Dimensioning circular features

Diameters

- Dia of single feature or concentric features – shown in longitudinal view
- Is limited in space or partial view is used (8-2-2)

The size of the dimension is always preceded by \( \Phi \)

Fig. 8-2-1 and 8-2-2
Dimensioning circular features

Radii

- Circular arc is dimensioned with radius
- Radius dim line is in line with or passes through the centre and ends in a arrow head
- Arrow head is never used in centre and size of dim is preceded by R
- When limited space, radial dimension may extend through centre

- If it is inconvenient to place arrowhead between centre and arc it can be placed outside or a leader may be used
- When a dimension is given to the centre of the radius a small cross is drawn at centre
- Extension and dimension lines are used to locate centre
- When centre is unimportant, arc may be located by tangent lines (8-2-3 E)

Fig 8-2-3
Dimensioning circular features

Radii

- When the radius is outside the drawing or interfering with another view, foreshortened dimension line may be used (8-2-3 D)
- Portion of dimension line next to arrowhead should be radial relative to the curved line
- When the radius dimension line is foreshortened and centre is located by coordinate dimensions, the dimensions locating the centre should be shown as ‘not to scale’ or foreshortened

Simple fillet and radii may be dimensioned with a general note

ALL ROUNDS AND FILLETS UNLESS OTHERWISE SPECIFIED
r.20

Or

ALL RADII R5

Fig 8-2-3
Dimensioning circular features

Rounded Ends

- Overall dimensions need to be used for parts having rounded ends
- For fully rounded ends the radius (R) is shown but not dimensioned
- For parts with partially rounded ends the radius (R) is dimensioned
- When a hole and radius have same centre and hole location is more critical than the radius location, either the radius or the overall length should be shown as reference dimension

Fig. 8-2-5 Dimensioning external surfaces with rounded ends.
Dimensioning circular features

Dimensions Chords, Arcs and Angles

- Dimensioning of chords, arcs and angles are shown below

**Fig 8-2-6**

Spherical Features

- Spherical features can be dimensioned either as radius or as diameter but should be used with abbreviation SR or SΦ

**Fig 8-2-7**
Dimensioning circular features

**Cylindrical Holes**

- Plain round holes are dimensioned based on design and manufacturing requirements.
- Leader method is most commonly used.
- When leader is used to specify diameter, Φ is placed before the numerical value.
- Size quantity and depth may be shown in single line.
- For through holes THRU should follow numerical value (if it is not clear in drawing).
- Blind hole – depth of full diameter is included.
- When more than hole of a size is required, the number of holes should be specified.
- Adequate spacing need to be given between the size and quantity of holes.

*Fig 8-2-8* Dimensioning cylindrical holes.
Dimensioning circular features

Minimizing Leaders

- If too many leaders will impair the legibility of the drawing, letters or symbols can be used to identify features.

- Slots are used for compensating inaccuracies.

- Method for location depends on method of manufacture.

- (A) machined out [centre need not be located] and (B) punched out.

Slotted Holes
Dimensioning circular features

Countersinks, Counterbores and spotfaces

- Counterbores, Spotfaces and Countersinks are specified on drawings by dimension symbols or abbreviations.
- They indicate the form of the surface only and do not restrict method of fabrication.
- The dimensions are usually given as a note, preceded by the size of through holes.
- Countersink - angular-sided recess to accommodate the head of flathead screws, rivets, and similar items.
  - Surface diameter and included angle are given. If the tapered section depth is critical – given in note.
  - For counter drilled holes, dia, depth and included angle are given.

Fig 8-2-11

Counterbored and spotfaced holes.
Dimensioning circular features

- **Counterbore** - flat-bottomed, cylindrical recess that permits the head of fastening devices (bolt) to lie recessed to part
  - The dia, depth and corner radius are specified in the note
  - In some cases the thickness of remaining stock can be dimensioned instead of depth

- **Spotface** - area where the surface is matched just enough to provide smooth, level seating for bolt head, nut or washer
  - Dia of the faced area, and depth or remaining thickness given
  - Spotface may be specified by a general note and not delineated in the drawing
  - If no depth is given, minimum depth for specified diameter is taken

Countsunk and counterdrilled holes. Fig 8-2-12
Dimensioning common features

Repetitive features and dimensions (Fig 8-3-1)

- Repetitive features can be specified by use of X with the numeral to indicate number of times or places they are required
  - 8X Φ.281 EQL SP ON Φ2.34
- It can also be specified by using descriptive notes
  - Φ.281 8 HOLES EQL SP ON Φ2.34

Chamfers (Fig 8-3-2)

