Dimensioning a Drawing

OVERVIEW
Dimensioning a drawing is an important step in creating a technical drawing. AutoCAD provides numerous commands and variables to assist in the creation of these dimensions. Dimensioning a drawing can be a very intimidating process. With practice and knowledge of dimensioning rules, AutoCAD can assist in creating correct and accurate dimensions.

OBJECTIVES
• Understand dimensions and their use.
• Understand the importance of associative dimensioning.
• Create and use dimension styles and style families.
• Modify existing dimensions.
• Create oblique dimensions.
• Create leaders.

INTRODUCTION
Many drawings include dimensions. Dimensions display measurement information about the object that is being described in the drawing. A sample of a drawing with dimensions can be found in Figure 13.1. Note that there are enough dimensions to accurately describe the object.

Dimensions can be used to assist in the creation of a drawing. Dimensions can also be used in the manufacture of a part or in the construction of a building. If dimensions are not accurate or if they are not placed appropriately, it will be very difficult to read and obtain the information needed from the drawing during manufacture or construction.

This unit will go over the commands used in AutoCAD to create dimensions. It will also provide a brief introduction to dimensions themselves. While placement of dimensions is very important, it is beyond the scope of this course to describe how to properly dimension a drawing. Depending on the discipline of study, there may be certain standards that have to be adhered to when locating dimensions on a drawing. Two of the most common standards, ANSI and AIA, will be discussed.

Figure 13.1
Dimensions can be used to visualize the size of an object.
As you begin to create dimensions you will find that they are only as accurate as the drawing being dimensioned. If you need a precision of .0001, for example, you must create the object with the same precision before AutoCAD can dimension it accurately. You cannot assume that because AutoCAD displays a dimension incorrectly that it is the software's fault. In most cases the problem is with the geometry of the drawing or incorrect placement of the dimension.

This unit will also discuss how to modify the placement of a dimension and its appearance. Avoid modifying the dimension text to make it appear correct. Concentrate on correcting the object, not the dimension itself. If you attempt to change only the dimension, you will often find that other dimensions will not add up. This can begin a chain reaction that can take longer to fix than simply correcting the objects themselves.

Parts of a Dimension
In order to make changes to the different features that comprise dimension and affect its appearance, it is important to know basic dimension terminology. Figure 13.2 displays the basic components that make up a dimension. After reviewing the figure, the rest of this section will explain each feature’s use.

- **Terminator**
  Terminates the dimension line where it intersects with the extension line. Used to assist in the visual display of the dimension. A terminator can be an arrow, tick mark, dot, or custom-made object.

- **Dimension Text**
  Indicates the measurement of the distance specified by the dimension line. The dimension text can be placed on top of the dimension line or within the dimension line.

- **Dimension Line**
  Displays the direction and distance of the measurement.

- **Extension Line**
  Extends from the measurement to the dimension line to assist in guiding the eye to the proper dimension.

- **Leader**
  Used to point to a feature that requires further annotation beyond dimensioning. It is composed of an extension line and a terminator, typically an arrow.

- **Center Lines**
  The intersection of the horizontal and vertical line represents the center of a circle or arc.

Figure 13.2
The basic parts of a dimension.
Dimensioning Standards
Two popular American associations, the American National Standards Institute (ANSI) and the American Institute of Architects (AIA), provide guidance on how dimensions are to be used and represented. For international drawings the International Standards Organization (ISO) standard is commonly used. Each organization provides standard guidelines for the creation of technical drawings in their respective field. ANSI and ISO provide guidance for mechanical drawings, and AIA for architectural and interior design drawings. By comparing the standards the appearance of the dimension will differ. Figure 13.3 shows the differences between AIA and ANSI dimensions.

Fortunately, the terminology for the parts of the dimensions are the same. Only the appearance changes. Note that the differences in appearance are with the terminators and the location of the dimension text. The extension lines themselves do not change.

**Figure 13.3**
Examples of AIA and ANSI dimensions.

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**Associative and Non Associative Dimensions**

By default dimensions created in AutoCAD are associative. Associative dimensions are complex objects that can change and reflect modifications made to the objects they describe. For instance, you create a line four units long and place a dimension on that object. Next use the stretch command to modify the length of the line to six units. The dimension will automatically change to reflect the new length of the line as shown in Figure 13.4. No moving, erasing or recreating the dimension is needed.

Associative dimensions also allow other features as listed. Each feature will be discussed later in this unit.

- **Dimension styles**
  Similar to text styles, they can be created to modify the appearance of the dimensions and to quickly change that appearance.

- **Grips**
  Can be used to modify the dimension.

**Figure 13.4**
Changing an associative dimension of 4 units to 6 units using the stretch command.

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Associative dimensions can be changed to nonassociative dimensions. To change an associative dimension to a nonassociative dimension use the explode command. This will nonassociate the dimension with an object, as well as break the dimension into individual components instead of one object.

While associative dimensions support most objects they do not support the following:

- Hatching
- Images inserted in to a drawing
- Underlays
- Objects created with the Multiline function.

If a dimension has lost association through exploding it or the geometry it was associated with was modified to the extent the dimension no long is associated with it, the DIMREASSOCIATE command can be used. DIMREASSOCIATE can be used to reassociate dimension. After reassociation the DIMREGEN command may be needed to updated the association.

