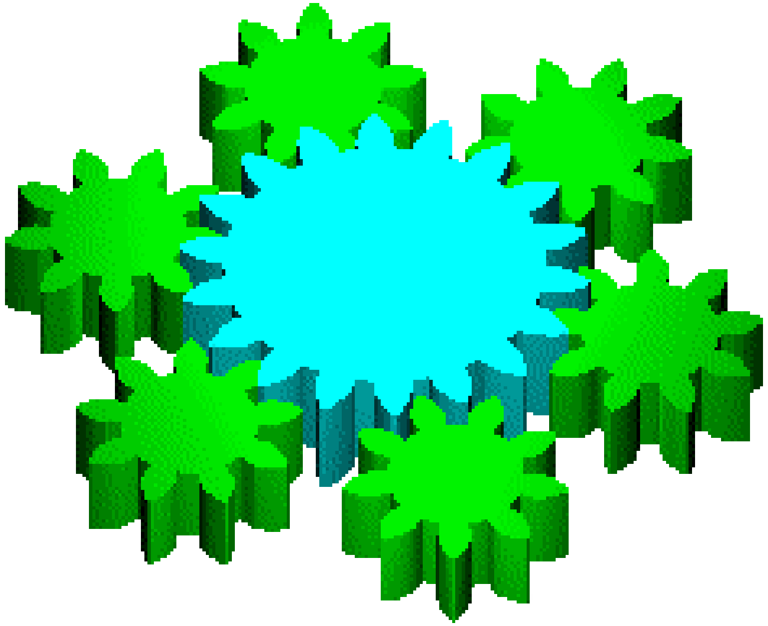


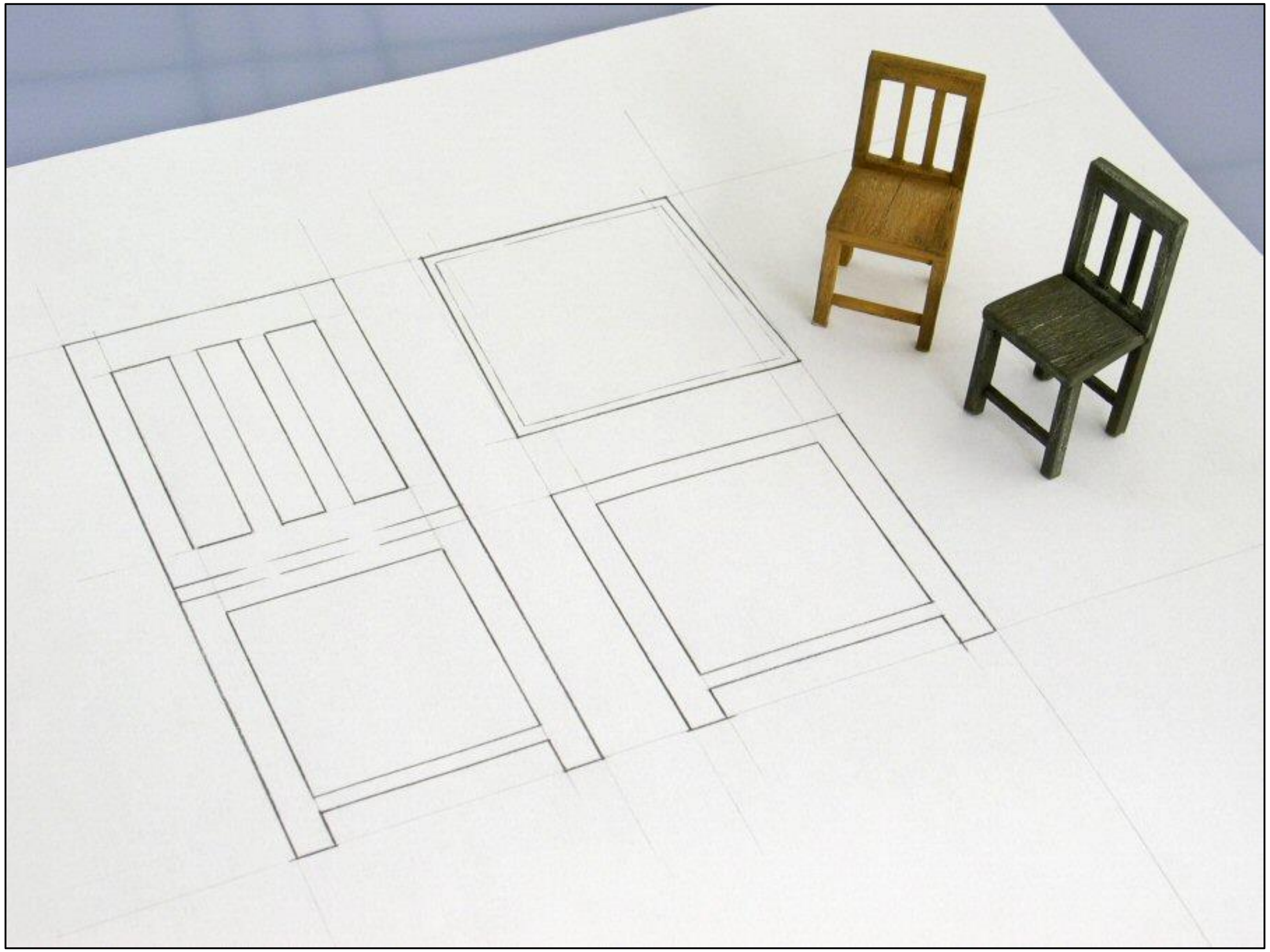
Tolerances in Dimensioning

From Drawings to Products

- Engineering drawings serve as documents for manufacturing, purchasing, construction, inspection, and legal contracts.

