

## Basics of Annotation in AutoCAD

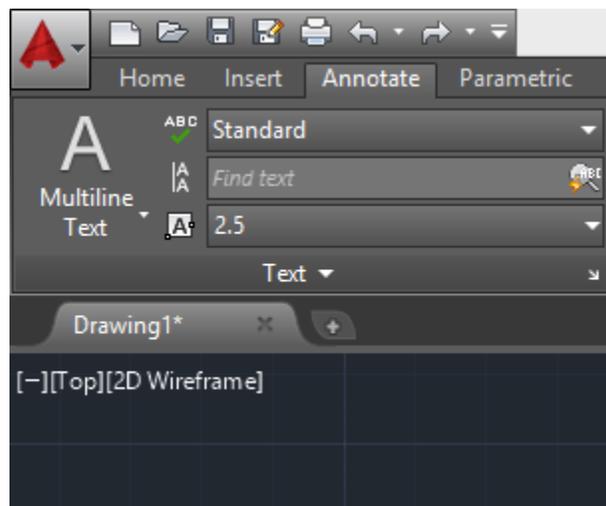
AutoCAD ANNOTATION is what makes a project have all its meaning and its reason to the eyes of someone looking at it when the creator of the project is not around. Annotations can also help CAD drafters keep track of all the important information a plain image cannot express in a project.

In AutoCAD, a few of the many ways to Annotate a draft are to use DIMENSIONS pallet tools and the TEXT pallet tools.

### TEXT

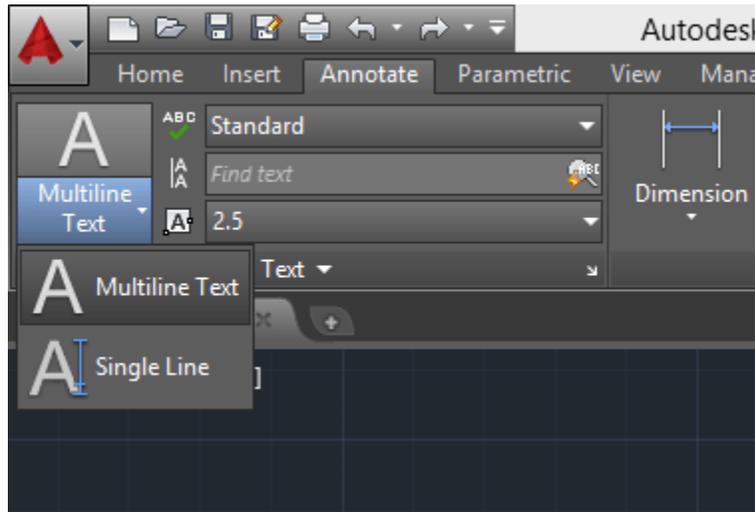
AutoCAD supplies two different types of text objects:

1. The Dtext command inserts one or more lines of text as single line objects. The Dtext command enables you to display Auto CAD text dynamically in the drawing as you type it. it also allows you to justify or align, the text in several ways, including left, right, and center justification.
2. The Mtext Command, treats one or more paragraphs of text as single objects. The Mtext command creates a multiple-line object called Mtext.



This pallet provides tools that help you write text in the drawing area. A text after creation will be just behaving like an ordinary object in AutoCAD. You will be able to move it, duplicate it, or do any other things you would with an image.