- The process of chamfering, that is, cutting away the inside or outside piece, is done to facilitate assembly
  - Dimensioned by angle and linear length (if angle not given it is 45°)
  - When small chamfer is permissible (to break sharp corners) no need to draw, only dimension will suffice
  - Chamfers are never measured along angular surface
Dimensioning common features

Slopes

- Slope – slant line representing inclined surface
- Expressed as ratio of height difference at right angles
  - Slope defined as ratio combined with slope symbol
  - Specified by angle
  - Dimensions showing difference in heights of two points from baseline
- D is the preferred method for architectural and structural drawings

Fig. 8-3-3 Dimensioning slopes.
Dimensioning common features

- **Slope** – ratio of the difference in the diameters of two sections (perpendicular to the axis of a cone to the distance between these two sections)
- In taper symbol vertical leg is always left and precedes ratio figures following may be used in suitable combinations to dimension tapered features
  - Diameter (or width) at one end of the tapered feature
  - Length of the tapered feature
  - Rate of taper
  - Included angle
  - Taper ratio
  - Diameter at a selected cross section
  - Dimension locating the cross section

![Fig 8-3-4 Taper](image)
Dimensioning common features

Knurls

- Knurling – specified in terms of type, pitch and diameter before and after knurling.
  - The letter P proceeds the pitch number
  - Diameter is omitted if control not required
  - If only portions of feature require knurling, axial dimensions must be provided
  - If fit is required, knurling is indicated by note including type, pitch, tolerated dia before knurling and minimum acceptable dia after
  - Pitch is expressed in terms of teeth per inch

Fig 8-3-5
Dimensioning common features

**Formed parts**

- In dimensioning formed parts, the inside radius is usually specified, rather than the outside radius.

- All forming dimensions should be shown on the same side if possible.

- Dimensions apply to the side on which it is shown unless otherwise specified.

---

**Fig 8-3-6**

Dimensioning theoretical points of intersection.
Dimensioning common features

Undercuts

- The operation of undercutting or necking, i.e. cutting a recess in a diameter, is done to permit two parts to come together.
- Indicated in the drawing by note listing the width first and then the diameter.
- If radius shown at bottom of undercut, assume radius is half of width if not specified (dia applies to centre).
- When undercut size is unimportant it may be left off in the drawing without dimension.

Dimensioning undercuts.

Fig 8-3-7

(A) CHAMFER AND UNDERCUT APPLICATION

(B) PLAIN UNDERCUT

(C) UNDERCUT WITH RADIUS

METRIC
Dimensioning common features

Limited length and areas

- Sometimes dimensioning limited length or area may be required.
- Length of a surface is indicated by a chain line drawn parallel and adjacent to the surface.
- When indicating an area of surface, the area is cross-hatched within the chain line boundary.

Fig. 8-3-8  Dimensioning limited lengths and areas.

Fig 8-3-8
Dimensioning common features

Wire, Sheet Metal, and Drill Rod

- Wire, sheet metal and drill rods, which are manufactured to gage or code sizes should be shown by their decimal dimensions; but gage numbers, drill letters, and so on, may be shown in parantheses following the dimensions.

Sheet -- .141 (NO 10 USS GA)

-- .081 (NO 12 B & S GA)
Dimensioning methods

- Choice dimensions and dimensioning methods depend (to some extent) on how the parts will be produced (unit or mass production).
- Unit production: each part is made separately using general purpose tools and machines.
- Mass production: parts produced in quantity using special tools.
- Linear or angular dimensions locate feature from one another (or from a datum).
- Point to point dimensions are done for simple parts and dimension from datum. May be need for parts with more than one critical dimension must mate with another part.
- Hence method of dimensioning becomes critical.
Dimensioning methods

Rectangular Coordinate Dimensioning

This is a method for indicating, distance, location and size by means of linear dimensions measured parallel or perpendicular to reference axes or datum planes that are perpendicular to one another.

Co-ordinate dimensioning with dimension lines must clearly identify the datum features from the which the dimensions originate.

Fig. 8-4-1  Rectangular coordinate dimensioning.

Fig 8-4-1
Dimensioning methods

Rectangular Coordinates for arbitrary points

- Coordinates for arbitrary points of reference without a grid appear adjacent to each point (Fig. 8-4-2) or in tabular form (Fig. 8-4-3).
Dimensioning methods

Rectangular Coordinate Dimensioning without Dimension Lines

- Dimensions may be shown on extension lines without the use of dimension lines or arrowheads. The base lines may be zero coordinates, or they may be labeled as X, Y, and Z.

Fig 8-4-4

Fig. 8-4-4 Rectangular coordinate dimensioning without dimension lines (arrowless dimensioning).
Dimensioning methods

Tabular Dimensioning

- Tabular dimensioning is a type of coordinate dimensioning in which dimensions from mutually perpendicular planes are listed in a table on the drawing rather than on the pictorial delineation.