- They must be clear, with enough information so that the product is fully defined and can be manufactured in a way as to fulfill the intended functionality and design.

- The information on the drawings is given in the form of graphics, general notes, local notes, dimensions and tolerances.



Tolerances

Each dimension on a drawing is permitted to vary within a specified amount.

The **Tolerance** is the amount of size variation permitted.

Tolerances enable parts to be manufactured by different companies and maintain their intended functionality and design.

Tolerancing

Tolerancing is the process of specifying the degree of precision

- large tolerances reduce production costs
- small tolerances = higher precision = higher quality = higher cost

Considerations:

- sustainability
- function
- manufacturing operations
- material
- quantity (run size)
- cost

Engineers must balance considerations.
A definition of Engineering:
"The Art of Compromise"

The **precision** of a dimension on an engineering drawing is the number of digits after the decimal point.

The **accuracy** is the number of sig figs.

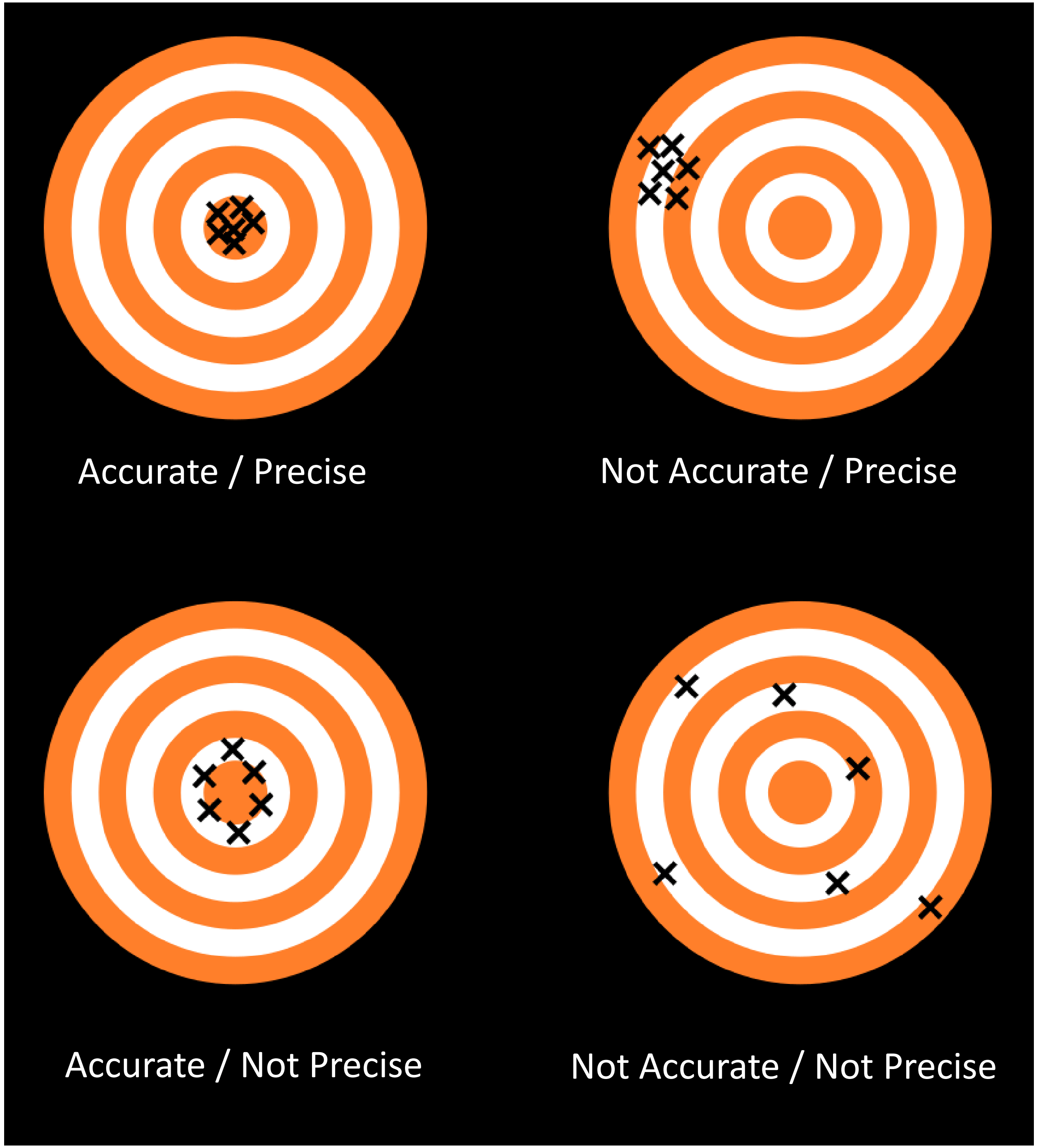
Precision vs. Accuracy

Good Videos

(if you can ignore the lack gender and racial diversity)

https://www.youtube.com/watch?v=KEeSQvMCPLg

https://www.youtube.com/watch?v=EeHtK5UYEMM



Tolerance Standards are set by ANSI/ASME or ISO

General tolerances are in the title block.
They are applied to all dimensions unless otherwise noted.

