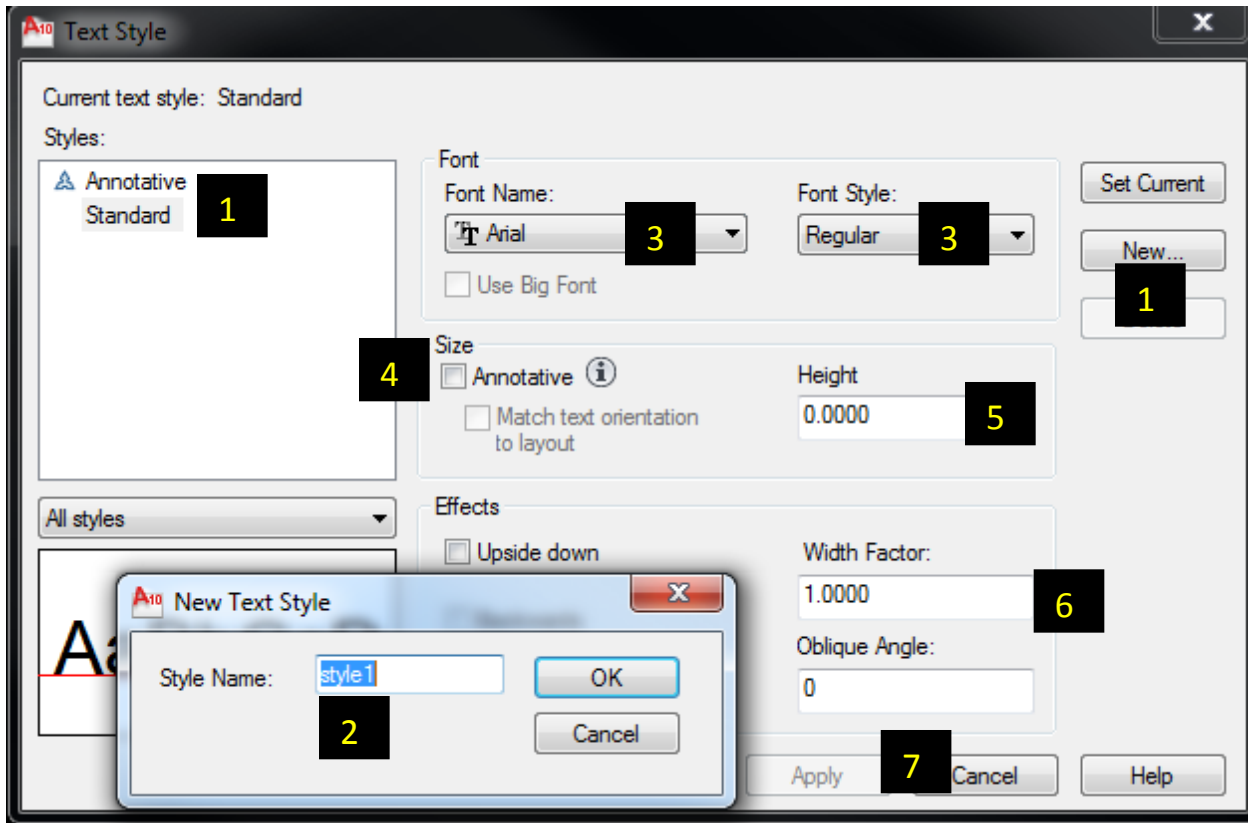


Creating AutoCAD text style:

In AutoCAD you can define many different text styles that will set unique fonts, sizes, angles, orientations and other characteristics. To create a text style:




In AutoCAD, go to **Format**. Select **Text Style**. When the Text style dialogue box appears:

1. You can either select Standard style or create a new style.
2. Enter the name for your new style.
3. Select a **Font Name and Style** from the drop down menu.
4. Check the **Annotative box** if you want annotation text.
5. Enter the **Text Height** for your style. This is the size of the text that will be displayed each time you apply your style.
6. Enter a **Width Factor**.
7. Click **Apply**.

You can edit the style at any time by repeating the steps above.

Method 2:

Select the text style symbol  from the format menu or type **Style** in the command line.

1. You can either select Standard style or create a new style.
2. Enter the name for your new style.
3. Select a **Font Name and Style** from the drop down menu.
4. Check the **Annotative box** if you want annotation text.
5. Enter the **Text Height** for your style. This is the size of the text that will be displayed each time you apply your style.
6. Enter a **Width Factor**.
7. Click **Apply**.

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