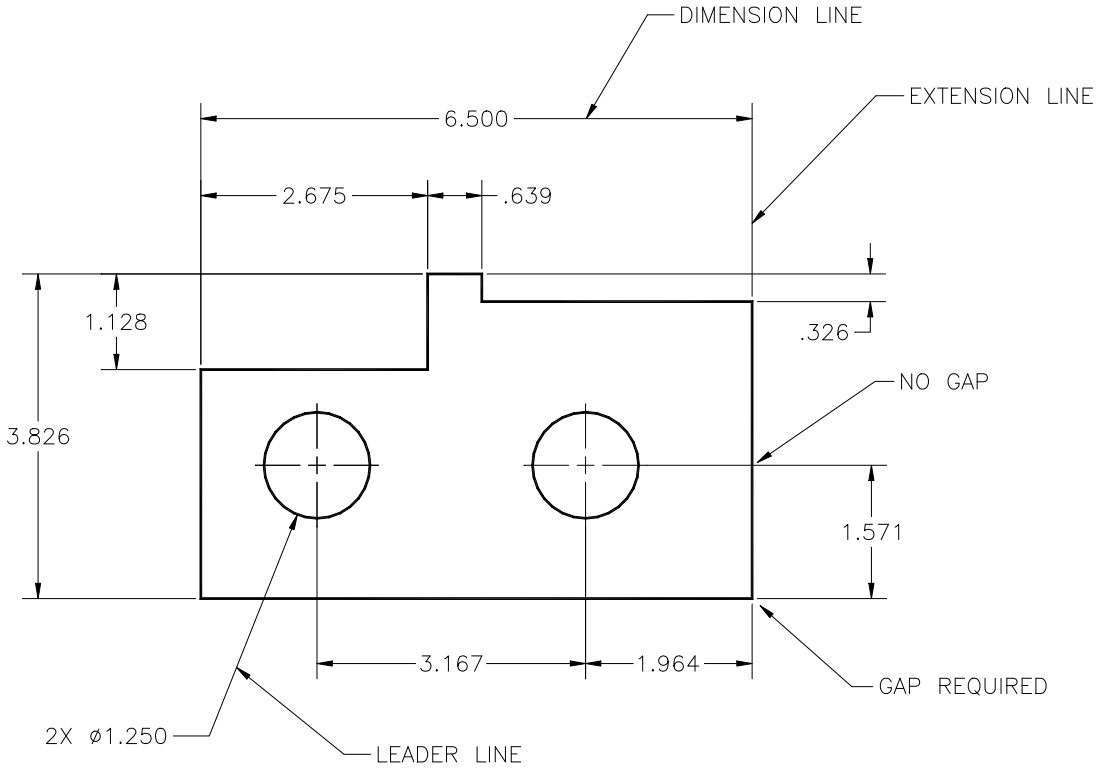
Dimensioning Fundamentals

Terminology

Dimension lines: Lines located between extension lines ending with an arrow and to include a numerical value. They should be spaced uniformly approximately .375 to .500 inches apart.

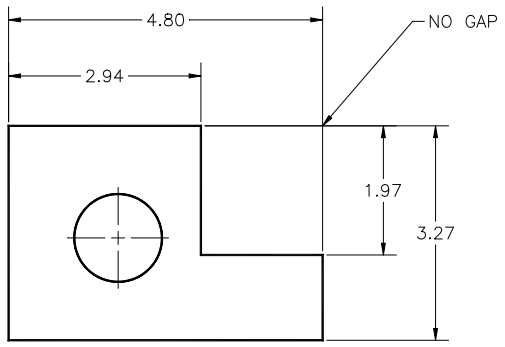
Extension lines: Extend away from a view to indicate a size or location constraint origin. When extension lines cross object or extension lines, no gap in either line should be made.

Leader Lines: Lines drawn at an angle (never horizontal or vertical) extending from a note to a feature to which the note applies.

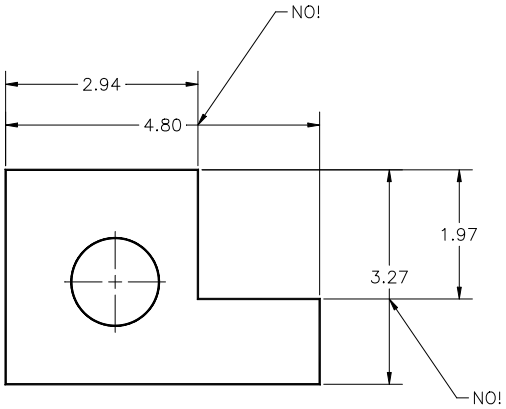


Dimension and Extension line placement

1. Always place shorter dimensions nearest to the object lines. Dimension lines should never cross. However, extension lines may cross each other.

