines cross or are close to arrowheads, they are broken for the arrowhead.
Standard Practices - Extension lines

- When the location of the center of a feature is being dimensioned, the center line of the feature is used as an extension line.

- When a point is being located by extension lines only, the extensions lines must pass through the point.
Standard Practices- Limited length or areas

- When it is necessary to define a limited length or area that is to receive additional treatment (such as the knurled portion of a shaft), the extent of the limits may be shown by a chain line. The chain line is drawn parallel to the surface being defined.
Standard Practices - Reading Direction

- All dimension and note text must be oriented to be read from the bottom of the drawing (relative to the drawing format).
- Placement of all text to be read from the bottom of the drawing is called **unidirectional dimensioning**.
- **Aligned dimensions** have text placed parallel to the dimension line with vertical dimensions read from the right of the drawing sheet.

![Diagram showing unidirectional and aligned dimensions](image)
Standard Practices- View Dimensioning

- Dimensions are to be kept outside of the boundaries of views of objects wherever practical.
- Dimensions may be place within the boundaries of objects in cases where extension or leader lines would be too long, or where clarity would be improved.
Standard Practices - Out-of-Scale Dimensions

- If it is necessary to include a dimension which is out of scale, the out of scale dimension text must be underlined.
The symbol X is used to indicate the number of times a feature is to be repeated. The number of repetitions, followed by the symbol X and a space precedes the dimension text.
Detail Dimensions

- Holes
  - Diameters must be dimensioned with the diameter symbol preceding the numerical value.
  - When holes are dimensioned with a leader line, the line must be radial. A radial line is one that passes through the center of a circle or arc if extended.
- **Chamfers**

  - Internal Chamfers

- **Slotted holes**
Keyseat and Keyway
Summary
Concentric circles
Arcs
Screw Threads
Grooves
<table>
<thead>
<tr>
<th>No. of Sheet-Metal Gage</th>
<th>Manufacturers' Standard Gage for Steel</th>
<th>Birmingham Gage (B. G.) for Sheets, Hoops</th>
<th>Galvanized Sheet Gage</th>
<th>Zinc Gage</th>
<th>No. of Sheet-Metal Gage</th>
<th>Manufacturers' Standard Gage for Steel</th>
<th>Birmingham Gage (B. G.) for Sheets, Hoops</th>
<th>Galvanized Sheet Gage</th>
<th>Zinc Gage</th>
</tr>
</thead>
<tbody>
<tr>
<td>150</td>
<td>1.000</td>
<td></td>
<td></td>
<td></td>
<td>20</td>
<td>0.0350</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>140</td>
<td>0.9680</td>
<td></td>
<td></td>
<td></td>
<td>21</td>
<td>0.0329</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>130</td>
<td>0.9167</td>
<td></td>
<td></td>
<td></td>
<td>22</td>
<td>0.0299</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>120</td>
<td>0.8760</td>
<td></td>
<td></td>
<td></td>
<td>23</td>
<td>0.0269</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>110</td>
<td>0.8333</td>
<td></td>
<td></td>
<td></td>
<td>24</td>
<td>0.0239</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>100</td>
<td>0.7917</td>
<td></td>
<td></td>
<td></td>
<td>25</td>
<td>0.0209</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>90</td>
<td>0.7500</td>
<td></td>
<td></td>
<td></td>
<td>26</td>
<td>0.0179</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>80</td>
<td>0.7063</td>
<td></td>
<td></td>
<td></td>
<td>27</td>
<td>0.0149</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>70</td>
<td>0.6666</td>
<td></td>
<td></td>
<td></td>
<td>28</td>
<td>0.0119</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>60</td>
<td>0.6250</td>
<td></td>
<td></td>
<td></td>
<td>29</td>
<td>0.0090</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>50</td>
<td>0.5833</td>
<td></td>
<td></td>
<td></td>
<td>30</td>
<td>0.0060</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>40</td>
<td>0.5416</td>
<td></td>
<td></td>
<td></td>
<td>31</td>
<td>0.0030</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>0.5000</td>
<td></td>
<td></td>
<td></td>
<td>32</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>0.4452</td>
<td></td>
<td></td>
<td></td>
<td>33</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>0.3964</td>
<td></td>
<td></td>
<td></td>
<td>34</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>0.3532</td>
<td></td>
<td></td>
<td></td>
<td>35</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.3147</td>
<td></td>
<td></td>
<td></td>
<td>36</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.2591</td>
<td></td>
<td></td>
<td></td>
<td>37</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.2242</td>
<td></td>
<td></td>
<td></td>
<td>38</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>0.2048</td>
<td></td>
<td></td>
<td></td>
<td>39</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.1943</td>
<td></td>
<td></td>
<td></td>
<td>40</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.1793</td>
<td></td>
<td></td>
<td></td>
<td>41</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>0.1644</td>
<td></td>
<td></td>
<td></td>
<td>42</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>0.1495</td>
<td></td>
<td></td>
<td></td>
<td>43</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>0.1345</td>
<td></td>
<td></td>
<td></td>
<td>44</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>0.1196</td>
<td></td>
<td></td>
<td></td>
<td>45</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>0.1046</td>
<td></td>
<td></td>
<td></td>
<td>46</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>0.0897</td>
<td></td>
<td></td>
<td></td>
<td>47</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>0.0747</td>
<td></td>
<td></td>
<td></td>
<td>48</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>0.0573</td>
<td></td>
<td></td>
<td></td>
<td>49</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>0.0528</td>
<td></td>
<td></td>
<td></td>
<td>50</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>0.0478</td>
<td></td>
<td></td>
<td></td>
<td>51</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>0.0428</td>
<td></td>
<td></td>
<td></td>
<td>52</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>0.0418</td>
<td></td>
<td></td>
<td></td>
<td>53</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Dimension Techniques

