

Chapter 14

Text and Mtext Annotation

Annotation is a very important component of any document. On any given drawing, annotation can range from a single word to a column of paragraphed text. Being able to efficiently create and edit text directly affects your productivity. In this chapter, you learn how to do the following:

- Determine your text height
- Define a text style
- Use single-line text
- Create and modify mtext
- Perform a spelling check
- Look at additional text options
- Perform text property modification
- Invoke Quick Text mode, mapping fonts, and the Windows clipboard

Choosing the Correct Text Height

The hardest part of adding text is deciding on the correct text height for the scale for which the drawing is set up. Use Tables 14.1 and 14.2 to help you determine the correct text height. To use these tables, go to the row associated with your drawing scale, and then move along the row to the column associated with the height you want your text to have on your plot.

Table 14.1 Text Heights for Architectural Scales

Drawing Scale	Plotted Text Heights				
	3/32"	1/8"	3/16"	1/4"	3/8"
1/16"=1'	18"	24"	36"	48"	72"
3/32"=1'	12"	16"	24"	32"	48"
1/8"=1'	9"	12"	18"	24"	36"
3/16"=1'	6"	8"	12"	16"	24"
1/4"=1'	4.5"	6"	9"	12"	18"
1/2"=1'	2.25"	3"	4.5"	6"	9"

Table 14.2 Text Heights for Decimal Scales*

Drawing Scale	Plotted Text Heights				
	3/32"	1/8"	3/16"	1/4"	3/8"
1:10	0.9375	1.25	1.875	2.5	3.75
1:20	1.8750	2.50	3.750	5.0	7.5
1:50	4.6875	6.25	9.375	12.5	18.75
1:100	9.3750	12.50	18.750	25.0	37.5

*All values in basic drafting units

INSIDER TIP

The simple equation to determine the size to make your text is the plotted scale multiplied by the desired text height. For example, at the scale 1/4"=1'-0", divide 12" by 1/4", or 12.0/0.25, which equals 48.0. Then multiply by the desired text height, 1/8", or 48.0×0.125, which equals 6.0. This calculation can be done directly in AutoCAD using the CAL command >> Expression: 12/(1/4)*0.125.

Choosing a Text Style

The appearance of text is controlled through a drawing named object known as a text style. The default text style supplied in a blank drawing is STANDARD, whereas in other template drawings, several text styles can be predefined for you. Although the default text style STANDARD is available, most users will generally want to customize the appearance. You use the STYLE command to create a new text style definition prior to creating new text objects.

Defining Text Styles

You can define as many text styles as you need in a drawing. You can access the STYLE command by choosing Text Style from the Format menu. Figure 14.1 shows the Text Style dialog box. The various settings within the Text Style dialog box are explained in more detail later in the chapter.

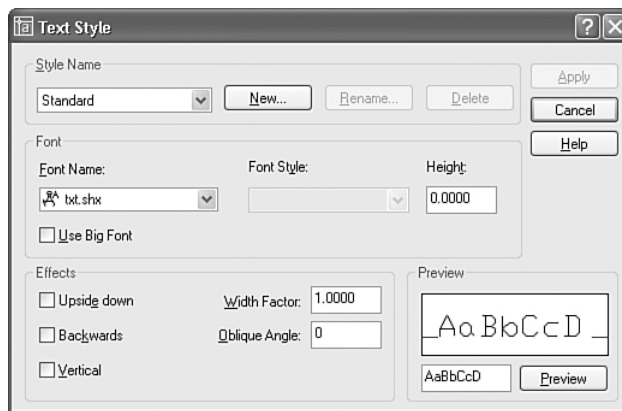


Figure 14.1 The Text Style dialog box.

To create a new style, click the New button. It uses the current style as a prototype for the new one. If the current style is not the style you want to begin with, select the desired style from the list of existing styles (thereby making it the current style).

After clicking the New button, you type a name in the New Text Style dialog box. A duplicate style is then created from the selected style and assigned the name you provided. To rename an existing style, select the style from the list of existing styles, click the Rename button, and enter a new name. To delete an existing style, highlight the name from the list of existing styles and click the Delete button. The STANDARD text style cannot be deleted or renamed. The current text style cannot be deleted either.

INSIDER NOTE

When a text object is created, the style with which it is created is stored as a property of the object. A text style can be deleted only if no existing text objects reference the style.

INSIDER CAUTION

The STANDARD text style cannot be renamed using the STYLE command but can be renamed using the RENAME command. However, doing so has been known to cause AutoCAD instability—it is suggested you avoid renaming it.

Text styles are stored in the drawing in which they are defined. If you want multiple styles to be available in a new drawing immediately, define the styles in your template drawings. If you want to import a style from another drawing, use AutoCAD's DesignCenter. For more information on importing text styles into the current drawing, refer to Chapter 12, "Applications for DesignCenter."

When defining a new style or modifying an existing style, you must choose a font file, a text height, a width factor, an oblique angle, and the special effects you want enabled. Choosing these settings and previewing the results of these settings are covered in the following sections.

Choosing a Font and Style

The font file is the file that contains the information that determines the look of each character. Table 14.3 lists the various types of font files supplied with AutoCAD.

Table 14.3 Font Files Types

Filename Extension	Font Type
SHX	AutoCAD's native fonts, known as an SHX font file.
TTF	TrueType font file or TTF.

TrueType fonts are supplied with Windows or other Windows applications.

INSIDER NOTE

An SHX file may also be a Big Font file. Big Fonts are used to support text shapes that are non-ASCII. For instance, you can create symbols to use as text in a Big Font file without affecting a text style's normal ASCII characters. Big Fonts are also used to support other languages, such as providing the Kanji character set.

You can use PostScript files in AutoCAD. To do so, you must first use the `COMPILE` command to compile the PostScript (PFB) font file into a shape (SHX) file.

AutoCAD supports TrueType font families, which means that for some TrueType fonts, you can choose a font style such as regular, italic, bold, or bold italic. Note that not all TrueType fonts have more than the regular style defined.

Two system variables affect the plotting of text using TrueType fonts: `TEXTFILL` and `TEXTQLTY`. When `TEXTFILL` is disabled, the characters are plotted in outline form only. If `TEXTFILL` is enabled, the characters are filled in. The value of `TEXTQLTY` affects the smoothness of the characters at plot time. The value of `TEXTQLTY` can be set from 0 to 100, with the default value set to 50. The higher the value, the better the resolution of the characters, but the longer it takes to process the drawing for plotting. Both system variables can be typed at the command prompt and then set to the desired value.

