

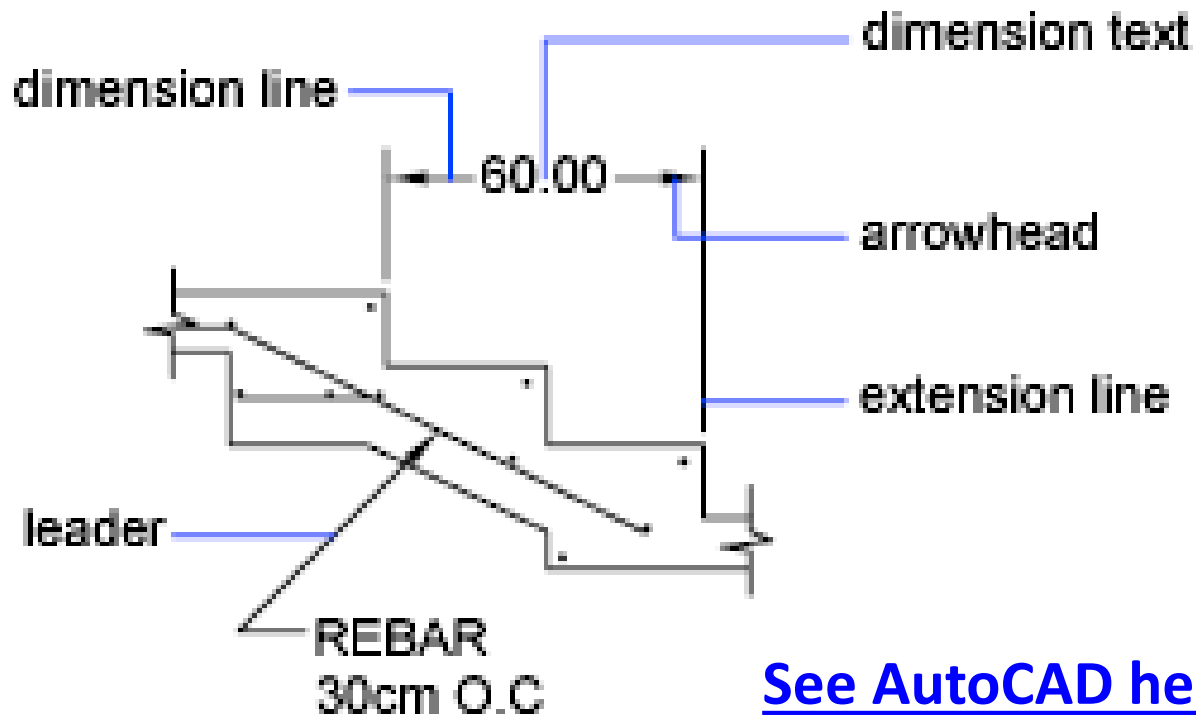
Elementary Dimensioning

Standards Institutions

- **ANSI** - American National Standards Institute - creates the engineering standards for North America.
- **ISO** - International Organization for Standardization - a world-wide organization that creates engineering standards with approximately 100 participating countries.

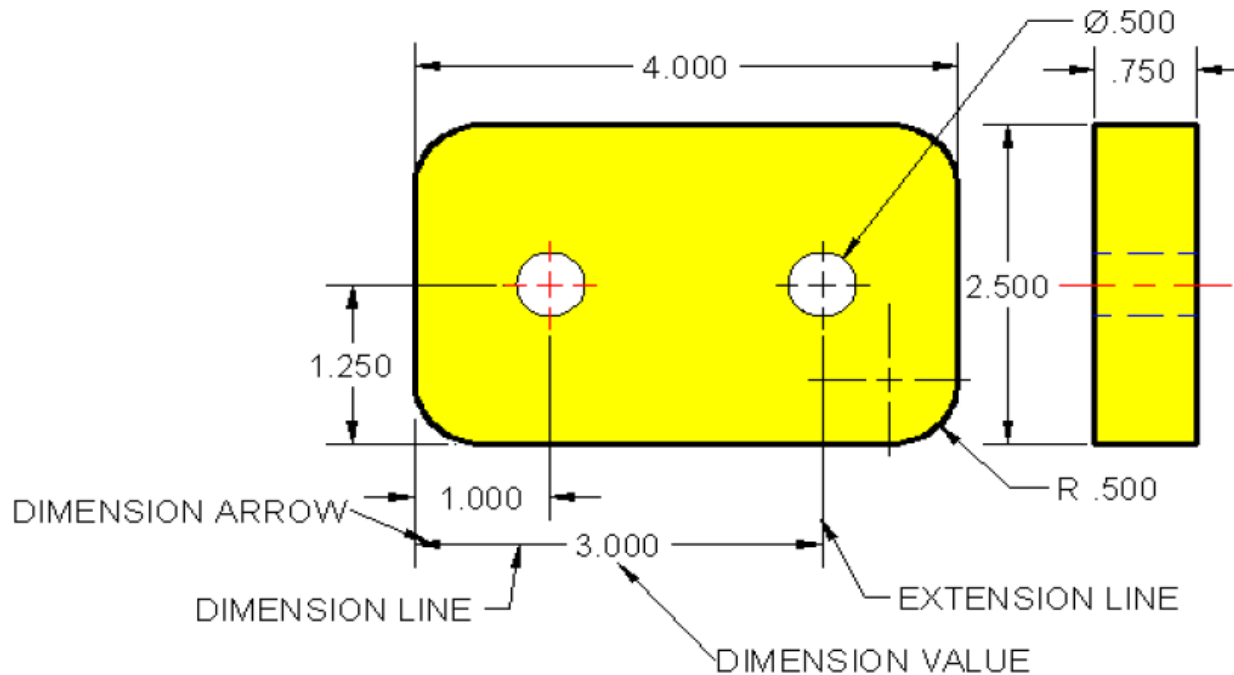
Parts of a Dimension

- Extension line
- Dimension line and arrow head
- Dimension text



[See AutoCAD help](#)

ANSI Drafting Standards - Dimensions



A dimension in a drawing consist of the following items: *dimension arrow*, *dimension line*, *dimension value*, and *extension line*.

Dimensioning Symbols

° Degree Symbol

() Reference Symbol

∅ Diameter Symbol

R Radius Symbol

⌊ Counter Bore or Spot Face Symbol

∨ Counter Sink Symbol

⌞ Depth or Deep Symbol

X Places or By Symbol

± Plus/Minus Symbol

⌀ Center Line Symbol

□ Square(shape) Symbol

⌒ Arc Symbol

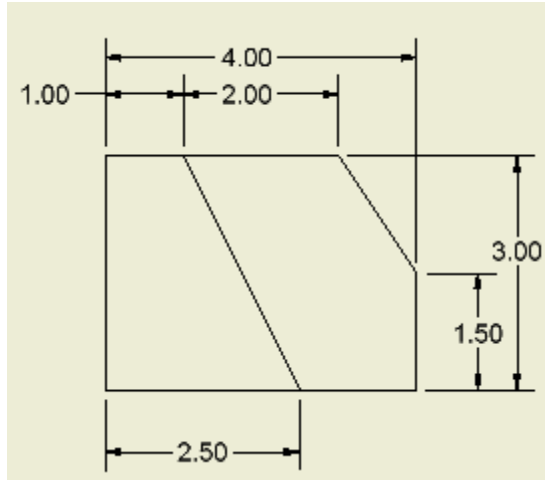
▴ Slope Symbol

Dimensioning Methods

- Dimensions are represented on a drawing using one of two systems, unidirectional or aligned.
- The *unidirectional* method means all dimensions are read in the same direction.
- The *aligned* method means the dimensions are read in alignment with the dimension lines or side of the part, some read horizontally and others read vertically.

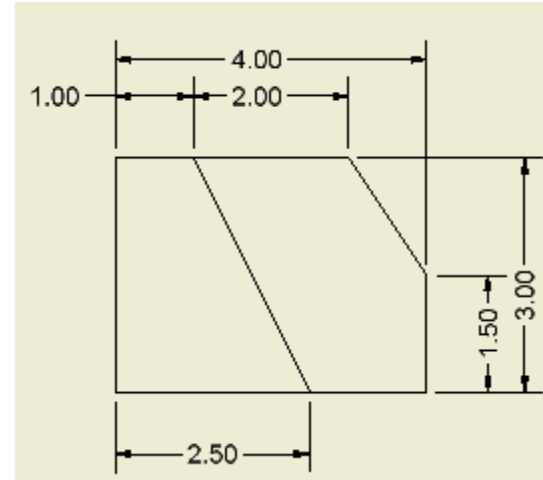
Dimension Text

Unidirectional vs. Aligned



Unidirectional

dimensions are placed so they can be read from the bottom of the drawing sheet. This method is commonly used in mechanical drafting.

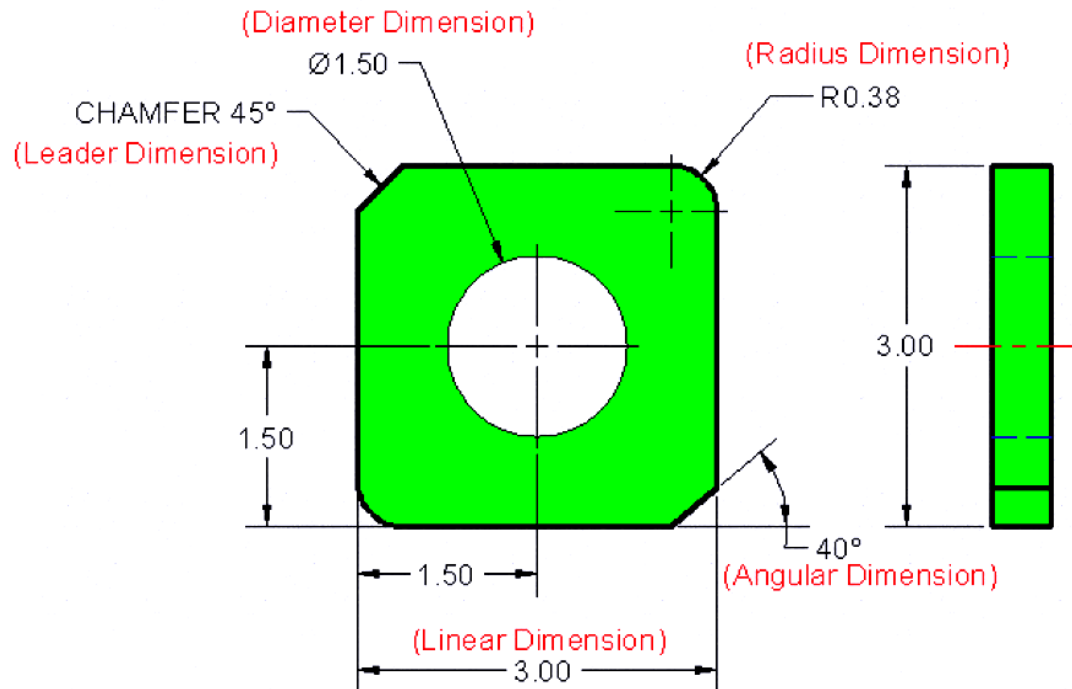


Aligned dimensions are placed so the horizontal dimensions can be read from the bottom of the drawing sheet and the vertical dimensions can be read from the right side of the drawing sheet. This method is commonly used in architectural and structural drafting.

Two Types of Dimensions

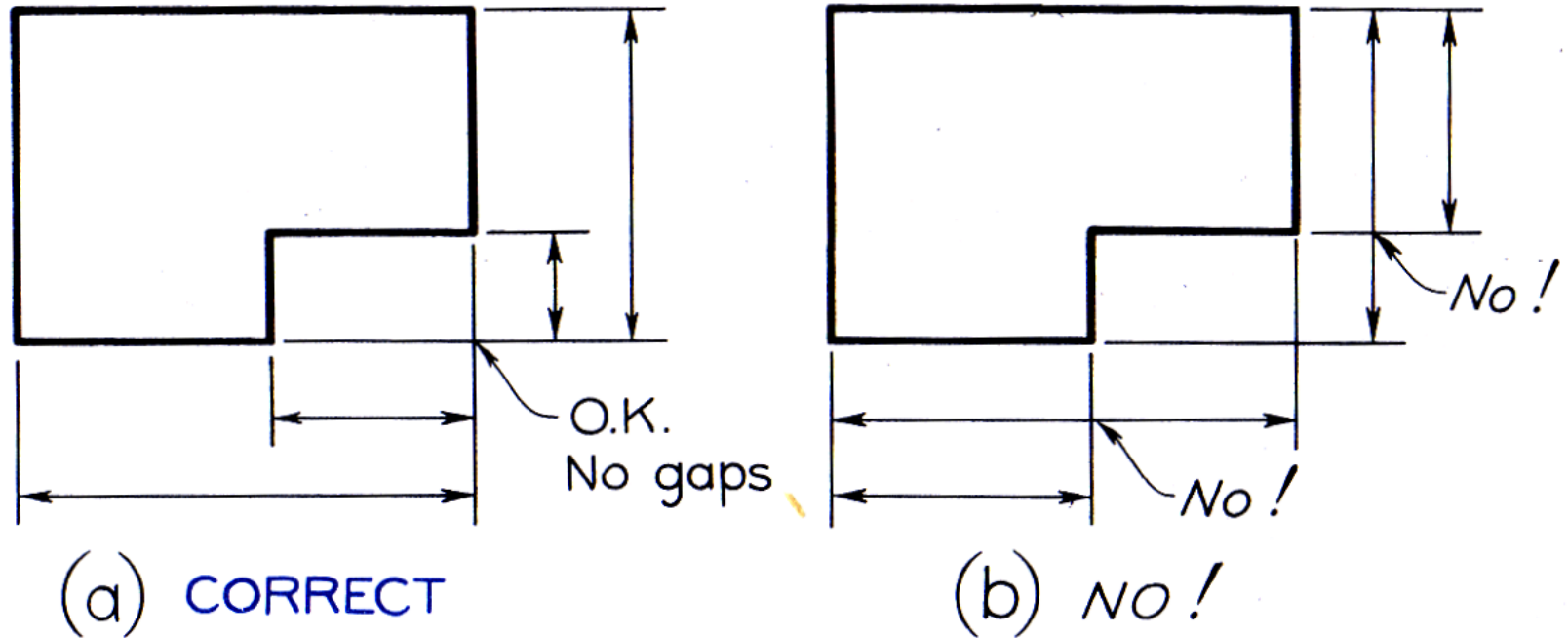
- There are two classifications of dimensions: size and location.
- *Size* dimensions are placed in direct relationship to a feature to identify the specific size.
- *Location* dimensions are used to identify the relationship of a feature to another feature within an object.

ANSI Drafting Standards - Dimensions



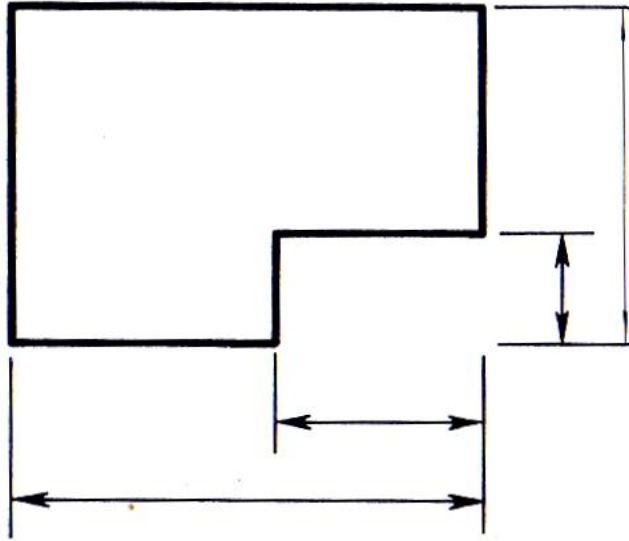
Remember - a **location dimension** locates holes or other part features.
A **size dimension** provides a radius, diameter, length, width, thickness, etc.

Elementary Dimensioning

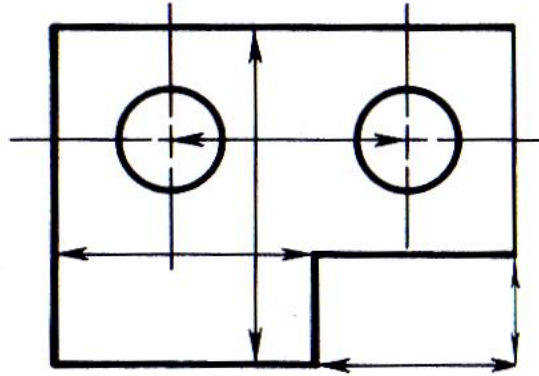


Dimensions should be stacked with shorter dimensions closer to the object and longer dimensions farther away so that dimension lines will not cross extension lines. Extension lines crossing extension lines is OK

Elementary Dimensioning



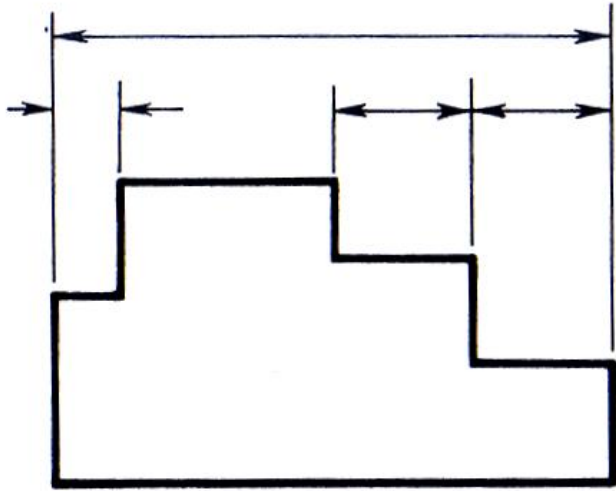
(c) *NO!*



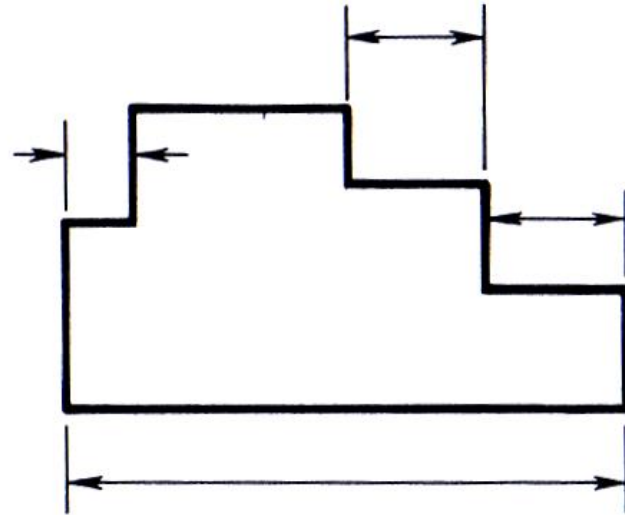
(d) *NO!*

Extension lines should extend to the profile of the object with a small gap between the end of the extension line and the object. **No line of the drawing should be used as a dimension line or coincide with a dimension line.**

Elementary Dimensioning



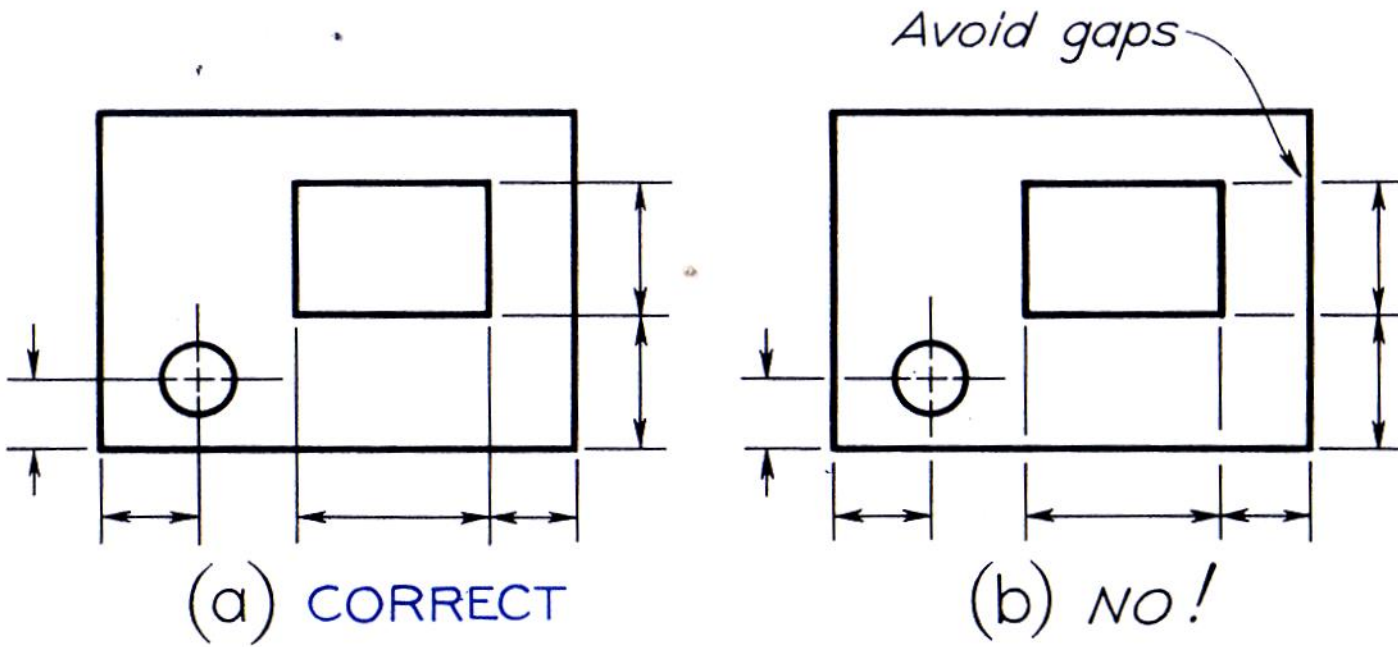
(a) CORRECT



(b) NO!