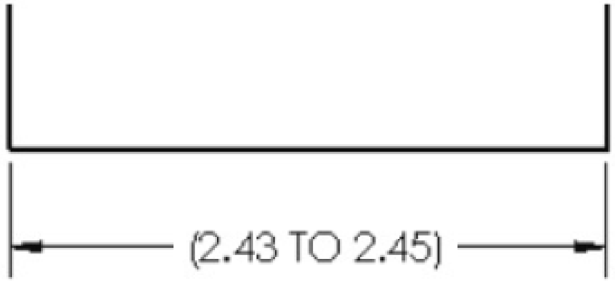
UNLESS OTHERWISE SPECIFIED:	
DIMENSIONS ARE IN INCHES	
TOLERANCES:	
ANGULAR: ± 1°	
ONE PLACE DECIMAL	± .1
TWO PLACE DECIMAL	± .01
THREE PLACE DECIMAL	± .005

UNLESS OTHERWISE SPECIFIED:	
DIMENSIONS ARE IN MILLIMETERS	
TOLERANCES:	
ANGULAR: MACH ± 0°30'	
ONE PLACE DECIMAL	±0.5
TWO PLACE DECIMAL	±0.15

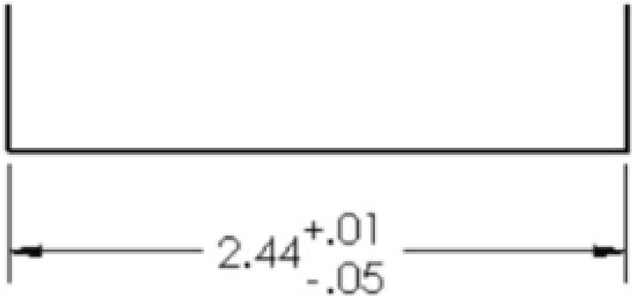
Local Tolerances indicate a special situation not covered in Title Box.
They are shown with the dimension.

3 Common Tolerance Types

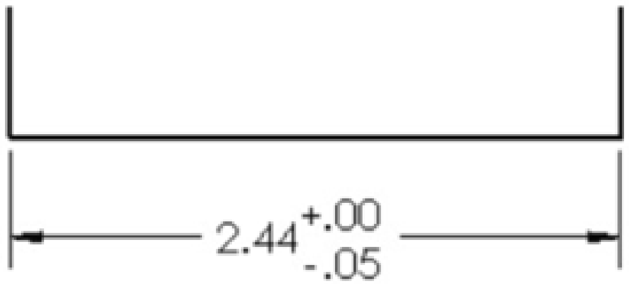
Limit Tolerance
the max and min size allowed

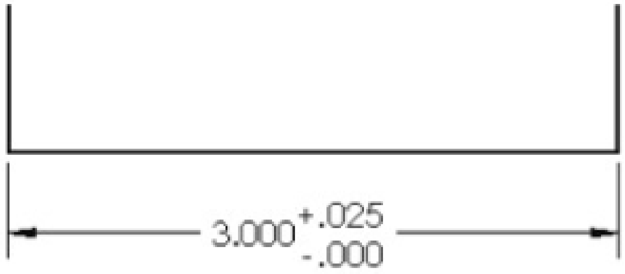


Bilateral Tolerance
the variation of size is in both directions



Unilateral Tolerance
the variation of size is only in a single direction (can be + or -)





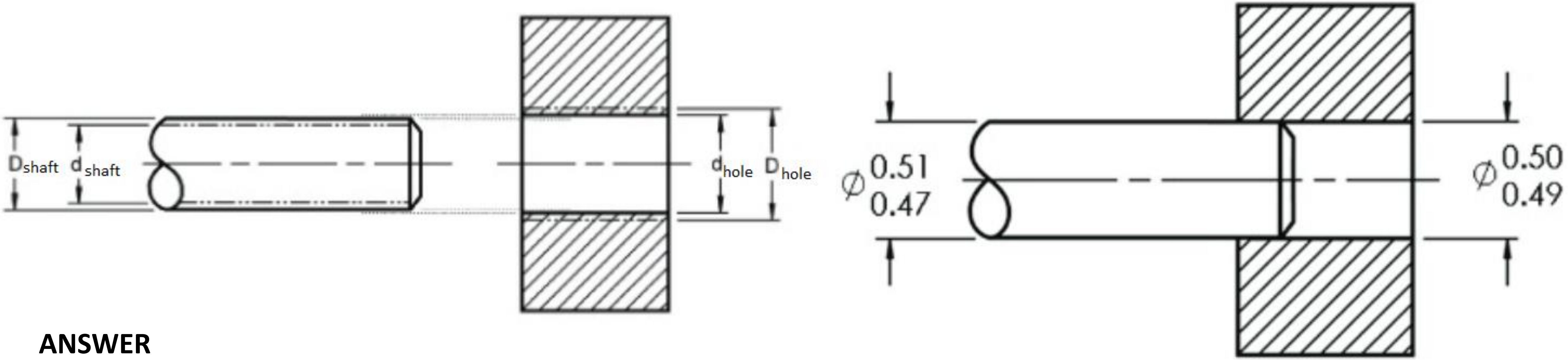
Fit Hole Tolerance

What is the minimum clearance?