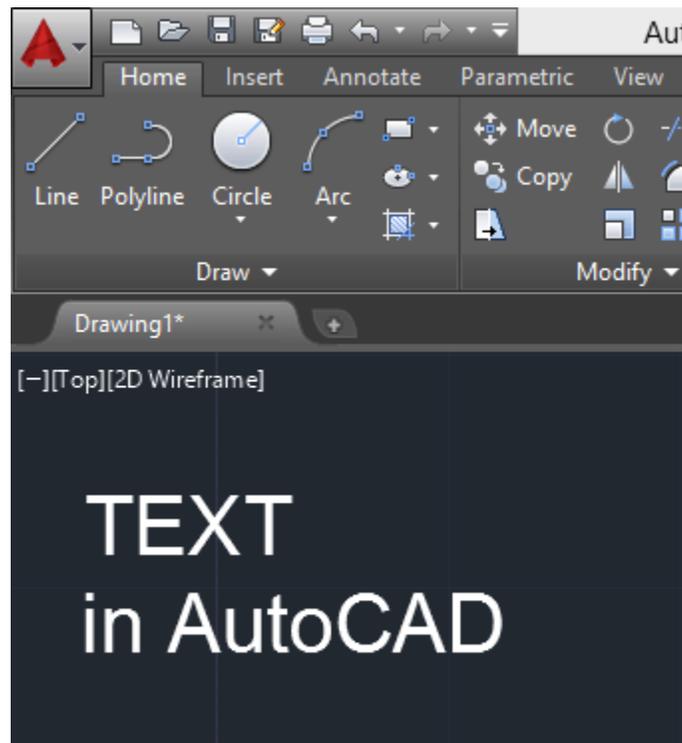
## Create a text in AutoCAD



### ***Multiline TEXT (MTEXT command)***

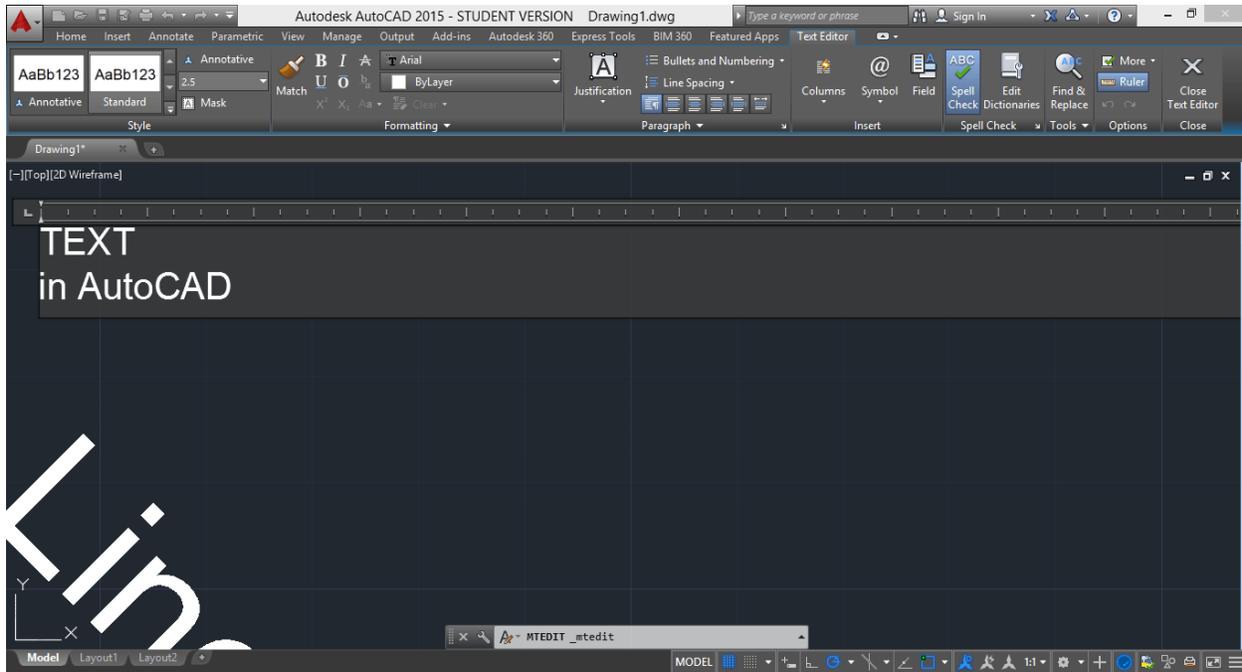
This command will simply allow you to create a multiline text object in AutoCAD as you would if you are using Microsoft word. To use it, you have to first specify the area in the drawing window where you would like the text to appear, then you will be able to enter your text.

The following shows a result of a text created using MTEXT



After creating a text (or while), you will be able to control its style and formatting, you will be able to create paragraphs, insert symbols, break the text into columns, and more.

