

Dimensioning and Notes

Design Drafting I / II
Heritage High School
Fall 2007

Objectives

- ▶ Explain why dimensions and notes are needed on drawings.
- ▶ Describe the difference between unidirectional and aligned dimensioning.
- ▶ Apply the general rules for dimensioning inch and/or metric drawings.
- ▶ Dimension circles, holes, and arcs.
- ▶ Explain the five methods used in the changeover from conventional inch measurements to metric measurements.

Measurements are used to tell us:

* *How tall?*



* *How heavy?*



* *How many?*

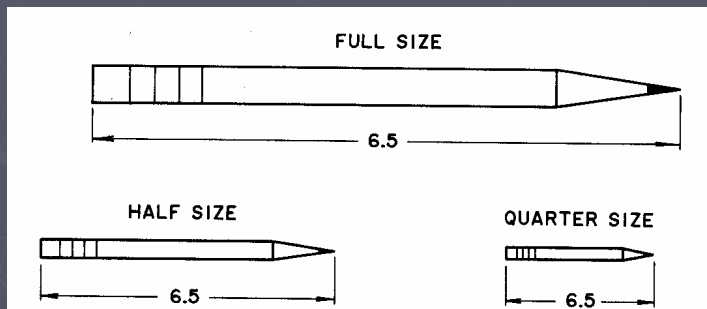


* *How much?*



Scaling vs. Dimensioning

- ▶ Drawing can be a different scales, but dimensions are ALWAYS as full scale.

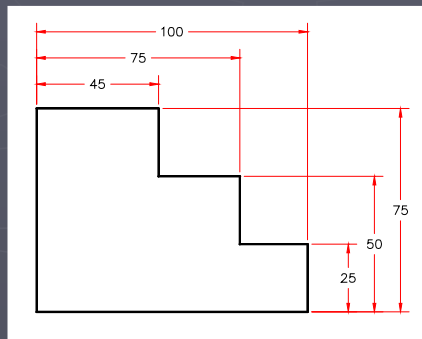


Basic Dimensioning

- ▶ Provide necessary information necessary for manufacturing.
- ▶ Dimensions define the sizes of the geometrical features of an object.
- ▶ Notes provide additional information not found in the dimensions.
- ▶ Dimensions and Notes can be added by hand or by the keyboard in CADD applications.

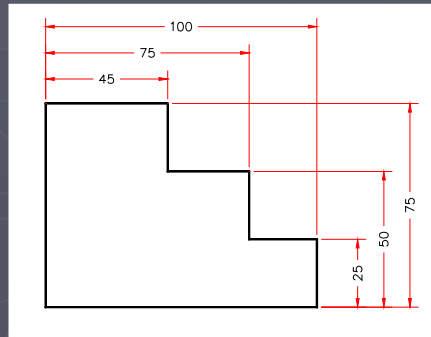
Unidirectional Dimensions

- ▶ Orientated to read from the bottom of the drawing.