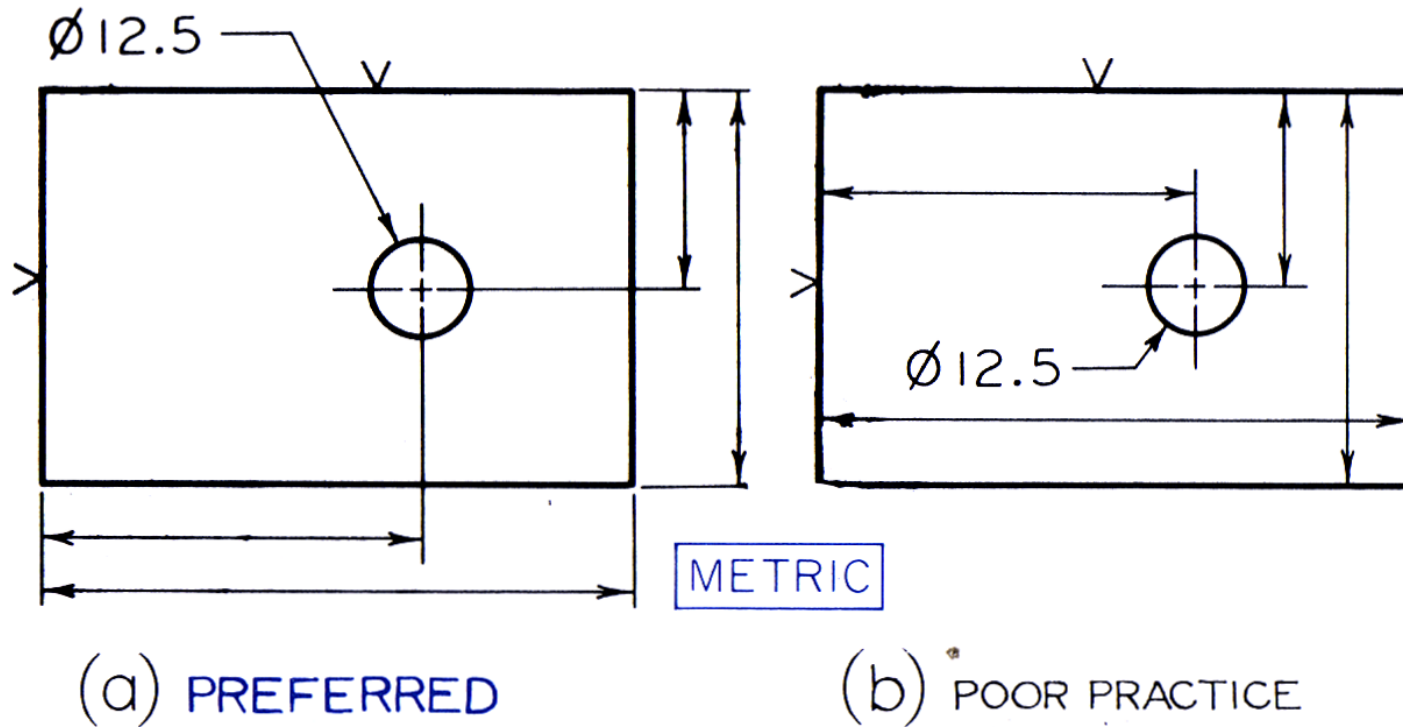
Dimensions at each stacking level should be aligned.

Elementary Dimensioning



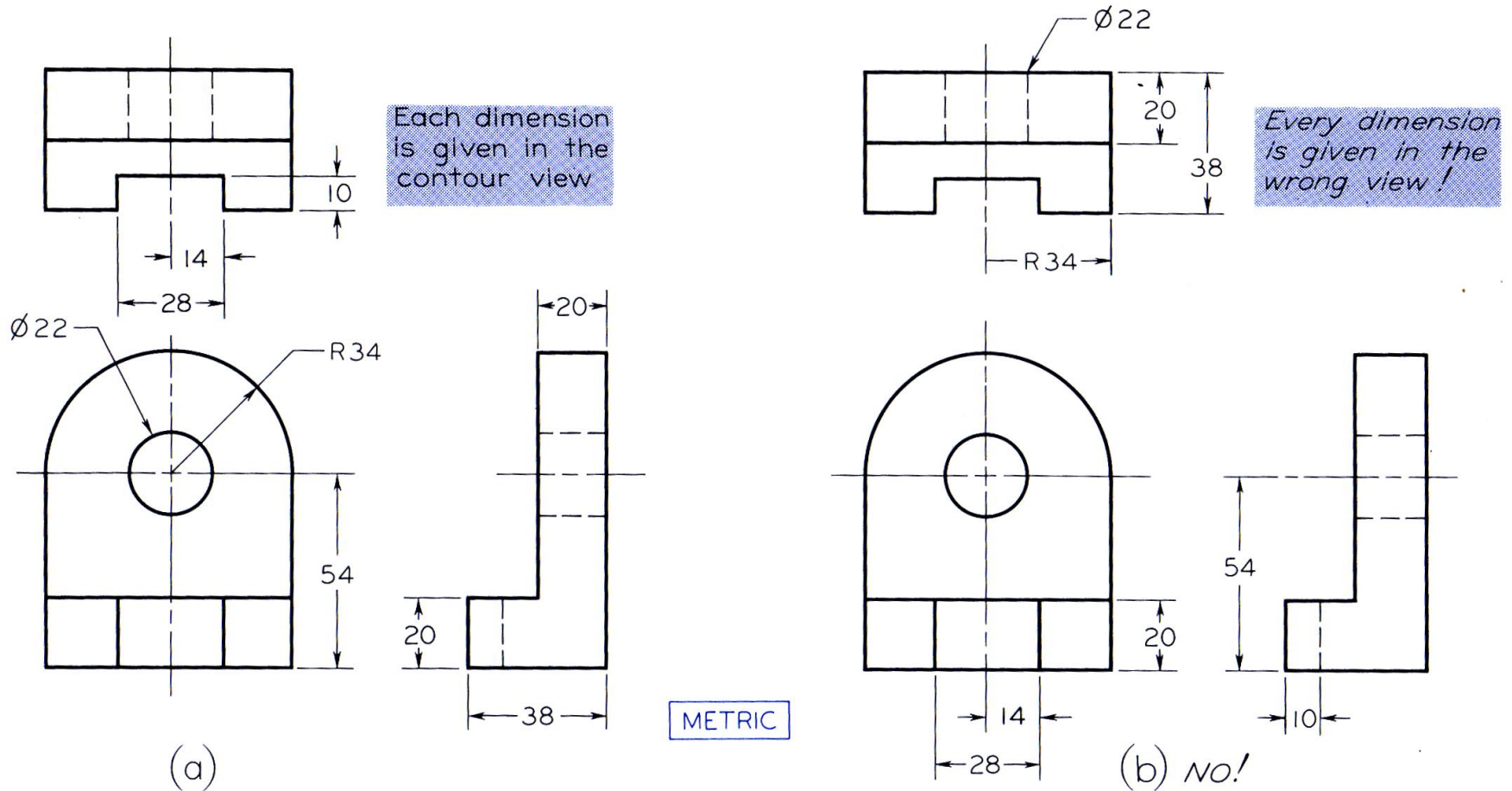
Don't break extension lines.

Elementary Dimensioning



Dimensions shouldn't be placed on a view unless doing so is unavoidable or if it enhances clarity.

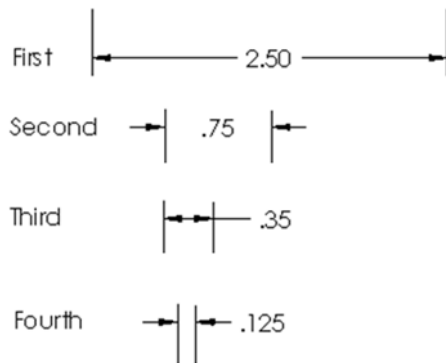
Elementary Dimensioning



Dimensions should be placed in views where the dimensioned features are shown in their true shapes. Avoid dimensioning to hidden lines wherever possible.

ANSI Drafting Standards - Dimensions

- Placement of Linear Dimensions - Order of Preference. When there is **not enough room** on the drawing view between the extension lines to accommodate either the dimension value or the dimension lines - located the outside extension lines as illustrated.



1. Arrows in / dimension in
2. Arrows out / dimension in
3. Arrows in / dimension out
4. Arrows out / dimension out

Dimensioning Checklist

- Each dimension should be written clearly with only one way to be interpreted.
- A feature should be dimensioned only once.
- Dimension and extension lines should not cross.
- Each feature should be dimensioned.
- Dimension features or surfaces should be done to a logical reference point.

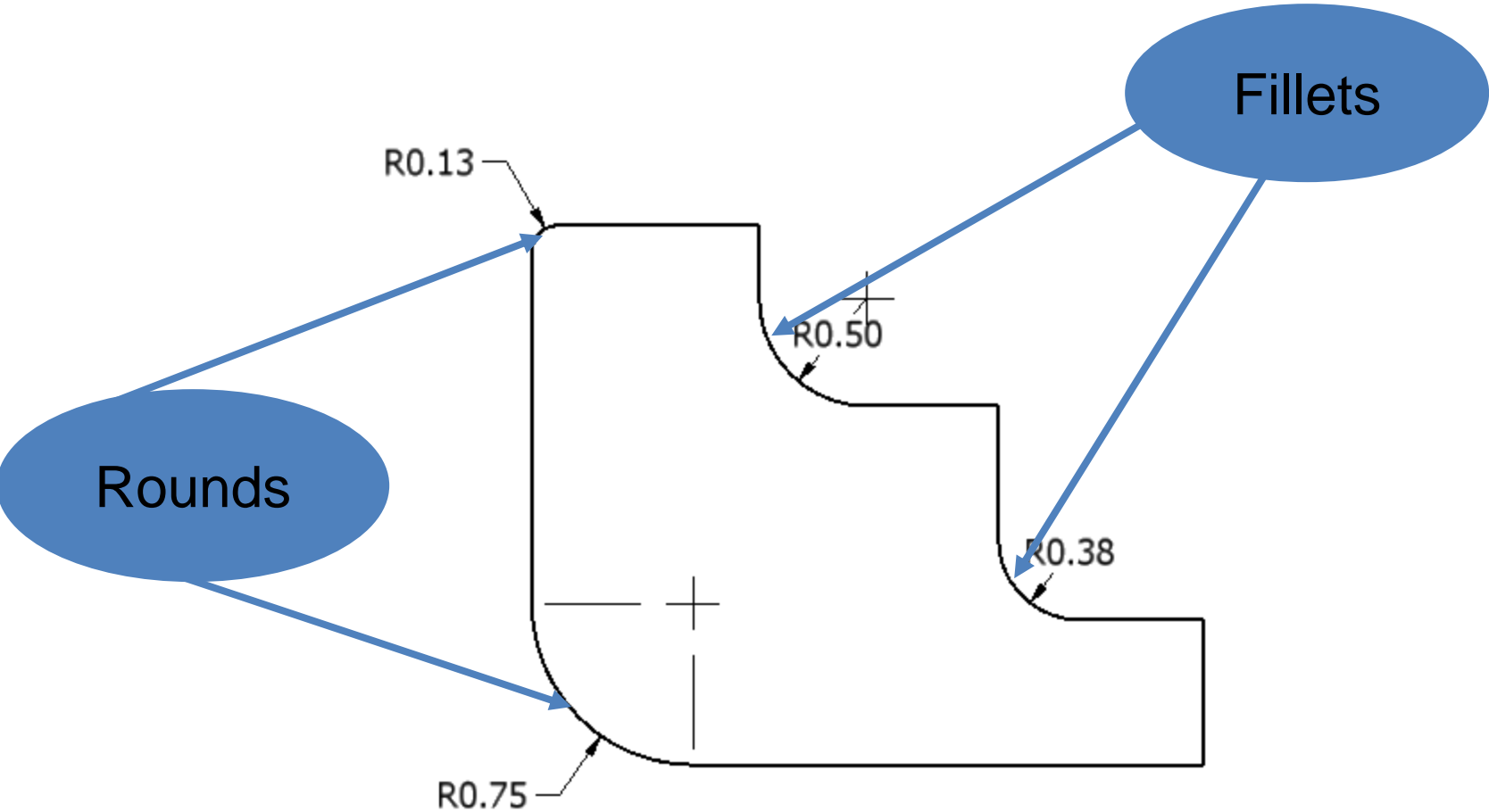
Dimension Checklist

- Dimension a circle with a diameter and an arc with a radius.
- A center line should be extended and used as an extension line.
- Dimension features on a view should clearly show its true shape.
- Enough space should be provided to avoid crowding and misinterpretation.

Dimension Checklist

- Extension lines and object lines should not overlap.
- Dimensions should be placed outside the part.
- Center lines or marks should be used on all circles and holes.

Fillets and Rounds

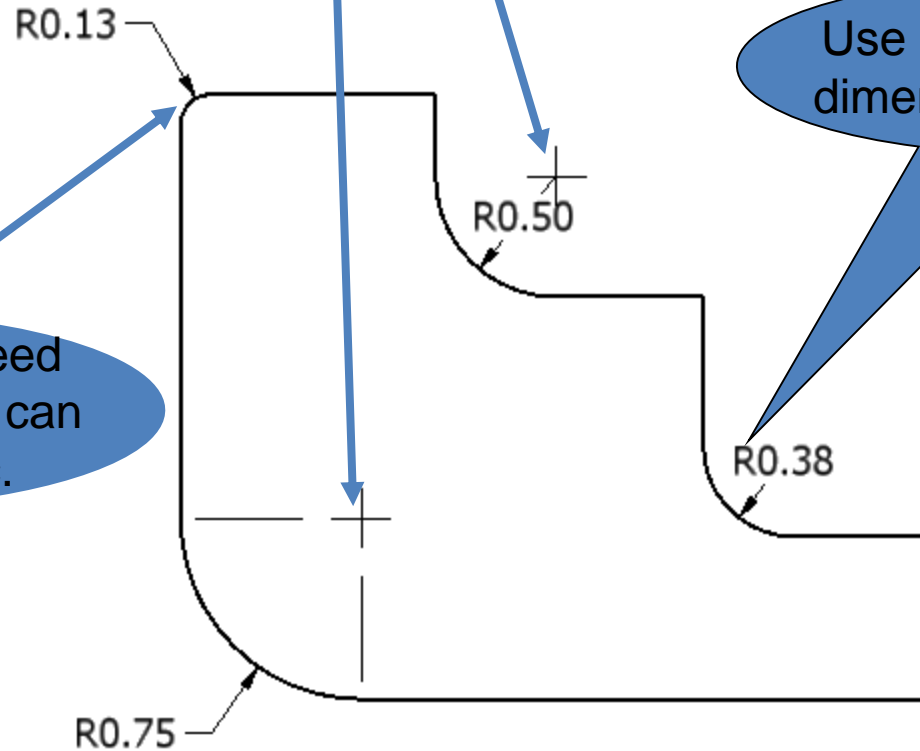


Fillets and Rounds

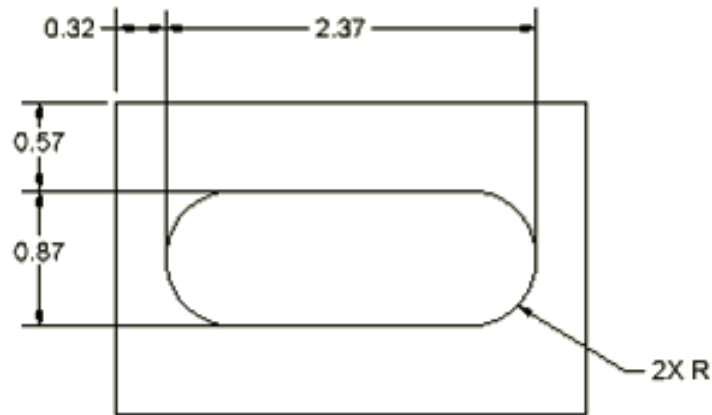
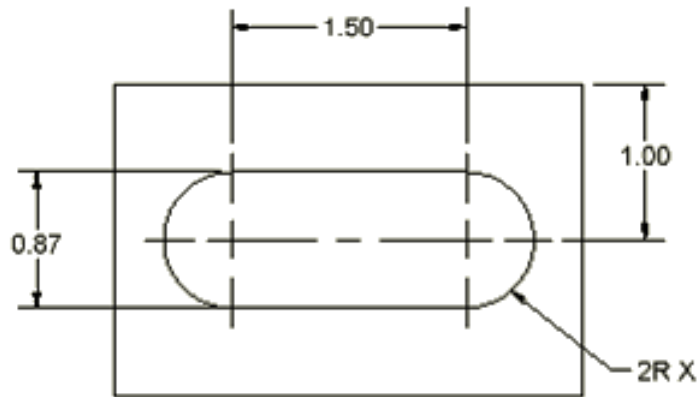
Large arcs use center marks.

Use a capital "R" for dimensioning the arc.

Small arcs do not need center marks. Arrow can be outside the arc.