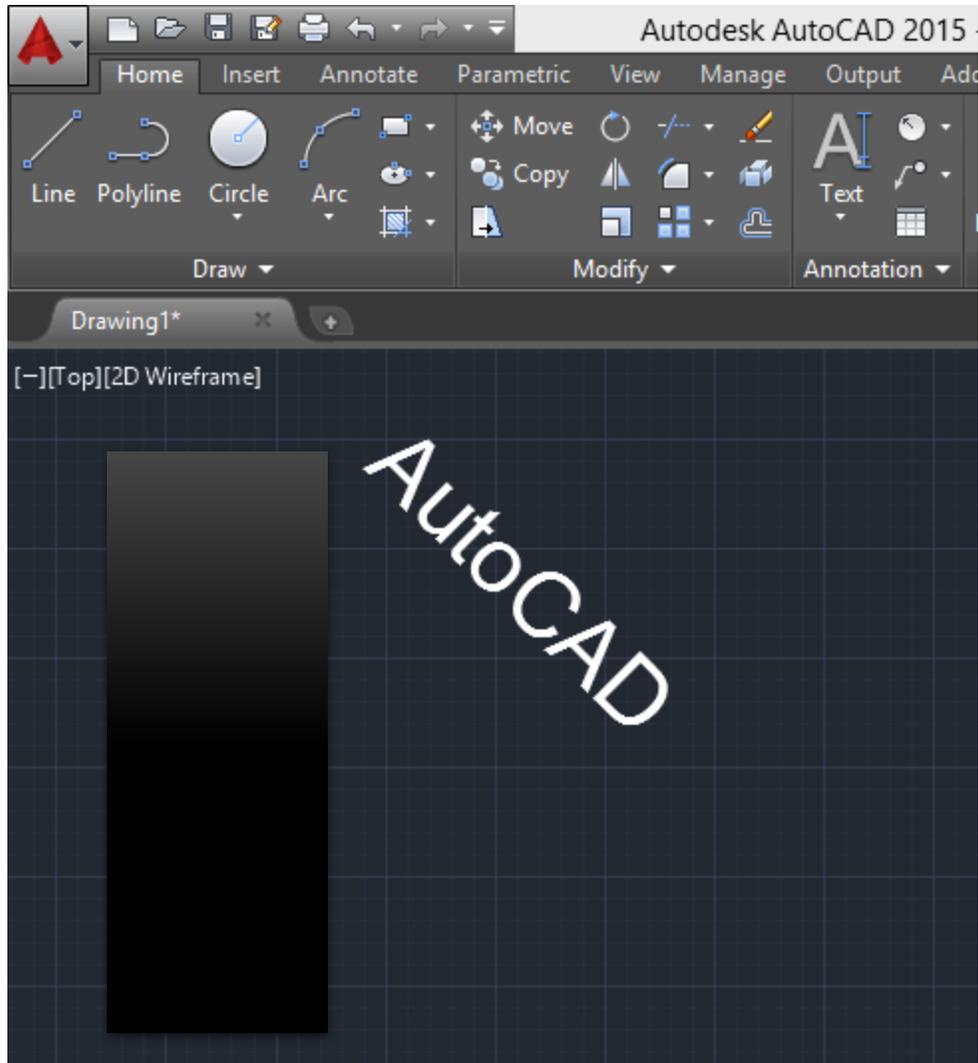
To access the formatting tools, double-click on the text you created.