(i.e. the minimum amount of space between the hole and the shaft?)

What is the maximum clearance?

(i.e. the maximum amount of space between the hole and the shaft?)



ANSWER

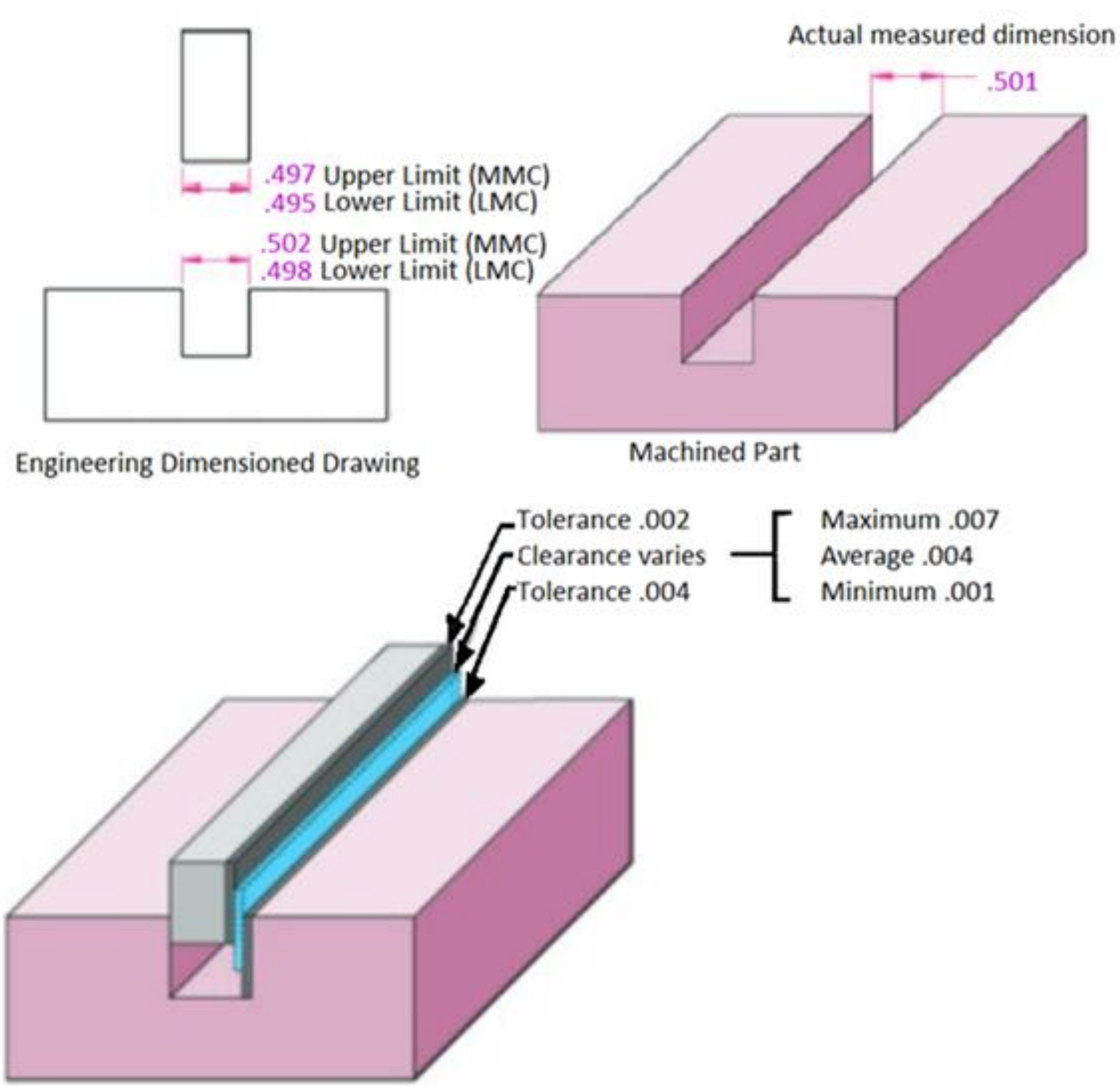
Minimum Clearance is the difference between the smallest hole diameter and the largest shaft diameter

$= d_{hole} - D_{shaft} = 0.49 - 0.51 = -0.02$in.

Maximum Clearance is the difference between the largest hole diameter and the smallest shaft diameter.

$= D_{hole} - d_{shaft} = 0.50 - 0.47 = 0.03$in.