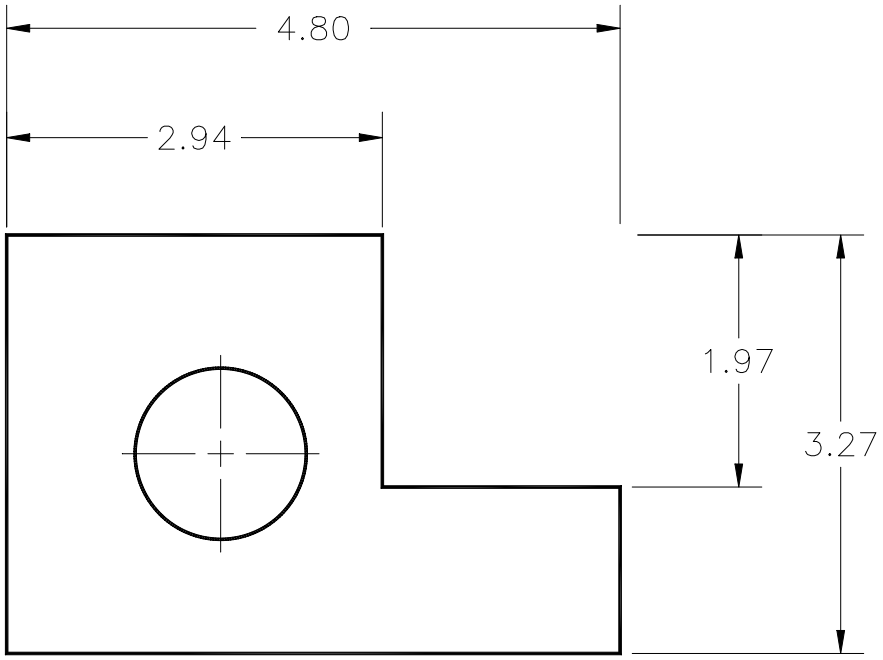


CORRECT



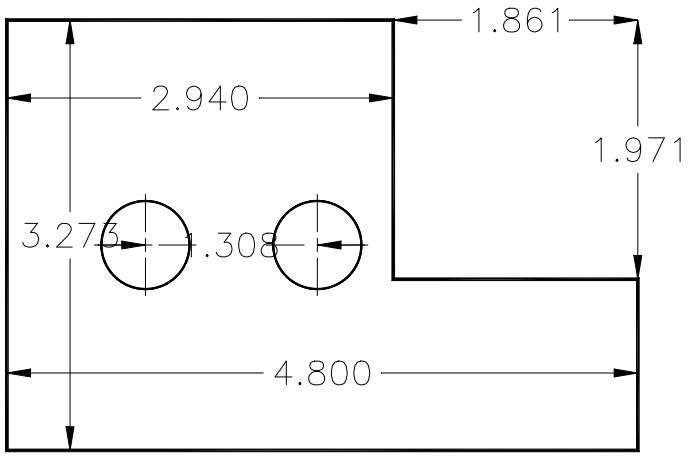
NO!

2. Extension lines should never be shortened.



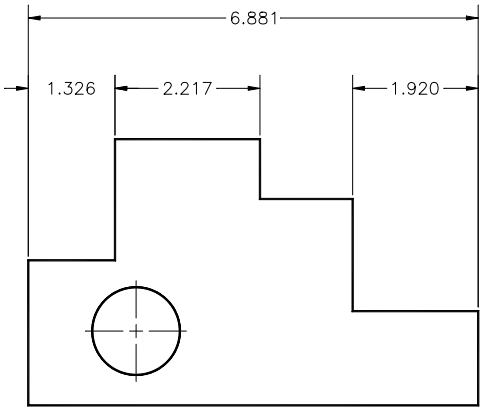
NO!

3. Dimension lines should never coincide with any object line or center line of the drawing. Therefore no line should be used as dimension line nor coincide with a dimension line.

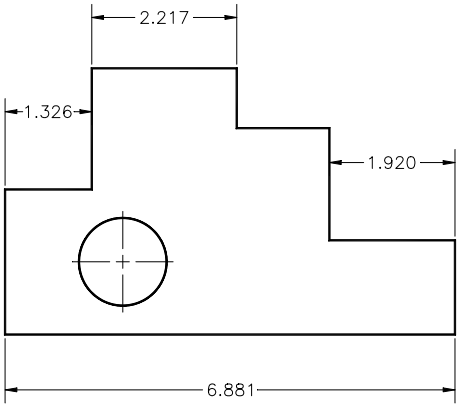


NO!

4. Dimensions should line up in chain fashion or be grouped together as much as possible.



CORRECT



NO!

5. Do not repeat a dimension. By repeating a dimension will only confuse the reader and could cause errors in the manufacturing process. In Figure 5A below, there are many cases where dimensions are duplicated. Figure 5B correctly shows only those dimensions needed to describe the part.

