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2.6 - USING TEXT TO WRITE ABOUT YOUR DRAWING

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Creating Text

Like most things in AutoCAD, there is more than one way to do it. Creating Text is no exception. Below are the common commands for working with text.

Command	Keystroke	Icon	Menu	Result
Text Style	STYLE / ST		Format > Text Style	Opens the Text Style Dialog
Single Line Text	DTEXT / DT / TEXT		Draw > Text > Single Line Text	Creates a single line of text
Multiline Text	MTEXT / T / MT		Draw > Text > Multiline Text	Creates formatable multiline text
Edit Text	DDEDIT / ED (or double-click)		Modify > Object > Text > Edit	Edits and formats text - edits attributes
Spell Check	SPELL		Tools > Spelling	Checks for spelling errors in Text, Attributes and Xrefs
Text along an arc	ARCTEXT		Express > Text > Arc-Aligned Text	Aligns text along a selected arc

Creating text in AutoCAD is easy. You may have already done this in the first level of tutorials, but this lesson will go into more detail and explore more options.

Formatting Text Styles

Most template drawings will have your text styles defined. Generally you won't change these. Occasionally you may need to create a new text style (definition of the way the text will look).

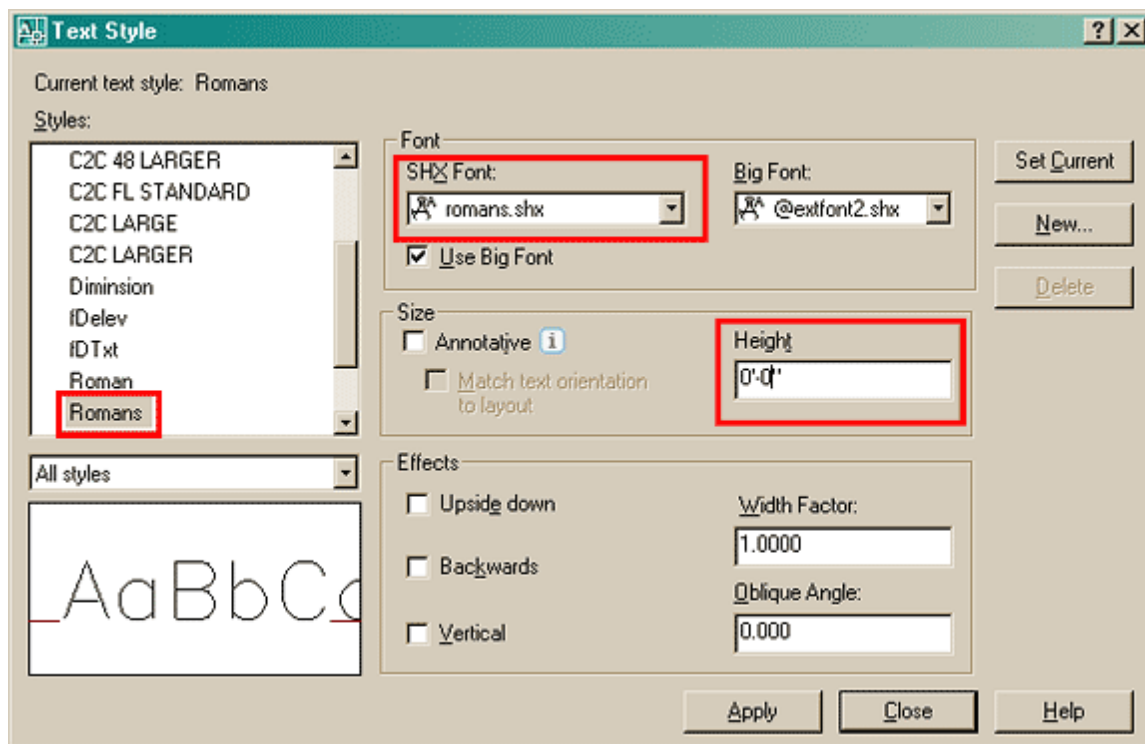
To format text in AutoCAD, you have to create a new text style for each different font and style of text.