<table>
<thead>
<tr>
<th>HOLE DIA</th>
<th>HOLE SYMBOL</th>
<th>LOCATION</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>5.6</td>
<td>A₁</td>
<td>X 60</td>
<td>Y 40 Z 18</td>
</tr>
<tr>
<td>4.8</td>
<td>B₁ B₂ B₃ B₄</td>
<td>X 10 75 60 80</td>
<td>Y 40 THRU</td>
</tr>
<tr>
<td>4</td>
<td>C₁ C₂ C₃ C₄</td>
<td>X 18 55 10 30</td>
<td>Y 40 THRU</td>
</tr>
<tr>
<td></td>
<td>C₅ C₆ C₆</td>
<td>X 75 18 18</td>
<td>Y 20 THRU</td>
</tr>
<tr>
<td>3.2</td>
<td>D₁</td>
<td>X 55</td>
<td>Y 8 Z 12</td>
</tr>
<tr>
<td>8.1</td>
<td>E₁</td>
<td>X 42</td>
<td>Y 20 Z 12</td>
</tr>
</tbody>
</table>

- This method is mainly used when drawings require the location of large number of similarly shaped features when parts for numerical control are dimensioned.
Dimensioning methods

Polar Coordinate Dimensioning

- It is commonly used in circular planes of circular configurations of features.
- This method indicates the position of a point, line, or surface by means of linear dimension and an angle (other than 90°) – implied by horizontal and vertical lines.

Fig 8-4-7
Dimensioning methods

Chordal Dimensioning

- It may also be used for the spacing of points on circumference of a circle relative to a datum, where manufacturing methods indicate that this will be convenient

Fig 8-4-7
Dimensioning methods

True Position Dimensioning

- This has many advantages over coordinate dimensioning system.
- True position of the hole is dimensioned with respect to 3 datums.
- Used while tolerancing.

Fig 8-4-8

True-position dimensioning.

Fig 8-4-8
Dimensioning methods

Chain Dimensioning

It is applied on a point to point basis.

Disadvantage - this may result in an undesirable accumulation of tolerances between individual features.
Dimensioning methods

Datum or Common-Point Dimensioning

- When several dimension emanate from a common reference point or line it called common point or datum dimensioning
- Here the dimensioning can be either parallel or super imposed dimensioning

Fig 8-4-10

Common-point (baseline) dimensioning
Dimensioning methods

Super imposed running Dimensioning

- This simplified parallel dimensioning used when space is a constraint
- Dimensions to be placed near arrowhead in line with corresponding extension line
- Origin is indicated by circle
- It may be advantageous to use superimposed running dimensions in two directions
- Origins can also be centre of holes or other features

Fig 8-4-11
Outline

- Limits and tolerances
- Fits and allowances
- Surface texture
Why give Tolerance?

- Manufacturing Practice is 6000 years old, while tolerancing is only 80 years old.
- Because only then, they came to realize that exact dimensions and shapes cannot be attained.
- What people thought were exact, have deviations, given the advanced metrology equipment that we have now.
- Once, it is understood that the variations cannot be avoided, the best way is to limit the variations.
- Tolerances are permissible variations in the specified form, size or location of individual features or a part from that shown on a drawing.
- If 1.500±.004 is given, the part can be anywhere between 1.496 to 1.504.
Limits and tolerances

- In a same part different portions have different Accuracy
- Greater accuracy = Increase in cost. Not necessary needed to on all parts.
- Accuracy is mainly high on assembled parts so that they function properly
Limits and tolerances

Key Concepts

- **Actual Size** - is the measured size

- **Basic Size** - The theoretical size from which the limits for that dimension are derived by the application of the allowance and tolerance.

- **Design Size** - refers to the size from which the limits of size are derived by the application of tolerances.

- **Limits of Size** - are the maximum and minimum sizes permissible for a specific dimension.

- **Nominal Size** - is the designation used for the purpose of general identification.
Limits and tolerances

Key Concepts

- **Tolerance** - The tolerance on a dimension is the total permissible variation in the size of a dimension. The tolerance is the difference between the limits of size. ± .004

- **Bilateral Tolerance** - Variation is permitted in both directions from the specified dimension.

- **Unilateral Tolerance** - Variation is permitted in only one direction from the specified dimension.

