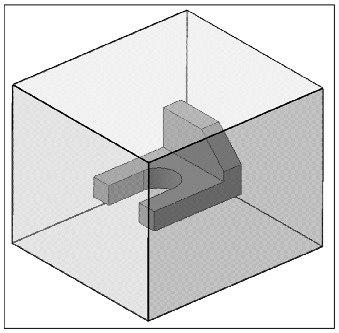
CH 16 & 17

Detailed Drawings

- Orthographic Projection: A shape description of an object (front, top, right side views).
- Detailed Drawing: An orthographic projection, complete with all the dimensions and specifications needed to manufacture the object.

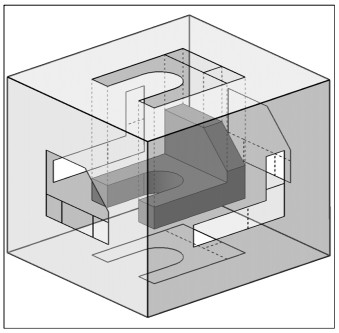
Glass Box Method

- The object is placed in a glass box.
- The side of the box represent the 6 principle planes.



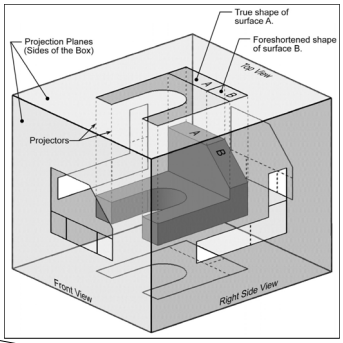
Glass Box Method

- The image of the object is projected on the sides of the box.



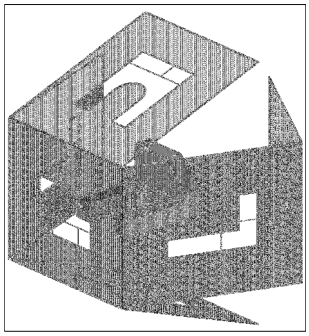
Glass Box Method

- Things to notice!
  - The projection planes.
  - The projectors.
  - How surfaces A and B are projected.

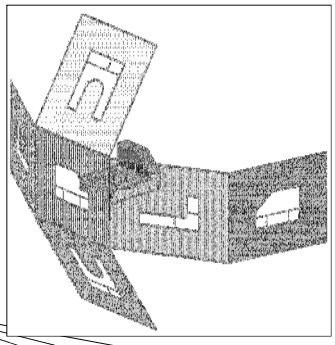


Glass Box Method

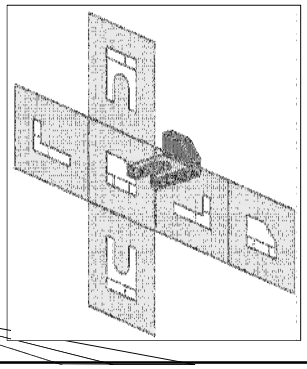
- The box is unfolded creating the 6 principle views.



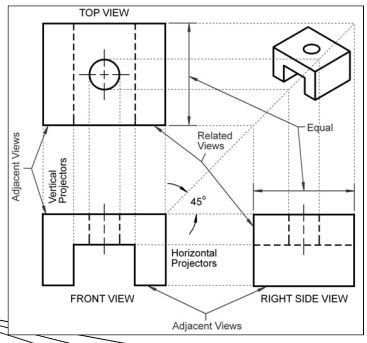
Glass Box Method

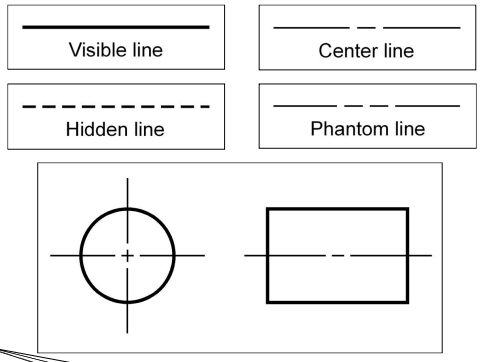


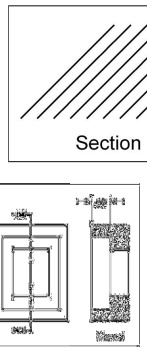
Glass Box Method



Creating an Orthographic Projection







This is how section lines might look on a drawing

Learning to Dimension

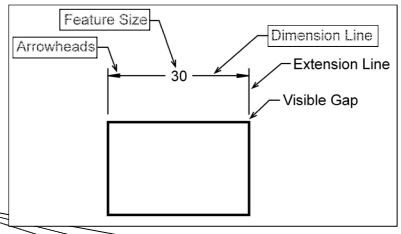
- What is our goal when dimensioning a part?
  - Basically, dimensions should be given in a clear and concise manner and should include everything needed to produce and inspect the part exactly as intended by the designer.