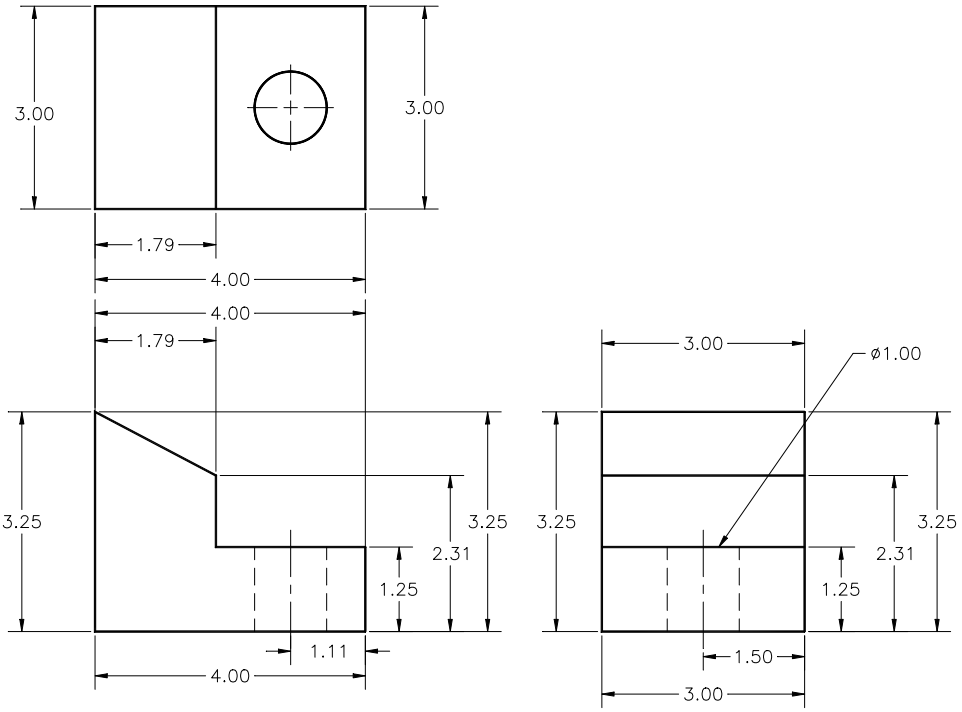


Figure 5A

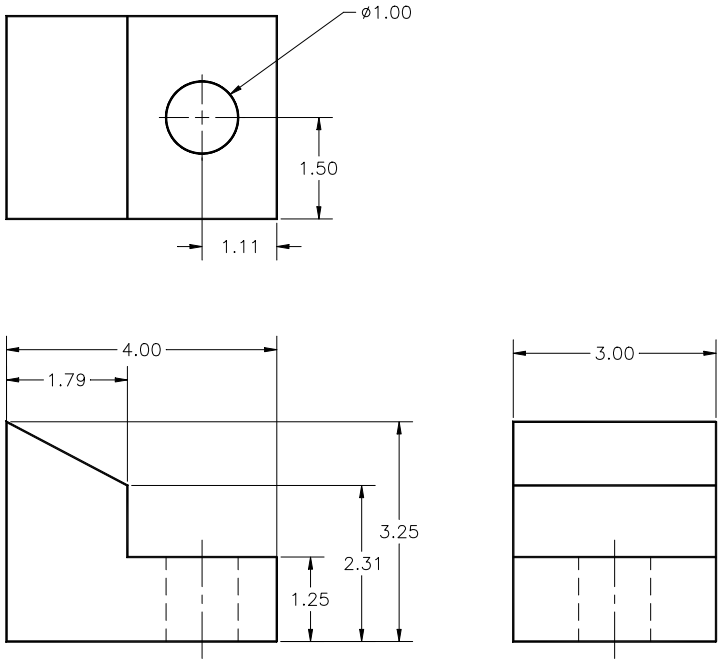


Figure 5B

6. Dimensions should be given in views where the shapes are presented in profile and/or contour. Figure 6A below shows each dimension given in the correct profile view. Figure 6B on the other hand, shows each dimension given in the wrong view.