- **Maximum Material Size** - The limit of size of a feature that results in the part containing the maximum amount of material. Thus it is the maximum limit of size for a shaft or an external feature, or the minimum limit of size for a hole or internal feature. MMC
## Limits and tolerances

### Terminology

<table>
<thead>
<tr>
<th>TERMINOLOGY</th>
<th>EXAMPLE</th>
<th>EXPLANATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>BASIC SIZE</td>
<td>1.500</td>
<td></td>
</tr>
<tr>
<td>BASIC SIZE WITH TOLERANCE ADDED</td>
<td>1.500 ± 0.004</td>
<td>HALF OF TOTAL TOLERANCE</td>
</tr>
<tr>
<td>LIMITS OF SIZE</td>
<td>1.504</td>
<td>LARGEST AND SMALLEST SIZES PERMITTED</td>
</tr>
<tr>
<td>LIMITS OF SIZE</td>
<td>1.496</td>
<td></td>
</tr>
<tr>
<td>TOLERANCE</td>
<td>0.008</td>
<td>DIFFERENCE BETWEEN LIMITS OF SIZE</td>
</tr>
</tbody>
</table>
Limits and tolerances

Tolerance Specification

- All dimensions require tolerance (except those identified as reference, max or min, or stock) - No critical areas.
- As specified limits of tolerances shown directly on the drawing for a specified dimension.
- As plus-and-minus tolerancing.
- Combining a dimension with a tolerance symbol class of fit. (Unit 8-6.)
Limits and tolerances

Tolerance Specification

- In a general tolerance note, referring to all dimensions on the drawing for which tolerances are not otherwise specified.
- In the form of a note referring to specific dimensions.
- Tolerances on dimensions that locate features may be applied directly to the locating dimensions or by the positional tolerancing method described in Chap. 16.
- Tolerancing applicable to the control of form and run out, referred to as geometric tolerance, is covered in detail in Chap. 16.
Limits and tolerances

Direct Tolerancing Methods

- Tolerance applied directly to dimension is expressed
  - Limit Dimensioning
  - Plus and Minus Tolerancing

- Limit Dimensioning - The high limit (max value) is placed above the low limit (min value). If placed in single line, low precedes the high and separated by a -, e.g. Ø .252 - .250

- If high or low dimension has digits to the right of the decimal, both are expressed in same number of decimal places

<table>
<thead>
<tr>
<th>Dimension 1</th>
<th>Dimension 2</th>
<th>Tolerance 1</th>
<th>Tolerance 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>30.75</td>
<td>30</td>
<td>.750</td>
<td>.75</td>
</tr>
<tr>
<td>30.00</td>
<td>30</td>
<td>.748</td>
<td>.748</td>
</tr>
</tbody>
</table>

Limit dimensioning application.
Limits and tolerances

Direct Tolerancing Methods

- **Plus & Minus tolerancing**
  - Dimension is placed first followed by the ± expression of tolerance
  - Plus should be above the minus value
  - This can be divided further into
    - **Bilateral Tolerancing**
    - **Unilateral Tolerancing**

*Fig. 8-5-5*  Plus-and-minus tolerancing.
Limits and tolerances

Metric Tolerancing

An application of unilateral tolerancing is shown in Fig. 8-5-6B.

Special case

Fig. 8-5-6 Application of tolerances.
Limits and tolerances

Metric Tolerancing

- The dimension need not to be shown to the same number of decimal places as its tolerance. For example:

- When bilateral tolerancing is used, both the + and the – values have the same number of decimal places, using 0s when necessary.

- When unilateral tolerancing is used, and either the + and the – value is nil, a single 0 is shown without the + or – sign.
Inch Tolerancing

- The dimension need to be shown to the same number of decimal places as its tolerance. For example:

- .500 ± .004 not .5 ± .004
Limits and tolerances

General Tolerance Notes

- This simplifies the drawing and saves layout. For Example

<table>
<thead>
<tr>
<th>Dimension (mm)</th>
<th>Tolerance</th>
</tr>
</thead>
<tbody>
<tr>
<td>UP TO 100</td>
<td>± 0.1</td>
</tr>
<tr>
<td>FROM 100.01 TO 300</td>
<td>± 0.75</td>
</tr>
<tr>
<td>FROM 300.01 TO 600</td>
<td>± 0.5</td>
</tr>
<tr>
<td>OVER 600</td>
<td>± 1</td>
</tr>
</tbody>
</table>

EXCEPT WHERE STATED OTHERWISE, TOLERANCES ON FINISHED DECIMAL DIMENSIONS ±0.1

Or

EXCEPT WHERE STATED OTHERWISE, TOLERANCES ON FINISHED DECIMAL DIMENSIONS TO BE AS FOLLOWS
Limits and tolerances

Tolerance Accumulation

- It is important to consider the effect of each tolerance with respect to other tolerance.
- Not to permit chain of tolerances to build up cumulative tolerance between surfaces or points that have important relation to one another.
- When position of surface is controlled by more than one tolerance, the tolerances are cumulative.
- Tolerance accumulation in different dimensioning methods:
  - Chain dimensioning
  - Datum dimensioning
  - Direct dimensioning
Limits and tolerances