INSIDER TIP

Using the simplest-shaped characters minimizes the drawing size and speeds up opening and working with the drawing file. Simpler-shaped characters are those that use very few elements, or line segments, to define a character's shape. The characters in the Simplex and Romans SHX files are quite simple in appearance and are similar to the Leroy characters used previously in manual drafting. Some shape files contain the alphabet of foreign languages, such as `GREEKS.SHX`, or even symbols, such as `SYMUSIC.SHX`.

After you change the font file associated with an existing style and apply the change, all text that has already been created with the modified style is updated to reflect the change.

INSIDER NOTE

The density of the fill of a given text font, including solid fill TrueType fonts, can increase the size of a plot file dramatically. It has no effect on the drawing file size itself, however.

Setting a Height

Also found in the Font area of the Text Style dialog box is the text height setting. The default height of 0 dictates that the user is allowed to set the text height at the time the text is created. A height other than 0 forces the text height for that particular style to that height. The style is then referred to as a fixed height style, and any text command height prompts are suppressed.

Changing the text height setting of an existing style does not affect the appearance of existing text objects.

Previewing the Text Style Settings

The character Preview area enables you to view a sample of the selected style and the results of changing the various settings. To view your own sample text, type your sample text in the text edit box and click the Preview button.

Specifying Special Effects

The Effects section of the Text Style dialog box contains the Upside Down, Backwards, Vertical, Width Factor, and Oblique Angle settings. These settings are covered in detail in the following sections.

Upside Down, Backwards, and Vertical Text

In the Effects area, you can enable the Upside Down, Backwards, and Vertical settings. See Figure 14.2 for an example of how these settings affect the appearance of text.



UPSIDE DOWN

BACKWARDS

VERTICAL

Figure 14.2 The effects of Upside Down, Backwards, and Vertical text settings.

INSIDER NOTE

Although the Upside Down and Backwards options work with all font files, the Vertical setting works only with specific SHX files.

Unlike the font file setting, the Upside Down and Backwards settings for an existing style can be changed without affecting text that has already been typed in that style; the text is not automatically updated to reflect setting changes. Changing the Vertical setting, however, does affect existing text objects, so you may want to create a new style before changing the Vertical setting.

INSIDER TIP

The Backwards option is useful if you want to plot text on the back side of a transparent plot sheet so that the text is readable when viewed from the front.

The Vertical option is useful when you need to add text in a column format like a schedule heading.

Setting a Width Factor

The Width Factor determines the width of the characters. A factor of 1 results in the characters being displayed with the width as defined in the chosen font file. A factor greater than 1 results in wider characters; a factor less than 1 results in thinner characters. Figure 14.3 illustrates the effects of using different width factors. All three text objects were added with the same text height.

Skinny Letters With Width Factor of 0.5
Normal Letters With Width Factor of 1
Fat Letters With Width Factor of 1.5

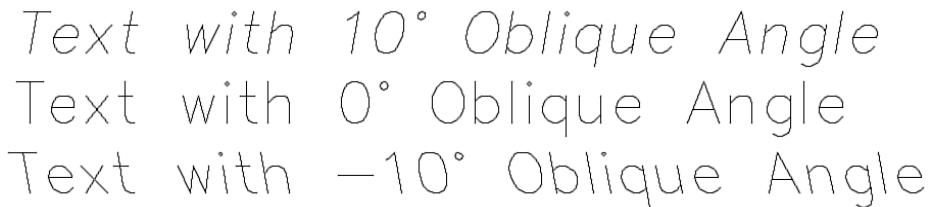
Figure 14.3 The effects of the Width Factor setting on text objects.

INSIDER TIP

Adding text with a width factor that is less than 1 may make it easier to squeeze text into an already crowded drawing while still keeping the text readable.

Setting an Oblique Angle

The Oblique Angle setting affects the slant of the characters. It is often used to add italic text when the characters in the font file being used are not naturally italic. Unlike the text rotation angle, the oblique angle of 0 refers to a vertical direction (see Figure 14.4). A positive angle value makes the letters lean to the right, and a negative value makes the letters lean to the left.



Text with 10° Oblique Angle
Text with 0° Oblique Angle
Text with -10° Oblique Angle

Figure 14.4 The effects of the Oblique Angle setting on text objects.

Now you have an understanding of the `STYLE` command and how to define a new text style and adjust the properties. Next, you learn about the various types of text objects.

Single-Line Text

A single line of text can consist of a single character, a word, or a complete sentence. The simplest form of text is created by using the `TEXT` command. To insert a single line of text, use the Draw pull-down menu and choose Text, Single Line Text. The initial prompt displayed in the command window presents a few options:

Specify start point of text or [Justify/Style]

INSIDER NOTE

A non-dynamic version of the `TEXT` command is available by typing `-TEXT`. This other command version can be used to create text by using AutoLISP or scripts.

INSIDER TIP

When you enter text, you can take advantage of the command history to repeat previously entered text. You use the up and down arrow keys to scroll through the command history.

The default option is to specify the lower-left corner, otherwise known as the start point, of the new line of text. After picking the start point, you are prompted to supply the height (unless the height is defined in the style being used), the rotation angle of the text, and the actual text to be created. As you type the text, it is displayed on your drawing. If you make a typographical error, you can use the Backspace key to delete the error and retype the text. You signify the end of the line of text by pressing the Enter key, at which point you can begin a new line of text immediately below the text object just created. To stop adding lines of text, press the Enter key without typing any new text. You can also relocate the next line of text by picking a point with the cursor. You may skip a line of text by pressing the spacebar and then pressing Enter. Note that this does not create a text object for the blank line.

INSIDER TIP

The spacing between successive lines of text is fixed by the font used for the text style. In many cases, this spacing is at a factor of approximately 1.666 of the text height. Each line of text created with the TEXT command is a separate object and can be moved independently.

If, after you end the TEXT command, you want to add an additional line of text below the last line created, you can easily do so by issuing the command again and pressing the Enter key instead of picking a new start point. TEXT then adds the new line of text right below the last one, using the style, height, and rotation angle of the previous line. You can also do other activities, such as drawing lines, erasing objects, and placing dimensions, and still continue with a new line of text below the last one.