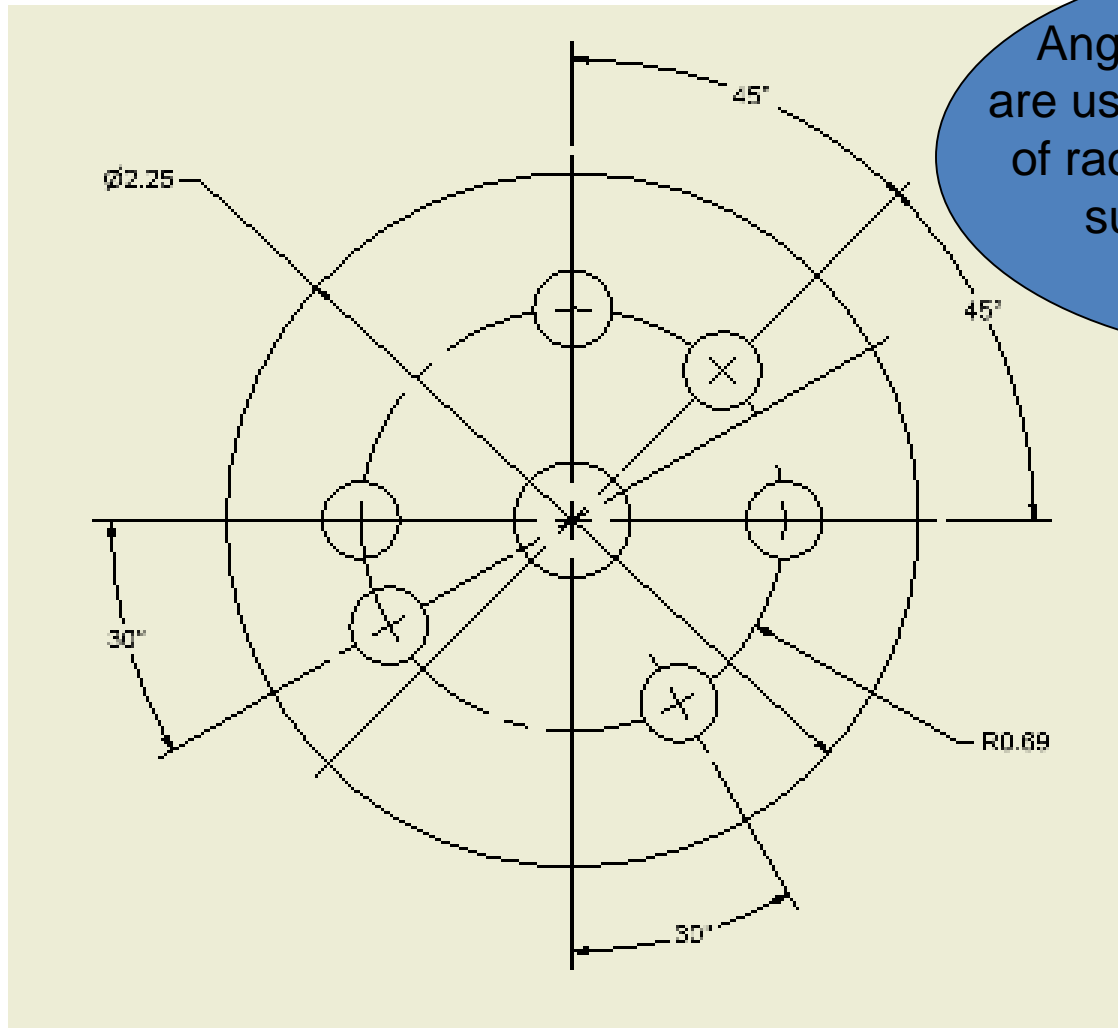


Slot Dimensioning



Two acceptable methods for dimensioning slotted holes.

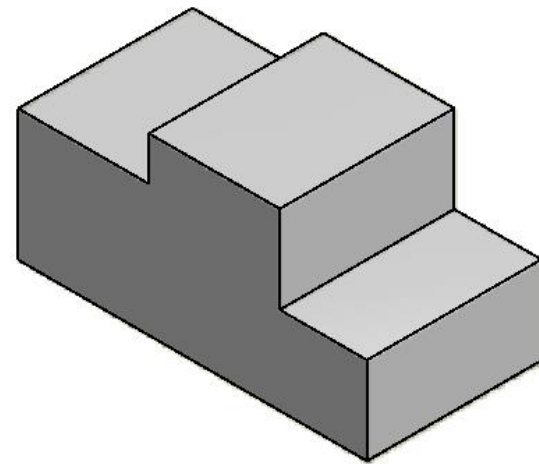
Dimensioning Radial Patterns



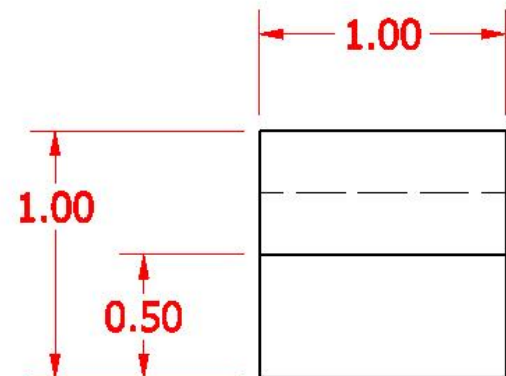
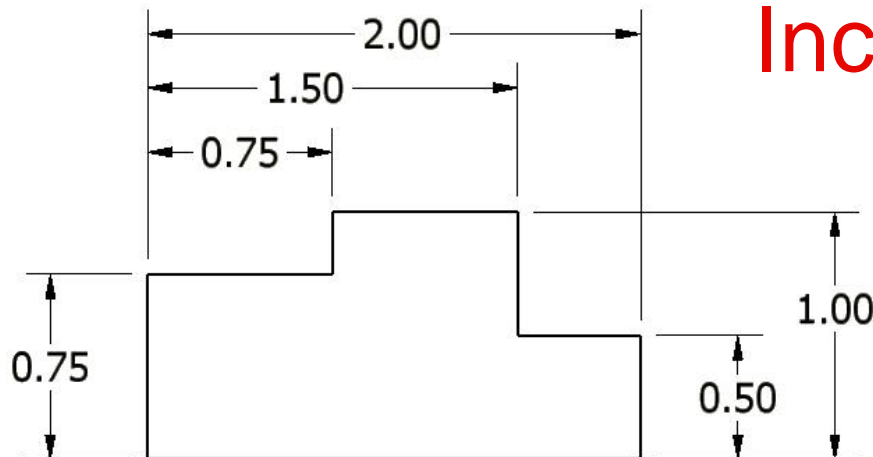
Angles and radius values are used to locate the centers of radial patterned features, such as the holes on this plate.

Dimension Guidelines

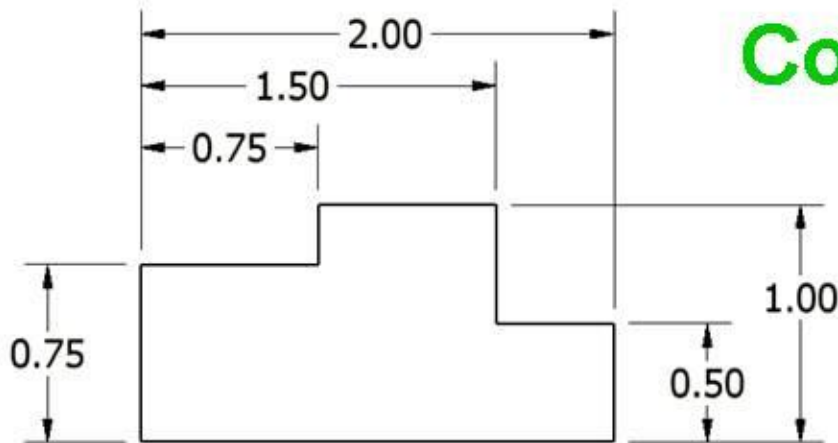
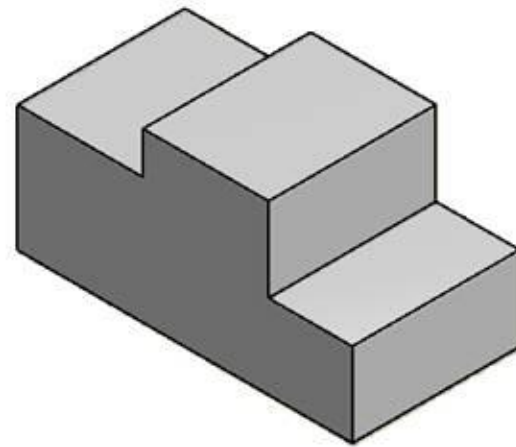
1. Dimensions should NOT be duplicated, or the same information given in two different ways.



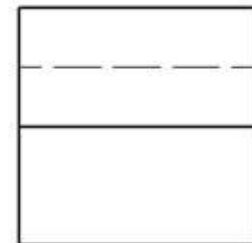
Incorrect



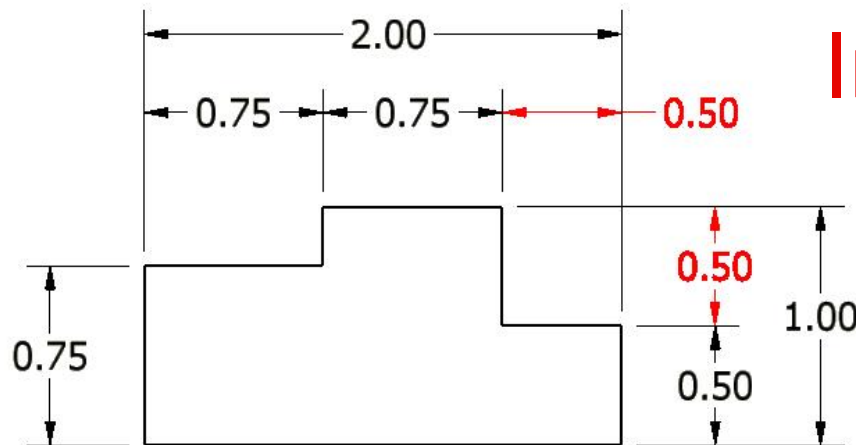
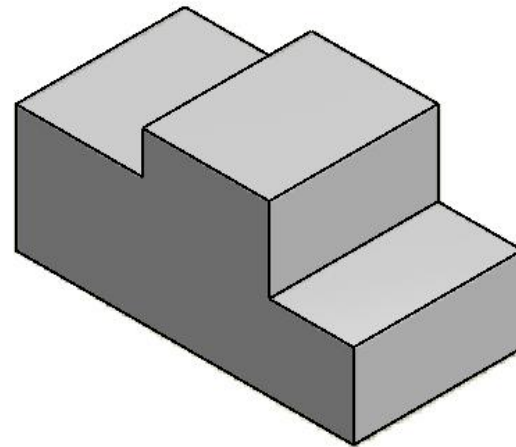
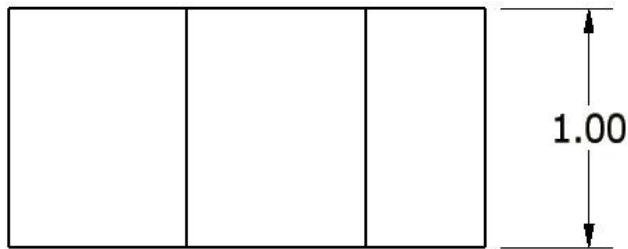
1. Dimensions should NOT be duplicated, or the same information given in two different ways.



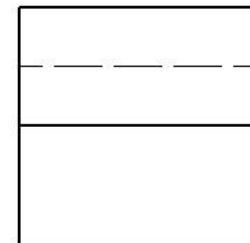
Correct



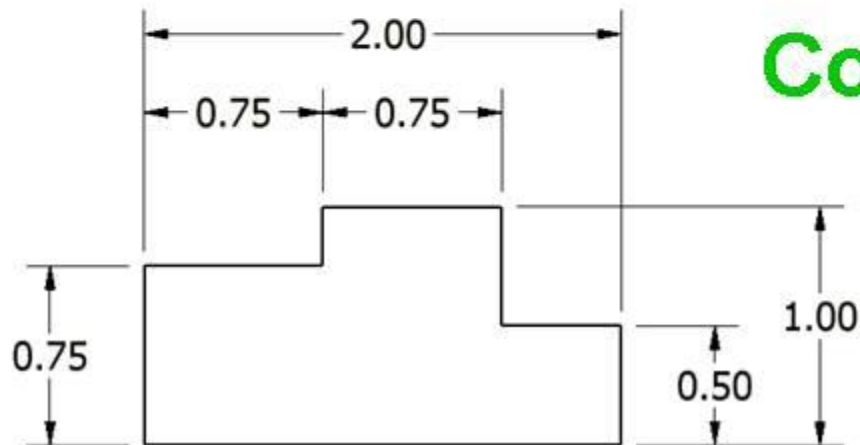
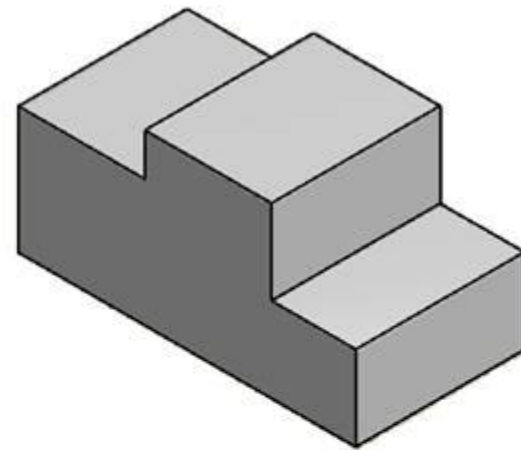
2. No unnecessary dimensions should be used – only those needed to produce or inspect the part.



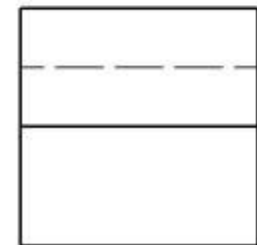
Incorrect



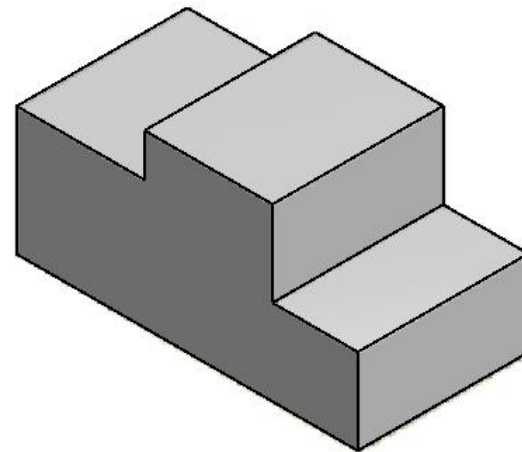
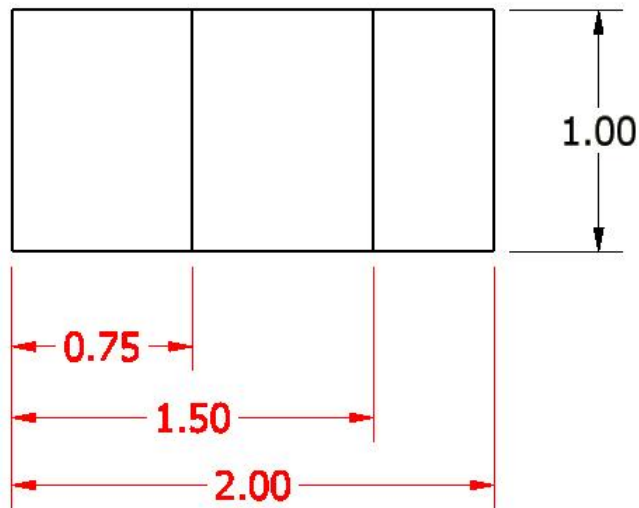
2. No unnecessary dimensions should be used – only those needed to produce or inspect the part.



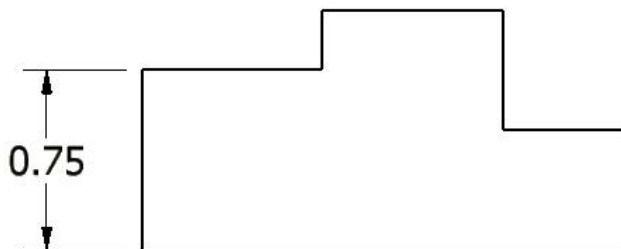
Correct



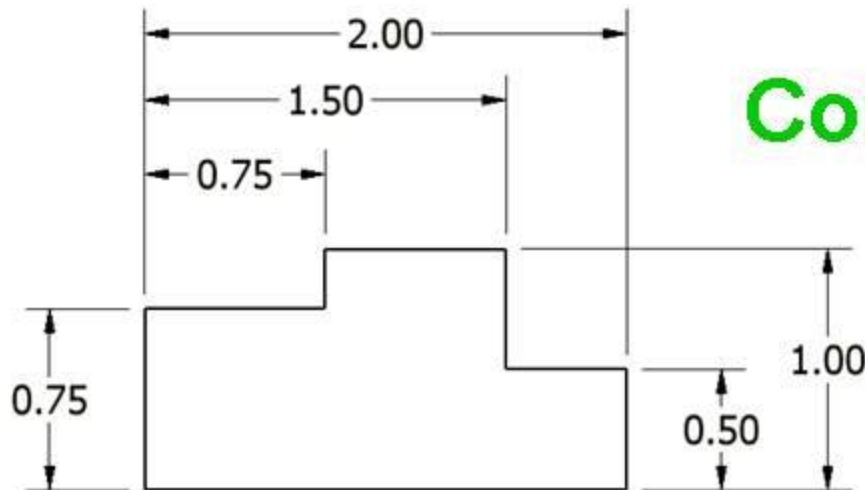
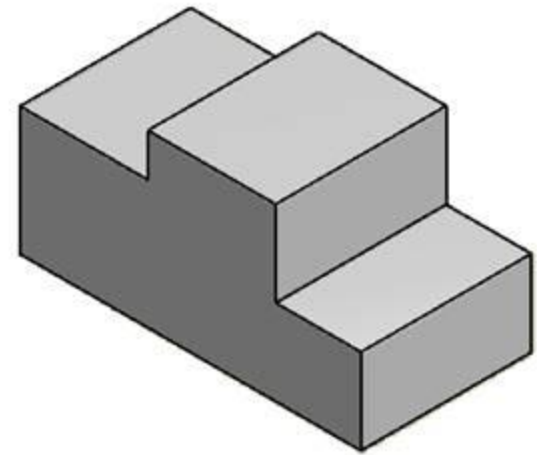
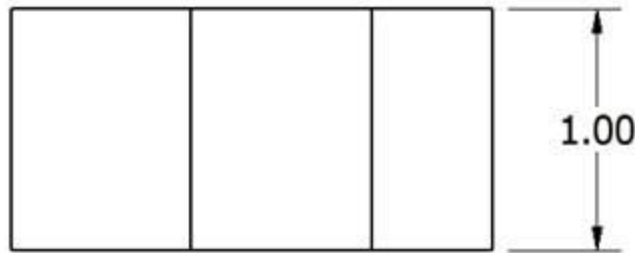
3. Dimensions should be attached to the view that best shows the *contour* of the feature to be dimensioned.



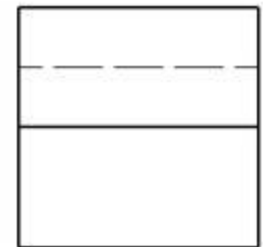
Incorrect



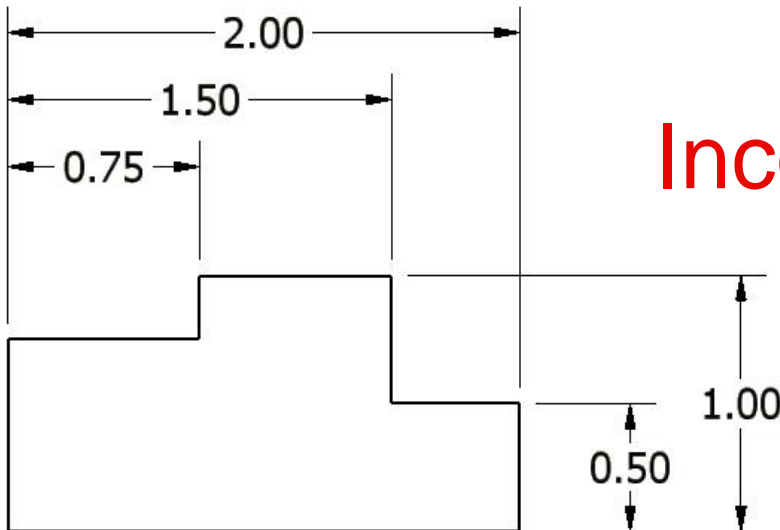
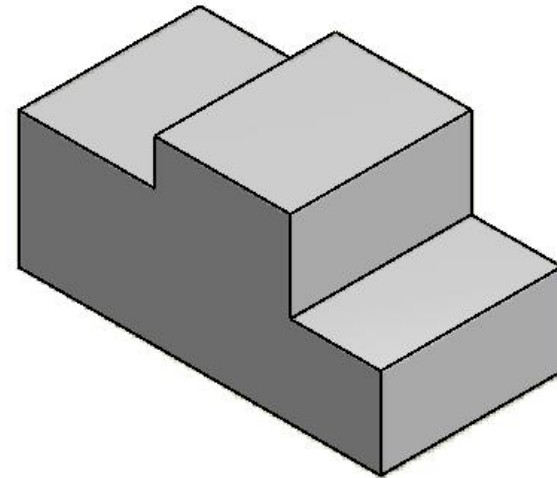
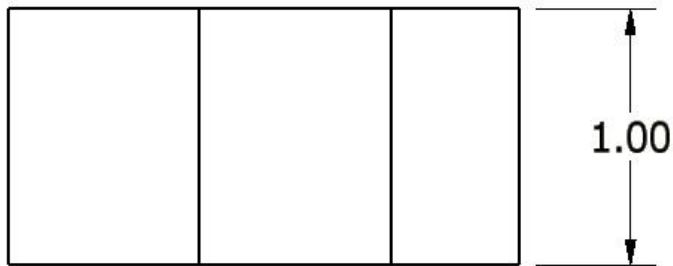
3. Dimensions should be attached to the view that best shows the *contour* of the feature to be dimensioned.