Lines used in Dimensioning

- Dimensioning requires the use of
  - Dimension lines
  - Extension lines
  - Leader lines
- All three line types are drawn thin so that they will not be confused with visible lines.

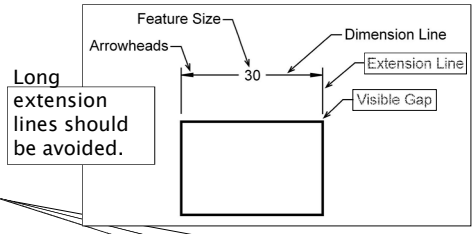
Dimension Line

- Dimension line: A line terminated by arrowheads, which indicates the direction and extent of a dimension.



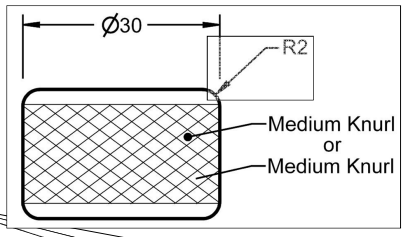
Extension Line

- Extension line: An extension line is a thin solid line that extends from a point on the drawing to which the dimension refers.



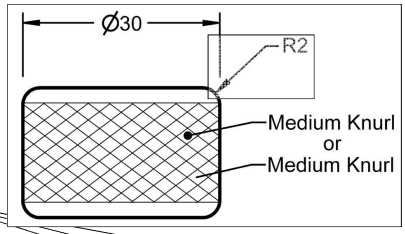
Leader Line

- Leader Line: A straight inclined thin solid line that is usually terminated by an arrowhead.



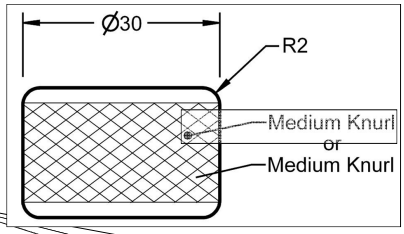
Leader Line

- Leaders may be terminated:
  - with an arrow, if it ends on the outline of an object.



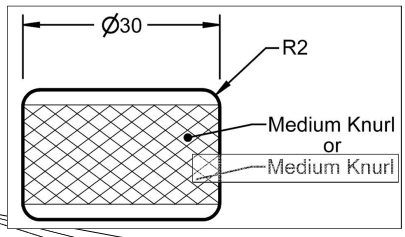
Leader Line

- Leaders may be terminated:
  - with a dot if it ends within the outline of an object.



Leader Line

- Leaders may be terminated:
  - without an arrowhead or dot, if it ends within the outline of an object.

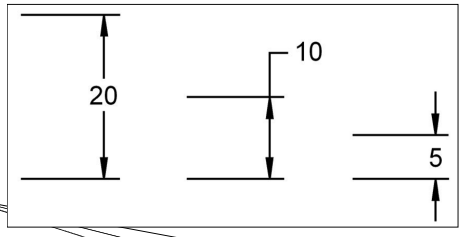


Leader Line

- Avoid!
  - Crossing leaders.
  - Long leaders.
  - Leaders that are parallel to adjacent dimension, extension or section lines.
  - Small angles between the leader and the terminating surface.

Arrowheads

- Arrowheads are drawn between the extension lines if possible. If space is limited, they may be drawn on the outside.

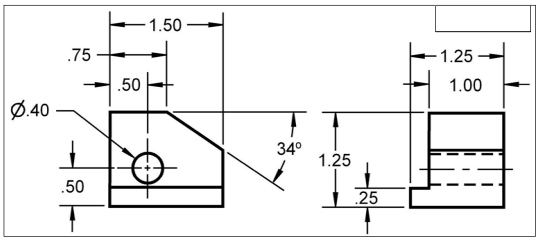


Types of Dimensions

- Dimensions are given in the form of *linear distances*, *angles*, and *notes*.
  - Linear distances: They are usually arranged horizontally or vertically, but may also be aligned with a particular feature of the part.
  - Angles: Used to give the angle between two surfaces or features of a part.

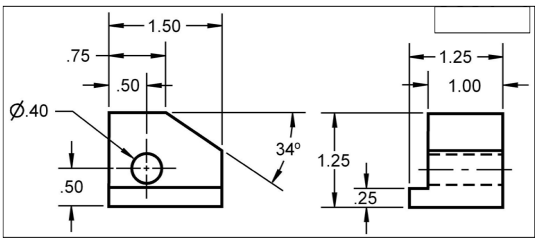
- Count the different types of dimensions.
  - How many linear horizontal dimensions are there?