INSIDER TIP

To help you spot the last line of text created, that line is highlighted when you begin TEXT. The highlighting, however, may not be apparent if the text is too small or off the screen.

Typing the text you want to create is the easy part. It is also important to know how to format the text according to your needs. The following sections discuss how to choose the correct text height, justification, and text style.

Choosing a Justification

The default option of TEXT is to specify the left endpoint, or the start point, of the line of text. Specifying the Justify option at the initial prompt displays the following prompt:

Enter an option [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]

Figure 14.5 shows the various TEXT justification options and their corresponding locations.

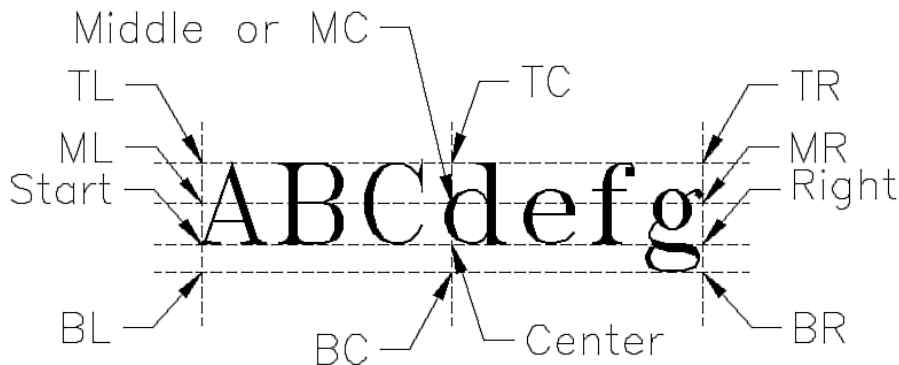


Figure 14.5 Justification points for a standard piece of text.

Unlike the justification options illustrated in Figure 14.5, the Align and Fit options require you to define two points.

Use the Align option when you want to specify the left and right endpoints of the text and do not care about the resulting height. The text height is automatically set to make the text fit between the specified points. Also, the angle from the first point to the second point is used as the rotation angle of the text.

Use the Fit option when you want to specify the left and right endpoints and the height of the text. To make the text fit between the specified points, AutoCAD varies the width of the text characters. Therefore, you may end up with skinny-looking characters on one line and very fat-looking characters on the next.

INSIDER TIP

You can enter the required text justification option when the TEXT command prompts for the start point, which eliminates the need for first selecting the Justify option.

When the text is initially added with one of the alternate justification options specified, it is displayed left-justified, as if the default justification were being used. When the TEXT command ends, the text is relocated and set with the correct justification.

INSIDER NOTE

You can snap to the justification point of a text object by using the Insert object snap mode.

Editing Single-Line Text

Two commands are of particular use for editing existing text: DDEDIT and PROPERTIES. DDEDIT is easier to use than PROPERTIES when all you want to do is change the text string. PROPERTIES is more powerful than DDEDIT in that it displays the Properties dialog box, which enables you to change all the properties of the selected text.

Using DDEDIT

From the Modify menu, click Object, choose Text, Edit to issue the DDEDIT command. Select the text object to be changed, and the Edit Text dialog box appears, displaying the selected text (see Figure 14.6).

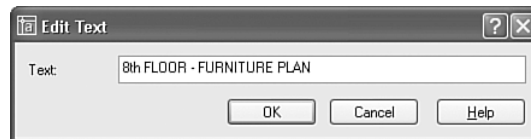


Figure 14.6 The Edit Text dialog box.

INSIDER TIP

If you double-click on a text object, it opens up in the appropriate text editor—a handy thing to use when you need edit your text objects.

Initially, the entire line of text is highlighted and is replaced by whatever you type. If you want to edit a specific portion of the text, position the cursor at the desired point in the text and pick it. You also use the Insert, Delete, and Backspace keys to add and delete characters.

If you want to replace a portion of the text displayed in the Edit Text dialog box, highlight the portion to be replaced and then type the new text.

Using Special Formatting Codes and Symbols

You can do a limited amount of formatting with the TEXT command. For instance, you can add a line under or above the text simply by adding the codes %u (for underlining) and %o (for overlining) to the text as you enter it. The codes act as toggle switches; the first time you include the code in a text object, it turns that effect on and is applied to the successive text characters. The second time you enter the code in the same text object, the effect is turned off. If you do not enter the code a second time in the text object, the effect is continued to the end of the text object but is not continued to the next text object. For example, to add the text shown in Figure 14.7, you would type this line of text: %uUnderlining%u and %oOverlining%o can be used separately or %o%utogether.

Underlining and Overlining can be used separately or together

Figure 14.7 Using underline and overline formatting codes.

Using Properties

Choose Properties from the Standard toolbar or the Modify pull-down menu to launch the Properties palette, as shown in Figure 14.8. The Properties palette enables you to change the text contents, style, justification point, and various settings that control the appearance of the text object.

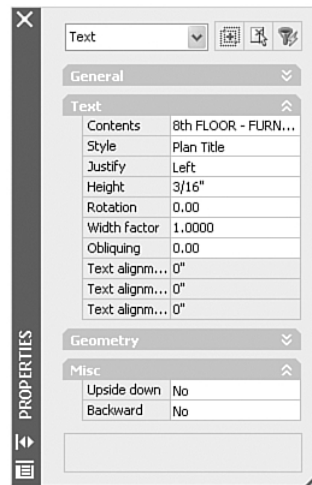


Figure 14.8 The Properties palette.

As you can see, the Properties palette provides an edit area for changing the text value, although it is generally used only for manipulating the format of text objects. The next section delves into the mainstream annotation type, MTEXT.

Multiline Text

The TEXT command can be used to add multiple lines of text, yet each text object is added as a separate object. Often, you want to add multiple lines of text as a single object. In such cases, you need the MTEXT command, issued by choosing Multiline Text from the Draw toolbar or from the Text submenu of the Draw pull-down menu.

