

DIMENSIONING ENGINEERING DRAWINGS

An engineering drawing must be properly dimensioned in order to convey the designer's intent to the end user. Dimensions provide the information needed to specify the *size* and *location* of every feature on the object. A properly dimensioned drawing helps ensure that the part produced in the manufacturing phase matches the part you thought you asked for. There are a few simple guidelines to be followed when dimensioning a drawing and these guidelines cover the majority of cases you will encounter. Before we can begin to go over the rules for dimensioning, we must learn a little bit about the anatomy of a dimension.

A dimensioned view of a part is shown in Figure 1. The following terms refer to this figure:

- The *dimension* is the numerical value that is being assigned to the size, shape or location of the feature being described.
- *Dimension lines* are thin lines that show the extent and direction of the dimension. Arrowheads are placed at the ends of the dimension line. The dimension line is normally broken to insert the dimension. In cases where the space prevents the dimension from being placed between the dimension lines, the dimension may be placed outside of the dimension line.
- *Extension lines* are drawn perpendicular to the dimension line and associate the dimension with a particular feature or location. Extension lines should not touch the object lines - there should be a visible gap between the extension line and the object line.
- A *leader line* is a thin solid line that is used to associate a note, symbol or dimension with a feature. Leader lines are typically used to call out radii and diameters.
- A *centerline* is a light broken line used to indicate the centerline of a feature. The line has a long - short - long sequence as indicated in Figure 1.

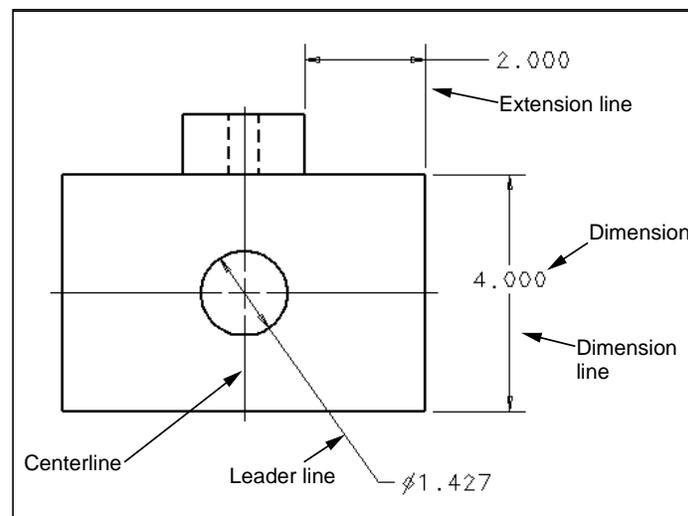


Figure 1 The anatomy of a dimension.

Dimensions are used to indicate the theoretical or exact size or location of a feature. It is impossible to produce exact parts, so every dimension must have an associated *tolerance*. The tolerance specifies the amount that a dimension may deviate from the basic dimension. The tolerance applied to a particular dimension may be specified along with the dimension or it may be specified in a note or in the title block of the drawing as shown in Figure 2. The specific value selected for the tolerance depends upon a number of factors. The principal factor used to set a tolerance for a dimension should be the function of the feature being controlled by the dimension. Unnecessarily tight tolerances lead to high cost of manufacture resulting from more expensive manufacturing methods and from higher reject rates. The tolerance required for the rough opening of a window in a house is considerably different from the tolerance required for a rotating shaft journal.

- The system of units used for the dimensions should be clearly identified on the drawing, usually in the title block along with the default tolerance values for all dimensions.
 - Every dimension must have a tolerance.

