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AUTODESK AUTOCAD

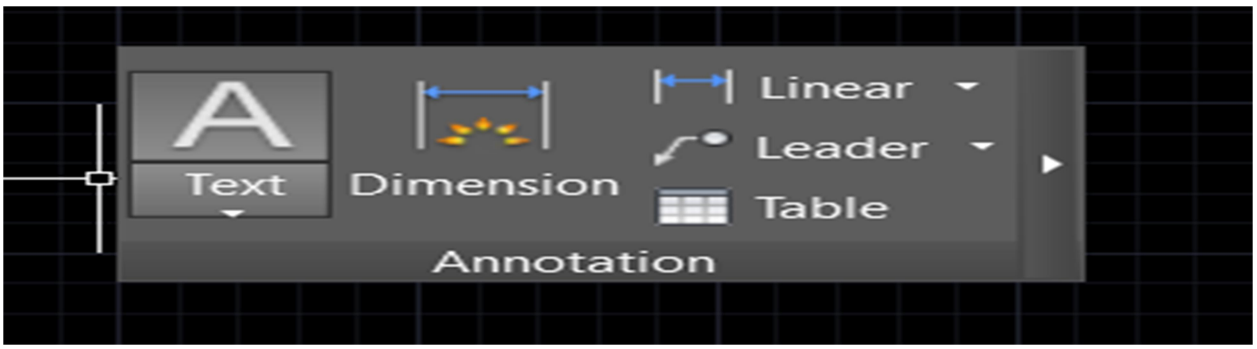
AutoCAD(CIVIL)

Annotation Panel Part-1

DETAILS OF THE DRAWING
ANNOTATION PANEL PART-1

Annotation Tools

Annotation objects include dimensions, notes, and other types of explanatory symbols or objects commonly used to add information to your drawing. Annotation objects provide information about a feature, such as the length of a wall, the diameter of a fastener, or a detailed callout.



The annotation tools mainly use in detailing about the drawing. The annotations tools containing dimension, text, leaders, and tables.

TEXT

The text is a specification for the drawing. The text is classified in two types

- Single Line Text
- Multiline Text

SINGLE LINE TEXT

You can single-line text to create one or more lines of the text, where each text line is an independent object that can move, format or otherwise modify. The single command for DT-Enter.

For short, simple notes and labels, using single line-text.

Click Home tab ➤ Annotation panel ➤ Single Line Text. Find

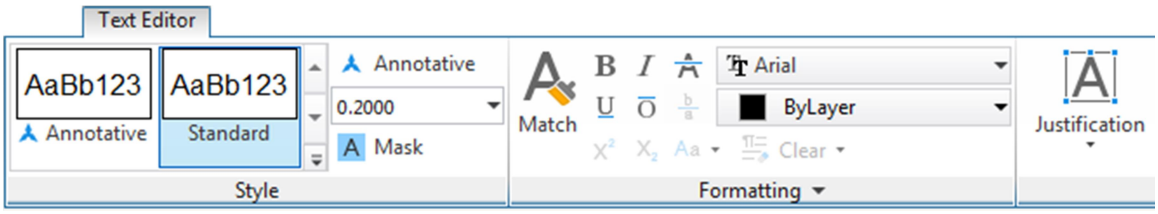
- Specify the insertion point
- Enter a height or click to specify height of the text
- Enter an angle value or click to specify the rotation angle
- Enter the text (The text may be displayed horizontally and at legible size
- Press ENTER on a blank line to end the command.

MULTILINE LINE TEXT

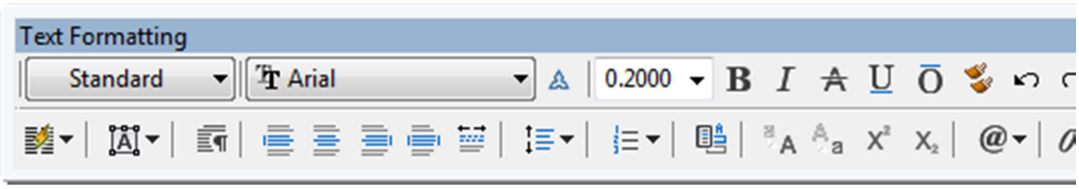
You can create several paragraphs of text as a single multiline text (**MTEXT**) object. With the built-in editor, you can format the text appearance, columns, and boundaries.