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While associative dimensions are very powerful and provide a lot of flexibility there will be times you need to explode a dimension and modify it. Be advised it is up to the CAD operator to ensure a drawing is dimensioned correctly. While it is possible to set up dimension styles meeting the ANSI, AIA and ISO standards, AutoCAD will not correctly apply the dimension to a drawing. Just because AutoCAD created a dimension a specific way does not mean it is correct. The CAD operator must be familiar with the dimension style being used, and ensure the object is dimensioned correctly.

Dimension Styles and Their Families

Modifying the style of a dimension and creating a new dimension style is done with the Dimension Style Manager dialog box found in Figure 13.5. To access the Dimension Style Manager dialog box use one of the following procedures:

- Select the small down arrow to the right of the word Dimensions in the Dimensions/Annotate ribbon.
- Select Dimension Style from the Dimension pull-down menu
- Type `dimstyle` at the Command: prompt.

Once the command is selected, the Dimension Style Manager dialog box will be displayed. The components of the dialog box are described below:

- **Styles**
  This text window displays the various names of the dimension styles that are available.

- **List**
  Using this drop-down menu, you can choose to list all styles that have been created and are available, or a list of only those that are in use in the Styles text window.

- **Preview of**
  This image tile allows you to see an illustration of the current dimension style. In figure 13.5, the settings for the STANDARD dimension style are being displayed.

- **Description**
  Provides a brief list of the various dimension style settings. This information is provided as a reminder of the type of units and settings that have been selected for that style. This description is not all inclusive but rather gives a snapshot of the current dimension style.

- **Set Current**
  This button sets the selected style highlighted in the Styles text box as the current style. In other words, any dimension that is created will inherit the attributes of that style.

- **New**
  This button displays the Create New Dimension dialog box which will allow you to create a new dimension style.
- **Modify**
  This button displays the Modify Dimension Style dialog box which will allow you to make changes to the current dimension style.

- **Override**
  This button displays the Override Current Style dialog box.

- **Compare**
  This button displays the Compare Dimension Styles dialog box.

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**Figure 13.5**
The Dimension Style Manager dialog box.

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Creating a New Dimension Style

To create a new dimension style, you will use the Create New Dimension Style dialog box as shown in Figure 13.6. This dialog box can be accessed using the New button in the Dimension Style dialog box as shown in Figure 13.5 earlier.

Once the Create New Dimension Style dialog box is displayed, the three following options will be presented.

- **New Style Name**
  Use this text edit box to name the new style to be created. You are allowed up to 256 characters, and spaces are allowed.

- **Start With**
  This drop-down menu displays a list of current styles. Selecting one of the current styles will allow the creation of a new style using a currently existing style's settings.

- **Use for**
  Use this drop-down menu to select the type of dimension for which to apply this style. For instance, you may want particular settings for linear dimensions, but completely different settings for radial dimensions.

Once these options have been selected, selecting the Continue button will accept the changes and the Dimension Style Manager dialog box will be displayed. Now it is time to modify the dimension settings for the newly created dimension style.

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**Figure 13.6**
The Create New Dimension Style dialog box.
The Lines tab contains settings that govern the Extension Line and Dimension Line of a dimension as shown in Figure 13.7. Changing the Color, Linetype or Lineweight in either the Dimension lines or Extension lines area will override any color, linetype or lineweight assigned to the current layer the dimension is created on, essentially meaning you can assign different colors, linetypes or lineweights to parts of a dimension while keeping it on the same layer.

**Dimension Lines:**
- **Color**
  Modifies the color of a dimension line.
- **Linetype**
  Modifies the linetype of a dimension line.
- **Lineweight**
  Modifies the lineweight of a dimension line. Lineweights are presented in mm, and selecting by block or by layer will modify the lineweight to match accordingly.
- **Extend beyond ticks**
  Determines the distance that the dimension line will travel beyond the terminator. The normal value is 0.
- **Baseline spacing**
  Specifies the distance that the next dimension line in a baseline dimension system will be placed from the previous line as shown in Figure 13.8.
- **Suppress**
  Select either the Dim line 1 or Dim line 2 check box to suppress the beginning or ending portion of the dimension line, as shown in Figure 13.9.
Extension Lines:
- Color
  Modifies the color of the extension lines.
- Linetype ext line 1
  Modifies the linetype of the first extension line.
- Linetype
  Modifies the linetype of the second extension line.
- Lineweight
  Modifies the lineweight of the extension lines.
- Suppress
  Select either the Ext Line I or Ext Line 2 check box to suppress the beginning or ending extension line, as shown in Figure 13.10.
- Extend beyond dim lines
  Determines the distance that the extension lines will travel beyond the dimension lines. The normal value is .180.
- Offset from origin.
  Modifies the extension line offset from the point selected to define the beginning and end of the dimension. An example is shown in Figure 13.11.

Figure 13.10
The effects of extension line suppression on a horizontal dimension.

Figure 13.11
The extension line offset.

Symbols and Arrows Tab
The Symbols and Arrows tab shown in Figure 13.12 allows you to change the type of terminator used at the end of the first and second dimension lines.