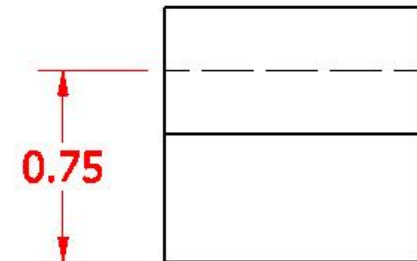
Correct



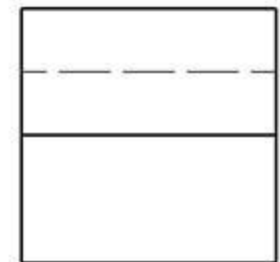
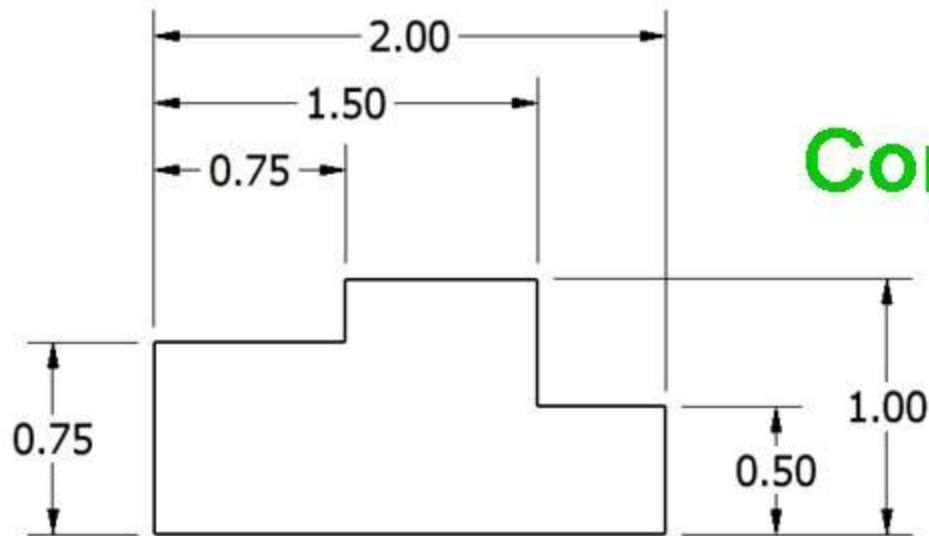
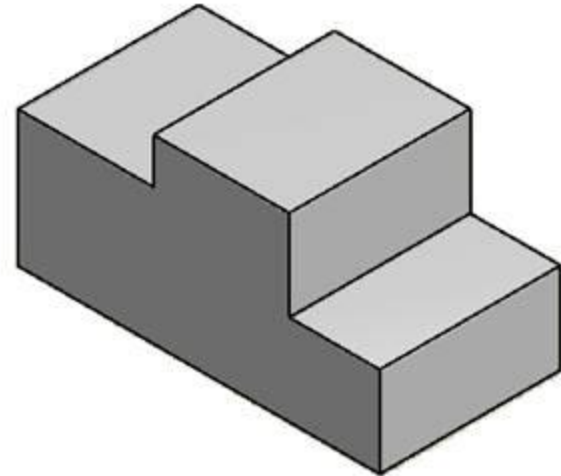
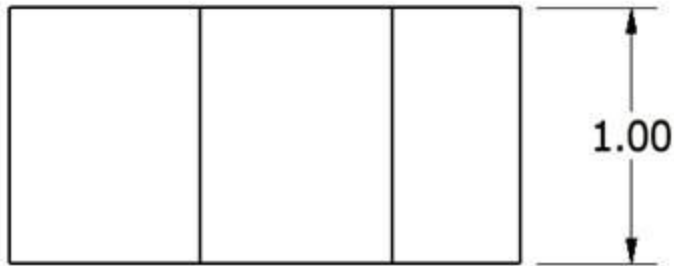
4. Whenever possible, avoid dimensioning to hidden lines and features.



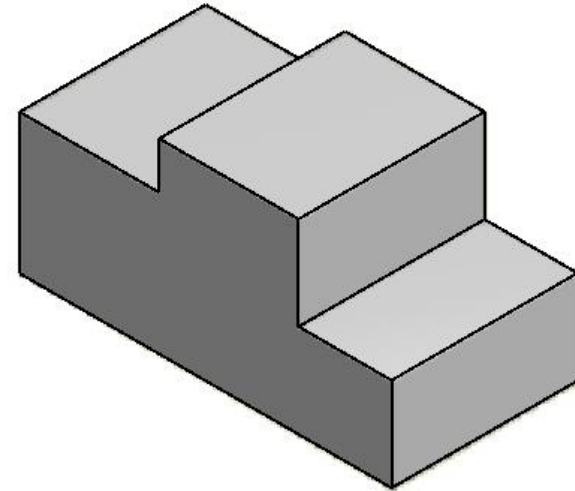
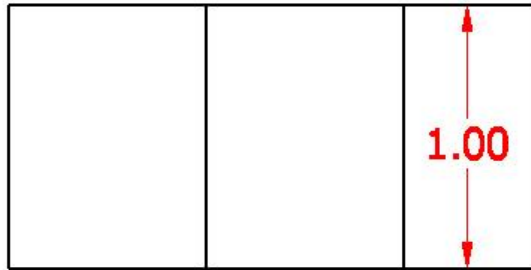
Incorrect



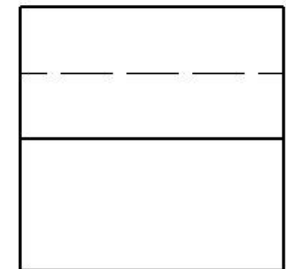
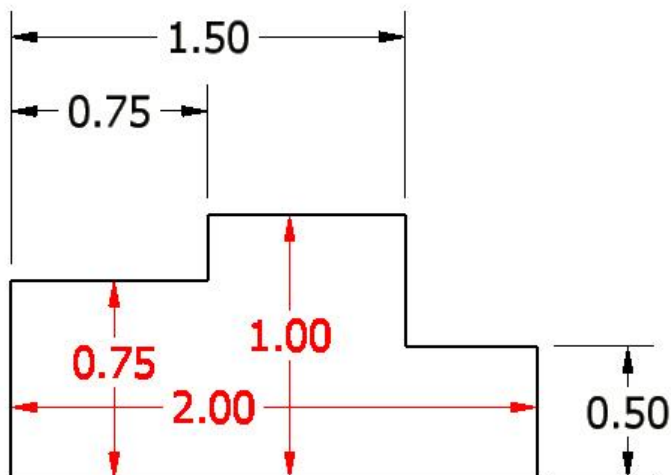
4. Whenever possible, avoid dimensioning to hidden lines and features.



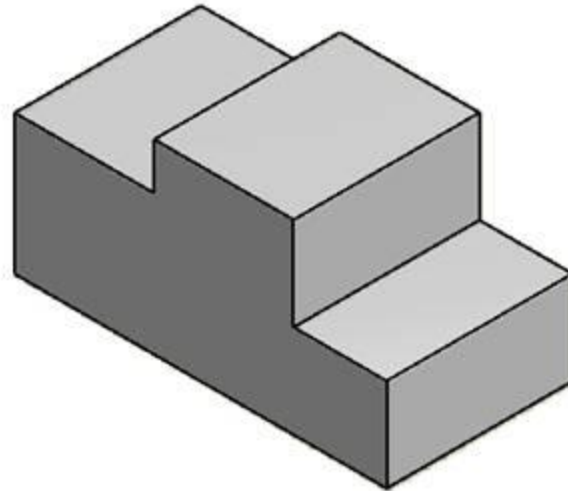
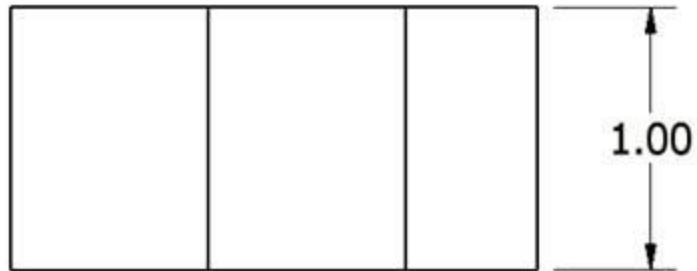
5. Avoid dimensioning over or through the object.



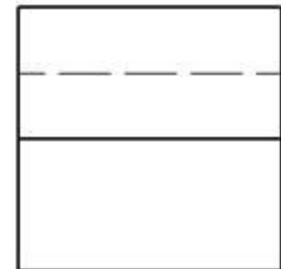
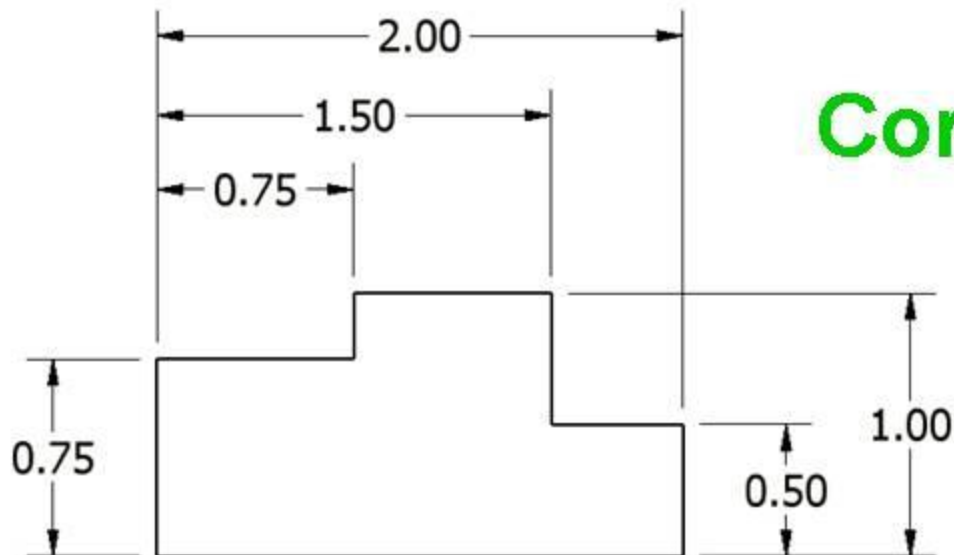
Incorrect



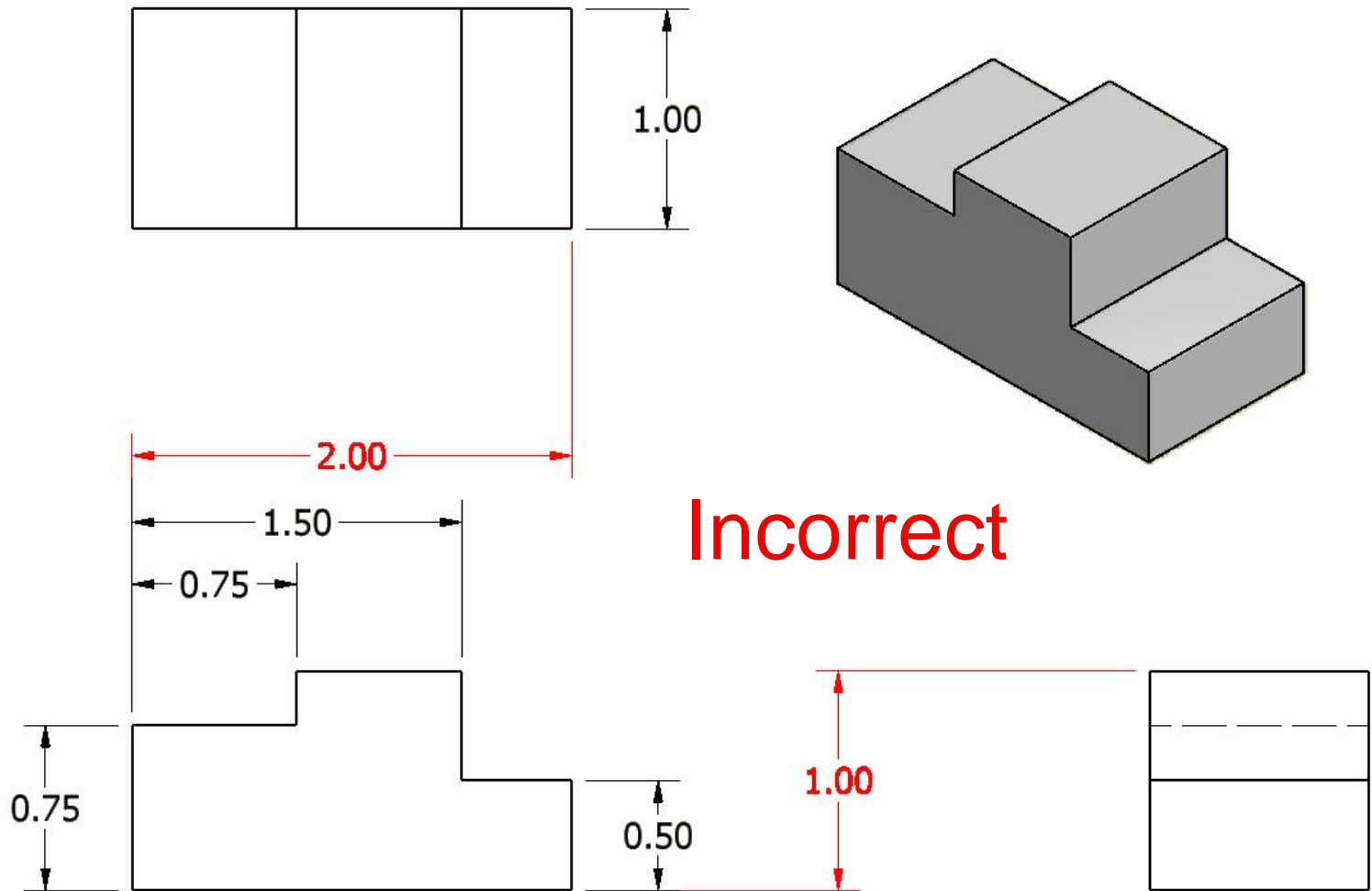
5. Avoid dimensioning over or through the object.



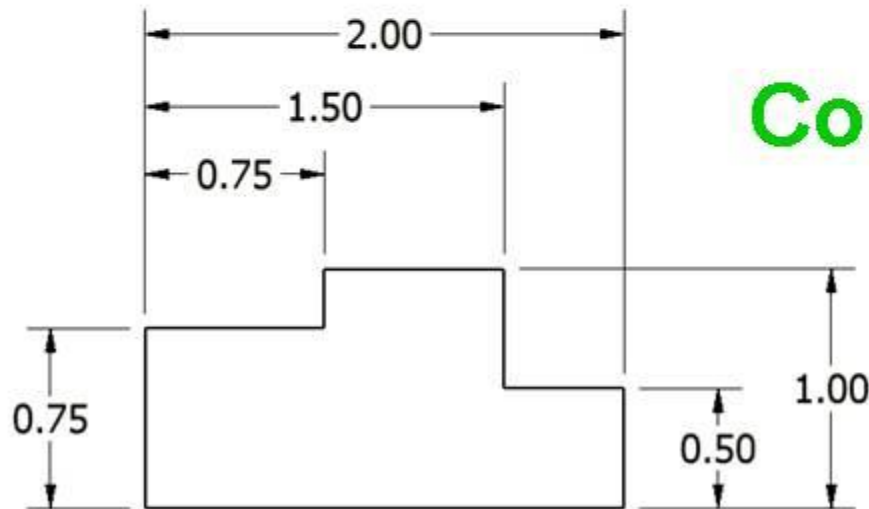
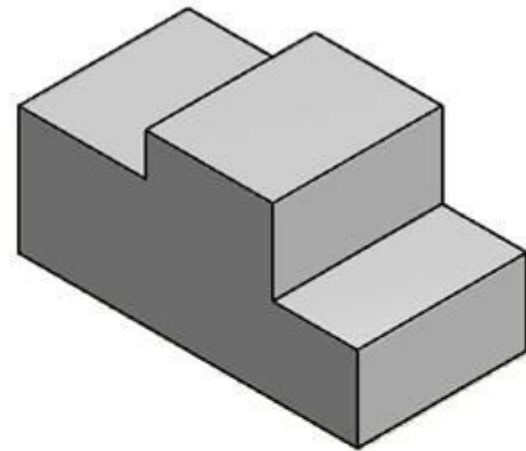
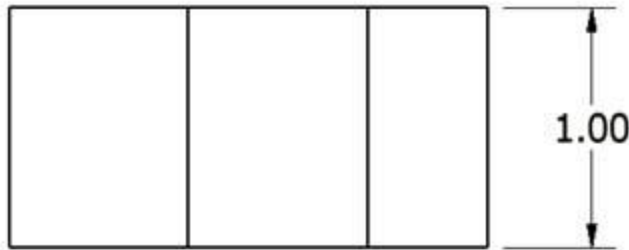
Correct



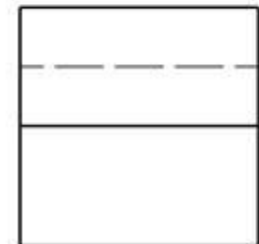
6. A dimension should be attached to only one view; for example, extension lines should not connect two views.



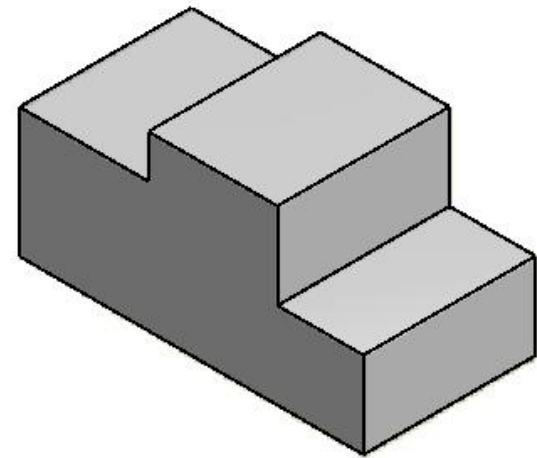
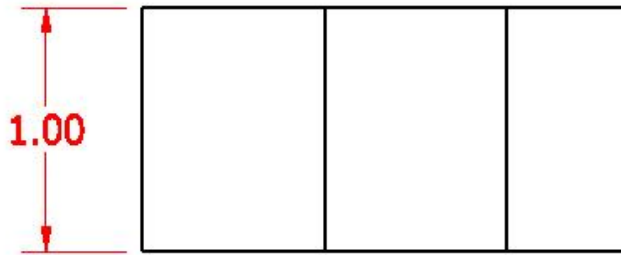
6. A dimension should be attached to only one view; for example, extension lines should not connect two views.