- Click Home tab ➤ Annotation panel ➤ Multiline Text. Find
- Specify opposite corners of a bounding box to define the width of the multiline text object.

If the ribbon is active, the Text Editor contextual tab displays.



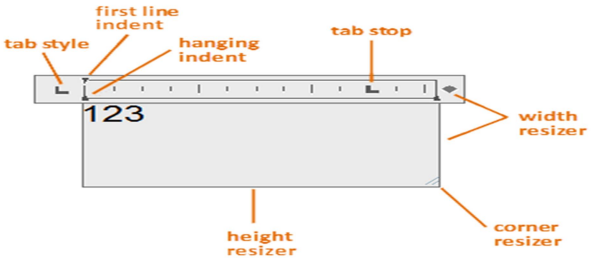
If the ribbon is not active, the Text Formatting toolbar displays.



Note: The MTEXTTOOLBAR system variable controls the display of the Text Formatting toolbar.

- Specify the initial formatting.

a. To indent the first line of each paragraph, drag the first-line indent slider on the ruler. To indent the other lines of each paragraph, drag the hanging indent slider.
b. To set tabs, click the ruler where you want a tab stop.
c. To change the current text style, select the desired text style from the drop-down list.



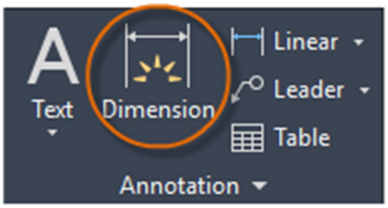
- Enter the text.
  Note: While typing, the text may be displayed horizontally and at a legible size.
- To change individual characters, words, or paragraphs, highlight the text and specify the formatting changes.
  Note: SHX fonts do not support boldface or italics.
- To save your changes and exit the editor, use one of the following methods:

a. On the Text Editor Ribbon contextual tab, in the Close panel, click Close Text Editor.
b. Click OK on the Text Formatting toolbar.
c. Click in the drawing outside the editor.
d. Press Ctrl--Enter.

- Note: Press Esc to exit the editor without saving your changes.

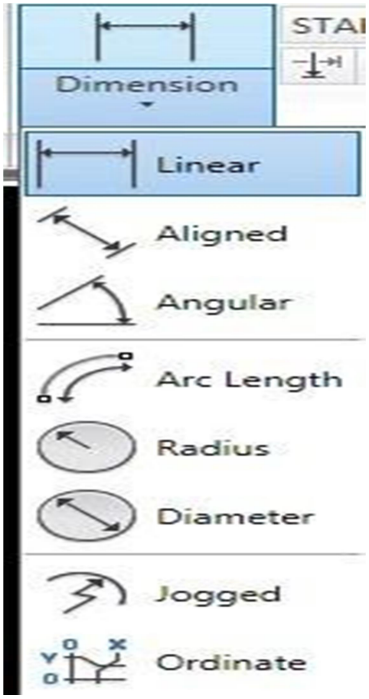
DIMENSION

The correct use of AutoCAD dimension tools is the key to producing clear and concise measured drawings.



The dimension consist of the Dimensions panel

- Linear
- Aligned
- Angular
- Arc length
- Radius
- Diameter
- Jogged
- Ordinate

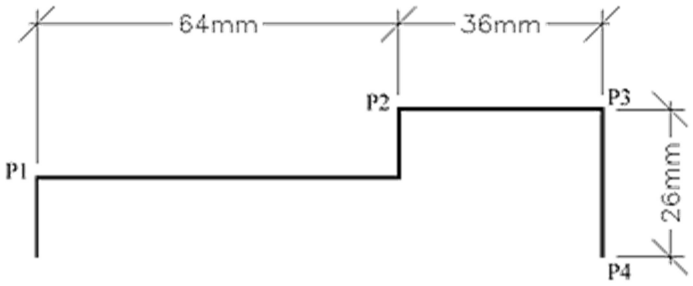


LINEAR DIMENSIONS:

As the name suggests the Linear dimension commands are used to dimension along straight lines. There are five linear dimension commands, namely: DIMLINEAR, DIMCONTINUE, DIMBASELINE, DIMALIGNED and DIMROTATED. The DIMLINEAR command is probably the most common dimension command you will use. You can use this command to generate horizontal and vertical dimensions. Creating a linear dimension is easy. All you have to do is start the command, specify the two points between which you want the dimension to be drawn and pick a point to fix the position of the dimension line.