Arrowheads
- First
  This drop-down menu is used to select the type of arrowhead (terminator) on the first dimension line.
- Second
  Use this drop-down menu to select the arrowhead by name for the second arrowhead created on a dimension. This selection defaults to the 1st option, and it is not necessary to modify this setting if both arrowheads are to be the same.
- Leader
  This drop-down menu specifies the type of arrowhead to use for a leader.
- Arrow size
  Use this text edit scroll box to modify the size of the arrowheads. Default is .1800

Center Marks
- Type
  These check marks allow you to specify the type of mark to use when center marks are automatically placed using AutoCAD. The options are Mark (cross with breaks), Line (cross with no breaks), and None (do not place marks).
- Size.
  Specifies the size of the mark and the distance that the center mark extends past the arc or circle being marked. Default is .090.
Tutorial 13.1: Creating a Dimension Style and Changing the Dimension Settings

In this tutorial you will create a new dimension style called TUTORIAL and modify the values in the Create New Dimension Style dialog box.

1. Click the Application menu, then select Open/Drawing. Open drawing Tutorial13.1.dwg. This will open the drawing shown in Figure 13.13.

2. Select the Format/Dimension Style pull-down menu to access the Dimension Style Manager dialog box.

3. Select the New button to access the Create New Dimension Style dialog box shown. You will create a new dimension style called Tutorial, starting with the Standard style. Your Create New Dimension Style dialog box should look like Figure 13.14. Press the Continue button.

4. In the Lines tab change the Baseline spacing to .375 and the Extend beyond dim lines to .125. Your Lines tab should look like Figure 13.15.

5. In the Symbols and Arrows tab change the Arrow size to .125, and the Center marks to None. Your Symbols and Arrows tab should look like Figure 13.16.
Figure 13.15
Modifying the Lines tab.

Figure 13.16
Modifying the Symbols and Arrows tab.

Text Tab
To display the Text tab, shown in Figure 13.17, simply use the arrow and click on the Text tab. Again, any changes to the settings are reflected on the dialog box image tile.

Figure 13.17
The Text tab.
Text Appearance

- **Text style**
  Used to select the style of text to use within a dimension. A style may have previously defined, or the ... button can be used to display the Text Style dialog box to define a new style. Refer to Unit 7 for information on creating a text style.

- **Text Color**
  Modifies the color of the text.

- **Fill Color**
  This will allow you to set a color for the text background in dimensions.

- **Text height**
  Modifies the height of the text.

- **Fraction height scale**
  Modifies the scale of fractions based on the overall text height specified previously.

- **Draw frame around text**
  Use this check box to specify whether or not a box should be placed around the dimension text.

Text Placement

- **Vertical**
  Specifies the vertical placement of the text in relationship to the dimension line. Options include centered, above, outside, and Japanese Industrial Standard (JIS). The JIS ensures that the dimension text is always aligned to be on top of the dimension line no matter at what angle the dimension is created.

- **Horizontal**
  Specifies the horizontal placement of the text in relationship to the dimension line. Examples of the options are displayed in Figure 13.16.

- **View Direction**
  Options are Left-to-Right (default) which places the text to enable reading from left to right. You can also change the viewing direction from Right to Left.

- **Offset from dim line**
  Specifies the distance that the text is offset from the dimension line. Default is .090

Text Alignment

Select the radio box that best serves the need of the particular drawing being created. Examples of Horizontal and Aligned with dimension line are displayed in Figure 13.17. The International Standards Organization (ISO) option aligns the text with the dimension line when text is inside the extension lines, but aligns it horizontally when the text is outside the extension lines.

**Figure 13.18**
The various horizontal options for dimension text.

**Figure 13.19**
Text can be aligned or horizontal with the dimension line.
**Tutorial 13.2: Changing the Text Format**

This tutorial continues from the previous tutorial. In this tutorial you will modify the text format.

1. Select the Text tab. The Text tab settings will be displayed as shown in Figure 13.20.

2. Change the Text Height to .125.

3. Change the Offset from dim line to .0625


The next tutorial will continue from here.

**Figure 13.20**
Modifying the Text tab.

**Fit Tab**

The Fit tab, shown in Figure 13.21, can be used to set the overall scale of the dimension style. It can also be used to set different fit options in the event the dimension text and arrows does not fit correctly between the dimension lines.

- **Fit Options**
  Specifies how text should be placed in the event that there is not enough room between the extension lines. Normally you should leave this setting at the default Either text or arrow option. This will allow AutoCAD to determine the best fit for the arrows and text.

- **Text Placement**
  Specifies where text should be placed if it cannot fit in its default position because of space constraints.

- **Scale for Dimension Features**
  Modifies the overall scale of the dimensions based on either real units or paper space units. For example, a scale of 2 would modify all dimension attributes to be double their specified values.

- **Fine Tuning**
  Two additional specifications of how text and extension lines should be placed. Place text manually when dimensioning allows you to specify exactly where the text will be placed. Always draw dim line between ext lines does just what it says.