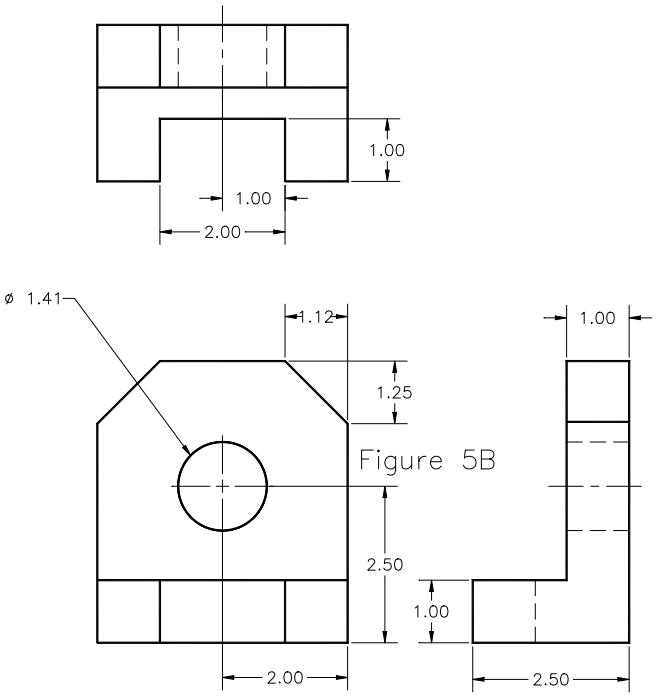


Figure 6A

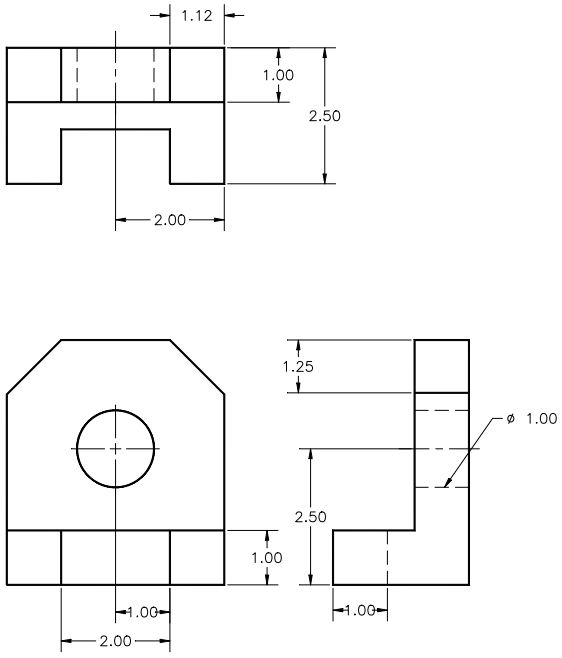
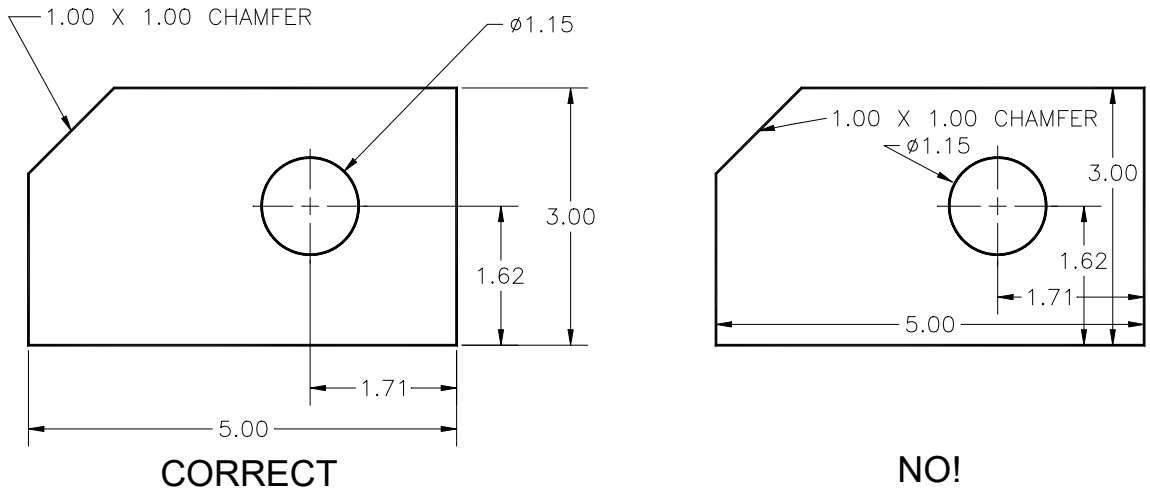
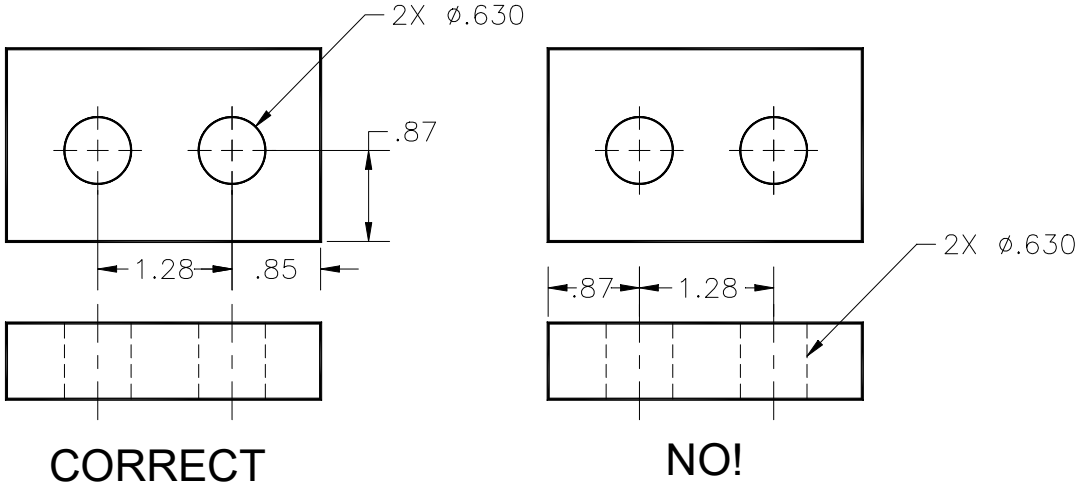