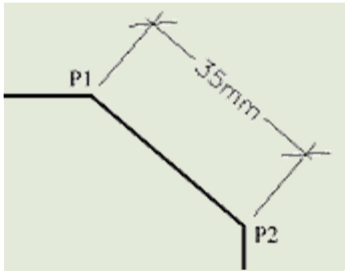
Command: DIMLINEAR

1. First extension line or origin or press enter
2. Second extension line origin
3. Dimension line location



ALIGNED DIMENSIONS:

You can use this command to generate aligned dimensions. These are dimensions along inclined lines which cannot be dimensioned with the DIMLINEAR dimension command because that command will only give a measured dimension in either a horizontal or vertical direction. However, as you can see from the command sequence below, this command works in exactly the same way.



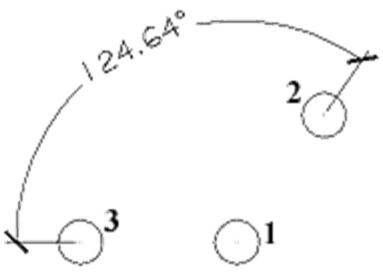
Command: DIMALIGNED

1. First extension line origin or press ENTER to select
2. Second extension line of origin
3. Dimension line location

ANGULAR DIMENSIONS:

The Angular command is amazingly flexible and can be used to indicate an angle in almost any situation. Just like the other dimension commands, all parts of the process are rubber banded so you can see the results of your actions before you make the final pick.

Command: DIMANGULAR
Select arc, circle, and line or press enter: (pick a line)
Second line :(picks another)
Dimension arc line location (Mtext/Text/angle): pick point

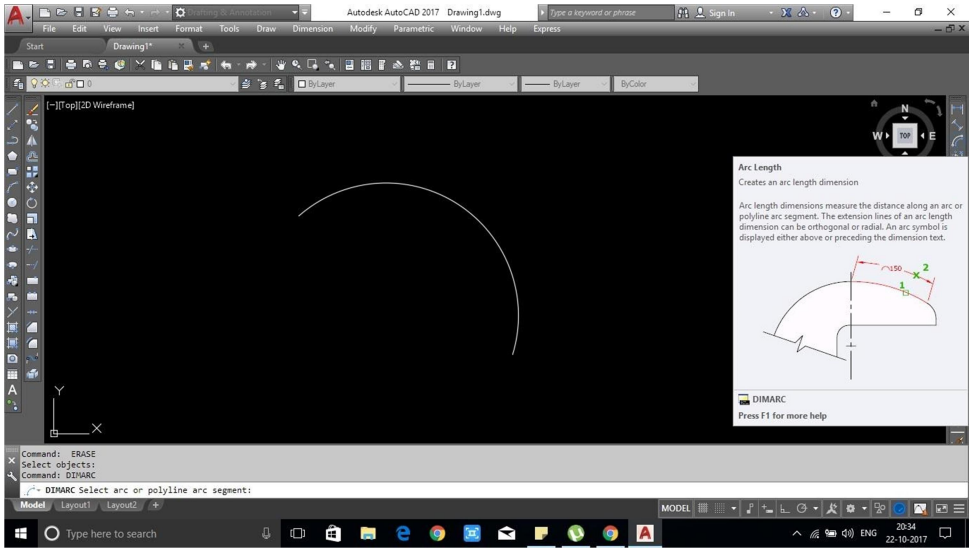


- Move the cursor position until you are happy with the result. Notice that you can move the cursor to either side of the lines and the angular dimension will change accordingly.
- You may have noticed that at the first prompt you are given the option to press ENTER. If you use this option you will be prompted to pick the angle vertex and then the two angle endpoints. This is quite useful if the angle you need to dimension is not defined by physical lines on the drawing. The illustration on the right shows the result of this option. The centre point of circle 1 was picked as the angle vertex and the centre points of circles 2 and 3 were picked for the two angle endpoints.

