Chapter 5

What's New or Impoved

Summary

In this chapter, you learn about:

Extended Symbol Names

Dimensioning

Text

Quickly Selecting Objects

In-Place Block/Xref Editing

Hatch Improvements

Partially Opening a Drawing

Other New Commands

Extended Symbol Names

You learned about long layer names in Chapter 4. Using AutoCAD 2000, the length of symbol names (layers, blocks, text styles, dim styles, views, UCS names etc.) has been expanded from 31 characters to 255. The following characters are not allowed: **< > / \ " : ? * | , = '**

AutoCAD 2000 preserves and displays the symbol names exactly as you type them, including uppercase/lowercase treatment. Internally, AutoCAD 2000 remains case-insensitive to symbol names, which means you cannot create two symbol names for the same symbol type that differ only in case.

You can enable or disable extended symbol names in a drawing: click **Tools ➔ Options ➔ System** tab, use the check box **Allow long symbol names**. This setting effects the current drawing only.

Dimensioning

Dimensioning enhancements include a redesigned Dimension Style Manager, which uses a preview pane to show you the effects of dimension style changes.

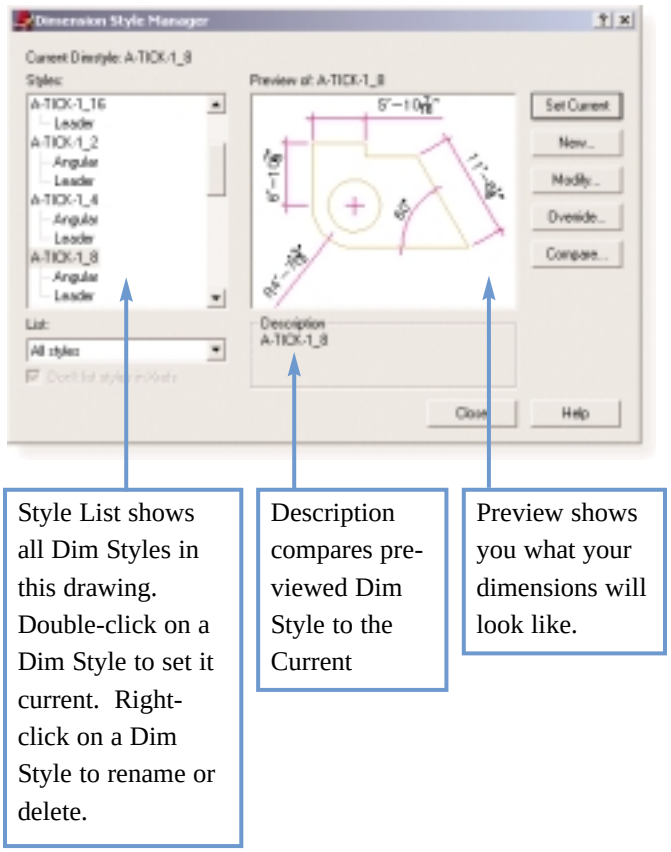
Other dimensioning enhancements include:

- Import dimstyles from other drawings using DesignCenter
- The Dimension toolbar includes a new DimStyle Control pull-down list.
- New Quick Dimension and Quick Leader commands (used to be Express Tool commands) accessible from the Dimension toolbar
- Modify dimensions using the Properties window

Dimstyle Setup

(This section is purposely brief, since you have Dimension Wizard to create Dim Styles for you.)

The Dimension Style Manager window (click Format ➔ Dimension Style from the menu bar), includes a preview that shows you the effects of changes to settings.



Buttons

Set Current: Highlight a Dim Style in the Style list, then click this button to make it current.

New: Creates a new Dim Style. If you highlight an existing Dim Style before clicking new - your new style will be based on that style.