5

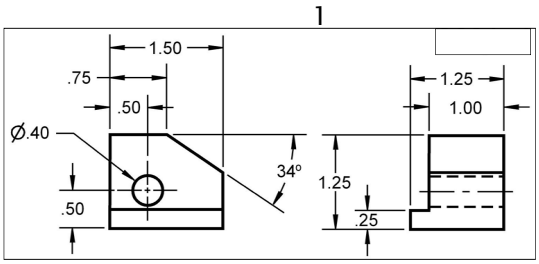


- Count the different types of dimensions.
  - How many linear vertical dimensions are there?

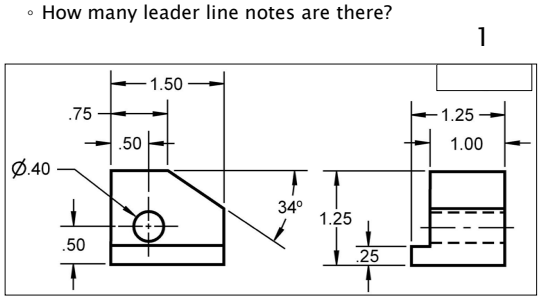
3



- Count the different types of dimensions.
  - How many angular dimensions are there?



- Count the different types of dimensions.
  - How many leader line notes are there?



Dimensioning Symbols

- Dimensioning symbols replace text.
  - The goal of using dimensioning symbols is to eliminate the need for language translation.
- Why is it important to use symbols.
  - How many products are designed in the United States?
  - How many products are manufactured or assembled in the United States?

Dimensioning Symbols

Term	Symbol
Diameter	∅
Spherical diameter	S∅
Radius	R
Spherical radius	SR
Reference dimension	(8)
Counterbore / Spotface	⌴
Countersink	⌵
Number of times or places	4X

Dimensioning Symbols

Term	Symbol
Depth / Deep	↧
Dimension not to scale	10
Square (Shape)	□
Arc length	⌒5
Conical Taper	⧐
Slope	⏥
Symmetry	⌯

Spacing and Readability

- Dimensions should be easy to read, and minimize the possibility for conflicting interpretations.
- The spacing between dimension lines should be uniform throughout the drawing.

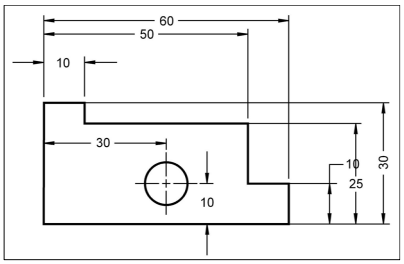
  Do not dimension inside an object or have the dimension line touch the object unless clearness is gained.

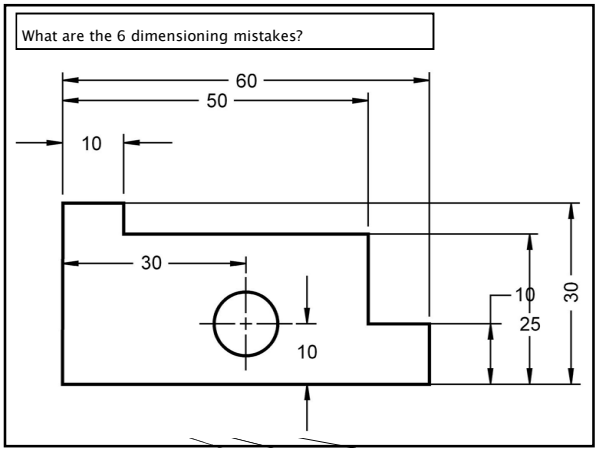
Spacing and Readability

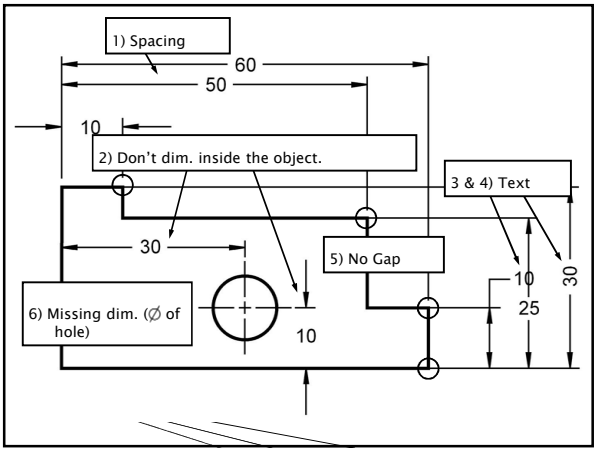
Dimension text should be horizontal which means that it is read from the bottom of the drawing.

