



Andhra Pradesh State Skill Development Corporation



AutoCAD(CIVIL)

Annotation Panel
Part-2



DETAILS OF THE DRAWING ANNOTATION PANEL PART-2

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.



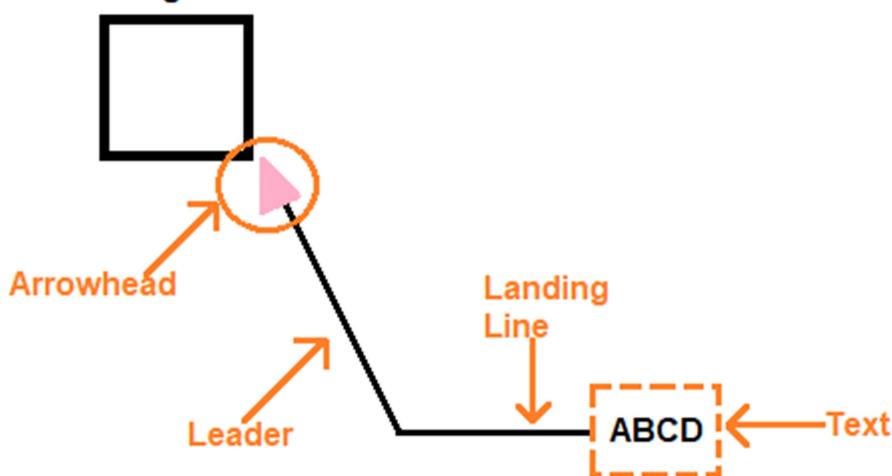
The annotation tools mainly use in detailing about the drawing. The annotations tools containing dimension, text, leaders, and tables.

MULTILEADER

- The MLEADER command in AutoCAD is used to create leader objects. It includes an arrowhead, a leader line or curve, and a horizontal landing. It also consists of a block or multiline text object.
- It creates a line, which connects it to the dimensioning text. We can insert the text either in single or multiple lines.
- This command creates a leader line segment to the specified point and continues to extend to the last specified point.

The different parts of a Multileader are shown in the below image:

Figure

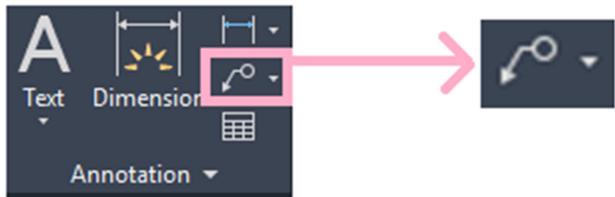


Let's understand with an example.

Consider the below figure:

The steps are listed below:

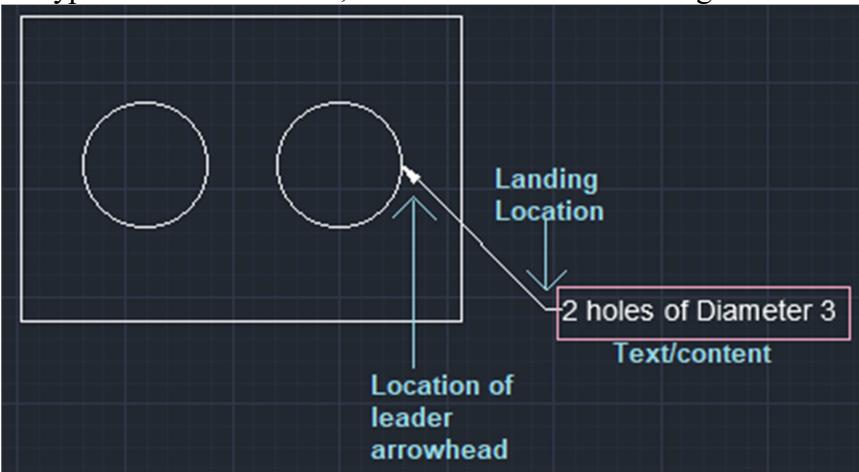
1. Select the Multileader command from the ribbon panel, as shown below:



Or

Type MLEADER on the command line or command prompt and press Enter.

2. Specify the point location of the leader arrowhead. The position of the leader can be on or near the particular object.
3. Specify the landing location of the leader.
4. Type the content or text, as shown in the below image:



The Multileader is created.

