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Dimensioning your drawing

The BtoCAD dimensioning tools let you add measurements to a drawing. You can quickly add dimensions by simply pointing to entities. You can also add tolerance symbols to your drawings.

The program's many dimensioning variables let you control the appearance of the dimensions. With dimension styles, you can save dimension variable settings so you can reuse them without having to re-create them.

This section explains how to:

- Create dimensions as linear, angular, diametral, radial, and ordinate.
- Create leaders and annotations.
- Edit dimensions.
- Use dimension styles and variables.
- Add geometric tolerances.
- Control dimension tolerance.
- Control alternate dimension units.

Topics in this chapter

Understanding dimensioning concepts

You can create five basic types of dimensions: linear, angular, radial, diametral, and ordinate. You can create dimensions for existing entities by selecting them, or you can create dimensions by selecting points within a drawing. For example, you can create a linear dimension either by selecting the entity to be dimensioned or by speci- fying the first and second extension line origins.
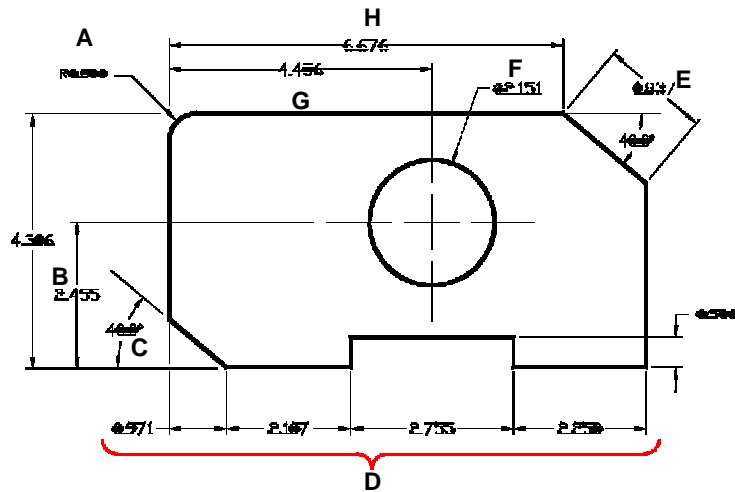

A Radial dimension.
B Vertical linear dimension.
C Angular dimension.
D Linear continued dimensions.
E Aligned dimension.
F   Diametral dimension.
G Horizontal linear dimension.
H Linear baseline dimension.

When you create a dimension, the program draws it on the current layer, using the current dimension style. Each dimension has an associated dimension style, which controls the appearance of the dimension, such as the types of arrowheads, text style, and colors of various components. You can modify existing dimension styles by changing one of the dimension variable settings and then updating the dimension style to reflect the new settings.

Each dimension you create consists of several parts. A dimension line shows where a dimension begins and ends. When you create an angular dimension, the dimension line is a dimension line arc that subtends the measured angle.

Extension lines, also called projection lines, are lines that extend away from the entity for which you are creating a dimension, so that you can place the dimension line away from the entity. Arrowheads form the termination at each end of the dimension line.

Dimension text contains the measured dimension and can also include prefixes, suf- fixes, tolerances, and other optional text. As you insert dimensions, you can control the dimension text and specify its position and orientation.

A   Extension line.
B   Arrowhead.
C   Dimension line.
D   Dimension text.
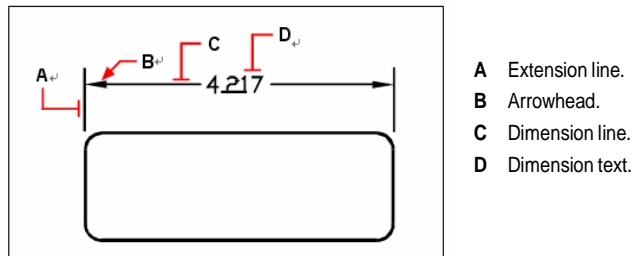
Dimensions can also contain other optional components. A leader is a line leading from a feature of the drawing to an annotation. Leaders begin with an arrowhead, and you can use them to place a dimension away from the dimension line or to add notes. When you create a radial dimension, you can add a center mark, which is a small cross that marks the center of a circle or an arc, or you can add centerlines, which are crossing lines that extend out from the center of a circle or an arc.

Leader.   Center mark.   Centerlines.

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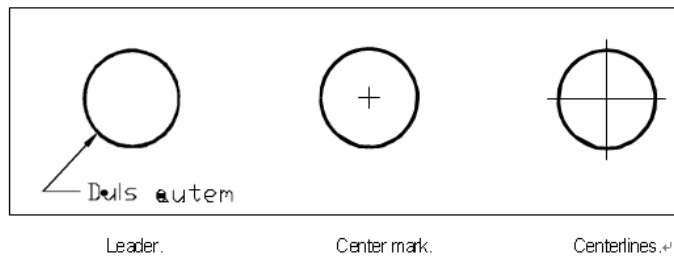
Creating dimensions

You can create dimensions by:

• Selecting the entity to dimension and specifying the dimension line location.

• Specifying the extension line origins and the dimension line location.

When you create dimensions by selecting an entity, the program automatically places the extension line origins at the appropriate definition points based on the type of entity you select. For example, the definition points are located at the endpoints of arcs, lines, and polyline segments. When you create dimensions by specifying the extension line origins, the points you specify determine the definition points. To establish these points precisely, use entity snaps.

Creating linear dimensions

Linear dimensions annotate linear distances or lengths and can be oriented horizon- tally, vertically, or aligned parallel to an existing entity or to the selected extension origin points. After you create a linear dimension, you can add a baseline dimension or a continued dimension. A linear baseline dimension inserts an additional dimen- sion from a common first extension line origin of a previous linear dimension. A lin- ear continued dimension continues a linear dimension from the second extension line of a previous linear dimension.
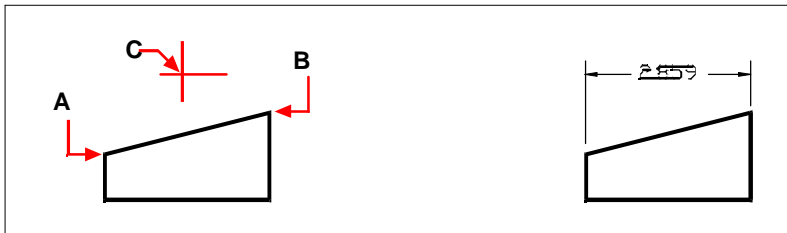
TIP   *To select precise ordinate points, use entity snaps.*

To create a horizontal or vertical dimension

1   Do one of the following:
   • Choose Dimension > Linear.
   • On the Dimension toolbar, click the Linear tool ().
   • Type *dimlinear* and then press Enter.