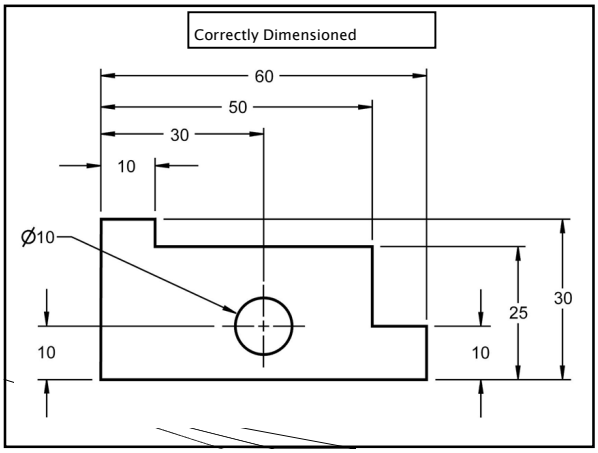
Dimension text should not cross dimension, extension or visible lines.

- List the dimensioning mistakes and then dimension the object correctly.









Spacing and Readability

Dimension lines should not cross extension lines or other dimension lines.

- Extension lines can cross other extension lines or visible lines.

Extension lines and centerlines should not connect between views.

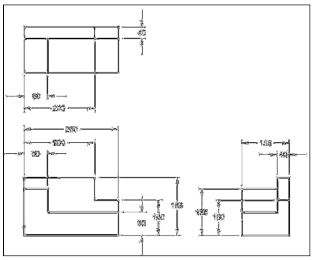
Spacing and Readability

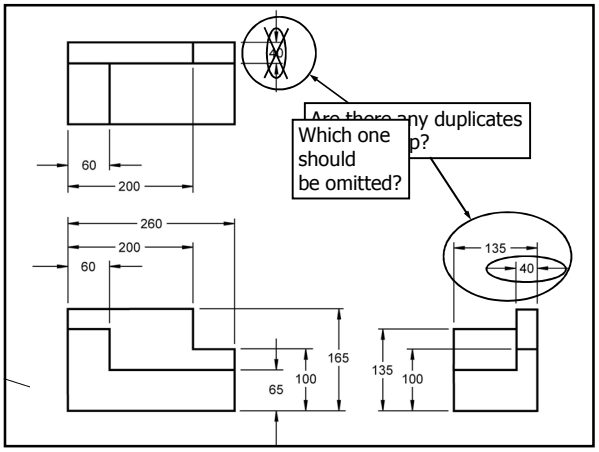
Leader lines should be straight, not curved, and point to the center of the arc or circle at an angle between 30º – 60º.

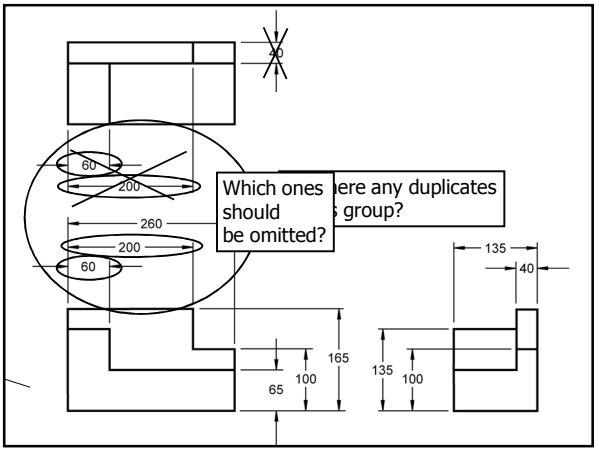
Dimensions should not be duplicated or the same information given in two different ways.

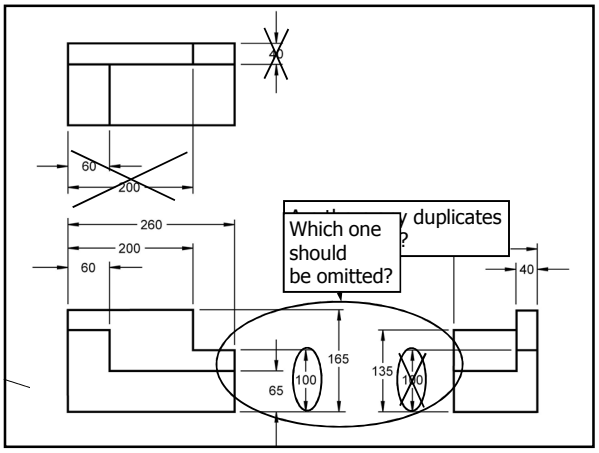
- If a reference dimension is used, the size value is placed within parentheses (e.g. (10)).