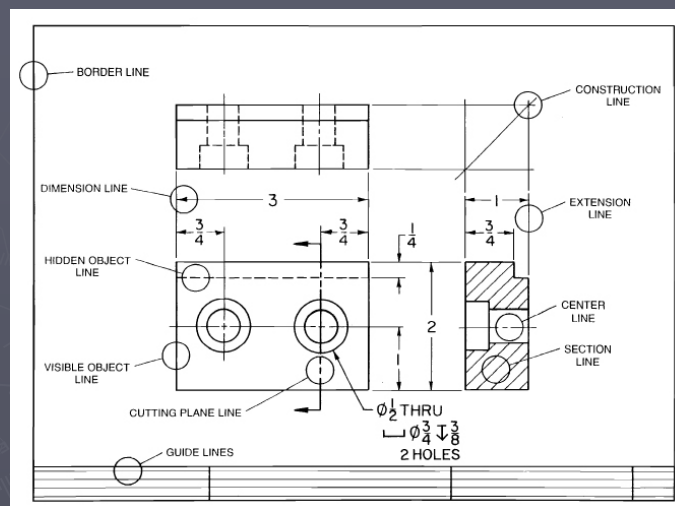


Aligned Dimensions

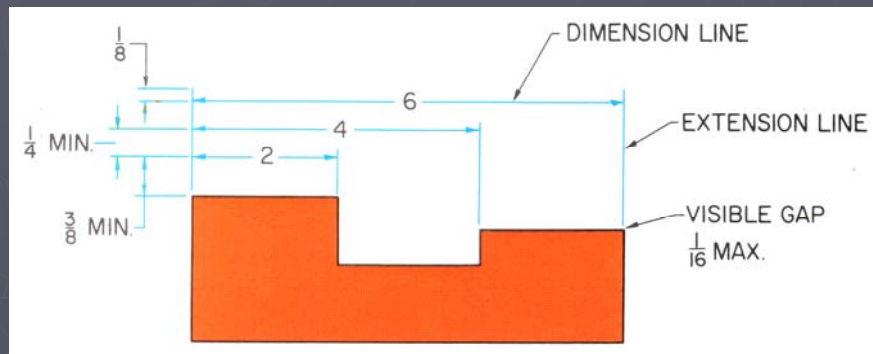
- Orientated to be read from the bottom and the right of the drawings (Parallel to the dimension line.)



Alphabet of Lines



Dimensioning a Drawing



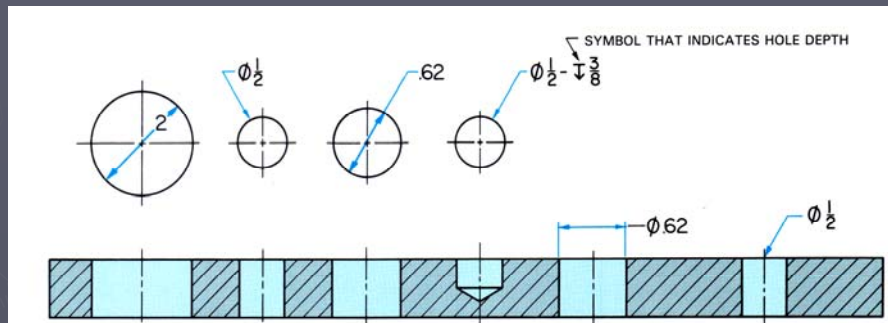
Dimension Lines Are:

- ▶ Fine solid lines
- ▶ Capped with arrowheads
- ▶ Placed on views that show the true shape of the drawing
- ▶ Should not be placed within the views (on top of the objects)
- ▶ Must be complete so no scaling is required

Dimension Lines Are:

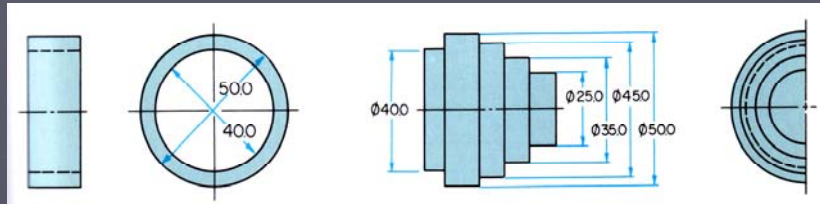
- ▶ Grouped together rather than scattered about the drawing
- ▶ Should not be duplicated unless they are absolutely necessary
- ▶ Should not cross extension lines
- ▶ When in inches, omit the inch symbol (")
- ▶ When in metric are in millimeters unless otherwise noted

Dimension Circles



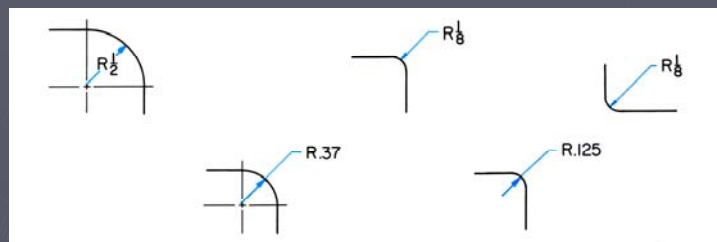
- ▶ The Greek letter ϕ (phi pronounced "fī") indicates that the dimension is a diameter.
- ▶ This symbol is placed *before* the dimension.
- ▶ The leader *always* points to the center.