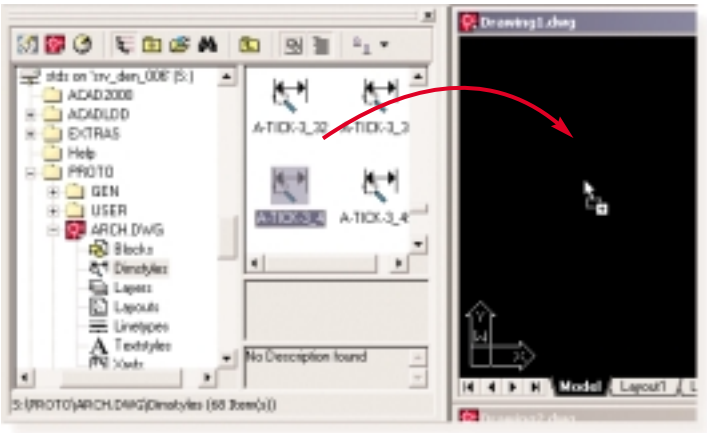
Modify: Modify an existing Dim Style. Remember - changes here are saved to the Dim Style and existing dimension objects using the modified Dim Style

Override: You can only override the current Dim Style - if you highlight a style in the style list that is not current, this button is grayed out. I advise you NOT to use overrides. Instead, use the Modify button to make permanent changes to your Dim Styles. If you don't want to affect existing dimension objects with Modify, create a new Dim Style and Modify it.

Compare: Lets you select two Dim Styles, then lists all dimension variables side-by-side.

Import Dim Styles Using DesignCenter

In Exercises 5 & 6 of Chapter 2, you learned how to use DesignCenter to import layers and blocks into your drawing. You can also import Dim Styles from other drawings into your current drawing.

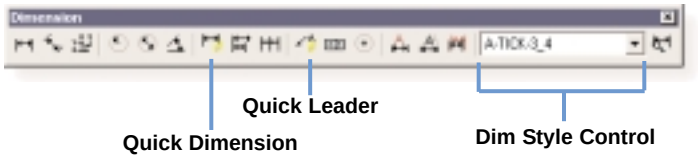


Locate your drawing in the Tree pane of DesignCenter. Click Dimstyles in the Tree pane so that you see Dim Styles listed in the Palette. Now just drag the desired Dim Style from the Palette to your drawing. Remember, this loads the Dim Style, but does not set it current. Click **Format ➔ Dimension Style** to set your current Dim Style once it's loaded.

Dim Toolbar

The Dimension Toolbar is useful, but it's large - try turning it on while you're dimensioning, then back off when you're finished.

To quickly display the Dim toolbar, right-click on any AutoCAD toolbar button, then pick Dimension from the shortcut menu.



You'll see all of the familiar buttons from release 14, and some new buttons as well.

New on the Dim toolbar are Quick Dimension, Quick Leader and Dim Style Control.

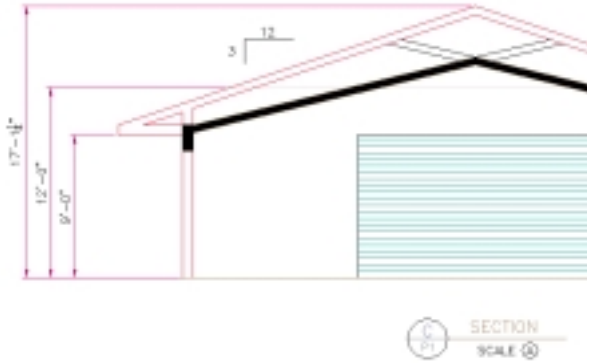
Quick Dimension

AutoCAD's Express Tools used to include a Quick Dimension command - it's now an AutoCAD command and has been removed from the Express Tools menu. You'll find the Quick Dimension button on the AutoCAD Dimension toolbar.

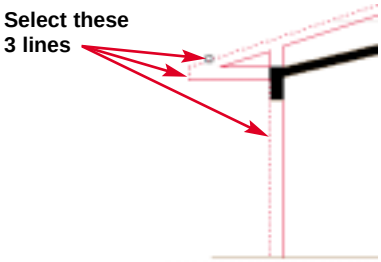
Quick Dimension creates multiple dimensions quickly, you select the objects to dimension, then pick a location for the dimension lines.

Exercise 1 - Using Quick Dimension

In this exercise, we'll use the Qdim command to quickly create the three dimension objects shown below.

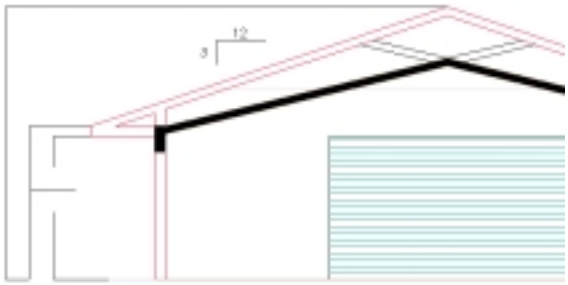


1. Open Chap05Ex01.dwg.
2. If the Dimension toolbar is not visible, right-click on any AutoCAD toolbar button, then pick Dimension from the shortcut menu.
3. Click the Quick Dimension toolbar button.
4. At the **Select geometry to dimension** prompt, pick the three lines shown here, then press **Enter**.