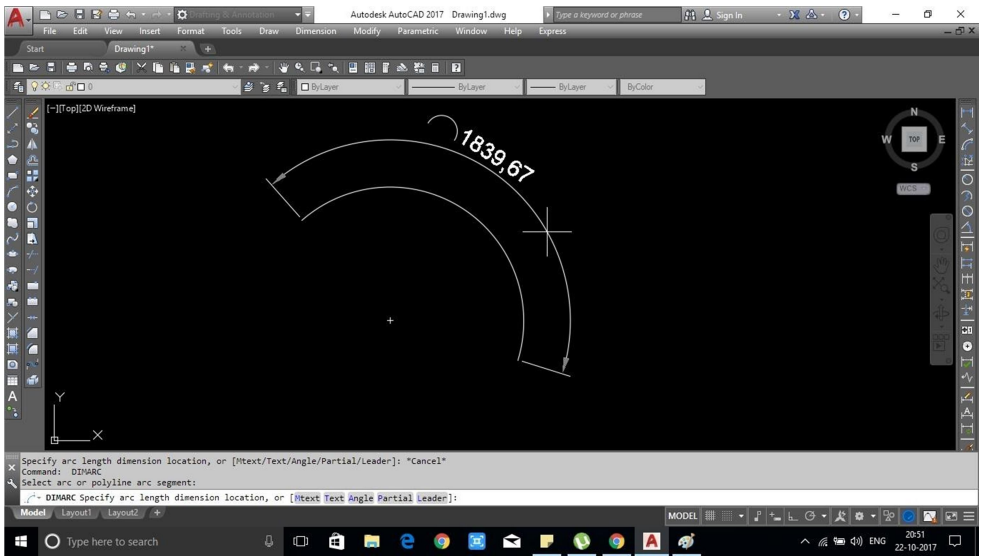
ARC LENGTH DIMENSIONS:

Many times users need to give the dimension of the arc as an arc length dimension & need to check arc length in AutoCAD. Users use list commands to check arc length in AutoCAD as well. Users can give arc length dimension in AutoCAD by using ARC LENGTH command. Also, users can use arc length dimension command for polyline arc segments.

- Type DIMARC in the command line and press enter or users may take command from the dimension toolbar as shown in figure.



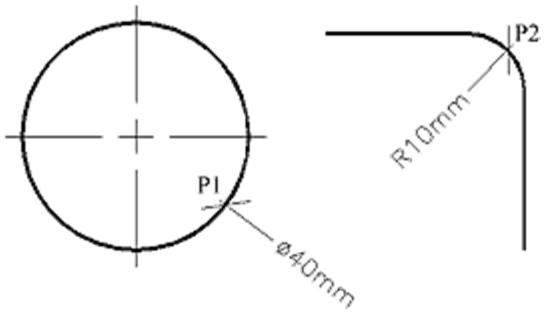
- Select the arc or polyline arc segment as shown in figure.
- Take the cursor upside and click on the screen.



RADIAL DIMENSIONS:

There are two main radial dimension commands, DIMDIAMETER and DIMRADIUS. Both commands result in a similar looking dimension so AutoCAD automatically inserts a "R" to indicate a radius and the dimension symbol to indicate a dimension. The Diameter and Radius commands are supplemented by the DIMCENTER command which can be used to add a center mark to any circle or arc. The DIMDIAMETER and DIMRADIUS commands do not automatically draw a center mark.

Command: DIMRADIUS
Select arc or circles :(pick the circumference of P1)
Dimension line location (mtext/angle/text):(move the cursor until you are happy with the text position)



DIAMETER DIMENSION:

You can use the Diameter command to annotate a circle or an arc with a diameter dimension. To achieve this simply start the command, pick a point on the circumference of the circle, pick a second point to determine the length of the leader and then add the dimension text or Return to accept the default.

Command: DIMDIAMETER
Select arc or circle :(Pick the circumference of P1)
Dimension line location (Text/angle): move the cursor until you are happy with the text position and they pick to complete the sequences).