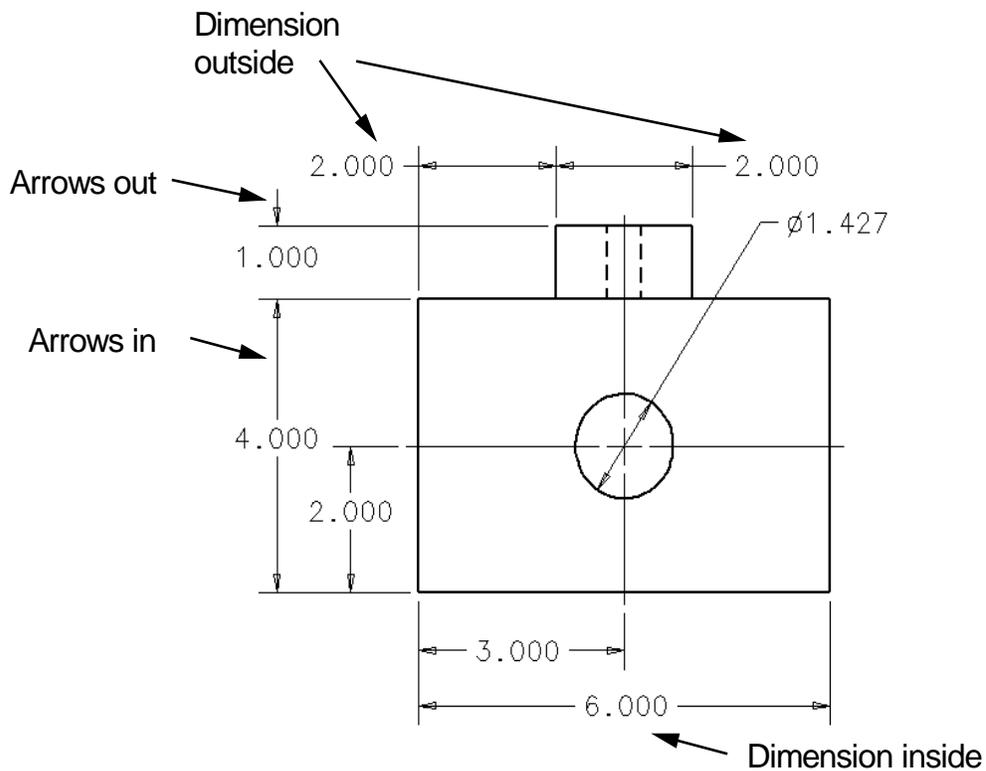
		FSCM	PART OR	
QTY REQD		NO	IDENTIFYING NO	
UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES TOLERANCES ARE:				CONTRACT NO.
FRACTIONS DECIMALS ANGLES ± 1/32 .XX=±.010 ± .5° .XXX=±.005				APPROVALS
DO NOT SCALE DRAWING				DRAWN
TREATMENT				CHECKED
FINISH				ISSUED
SIMILAR TO		ACT. WT	CALC WT	

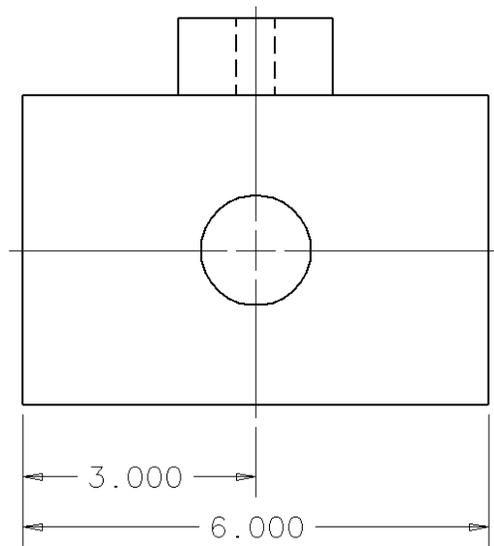
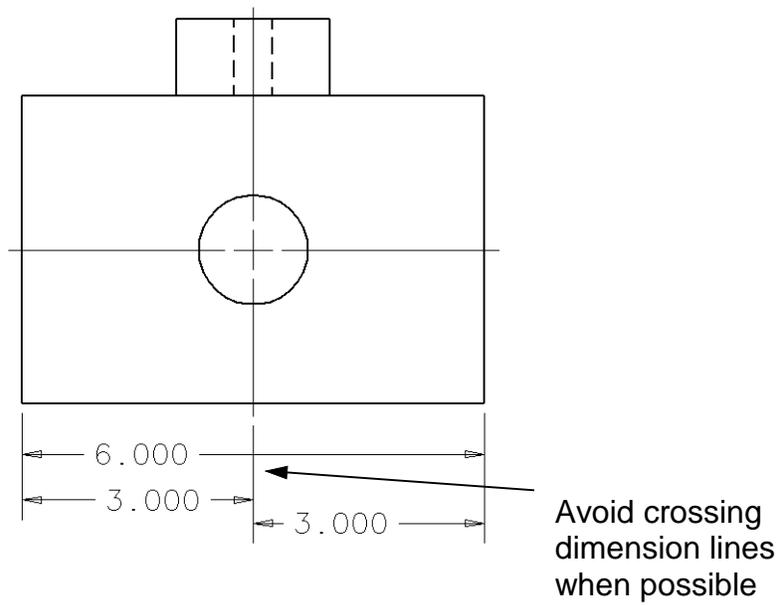

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Figure 2 Title block showing the default tolerance for all dimensions.