- Find the duplicate dimensions and cross out the ones that you feel should be omitted.

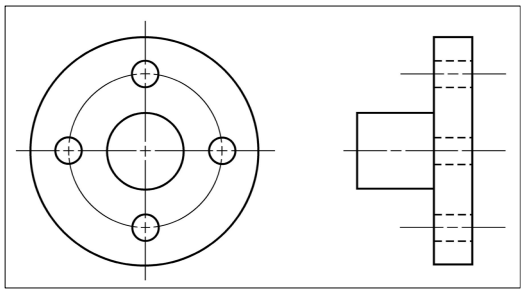




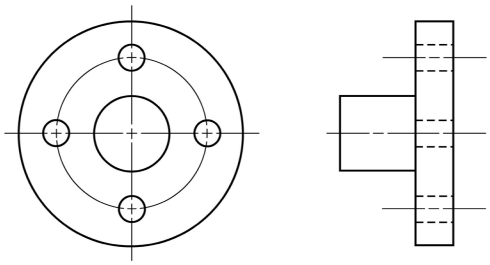




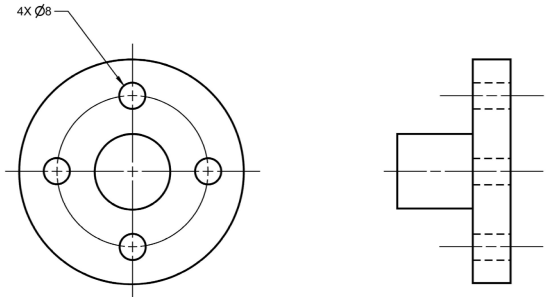
- Dimension the object shown.



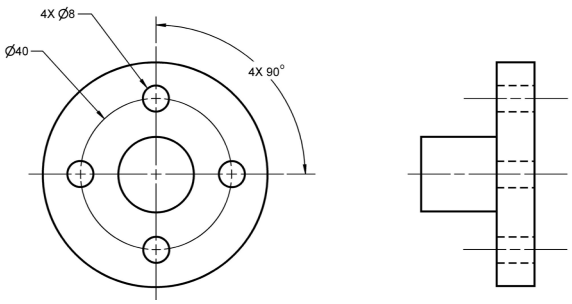
How do we dimension the hole diameters?



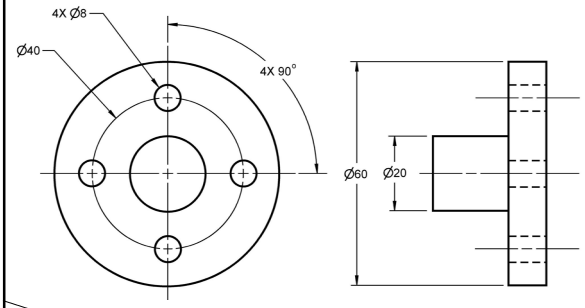
How do we locate the holes?



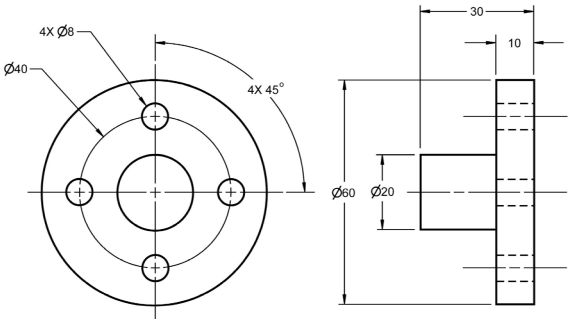
How do we dimension the cylinder diameters?



How do we dimension the cylinder heights?

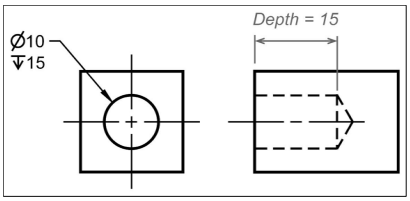


Completely dimensioned.



Dimensioning Features

The depth of a blind hole may be specified in a note and is the depth of the full diameter from the surface of the object.

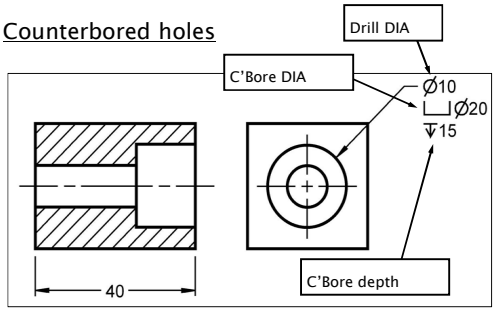


Dimensioning Features

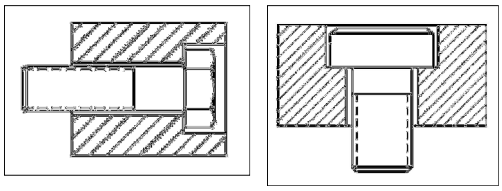
d) If a hole goes completely through the feature and it is not clearly shown on the drawing, the abbreviation "THRU" follows the dimension.