- Contour Dimensioning
  - contours or shapes of the object are dimensioned in their most descriptive view. For example, the radius of a arc would be dimensioned where it appears as an arc and not as a hidden feature.
Dimension Techniques

- Geometric Breakdown
  - A part is to break the part into its geometric configurations.
Dimension Process

Step 1

Step 2

Step 3

Step 4

Alternate Method
Dimension Guidelines

- The primary guideline is that of **clarity** and whenever two guidelines appear to conflict, the method which most clearly communicates the size information shall prevail.
  - Every dimension must have an associated tolerance, and that tolerance must be clearly shown on the drawing.
  - Avoid over-dimensioning a part. Double dimensioning of a feature is not permitted.
  - Dimensions should be placed in the view which most clearly describes the feature being dimensioned.
Dimension Guidelines

- A minimum spacing between the object and dimensions and between dimensions must be maintained.
- A visible gap shall be placed between the end of extension lines and the feature to which they refer.
- Manufacturing methods should not be specified as part of the dimension unless no other method of manufacturing is acceptable.
- Placing dimensions within the boundaries of a view should be avoided whenever practicable.
Dimension Guidelines

- Dimensions for materials typically manufactured to gages or code numbers shall be specified by numerical values.
- Unless otherwise specified, angles shown on drawings are assumed to be 90 degrees.
- Dimensioning to hidden lines should be avoided whenever possible. Hidden lines are less clear than visible lines.
- The depth of blind, counterbored, or countersunk holes may be specified in a note along with the diameter.
Dimension Guidelines

- Diameters, radii, squares, counterbores, spotfaces, countersinks, and depth should be specified with the appropriate symbol preceding the numerical value.
- Leader lines for diameters and radii should be radial lines.
Tolerancing

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

- Tolerances are used to control the amount of variation inherent in all manufactured parts. In particular, tolerances are assigned to mating parts in an assembly.
One of the great advantages of using tolerances is that it allows for interchangeable parts, thus permitting the replacement of individual parts.