2   Press Enter, and then select the entity to dimension.

   Or, you can insert the dimension by specifying the first and second extension line origins.

3   Specify the dimension line location.

To insert a linear dimension by selecting the entity, select the
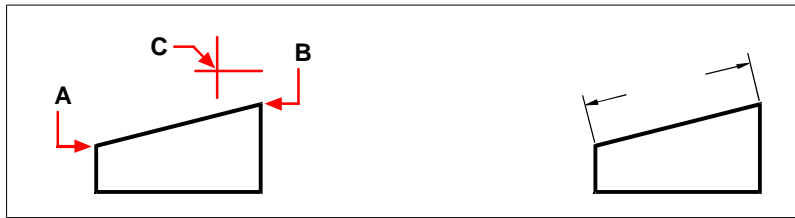entity (**A**) to dimension, and then specify the dimension line location (**B**).   Result.

To insert a linear dimension by selecting the extension line   Result.
origins, select the first extension origin (**A**),
select the second extension origin (**B**),
and then specify the dimension line location (**C**).

To create an aligned dimension

1   Do one of the following:
   •   Choose Dimension > Aligned.

   •   On the Dimension toolbar, click the Aligned tool ( ).
   •   Type *dimaligned* and then press Enter.

2   Press Enter, and then select the entity to dimension.

   Or, you can insert the dimension by specifying the first and second extension line origins.

3   Specify the dimension line location.

To insert an aligned dimension by selecting the entity, select   Result
the entity (**A**) to dimension, and then select the dimension linelocation (**B**).   .

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To insert an aligned dimension by selecting the extension line
origins, select the first extension origin (**A**), select the second
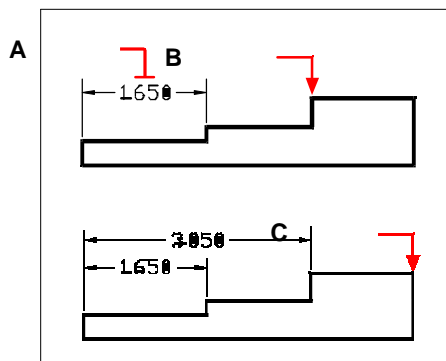extension origin (**B**), and then specify the dimension line location (**C**).

Result

To create a linear baseline dimension

NOTE *Before you can use this procedure, you must first create a dimension.*
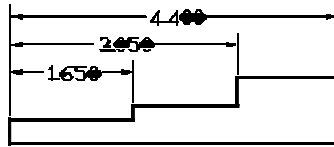
1   Do one of the following:
   •   Choose Dimension > Baseline.

   •   On the Dimension toolbar, click the Baseline tool ( ).
   •   Type *dimbaseline* and then press Enter.

2   To select a starting dimension, press Enter.

3   Select the next extension line origin, and then press Enter.

Or press Enter, and then select an existing dimension for the baseline. Select the origin of the next extension
line, and then press Enter.

The program automatically places the new baseline dimension above or below the previous dimension line.
The distance between the two dimension lines is determined by the Baseline Offset value in the Dimension
Settings dialog box.

To add a baseline dimension to an existing linear dimension, select the existing dimension (**A**), select the next extension
line origin (**B**), and select as many additional points as you want (**C**).

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Result.

To create a linear continued dimension

NOTE *Before you can use this procedure, you must first create a dimension.*

1   Do one of the following:
   • Choose Dimension > Continue.

   • On the Dimension toolbar, click the Continue tool (⊢⊣).
   • Type *dimcontinue* and then press Enter.

2   To select a starting dimension, press Enter.

3   Select the next extension line origin, and then press Enter.

   Or press Enter, and then select an existing dimension to continue.

4   To add continued dimensions, continue selecting extension line origins.

5   To end the command, press Enter twice.
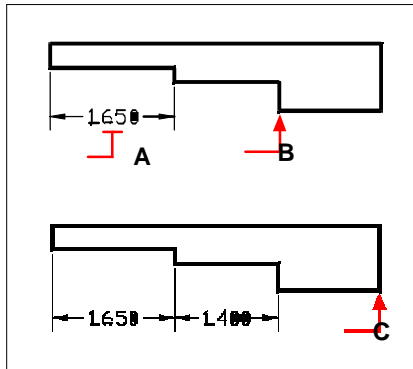
To add a continued dimension to an existing linear dimension, select the existing dimension (**A**), select
the next extension line origin (**B**), and select another extension line origin (**C**).

Creating angular dimensions

Angular dimensions annotate the angle measured between two lines. You can also dimension an angle by selecting an angle vertex and two endpoints. After you create an angular dimension, you can add a baseline dimension or a continued dimension.

An angular baseline dimension inserts an additional dimension from a common first extension line origin of a previous angular dimension. An angular continued dimen- sion continues an angular dimension from the second extension line of a previous angular dimension.

To dimension an angle encompassed by an arc

1   Do one of the following:
  •   Choose Dimension > Angular.

  •   On the Dimension toolbar, click the Angular tool ( ).

  •   Type *dimangular* and then press Enter.

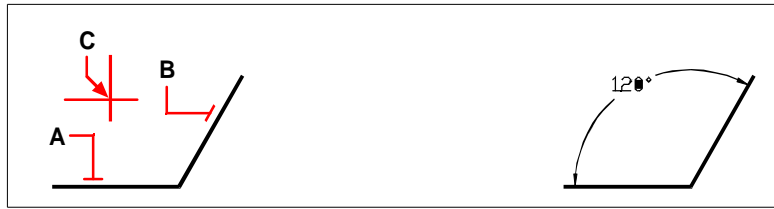2   Select the arc.

3   Specify the dimension arc location.

To dimension the angle subtended by an arc, selectthe arc (**A**), and then specify the dimension arc location (**B**).

To dimension an angle between two lines

1   Do one of the following:
  •   Choose Dimension > Angular.

  •   On the Dimension toolbar, click the Angular tool ( ).

  •   Type *dimangular* and then press Enter.

2   Select one line.

3   Select the other line.

4   Specify the dimension line location.

Select one line (**A**), select the other line (**B**), and then specify the dimension line location (**C**).

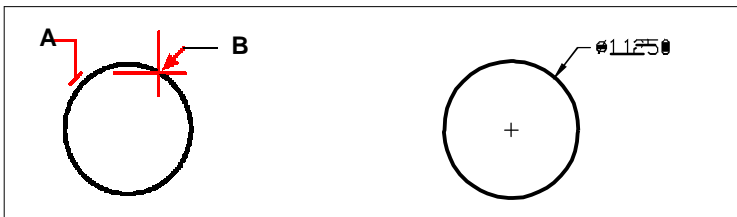Result.