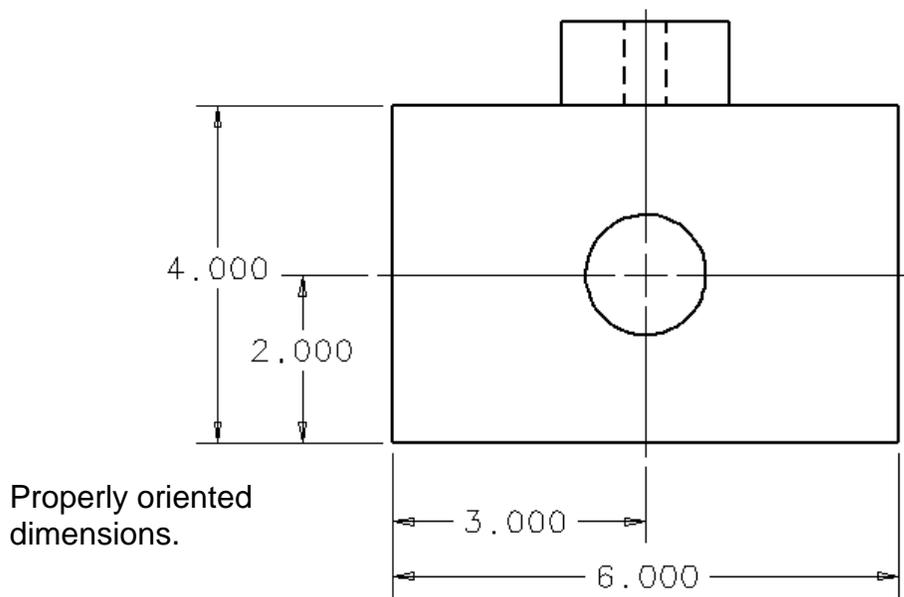
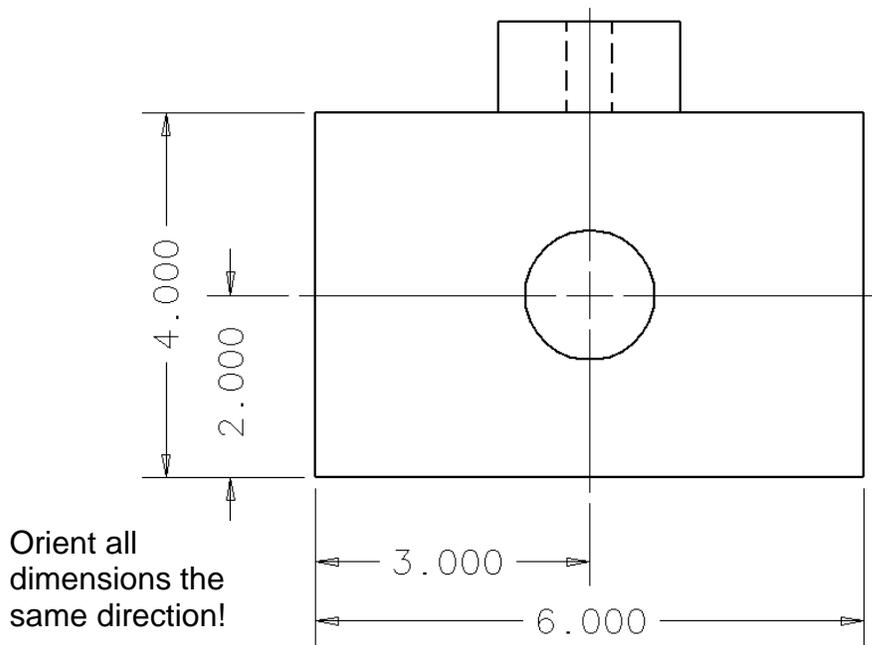
A simple procedure for laying out the dimensions of a part is to break the part down into a series of geometric features, apply dimensions to *size* each of the features, then apply dimensions to control the *location* of the features. There are usually several different ways to dimension any given object. Dimensions should be selected based on the function of the part. Make sure that you directly control the most important features from a functional viewpoint. The dimensions that are selected for describing the part can have a significant impact on the way in which an object is manufactured. The following rules provide some simple guidelines to be followed when placing dimensions on a drawing.