5. Press Enter or Esc to exit from the MLEADER

ADD LEADER:

Select the multileader.

- Click Annotate tab ▶ Multileaders panel ▶ Add Leader.  Find.
- Specify the endpoint for the new leader.

REMOVE LEADER:

Use this procedure to remove leader lines from the following Annotation tools:

1. Select an existing multiline leader.
2. Select remove leader from drop down list.
3. Select the leader to remove.

TABLE

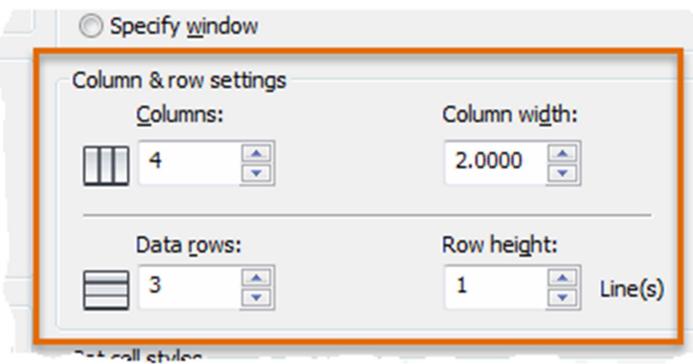
The table command in AutoCAD is used to insert the table containing rows and columns. We can resize the rows and columns, and can also stretch the entire table.

We can insert the appropriate data in rows and columns according to the requirements.

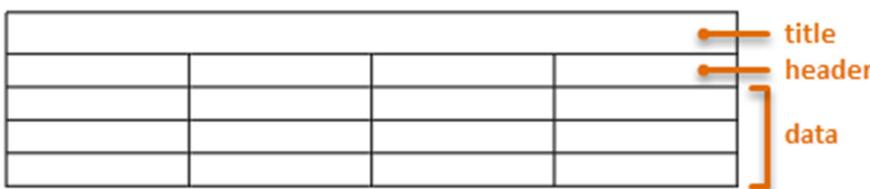


The steps are listed below:

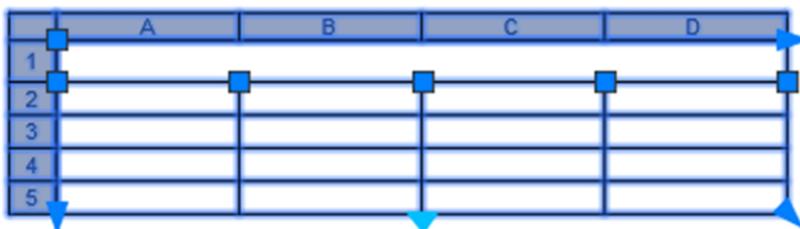
1. Enter TABLE at the Command prompt.
2. In the Insert Table dialog box, enter 4 columns and 3 data rows. Specify a location for the table.



By default, there are three styles of cells that appear when you use the Standard table style:



3. Click outside the table, and then select it on an edge to display its grips.

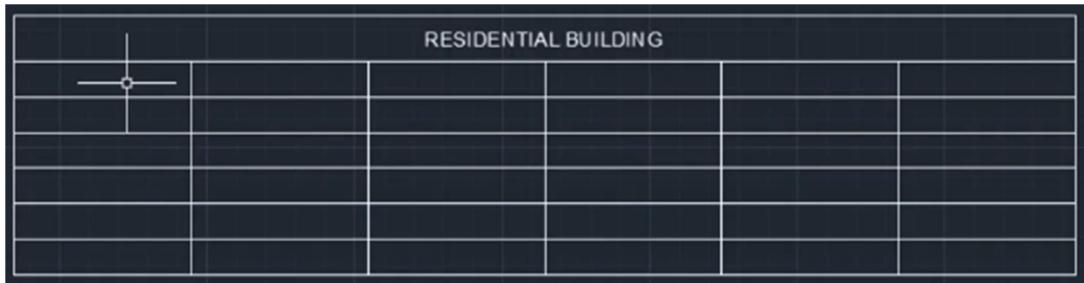


4. To change the size and shape of the table, click the dark blue triangular grips. You can click the square grips to adjust the width of the columns. You can also use object snap tracking to align the grips to existing geometry.