**Figure 13.21**
The Fit settings.
Primary Units Tab
This tab allows you to set the units and precision for the dimension style. Both linear and angular dimensions are covered here as shown in Figure 13.22

Linear Dimensions:
- **Unit format**
  Specifies the type of units to be displayed within the text. Note that this is NOT the same as the units that were selected during drawing setup. They must be changed in order for the proper format to appear within the dimension.
- **Precision**
  Specifies the precision with which the dimension should be displayed. The format of the precision is based on the type of units selected.
- **Fraction format**
  Specifies how fractions should be displayed: Horizontal, Diagonal, and not stacked.
- **Decimal separator**
  Specifies how decimals should be separated from whole numbers. Options include a period, comma, and a space.
- **Round off**
  Specifies the precision with which numbers should be rounded.
- **Prefix**
  Any characters entered within this text edit box will appear at the beginning of every dimension.
- **Suffix**
  Any characters entered within this text edit box will appear at the end of every dimension. This can be handy if you want to create a dimension style that reads in inches only. To do this select Fractional as the Unit format, then enter a double quote (") for inches in the suffix box.
- **Scale Factor**
  Sets a scale factor for all linear dimensions. AutoCAD multiplies the dimension measurement by the value entered. For example, if a value of 2 is entered, AutoCAD displays a 1-inch dimension as 2 inches. This changes what the dimension reads and does NOT change the geometry.
- **Apply to layout dimensions only**
  Applies the linear scale value only to dimensions created in the layout tab.
- **Zero Suppression**
  Select the appropriate check box to suppress Leading, Trailing, 0 feet, and 0 Inches when they appear within a dimension. The last two options are only available when architectural units have been selected.

Angular Dimensions:
Specifies the units, precision, and zero suppression for all angular dimensions.

Figure 13.22
The Primary Units tab is used to control the type of units used when dimensioning.
Alternate Units Tab
This tab is displayed in Figure 13.23. The use of alternate units allows two dimension values to be displayed on one dimension. For example, a dimension could contain both a decimal and metric value.

Figure 13.23
The Alternate Units tab.

Tolerances Tab
Used primarily in mechanical drafting, tolerances allow a part to be dimensioned in a series of limits or tolerances. As this is specific to the mechanical discipline, the use of this tab (as displayed in Figure 13.23) can be found in the accompanying mechanical specialty.

Figure 13.24
The Tolerances tab.
Tutorial 13.3: Changing the Fit and Units

This tutorial continues from the previous tutorial. In this tutorial you will complete the dimension style by setting the fit and units.

1. Select the Fit tab. Change the Use overall scale to 2. The Fit tab should look like Figure 13.25.

2. Select the Primary Units tab. Change the Precision to three decimal places. Check Leading in the Zero suppression box. The Primary Units tab should look like Figure 13.26.

3. Select OK then Close to close the Dimension Style dialog box. You should now have the TUTORIAL dimension style created. You will use this style in the next tutorial where you will dimension the object.

Figure 13.25
Modifying the Fit tab.

Figure 13.26
Modifying the Primary Units tab.
After creating a dimension style the next step is to create dimensions. Linear and radial dimensions are the most common dimensions created. Linear dimensions consist both of vertical and horizontal dimensions, as shown in Figure 13.26.

Very similar to linear dimensions are aligned dimensions. These are dimensions that are not vertical or horizontal. They are also shown in Figure 13.27.

Not all geometry on a drawing will be linear. Many drawings include arcs and circles to describe fillets or holes or even a curved wall. These objects need to be dimensioned if the object is going to be manufactured or constructed. Because of this, AutoCAD provides radial dimensions. A radial dimension is shown in Figure 13.27 also. Let’s begin this section with a study of basic horizontal and vertical, or linear dimensions.

**Linear Dimensions**

Linear dimensions are used to create dimensions aligned with vertical and horizontal objects. The command to create horizontal and vertical dimensions can be accessed using one of the methods described here:

- Select the Linear Dimension button from the Dimension/Annotate ribbon
- Select Linear from the Dimension pull-down menu
- Type `dimlinear` at the Command: prompt.

Once activated, you will see the following prompt:

Specify first extension line origin or <select object>:

At this point you can select the first extension line origin. Pressing Enter will access the Select object to dimension prompt, allowing you to select an object to dimension. When you select a line, the first and second extension line origins will be selected for you automatically, based on the start and endpoint of the object. Using the select object option has many advantages. It will create an accurate dimension as AutoCAD locates the end points of the object for you. Select objects can also make linear dimensioning very fast as all you need to do is select the object to dimension. You cannot, however, use the object selection method if the total length of a distance to be dimensioned is broken by two or more objects. To use select object the object to be dimensioned must be continuous. If you used select object the dimension is created and the command ends.

If you selected the first extension line origin, the following prompt is displayed:

Specify dimension line location or
[Mtext/Text/Angle/Horizontal/Vertical/Rotated]:

At this prompt you can select the second extension line origin to specify the overall length of the linear dimension.
Other options allow you to modify the content of the dimension text, the angle of the text, or the angle of the dimension line.

- **Mtext**
  Creates or modifies dimension text using Mtext features.
- **Text**
  Modifies the default text.
- **Angle**
  Rotates the text.
- **Horizontal**
  Forces the text to be horizontal.
- **Vertical**
  Forces the text to be vertical.
- **Rotated**
  Rotates the dimension line at an angle you specify.

### Tutorial 13.4: Creating Linear Dimensions

In this tutorial, you will create horizontal and vertical linear dimension on the Tutorial 13.1.dwg file. In order for this tutorial to work correctly, you must first have completed the tutorials prior to this one and setup the TUTORIAL dimension style.

1. Ensure that the Dimensions layer is the current layer.
2. Ensure that Tutorial is the current dimension style shown in the Dimensions ribbon on the Annotate tab as shown in Figure 13.28
3. Select the Linear button on the Dimensions tab of the Annotate ribbon.
4. At the prompt press the Enter key. The cursor will change to pick box. Select the horizontal line in the top view as shown in Figure 13.29.
5. Locate the dimension approximately .75 units from the line being dimensioned. The Snap increment for this tutorial drawing is set to .25.