Figure 6B

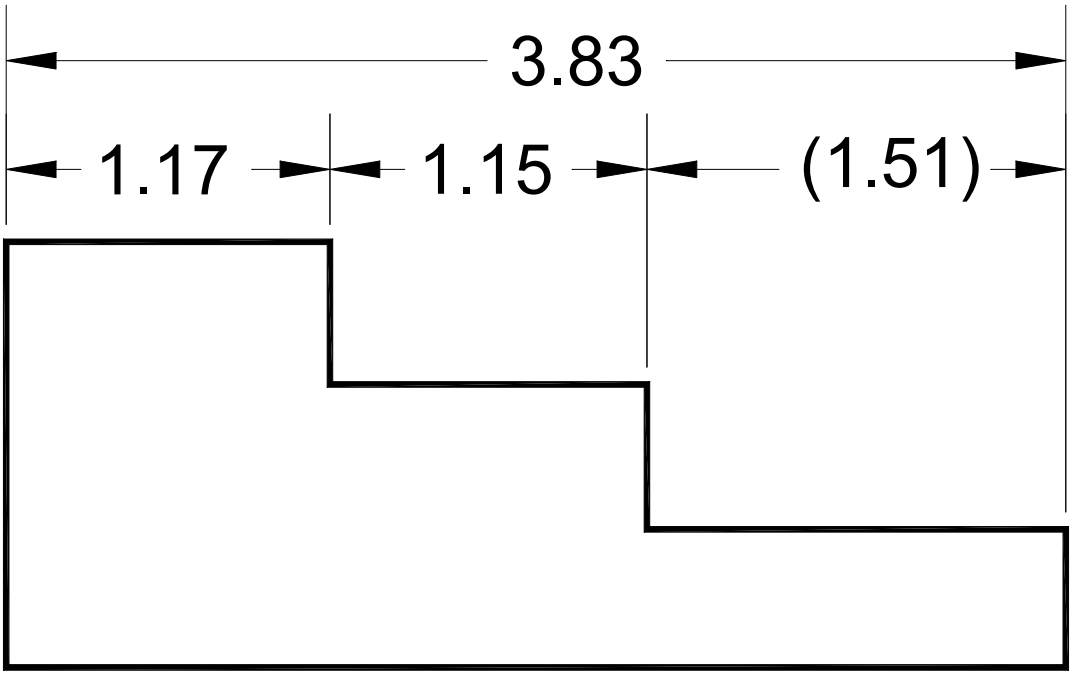
7. Dimensions should always be placed off or outside of a view where possible.



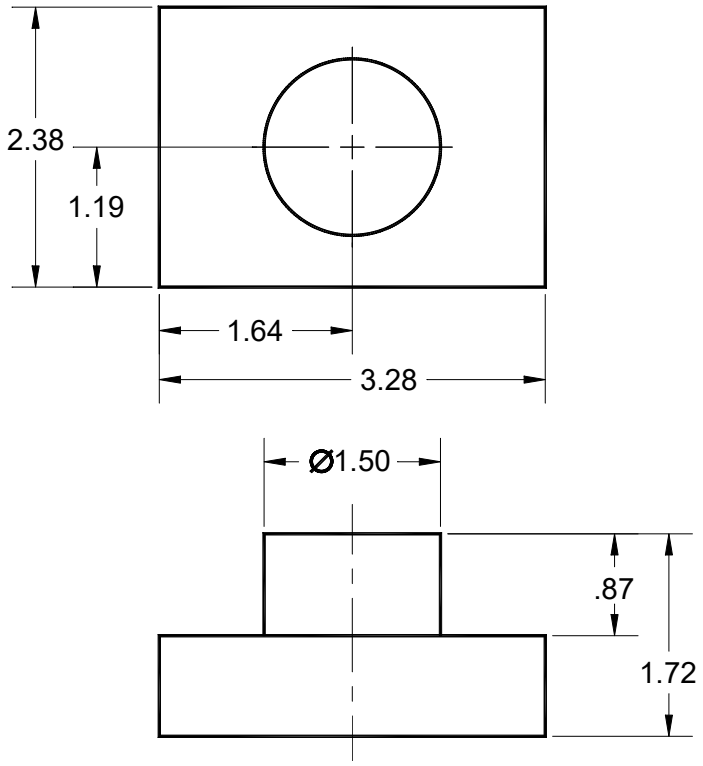
8. Hole features must be located and given size in the view where they appear as a circle not as a rectangle or hidden lines. **Never dimension to hidden lines.**



9. Avoid a complete chain of dimensions. Either omit one or use reference notation. Reference notation indicates that a dimension is used for information purposes only. Indicate a reference dimension by placing a parenthesis around the dimension. For example, (1.51) would indicate a reference dimension. See example below.



10. Cylinder location and size constraint. Locate cylinders in the circular view but give its diameter and length in the rectangular view. See illustration below.