### Single LINE (TEXT command)

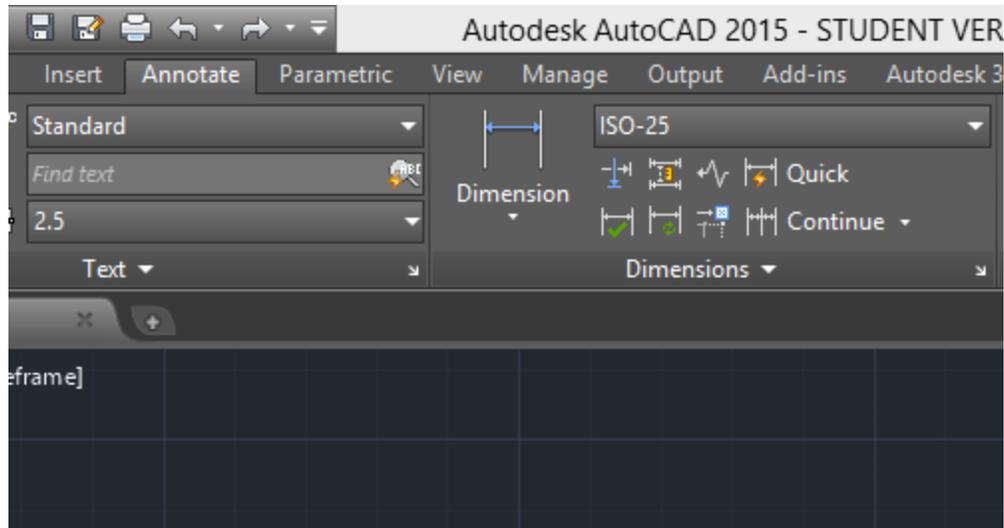
TEXT allows you to create a single-line text object in AutoCAD. The major difference between the TEXT command and the MTEXT command is that while creating a text object with the TEXT command, each time you press the ENTER key, you are creating a piece of text not in the same entity as the previous one. Both can be selected separately and move independently one from another.

The TEXT command also easily allows creating texts while playing with their orientation.



## Dimensions

AutoCAD dimensioning is semi-automatic, when you invoke a command to create a linear dimension, AutoCAD only requires that you Pick an object or specify the extension line origins, and pick the location of the dimension line. AutoCAD then measures the feature and draws the extension line, arrowheads, and dimension text.

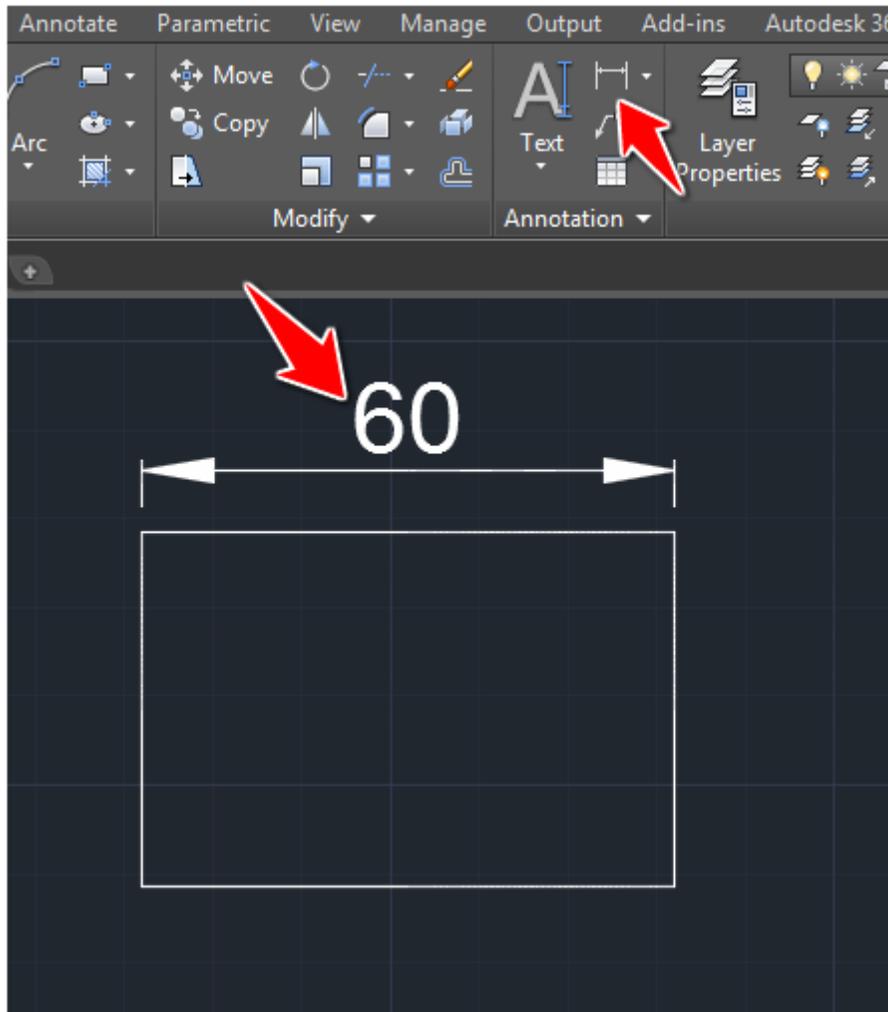


- **Linear dimensions:** are those with horizontal, vertical, or aligned dimension lines. A dimension line is the part of the dimension that typically contains arrowheads at each of its ends.

