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:

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 $\text{A}$ from the format menu or type *Style* in the command line.

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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.2learnacad.com/forum](http://www.2learnacad.com/forum).*