Tolerances are used in production drawings to control the manufacturing process more accurately and control the variation between parts.
Tolerancing

- Tolerance representation
  - Direct limits or as tolerance values applied directly to a dimension.
  - Geometric tolerances
  - Notes referring to specific condition.
Tolerancing

- Tolerance representation
  - Plus/Minus

(A) Unilateral tolerancing
(B) Bilateral tolerancing
Tolerancing

- Important terms
  - **Nominal size** a dimension used to describe the general size usually expressed in common fractions.
  - **Basic size** the theoretical size used as a starting point for the application of tolerances.
  - **Actual size** the measured size of the finished part after machining.
Tolerancing

- Important terms
  - **Limits** the maximum and minimum sizes shown by the toleranced dimension.
  - **Allowance** is the minimum clearance or maximum interference between parts.
  - **Tolerance** is the total variance in a dimension which is the difference between the upper and lower limits. The tolerance of the slot in Figure 14.50 is .004" and the tolerance of the mating part is .002".
Tolerancing

- Important terms
  - **Maximum material condition (MMC)** is the condition of a part when it contains the most amount of material. The MMC of an external feature such as a shaft is the upper limit. The MMC of an internal feature such as a hole is the lower limit.
Tolerancing

Important terms

- **Least material condition (LMC)** is the condition of a part when it contains the least amount of material possible. The LMC of an external feature is the lower limit of the part. The LMC of an internal feature is the upper limit of the part.
Tolerancing

- Upper Limit (MMC)
- Lower Limit (LMC)
- Actual measured dimension
- Engineering Dimensioned Drawing
- Machined Part
- Tolerance .002
- Clearance varies
- Tolerance .004
  - Maximum .007
  - Average .004
  - Minimum .001
Tolerancing

- Fit types
  - **Clearance fit** occurs when two toleranced mating parts will always leave a space or clearance when assembled.
  - **Interference fit** occurs when two toleranced mating parts will always interfere when assembled.
  - **Transition fit** occurs when two toleranced mating parts will sometimes be an interference fit and sometimes be a clearance fit when assembled.
Allowance always equals smallest hole minus largest shaft.
Tolerancing
Tolerancing

- Metric Limits and Fits
  - Basic size
  - Deviation
  - Upper Deviation
  - Lower Deviation
  - Fundamental Deviation
Tolerancing

- Tolerance
- Tolerance zone
- International tolerance grade
- Hole basis
- Shaft basis
Tolerancing

- Symbols and Definitions
- Methods

\[
\begin{align*}
40H8 & \quad 40H8 \quad \left( \begin{array}{c}
40.039 \\
40.000
\end{array} \right) \quad \left( \begin{array}{c}
40.039 \\
40.000
\end{array} \right) \quad 40H8 \\
(A) & \quad (B) & \quad (C)
\end{align*}
\]
**Tolerancing**