**Command: dim linear**

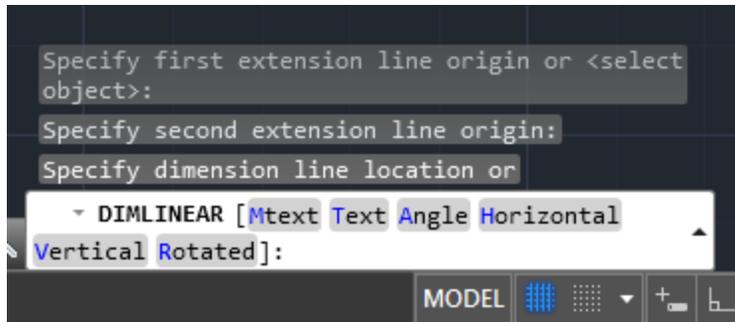
**Specify first extension line origin or <select object>:**

**Select object to dimension:**

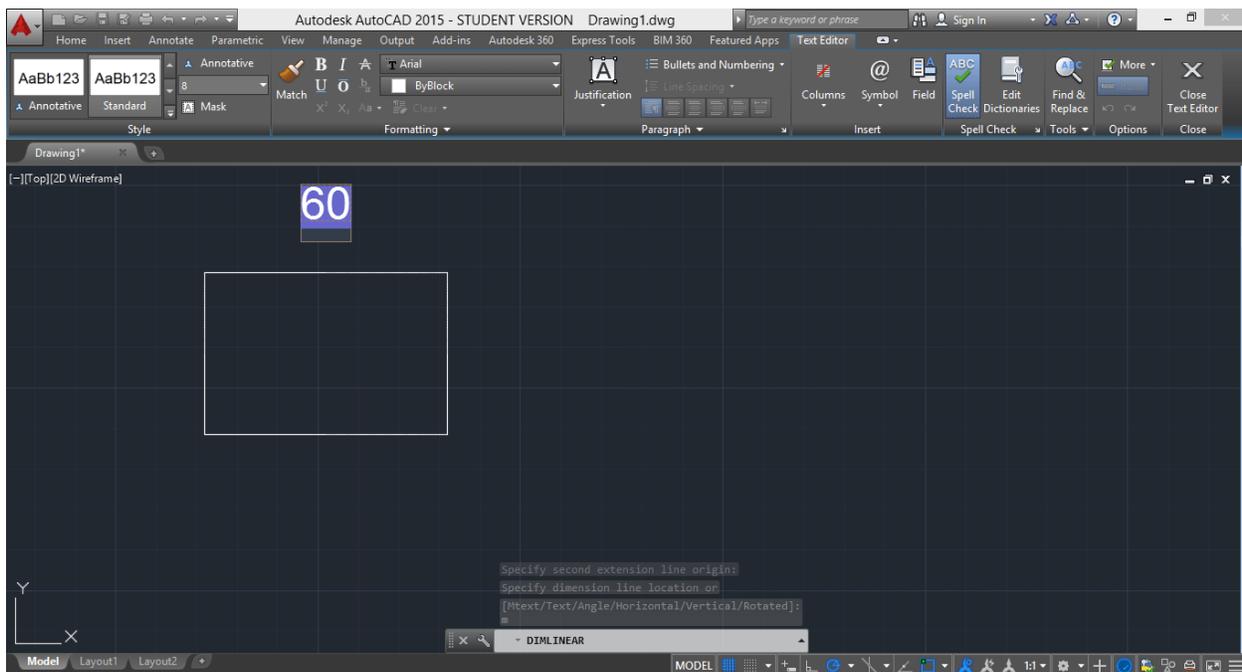


### **DIMLINEAR settings**

While using this command you can choose to make use of the following via the command window.