After you start the MTEXT command, you are prompted to select the first corner point of a window. This window is used to determine the point where the mtext object will exist. By default, the mtext object is created with a top-left justification. If you want, you can change the justification type to one of eight others: TC (Top Center), TR (Top Right), ML (Middle Left), MC (Middle Center), MR (Middle Right), BL (Bottom Left), BC (Bottom Center), or BR (Bottom Right). These justification types are similar to those available with the TEXT command (refer to Figure 14.5), except that they apply to the whole mtext object and not just a single line of text.

You then indicate the desired width of the mtext window. A width of zero disables the automatic word wrap feature of the Text Formatting dialog box. With that setting, you must press the Enter key when you want to start a new line of text.

Several other command line options appear, but many of these are easier to set through the Text Formatting dialog box (see Figure 14.9). The Text Formatting interface is divided into two parts. The bottom part is the editor, and the top part controls the properties, which are described in detail in later sections.

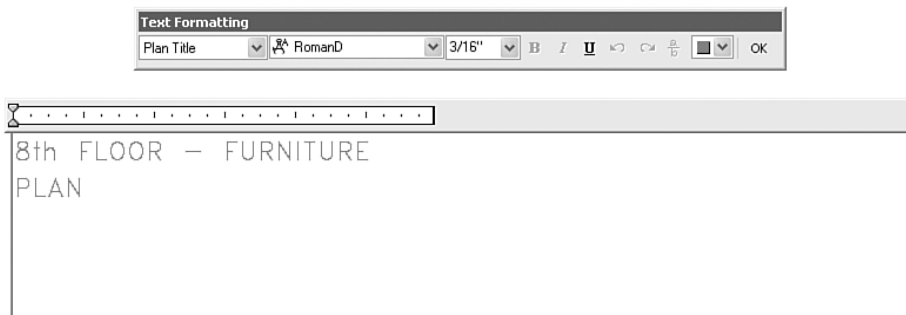


Figure 14.9 The Text Formatting dialog box.

Creating Mtext Objects

Previous versions of the Text Formatting editor were somewhat crowded with icons and tabs. It was revamped in AutoCAD 2004 and reduced to the bare essentials needed to produce text. The plethora of options still exists, however, and the options are now located in the right-click shortcut menu. As shown in Figure 14.10, there are many options related to creating mtext that can be taken advantage of.

INSIDER NOTE

Several options, Line spacing and Rotation, are still only available at the command line before the dialog box appears.

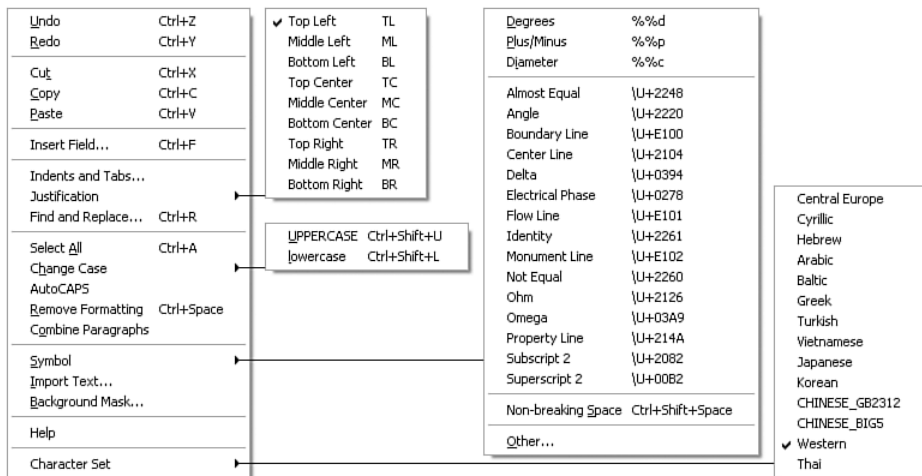


Figure 14.10 The right-click shortcut menu has many features, as well as several submenus.

In the following exercise, you learn about the various aspects of creating and editing mtext objects.

Exercise 14.1 Using MTEXT to Create Paragraphs of Text

1. Open the drawing 14EX01.dwg from the accompanying CD. It contains several predefined text styles that we will utilize.
2. Start the MTEXT command by choosing the Multiline Text tool from the Draw toolbar. AutoCAD reports at the command line:
Current text style: "Standard" Text height: 0.2000

This indicates which text style is current and what height will be active in the editing dialog.

3. At the Specify first corner prompt, go ahead and pick the upper-left corner mark.

INSIDER TIP

The little text sample attached to the cursor is adjustable. Normally it reads abc, but you can change the MTJIGSTRING system variable to whatever you would like it to read.

After you pick the first corner, you then are permitted to make changes to the current mtext settings, as shown in the prompt:

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]

4. Similar in nature to the TEXT command, you adjust the size of the text, its justification, and other properties prior to entering the editor. We pass on this now; go ahead and specify the opposite corner by using the lower-right corner mark. The Text Formatting dialog then appears.

INSIDER TIP

The AutoCAPS item on the right-click menu permits you to leave Caps Lock off, yet type text in the editor window in uppercase letters.

5. Within the editor window, type the following note:

**6"x1/2" CONT. EMBED PLATE WITH 3/4" DIA. x 5" LONG H.S. @ 24"
 ↳O.C. (STAGGER LOCATIONS OF STUDS WITH BRIDGING ANCHORS)**

Depending on how your system is configured, as you typed the note, you may have gotten the AutoStack Properties dialog, as shown in Figure 14.11. This dialog enables you to control how your fractions should appear and enables you to “set it and forget it.” Select your preference; the examples used here are similar to the Architectural style of drafting.



Figure 14.11 The AutoStack Properties dialog box is where you can tweak your fraction's appearances.

INSIDER TIP

If you need to adjust the AutoStack Properties dialog settings and have instructed it to not show again, you can still get to it. Highlight an existing stacked fraction and, using the right-click shortcut menu, choose Properties. Then on the Stack Properties dialog box, as shown in Figure 14.12, choose the AutoStack button to gain access once again.

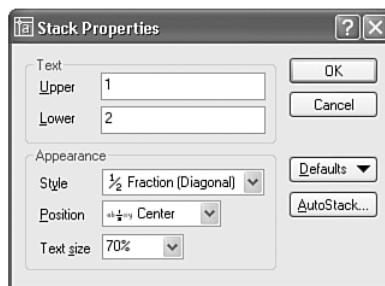


Figure 14.12 The Stack Properties dialog box enables custom stacking styles to be applied to an individual fraction.