- Standard Hole basis table; limits

<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>Fit</td>
<td>Hole H9</td>
<td>Shaft d9</td>
</tr>
<tr>
<td>40 MAX</td>
<td>40.160</td>
<td>39.880</td>
<td>0.440</td>
<td>40.062</td>
<td>39.920</td>
</tr>
<tr>
<td>40 MIN</td>
<td>40.000</td>
<td>39.720</td>
<td>0.120</td>
<td>40.000</td>
<td>39.858</td>
</tr>
<tr>
<td>50 MAX</td>
<td>50.160</td>
<td>49.870</td>
<td>0.450</td>
<td>50.062</td>
<td>49.920</td>
</tr>
<tr>
<td>50 MIN</td>
<td>50.000</td>
<td>49.710</td>
<td>0.130</td>
<td>50.000</td>
<td>49.858</td>
</tr>
<tr>
<td>60 MAX</td>
<td>60.190</td>
<td>59.860</td>
<td>0.520</td>
<td>60.074</td>
<td>59.900</td>
</tr>
<tr>
<td>60 MIN</td>
<td>60.000</td>
<td>59.670</td>
<td>0.140</td>
<td>60.000</td>
<td>59.826</td>
</tr>
<tr>
<td>80 MAX</td>
<td>80.190</td>
<td>79.550</td>
<td>0.530</td>
<td>80.074</td>
<td>79.900</td>
</tr>
<tr>
<td>80 MIN</td>
<td>80.000</td>
<td>79.660</td>
<td>0.150</td>
<td>80.000</td>
<td>79.826</td>
</tr>
<tr>
<td>100 MAX</td>
<td>100.220</td>
<td>99.830</td>
<td>0.610</td>
<td>100.087</td>
<td>99.890</td>
</tr>
<tr>
<td>100 MIN</td>
<td>100.000</td>
<td>99.610</td>
<td>0.170</td>
<td>100.000</td>
<td>99.793</td>
</tr>
<tr>
<td>120 MAX</td>
<td>120.220</td>
<td>119.820</td>
<td>0.620</td>
<td>120.087</td>
<td>119.880</td>
</tr>
<tr>
<td>120 MIN</td>
<td>120.000</td>
<td>119.600</td>
<td>0.180</td>
<td>120.000</td>
<td>119.793</td>
</tr>
<tr>
<td>160 MAX</td>
<td>160.250</td>
<td>159.790</td>
<td>0.710</td>
<td>160.100</td>
<td>159.855</td>
</tr>
<tr>
<td>160 MIN</td>
<td>160.000</td>
<td>159.540</td>
<td>0.210</td>
<td>160.000</td>
<td>159.755</td>
</tr>
<tr>
<td>200 MAX</td>
<td>200.290</td>
<td>199.760</td>
<td>0.820</td>
<td>200.115</td>
<td>199.830</td>
</tr>
<tr>
<td>200 MIN</td>
<td>200.000</td>
<td>199.470</td>
<td>0.240</td>
<td>200.000</td>
<td>199.715</td>
</tr>
<tr>
<td>250 MAX</td>
<td>250.290</td>
<td>249.720</td>
<td>0.860</td>
<td>250.115</td>
<td>249.830</td>
</tr>
<tr>
<td>250 MIN</td>
<td>250.000</td>
<td>249.430</td>
<td>0.280</td>
<td>250.000</td>
<td>249.715</td>
</tr>
<tr>
<td>300 MAX</td>
<td>300.320</td>
<td>299.670</td>
<td>0.970</td>
<td>300.130</td>
<td>299.810</td>
</tr>
<tr>
<td>300 MIN</td>
<td>300.000</td>
<td>299.350</td>
<td>0.330</td>
<td>300.000</td>
<td>299.690</td>
</tr>
<tr>
<td>400 MAX</td>
<td>400.360</td>
<td>399.600</td>
<td>1.120</td>
<td>400.140</td>
<td>399.790</td>
</tr>
<tr>
<td>400 MIN</td>
<td>400.000</td>
<td>399.240</td>
<td>0.400</td>
<td>400.000</td>
<td>399.650</td>
</tr>
<tr>
<td>500 MAX</td>
<td>500.400</td>
<td>499.520</td>
<td>1.280</td>
<td>500.155</td>
<td>499.770</td>
</tr>
<tr>
<td>500 MIN</td>
<td>500.000</td>
<td>499.120</td>
<td>0.480</td>
<td>500.000</td>
<td>499.615</td>
</tr>
</tbody>
</table>
Tolerancing