Next you will place a horizontal dimension. Make sure the ENDpoint object snap is activated to ensure you select the objects accurately.

1. Select the Linear button on the Dimensions tab of the Annotate ribbon.

   **Specify first extension line origin:**
   Select the endpoint of the center line in the front view as shown in Figure 13.29

   **Specify second extension line origin:**
   Select the bottom left corner of the front view as shown in Figure 13.29

2. Locate the dimension approximately .75 units from the line being dimensioned.

   You should now have a horizontal and vertical dimension as shown in Figure 13.30
Aligned Dimensions

Aligned dimensions are identical to linear dimensions in appearance. The only difference is the dimension line orientation. Aligned dimensions are typically slanted at an angle. Creating aligned dimensions is very similar to creating linear dimensions. Use one of the following methods to execute the command:

- Select the Aligned Dimension button from the Dimension/Annotate ribbon. The dimension buttons are located in a drop-down menu on the Dimensions tile on the Annotate ribbon.
- Select Aligned from the Dimension pull-down menu
- Type dimaligned at the Command: prompt.

Once the command is entered, the procedure for creating aligned dimension is the same as for creating a linear dimension.

Tutorial 13.5: Creating Aligned Dimensions

This tutorial continues from the previous tutorial. In this tutorial, you will create an aligned dimension on the top view of the tutorial drawing.

1. Ensure that the Ortho is off.
2. Select the Aligned button on the Dimensions tile of the Annotate ribbon.
3. At the prompt press the Enter key. The cursor will change to pick box. Select the angled line in the top view as shown in Figure 13.31.
4. Locate the dimension approximately .75 units from the line being dimensioned. An aligned dimension will be placed.

Figure 13.30
A horizontal and vertical dimension created on the tutorial drawing in this figure.

Figure 13.31
Select the angled line to create an aligned dimension.
Continued Dimension Strings

It is common to find a string of continued dimensions on a drawing since it is very rare that a single dimension will be able to describe the complete profile of an object. AutoCAD provides two commands that allow you to continue your dimensions using previously created dimensions as the base point; Continue and Baseline dimensions.

Continued Dimensions

Continued dimensions are multiple dimensions placed end to end. The dimension lines of the previous dimension and the new dimension will be aligned. Before creating a continued dimension, ensure that a previous, or base dimension has been created. An example of a continued dimension is shown in Figure 13.32. After creating a base dimension create a continued dimension using one of the following procedures:

- Select the Continue button from the Dimension/Annotate ribbon
- Select Continue from the Dimension pull-down menu
- Type `dimcontinue` at the Command: prompt.

Once activated, you will see the following prompt:

Select continued dimension:

Select the dimension you want to continue. This dimension must already exist. By default the last dimension created will be used as the base dimension. Once the continued dimension is selected the following prompt appears

Specify a second extension line origin or [Undo/Select] <Select>:

At this prompt you can select the second extension line origin. You can continue to select endpoints for the continued dimension as needed. To end the command press the Enter key.

Figure 13.32
Continued dimensions are multiple dimensions placed end to end.
This tutorial continues from the previous tutorial. In this tutorial, you will create a continued dimension on the front view of the tutorial drawing. As continued dimensions work off of a base dimension you must first create a linear dimension.

1. Select the Linear button on the Dimensions tab of the Annotate ribbon.

2. At the prompt press the Enter key. The cursor will change to pick box. Select the horizontal line in the front view as shown in Figure 13.33.

3. Locate the dimension approximately .75 units from the line being dimensioned.

Next you will create two continued dimensions. Make sure the ENDpoint object snap is activated to ensure you select the objects accurately.

1. Select the Continue button on the Dimensions tab of the Annotate ribbon.

   Specify a second extension line origin
   Select First point as shown in Figure 13.34
   Specify second extension line origin:
   Select Second point as shown in Figure 13.34
   Specify second extension line origin:
   Press Enter twice to end the command.

   Your drawing should now look like Figure 13.35
Baseline Dimensions

Baseline dimensions are multiple dimensions measured from the same location. The dimension line of the baseline dimension is placed away from the previous dimension at the distance specified in the Baseline Spacing text edit box of the Dimension Style dialog box. By default, the origin of the last linear or angular dimension created is used as the first extension line for the new baseline dimension. To invoke Baseline Dimensions, use one of the following procedures:

- Select the Baseline button from the Dimension/Annotate ribbon. The Baseline button is below the Continue button on the Dimension/Annotate ribbon
- Select Baseline from the Dimension pull-down menu
- Type `dimbaseline` at the Command: prompt.

Once activated the following prompt appears:

**Specify a second extension line origin or [Undo/Select] <Select>:**

Keep in mind the baseline dimensions use first extension line of the original dimension for the new baseline dimension. To create a baseline dimension you need to know which extension line was created first. You can continue creating baseline dimensions by selecting the second extension line origin.

**Figure 13.36**

Baseline dimensions are multiple dimensions measured from the same location.
This tutorial continues from the previous tutorial. In this tutorial, you will create baseline dimensions on the front view of the tutorial drawing. As continued dimensions work off of a base dimension you must first create a linear dimension.