Chain Dimensioning

- Maximum variation between any two features is equal to sum of tolerances on the intermediate distances.
- This results in greatest tolerance accumulation
- Accumulated tolerance between X and Y is ±0.08
Limits and tolerances

Datum Dimensioning

- Maximum variation between any two features is equal to sum of tolerances on two dimensions from the datum to feature
  - This results in lesser tolerance accumulation
  - Accumulated tolerance between X and Y is ±0.04
Limits and tolerances

Direct Dimensioning

- Maximum variation between any two features is controlled by tolerance on the dimensions between features
- This results in least tolerance accumulation
- Accumulated tolerance between X and Y is ±0.02
Limits and tolerances

Additional rules for dimensioning

- The engineering intent must be clearly defined.
- Dimensions must be complete enough to describe the total geometry of each feature. Determining a shape by measuring its size on a drawing or by assuming a distance or size is not acceptable.
- Dimensions should be selected and arranged to avoid unsatisfactory accumulation of tolerances, to preclude more than one interpretation, and to ensure a proper fit between mating parts.
- The finished part should be defined without specifying manufacturing methods. Thus only the diameter of a hole is given, without indicating whether it is to be drilled, reamed, punched, or made by any other operation. – Consultation.
- Dimensions must be selected to give required information directly.
Limits and tolerances

Additional rules for dimensioning

- Dimensions should preferably be shown in true profile views and refer to visible outlines rather than to hidden lines.

- Drawings that illustrate part surfaces or center lines at right angles to each other, but without an angular dimension, are interpreted as being $90^\circ$ between these surfaces or center lines. Actual surfaces, axes, and center planes may vary within their specified tolerance of perpendicularly.

- Dimension lines are placed outside the outline of the part and between the views unless the drawing may be simplified or clarified by doing otherwise.

- Dimension lines should be aligned, if practicable, and should be grouped for uniform appearance.
Fits and Allowances

- For assembled parts to function properly and to facilitate interchangeable manufacturing, there is a necessity to permit only certain amount of tolerances and allowances between them.

**Fits**

- The fit between two mating parts is the relationship between them with respect to the amount of clearance or interference present when they are assembled. There are three basic types of fits: clearance, interference, and transition.
Fits and Allowances

Fits

- **Clearance Fit** - A fit between mating parts having limits of size so prescribed that a clearance always results in assembly.

- **Interference Fit** - A fit between mating parts having limits of size so prescribed that an interference always results in assembly.

- **Transition Fit** - A fit between mating parts having limits of size so prescribed as to partially or wholly overlap, so that either a clearance or an interference may result in assembly.
Fits and Allowances

**Allowance**

- is difference between maximum material limits of mating parts (minimum clearance is positive allowance, Maximum interference is negative allowance).

- **Basic Size** - The size to which limits or deviations are assigned. The basic size is the same for both members of a fit.

- **Deviation** - The algebraic difference between a size and the corresponding basic size.
Fits and Allowances

Allowance

- **Upper Deviation** The difference between the maximum limit of size and the corresponding basic size.

- **Lower Deviation** The difference between the minimum limit of size and the corresponding basic size.

- **Tolerance** The difference between the maximum and minimum size limits.

- **Tolerance Zone** A zone representing the tolerance and its position in relation to the basic size.

- **Fundamental Deviation** The deviation closest to the basic size.

Fig. 8-6-1 Illustration of definitions.
Fits and Allowances

Description of Fits

- **Running and Sliding Fits** - A special type of clearance fit. These are intended to provide a similar running performance, with suitable lubrication allowance, throughout the range of sizes.

- **Locational Fits** – Intended to provide the location of mating parts. They may provide rigid or accurate location, as with interference fits, or some freedom of location, as with clearance fits. Accordingly, they are divided into three groups: clearance, transition, and interference fits.

- **Locational clearance fits** - are intended for parts that are normally stationary but that can be freely assembled or disassembled.
  - Snug fit for parts that require to be located with high accuracy
  - Medium clearance for parts like ball, race, and housing
  - Loose fit for ease of freedom of assembly (example for fasteners)
Fits and Allowances

Description of Fits

- **Locational transition fits** - are a compromise between clearance and interference fits when accuracy of location is important but a small amount of either clearance or interference is permissible.

- **Locational interference fits** - are used when accuracy of location is of prime importance and for parts requiring rigidity and alignment with ‘no special requirements for bore pressure’ – not used for transmitting frictional loads from one part to another.