Creating diametral and radial dimensions

Diameter and radius dimensions annotate the radii and diameters of arcs and circles. You can optionally include centerlines or center marks.

To create a diametral dimension

1 Do one of the following:
- Choose Dimension > Diameter.
- On the Dimension toolbar, click the Diameter tool ( ).
- Type *dimdiameter* and then press Enter.

2 Select the arc or circle.

3 Specify the dimension line location.

Select the circle (**A**), and then specify the dimension line location (**B**).Result.

To create a radial dimension

1 Do one of the following:
- Choose Dimension > Radius.
- On the Dimension toolbar, click the Radius tool ( ).
- Type *dimradius* and then press Enter.

2 Select the arc or circle.

**3** Specify the dimension line location.

Select the circle (**A**), and then specify the dimension line location (**B**).Result.

Creating ordinate dimensions

An ordinate dimension annotates the perpendicular distance from an origin or base point (the origin of the current user coordinate system [UCS]). Ordinate dimensions consist of an x- or y-coordinate and a leader. An x-ordinate dimension measures dis- tances along the x-axis; a y-ordinate dimension measures distances along the y-axis.

As you select ordinate points, the program automatically determines whether the point is an x- or y-ordinate based on which direction you drag the second point. You can also specify whether the ordinate represents an x- or y-ordinate. Ordinate dimen- sion text is always aligned with the ordinate leader lines, regardless of the text orien- tation specified by the current dimension style.
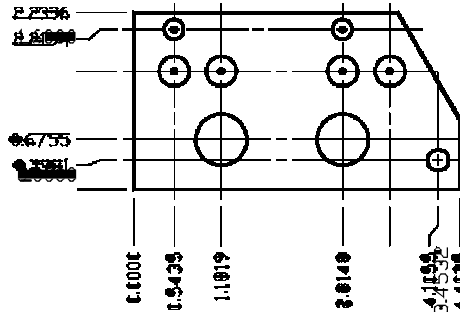
Ordinate dimensions measure the distance along the x- or y-axis from an origin to a selected ordinate point.

To create an ordinate dimension

1   Do one of the following:
- Choose Dimension > Ordinate.

- On the Dimensioning toolbar, click the Ordinate tool ( ).
- Type *dimordinate* and then press Enter.

2   Select the point for ordinate dimension.

3   Specify the ordinate leader endpoint.
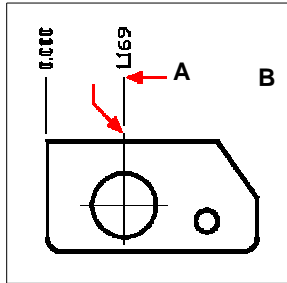
Select the ordinate point (**A**), and then specify the ordinate leader endpoint (**B**).

TIP   *To select precise ordinate points, use entity snaps.*

Creating leaders and annotations

Leaders consist of a line or series of lines that connects a feature in a drawing to an annotation. Generally, you place an arrowhead at the first point. An annotation, cre- ated as dimension text, is placed immediately adjacent to the last point. By default, the text placed at the end of the leader line consists of the most recent dimension. You can also type an annotation as a single line of text.

To create a leader and an annotation

1   Do one of the following:
   • Choose Dimension > Leader.
   • On the Dimension toolbar, click the Leader tool ( ).
   • Type *dimleader* and then press Enter.

2   Specify the starting point of the leader.

3   Specify the endpoint of the leader line segment.

4   Specify additional leader line segment endpoints.

5   After you specify the last endpoint, press Enter.

6   Type the annotation, or press Enter to accept the most recent dimension as the default annotation.
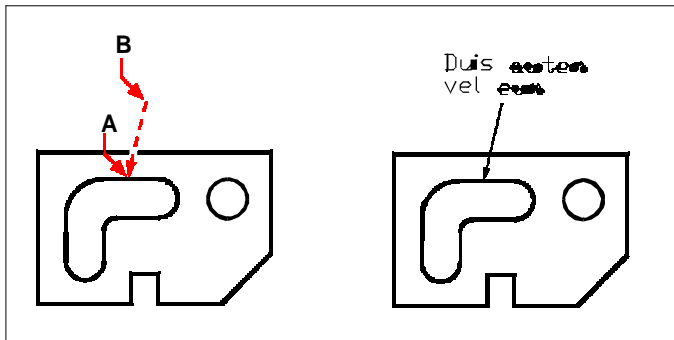
Specify the starting point of the leader (**A**) and the endpoint of the leader line segment(**B**).

Editing dimensions

You can use grips to edit entity dimensions. You can also edit the dimension text.
You can rotate dimension lines and dimension text at any angle, and you can reposi- tion the dimension text anywhere along the dimension line.

Making dimensions oblique

Extension lines are normally created at a perpendicular angle to the dimension line. You can change the angle of the extension lines, however, so that they tilt relative to the dimension line.

To make oblique extension lines

1 Do one of the following:
   - Choose Dimension > Make Oblique.
   - On the Dimension toolbar, click the Make Oblique tool ( ).
   - Type *dimedit,* press Enter, and then in the prompt box, choose Oblique Lines.

2 Select the linear dimension, and then press Enter.

3 Type the obliquing angle, and then press Enter.
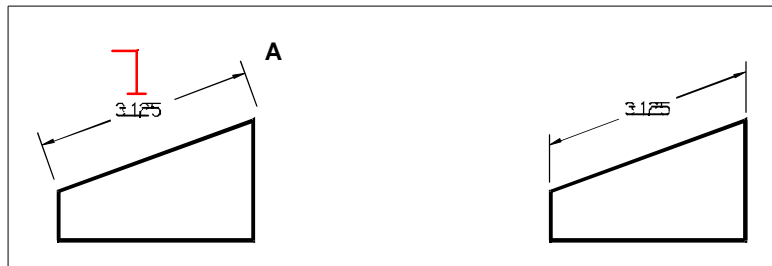
Select the dimension to be made oblique (**A**), and then type the obliquing angle.   Result.

TIP *To align the oblique angle if you don't know the exact measurement, use snaps to pick two points on the entity.*

NOTE *The dimension text angle is relative to the dimension line. If the dimension text rotation is set to zero, the text angle is defined by the dimension type and the dimension style.*

To move dimension text

Advanced experience level

1   Do one of the following:

- On the Dimension toolbar, click the Reposition Dimension Text tool (   ).
- Type *dimtedit* and then press Enter.