1. Select the Linear button on the Dimensions tab of the Annotate ribbon.

Specify first extension line origin:
Select the first point shown in Figure 13.37

Specify second extension line origin:
Select the second point as shown in Figure 13.37

3. Locate the dimension approximately .75 units from the line being dimensioned.

Next you will create two baseline dimensions.

1. Select the Baseline button on the Dimensions tab of the Annotate ribbon.

Specify a second extension line origin
Select First point as shown in Figure 13.38
Specify second extension line origin:
Select Second point as shown in Figure 13.38
Specify second extension line origin:
Press Enter twice to end the command.

Your drawing should now look like Figure 13.39

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**Figure 13.37**
Selecting the first and second points of the base dimension.

**Figure 13.38**
Selecting the points of the baseline dimension.

**Figure 13.39**
The completed drawing with linear, aligned, continued and baseline dimensions.
Radial Dimensions

Not every object located on a drawing is linear. Many objects you find on a drawing are circular. Normally a diameter is used to describe a circle. A circle on a drawing can represent a cylinder or hole. The term radius is used to describe a partial circle or an arc. An arc may represent a fillet or a curve in a driveway on a site plan. AutoCAD provides the Radius and Diameter commands to create these dimensions. These commands will automatically add the appropriate radius and diameter symbol. Figure 13.40 shows both radius and diameter dimensions.

To enter the radius command, use one of the following procedures listed:

- Select the Radius button from the Dimension/Annotate ribbon.
- Select Radius from the Dimension pull-down menu.
- At the Command: prompt, type `dimradius`.

To enter the diameter command use one of the following procedures:

- Select the Diameter button from the Dimension/Annotate ribbon.
- Select Diameter from the Dimension pull-down menu, or
- At the Command: prompt, type `dimdiameter`.

Once either command has been selected, you must choose an arc or a circle. Depending on your dimension style settings, you may or may not have to place the location of the dimension text.

![Figure 13.40](image)

Examples of radius, diameter, and center lines.

**Tutorial 13.8: Adding Radius and Diameter Dimensions**

In this tutorial you will create a radius and diameter dimension on the right-side view of the tutorial drawing.

1. Select the Radius button from the Dimension/Annotate ribbon.
   
   Select the arc as shown in Figure 13.41. Place the radial dimension on the right side of the object.

2. Select the Diameter button from the Dimension/Annotate ribbon.
   
   Select the circle as shown in Figure 13.41. Place the diameter dimension on the right side of the object.

3. Your radius and diameter dimensions should look like Figure 13.42.

![Figure 13.41](image)

Creating Radius and Diameter dimensions.

![Figure 13.42](image)

The completed Radius and Diameter dimensions.
Just as the world is not composed of straight lines, it is also not composed of 90° angles. Many objects contain angles that are greater than or less than a right angle. Consider an octagonal gazebo or a pyramid. Each of these objects contains angles other than right. This section will show you how to create dimensions that specify an angular dimension.

Creating Angular Dimensions
Before creating an angular dimensions there must be an angle defined in a drawing. You will typically not create an angular dimension on two objects that form a right angle. If no angular dimension is given the angle is assumed to be a right angle. To create angular dimensions, use the dimangular command. Use one of the following procedures for starting the command:

- Select the Angular button from the Dimension/Annotate ribbon.
- Select Angular from the Dimension pull-down menu.
- Type dimangular at the Command: prompt.

Once the command is entered, you will be asked to select the arc, circle, or line that defines the angle to receive the dimension. There are four ways to define angles using AutoCAD:

- **Two nonparallel lines**
  Select any two lines that are not parallel. Once selected, the angular dimension text is placed. The text will generally fall within the angular dimension unless there is not enough room for the text. Moving the text to different areas can lead to completely different angular measurements. Experiment with this command on two nonparallel lines to see the different values and orientations that this selection will create.

- **Arcs**
  Selecting an arc will cause the angular measurement to measure the angle between the start and ending points of the arc. The placement of the text will once again affect the appearance of the angular dimension.

- **Circle and a defined point**
  The point at which you select the circle becomes the start point for the arc that is going to be defined. Next, pick a point on the circle to define the ending point for the arc. This option is identical to the arc, except that you are starting with a circle and specifying a portion of that circle to be an arc.

- **Three defined points**
  Use this method to select three points that define an angle. To use this option, press Enter at the Select arc, circle, line, or <specify vertex>: prompt. Then select the first point to define the start point of the angle. The next point you select will be the vertex of the arc and, finally, the third point will be the end point for the arc. Figure 13.43 displays this and the previous three options for selecting objects to create an Angular Dimension.

**Figure 13.43**
Various methods that can be used to create angular dimensions.
Tutorial 13.9: Adding Angular Dimensions

In this tutorial you will create an angular dimension on the top view of the tutorial drawing.

1. Select the Angular button from the Dimensions/Annotate ribbon.

   Select arc, circle, line, or <specify vertex>:
   Select the first line as shown in Figure 13.44.
   Select second line:
   Select the second line as shown in Figure 13.44.

2. Place the dimension as shown in Figure 13.45.

EDITING DIMENSIONS

Creating dimensions is one of the advantages of using CAD software over the traditional manual drawing method. There are times, however, when you want to change the dimension text or its location, suppress extension lines, or even change the type of terminator. You do not need to erase the dimension and then recreate it. You can simply modify it. This section will discuss several methods that can be used to edit an existing dimension.