5. At the **Specify dimension line position, or [Continuous/Staggered/Baseline/Ordinate...** prompt, type **B<Enter>**.

   This puts you in the baseline mode (baseline dimensions share a common datum point).

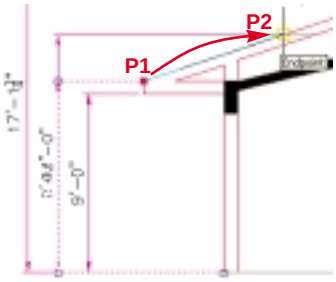


As you drag your mouse to the left, notice the ghosted image showing an outline of what the dimensions will look like.

7. Pick a point on the screen to locate the dimensions.

   We're not done, since the middle dimension is actually not dimensioning the correct upper point. Let's use grips to modify the second dimension...

8. Use grips to stretch point **P1** to **P2** (the endpoint of the horizontal construction line). Use object snap so it's accurate.



   This exercise illustrates how Qdim might not initially locate dimensions exactly where you want them, but it's often easier to get the dimensions on the screen quickly with Qdim, then modify them afterwards.

9. Your drawing should look like the example shown at the beginning of this exercise.

   You can use Qdim to create continuous, staggered, baseline, ordinate, radius and diameter style dimensions - use the appropriate keyword option to change the style, as you did in step 5.

This completes Exercise 1. Close Chap05Ex01.dwg, discard changes.

Quick Leader

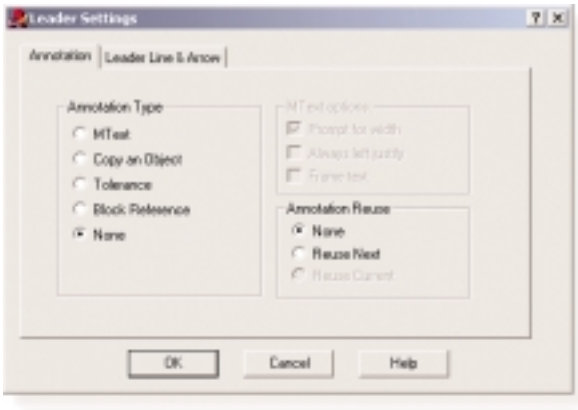
Like Quick Dimension, Quick Leader has been moved from the Express Tools menu to the AutoCAD Dimension toolbar.

By customizing your Qleader settings, you can quickly create leaders in straight-line or spline format, text or no text, arrowhead type and more.

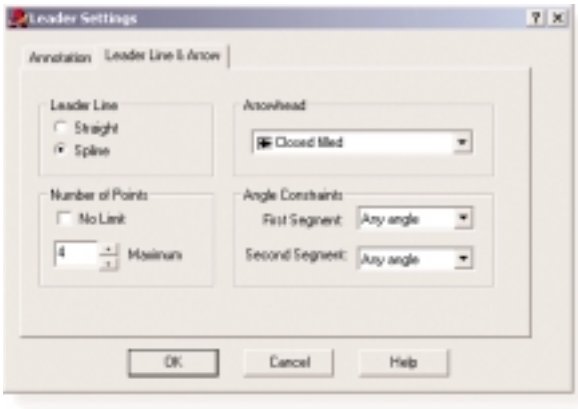
Exercise 2 - Using Quick Leader

In this exercise, you'll use the Qleader command to quickly create straight-line and curved leaders.

1. Open Chap05Ex02.dwg.
2. If the Dimension toolbar is not visible, right-click on any AutoCAD toolbar button, then pick Dimension from the shortcut menu.
3. Click the Quick Leader toolbar button.
4. At the **Specify first leader point,...** prompt, press **Enter** on the keyboard to accept the Settings default.