2   Select the dimension to reposition text.

3   Select the new text position.
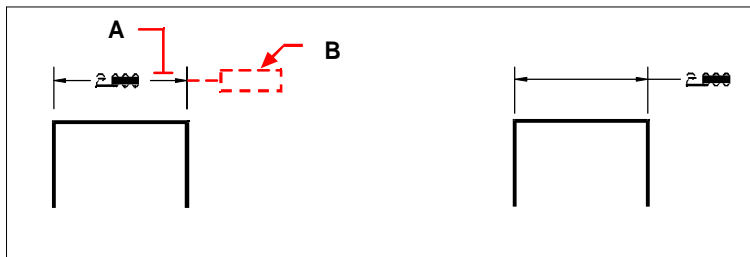
Select the dimension to be moved (**A**), and then select   Result
the new text position (**B**).   .

To restore dimension text to its home position

Advanced experience level

1   Do one of the following:
- Choose Dimension > Align Text > Home.
- Type *dimedit,* press Enter, and then in the prompt box, choose Restore Text.

2   Select the dimension text to restore, and then press Enter.

To replace existing dimension text with new text

Advanced experience level

1   Do one of the following:

- On the Dimension toolbar, click the Edit Dimension Text tool (   ).
- Type *dimedit* and then in the prompt box, choose Edit Text.

2   Type the new dimension text, and then press Enter.

3   Select the dimension to be replaced, and then press Enter.

Understanding dimension styles and variables

Dimensions that you insert are created using the current dimension style. You can create, save, restore, and delete named dimension styles.

To display information about the current dimension style and compare it with other style names, you can use the dimstyle command.

Dimension styles provide a way for you to change various settings that control the appearance of dimensions. You can then save those settings for reuse. If you don't define a dimension style before creating dimensions, the program uses the Standard dimension style, which stores the default dimension variable settings. Each option in the Dimension Settings dialog box relates to a variable that you can set manually. See the online Command Reference for more information.

To create a dimension style

1   Do one of the following:
    - Choose Dimension > Dimension Settings.
    - On the Settings toolbar, click the Dimension Settings tool ( ).
    - Type *setdim* and then press Enter.

2   In the Dimension Settings dialog box, click New.
3   Type the name of the new dimension style.
4   Click Create.
5   In the Dimension Settings dialog box, click one of the other tabs, and then change the dimension settings as necessary. Repeat this step for each tab, as needed.
6   To end the command, click OK.

To select a dimension style

1   Do one of the following:
    - Choose Settings > Dimension Settings, select a dimension style from the list, and then click OK.
    - On the Dimensioning toolbar, select a dimension style from the list.
    - Type *setdim*, press Enter, select a dimension style from the list, and then click OK.

To rename a dimension style

1   Do one of the following:
   • Choose Dimension > Dimension Settings.
   • On the Settings toolbar, click the Dimension Settings tool ().
   • Type *setdim* and then press Enter.

2   In the Dimension Settings dialog box, click Rename.

3   In the Rename list, click the dimension style to be renamed.

4   In the To box, type the new dimension style name.

5   Click Rename.

6   Click OK.

To delete a named dimension style

1   Do one of the following:
   • Choose Dimension > Dimension Settings.
   • On the Settings toolbar, click the Dimension Settings tool ().
   • Type *setdim* and then press Enter.

2   In the Dimension Settings dialog box, click Delete.

3   Select the dimension style to delete.

4   Click Delete.

5   Click OK.

To display information about the current style

1   Type *dimstyle* and then press Enter.

2   Type *v* and then press Enter to display information about the variables.

3   Type the dimension style name, and press Enter.

Controlling dimension arrows

You can control the appearance and size of arrowheads or tick marks placed at the ends of dimension lines. Any changes you make affect the current dimension style. The arrowheads you choose display in the image tile on the right side of the Dimen- sion Settings dialog box.

You can choose from a number of arrowhead types. You can specify different arrow- heads for each end of a dimension line and for leader lines. The Starting arrow corre- sponds to the first extension line; the Ending arrow corresponds to the second extension line. Blocks defined in the drawing also display in the three Arrowhead lists as user-defined arrows. You can use these blocks to create and assign your own arrowheads.

The Arrow Size value determines the size of the arrowhead, measured in drawing units. You can also use tick marks instead of arrowheads.

To choose an arrowhead

1   Do one of the following:
- Choose Dimension > Dimension Settings.
- On the Settings toolbar, click the Dimension Settings tool (□).
- Type *setdim* and then press Enter.

2   Click the Arrows tab.

3   In the Starting Arrowhead or Ending Arrowhead list, click to select the Starting or Ending arrowhead, respectively.

4   In the Leader Arrowhead list, click to select a leader arrowhead for leader lines.

5   Click OK.

NOTE   *You can also specify leader arrow types using the DIMLDRBLK system variable.*
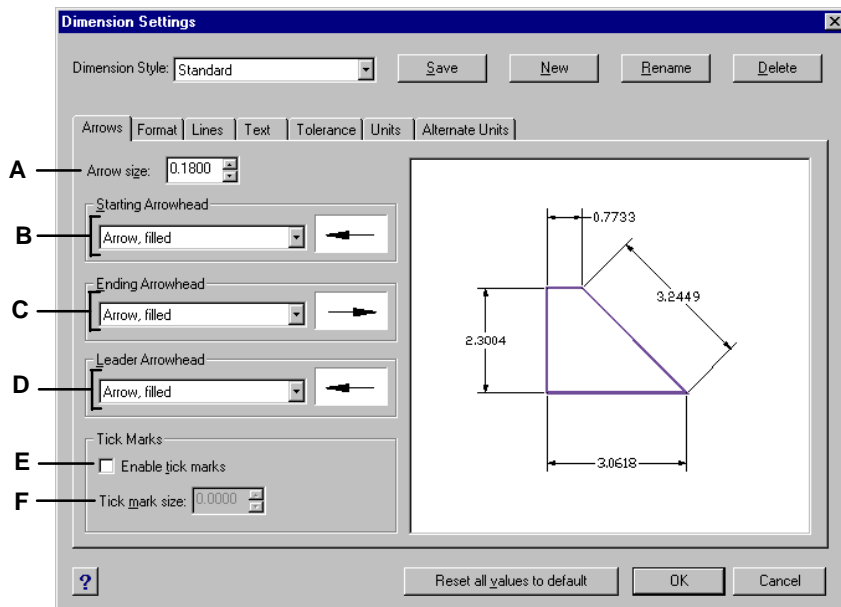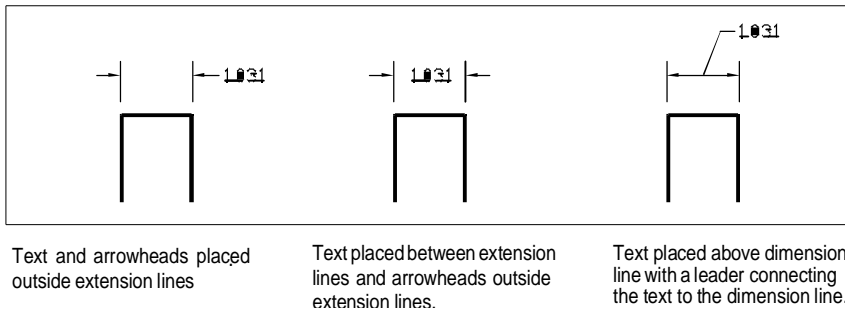
A Type or select arrowhead size.