- **Drive or Force Fits** – a special type of interference fit, normally characterized by maintenance of constant bore pressures throughout the range of sizes. The difference between maximum and minimum values is small to maintain resulting pressure.
Fits and Allowances

Interchangeability of Parts

- Basis for mass production and low cost manufacturing
- Today it is possible to produce parts with 100% interchangeability
- No part can be produced to ‘exact’ dimensions
- Machine variations, tool wear and human errors contribute to deviation
- Necessary to determine permissible clearance or interference to facilitate fit between parts
  - Completely interchangeable assembly – all mating parts are tolerated to permit assembly and proper function without need for machining or fitting at assembly
  - Fitted assembly – all mating parts are fabricated simultaneously or with respect to one another. Individual members of mating parts are not interchangeable
  - Selected assembly – all parts are mass produced but members of mating features are individually selected to provide required relationship with one another
Clearance fit

Maximum

Minimum

CLEARANCE = ALLOWANCE = 0.0003

0.005 HOLE TOLERANCE

MAX DIAMETER OF HOLE = 0.7505

MIN OR DESIGN SIZE OF HOLE = BASIC SIZE = 0.7500

EXAMPLE - 0.7500 RC2 FIT (BASIC HOLE SYSTEM)

(A) CLEARANCE FIT
Transition fit

MIN OR DESIGN SIZE OF SHAFT = \(0.7504\)

0.008 SHAFT TOLERANCE

0.0016 MAX CLEARANCE

MIN DIAMETER OF SHAFT = \(0.7496\)

MAX INTERFERENCE = 0.004 - allowance

0.0012 HOLE TOLERANCE

MIN OR DESIGN SIZE OF HOLE = \(0.7500\)

MAX DIAMETER OF HOLE = \(0.7512\)

EXAMPLE - 0.7500 LT2 FIT (BASIC HOLE SYSTEM)

MAX

(B) TRANSITION FIT
Interference Fit

(B) TRANSITION FIT

MAX

MIN OR DESIGN SIZE OF SHAFT = Ø 0.7519
SHAFT TOLERANCE = .0005
MIN DIAMETER OF SHAFT = Ø 0.7514

MAX

MAX INTERFERENCE = ALLOWANCE = .0019
MIN INTERFERENCE = .0006

.0008 HOLE TOLERANCE
MIN OR DESIGN SIZE
MAX DIAMETER OF HOLE = Ø 0.7508
OF HOLE = BASIC SIZE = Ø 0.7500

EXAMPLE - Ø 0.7500 FN2 FIT (BASIC HOLE SYSTEM)

(C) INTERFERENCE FIT

Types of inch fits.
Fits and Allowances

Standard inch Fits

- Standard fits are designated to specify on design sketches with symbols.
- They are not shown on shop drawings but actual limits of size are determined and shown.

<table>
<thead>
<tr>
<th>RC</th>
<th>LC</th>
<th>LT</th>
<th>LN</th>
<th>FN</th>
</tr>
</thead>
<tbody>
<tr>
<td>RC4</td>
<td>LC6</td>
<td>RC3S</td>
<td>LN3</td>
<td>FN2</td>
</tr>
</tbody>
</table>

- The two letters and the number express complete fit, the limits of size are given in appendix.

Fig. 8-6-2 Typical design sketches showing classes of fits.
Fits and Allowances

Running and sliding Fits

- **RC1 Precision Sliding Fit** - This fit is intended for the accurate location of parts that must assemble without perceptible play, for example, for high-precision work such as gages.

- **RC2 Sliding Fit** - This fit is intended for accurate location, but with greater maximum clearance than class RC 1. Parts made to this fit move and turn easily but are not intended to run freely, and in the larger sizes may seize with small temperature changes. Note: LCI and LC2 locational clearance fits may also be used as sliding fits with greater tolerances.

- **RC3 Precision Running Fit** - This fit is about the closest fit that can be expected to run freely and is intended for precision work for oil-lubricated bearings at slow speeds and light journal pressures, but is not suitable where appreciable temperature differences are likely to be encountered.
Fits and Allowances

Running and sliding Fits

- **RC4 Close Running Fit** - This fit is intended chiefly as a running fit for grease- or oil-lubricated bearings on accurate machinery with moderate surface speeds and journal pressures, where accurate location and minimum play are desired.

- **RC5 and RC6 Medium Running Fits** - These fits are intended for higher running speeds and/or where temperature variations are likely to be encountered.

- **RC7 Free Running Fit** - This fit is intended for use where accuracy is not essential and/or where large temperature variations are likely to be encountered.