Where **Mtext** will help access the properties of the dimension text,



**Text** will help edit the dimension text,

**The angle** will help rotate the dimension text,

**Horizontal and Vertical** will help specify the orientation of the dimension and

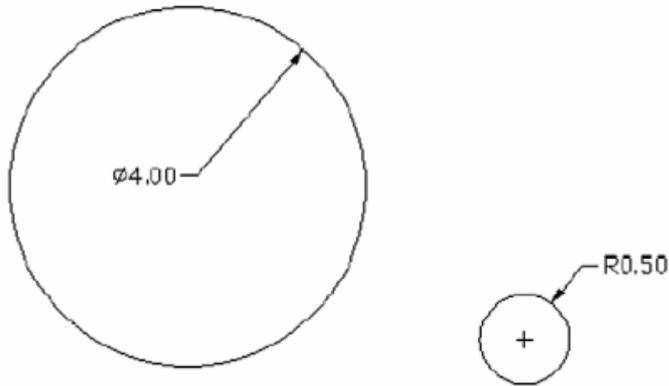
**Rotate** will help rotate the dimension line.

- **Dimensioning Round Features:** use radius dimensions for features such as fillets and rounds.

*Command: dim Diameter*

*Select arc or circle:*

*Select object to dimension:*



- **Dimensioning Angles:** Dimension the angles.