B Click to select a starting arrowhead, or click the arrowhead picture to scroll the list automatically.

C Click to select an ending arrowhead, or click the arrowhead picture to scroll the list automatically.
D Click to select a leader arrowhead, or click the arrowhead picture to scroll the list automatically.

E Select to enable tick marks instead of arrowheads.

F Type or select tick mark size.

Controlling dimension format

You can control the way dimension text and arrowheads are placed in relation to the dimension lines. Any changes you make affect the current dimension style. The image tile on the right side of the Dimension Settings dialog box shows the appear- ance of dimensions based on the current dimension style settings.

The program determines whether both dimension text and arrowheads will fit between the extension lines by comparing the distance between the extension lines to the size of the dimension text, the size of the arrowheads, and the amount of space required around dimension text. The program applies the best fit method based on the available space. If possible, both the dimension text and arrowheads are placed between the extension lines. If both will not fit between the extension lines, you can determine how text and arrowheads are placed using the Fit Method settings on the Dimension Settings dialog box.

Text and arrowheads placed outside extension lines

Text placed between extension lines and arrowheads outside extension lines.

Text placed above dimension line with a leader connecting the text to the dimension line.

To format dimensions

1 Do one of the following:
  • Choose Dimension > Dimension Settings.
  • On the Settings toolbar, click the Dimension Settings tool ().
  • Type *setdim* and then press Enter.

2 Click the Format tab.

3 Click the Fit option that you want.

**4** Specify the Distance option that you want.

5 Click OK.

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NOTE *You can also specify how dimension text and arrows are arranged using the DIMATFIT system variable. You can specify how dimension text is moved using the DIMTMOVE system variable.*
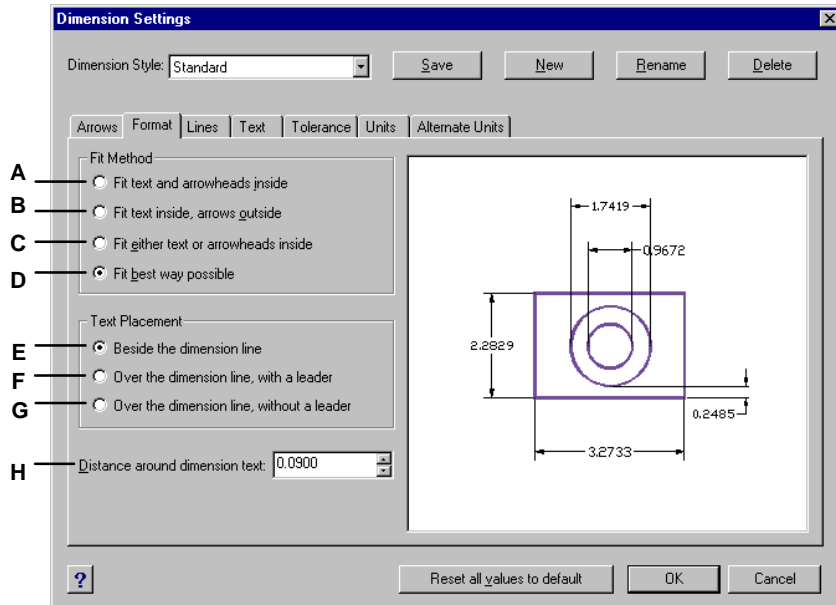
A Click to place both text and arrowheads insidethe extension lines when both do not automatically fit between them.

B Click to place text only between extension lines and arrowheads outside extension lines when both do not fit between them.

C Click to fit either text or arrowheads between extension lines when both do not fit between them.

D Click to automatically determine the best fit method.

E Click to place text beside the dimension line witha leader when both text and arrowheads do not fit between the extension lines.

F Click to place text above the dimension line with a leader connecting the text to the dimension line when both text and arrowheads do not fit between the extension lines.

G Click to place text above the dimension line without a leader when both text and arrowheads do not fit between the extension lines.

H Type or select the distance around the dimension text.

Controlling line settings

You can control settings affecting dimension lines, extension lines, and center marks. Any changes you make affect the current dimension style. The image tile on the right side of the Dimension Settings dialog box shows the appearance of the dimensions based on the current dimension style settings.
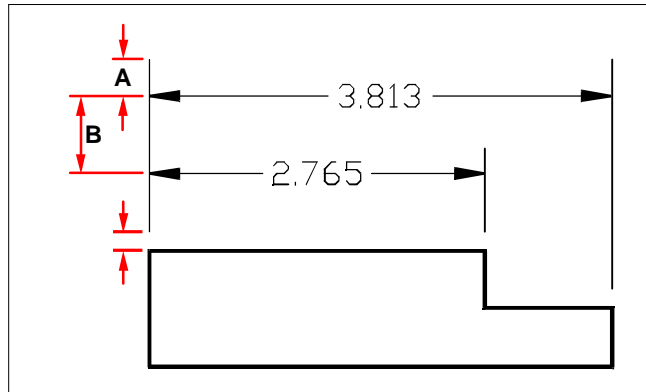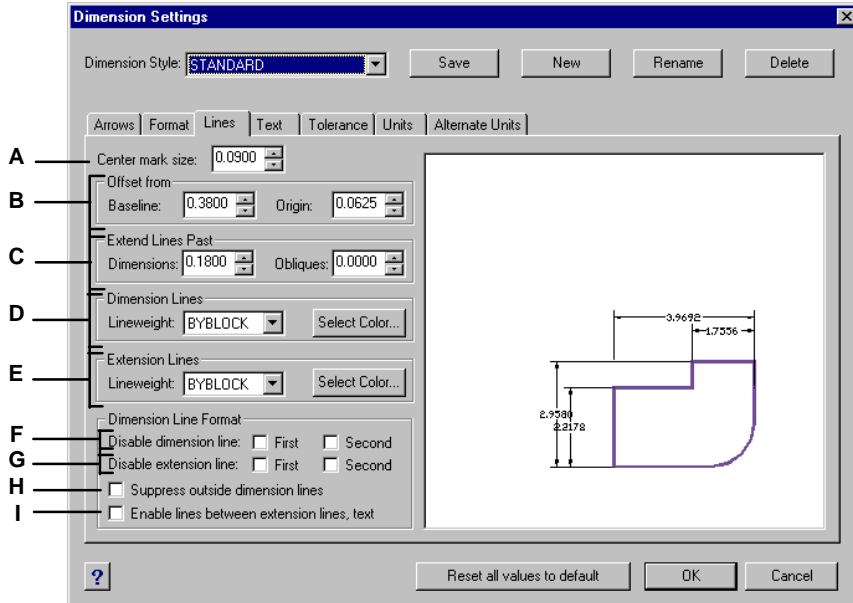

