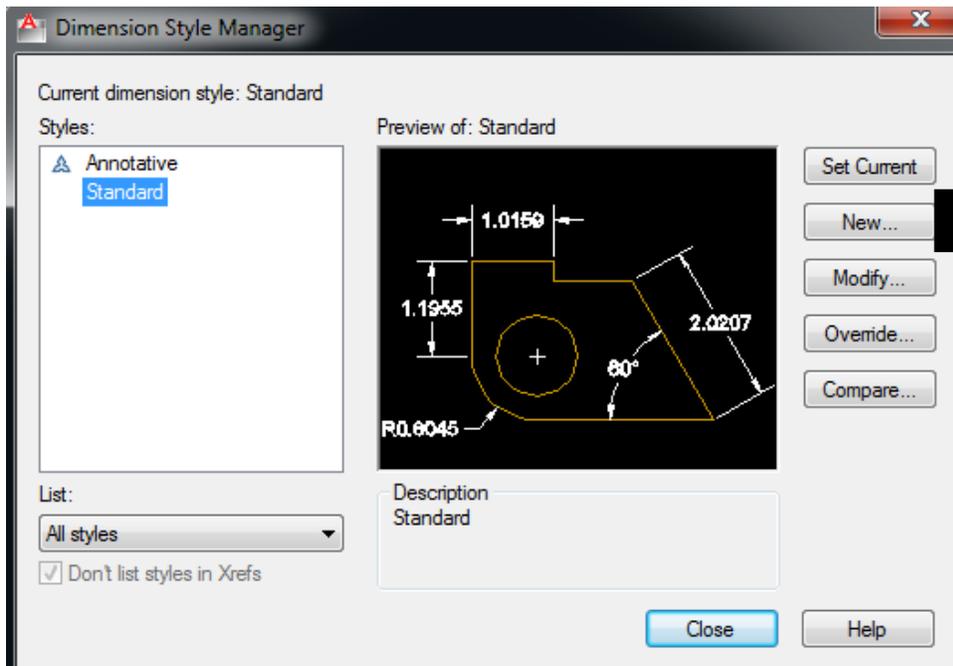


Creating AutoCAD Dimension style:

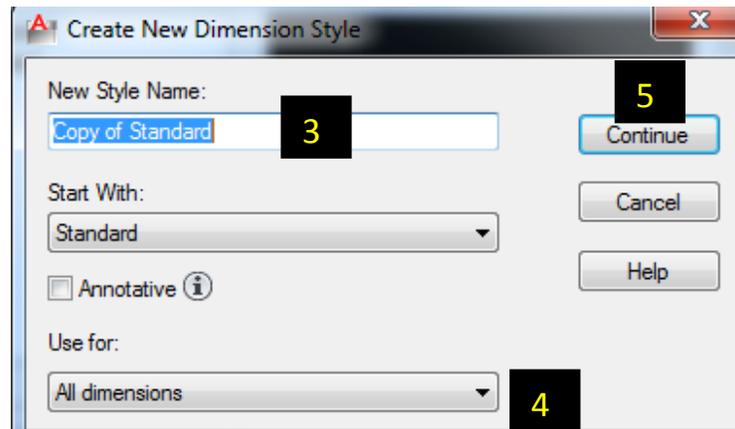
When using dimensions we constantly have to modify the standard style from AutoCAD to fit our needs. Creating a dimension style is one of the answers to this problem. By creating a dimension style we can have different styles for specific scales, etc... Plus we can always transfer the style to another drawing or save it as a template for future use. Not to mention, that also saves you time.



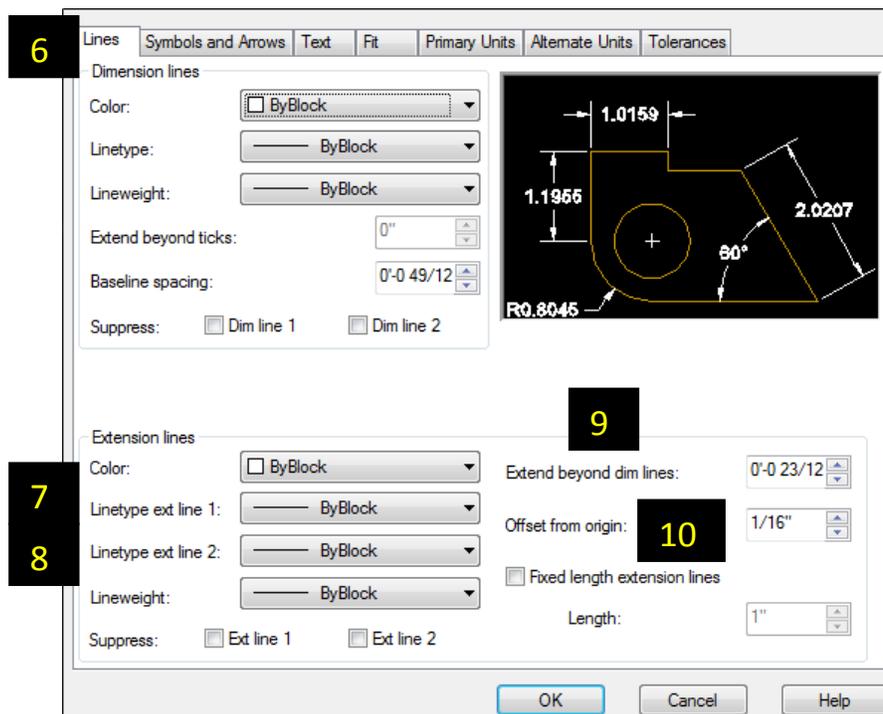
To create a dimension style, follow the steps below:

1. Go to **Format** then select  **Dimension style** or type **Dimstyle** in the command line.
2. Select **New** to start from scratch, otherwise select any existing *styles* on the left to modify. For this tutorial we will create a new style.
3. When prompted, specify a name for the new style or create the style based on the existing standard from AutoCAD by default or any other existing styles. Use the **Standard** one for now.
4. Specify the use for this new style on the pull down list. Use **All dimensions** for now, and no annotative for this tutorial.
5. Click on **Continue**.