*Command: dim Angular*

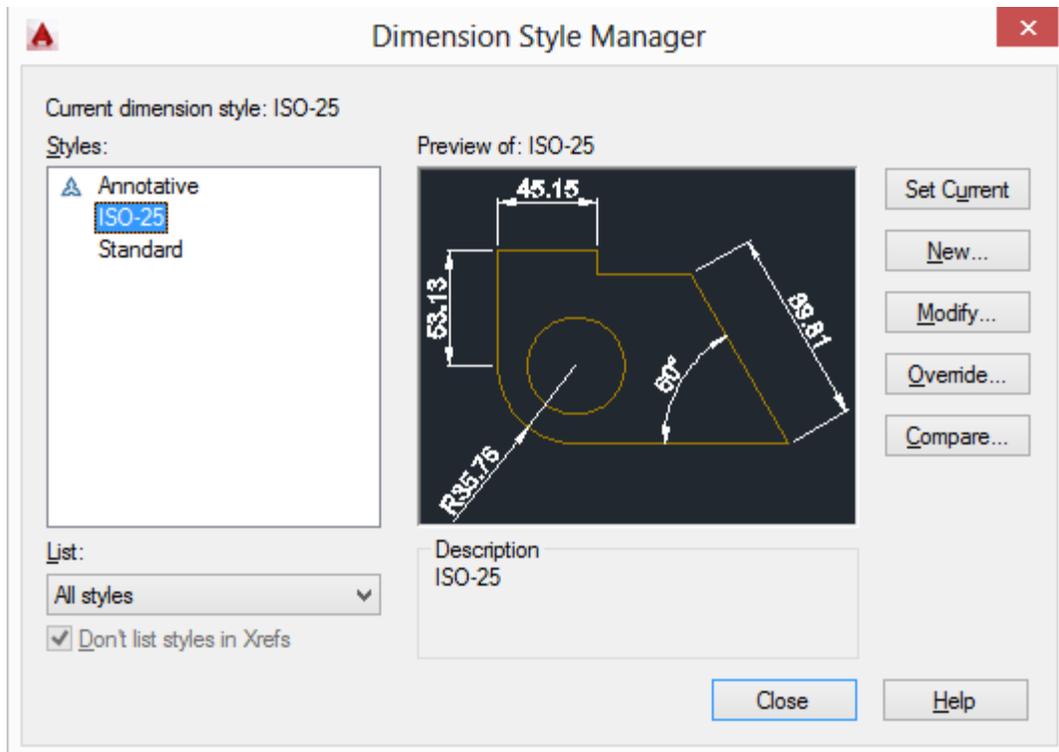
*Select arc, line or (specify vertex):*

*Specify second angle end point:*

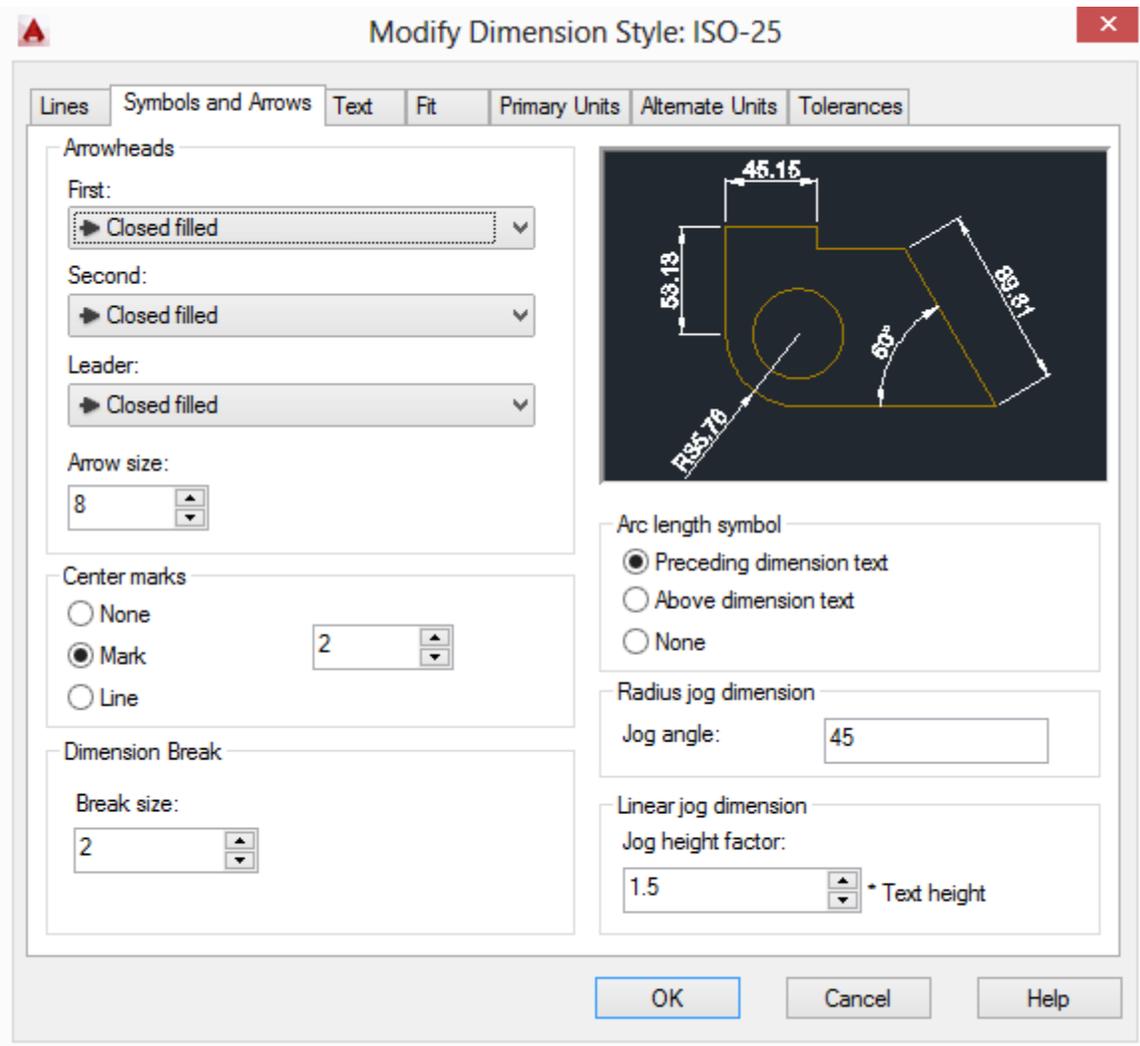
*Specify dimension arc line location or [Mtext ,text, angle]:*

## **DIMSTYLE command**

DIMSTYLE is the command that helps you access the Dimension Style Manager.



To make dimension setting in preparation for dimensioning a drawing click on the style, option in the dimension pull-down menu at the top of the screen, then click on the modify button to open the diminution style dialog box.



That can help you modify the whole dimension style properties of your project.