- The dimension should be applied in the view that provides the best description of the feature being dimensioned. For example, holes should be dimensioned in a view where they appear round. A slot should be dimensioned in a view where the contour of the slot is visible.
- Apply dimensions in a view where the feature appears true size.
- Reference dimensions should be placed in parenthesis.
- The overriding principle of dimensioning is clarity.
- Angles shown on drawings as right angles are assumed to be 90 degrees unless otherwise specified and they need not be dimensioned.
- The dimension should be centered between the extension lines. The dimension may be placed outside of the dimension lines if there is insufficient space. The arrows may point out or in depending on the available space.

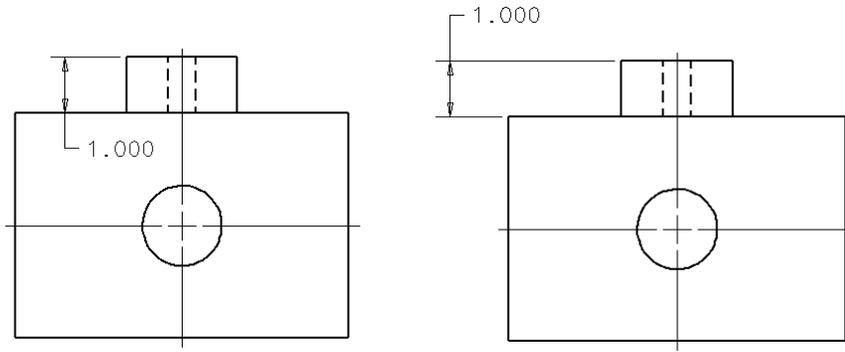




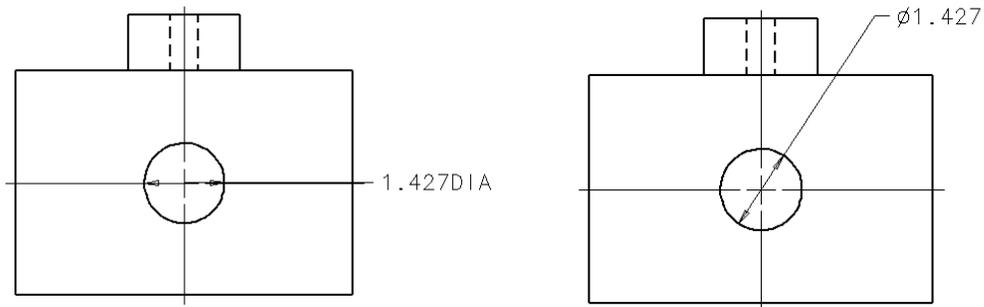
- Place larger dimensions towards the outside so that extension lines don't cross dimension lines.
- Do not over-dimension. Each feature should be dimensioned once and only once.



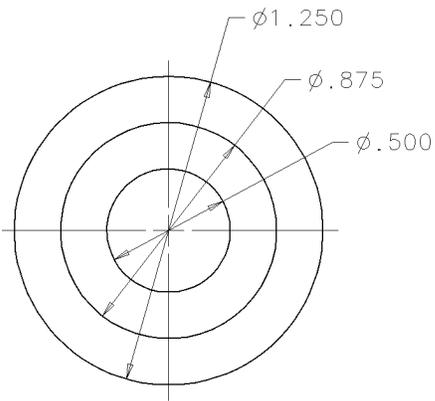
- The dimensions on a drawing should be aligned so that the dimensions are all oriented in the same direction,