- After typing the note, it is obvious that the current font is rather unappealing. Use the Style drop-down list and select Romans. Instantly, the text within the editor changes appearance to that of the Romans font.

INSIDER NOTE

The other drop-down list to the right of the Style drop-down is Font. It is not related to the Style control; in reality, it is similar to a dimension override, enabling specific portions of the mtext object to use another font. It should only be used if you must have a differing font embedded within the rest of the text.

Your multiline text is now complete. Press the OK button to close the Text Formatting dialog box. You should now have a text object that looks like Figure 14.13.

Leave this drawing open for the following exercise.

6"x1/2" CONT. EMBED PLATE WITH
3/4" DIA. x 5" LONG H.S. @ 24"
O.C. (STAGGER LOCATIONS OF
STUDS WITH BRIDGING
ANCHORS)

Figure 14.13 A typical mtext object.

This short exercise exemplifies the ease of use of the Text Formatting interface. Although the control area of the dialog is available, facilitating immediate changes to text properties, productivity can be gained by presetting the various properties prior to actually showing the interface. Some of the controls are also available through toolbars and system variables.

Customizing for Text Styles

AutoCAD 2005 comes with two toolbars for creating and manipulating text objects. Shown in Figure 14.14, the Styles toolbar contains a drop-down list for setting the current text style. Also shown in Figure 14.14 is the Text toolbar, which has tools to create single and multiline text as well as other editing features.

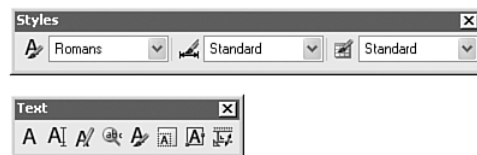


Figure 14.14 Toolbars available to control, create, and edit text and mtext objects.

System variables affect the creation of text objects and their preset properties (see Table 14.4). These can be used in your customization methods to speed setting up your environment.

Table 14.4 Text Formatting System Variables

System Variable	Description
TEXTSIZE	Sets the text height for all new text-based objects created.
TEXTSTYLE	Sets the current text style.

After you begin to customize your AutoCAD environment, you can take advantage of ways to set to specific properties and start the MTEXT command in one click. The following menu macro is an example:

```
^C^C_. -style;romans;romans.shx;;0.8;;;;;
```

When executed, this creates the text style Romans, assigning ROMANS.SHX to it and leaving all properties except for width set to 0 or off. The width factor is set to 0.8, providing a slightly narrower text style:

```
^C^C(setvar "TEXTSIZE" (* (getvar "DIMSCALE") (getvar "DIMTXT")))
```

This short AutoLISP code first retrieves the current DIMSCALE value, which is used to control how large the parts in a dimension object appear. The code then gets the current dimension style text height, multiplies these two together, and applies that size to the system variable TEXTSIZE. Click this button, and the next time you create text, it will match your current dimension style appearance.

Customization such as this can be placed within your Custom menu, on a Tool palette, or on a toolbar, or it can even be coded as small AutoLISP programs. As you develop the formatting specifics, you will learn where best to house them for the greatest effectiveness.

Editing Multiline Text

Very often, you will have to change the content in a mtext object. Early versions of AutoCAD required the user to erase the text and re-create it. Luckily for you, editing isn't that bad now. Just as easily as you create the content, you can later edit it as needed.

Exercise 14.2 Editing Mtext Objects

1. Continue from the previous exercise. If needed you can open the drawing 14EX02.dwg from the accompanying CD. It contains a single mtext object and some reference polylines.
2. Double-click on the mtext object, utilizing AutoCAD's double-click editing feature. The MTEDIT command starts, and the Text Formatting dialog box appears.
3. Locate on the top line CONT. and highlight the period. Then type **INUOUS**, eliminating the abbreviation. You probably noticed that a few words moved down to the next line. That's a key feature of mtext—word wrapping.
4. Now highlight the word PLATE. Using the right-click shortcut menu, choose Symbol and select Property Line. The word PLATE is then replaced with a new AutoCAD 2005 PL symbol.

INSIDER TIP

As you work with the MTEDIT dialog, you may notice it moves around to located itself over the text being edited. If this functionality is bothersome, you can use the MTEXTFIXED system variable to eliminate that activity. When turned on (1), the dialog enables you to place it on-screen wherever you want, and it displays there until moved. Additionally, turning this system variable on makes the editor background match the display background color. When off, the dialog background is translucent to show objects behind it.

5. Next, locate the word DIA. and highlight it. Use the right-click shortcut menu and select Symbol. At the top of the Symbol submenu, select Diameter %%c. It doesn't look like much now; you will have to wait until the command ends.
6. Locate the O.C. and highlight it. Again use the right-click shortcut menu and select Change Case. Select lower case to change the case for the ON CENTER reference.

INSIDER TIP

You can also use Ctrl+Shift+U or Ctrl+Shift+L to change selected text case to upper or lower, respectively.

7. Click the OK button to close the editor. You can also click outside of the editing window to end the command.

You have edited the mtext object. You should also notice that the %%c now represents a diameter (\varnothing) symbol, as shown in Figure 14.15.

Leave this drawing open for the following exercise.

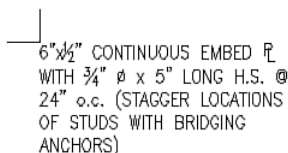


Figure 14.15 The edited mtext object with new text content and symbols.

There are a few specialty character combinations used by AutoCAD, as described in the following section.

Specialty AutoCAD Symbols

You can add symbols that are in the font file but that are not on the keyboard. Table 14.5 shows several formatting codes and the resulting symbols.

Table 14.5 Additional Formatting Codes

Formatting Code	Symbol	Meaning
%%c	\varnothing	Diameter
%%d	°	Degree
%%p	±	Plus/minus

The codes are not case-sensitive. In addition to the codes in Table 14.5, you can enter the code %%nnn, where nnn is a three-digit integer, to add any character in a font file.

AutoCAD 2005 comes with a series of new symbols. Shown previously in Figure 14.10, these new symbols require that you use new AutoCAD 2005 versions of SIMPLEX and ROMANS files as well new ISOCP and ISOCT series files.