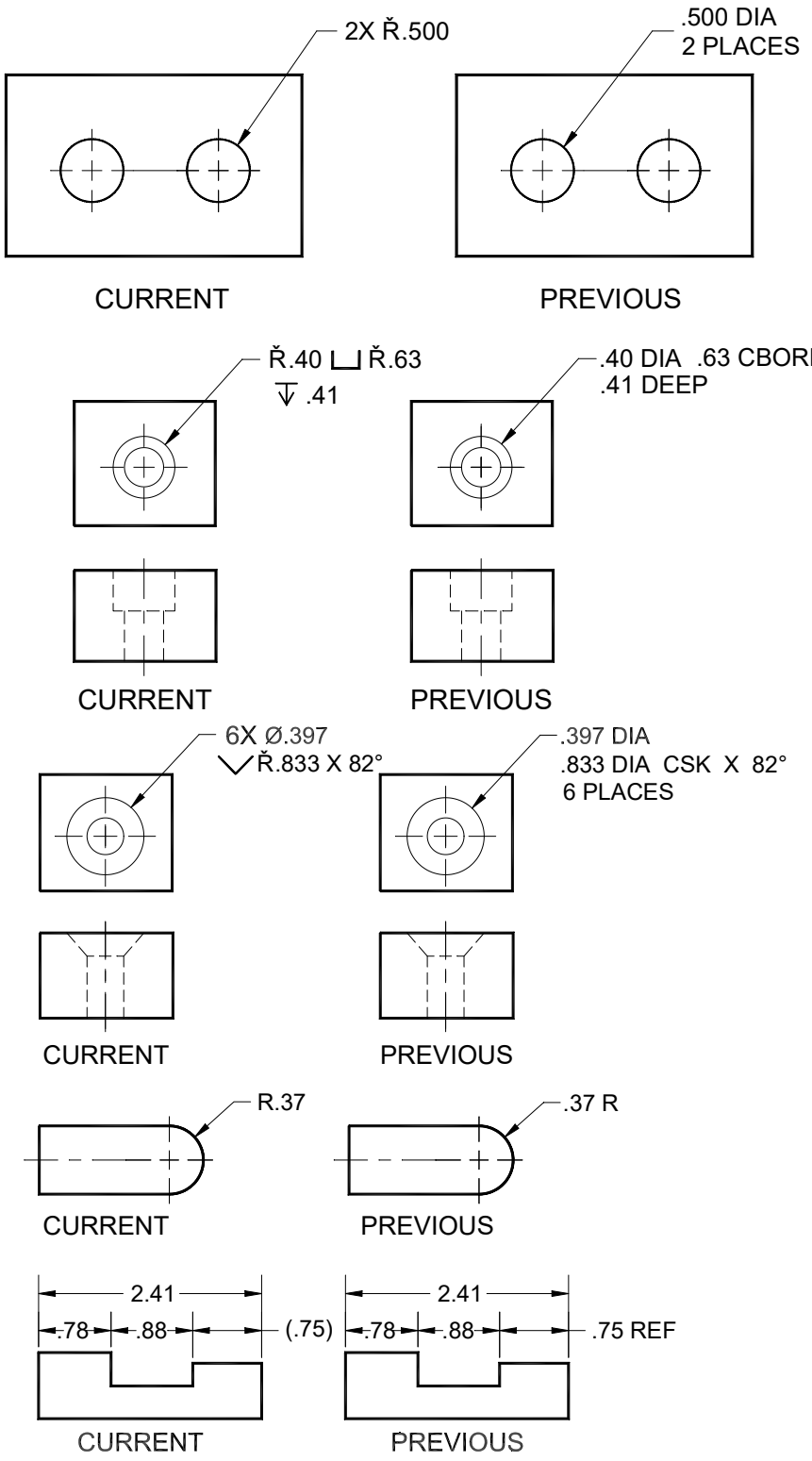
Dimensioning Symbols

1. Symbols represent specific information which could be difficult to duplicate in note form. They aid in clarity, ease of CAD drawing presentation, and above all, save time. Seven such symbols are show below.

⌴	⌵	↧	Ø
COUNTERBORE OR SPOTFACE	COUNTERSINK	DEPTH (OR DEEP)	DIAMETER

R	X	( )
RADIUS	PLACES OR BY	REFERENCE

2. Local Notes: Local notes apply to specific features only and are connected to a feature by a leader. Notes should always be lettered horizontally. Leader arrows for notes should always point toward the circular view of the hole feature and if extended, would pass through the center. Local notes showing hole features, using the above symbols, are presented below reflecting Current and Previous ANSI practices.