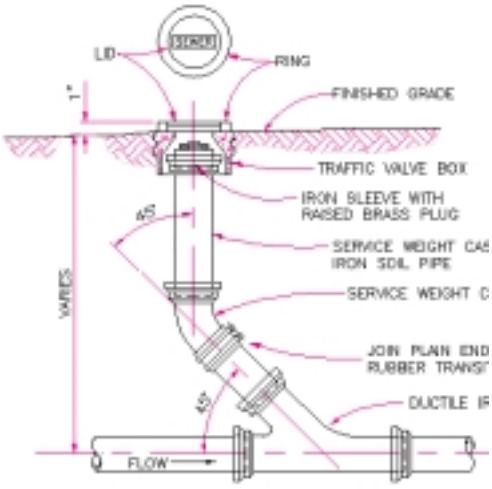


5. In the Leader Setting dialog, Annotation tab, select None for the Annotation type. (This drawing already has text, we're only drawing leaders.)



6. In the Leader Line & Arrow tab, select **Spline** for the Leader Line, in the Number of Points area, uncheck No Limit, enter **4** for Maximum. Arrowhead should be set to **Closed filled** and both angle constraints should be set to **Any angle**.

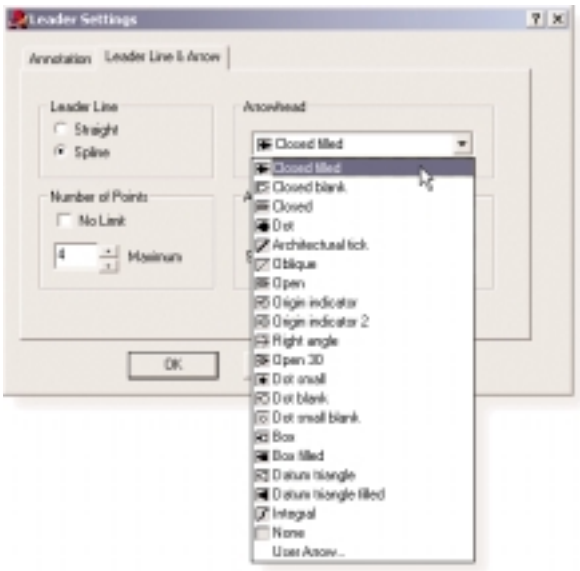
7. Click OK to exit Leader Settings.



8. Continue with the Qleader command - make your drawing look like the example above.

Tips:

- Your first point picked is at the arrowhead end of the leader.
- Your settings specify 4 points for each leader. Once you pick the 4th point, the Qleader command stops and your leader line is created.
- Don't worry about making the leader look perfect the first time. Get the leader placed, then use grips to adjust the vertices later, if necessary.

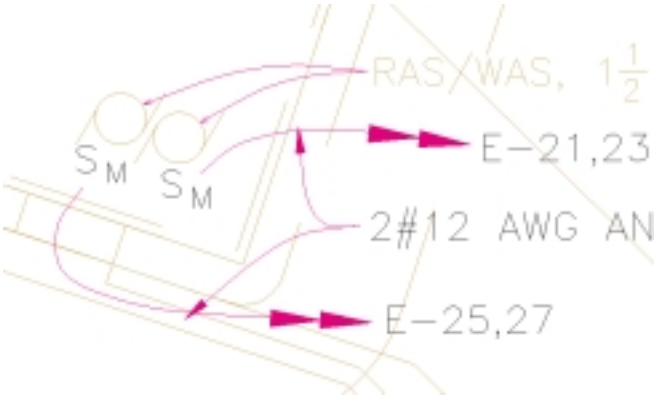


Using Settings in the Qleader command, you can select the arrowhead type. Choices include closed filled, dot, integral and user arrow, which you could use to make a loop-end leader.

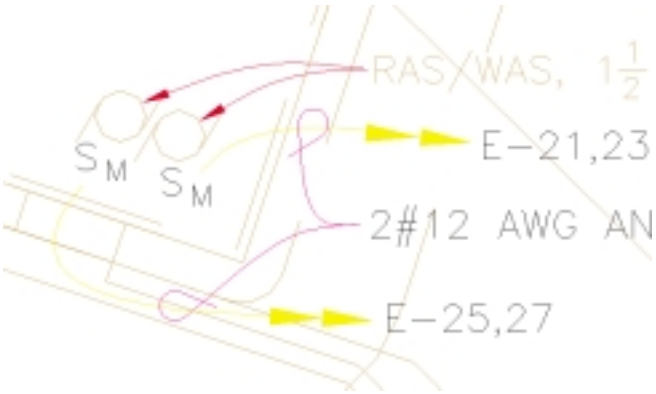
This completes Exercise 2. Close Chap05Ex02.dwg, discard changes.