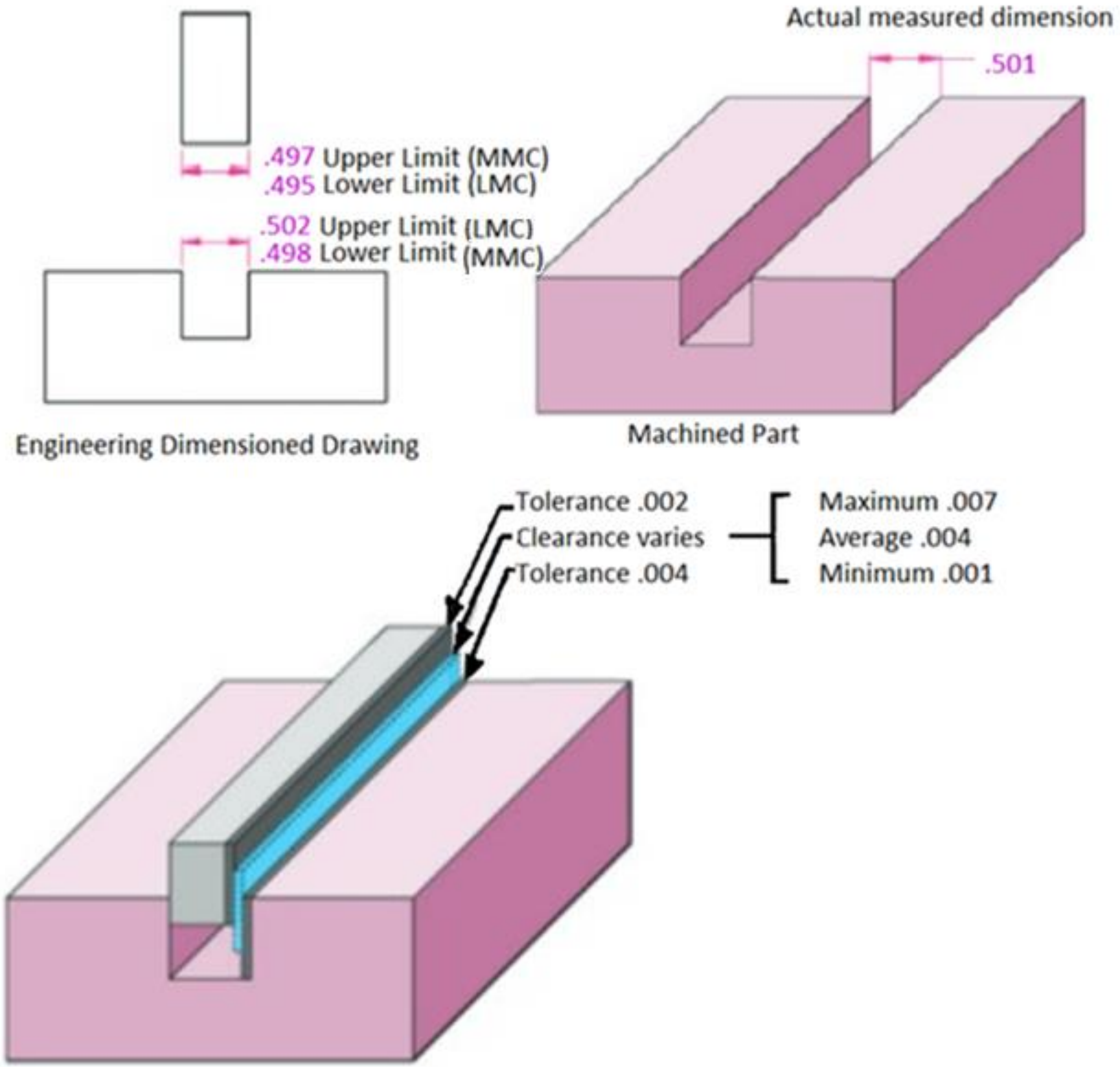
ASME Y14.5 2009 important terms



- **Tolerance** - the total allowable variance in a dimension; the difference between the upper and lower limits. The tolerance of the slot is .004″. (.502″ - .498″ = .004″) and the tolerance of the mating parts is .002″ (.497″ - .495″ = .002″).

- **Allowance** - the minimum clearance or maximum interference between parts, or the tightest fit between two mating parts. In the illustration, the allowance is .001″, meaning that the tightest fit occurs when the slot is machined to its smallest allowable size of .498″ and the mating part is machined to its largest allowable size of .497″. The difference between .498″ and .497″ or 001″ is the allowance.