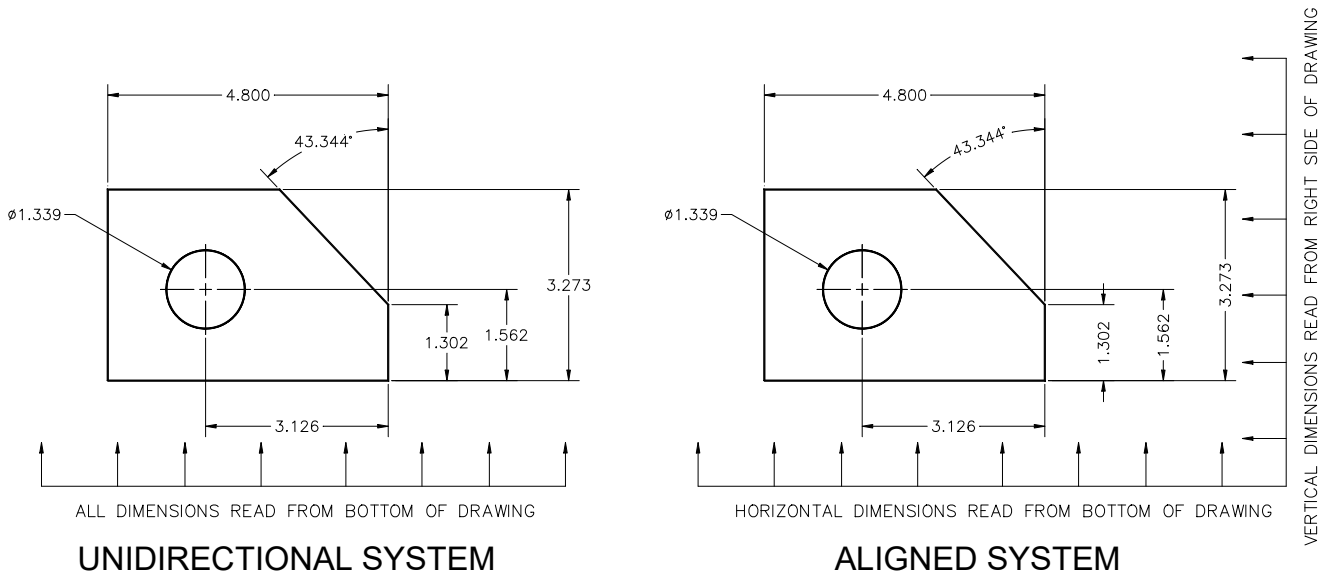

2X Ř.500
.500 DIA
2 PLACES
CURRENT
PREVIOUS
Ř.40 ⌴ Ř.63
↧ .41
.40 DIA .63 CBORE
.41 DEEP
CURRENT
PREVIOUS
6X Ø.397
⌵ Ř.833 X 82°
.397 DIA
.833 DIA CSK X 82°
6 PLACES
CURRENT
PREVIOUS
R.37
.37 R
CURRENT
PREVIOUS
2.41
.78
.88
(.75)
2.41
.78
.88
.75 REF
CURRENT
PREVIOUS

Direction of Dimension Figures

1. Two systems of placing dimensions are used—Unidirectional (Mechanical) or Aligned (Architectural).

Unidirectional: Dimensioned numeral values and text are placed so that they can be read only from the bottom of the drawing.

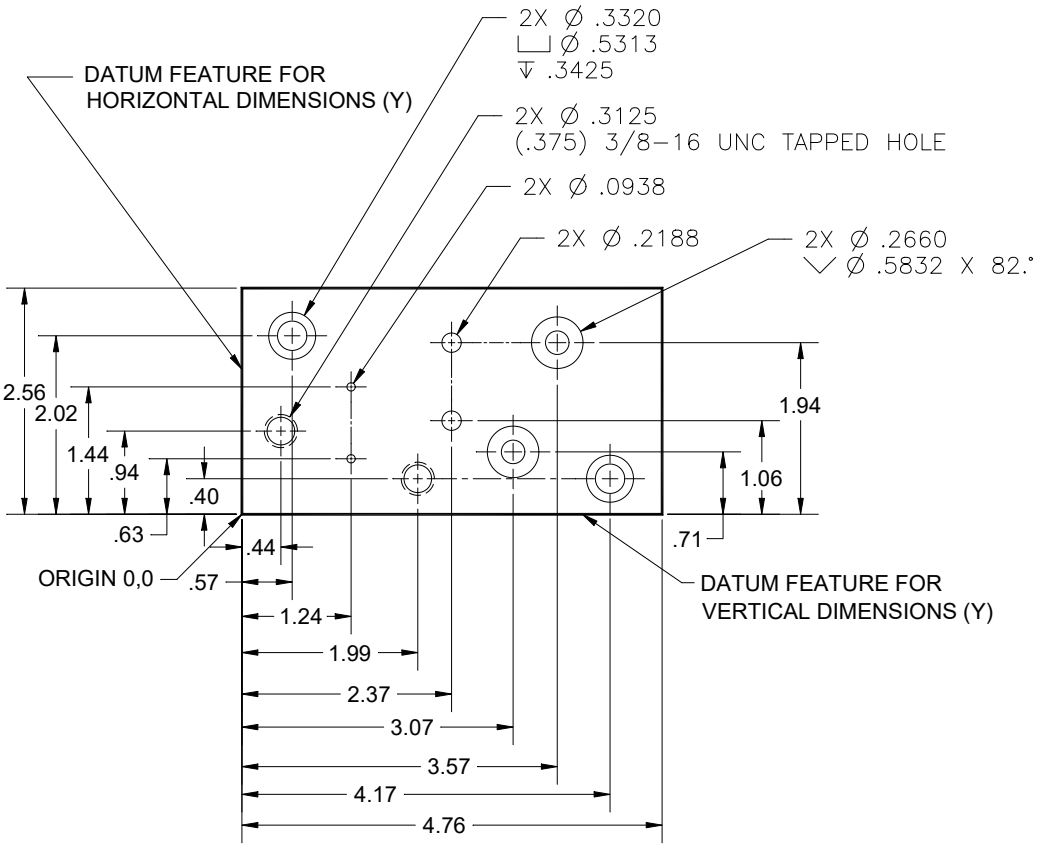
Aligned: Dimensioned numeral values and text are placed so that they can be read either from the bottom or right side of the drawing.