INSIDER CAUTION

In the previous exercise, if you change the font file to Arial, for example, the PL symbol becomes undefined and represents itself as small square. Due to this characteristic, caution should be taken when using new AutoCAD 2005 symbols with drawings that will be used by others who may not be on the same release.

A much easier way to add a symbol is to use the Windows Character Map program. To use the Character Map program in place of the %%nnn code, simply start the Character Map program by selecting Other on the Symbols submenu and select the TrueType font file you have specified in the current text style. Then select the character you want to add and copy it to the clipboard. You can then paste the character into the text you are typing.

INSIDER CAUTION

Not all font files contain the same characters, which is why it is important that the font file you choose to copy from in the Character Map program is the same font file specified in the text style you are drawing with in AutoCAD. What the Character Map program actually copies when you copy a character to the clipboard is the character's position number in the font chart. When you paste that character into AutoCAD, the character corresponding to the position number recorded in the clipboard is drawn. Therefore, if you are using a different font file in AutoCAD, you may end up with a different character altogether.

MTEDIT Right-Click Shortcut Features

As you have discovered, there are many options available on the MTEXT right-click shortcut menu. The following sections discuss some of these features in detail.

Insert Field

Fields are a new AutoCAD 2005 feature for embedding dynamic text values within text-type objects such as mtext. Refer to Chapter 21, "Using Fields and Tables," for more information.

Indents and Tabs

With the addition of indentation and tab stops, using AutoCAD as a limited word processor is easier than before.

As shown in Figure 14.16, the highlighted row of text in these notes is utilizing both features. It has a tab stop to push the 2. text over. It has another tab stop to push start of the REDUCTION text over. It then has an indent for all lines after the first one, helping it line up with the previous line.

INSIDER NOTE

There is a dialog specifically for setting indents and tabs. Accessed from the Text Formatting right-click shortcut menu, choose Indents and Tabs, showing the Indents and Tab dialog.

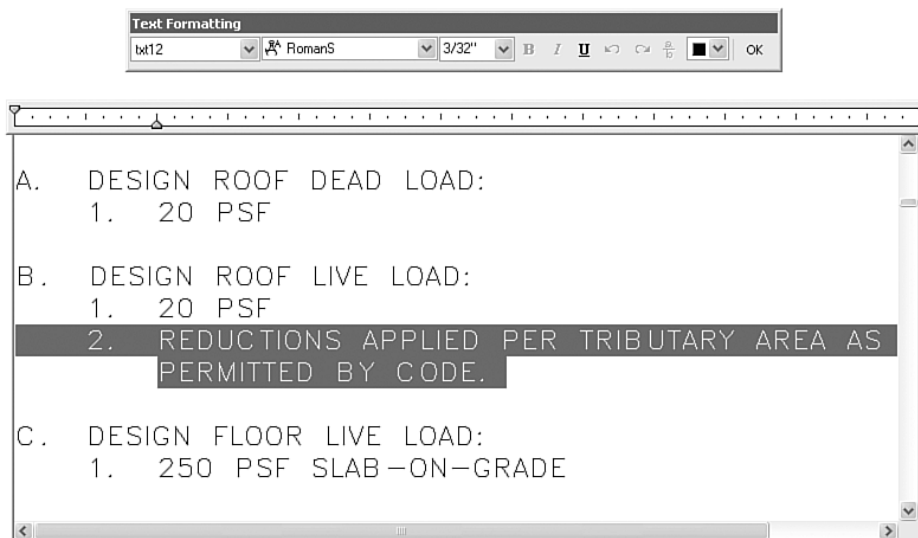


Figure 14.16 Indents and tab stops in mtext aid in alignment.

INSIDER TIP

You can adjust your indents within the editor window; just grab the appropriate indent marker and slide as desired. The top marker sets the first line indent; the bottom marker sets the indent for the rest of the paragraph. You can also place new tab stops by clicking on the ruler. Remove tabs by clicking on one and dragging it off the ruler.

Justification

After you are in the Text Formatting dialog, you can still change text justification if needed. The Justification submenu contains all available alignments for mtext objects. Just click the placement type you need, and the content of the editor instantly updates.

Find and Replace

Use the Find and Replace feature to search for a specific combination of characters and even to replace the text found with different text. If the Match Case setting is enabled, AutoCAD finds only text that exactly matches the case of the search string. If the Whole Word setting is enabled, AutoCAD finds only words that fully match the search string; otherwise, other words that contain the search string are located. After you specify the settings you want, use the Find button to start the search.

Stack and Unstack

The Stack/Unstack option is a slick feature in AutoCAD 2005 that is used to stack or unstack selected text. For example, you can designate selected text to be stacked by using special characters between the characters you want stacked. The text to the left of the character is placed on top of the text to the right of the character. To unstack stacked text, select it and then click the Unstack option.

AutoCAD 2005 provides for three stacked text types based on which of the following characters is used:

- **Carat (^)**. Converts selected text to left-justified tolerance-style stacking.
- **Forward slash (/)**. Converts selected text to center-justified text separated by a horizontal bar.
- **Pound sign (#)**. Converts selected text to a fraction-style stack separated by a diagonal bar.

You can edit stacked text and change the stack type, its alignment, or the size of stacked text in the Stack Properties dialog box, shown previously in Figure 14.12.

Import Text

If you have text in an existing TXT or RTF file, you can use the Import Text button to import the file into the editor, and then you can edit the text as needed. Click this option and then simply locate the text file to bring in. The content is read and placed within the editor at the current cursor location.

Background Mask

A great new feature in AutoCAD 2005 is text masking. Basically, this provides the capability have a colored background property applied to your mtext object. In addition, this mask can display the background of your screen—essentially blanking out other drawing content that may exist within the mtext object boundary.

Setting the Bold, Italic, and Underline Text Properties

The Bold and Italic buttons enable you to bold or italicize the text, but only if the chosen font file is a TrueType font. You can use the Underline button to underline any selected text, regardless of the font file used. All three buttons act as toggles, and you can turn their properties on or off by simply selecting the desired text and clicking the appropriate button.

INSIDER TIP

If you are using an SHX-based font, you can use the oblique angle to achieve the appearance of italics.

Color Settings

The Text Color list enables you to set the color for selected text. You can set the color to ByLayer, ByBlock, one of AutoCAD's 255 colors, or a True Color or Color Book setting.