Dimensioning Features

f) Counterbored holes

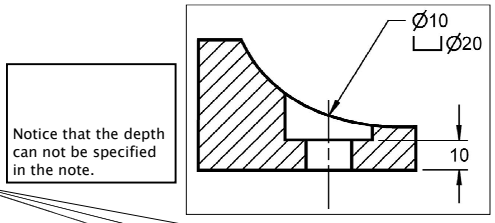


- What do you think a counterbored hole is used for?



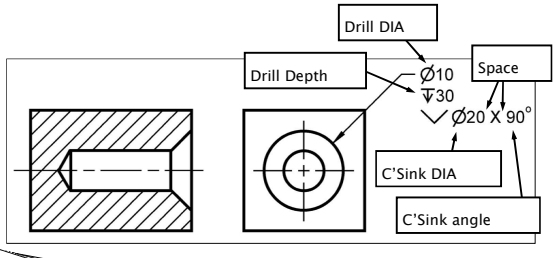
Dimensioning Features

g) Spotfaced Holes: The difference between a C'BORE and a Spotface is that the machining operation occurs on a curved surface.

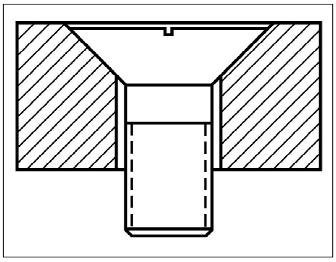


Dimensioning Features

h) Countersunk Holes

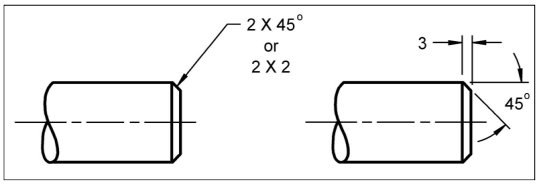


- What do you think a countersunk hole is used for?



Dimensioning Features

i) Chamfers: Dimensioned by a linear dimension and an angle, or by two linear dimensions.



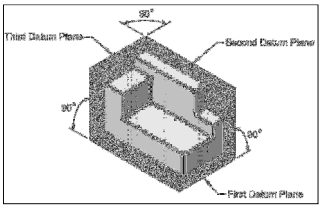
- What do you think a chamfer is used for?

Safety.

Improve engagement of mating parts.

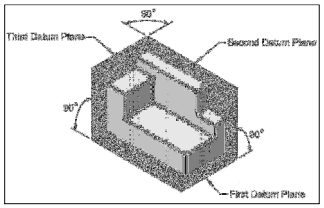
Locating Features Using Datums

- Consider three mutually perpendicular datum planes.
  - These planes are imaginary and theoretically exact.



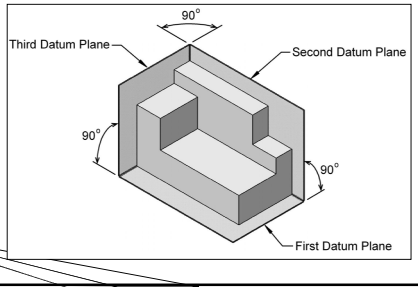
Locating Features Using Datums

- Now, consider a part that touches all three datum planes.
  - The surfaces of the part that touch the datum planes are called ***datum features***.

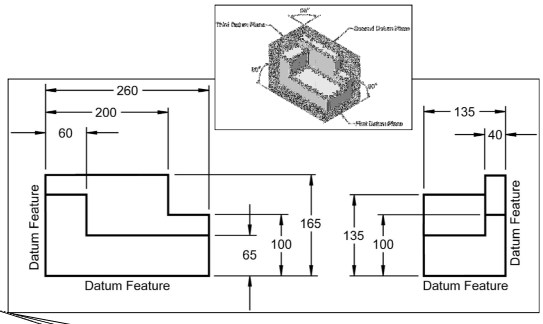


Locating Features Using Datums

- Most of the time, features on a part are located with respect to a datum feature.