**Relocating and Rotating Text**

There are several commands that can be used to change the location of text on existing dimensions. You can left, right and center justify text along with rotating the text. These commands can be found on the Dimensions drop-down menu on the Dimensions tile on the Annotate ribbon. Examples of the different dimension text editing functions are shown in Figure 13.46. The left, right and center justification options only works with linear, radius and diameter dimensions.

- **Text Angle**
  Used to rotate the dimension text to a specified angle. The text angle is measured from the X axis.
- **Left Justify**
  Justifies dimension text to the left.
- **Center Justify**
  Centers dimension text in the middle of the dimension line.
- **Right Justify**
  Justifies dimension text to the right.

**Figure 13.46**

Editing text on an existing dimension.
Modifying Dimensions using Grips
Like other objects, dimensions can be moved, copied, rotated, and stretched. A previous unit was completely devoted to grips and their operations. Grips provide an additional way to modify dimensions. Each type of dimension will have different grip points that can be used to modify specific attributes of a dimension. Using grips on dimensions is not unlike using them for other objects. At the Command: prompt, select a dimension. Grips will appear at key locations that affect the dimension. Select the grip point to modify and move, copy, rotate, or stretch it to another location. Using grips is beneficial for the following situations:

- Change the location of the dimension text.
- Modify the location of the dimension line.
- Modify the gap between extension line and object.
- Relocate the endpoints of the extension lines.

You may find many other uses other than the ones listed above. Experiment with grips and you may find that grips are a very convenient way to edit dimensions.

Modifying Dimension Properties
The Properties dialog box can be used to edit just about all of the settings on a single dimension. To access the properties dialog box first select a single dimension. With one dimension selected right-click and select Properties from the pop-up menu. The Properties dialog box appears as shown in Figure 13.47. By selecting the small + sign you can open up the various tabs and modify many of the currently selected dimension settings.

![Figure 13.47](image)
The properties dialog box can be used to change many settings of a single dimension.

SKILL BUILDER
Editing dimensions can become very complex and time consuming. An alternative to editing the dimension is using the Explode command and breaking the dimension into individual parts. Once the dimension is exploded you can edit it with any of your conventional editing tools.
Leaders are used to place notes on a drawing. These notes can be used to describe an object or feature in a way that dimensions may not be able to describe. Leaders are composed of a terminator, usually an arrowhead, a leader line, landing and leader text as shown in Figure 13.48.

As with dimensions the first step in working with leaders is to define the leader style. This is done in the Multileader Style Manager dialog box shown in Figure 13.49. You can access the dialog box by:

- Select the small down arrow to the right of the word Leaders in the Leaders/Annotate ribbon.
- Select Multileader Style from the Format pull-down menu.
- Type `mleaderstyle` at the Command line.

Once the command is selected, the Multileader Style Manager dialog box will be displayed.

The components of the dialog box are described below.

- **Styles**
  This text window displays the various names of the multileader styles that are available.
- **List**
  Using this drop-down menu, you can choose to list all styles that have been created and are available, or a list of only those that are in use in the Styles text window.
- **Preview of**
  This image tile allows you to see an illustration of the current multileader style. In Figure 13.49, the settings for the STANDARD multileader style are being displayed.
- **Set Current**
  This button sets the selected style highlighted in the Styles text box as the current style. In other words, any multileader that is created will inherit the attributes of that style.
- **New**
  This button displays the Create New Multileader dialog box which will allow you to create a new multileader style.
- **Modify**
  This button displays the Modify Multileader Style dialog box which will allow you to make changes to the current multileader style.
- **Delete**
  This will delete a selected multileader style.
Creating a New Style

To create a new multileader style, you will use the Create New Multileader Style dialog box as shown in Figure 13.50. This dialog box can be accessed using the New button in the Multileader Style Manager dialog box as shown in Figure 13.49 earlier. Once the Create New Multileader Style dialog box is displayed, the three following options will be presented:

- **New Style Name**
  Use this text edit box to name the new style to be created. You are allowed up to 256 characters, and spaces are allowed.

- **Start With**
  This drop-down menu displays a list of current styles. Selecting one of the current styles will allow the creation of a new style using a currently existing style's settings.

Once these options have been selected, selecting the Continue button will accept the changes and the Modify Multileader Style dialog box will be displayed as shown in Figure 13.51. Across the top are three tabs; Leader Format, Leader Structure and Content. Within each tab are various tiles. As the changes are made to the tiles, the image tile will present a visual inspection of the new settings. Let’s take a look at the settings for a newly created multileader style.

**Leader Format**

The Leader Format tab shown in Figure 13.51 governs several of the basic components of the leader.

- **Type**
  Governs the type of leader line. You can select a straight line (default), spline, or no leader line.

- **Color**
  Modifies the color of a leader line.

- **Linetype**
  Modifies the linetype of a leader line.

- **Lineweight**
  Modifies the lineweight of a leader line. Lineweights are presented in mm, and selecting by block or by layer will modify the lineweight to match accordingly.
- **Arrowhead Symbol**  
  Accesses a drop-down menu that allows you to select the arrowhead, or terminator for the multileader.

- **Size**  
  Sets the size for the symbol

- **Leader Break**  
  Controls the settings used when adding a dimension break to a multileader. Figure 13.52 shows a leader and dimension line with a break.