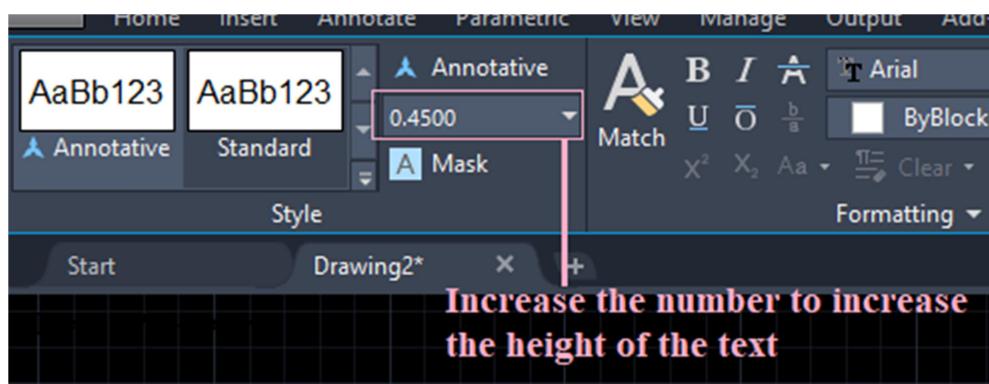
Adding Information to the created table:

After creating the table, we are required to insert text inside the table.

- Double-click on each cell.
- Write the text, as shown below:



- We can modify the size of the text, which is present on the top of the display, as shown below:



- Continue writing on other blocks by clicking.

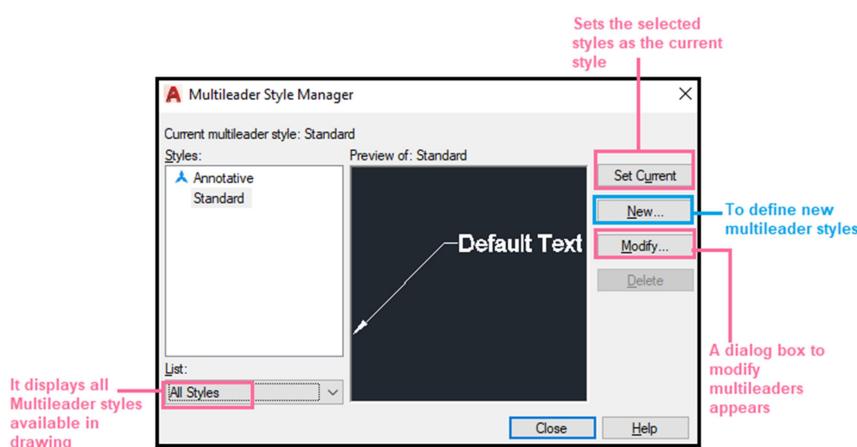
Multileader Style

The Multileader style or **MLEADERSTYLE** in AutoCAD is used to create and modify the Multileader styles. It controls the appearance of the Multileaders.

The Multileader style includes the formatting of the content, arrowhead, landing, and leader lines.

Let's discuss the steps to use the Multileader style.

- Type **MLEADERSTYLE** or **MLS** on the command line and press **Enter**. A Multileader Style Manager dialog box will appear, as shown in the below image:

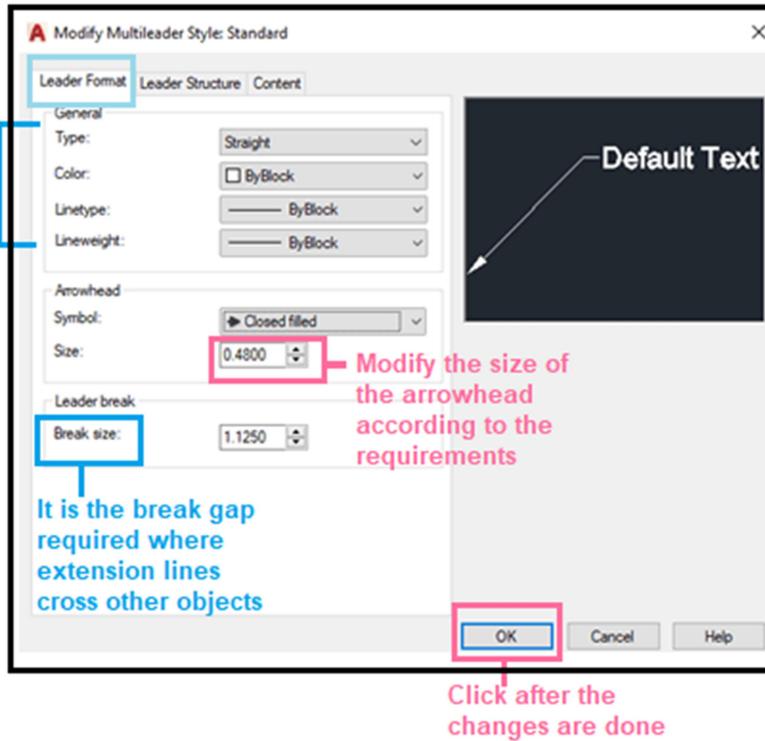




2. Click on the **Modify**

3. To modify the arrowhead size, click on the '**Leader format**' and change the value in front of the **Size**, as shown in the below image:

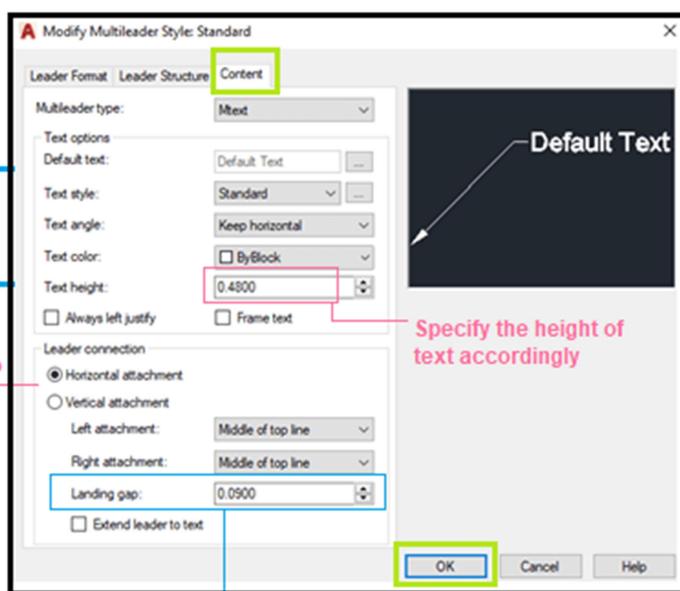
To modify the general characteristics



4. To modify the text height, click on the '**Content**' and change the value in front of the **Text Height**, as shown in the below image:

To modify the general characteristics

It places the text to the specified position



It is the distance between the landing line and multileader text



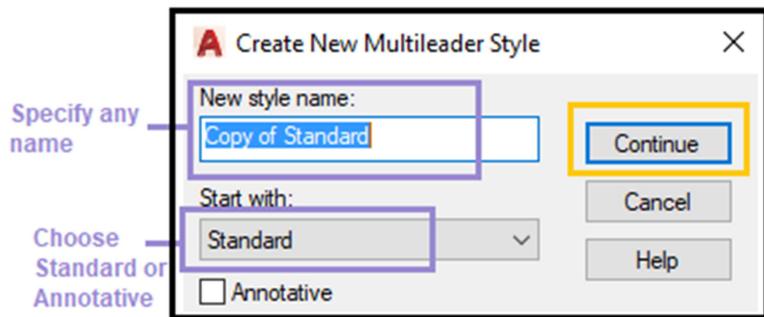
5. Click on the **OK** button.
6. Click on the **Close** button.
7. The changes will be updated in the figure.

New Multileader

The **NEW** button in the MLEADERSTYLE window is used to define new Multileader styles.

Let's discuss the steps to create a New Multileader style.