By default in the acad.dwt file, AutoCAD loads the **txt.shx** font because it is simple to display on the screen. Unfortunately, it is a very basic and 'ugly' font. Here is an example of the default TXT font compared to the 'standard' ROMANS font. Look at the O's in particular. The TXT font has no curves.


TOO UGLY DEFAULT TEXT
SMOOTH ROMANS.SHX FONT

It is easy to load in a new style for all of your text. Here's how:

Use your pull-down menus **Format > Text**, or type **ST**, and this dialog box appears.



Select the **New...** Button and type in a name for your new text style.

Click on the **Font Name** edit box and select "romans.shx"  as your new text style. This style will be used for all text and dimensioning in the next assignments. This is a common, clean font that AutoCAD can display quickly on the screen. No other adjustments are needed. Optionally you can set the width factor to a .8 - this allows you to fit text in narrower spaces at an 80% width.

IMPORTANT: Do **NOT** change the Height in this dialog box unless you really know why you are doing it. If you set a height here, AutoCAD will not be able to scale if for uses such as dimensioning, in other words, it is fixed at the height you set.

Annotative text is a new feature in AutoCAD 2008. Click on the  in the dialog to read more about it.

Once you get into more complex drawings, you will need more than one textstyle (if they aren't in the

template). One for notes, one for tables, one for the table header, two or three for the title block - I think you get the picture.

If you need more than one text style, click on the new button, type in a new name and press OK.



Now the Text Style dialog box (as shown above) will have the new name in the Style Name field. Select **RomanD.shx** for this new style.

Now when you add text to your drawing, you have 2 choices. Watch the command line to know when to change to another style.

Adding Text to a drawing

Of course, there are a few ways of entering text in AutoCAD. For simple one line text, use the **TEXT** command. There are a few parameters needed to add text, but if you follow the command line, it's quite easy.

Command: **TEXT**

Current text style: "Romans" Text height: 0'-6" Annotative: No
Specify start point of text or [Justify/Style]: <PICK>
Specify height <0'-6">: 12

Specify rotation angle of text <0.000>: <ENTER>

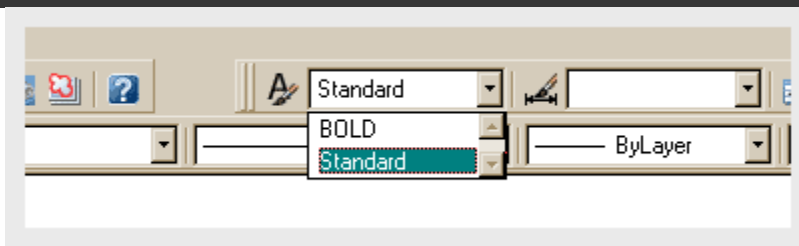
TYPE YOUR TEXT and press <ENTER><ENTER>

You should now have a line of text in your drawing. Start the command again and when prompted for Style, enter the name of the second text style you created.

Once you have used the two different styles, it should look something like this:



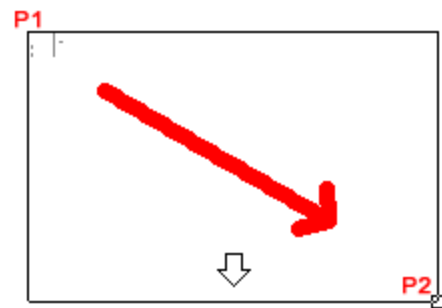
In later versions of AutoCAD, there is an easy way to change existing text to a new style. First, select the text, then go up to the Text Style Menu (usually towards the top of the screen) and select the style you want from the droplist (below) and the text will change automatically.