Secondary Editing of Mtext Objects

You can use grips to move or change the width of the mtext object. When you select the grip point that corresponds to the justification point, you can move the mtext object. If you select any other grip point, you stretch the width of the mtext object. The following exercise shows the ease with which other mtext properties can be changed.

INSIDER TIP

A new object snap opportunity exists in AutoCAD 2005. Now the corners (same locations as the grips) can be targeted using the Node osnap mode. Primarily used to align the corners or edges with other objects, this feature can be controlled through the OSNAPNODELEGACY system variable. When set to 0, it is active; when set to 1, AutoCAD does not provide nodes for an mtext object.

Exercise 14.3 Modifying Mtext Objects

1. Continue from the previous exercise. If needed you can open the drawing 14EX03.dwg from the accompanying CD. It contains a single mtext object and two reference polylines made in the previous exercise.
2. Double-click on the mtext object, invoking the MTEDIT command.
3. Press Enter to create a new line. Then move the cursor back up to the top line and type **NOTE:**. Then select that new text and then press the U (underline) button. Notice that the Bold font property is unavailable. The current style's font does not include a bold component.
4. Highlight (if needed) the NOTE: text and, using the Font drop-down list, select Arial. It is near the top of the list. Choosing it changes just the selected text to that font.
5. Now the Bold font property is available. Select it to make the NOTE: bold.
6. Select the NOTE: text (if needed). In the size list, type in **0.25**, making the text noticeably bigger than the rest of the text.
7. Again, with the NOTE: text selected, use the Color drop-down list and pick red. Now the added text is bigger and more distinctive.
8. Use the right-click shortcut menu and select Background Mask. The Background Mask dialog box is presented. Turn on the Use background mask option, and leave the other settings as shown in Figure 14.17. Close the dialog. Press OK to close the Text Formatting dialog box.

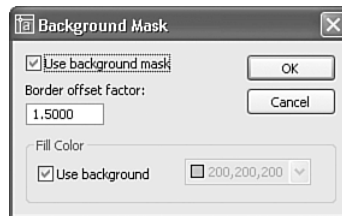


Figure 14.17 You can set an mtext background color with the new Background Mask feature.

You didn't see a visible change to the background of the mtext object. This is partly due to the fact that the background color and the mask are identical. To truly appreciate this feature, you need something to hide.

9. Draw a line from one corner mark to the other mark.

Again, no change. This is because the last object drawn is on top of previous items drawn. But that is easily corrected.

10. Select the mtext object so that its grips are activated. Then use the right-click shortcut menu and choose the Draw Order submenu, Bring to Front option. Instantly, the line is now hidden by the area of the mtext object, plus a little bit more. Notice in Figure 14.18 that the corner mark is also being clipped. This clipping is handled by the border offset factor, as shown in Figure 14.17. You can close this drawing without saving.

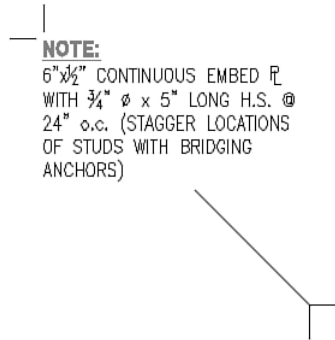


Figure 14.18 The revised mtext object, hiding all objects behind it.

INSIDER NOTE

It should be noted that the entire mtext boundary box is the space that gets filled with the mask. As the bottom line content shows, even though the text doesn't extend very far, the boundary masks that area. Also note that the Draw Order command is crucial to implementing this feature well. A given mtext object can mask all other things, including other mtext objects and dimensions that are below it in the draw order.

INSIDER TIP

Even geometry that is masked is still there, selectable and available.

Line Spacing Controls

Shown in Figure 14.19 is a portion of the Properties palette with the mtext object from the previous exercise selected. Most text properties can now be managed through the right-click shortcut menu found within the Text Formatting dialog box. A few exceptions exist, including the line spacing properties.

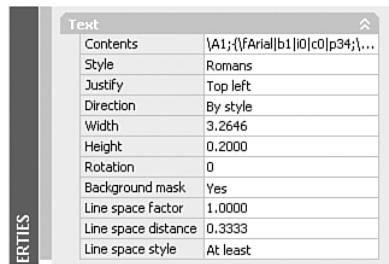


Figure 14.19 Mtext-related spacing features on the Properties palette.

The default Spacing Factor is 1.0, which sets the line spacing as a multiple of the single line spacing. Single spacing is 1.666 times the text height in most cases. The Space Distance value follows the factor value, but it shows the value as a true distance. For example, text with a height of 0.20 has a default distance of 0.333 (0.20×1.666).

The combination of these two settings enables the mtext object to have a minimum spacing while the Line Spacing Style is set to “At least.” If characters go above or below the line, such as a [or /, the lines adjust to maintain proper clearance. However, if you need a specific line spacing, for a schedule perhaps, you can set the Style to Exact and then the Spacing Distance to your desired setting.

Performing a Spelling Check

To check spelling within all your text-based objects, including attributes, issue the SPELL command by choosing Spelling from the Tools pull-down menu. When you start the command, AutoCAD prompts you to select the objects to check. If the SPELL command encounters an unknown word, the Check Spelling dialog box appears, and you must choose to replace the word, ignore the discrepancy, or add the word to your supplemental dictionary. If no errors are found, a message box appears, informing you that the spell check is complete.

Specifying the Dictionaries

The SPELL command looks up words in two dictionaries at any given time: a main dictionary and a supplemental dictionary. Several main dictionaries are supplied with AutoCAD; the default is the American English Dictionary. The default supplemental dictionary is SAMPLE.CUS (it contains a number of AutoCAD command words and terms). To change the dictionaries used by SPELL, issue the OPTIONS command. In the Options dialog box, select the Files tab, then change the Main Dictionary and Custom Dictionary File settings located under Text Editor, Dictionary, and Font File Names.

Unlike the supplemental dictionary, the main dictionary file cannot be modified or added to. You can, however, add words to update the current supplemental dictionary.

INSIDER NOTE

The SPELL command within AutoCAD 2005 can delve into all forms of annotation, including blocks, to locate misspelled text-type items. It also displays misspellings in xrefs but cannot correct them.