Locating Features Using Datums

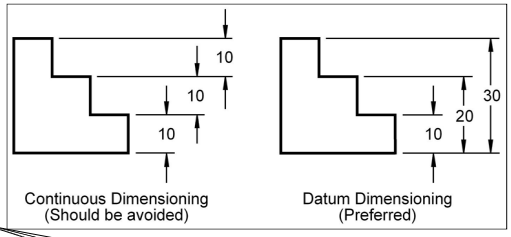


Locating Features Using Datums

- How do we choose which surface will be a datum feature?
- Good datum features are:
  - functionally important surfaces
  - mating surfaces
  - big enough to permit its use in manufacturing the part

Locating Features Using Datums

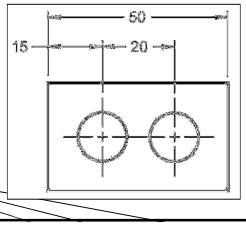
Datum dimensioning is preferred over continuous dimensioning.



Locating Features Using Datums

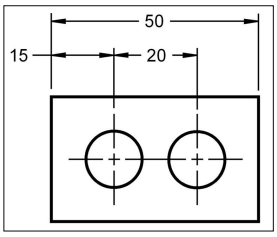
Dimensions should be given between points or surfaces that have a functional relation to each other

- Slots, mating hole patterns, etc...



- Why is the distance between the two holes functionally important?

If the hole pattern mates with 2 pins or bolts, the distance between the holes is more important than the distance from the edge to the second hole.



Dimension Accuracy

- There is no such thing as an "exact" measurement.
-
  - Every dimension has an implied or stated tolerance associated with it.
  - A tolerance is the amount a dimension is allowed to vary.

Rounding Off

- The more accurate the dimension the more expensive it is to manufacture.
  - To cut costs it is necessary to round off fractional dimensions.
- Round off the following fractions to two decimal places.

(5/16) .3125 → .31

(5/32) .1562 → .16

(1/8) .125 → .12

(3/8) .375 → .38

General Definitions

- Limits: The maximum and minimum diameters.
- Tolerance: The difference between two limits.
- Allowance: (Minimum Clearance) The difference between the largest shaft diameter and the smallest hole diameter.

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.

Tolerancing Standards

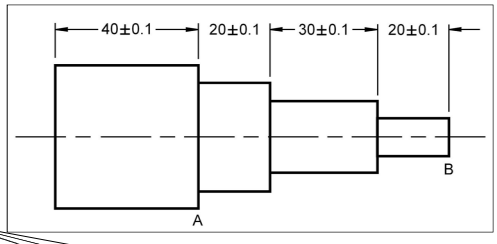
- The two most common standards agencies are;
  - American National Standards Institute (ANSI) / (ASME)
  - International Standards Organization (ISO).

Inch Tolerances Definitions

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

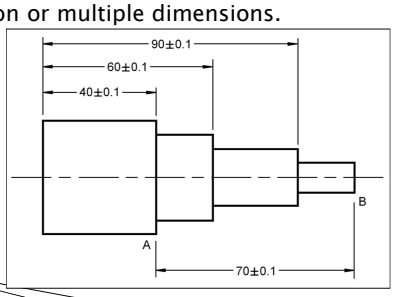
Tolerance Accumulation

- The tolerance between two features of a part depends on the number of controlling dimensions.



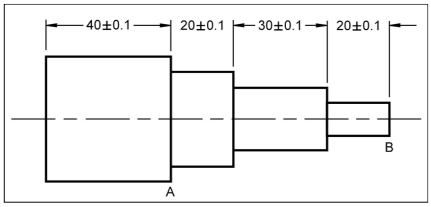
Tolerance Accumulation

- The distance could be controlled by a single dimension or multiple dimensions.



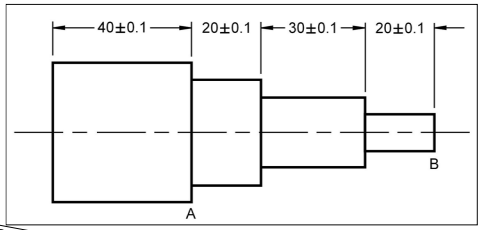
Tolerance Accumulation

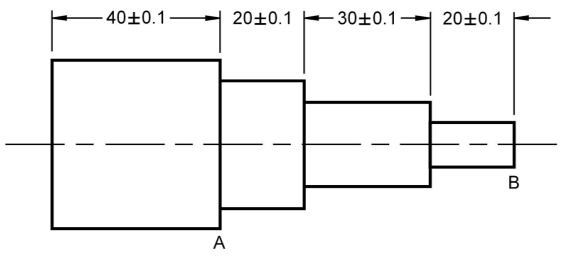
- The maximum variation between two features is equal to the sum of the tolerances placed on the controlling dimensions.