A Extend past dimension.
B Baseline offset.
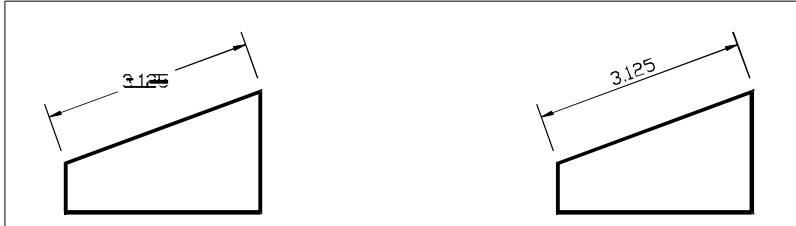
To set the color for dimension lines

1 Do one of the following:
- Choose Dimension > Dimension Settings.
- On the Settings toolbar, click the Dimension Settings tool ().
- Type *setdim* and then press Enter.

2 Click the Lines tab.

3 Make your selections.

4 Click OK.

A Type or select center mark size. Positive values create a center mark. Negative values create centerlines.

B Type or select the baseline offset distance (the distance to offset successive dimension lines when creating baseline dimensions) and the offset from origin (the distance extension lines are offset from their origin points).

C In Dimensions, type or select the distance that extension lines extend beyond dimension lines. In Obliques, type or select the distance that dimension lines extend beyond extension lines.

D Select the dimension lineweight and the dimension line color.
E Select the extension lineweight and theextension line color.

F Select to prevent the creation of the first and second dimension lines.

G Select to prevent the creation of the first and second extension lines.

H Select to prevent the creation of dimension lines outside extension lines.

I Select to draw dimension lines between extension lines when text and arrows are placed outside extension lines.

Controlling dimension text

You can control the settings affecting the appearance of dimension text. Any changes you make affect the current dimension style. The image tile on the right side of the Dimension Settings dialog box shows the appearance of the dimensions based on the current dimension style settings.

Text between extension lines aligned horizontally. Text between extension lines aligned with dimension line.

To align dimension text with the dimension line

1   Do one of the following:
     •   Choose Dimension > Dimension Settings.
     •   On the Settings toolbar, click the Dimension Settings tool ( ).
     •   Type *setdim* and then press Enter.

2   Click the Text tab.

3   Make your selections.

4   Click OK.
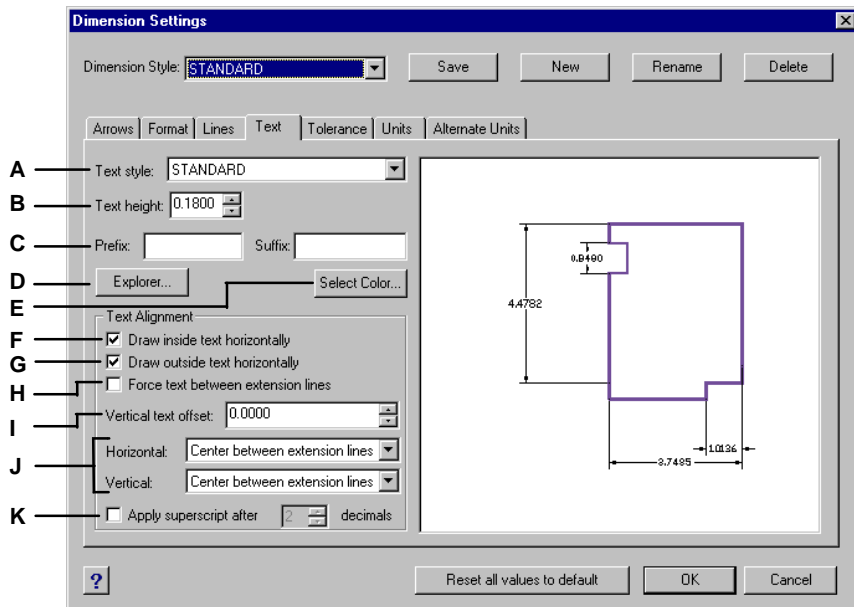
A Click to select the text style used for dimension text.

B Type or select the text height, measured in drawing units.

C Type a prefix or suffix to be appended to dimension text.

D Click to display the BtoCAD Explorer, Text Styles element.

E Click to select the dimension text color.

F Select to align text placed between the extension lines with the dimension line.

G Select to align text placed outside the extension lines with the dimension line.

H Select to force text between the extension lines.

I Type or select the vertical text offset distance.

J Click to select the horizontal or vertical justification of dimension text.

K Formats integers in superscript after the specified number of decimal places.

Controlling dimension units

You can determine the appearance and format of the primary and alternate dimension units. The image tile on the right side of the Dimension Settings dialog box shows the appearance of the dimensions based on the current dimension style settings.

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To round off dimensions

1   Do one of the following:
    • Choose Dimension > Dimension Settings.

    • On the Settings toolbar, click the Dimension Settings tool ( ).

    • Type *setdim* and then press Enter.

2   Click the Units tab.

3   In the Round Linear Distances field, type or select the nearest value to which you want to round off dimensions.

4   Click OK.

NOTE *You can also specify units for linear dimensions using the DIMLUNIT system variable. You can specify fraction formats using the DIMFRAC system variable.*
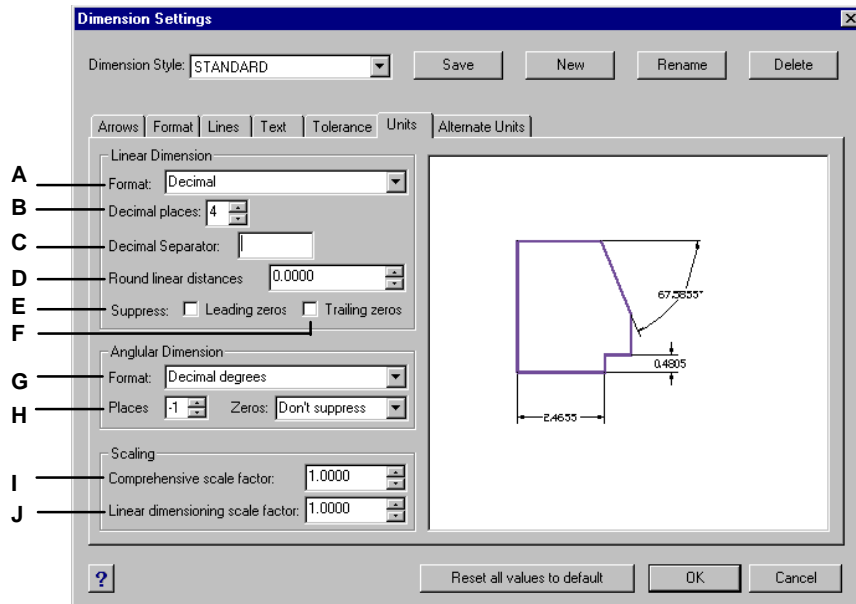
A Select the linear dimension format.