DIMENSION STYLE MANAGER

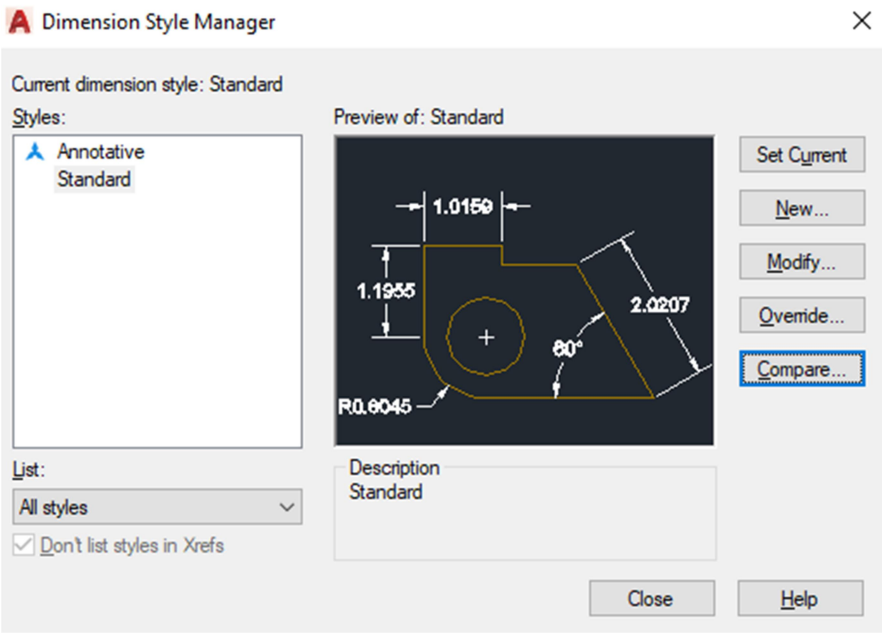
The Dimension Style Manager is used to create, modify, override, and compare the new styles and dimensions in AutoCAD.

The **Standard** dimension style is considered as the default dimension style.

To open the Dimension Style Manager,

- Type **D** or **DIMSTY** on the command line or command prompt and press **Enter**.

The dialog box will appear, as shown below:



The properties given on the right side of the Dimension Style Manager are listed below:

- **Set Current**

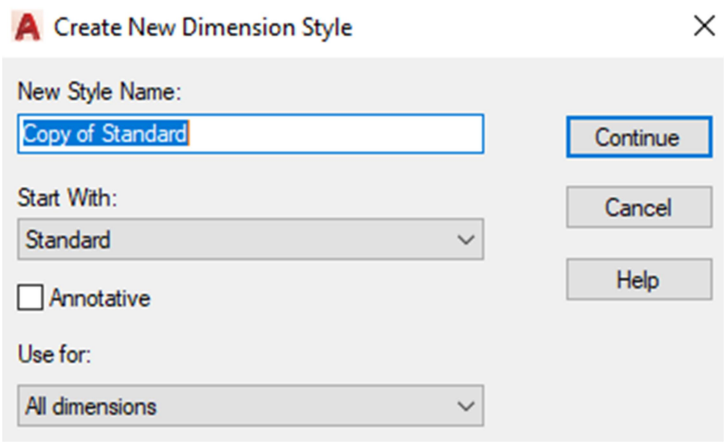
  It applies the current style to the dimensions created by us. The selected style will be set as the current style.