  To apply the break select the Break button on the Dimensions/Annotate ribbon. At the prompt
  
  Select dimension to add/remove break or [Multiple]:  
  
  pick the dimension or leader you want to apply the break to.

  ![Figure 13.52](Image)
  
  *Using the Break function on a dimension and multileader.*

- **Leader Structure**  
  Options in this Leader Structure tab shown in Figure 13.53 allow you to control the number of leader points, landing size, and multileader scale.

  **Constraints**
  
  - **Maximum leader points**  
    Determines how many segments will be in a leader.

  - **First segment angle**  
    Controls the angle for the first leader segment. Setting the first segment angle can insure standardization in your leaders. A common value is 45.

  - **Second segment angle**  
    Controls the angle for the second leader segment. Setting the second angle segment to 0 can be use to ensure the landing is straight.

  **Landing Settings**
  
  - **Automatically include landing**  
    Attaches a horizontal landing to the leader.

  - **Set Landing Distance**  
    Sets a fixed distance for the landing line.

  **Scale**
  
  Specifies the scale for the multileader.

  ![Figure 13.53](Image)
  
  *The Leader Structure tab.*
Content
Defines parameters associated with the multileader content. The Content tab is shown in Figure 13.54

Text Options
- **Default Text**
  Sets default text for the multileader content. Selecting the [...] button returns you to the drawing editor where you can define your own text using the Multiline Text editor. Text placed in this box becomes the default text for all leaders using this style.
- **Text Style**
  Lists the available text styles. A style must be previously defined to appear in the drop-down menu. Selecting the [...] button returns you to the Text Style dialog box where you can define a new text style. Refer to Unit 7 for information on creating text styles.
- **Text Angle**
  Allows you to set a default rotation angle of the multileader text.
- **Text Color**
  Specifies the color of the multileader text.
- **Text Height**
  Specifies the height of the multileader text.
- **Always Left Justify**
  When checked the multileader text is always left justified.
- **Frame Text**
  Frames the multileader text content with a text box.

Leader Connection
**Horizontal Attachment**
Inserts the leader to the left or right of the text content.
- **Left Attachment**
  A drop-down box that allows you to control how the landing line is attached to the multileader text when the TEXT is to the RIGHT of the leader.
- **Right Attachment**
  A drop-down box that allows you to control how the landing line is attached to the multileader text when the TEXT is to the LEFT of the leader.
- **Landing Gap**
  Distance between the landing line and the multileader text.
- **Extend Leader to Text**
  Extends the leader line to the text.

**Vertical Attachment**
Inserts the leader at the top or bottom of the text content.
- **Top Attachment**
  Attaches the leader to the top center of the text.
- **Bottom Attachment**
  Attaches the leader to the bottom of the text.

![Figure 13.54](image)
The Content tab.
In this tutorial you will create a new Multileader style called TUTLEADER and modify the values in the Create New Dimension Style dialog box.

1. Select the Format/Multileader Style pull-down menu to access the Multileader Style Manager dialog box.

2. Select the New button to access the Create New Multileader Style dialog box. You will create a new multileader style called TUTLEADER, starting with the Standard style. Your Create New Multileader Style dialog box should look like Figure 13.55. Press the Continue button.

4. In the Leader Format tab change the Arrow size to .125. Your Leader Format tab should look like Figure 13.56.

5. In the Leader Structure tab set the First segment angle to 45 and the Scale to 2.000. Your Leader Structure tab should look like Figure 13.57.

6. In the Content tab set the Text height to .125 and the Right attachment to Middle of Top line. Your Content tab should look like Figure 13.58. Select OK to close the Modify Multileader Style dialog box, then close the Multileader Style dialog box.
In this tutorial, you will create a multileader on the Tutorial 13.1.dwg file. In order for this tutorial to work correctly, you must have defined the TUTLEADER multileader style.

1. Ensure that the Dimensions layer is the current layer.

2. Ensure that TUTLEADER is the current multileader style shown in the Leaders ribbon on the Annotate tab as shown in Figure 13.28

3. Select the Multileader button from the Leaders tile on the Annotate ribbon.

4. At the prompt place the leader in the circle in the front view. Add the following text:

   REAM TO SIZE

5. Your drawing should look like Figure 13.60. Save the drawing. This completes the dimension tutorials.
Dimensions are very important to technical drawings. They are needed so that a part or building can be manufactured correctly. Without dimensions drawings are just that - drawings. They do not contain necessary information and just become works of art instead of legal manufacturing documents. Creating dimensions manually is a very time consuming procedure. AutoCAD allows you to create and modify dimensions quickly and correctly. Keep in mind AutoCAD will not correctly dimension a drawing for you. You must be knowledgeable in the appropriate standard for your field and know how to correctly dimension a drawing. Just because AutoCAD puts a dimension on a drawing in a particular manner does not mean it is correct.

Multileaders can also quickly add necessary information to a drawing that is not covered by dimensions. Before dimensioning or adding leaders you should define a style. Defining a style for dimensions and multileaders ensures standardization on a drawing. If you, or someone else later has to edit the drawing it is much easier adding dimensions and leaders if styles are already created.

There are many ways to edit existing dimensions and multileaders. Grips can easily change the location of dimensions and leaders as well as move the text. To edit many of the dimension and leader settings on a single object simply select one dimension or multileader, right-click, and select Properties in the pop-up menu. The properties menu will allow you to change many of the characteristics of the selected object.