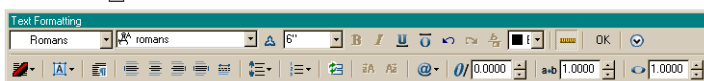
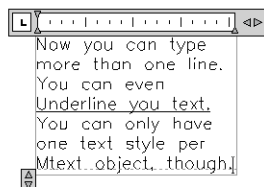
Professional looking and clear text is essential in any CAD drawing. Practice with different styles until you are comfortable with these concepts.

Multiline Text

Often you will want to enter a block of text, maybe a paragraph explaining a problem. In this case, MTEXT is what you need. Mtext allows you to format your text much like a word processor.



Start the Multiline Text command, and pick a spot in the drawing. Drag the cursor over to create a 'rectangle' to represent the area you want your text in, and pick the other corner. As soon as you pick the second point (P2), the full Multiline text editor will appear. Type your text, and press OK when done.



Click on the image above for a better look.

If you use a True Type font for your text style, you have more options for formatting (just don't get carried away).

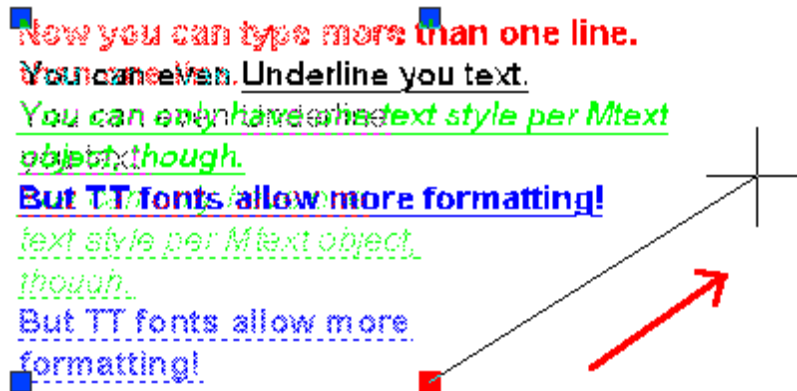


NOTE: AutoCAD uses two types of fonts: true-type and 'SHX' vector fonts. True-type fonts are indicated by a small **T** symbol beside the font name. As a rule, you will want to use the SHX fonts, as AutoCAD can draw these faster. Newer releases of AutoCAD handle true-type faster than before, but they can still slow down your display speed. Another concern when using True-type fonts is that if you are exchanging drawing files with other people, they may not have the same fonts installed on their system. This could lead to formatting problems when they open your files.

Editing text

The easiest way to edit the contents of your text (what you wrote) is to double click on it. Depending up your method of entering the text, a different editor will appear (Mtext or single line text).

If you want to change the shape of your Mtext box, just pick on the text, and use grips to drag the bottom right corner.



Single line text cannot be changed like this. When deciding when to use single or multiline text, think about how the text will be used, if it might be edited later, it's generally a safer bet to use Mtext.

Arc-Aligned Text

Recently AutoCAD added a selection of commands called "Express Tools". These are regular AutoCAD commands, but they aren't fully supported (meaning don't cry to Autodesk if they don't work). They are also sometimes installed separately from the regular AutoCAD installation, so you may not have them on your screen. But they do offer a nice option to some drawing problems. One of these Express Tools is Arc-Aligned Text. Just like the name suggests, you can place text along an arc.

Draw an arc and start the command as shown above. As prompted, select the arc, and this dialog box will appear:



As you can see, there are quite a few options. Use the default settings and you should have text that looks something like this:

TYPE YOUR TEXT HERE

You can also erase the arc and the text will remain. This isn't a command that you will use a lot, but it is comforting to know that it's there.



[View the video about using Text in AutoCAD.](#)

Review: Text in a drawing should be clear, concise and uncluttered. Always use CAPS unless you're told otherwise. Use different styles for different situations, but don't get carried away. Use Multiline text when creating more than one line of text. Check your spelling. Reread what you just wrote and ensure that it will make sense to someone else reading it that is not familiar with the drawing.



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