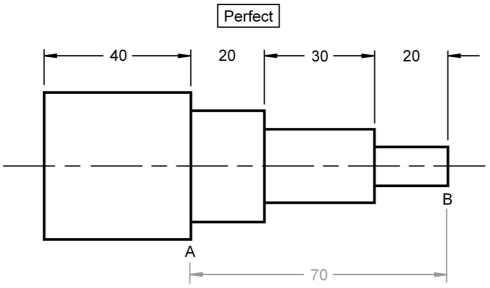
Tolerance Accumulation

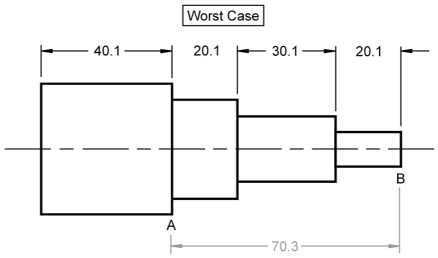
- As the number of controlling dimensions increases, the tolerance accumulation increases.

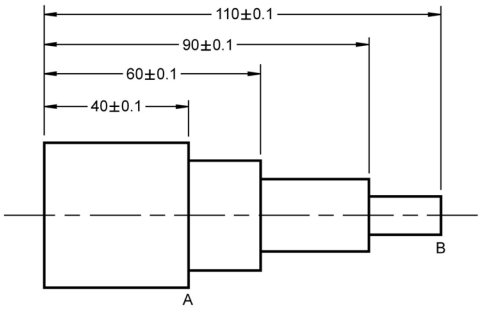




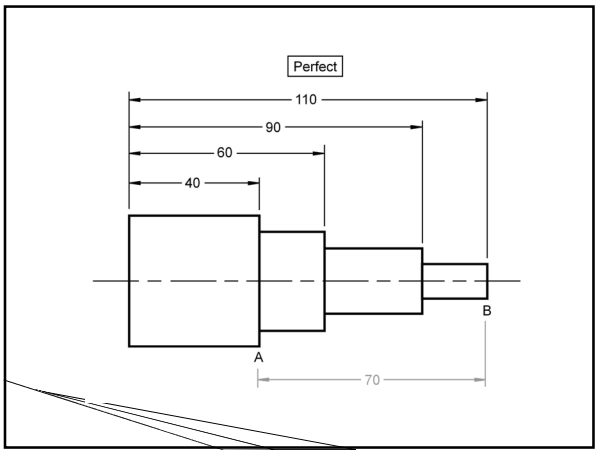
Tolerance accumulation between surface A and B = 0.3

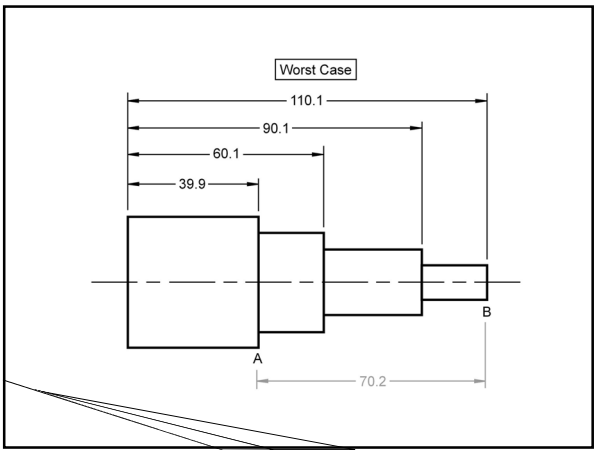


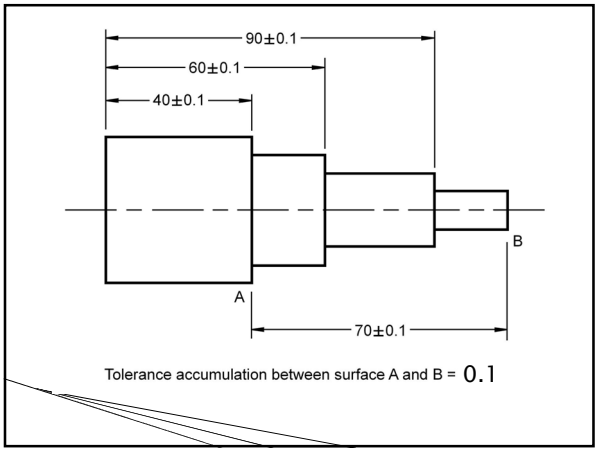




Tolerance accumulation between surface A and B = 0.2


Perfect
110
90
60
40
B
A
70


Worst Case
110.1
90.1
60.1
39.9
B
A
70.2


90±0.1
60±0.1
40±0.1
B
A
70±0.1
Tolerance accumulation between surface A and B = 0.1