- **RC8 and RC9 Loose Running Fits** - These fits are intended for use where materials made to commercial tolerances, such as cold-rolled shafting, or tubing, are involved.
Fits and Allowances

Locational clearance Fits

- Locational clearance fits are intended for parts that are normally stationary but that can be freely assembled or disassembled. Snug fit medium clearance fit – spigot for looser fasteners.

- LC1 to LC4 These fits have a minimum zero clearance

- LC5 and LC6 These fits have a small minimum clearance. Bolts

- LC7 and LC11 These fits have progressively larger clearances and tolerances and are useful for various loose clearances for the assembly of bolts and similar parts.
Fits and Allowances

Locational Transition Fits

- Either a small amount of clearance or interference is permissible. Accuracy of location – important.

- **LT.1 and LT2**: These fits average have a slight clearance, giving a light push fit, and are intended for use where the maximum clearance must be less than for the LCI to LC3 fits. Assembly – light pressure.

- **LT3 and LT4**: These fits average virtually no clearance and are for use where some interference can be tolerated, for example, to eliminate vibration. Inner bearing race, shaft key.

- **LT5 and LT6**: These fits average a slight interference, although appreciable assembly force will be required when extreme limits are encountered, and selective assembly may be desirable. Heavy key. Heavy duty, vibration.
Fits and Allowances

Locational interference Fits – Accuracy of location, but no transmitting of torque.

- **LN1 and LN2** - These are light press fits, with very small minimum interference, suitable for parts such as dowel pins.

- **LN3** - This is suitable as a heavy press fit in steel and brass, or a light press fit in more elastic materials and light alloys.

- **LN4 to LN6** - Although LN4 can be used for permanent assembly of steel parts, primarily suited for elastic or plastic parts.
Fits and Allowances

Force or shrink Fits – Maintenance of constant pressure, variable in size – small.

- **FN1 Light Drive Fit** - Requires light assembly pressure and produces more or less permanent assemblies. It is suitable for thin sections or long fits, or in cast iron external members.

- **FN2 Medium Drive Fit** - Suitable for ordinary steel parts or as a shrink fit on light sections. It is about the tightest fit that can be used with high-grade cast iron external members.

- **FN3 Heavy Drive Fit** - Suitable for heavier steel parts or as a shrink fit in medium sections.

- **FN4 and FNS Force Fits** - Suitable for parts that can be highly stressed and/or for shrink fits where the heavy pressing forces required are impractical.
Fits and Allowances

Basic Hole System

- In the basic hole system, which is recommended for general use, the basic size will be the design size for the hole, and the tolerance will be plus. The design size for the shaft will be the basic size minus the minimum clearance, or plus the maximum interference.

- eg. (table 43 appendix A 26) An 1” RC7 fit, values of +.0020 (hole tolerance), .0025 (min clearance), and -.0012 (shaft tolerance) the limits will be:

<table>
<thead>
<tr>
<th>Hole Ø</th>
<th>Shaft Ø</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0000</td>
<td>.9975</td>
</tr>
<tr>
<td>+.0020</td>
<td>+.0000</td>
</tr>
<tr>
<td>-.0000</td>
<td>-.0012</td>
</tr>
</tbody>
</table>
Fits and Allowances

Basic Shaft System

- Fits are sometimes required on a basic shaft system, especially when two or more fits are required on the same shaft. It is indicated (for design purposes) with an S following the fit symbol RC7S. Table not available for inch fit.

- eg. An 1” RC7 fit, values of +.0020 (hole tolerance), .0025 (min clearance), and -.0012 (shaft tolerance) the limits will be

  +.0020 +.0000
  Hole Ø 1.0025 Shaft Ø 1.0000
  -.0000 -.0012
Fits and Allowances
Fits and Allowances

Preferred Metric Limits and Fits

- General terms hole and shaft can also be taken as space contained by two parallel faces of parts (width of slot or thickness of key)
- International tolerance grades (IT) establishes the magnitude of tolerance zone (amount of part variation Appendix table 40)
- There are 18 tolerance grades identified by IT as prefix
- Smaller the grade No, smaller the tolerance zone (more precise)
- 1-4 are very precise used for gage making and high precision work
- 5-16 represent progressive series suitable for cutting
- Grade 5 is most precise obtained by grinding and lapping and grade 16 is most coarse obtained by sawing
Fits and Allowances

Preferred Metric Limits and Fits

- A fundamental deviation establishes the position of the tolerance zone wrt to basic size. FD is expressed by tolerance position letters (upper case for internal and lower for external)

![Diagram of Fits and Allowances](image)