Modifying Dimension Properties

Using AutoCAD's new Properties Manager and the Dim Style Control on the Dimension toolbar, it's very easy to modify existing dimensions. The examples below show before and after the changes you'll make in the next exercise.



Before: We'll use Properties Manager and the Dim Style Control on the dimension toolbar to modify quickly these leader lines



After modifying the leader lines

Exercise 3 - Using Modifying Dimensions

Remember that leaders are dimension objects, that is they use Dim Styles just like dimensions do. The techniques in this exercise can also be applied to dimension objects.

1. Open Chap05Ex03.dwg.

2. If the Dimension toolbar is not visible, right-click

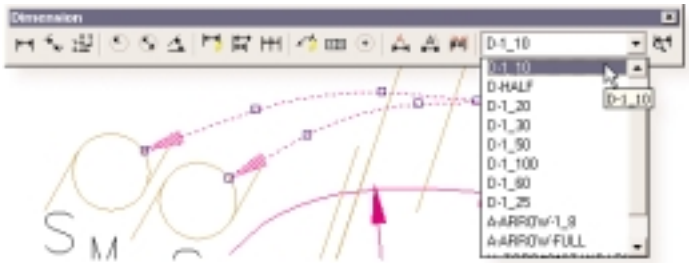
on any AutoCAD toolbar button, then pick Dimension from the shortcut menu.

In your drawing, see the pair of leaders at the "RAS/WAS..." annotation - currently they have the wrong arrowhead type and size, and they're the wrong color.

3. Use the LIST command, select one of the "RAS/WAS..." leaders. Here's what LIST says:

dimension style: "D-1_10$7"
dimension style overrides:
DIMLDRBLK Open30
DIMSCALE 3.00

4. Select both "RAS/WAS..." leaders. (See example below.)

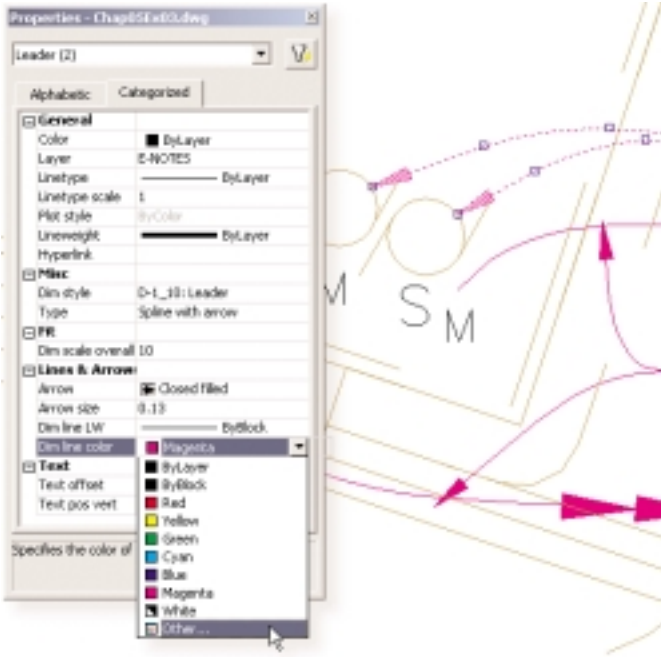


5. On the Dimension toolbar, pull down the Dim Style control and select D-1_10 from the list. (This drawing will be printed at a 1:10 scale, that's why you're picking the D-1_10 style.) You have just set the selected leaders to the D-1_10 style and cleared the leaders of all overrides. Use the LIST command, select one of the leaders, you'll see there are no longer overrides controlling these dimensions.

This illustrates how you can use the Dim Style control on the Dimension toolbar to change the style of a dimension or leader object.

To make the "RAS/WAS..." leaders match the After example, we still need to change their color. We cannot simply change the color value of the leader like we would a line or circle. Leaders and Dimensions behave differently. Continue with step 6 to change their color...

6. Click the Properties button on the Standard toolbar.



7. If they're not still selected, select the 2 leaders. Click **Other** from the **Dim line color** pull-down list in the Properties window.