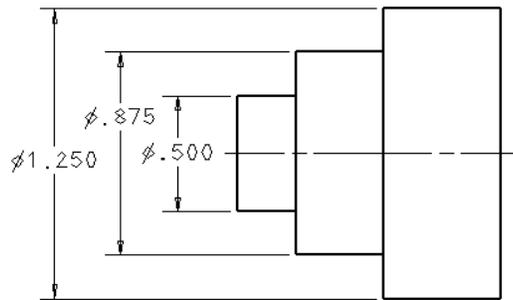
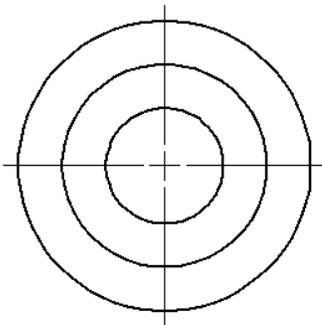
- Dimension lines should not end at object lines. Use extension lines to relate the dimension to the feature being described.
- Do not place dimensions in the view. Use extension lines from the feature and locate the dimension outside of the view.



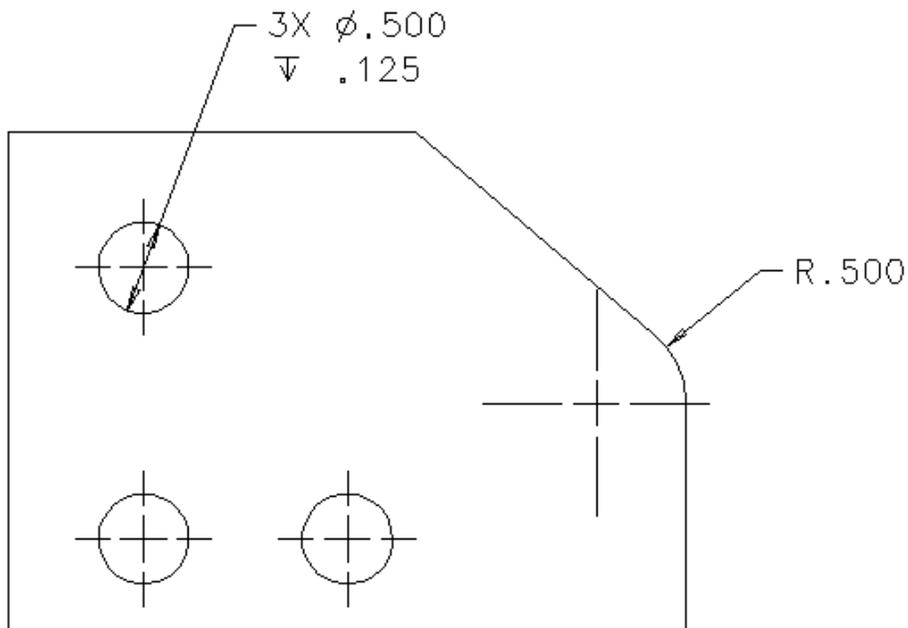
- Use the diameter dimension to specify the size of holes and cylinders. Precede the dimension with the diameter symbol, ϕ .
- Leaders should not be drawn horizontal or vertical.



Too many diameters are confusing with concentric circles



- Concentric circles should be dimensioned in a longitudinal view.
- Stagger the dimensions if they are stacked.



- Use the times symbol, \times , to indicate repeated dimensions or features.
- Use the radius to dimension an arc. The radius dimension is preceded by the symbol, R. A leader line is commonly used for diameters and radii. The leader line should be a radial line directed through the center of the arc or circle.