ASME Y14.5 2009 important terms



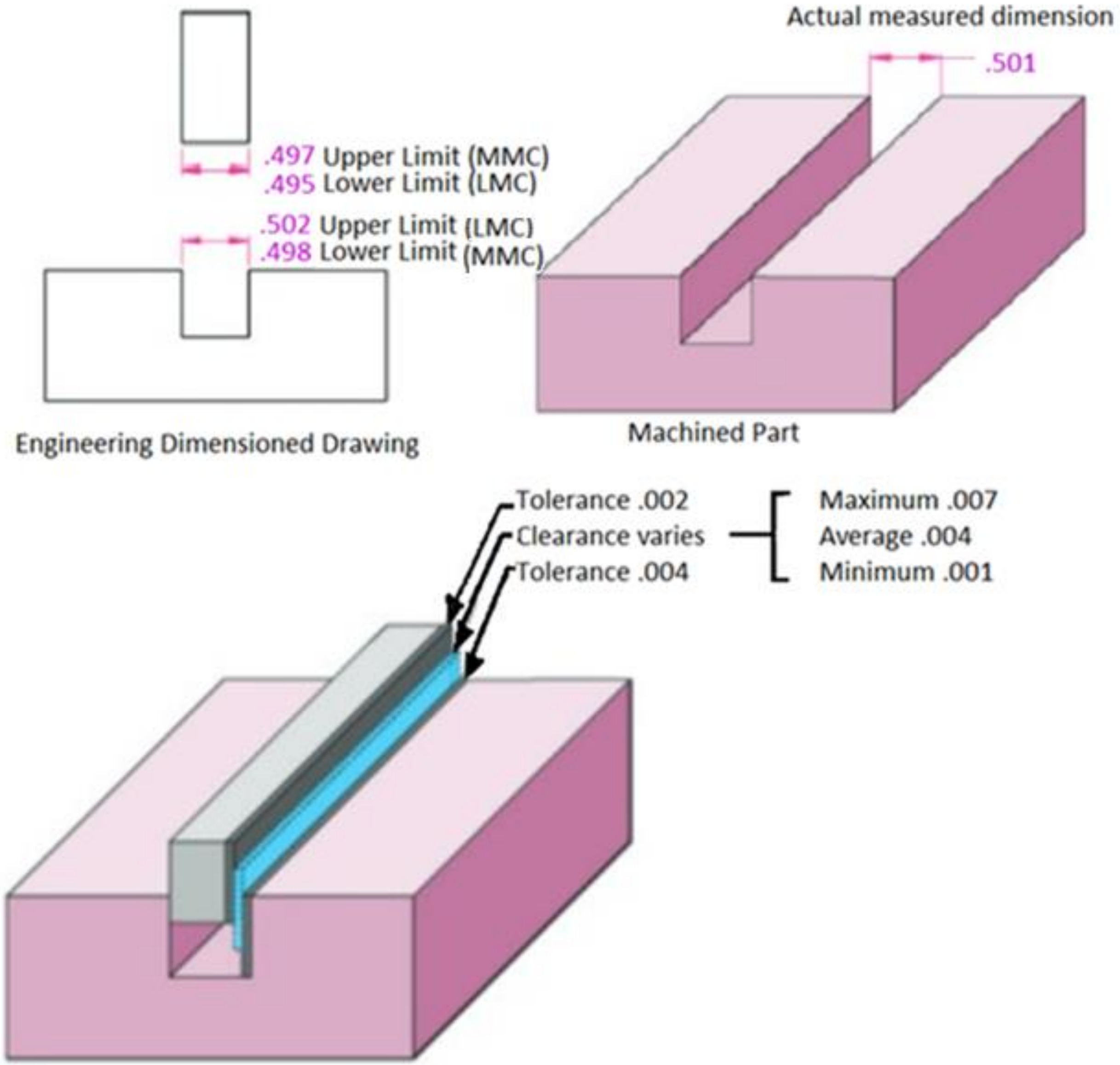
Nominal size - a dimension used to describe the general size, usually expressed in common fractions. The slot in the above illustration has a nominal size of ½″.

Basic size - the theoretical size used as a starting point for the application of tolerances. The basic size of the slot is .500″.

Actual size - the measured size of the finished part after machining is .501″.

Limits - the maximum and minimum sizes shown by the tolerance dimension. The slot has limits of .502″ and .498″, and the mating part has limits of .495″ and .497″. The larger value for each part is the upper limit, and the smaller value is the lower limit.

ASME Y14.5 2009 important terms



- **Maximum material condition (MMC)** - the condition of a part when it contains the greatest 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.
- **Least material condition (LMC)** - the condition of a part when it contains the least amount of material possible. The LMC of an external feature is the lower limit. The LMC of an internal feature is the upper limit.
- **Piece tolerance** - the difference between the upper and lower limits of a single part.
- **System tolerance** - the sum of all the piece tolerances.

Fit Types Between Mating Parts

Fit: degree of tightness between two parts.

Fit types:

Clearance Fit

tolerance of mating parts always leaves a space

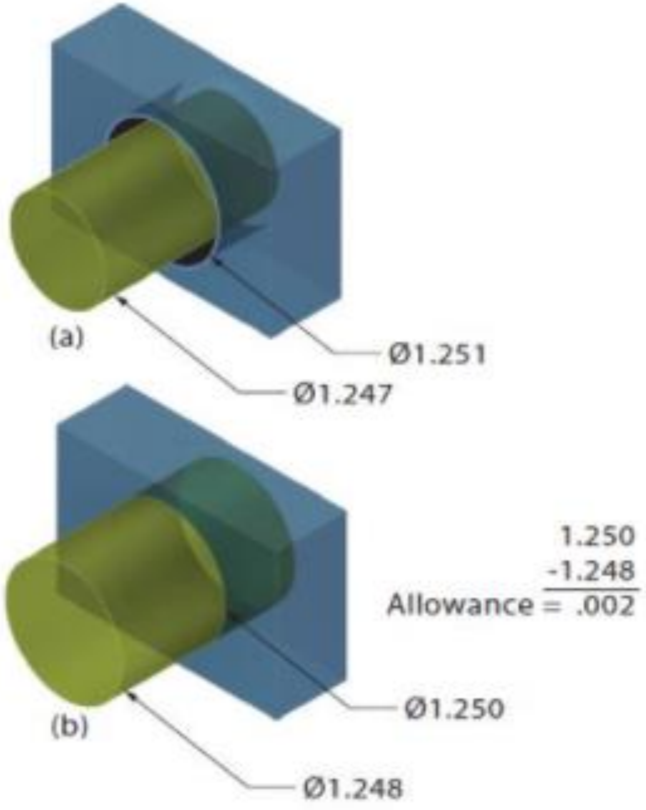
Interference Fit

tolerance of mating parts always results in interference

Transition Fit

sometimes interferes, sometimes clears

Fit is the range of tightness or looseness resulting from the allowances and tolerances in mating parts. The loosest fit, or maximum clearance, occurs when the smallest internal part (shaft) is in the largest external part (hole).