- Hole basis system: fits
Tolerancing

- Shaft basis system; fits
<table>
<thead>
<tr>
<th>ISO Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>H11/c11</td>
<td>Loose running fit for wide commercial tolerances or allowances on external members</td>
</tr>
<tr>
<td>H9/d9</td>
<td>Free running fit not for use where accuracy is essential, but good for large temperature variations, high running speeds, or heavy journal pressures</td>
</tr>
<tr>
<td>H8/f7</td>
<td>Close running fit for running on accurate machines and for accurate location at moderate speeds and journal pressures</td>
</tr>
<tr>
<td>H7/g6</td>
<td>Sliding fit not intended to run freely but to move and turn freely and locate accurately</td>
</tr>
<tr>
<td>H7/h6</td>
<td>Locational clearance fit provides snug fit for locating stationary parts but can be freely assembled and disassembled</td>
</tr>
<tr>
<td>H7/k6</td>
<td>Locational transition fit for accurate location; a compromise between clearance and interference</td>
</tr>
<tr>
<td>H7/n6</td>
<td>Locational transition fit for more accurate location where greater interference is permissible</td>
</tr>
<tr>
<td>H7/p6*</td>
<td>Locational interference fit for parts requiring rigidity and alignment with prime accuracy of location but without special bore pressure requirements</td>
</tr>
<tr>
<td>H7/s6</td>
<td>Medium drive fit for ordinary steel parts or shrink fits on light sections; the tightest fit usable with cast iron</td>
</tr>
<tr>
<td>H7/u6</td>
<td>Force fit suitable for parts that can be highly stressed or for shrink fits where the heavy pressing forces required are impractical</td>
</tr>
</tbody>
</table>

*Transition fit for basic sizes in range from 0 through 3 mm
Tolerancing

- Standard Precision Fit; English Units
  - Running and Sliding (RC)
  - Clearance Locational (LC)
  - Transition Locational (LT)
  - Interference Locational (LN)
  - Force and Shinks (FN)
Geometric Dimensioning and Tolerancing

- GDT is a method of defining parts based on how they function, using standard ASME/ANSI symbols.
Within the last 15 years there has been considerable interest in GDT, in part because of the increased popularity of statistical process control. This control process, when combined with GDT, helps reduce or eliminate inspection of features on the manufactured object. The flipside is that the part must be tolerated very efficiently; this is where GDT comes in.
Geometric Dimensioning and Tolerancing

- Another reason for the increased popularity of GDT is the rise of worldwide standards, such as ISO 9000, which require universally understood and accepted methods of documentation.
GDT-Symbols

Datum feature  Datum target  Target point  Concentricity  Circularity  MMC  Free state  Tangent plane

LMC  RFS  Projected tolerance zone  Parallelism  Flatness  Cylindricity  Between

Diameter  Position  All around (profile) (ISO-none)  Profile surface  Profile line  Straightness

Perpendicularity  Angularity  Runout circular  Runout total  Counterbore or spotface (ISO-proposed)

Countersink (ISO-proposed)  Depth (or deep) (ISO-proposed)  Dimension origin  Conical taper  Square (shape)

Reference  Arc length (ISO-none)  Slope  Places, times or by  Radius  Statistical Tolerance

SR  S  Spherical radius  Spherical diameter  Feature control frame  Symmetry

Dimension not to scale  Miscellaneous symbols
GDT

- Feature control frames
GDT

- MMC/LMC
- Datums
- Geometric Controls
  - Form
  - Orientation
  - Position
Forms

- Straightness
  - Line element
  - Axis
GDT

- Forms
  - Circularity

![Diagram of GDT Forms and Tolerance Zone with Inspection Method](image-url)
GDT

- Forms
  - Flatness

![Diagram showing drawing, tolerance zone, and inspection methods for GDT flatness.](image)
GDT

- Forms
  - Cylindricity
GDT

- Orientation
  - Parallelism
GDT

- Orientation
  - Perpendicularity
GDT

- Orientation
  - Angularity
GDT

- Orientation
  - Line profile
GDT

- Orientation
  - Surface profile
GDT

- Location
  - Concentricity
GDT

- Location
  - Runout
GDT

- Location
  - Position
GDT

- Location
- Position
Tolerance Calculation

- *Floating fastener tolerancing* is used to confirm that loose bolts, screws or other fasteners have the standard clearance in their holes.

- *Fixed fastener tolerancing* is measured the same as with floating fasteners except that the fastener is already fixed/located on one of the mating parts and the tolerance is now divided between the parts.
GDT

- Tolerance Calculation
  - *Hole diameter tolerancing* is used to calculate the MMC of the hole.
GDT

- Design Application
  - Five-Step
    - Isolate and define the functions of the features/part.
    - Prioritize the functions.
    - Identify the datum reference frame based on functional priorities.
    - Select the proper control(s).
    - Calculate the tolerance values.