- **New**

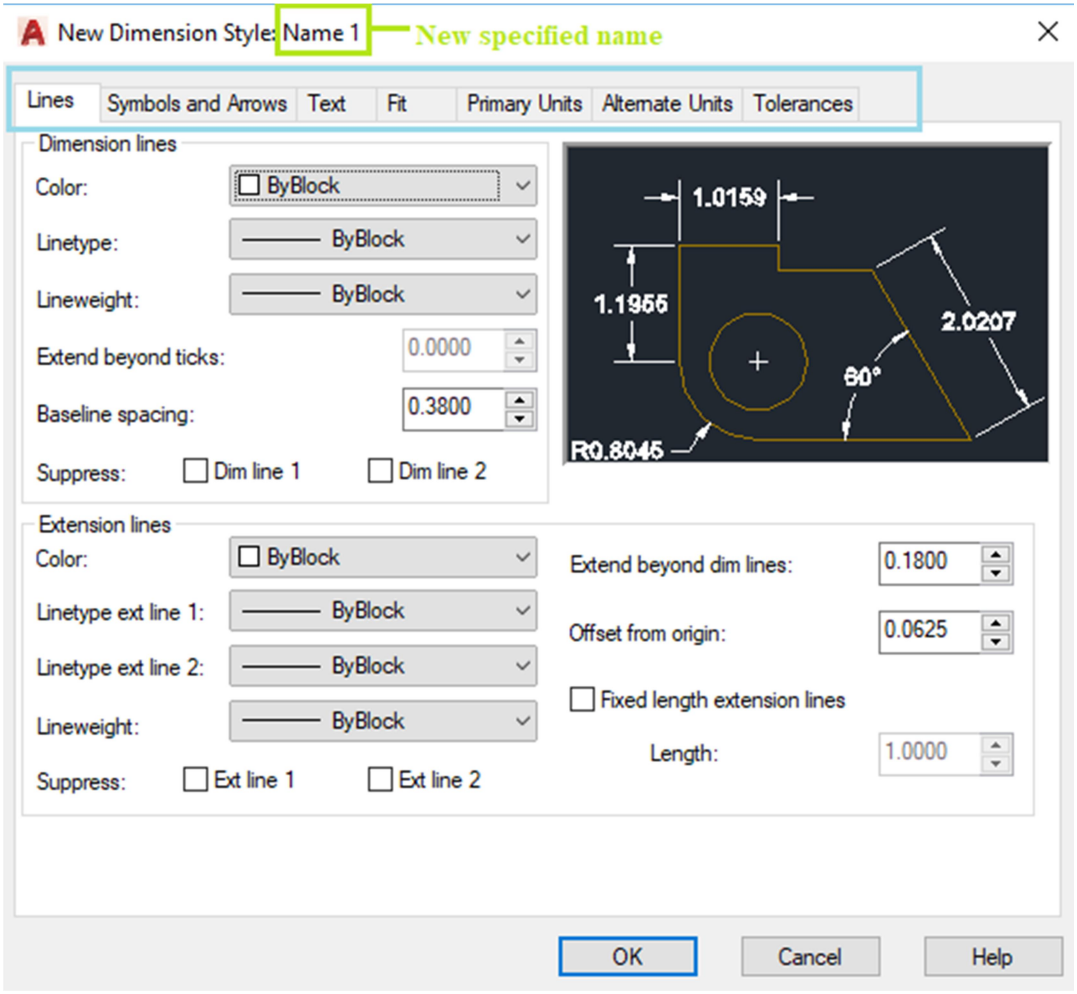
  It is used to create a new dimension style. The steps to create a new dimension style are listed below:

  1. Click on the '**New**' button.

  A dialog box will appear, which will look like the below image:



2. Specify the **New Style Name** and click on the **Continue** button. A dialog box will appear, as shown below:



The top row marked consists of the different characteristics of Dimensions. We can modify the values accordingly.

We need to click on the **OK** and **Close** to apply the dimensions on the drawing.