(a) Loosest and (b) Tightest Fit

Fit Types Between Mating Parts

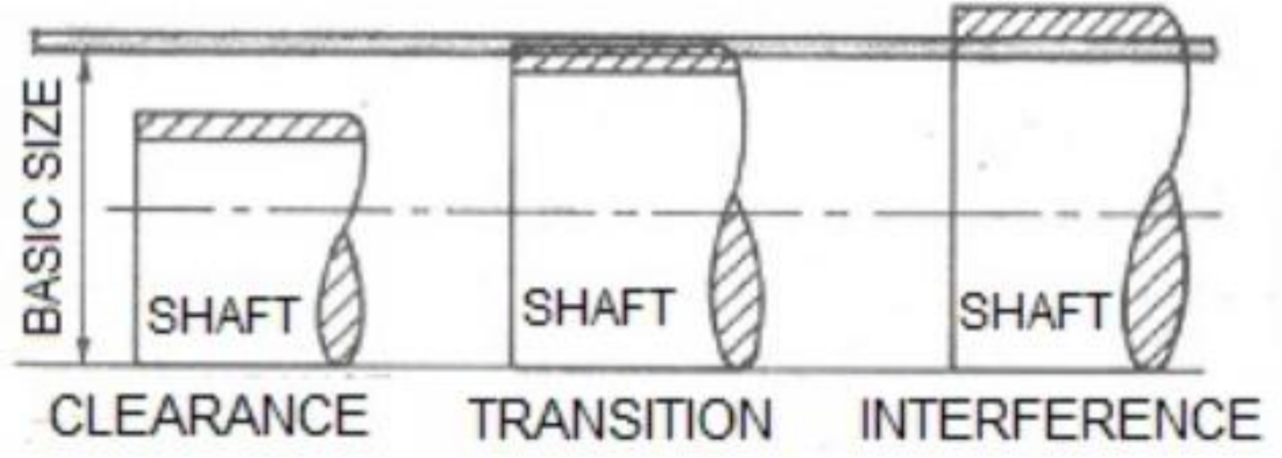
When two parts are to be assembled, the relationship resulting from the difference between their sizes before assembly is called a **Fit.**

Fits are broadly classified into three groups

1. ***Clearance fit***
   It always provides a positive clearance between the hole and shaft over the entire range of tolerances. In this case tolerance zone of the hole is entirely above that of the shaft.

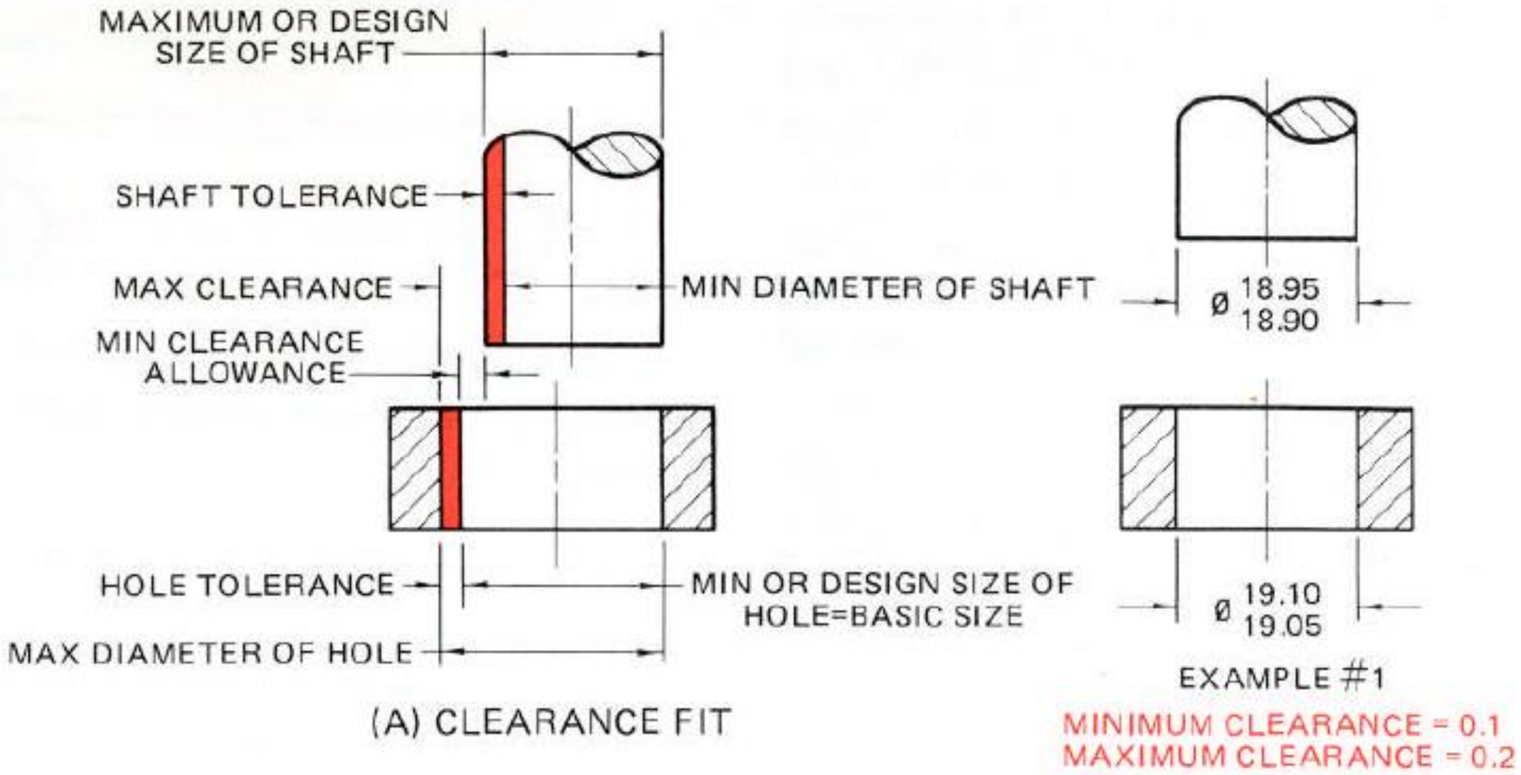
2. ***Transition fit***
   It may provide either a clearance or interference, depending upon the actual values of the individual tolerances of the mating components. In this case tolerance zone of hole and shaft overlaps.