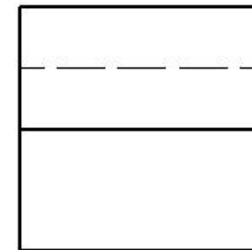
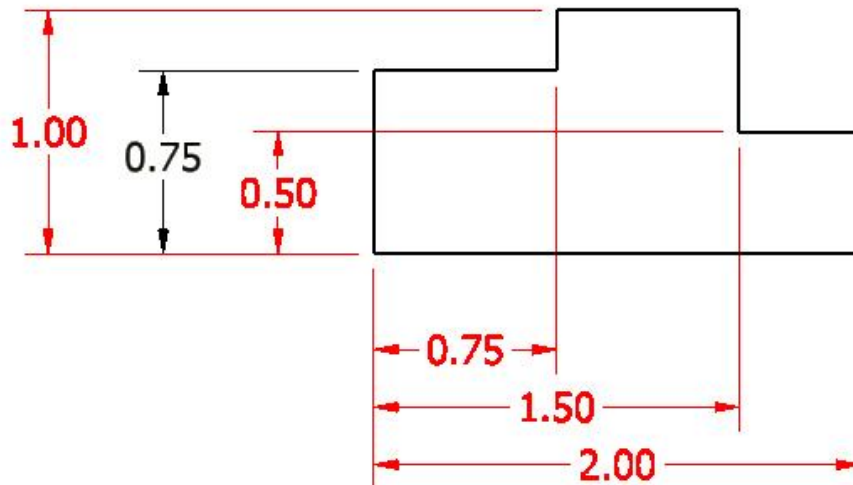
Correct



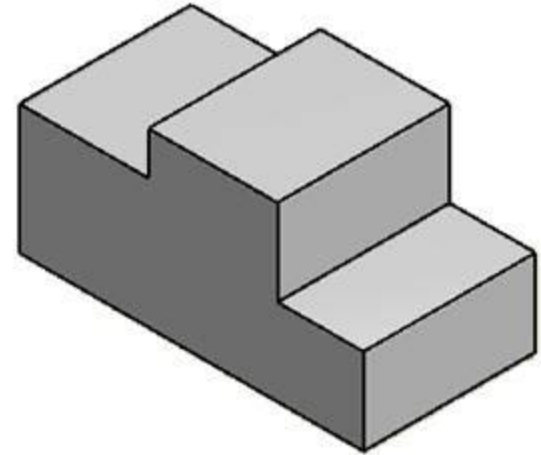
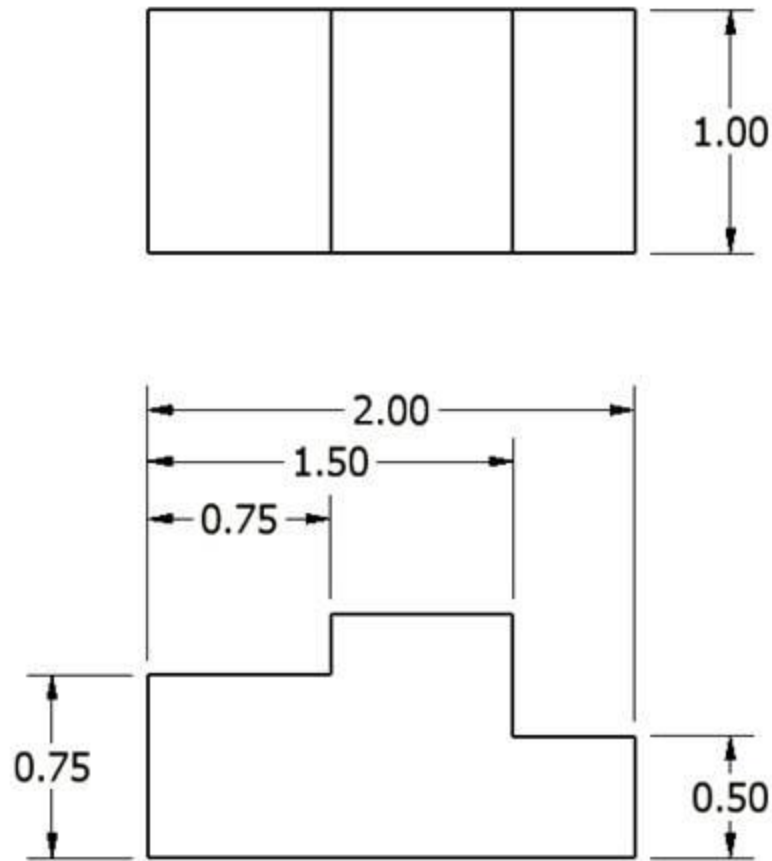
7. Whenever possible, locate dimensions between adjacent views.



Incorrect

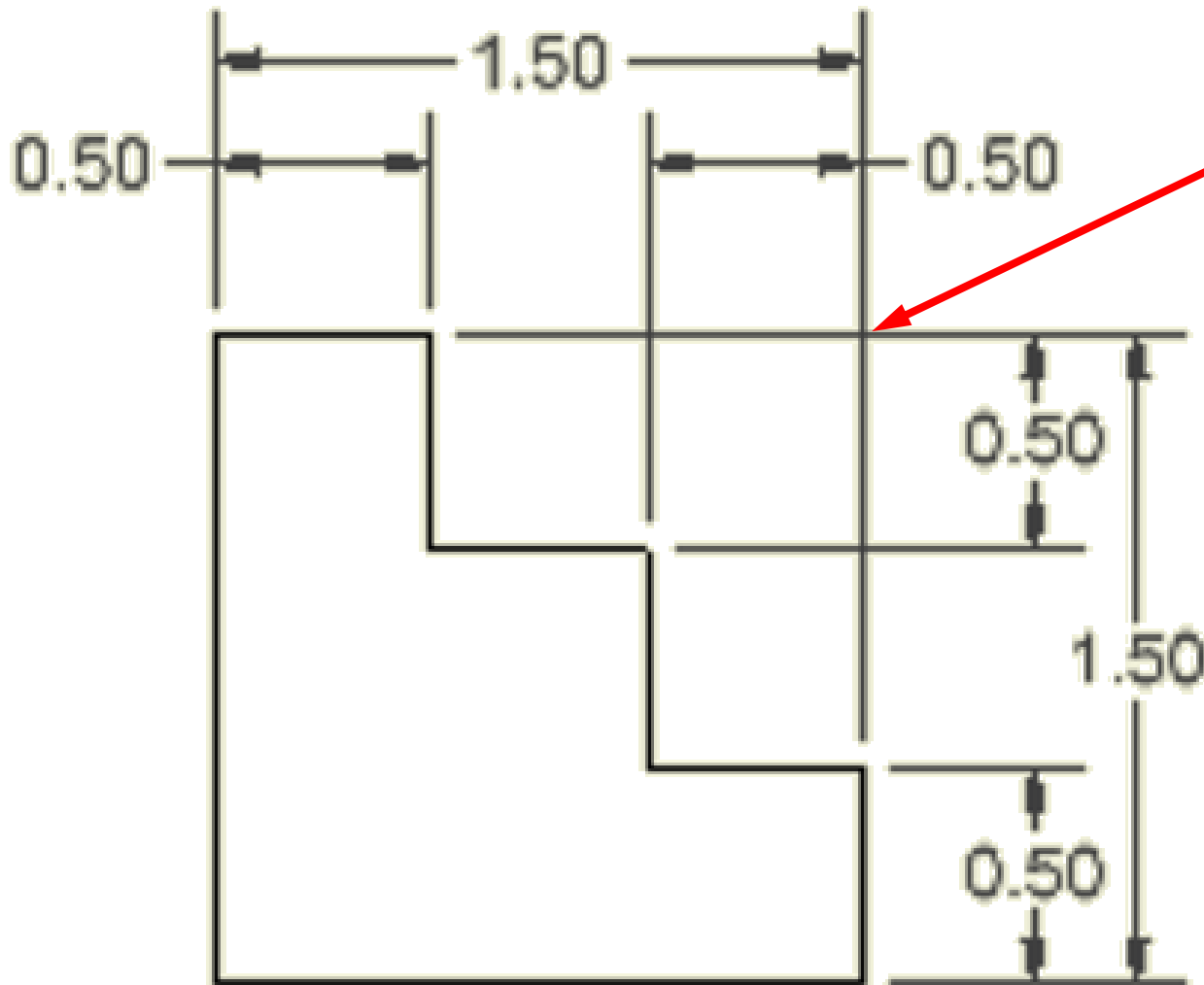


7. Whenever possible, locate dimensions between adjacent views.



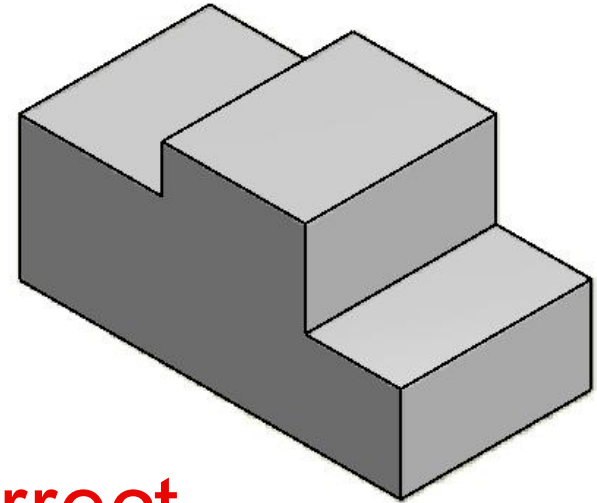
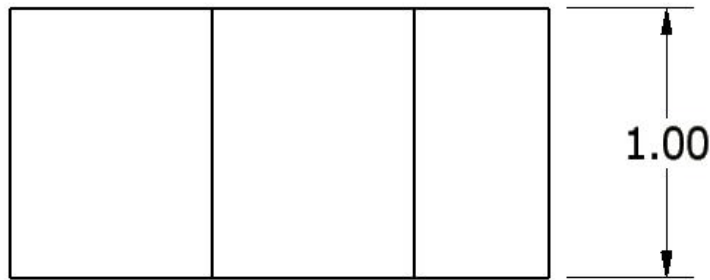
Correct

8. Avoid crossing extension lines, but do not break them when they do cross.

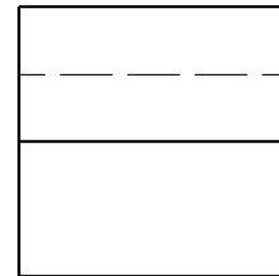
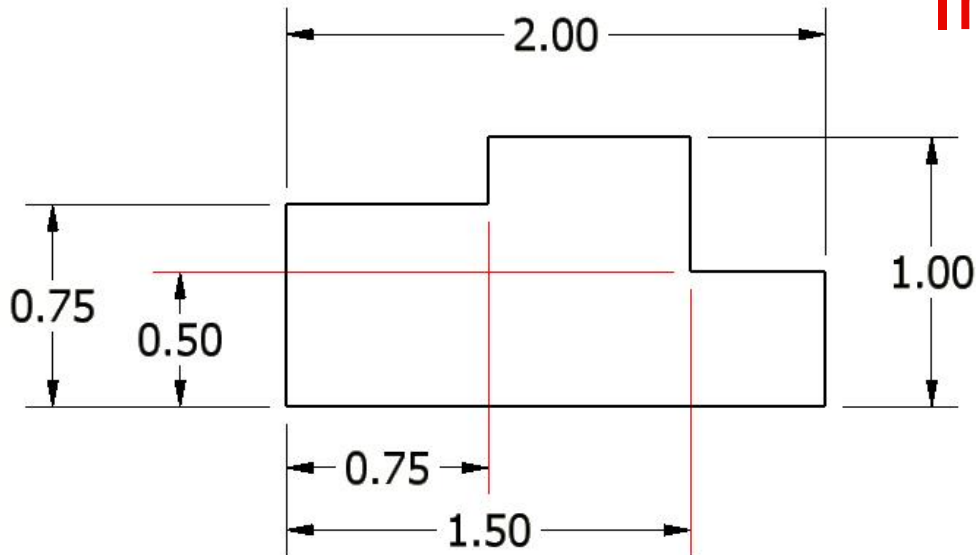


Multiple extension line crossings may be confused for the outside corner of the part.

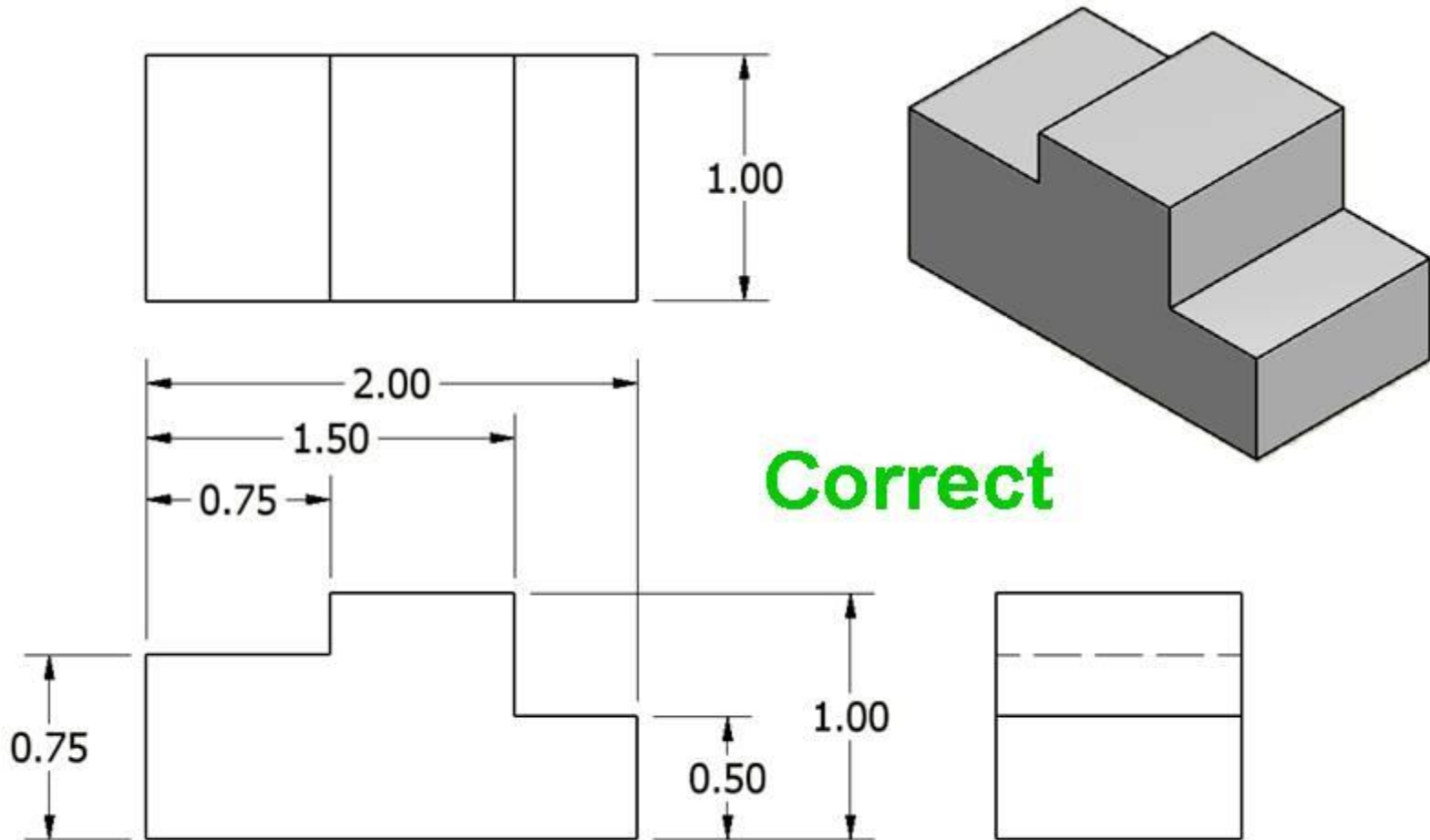
9. Whenever possible, avoid sending extension lines through object views.



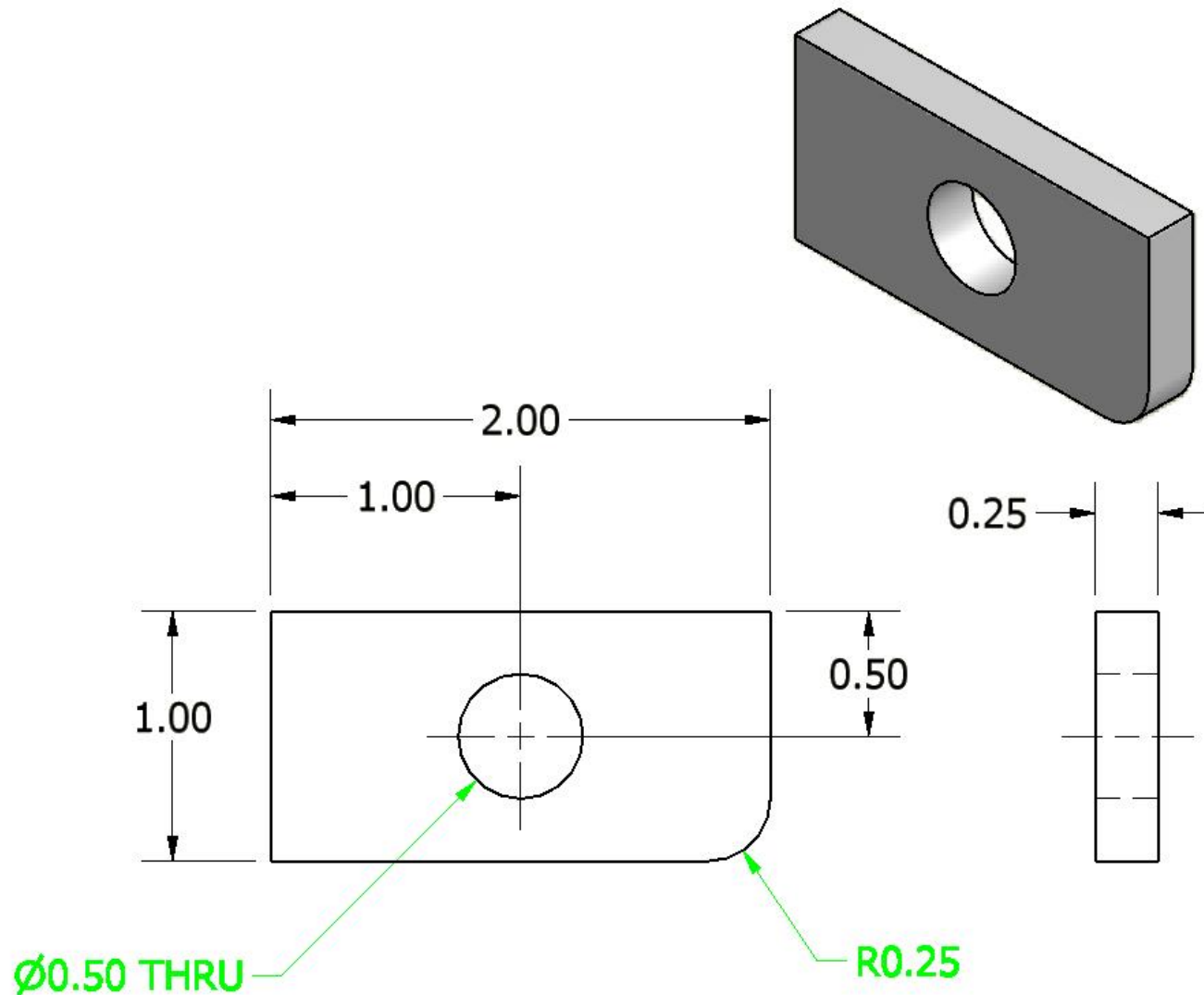
Incorrect



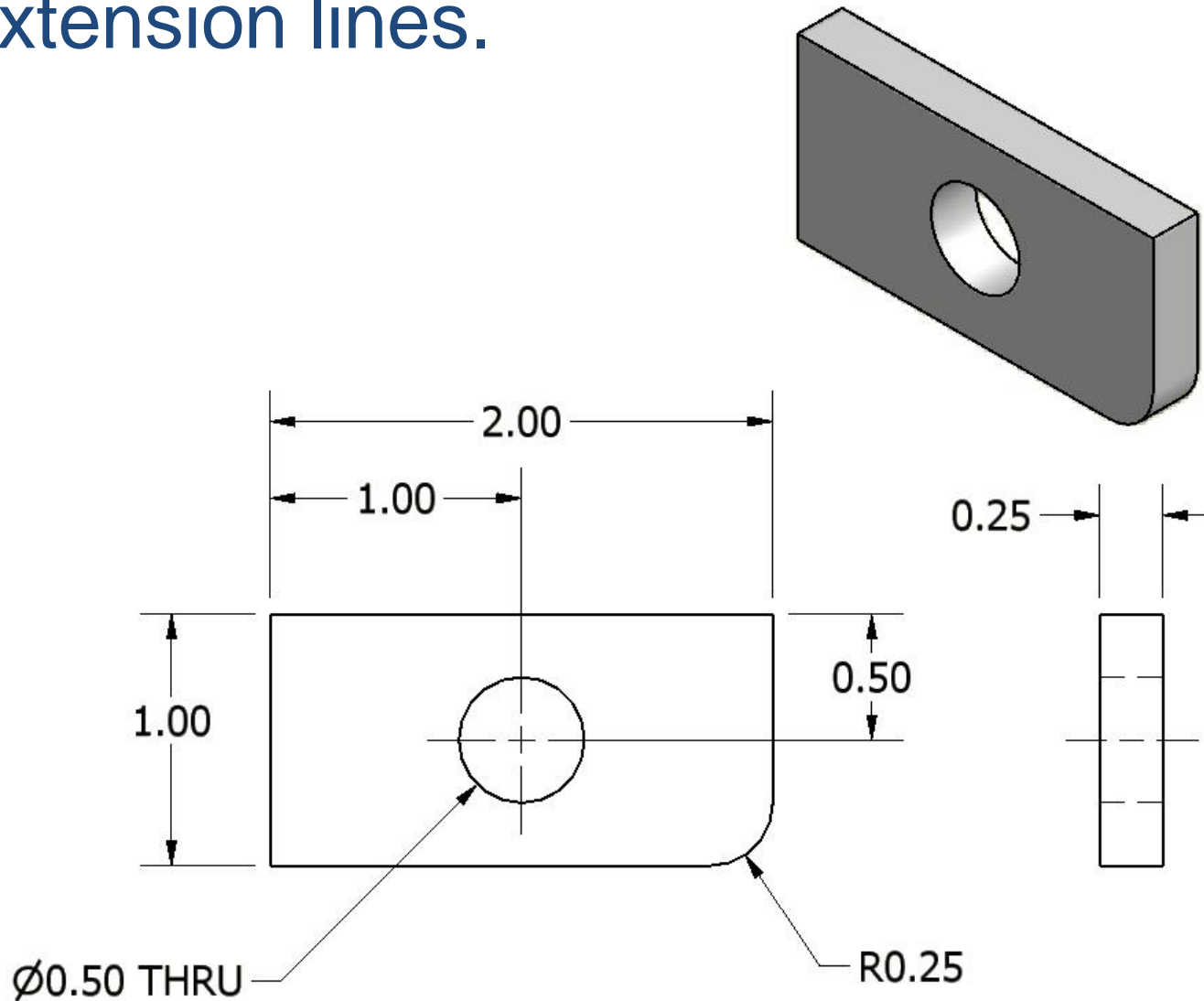
9. Whenever possible, avoid sending extension lines through object views.