Concentric Circles




- ▶ \emptyset is not needed if the dimension is on the circular view
- ▶ Half of a right side view is needed for concentric circles


Dimension Arcs





- ▶ The capital letter **R** indicates that the dimension is a radius
- ▶ It is placed *before* the dimension.


Symbols Used to Dimension

 Counterbore or spotface symbol

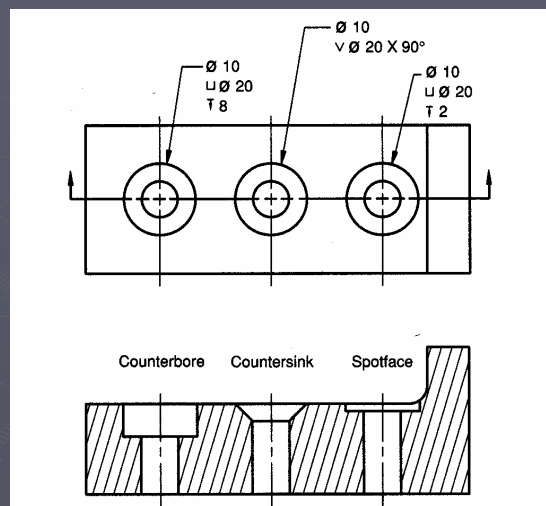
 Countersink symbol

 Diameter symbol

 Square symbol

 Depth symbol

Symbols Used to Dimension



Providing Information

1. Dual dimensioning
2. Dimensioning with letters and tabular chart.
3. Metric dimensioning with readout chart.
4. Dimensioning with metric units only
5. Undimensioned master drawings.

Dual Dimensioning

- ▶ First method devised to dimension engineering drawing with both inch and metric units.
- ▶ The dual dimensions are presented by the position method or the bracket method.
 - When drawn in the US the inch dimensions are placed on top. The metric measurement is placed in brackets.
 - Very complicated and seldom used today.

Dimensioning w/ Letters

- ▶ Letters are used in place of either inch or millimeter dimensions.
- ▶ A **tabular chart** is added to the drawing, showing both the metric and inch equivalents of each letter.

Dimension Chart		
	mm	inch
A	60.0	2.365
B	80.0	3.150

Metric Dimensions w/ Readout Chart

- ▶ Parts are designed/drawn to metric standards
- ▶ Only metric dimensions are placed on the drawing.
- ▶ A readout chart is added to the drawing to compare the values.

Metric Dimensions Only

- ▶ A method to get engineers, designers, drafters, and craftworkers to “think metric”
- ▶ There was a movement at one time in the US to move from inches to metrics, but it never really caught on.

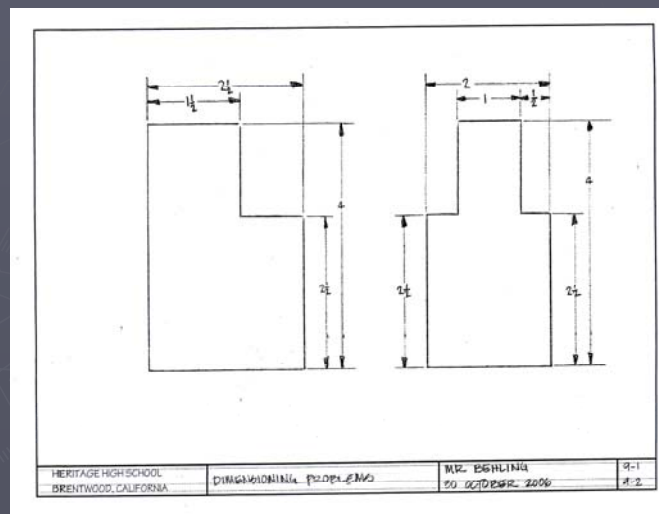
Undimensioned Master Drawings

- ▶ Drawn without dimensions.
- ▶ Copies are made and then dimensions are added
 - Metric on one
 - Inches on another

Geometric Dimensioning and Tolerancing (GD&T)

- ▶ Highly precise system of specifying the dimensions and tolerances
- ▶ Controls the interpretation of
 - Form
 - Profile
 - Orientation
 - Location
- ▶ A datum is an exact point, axis, plane, or surface from which features of a part are located.
 - (There is a type-o on the handout)

Drawings 1 and 2



Drawings 3 and 4