Ordinate Dimensioning

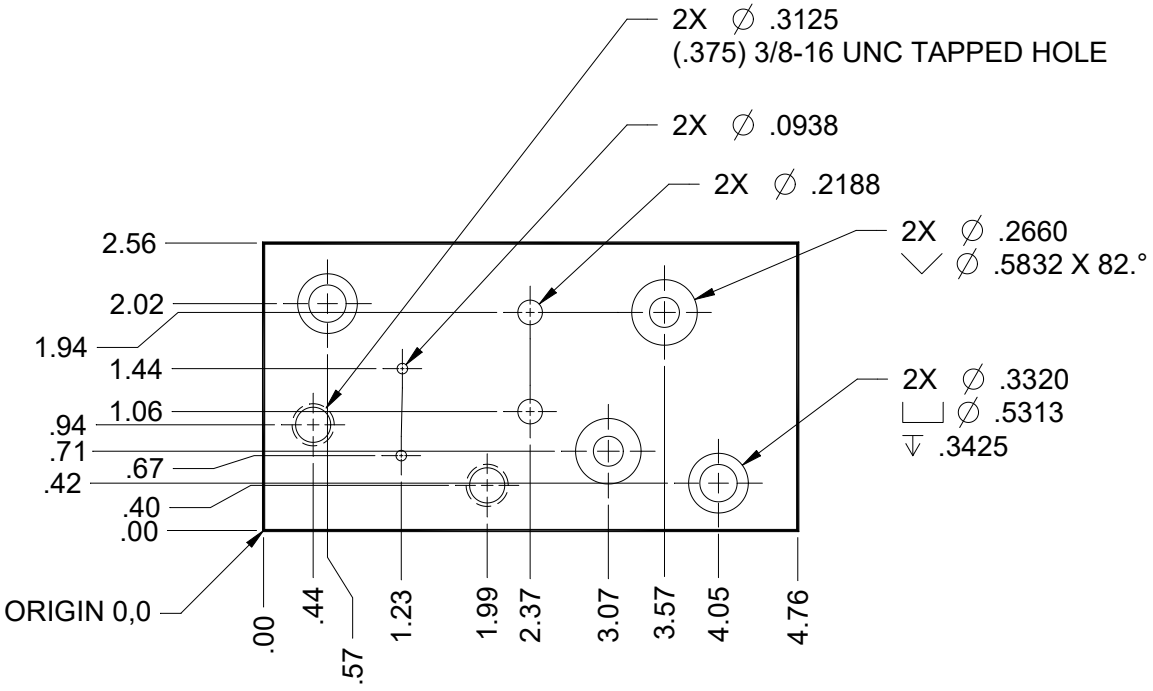
Ordinate dimensioning is also known as Datum dimensioning or Baseline dimensioning. Dimensions may be applied either using dimension lines or arrowheads (Figure 1) or without dimension lines or arrowheads (Figure 2).

A much clearer method of presenting a part consisting of numerous holes is Tabular Ordinate dimensioning. This method involves labeling each hole feature with a letter and then providing a Hole Chart indicating the X location constraint and Y location constraint with a complete size description of each hole feature (Figure 3). This eliminates the need for both dimension and extension lines to locate hole features.



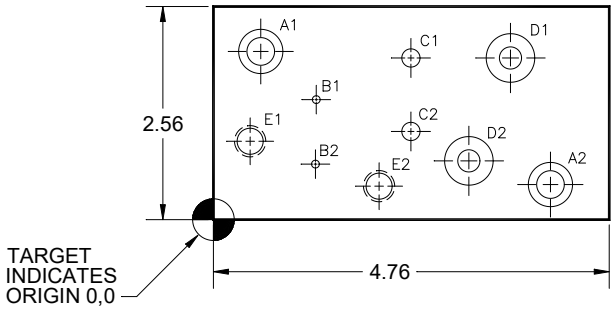
Using Dimension lines and Arrowheads

Figure 1



Omitting Dimension lines and Arrowheads

Figure 2



HOLE CHART

LABEL	X	Y	DESCRIPTION
A1	.569	2.019	C'BORE FOR 5/16 SOCKET CAP
A2	4.050	.422	"
B1	1.236	1.441	3/32 (.0938) DIAMETER HOLE
B2	1.226	.667	"
C1	2.373	1.941	7/32 (.2188) DIAMETER HOLE
C2	2.373	1.059	"
D1	3.570	1.941	C'SINK FOR 1/4 FLAT SOCKET (82)
D2	3.069	.706	"
E1	.441	.941	(.375) 3/8-16 UNC TAPPED HOLE
E2	1.991	.402	"

Tabular Ordinate Dimensioning

Figure 3