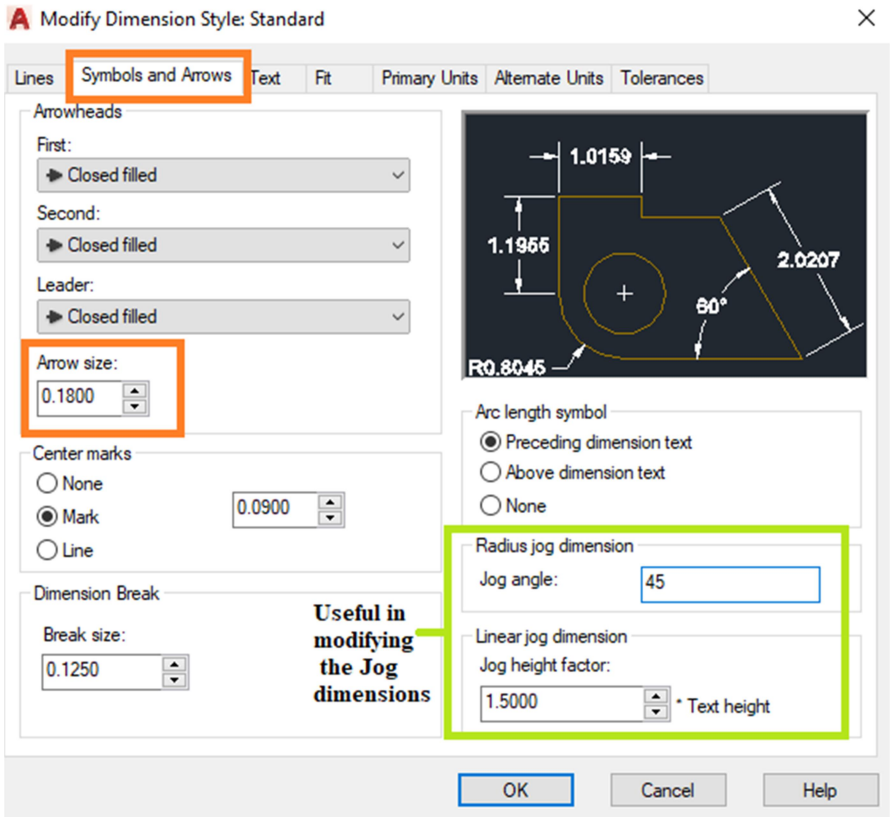
- **Modify**

The **modify** option will open the modification dialog box, which is the same as the New Dimension Style dialog box.

Here, we can modify the different characteristics of dimensions, according to the requirements.

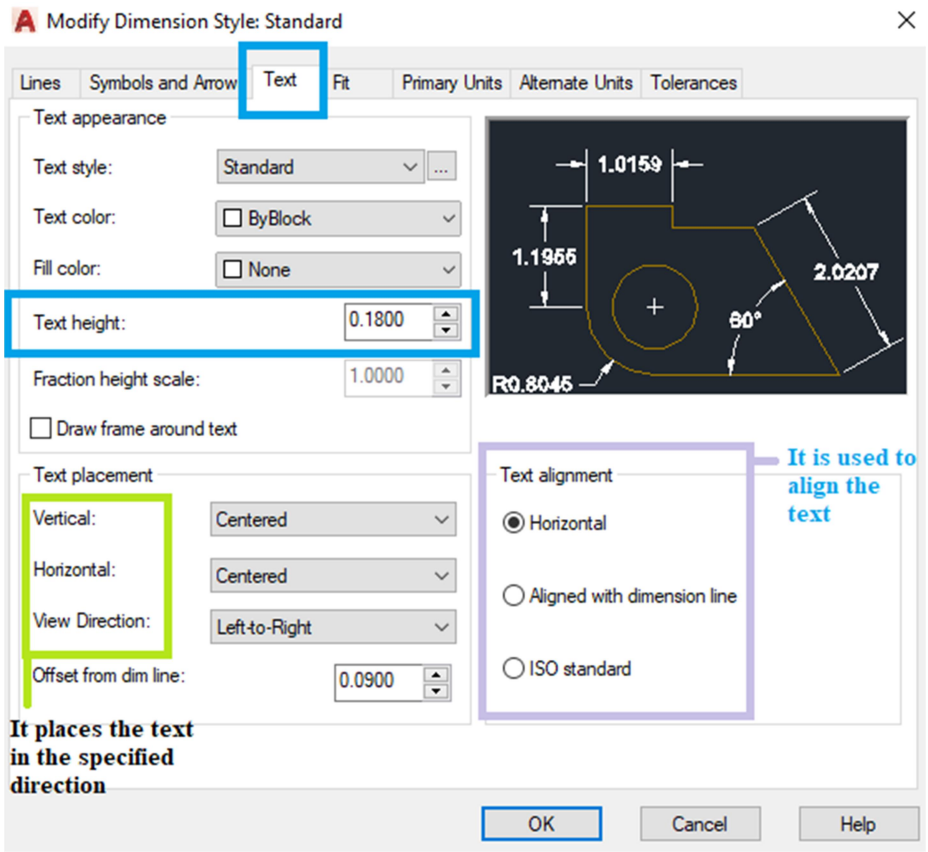
- To modify the size of the arrow,

Click on the **'Symbols and Arrows'** option, which will modify the value below the **Arrow size**, as shown below:



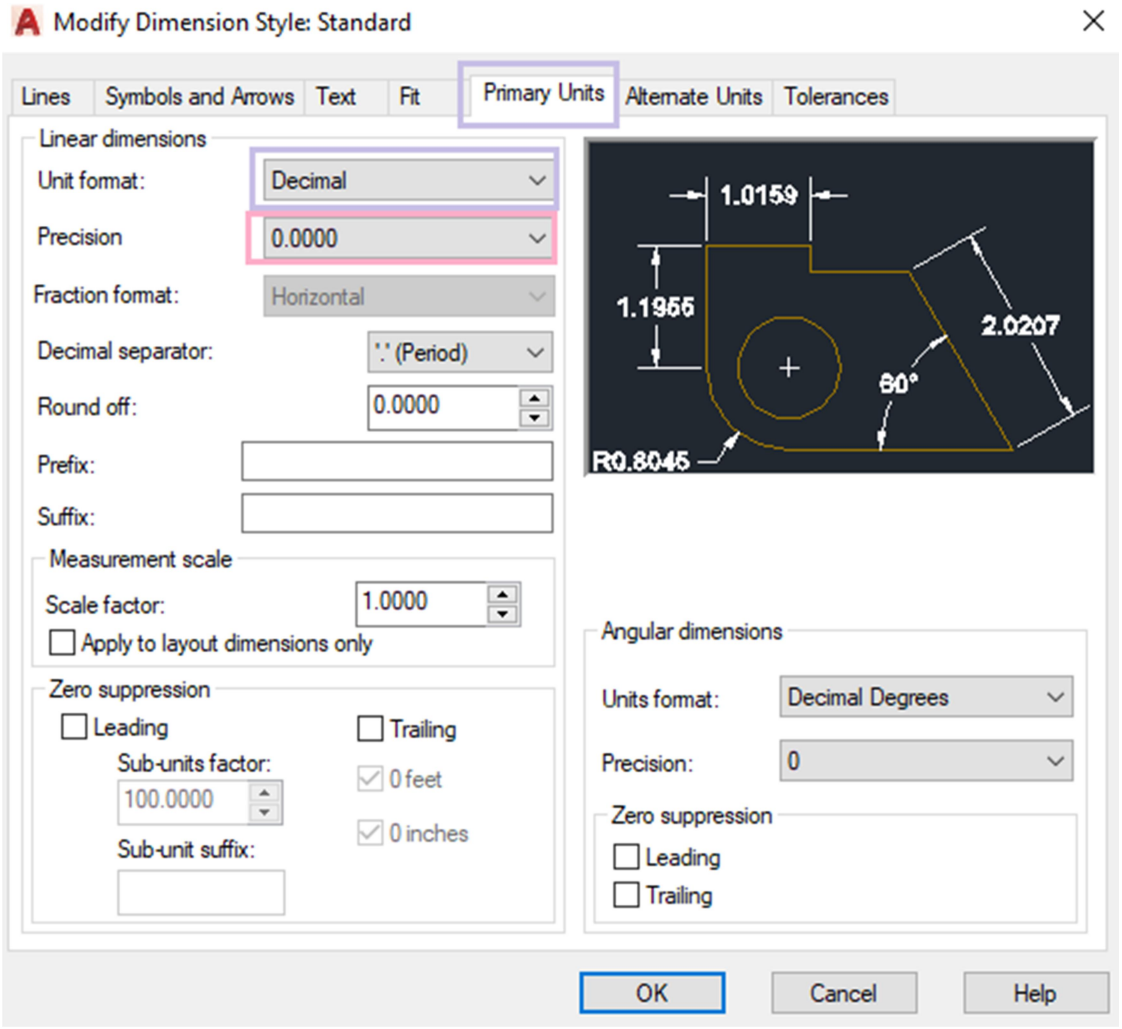
- To modify the text size,

  Click on the **'Text'** option and modify the value in front of the **Text Height**, as shown below:



- To change the **Units type** and the **precision** value,

  Click on the '**Primary Units**' option, modify the value in front of the **Precision** and **Unit Format**, as shown below:



The precision format is shown in the below image:

Precision	Example
0	1
0.0	1.2
0.00	1.20
0.000	1.198
0.0000	1.1966
0.00000	1.19660
0.000000	1.196601
0.0000000	1.1966010
0.00000000	1.19660148

The dimensions will be rounded to the specified precision value.