1. Type **MLEADERSTYLE** or **MLS** on the command line and press **Enter**. A Multileader Style Manager Dialog box will appear.
2. Click on the '**New...**' button present on the right. A dialog box will appear, as shown in the below image:

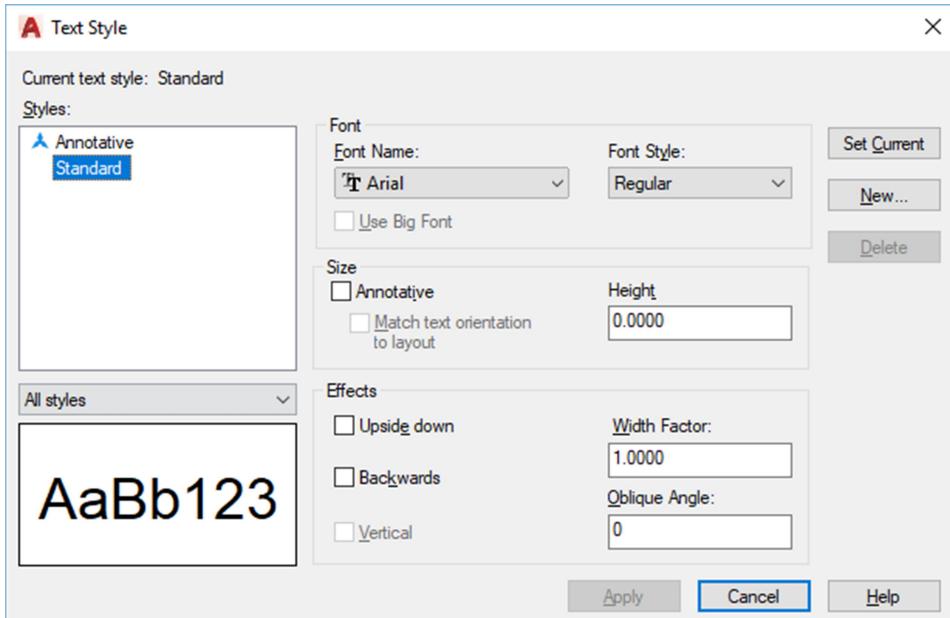


3. Specify the style name according to your choice.
4. Choose standard or Annotative.
5. Now, click on **Continue**.
6. A dialog box of Style Manager will appear. It will be the same as the **Modify dialog box**. We can change the text height, arrowhead size, etc. accordingly.
7. After the modification, click on the **OK** button at the bottom.

The new style will appear then Click on the **Close** button after finishing it.

Manage Text Style

STYLE (Command) Find



Creates, modifies, or sets named text styles.

List of Options

The following options are displayed.

Current Text Style

Lists the current text style.

Styles

Displays the list of styles in the drawing. A icon before the style name indicates that the style is annotative.

Style List Filter

The drop-down list specifies whether all styles or only the styles in use are displayed in the styles list.

Preview

Displays sample text that changes dynamically as you change fonts and modify the effects.

Font

Changes the style's font.

Note: If you change the orientation or font file of an existing text style, all text objects with that style use the new values when the drawing is regenerated.

Font Name

Lists the font family name for all registered TrueType fonts and all compiled shape (SHX) fonts in the Fonts folder.

When you select a name from the list, the program reads the file for the specified font. The file's character definitions are loaded automatically unless the file is already in use by another text style. You can define several styles that use the same font. For more information, see "Assign Text Fonts".



Font Style

Specifies font character formatting, such as italic, bold, or regular. When Use Big Font is selected, this option changes to Big Font Name and is used to select a Big Font file name.

Use Big Font

Specifies an Asian-language Big Font file. Only SHX files are valid file types for creating Big Fonts.

Size

Changes the size of the text.

Annotative

Specifies that the text is annotative. Annotative objects and styles are used to control the size and scale at which annotation objects are displayed in model space or a layout.

Match Text Orientation to Layout

Specifies that the orientation of the text in paper space viewports matches the orientation of the layout. This option is unavailable if the Annotative option is cleared.

Height or Paper Text Height

Sets the text height based on the value you enter. Entering a height greater than 0.0 sets the text height for this style automatically. If you enter 0.0, the text height defaults to the last text height used, or the value stored in the drawing template file.

TrueType fonts might be displayed at a smaller height than SHX fonts with the same height setting.

If the annotative option is selected, the value entered sets the text height in paper space.

Effects

Modifies characteristics of the font, such as its height, width factor, and obliquing angle and whether it is displayed upside down, backwards, or vertically aligned.