10. In general, a circle is dimensioned by its diameter and an arc by its radius.

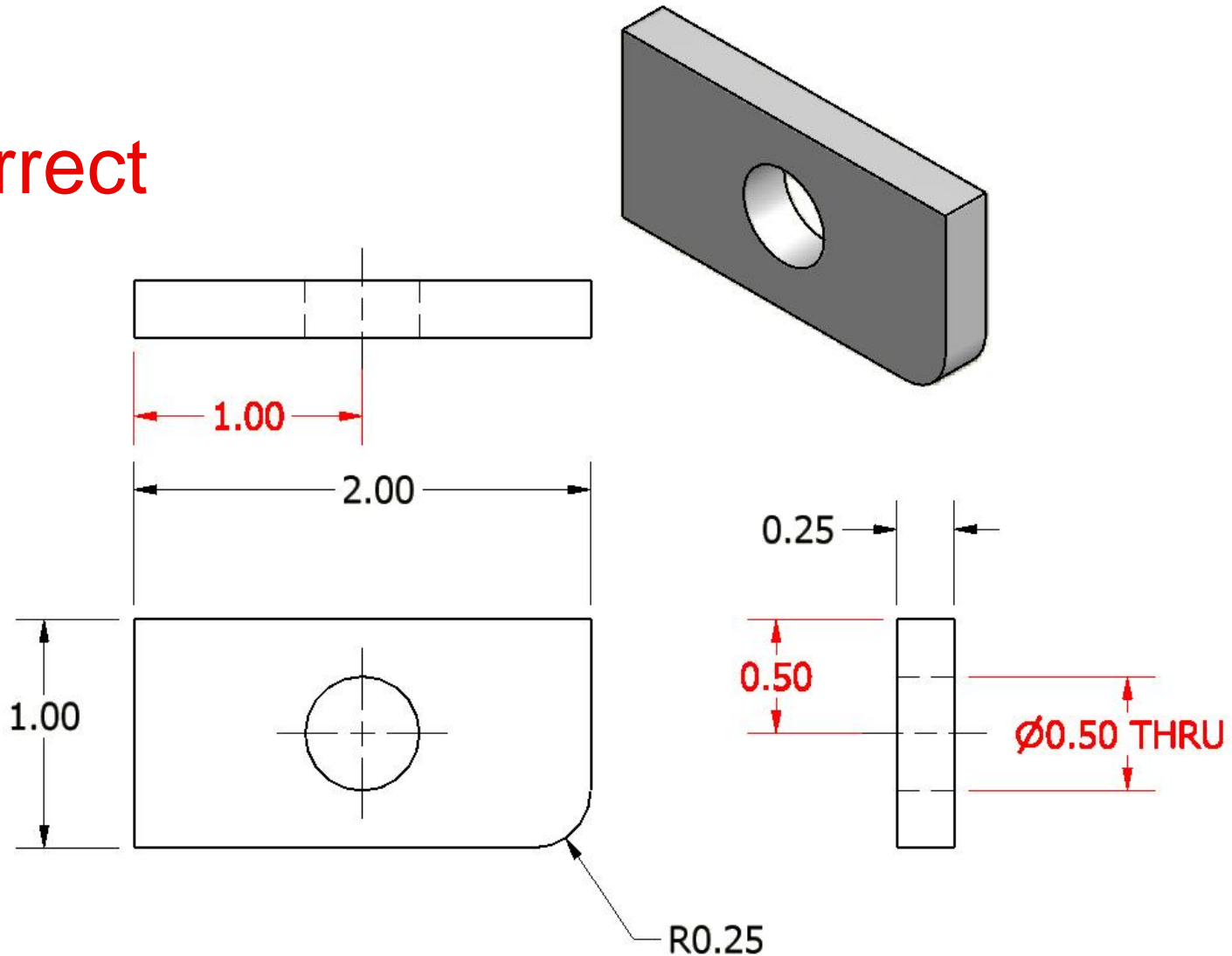


11. Holes are located by their centerlines, which may be extended and used as extension lines.



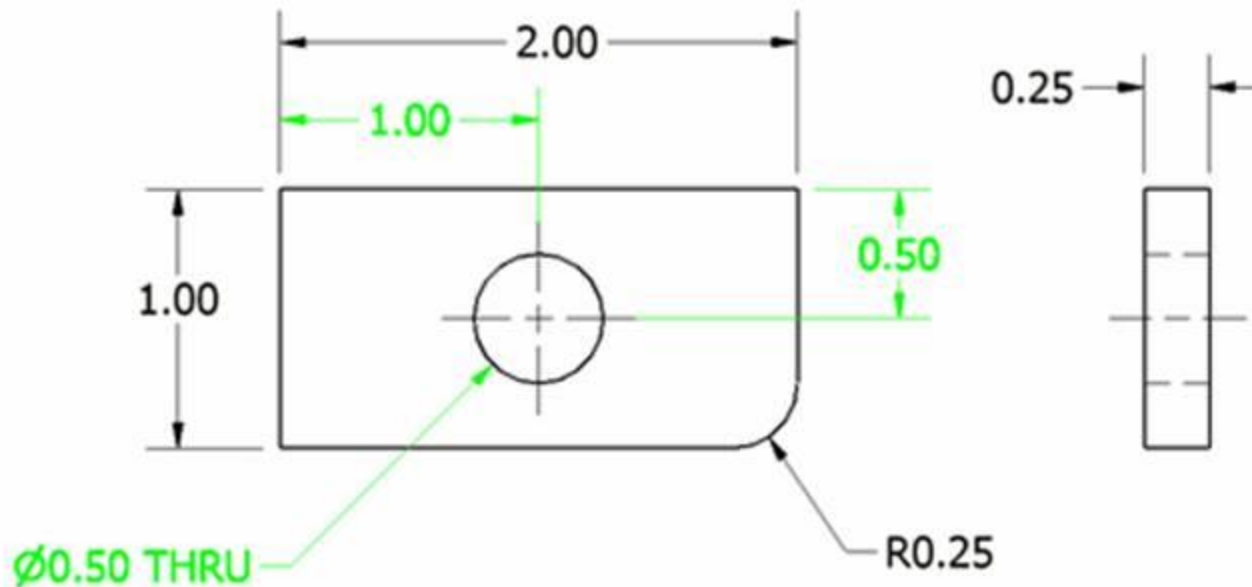
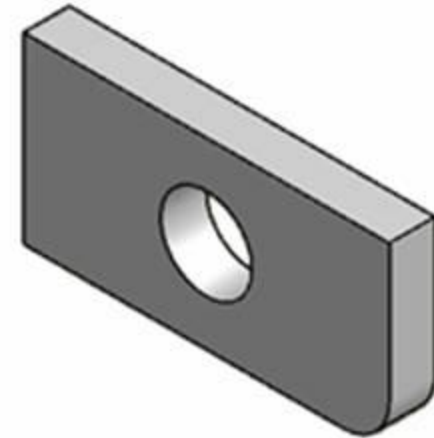
12. Holes should be located and sized in the view that shows the feature as a circle.

Incorrect



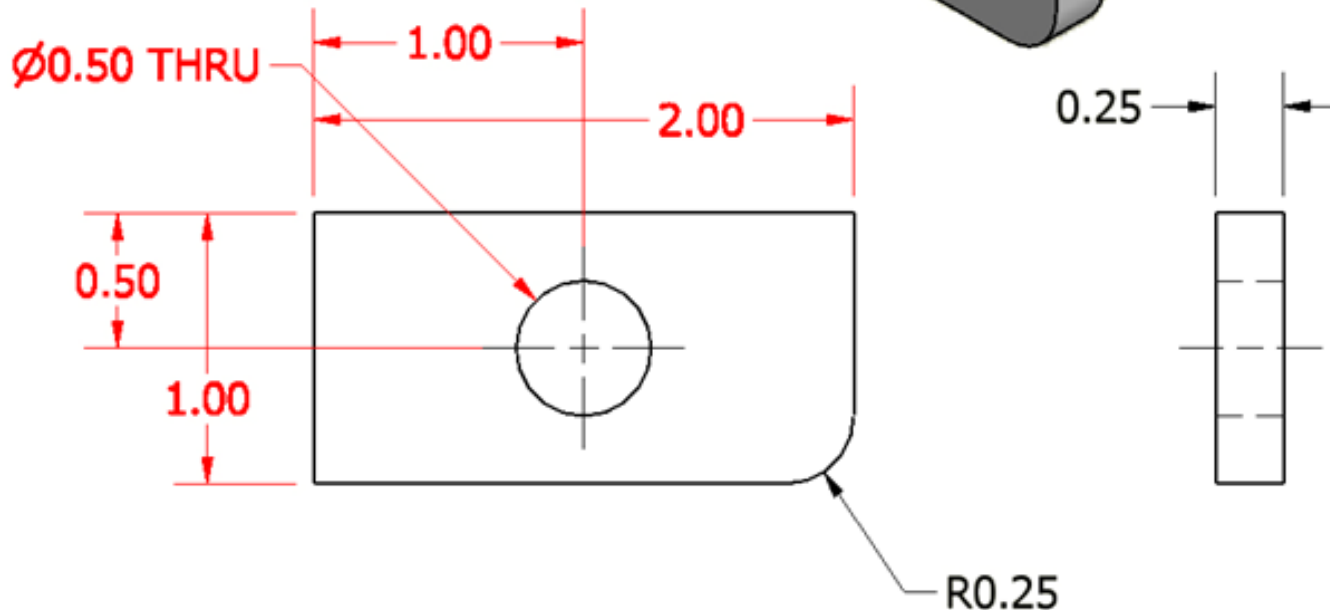
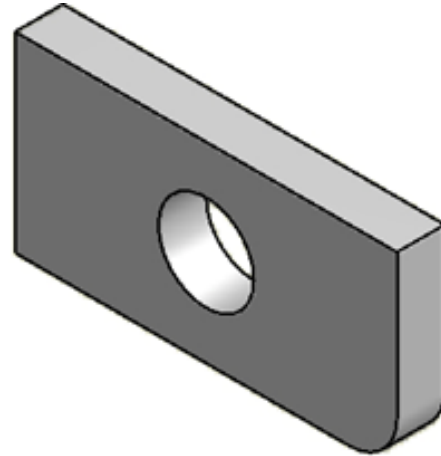
12. Holes should be located and sized in the view that shows the feature as a circle.

Correct



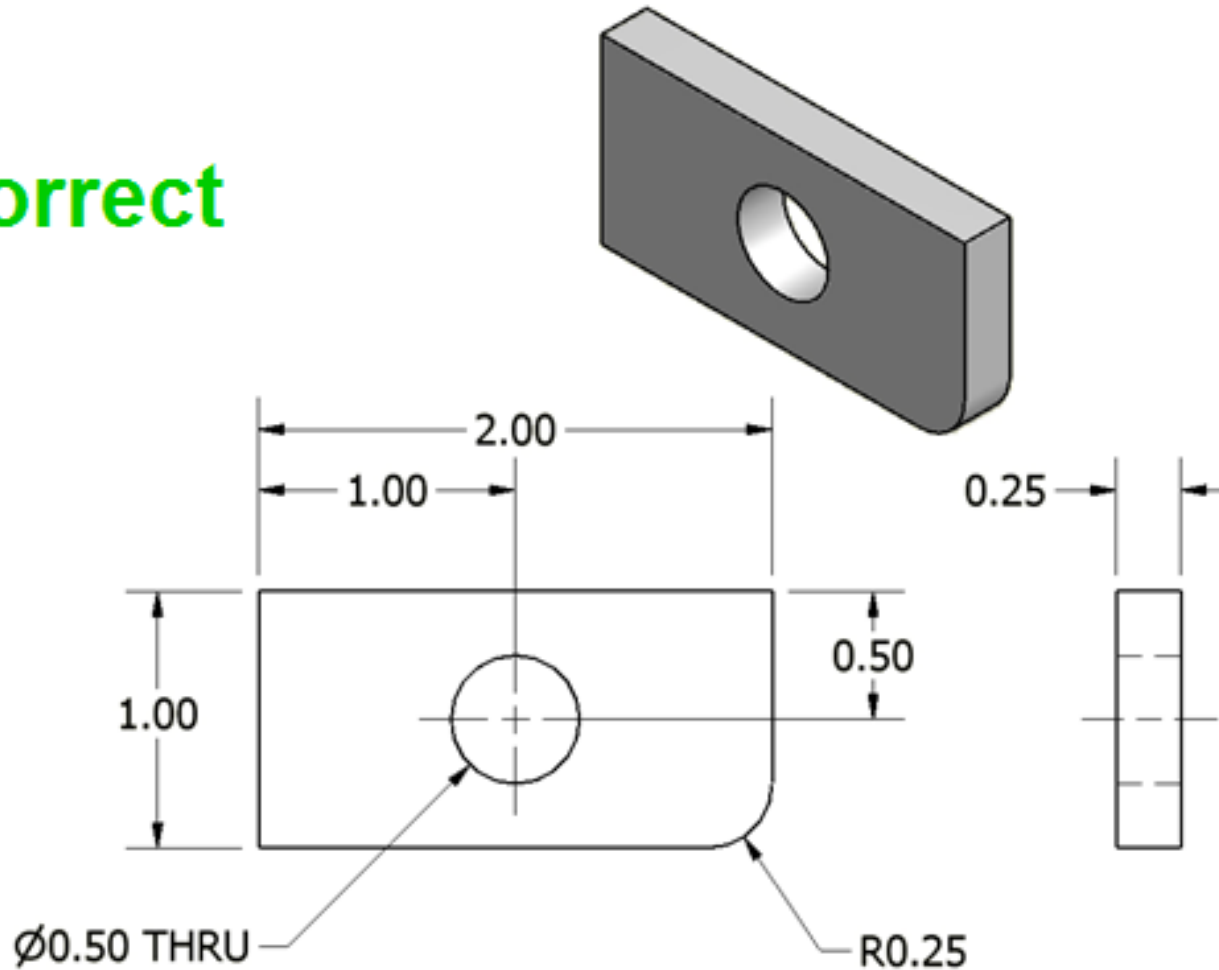
13. Do not cross a dimension line with an extension line, and avoid crossing dimensions with leader lines.

Incorrect



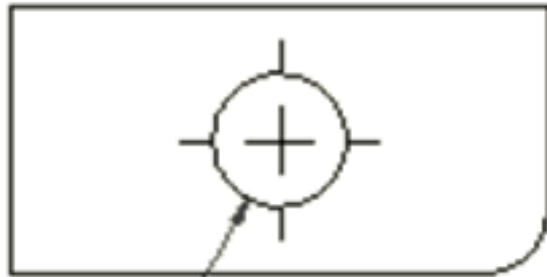
13. Do not cross a dimension line with an extension line, and avoid crossing dimensions with leader lines.

Correct



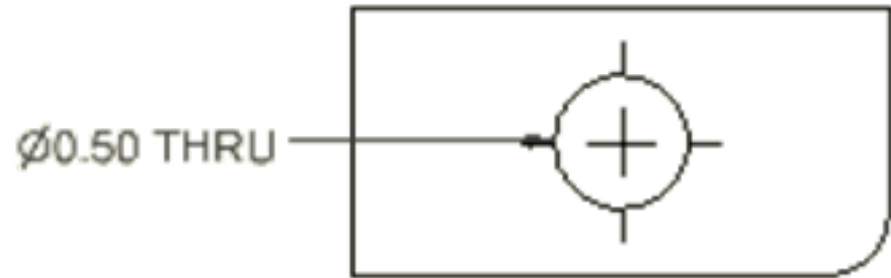
14. Leader lines point toward the center of the feature, and should not occur horizontally or vertically.