Creating a Supplemental Dictionary

A supplemental dictionary file is an ASCII-based text file that contains the additional words you want SPELL to use when checking for correct spelling. The supplemental dictionary contains one word per line. You can create as many supplemental dictionaries as you want, but you can use only one at time. When you create a supplemental dictionary, be sure to use a CUS filename extension.

Looking at Additional Text Options

This section covers several optional text handling features that may prove useful to you. These features enable you to speed up the display of text, handle missing font files, and insert text files into the current drawing.

Enabling the Quick Text Display

When AutoCAD opens or regenerates a drawing, if the drawing contains numerous text objects, it can take quite some time to complete a selection process, especially if the text uses complex fonts. If you want to speed up the process and do not need to actually see the existing text, enable the Quick Text mode (QTEXT). To enable the Quick Text mode, access the Display tab of the Options dialog box and select the Show Text Boundary

Frame Only check box. When Quick Text is enabled, text-based objects are displayed as simple rectangles that contain no characters. When you enable Quick Text mode, you need issue a REGEN command to redisplay the text as simple rectangles.

INSIDER NOTE

When Quick Text is enabled, new text objects are displayed as text characters while the TEXT and MTEXT commands are active, which makes it easier to read what you are typing.

Specifying an Alternate Font File

Font files are not stored with the drawing file. If a font file that is referenced in the drawing is not available when the drawing is opened, an error message is displayed. You are then prompted to choose a replacement font file. If you want to bypass all such error messages, you can specify a font file that is automatically used whenever a needed font file cannot be found. You specify this alternate font file in the Alternate Font File setting under Text Editor, Dictionary, Font File Names in the Files tab of the Options dialog box. The default alternate font is SIMPLEX.SHX.

INSIDER CAUTION

A couple of problems can occur when you use an alternate font. If the missing font file contains special characters that the alternate font file does not have, the text on the drawing may end up incomplete. Furthermore, because the space that a line of text occupies depends on the font file used to generate the text, you may find that when the alternate font is applied, the text on the drawing looks out of place or does not fit properly. The best solution is to obtain the correct font files and use them unless you are sure you have a suitable alternative font file.

Mapping Fonts

If you need to specify more than one alternate font file, use a font mapping file. A font mapping file is an ASCII-based text file in which each line in the file specifies the font filename to be replaced and its substitute font filename (separated by a semicolon). Note that you should include the extension if there could be any ambiguity. The default font map file is ACAD.FMP. You can identify a different font map file by changing the Font Mapping File setting under Text Editor, Dictionary, Font File Names in the Files tab of the Options dialog box.

Adding Text as Attributes

An alternate method to adding text objects that are to be incorporated into block definitions is to create attributes. Attributes behave much like text objects but have additional functions beyond displaying text. Attributes are discussed in more detail in Chapter 10, “Creating and Using Blocks.”

Dragging and Dropping Text Files

In Windows, you can drag a text file from the desktop or from Windows Explorer and drop it into your drawing. AutoCAD automatically inserts the file as an mtext object, using the current text settings for the text height, rotation angle, and text style.

Copying Text Using the Clipboard

You also can copy text from any application to your clipboard and paste the contents into your drawing. If you use the PASTE command, the contents are dropped into your drawing as an embedded object. If you use the PASTESPEC command, you can choose to paste the clipboard contents as text, in which case the text is an mtext object.

The clipboard operations depend on OLE (Object Linking and Embedding). For more information on AutoCAD’s OLE features, refer to Chapter 15, “Applications for OLE Objects.”

Text Property Modification

AutoCAD 2005 includes a few commands to aid the process of changing text properties such as height and justification. These new tools can help tasks that take hours to complete finish in mere seconds. Learning these features and how to leverage them in your everyday work is key to high productivity.

Scaling Text

Accessed through the Modify pull-down menu, Object, Text, the SCALETEXT command enables you to select as many text objects as needed to modify. After selecting your objects, you get a prompt

```
[Existing/Left/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR] <Existing>
```

where you can choose any number of methods to change the size of the text objects.

If you choose the default <Existing> option, the base point for the scaling procedure is based on the existing text insertion point. If you choose any other option, the scaling process is relative to that justification type. For example, TC would change the scale of the text relative to the Top Center of the text.

Following the choice of from where to scale the text, you get another prompt:

Specify new height or [Match object/Scale factor] <0.1000>

Here you decide how to change the size of the text. You can specify a literal size, pick an object on-screen to get a distance, or just provide a scale factor such as 1.5. This changes your text heights by 1.5 times. After that, the various text objects change.

INSIDER NOTE

The SCALETEXT command has no dialog interface.

Changing Text Justifications

Also accessed in the Modify pull-down menu under Object, Text, the new JUSTIFYTEXT command enables you to change the justification of multiple text objects very easily. You could of course use the Properties palette, but that adjusts the justification only about the insertion point. What this means is a piece of text that is left-justified at 0,0 when switched to right justification will graphically move to a -X value but keep the insertion at 0,0.

However, using the JUSTIFYTEXT command preserves all graphic locations of text and relocates the insertion points. If you have ever had this problem, you will recognize what a timesaver this is.

At the Select Objects prompt, gather all your text objects to change justification. Then another prompt appears:

[Left/Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR] <TC>

Choose which justification type you want and then press Enter. AutoCAD then changes the justification but doesn't move the text's first line visually. Of course, when you change the justification on mtext objects, the subsequent lines of text in the mtext object alter to honor the new justification.

INSIDER NOTE

The JUSTIFYTEXT command has no dialog interface.

Space Translation

SPACETRANS is a transparent command available in AutoCAD 2005. This command converts distances from space, such as model space, to an equivalent distance in paper space. This is typically needed for determining text heights between spaces. The program is transparent-capable, so it can provide the data to other commands.

INSIDER NOTE

A transparent command is any program that can be used while running another command. An example of this is the ZOOM command; you can zoom while drawing lines in the LINE command. To use a transparent-aware command, prefix it with a (') sign and press Enter.

Summary

AutoCAD provides a variety of tools that deal with adding and editing text. This chapter covered the various steps for creating and working with single lines of text in your drawings, as well as how to add multiple paragraphs of text using the MTEXT command. The chapter also provided detailed information on editing mtext and defining and changing text styles to control the appearance of text.