Upside Down

Displays the characters upside down.

Backwards

Displays the characters backwards.

Vertical

Displays the characters aligned vertically. Vertical is available only if the selected font supports dual orientation. Vertical orientation is not available for TrueType fonts.

Width Factor

Sets the character spacing. Entering a value less than 1.0 condenses the text. Entering a value greater than 1.0 expands it.

Oblique Angle

Sets the oblique angle of the text. Entering a value between -85 and 85 italicizes the text.

Note: TrueType fonts using the effects described in this section might appear bold on the screen.

Onscreen appearance has no effect on plotted output. Fonts are plotted as specified by applied character formatting.



Set Current

Sets the style selected under Styles to current.

New

Displays the New Text Style dialog box and automatically supplies a default name.

Style names can be up to 255 characters long. They can contain letters, numbers, and the special characters dollar sign (\$), underscore (_), and hyphen (-).

Delete

Deletes unused text styles.

Apply

Applies style changes made in the dialog box to the current style and to the text of the current style in the drawing.

Manage Table Style

You control the appearance of tables — both text and cells — with table styles in AutoCAD. Use the TABLESTYLE command to create and modify table styles. Follow these steps to create a table style:

1. On the Home tab, click the Annotation panel's label to open its slide out, and then choose Table Style.

The Table Style dialog box appears.

2. In the Styles list, select the existing table style whose settings you want to use as the starting point for the settings of the new style.

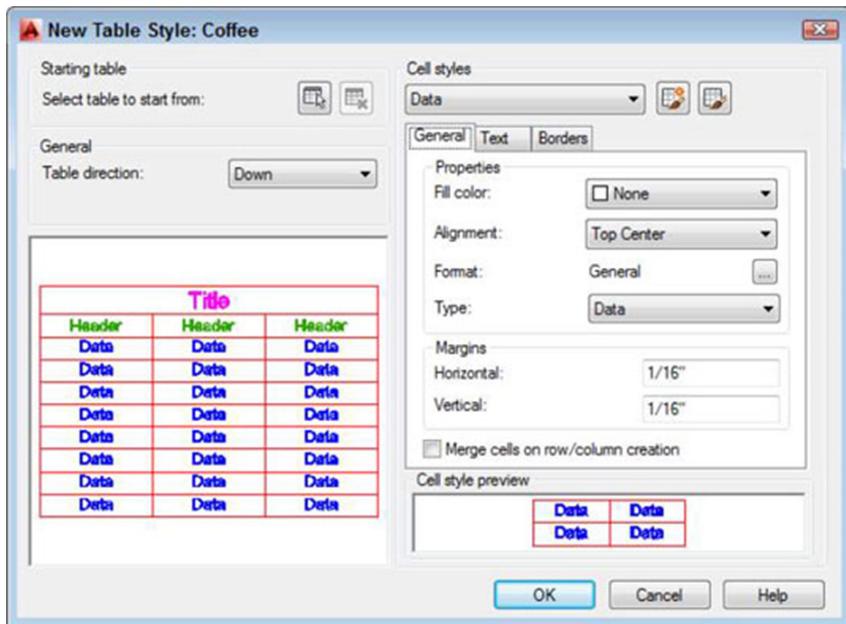
For example, select the default table style named Standard.

3. Click the New button to create a new table style that's a copy of the existing style.

The Create New Table Style dialog box appears.

4. Enter a new style name and click Continue.

The New Table Style dialog box appears.



5. In the Cell Styles area, with Data showing in the list box, specify settings for the data alignment, margins, text, and borders.

The settings you're likely to change are Text Style, Text Height, and perhaps Text Color (all three are on the Text tab) or Grid Color (on the Borders tab). If you leave colors set to ByBlock, the text and grid lines inherit the color that's current when you create the table. That color will be the current layer's color.

6. In the Cell Styles area, open the drop-down list and repeat Step 5 for the Headers (that is, the column headings) and the Title styles.
7. Click OK to close the New Table Style dialog box.

The Table Style dialog box reappears.

8. Click Close.

The new table style becomes the current table style that AutoCAD uses for future tables in this drawing, and the Table Style dialog box closes. Now you're ready to create a table.