**Tolerance Symbol**

- In metric system, the tolerance maybe indicated by a basic size and tolerance symbol
- Combination of IT grade and the position letter the symbol is established to define maximum and minimum clearance of the part
- Tolerance sizes are defined by basic size, followed by symbol composed of letter and number
Fits and Allowances

Hole Basis System

- The basic size will be the design size for the hole eg. (table 48 appendix) A Φ25H8/f7 fit, hole basis clearance fit

<table>
<thead>
<tr>
<th>Hole limits</th>
<th>Ø25.000 - Ø25.033</th>
<th>Minimum clearance</th>
<th>0.020</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shaft limits</td>
<td>Ø24.959 - Ø24.980</td>
<td>Maximum clearance</td>
<td>0.074</td>
</tr>
</tbody>
</table>

- A Φ25H7/s6 fit, hole basis interference fit

<table>
<thead>
<tr>
<th>Hole limits</th>
<th>Ø25.000 - Ø25.021</th>
<th>Minimum Interference</th>
<th>-0.014</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shaft limits</td>
<td>Ø25.035 - Ø25.048</td>
<td>Maximum Interference</td>
<td>-0.048</td>
</tr>
</tbody>
</table>
Fits and Allowances

Shaft Basis System

- Used when more than two fits are required on the same shaft. The basic size will be the maximum shaft size.
- for eg. (table 49 appendix) A Φ16 C11/h11 fit, shaft basis clearance fit

<table>
<thead>
<tr>
<th>Hole limits</th>
<th>Ø16.095 – Ø16.205</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shaft limits</td>
<td>Ø15.890 – Ø16.000</td>
</tr>
<tr>
<td>Minimum Clearance</td>
<td>-0.095</td>
</tr>
<tr>
<td>Maximum Clearance</td>
<td>-0.315</td>
</tr>
</tbody>
</table>
Fits and Allowances

Type of Metric Fits

- Hole basis have FD of ‘H’ on the hole, H11/C11
- Shaft basis have FD of ‘h’ on the shaft, C11/h11
- Hole basis is preferred
- Three common fits in Metric (shown in figure)
Fits and Allowances

Fit Symbol

- Fit is indicated by the basic size common to both components, followed by a symbol corresponding to each component, with internal preceding the external part symbol.
- The limits of size for a hole having tolerance symbol 40H8 (table 41 and then table 48).
Fits and Allowances

Fit Symbol

- limits of size for a shaft having tolerance symbol 40f7 (table 48)
  - Ø39.975 Maximum limit
  - Ø39.950 Minimum limit

- (A) Method is first introduced – limit dimensions are specified and basic size & tolerance symbol identified as reference

- (B) When some experience gained – basic size & tolerance symbol are specified and limit dimensions identified as reference

- (C) When system established – only basic size & tolerance symbol are specified
## Fits and Allowances

### Preferred Metric Fits

<table>
<thead>
<tr>
<th>ISO SYMBOL</th>
<th>HOLE BASIS</th>
<th>SHAFT BASIS</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>H11/11</td>
<td>C11/h11</td>
<td></td>
<td>LOOSE RUNNING FIT FOR WIDE COMMERCIAL TOLERANCES OR ALLOWANCES ON EXTERNAL MEMBERS.</td>
</tr>
<tr>
<td>H9/d9</td>
<td>D9/h9</td>
<td></td>
<td>FREE RUNNING FIT NOT FOR USE WHERE ACCURACY IS ESSENTIAL, BUT GOOD FOR LARGE TEMPERATURE VARIATIONS, HIGH RUNNING SLOWS, OR HEAVY JOURNAL PRESSURES.</td>
</tr>
<tr>
<td>H8/f7</td>
<td>F8/h7</td>
<td></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>G7/h6</td>
<td></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>H7/h6</td>
<td></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>K7/h6</td>
<td></td>
<td>LOCATIONAL TRANSITION FIT FOR ACCURATE LOCATION, A COMPROMISE BETWEEN CLEARANCE AND INTERFERENCE.</td>
</tr>
<tr>
<td>H7/n6</td>
<td>N7/h6</td>
<td></td>
<td>LOCATIONAL TRANSITION FIT FOR MORE ACCURATE LOCATION WHERE GREATER INTERFERENCE IS PERMISSIBLE.</td>
</tr>
<tr>
<td>H7/p6</td>
<td>P7/h6</td>
<td></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>S7/h6</td>
<td></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/u0</td>
<td>U7/h6</td>
<td></td>
<td>FORCE FIT SUITABLE FOR PARTS WHICH CAN BE HIGHLY STRESSED OR FOR SHRINK FITS WHERE THE HEAVY PRESSING FORCES REQUIRED ARE IMPractical.</td>
</tr>
</tbody>
</table>

*Description of preferred metric fits.*