This

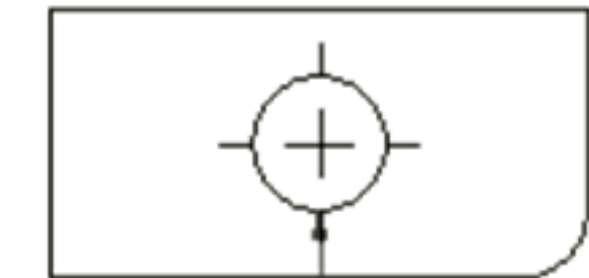


Ø0.50 THRU

Not This



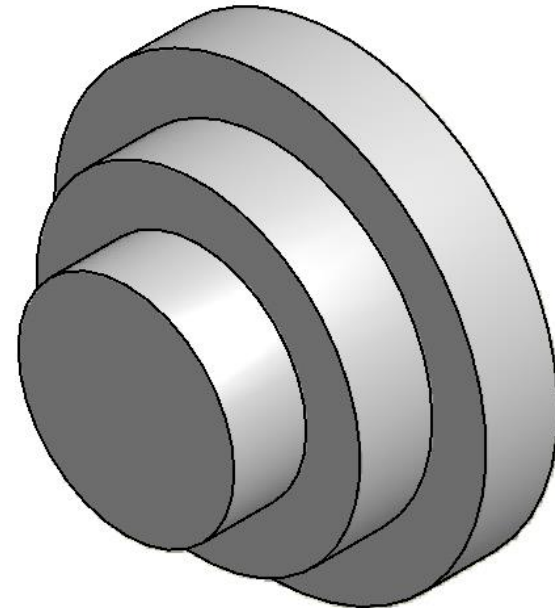
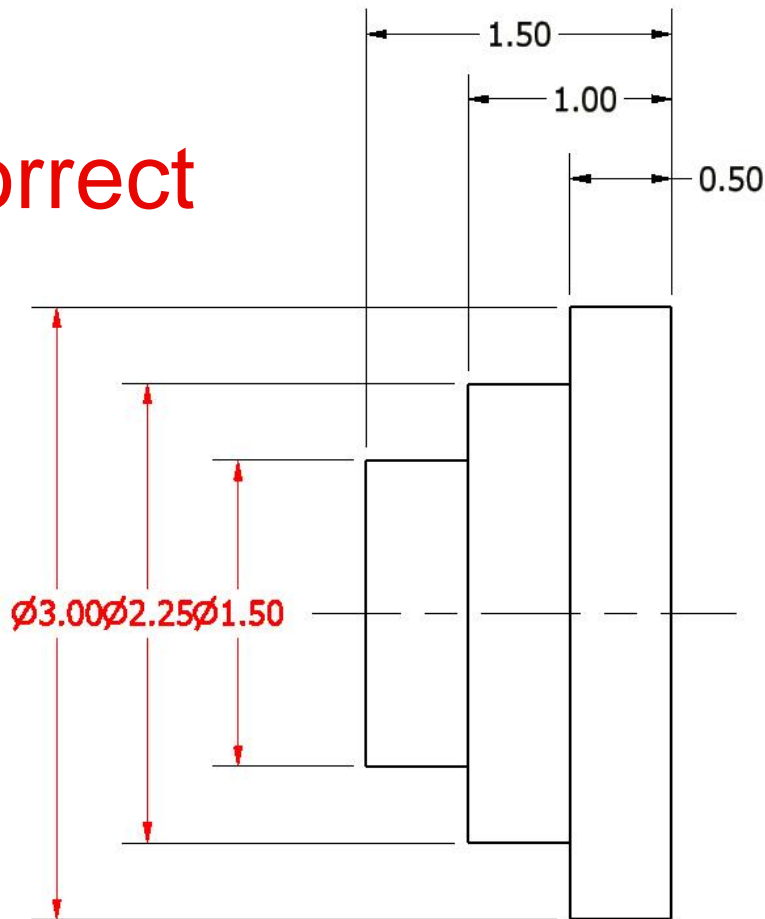
Ø0.50 THRU



Ø0.50 THRU

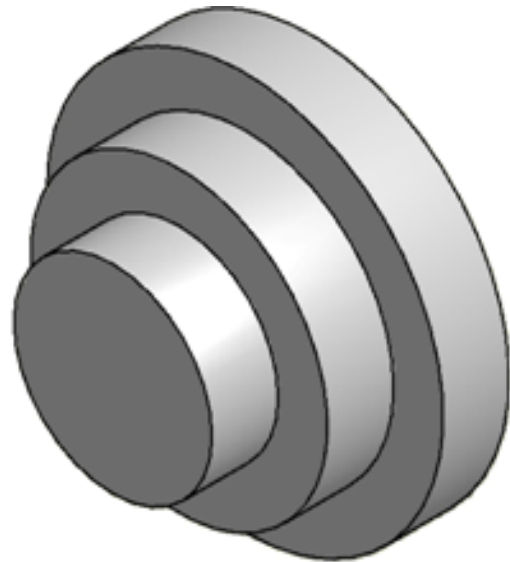
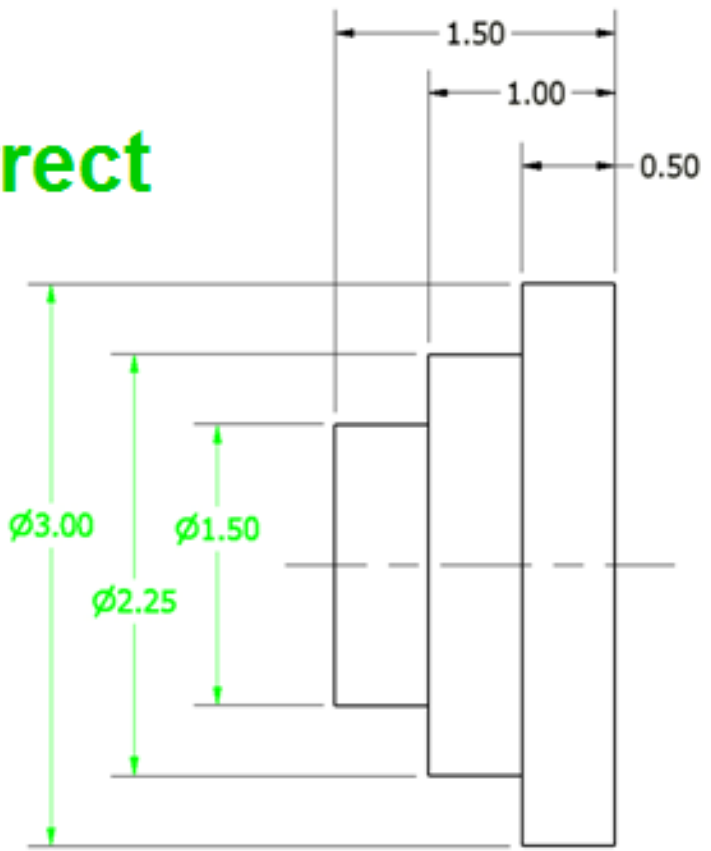
15. Dimension numbers should be centered between arrowheads, except when using stacked dimensions, and then the numbers should be staggered.

Incorrect



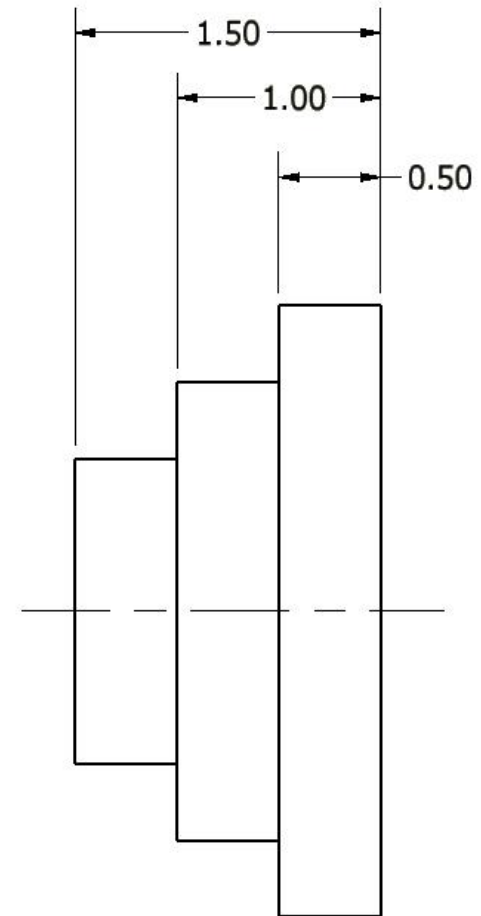
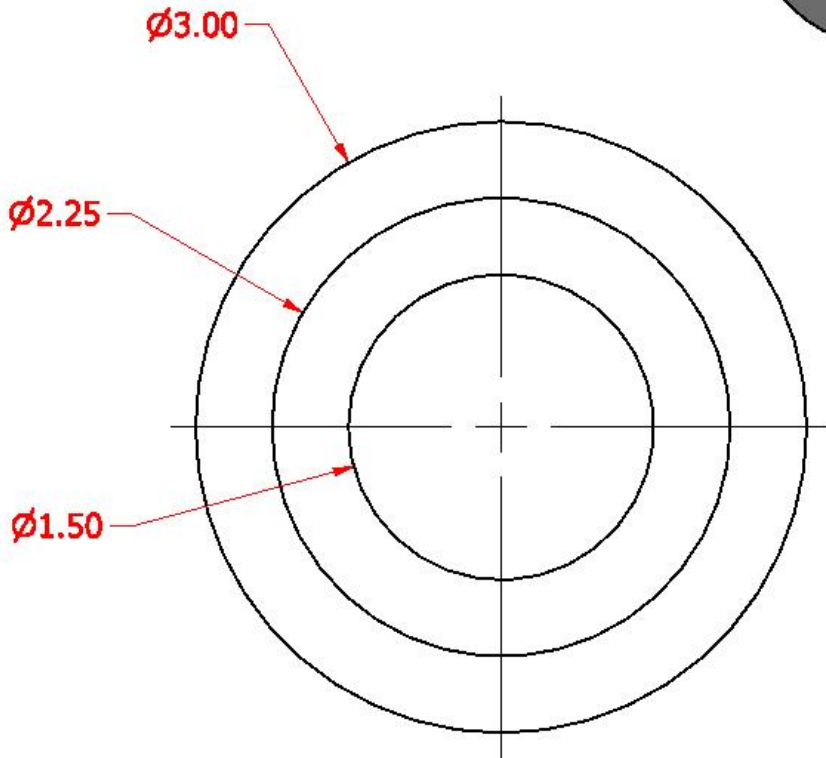
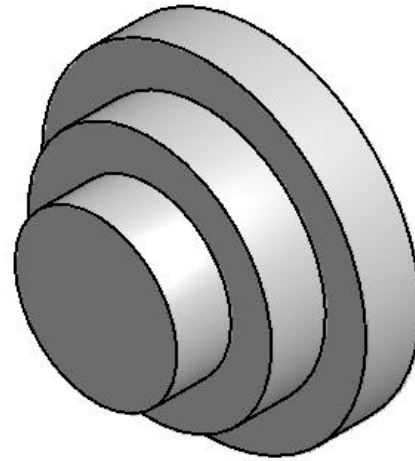
15. Dimension numbers should be centered between arrowheads, except when using stacked dimensions, and then the numbers should be staggered.

Correct



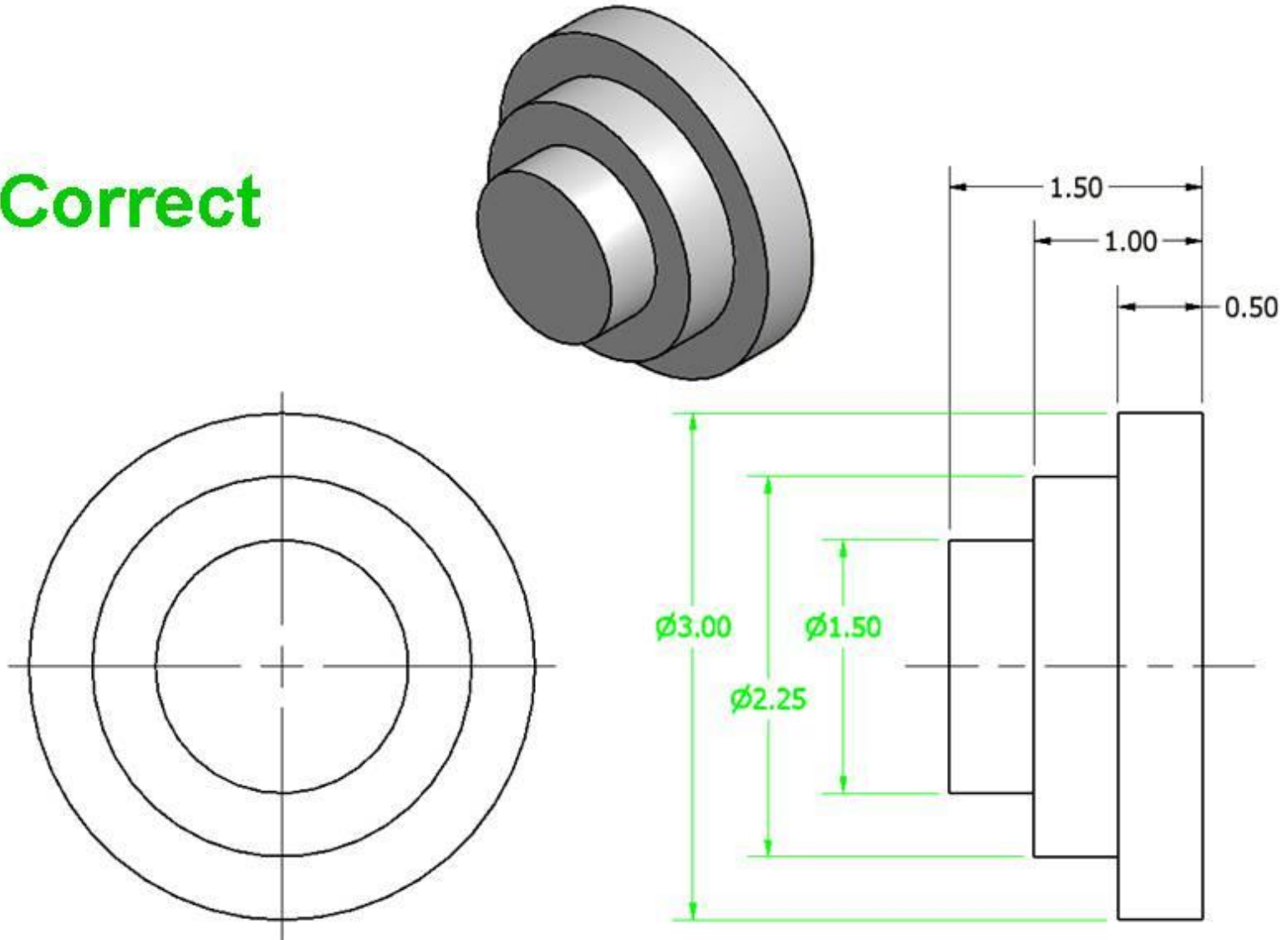
16. Concentric circles are dimensioned in the longitudinal view, whenever practical.

Incorrect



16. Concentric circles are dimensioned in the longitudinal view, whenever practical.

Correct



Tolerances

Variation is Unavoidable

- No two manufactured objects are identical in every way. Some degree of *variation* will exist.
- Engineers apply *tolerances* to part dimensions to reduce the amount of variation that occurs.

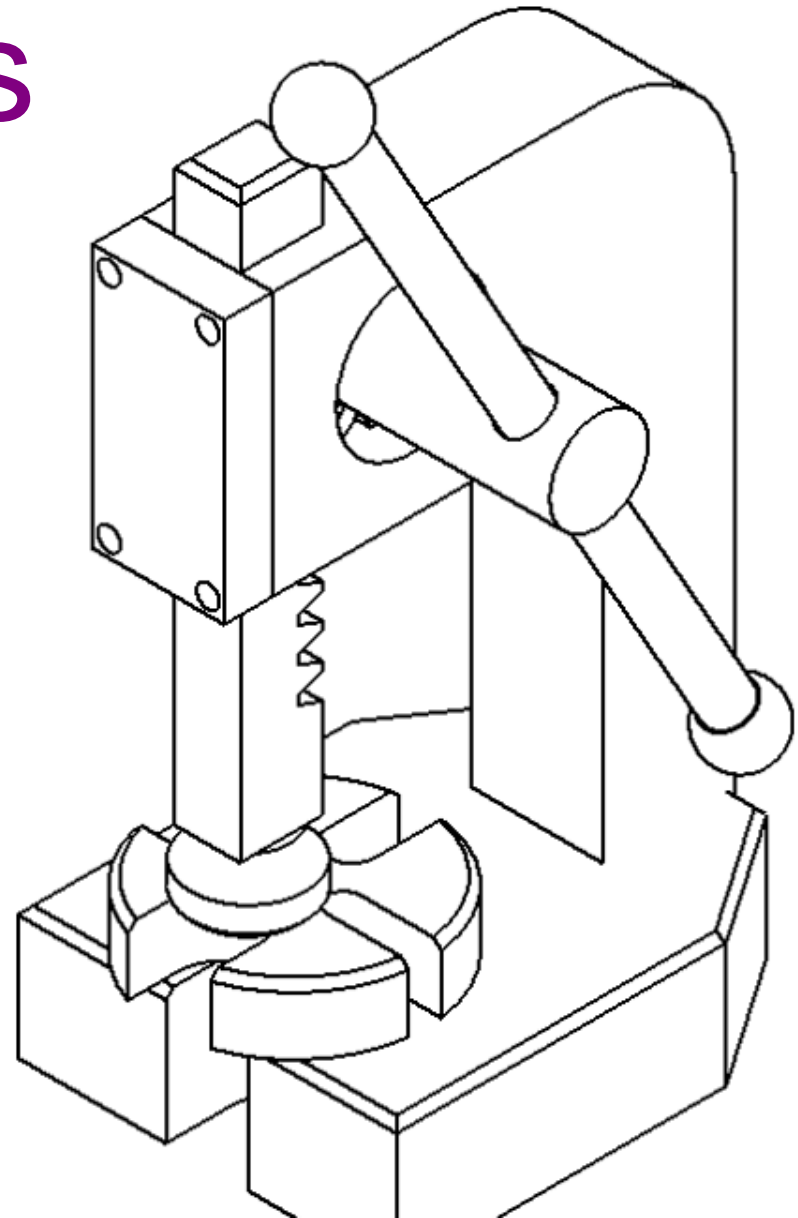
ANSI/ASME Standard

ANSI/ASME Standard Y14.5

Each dimension shall have a tolerance, except those dimensions specifically identified as reference, maximum, minimum, or stock. The tolerance may be applied directly to the dimension or indicated by a general note located in the title block of the drawing.

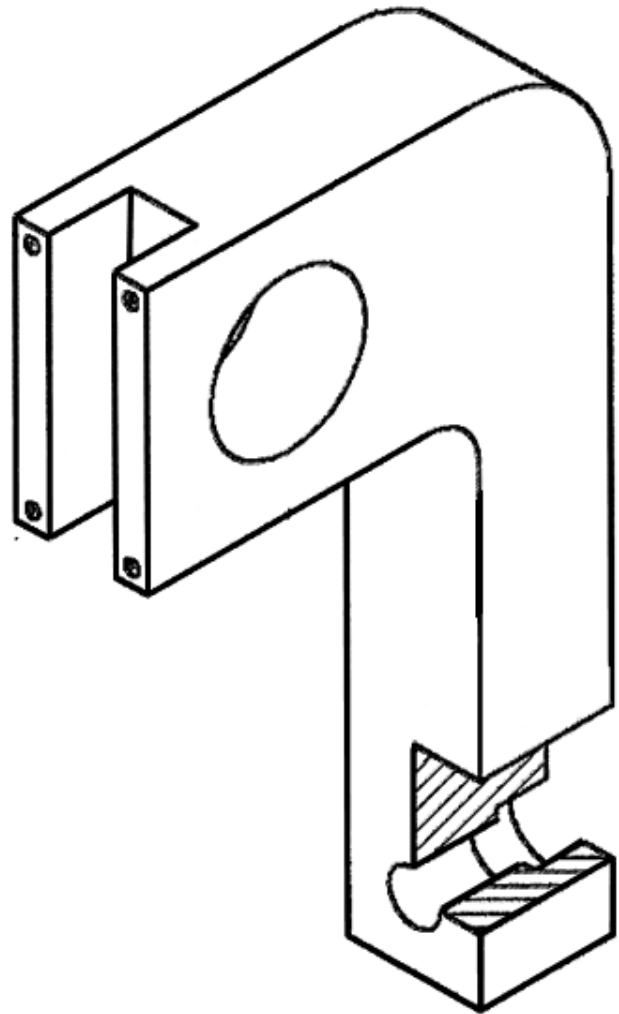
Tolerances

A *tolerance* is an acceptable amount of dimensional variation that will still allow an object to function correctly.