B Type or select the number of decimal places you want displayed in linear dimension text.

C Enter the marker symbol used for decimals.

D Type or select the nearest value to which you want to round off linear distances.

E Select to prevent the inclusion of leading zeros or the inclusion of feet in dimension text when the dimension is less than one foot.

F Select to prevent the inclusion of trailing zeros or the inclusion of inches in dimension text when the number of inches is zero.

G Select the angular dimension format.

H Type or select the number of decimal places you want displayed for angular dimension text. Select whether to suppress leading or trailing zeros.

I Type or select the scale factor applied to all dimensions.

J Type or select the linear scale factor applied to all lengths measured by dimensioning commands.

Adding geometric tolerances

Geometric tolerances indicate the maximum allowable variations in the geometry defined by a drawing. BtoCAD draws geometric tolerances using a feature control frame, which is a rectangle divided into compartments.

Each feature control frame consists of at least two compartments. The first compartment contains a geometric tolerance symbol that indicates the geometric characteristic to which the tolerance is applied, such as location,

orientation, or form. For example, a form tolerance may indicate the flatness or roundness of a surface. The geometric tolerance symbols and their characteristics are shown in the following table.
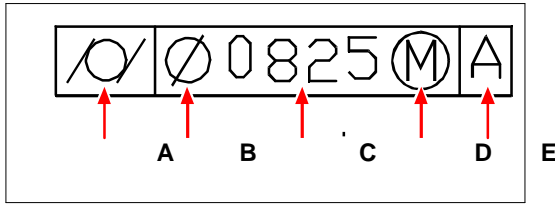
Geometric tolerance symbols

Symbol	Characteristic	Type
	Position	Location
	Concentricity or coaxiality	Location
	Symmetry	Location
	Parallelism	Orientation
	Perpendicularity	Orientation
	Angularity	Orientation
	Cylindricity	Form
	Flatness	Form
	Circularity or roundness	Form
	Straightness	Form
	Profile of a surface	Profile
	Profile of a line	Profile
	Circular runout	Runout
	Total runout	Runout

The second compartment contains the tolerance value. When appropriate, the tolerance value is preceded by a diameter symbol and followed by a material condition symbol. The material conditions apply to features that can vary in size. The material condition symbols and their meanings are shown in the following table.

Material conditions

bol	Definition
Ⓜ	At maximum material condition (MMC), a feature contains the maximum amount of material stated in the limits.
Ⓛ	At least material condition (LMC), a feature contains the minimum amount of material stated in the limits.
Ⓢ	Regardless of feature size (RFS) indicates that the feature can be any size within the stated limits.

The tolerance value can then be followed by primary, secondary, and tertiary datum reference letters, along with the material conditions of each datum. Datum reference letters are generally used as reference tolerances to one of up to three perpendicular planes from which a measurement is made, although datum reference letters can also indicate an exact point or axis.

A Geometric characteristic symbol.
B Diameter symbol.
C Tolerance value.
D Material condition symbol.
E Datum reference.

When two tolerances apply to the same geometry, you can also add a composite tolerance consisting of a primary tolerance value followed by a secondary tolerance value. To make a tolerance even more specific, it can also contain a projected tolerance consisting of a height value followed by a projected tolerance symbol. For example, you can use a projected tolerance to indicate the perpendicularity of an embedded part.

To add a geometric tolerance

1   Do one of the following:
•   Type *tolerance* and then press Enter.

2   To display the geometric tolerance symbols, on the first line, click the Sym box.

3   Click to select a geometric tolerance symbol.

4   Under Tolerance 1, click the Dia box to add a diameter symbol.

5   In the field, type the first tolerance value.

6   To display the material condition symbols, click the M.C. box.

7   Click to select a material condition.

8   Under Tolerance 2, repeat steps 4 through 7 to add a secondary tolerance value, if appropriate.

9   Under Datum 1, type the primary datum reference letter.

10   To display the material condition symbols, click the M.C. box.

11   Click to select a material condition.

12   Repeat steps 9 through 11 to add secondary and tertiary datum, if appropriate.

13   In the second row, repeat steps 2 through 12 to add composite tolerances, if appropriate.

14   In the Height box, type a projected tolerance zone height value, if appropriate.

15   To insert a projected tolerance zone symbol, click the Projected Tolerance Zone box, if appropriate.

16   Click OK.

17   In the drawing, specify the location of the feature frame.

A   Use this row to include composite tolerances.

B   Type a projected tolerance height value.

C   Type a datum identifier.

D   Click to include a projected tolerance symbol.

E   Click to select the material condition symbol for tertiary data.

F   Type the tertiary datum reference letter.

G   Click to select the material condition symbol for secondary datum.

H   Type the secondary datum reference letter.

I   Click to select the material condition symbol for primary datum.

J   Type the primary datum reference letter.

K   Click to select the material condition symbol for the second tolerance value.

L   Type the second tolerance value.

M   Click to include a diameter symbol for the second tolerance value.

N   Click to select the material condition symbol for the first tolerance value.

O   Type the first tolerance value.

P   Click to include a diameter symbol for the first tolerance value.

Q   Click to select a geometric tolerance symbol.

Controlling dimension tolerance

You can create dimensions as either tolerance or limits dimensions. When creating a tolerance dimension, you can control the upper and lower tolerance limits as well as the number of decimal places of the dimension text. The image tile on the right side of the Dimension Settings dialog box shows the appearance of tolerance and limits dimensions based on the current dimension style settings.

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Tolerance dimension.   Limits dimension.

To create a tolerance dimension

1   Do one of the following:
    • Choose Dimension > Dimension Settings.
    • On the Settings toolbar, click the Dimension Settings tool ().
    • Type *setdim* and then press Enter.