\sqcup .XXX - Counterbore or Spotface

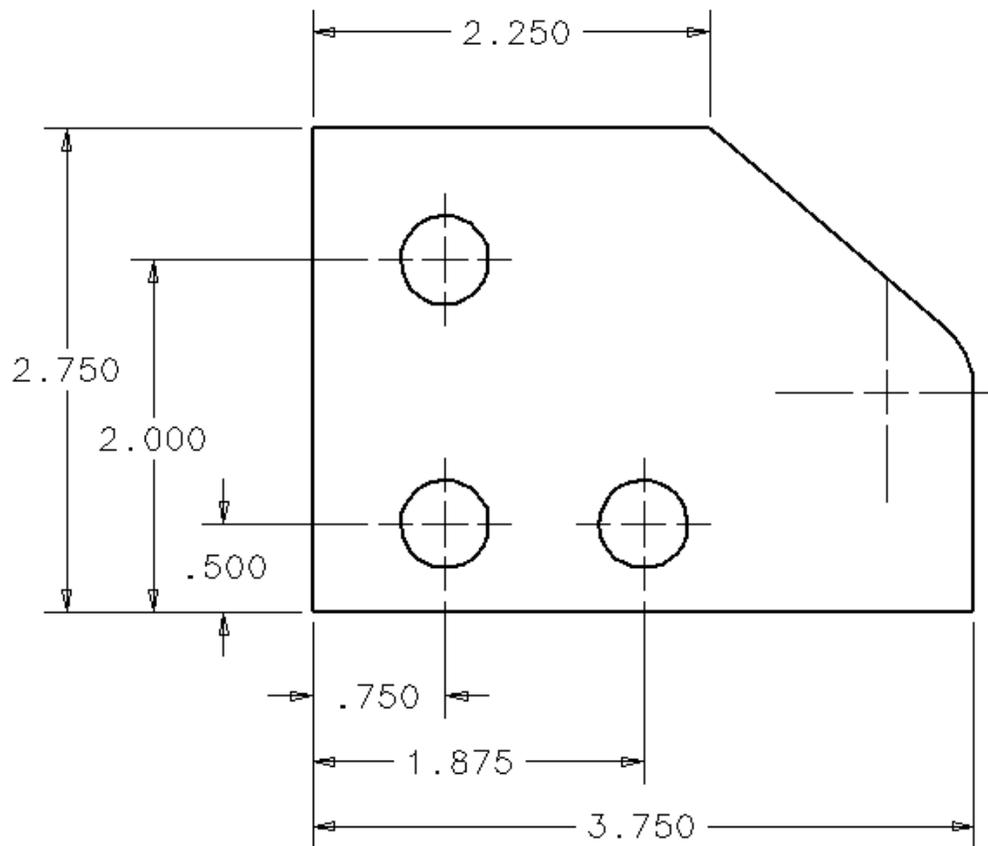
∇ .XXX - Depth

\sphericalangle .XXX - Countersink

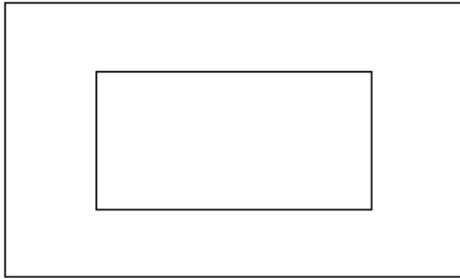
\square .XXX - Square

ϕ .XXX - Diameter

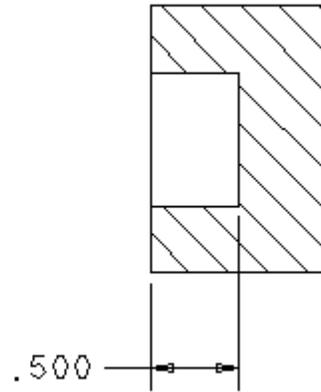
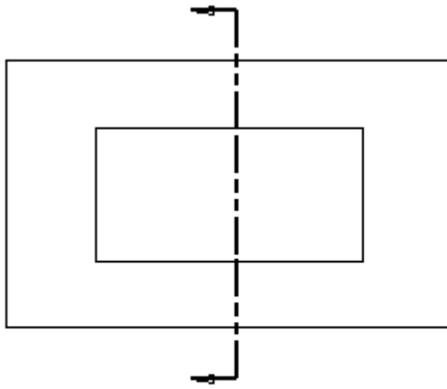
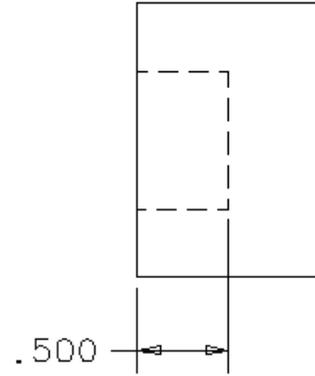
- Use appropriate symbols for describing common features.



- Begin dimensions from a common origin when possible (baseline dimensioning). Functional considerations of the feature override this principle.
- The depth dimension for a blind hole is the depth to which the hole maintains the full diameter.
- Dimensions should be uniformly spaced.
- Group associated dimensions.
- Circular features should be located by dimensioning the centerlines



Don't dimension to hidden lines



- Avoid dimensioning to hidden lines. Create a section view if necessary to dimension to an object line.

CHECKLIST OF DIMENSIONING DO'S AND DON'Ts

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