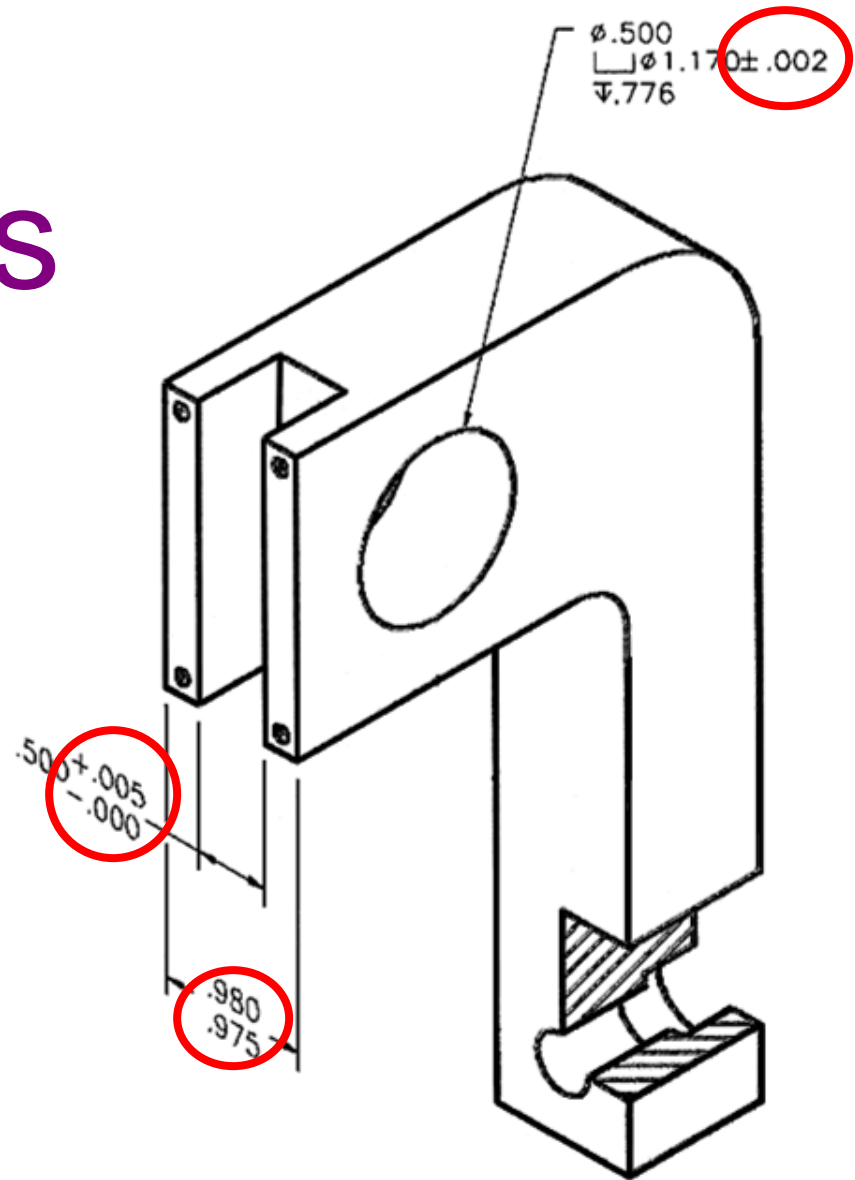
Tolerances

A *tolerance* is an acceptable amount of dimensional variation that will still allow an object to function correctly.



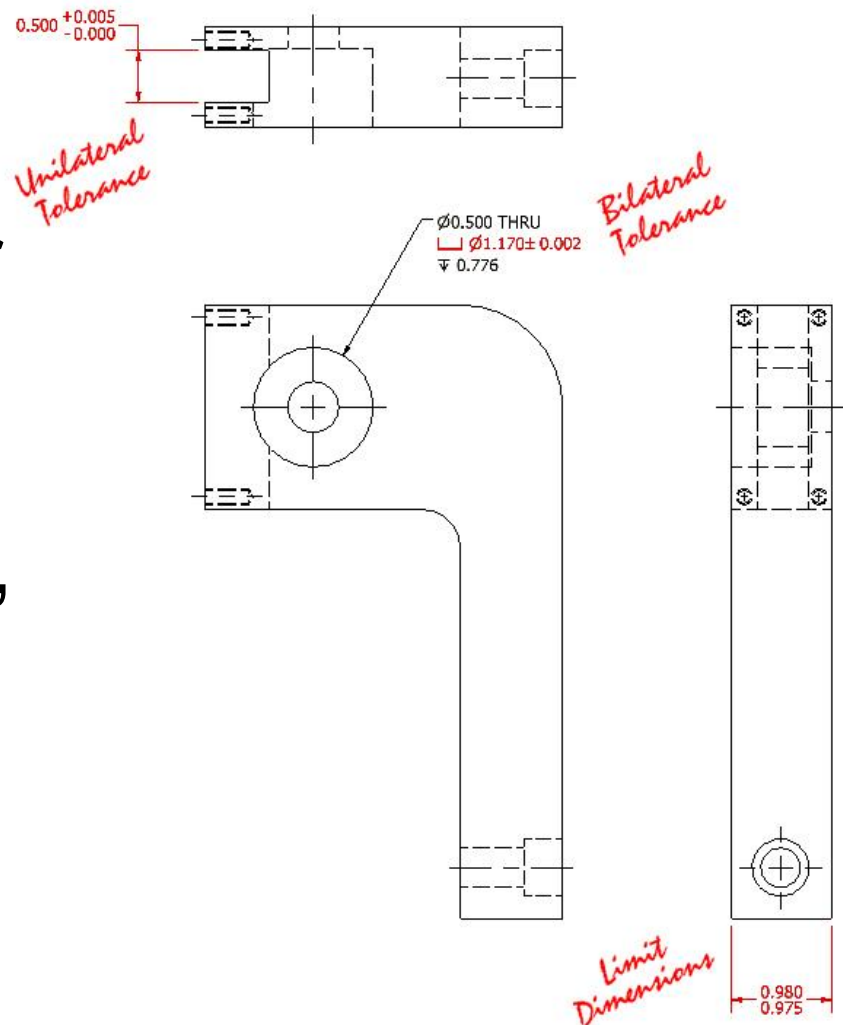
Tolerances

Three basic tolerances that occur most often on working drawings are: *limit dimensions*, *unilateral*, and *bilateral tolerances*.



Tolerances

Three basic tolerances that occur most often on working drawings are: *limit dimensions*, *unilateral*, and *bilateral tolerances*.

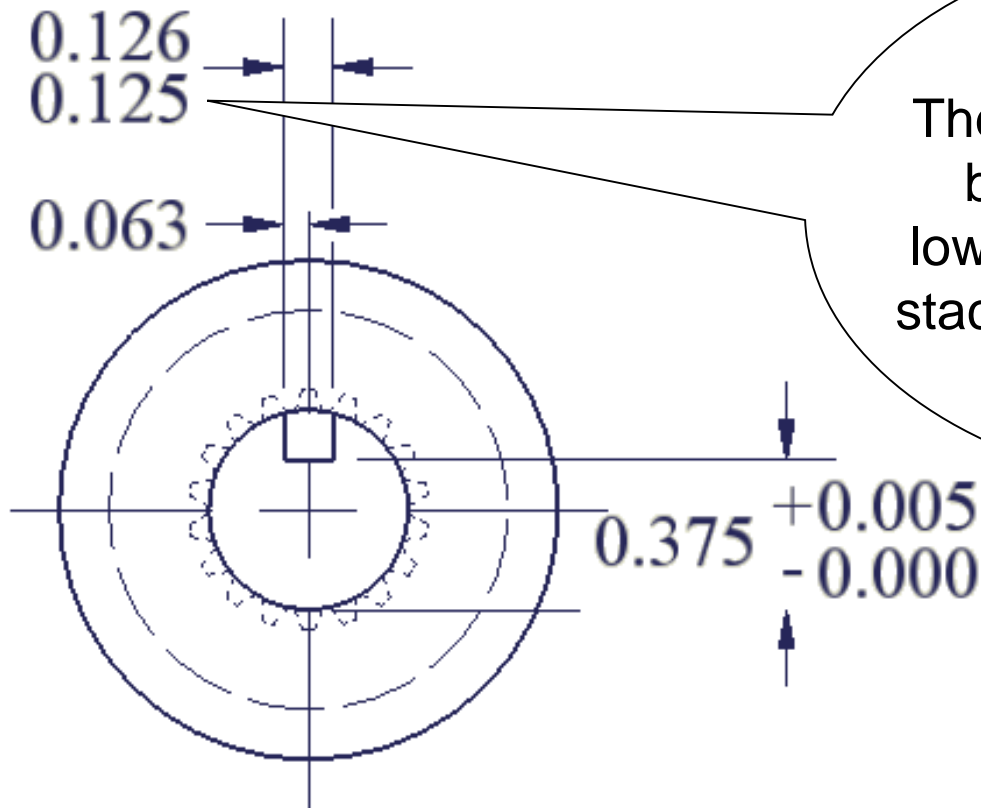


Limit Dimensions

Limit dimensions are two dimensional values stacked on top of each other. The dimensions show the largest and smallest values allowed. Anything in between these values is acceptable.



Limit Dimensions



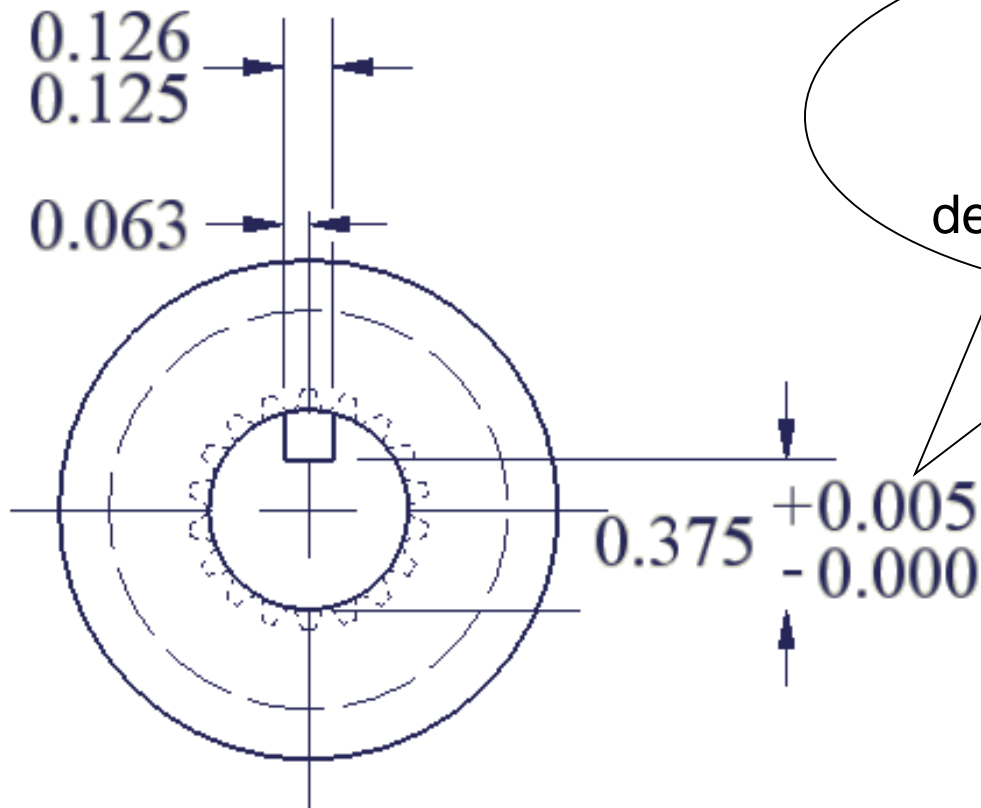
These are limit dimensions, because the upper and lower dimensional sizes are stacked on top of each other.

Unilateral Tolerance

A *unilateral tolerance* exists when a target dimension is given along with a tolerance that allows variation to occur in only one direction.

$500 \pm .005$
 $500 \begin{matrix} / \\ .000 \end{matrix}$

Unilateral Tolerance



This tolerance is unilateral, because the size may only deviate in one direction.

Bilateral Tolerance

A *bilateral tolerance* exists if the variation from a target dimension is shown occurring in both the positive and negative directions.

$\varnothing .500$
 $\sqcup \varnothing 1.170 \pm .002$
 $\nabla .776$

General Tolerances

If no tolerances are specified at the dimension level, then general tolerances may be applied by deliberately controlling the number of values past the decimal point on each dimension.

Angles = $\pm .5^\circ$

Linear Dimensions

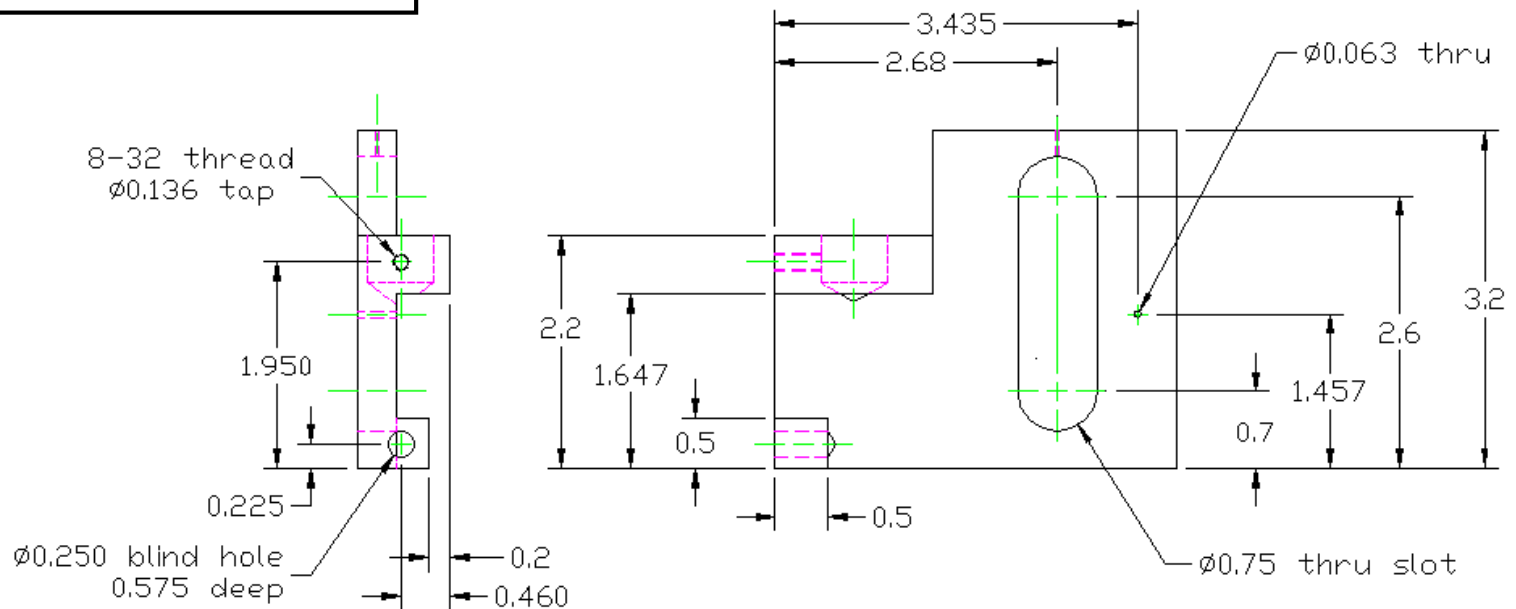
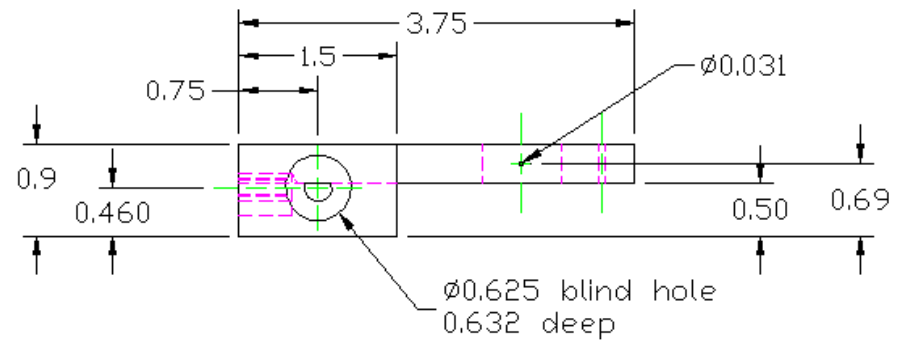
X.X = $\pm .020$

X.XX = $\pm .010$

X.XXX = $\pm .005$

General Tolerances

Tolerances	
X.X	= ± .020
X.XX	= ± .010
X.XXX	= ± .005

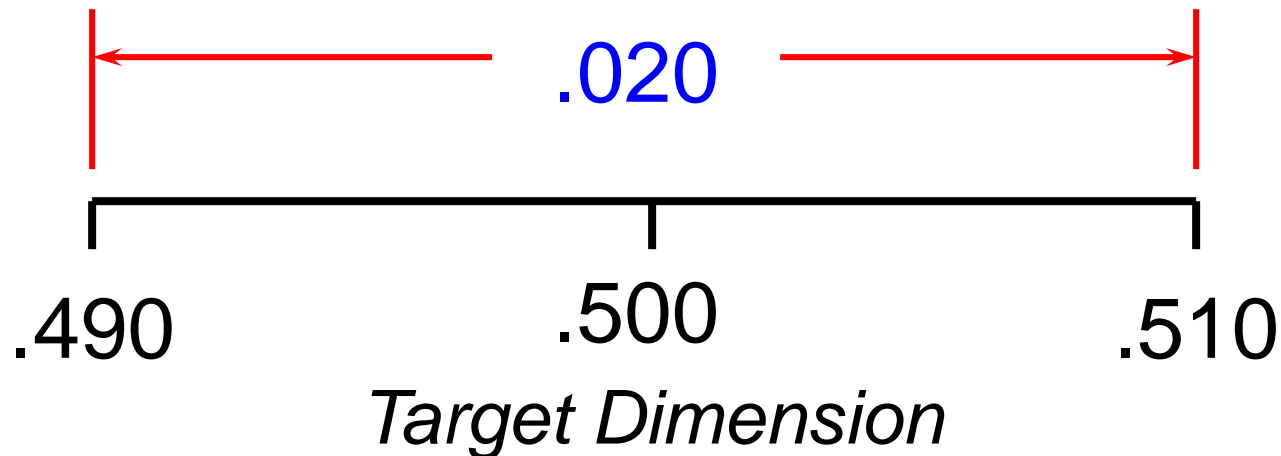


Total Tolerance

The total tolerance is a value that describes the maximum amount of variation.

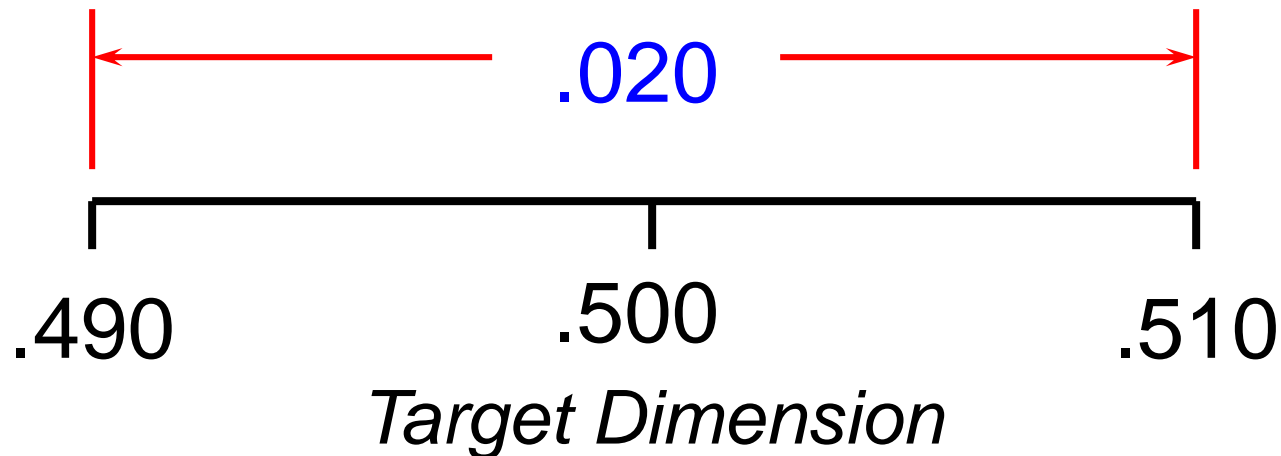
$$\text{Tolerance} = \pm .010$$

$$\text{Total Tolerance} = .020$$



Total Tolerance

A measuring device should be able to accurately measure within $1/10^{\text{th}}$ of the total blueprint tolerance identified.



Tolerances and Measuring

In this case, a measuring device should be able to take accurate measurements to within two thousandths of an inch.

$$\textit{Total tolerance} = .020$$

$$\frac{1}{10} = \frac{x}{.020}$$

$$10x = .020$$

$$x = .002$$

X = the minimum accuracy of the measuring device

Three Types of Fit

There are three types of fit that should be considered when working with tolerances.

- **Clearance Fit**- have limits of size so prescribed that a clearance always results when mating parts are assembled.
- **Interference Fit**- have limits of size so prescribed that an interference always results when mating parts are assembled.
- **Transition Fit**- have limits of size indicating that either a clearance or an interference may result when mating parts are assembled.