2   Click the Tolerance tab.

3   Select the Append Dimension Tolerances check box.

4   Type or select the lower tolerance limit.

5   Type or select the upper tolerance limit.

6   Click OK.

7   Insert the dimension.

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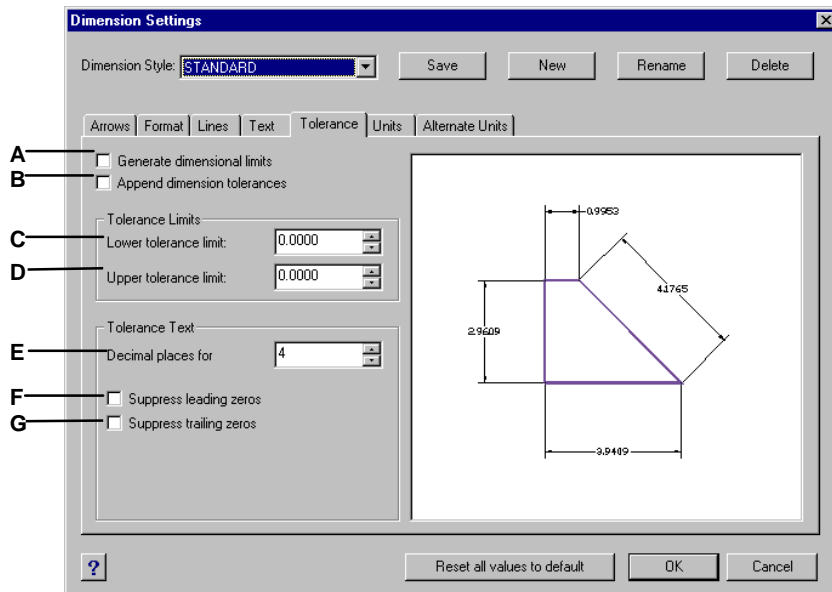
A   Select to insert dimensions as upper and lowertolerance limits.

B   Select to include plus and minus tolerance values with the dimension text.

C   Type or select the minimum tolerance or lower limit value.

D   Type or select the maximum tolerance or upper limit value.
E   Type or select the number of decimal places displayed in limits or tolerance dimensions.

F   Select to prevent the inclusion of leading zeros or the inclusion of feet in dimension limits when the dimension is less than one foot.

G   Select to prevent the inclusion of trailing zeros or the inclusion of inches in dimension limits when the number of inches is zero.

Controlling alternate dimension units

You can include alternate dimensions in addition to the primary dimension text. You can also determine the appearance and format of the alternate dimensions, including the scale factor applied to generate alternate dimensions. The image tile on the right side of the Dimension Settings dialog box shows the appearance of the dimensions based on the current dimension style settings.

Alternate dimension created using a scale factor of 25.4, with an appended suffix.
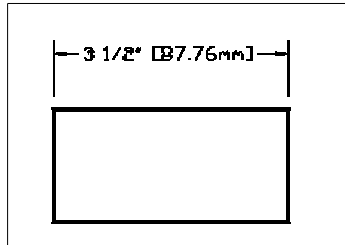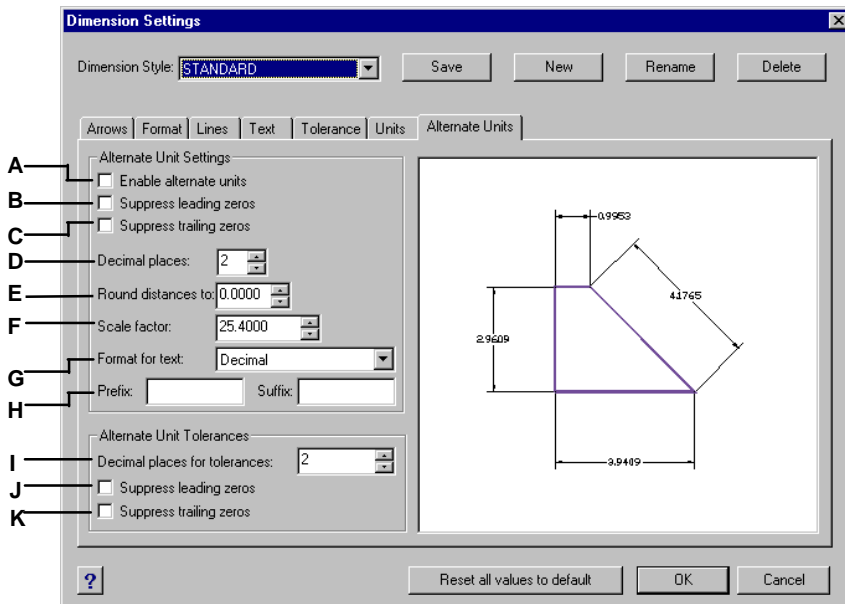
To create an alternate dimension

1   Do one of the following:
- Choose Dimension > Dimension Settings.
- On the Settings toolbar, click the Dimension Settings tool ().
- Type *setdim* and then press Enter.

2   Click the Alternate Units tab.

3   Select the Enable Alternate Units check box.

4   Type or select the scale factor.

5   In the Suffix field, type a suffix to be appended to the alternate dimension.

6   Click OK.

7   Insert the dimension.

A Select to include alternate dimensions.

B Select to prevent the inclusion of leading zeros or the inclusion of feet in alternate dimensions when the dimension is less than one foot.

C Select to prevent the inclusion of trailing zeros or the inclusion of inches in alternate dimensions when the number of inches is zero.

D Type or select the number of decimal places displayed in alternate dimensions.

E Type or select any rounding for alternate dimensions.

F Type or select the scale factor applied to measured dimensions to generate the alternate dimensions.

G Click to select the format for alternatedimensions.

H Type a prefix or suffix to be appended to alternate dimensions.

I Type or select the number of decimal places displayed in limits or tolerances included as part of alternate dimensions.

J Select to prevent the inclusion of leading zeros or the inclusion of feet in limits or tolerances included as part of alternate dimensions.

K Select to prevent the inclusion of trailing zeros or the inclusion of inches in limits or tolerances included as part of alternate dimensions.

Quick Dimension

To make a quick dimension

1 Do one of the following:

Choose Dimension→Quick Dimension

Type QDIM and then press *Enter*

2 Command Function

Use QDIM to quickly create or edit a series of dimension. This command is very useful when creating a series of baseline or continuous dimension markings, or mark the dimensions of a series of circles or arc.

3 Parameters:

After activating QDIM, BtoCAD will inform "select geometry to dimension", when finish the selection, system will inform "[Continuous/Staggered/Baseline/Radius/Diameter/datumPoint/Edit]<Continuous>: -"Liberate as follows:

Continuous: Mark a series of continuous dimensions.

Staggered: Mark a series of staggered dimensions.

Baseline: Mark a series of baseline dimensions.

Radius: Mark a series of radius dimensions.

Diameter: Mark a series of diameter dimensions.

DatumPoint: Set the new datumPoint for the baseline and coordinate dimension.

Edit: Edit a series of dimensions.