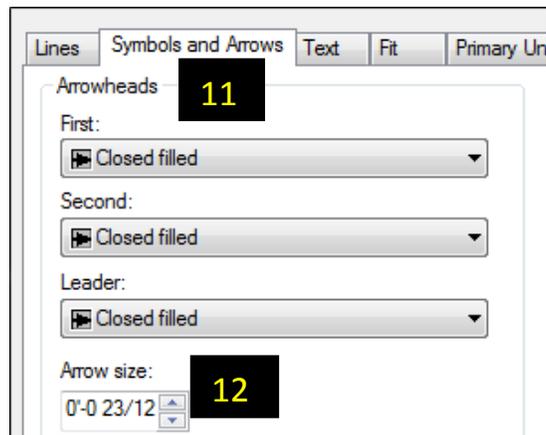
NB: There are many things to change in there. For this tutorial we will go over the most important ones.



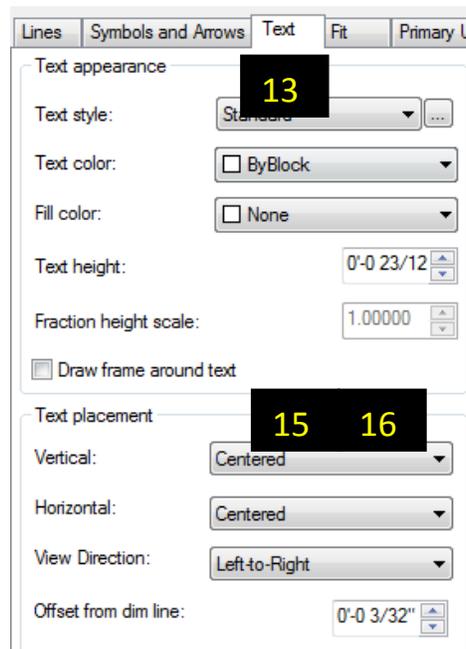
6. The tab **Lines** is divided into two categories: The **Dimension lines** and the **Extensions lines**. Under each section you can change their color and their [linetype](#) for the style.
7. Select **Suppress Dim line 1 or 2** to hide dimension lines from left to right.
8. Select **Suppress Ext line 1 or 2** to hide extension lines from left to right.
9. Under **Extension lines** category: Change **Extend beyond dim lines** accordingly.
10. Edit **Offset from origin** to adjust how far you want the dimension from the object.



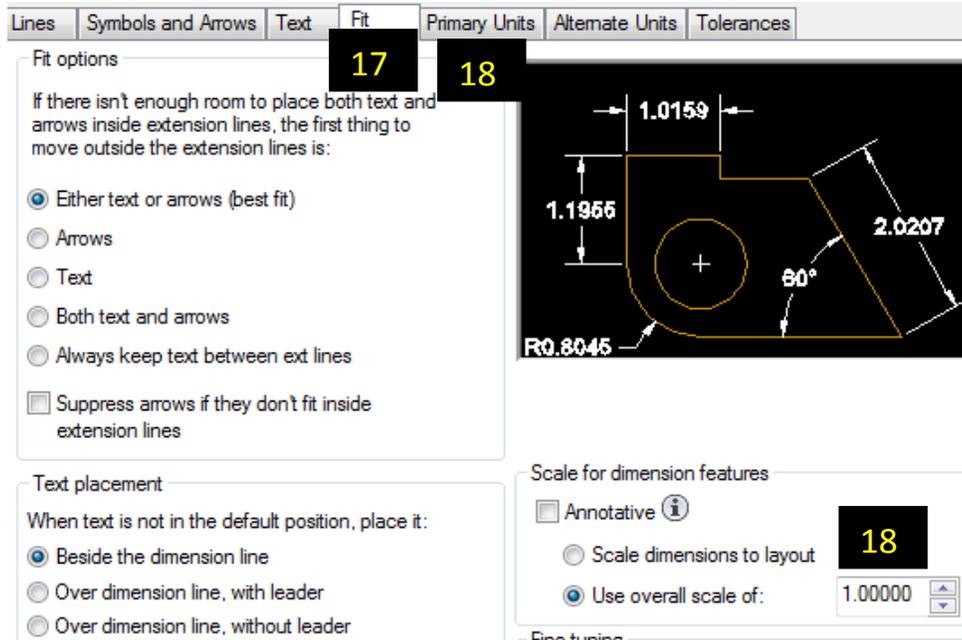
11. On **Symbols and arrows** select what kind of arrow heads you like for first, second dimensions and leader.
12. Specify how big the arrow head should be under **Arrow size**.
13. Under the **Text** tab you can select a text style if you have already created one or you can specify a font style. You can use that in conjunction with our [handy scale table list](#) if you are not sure what size to use for your drawing scale.



14. Specify a height for your text.
15. Specify text placement and orientation under the **Text placement** category.
16. Select your Text alignment.



17. Under **Fit** tab select either text or arrows (best fit) for fitting.



18. Use **1** for overall scale dimension.

19. Under **Primary units** tab, select your format unit from the pull down list.

Those are the most essential parts of creating the dimension style. You can change the values for the rest of the categories to fit your needs. It should not be that difficult since as you change the values the screen preview on the top right changes automatically to confirm your changes.

If you need any further help, contact me using the [contact page](#) or visit the discussion forum site <http://www.2learncad.com/forum>.