3. ***Interference fit***
   It always provides a negative clearance (or positive interference) over the whole range of tolerances. In this case, the tolerance zone of the hole is completely below that of the shaft.



Clearance Fit

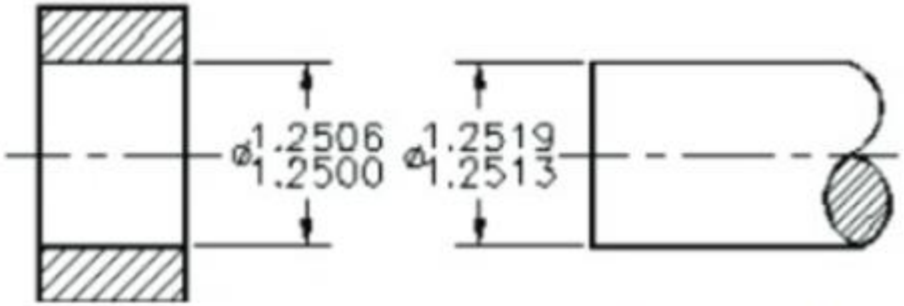
Clearance Fit: The difference between the hole and shaft sizes before assembly is positive. Clearance fits have limits of size prearranged such that a clearance always results when the mating parts are assembled. Clearance fits are intended for accurate assembly of parts and bearings. The parts can be assembled by hand because the hole is always larger than the shaft. Min. Clearance > 0. Two examples: Lock and Key, Door and Door frame.



(A) CLEARANCE FIT

Interference Fit

Interference Fit: (also referred to as Force fit or Shrink fit) - interference fit has limits of size that always result in interference between mating parts. The hole is always smaller than the shaft. Interference fits are for permanent assemblies of parts which require rigidity and alignment, such as dowel pins and bearings in casting, hinge pin or a pin in a bicycle chain. Max. Clearance ≤ 0.

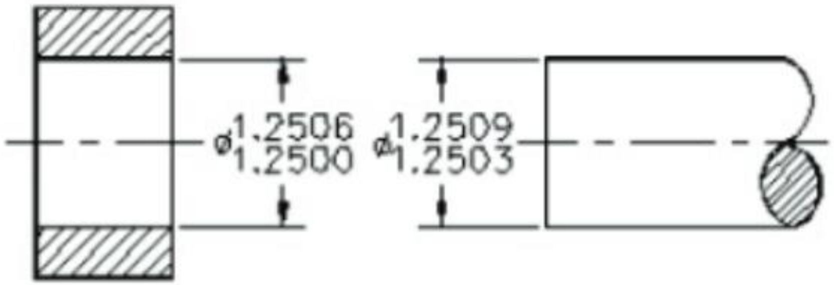


LMC Shaft	=	1.2513
- LMC Hole	=	1.2506
Min Interference	=	.0007
MMC Shaft	=	1.2519
- MMC Hole	=	1.2500
Max Interference	=	.0019

Transition Fit

Transition Fit: May provide either clearance or interference, depending on the actual value of the tolerance of individual parts. Transition fits are a compromise between the clearance and Interference fits. They are used for applications where accurate location is important, but either a small amount of clearance or interference is permissible. Max. Clearance > 0, Min. Clearance < 0.

Why is this information important? By specifying the correct allowances and tolerances, mating components in an assembly can be completely interchangeable.



LMC Hole	=	1.2506
- LMC Shaft	=	1.2503
Positive Clearance	=	.0003
MMC Shaft	=	1.2509
- MMC Hole	=	1.2500
Negative Allowance (Interference)	=	.0003