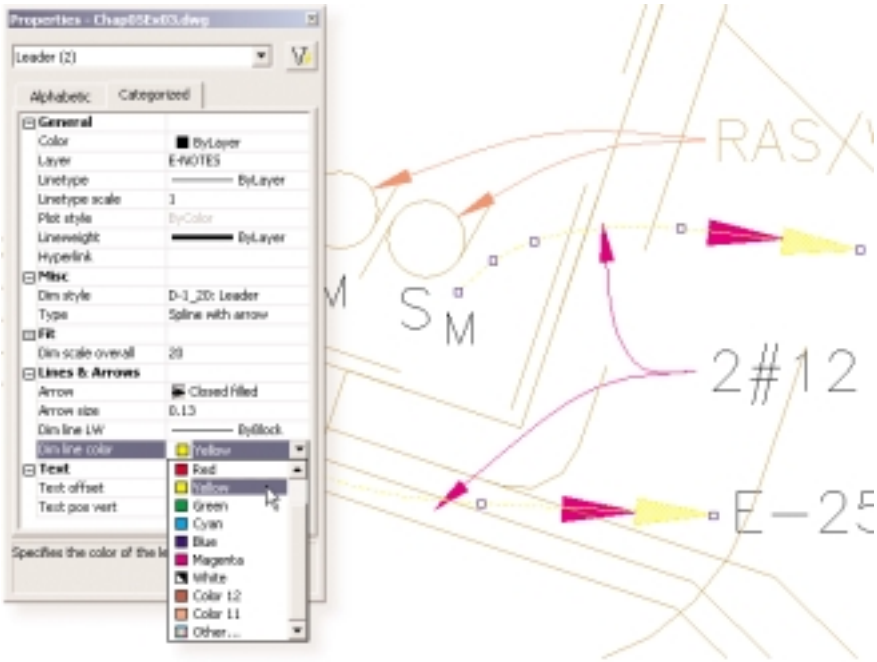
8. In the Select Color dialog, enter 11 for the color value, then click OK.

9. Press Esc on the keyboard twice to de-select the leaders. As you see, their color has been easily changed using the new Properties window.

10. Select both home-run leaders (shown at the right, highlighted), then use the Dim line color pull-down to change their color to yellow.

11. Press Esc on the keyboard twice to de-select the leaders, then select both magenta colored secondary arrow-heads on the home-run leaders.

12. Change the selected arrowheads color to yellow by using the Color pull-down in the Properties window, then press Esc twice to de-select the arrow-heads.

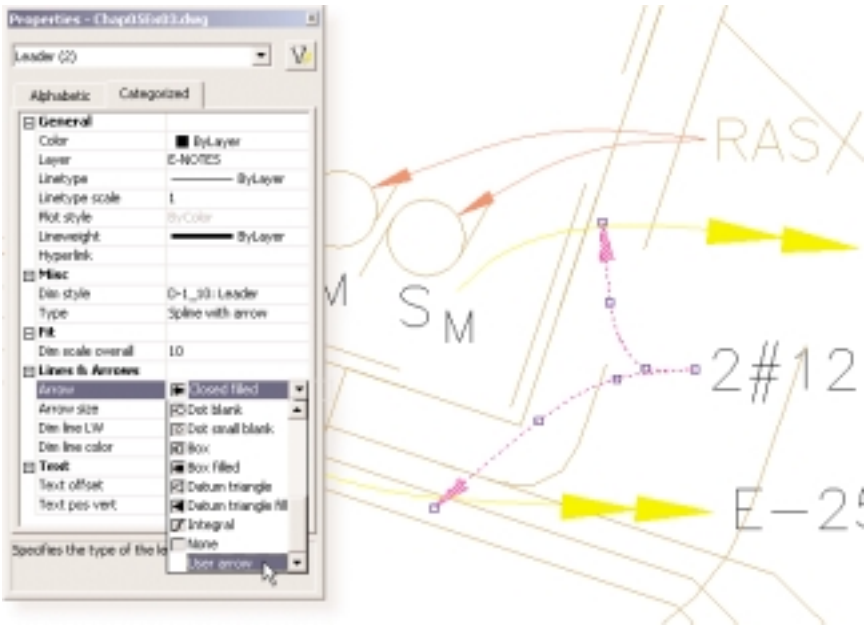


13. Select both "2#12..." leaders (shown at the right, highlighted), then select User arrow from the Arrow pull-down.

14. From the Select Custom Arrow Block dialog, select ZLPARW (loop-end block), then click OK.

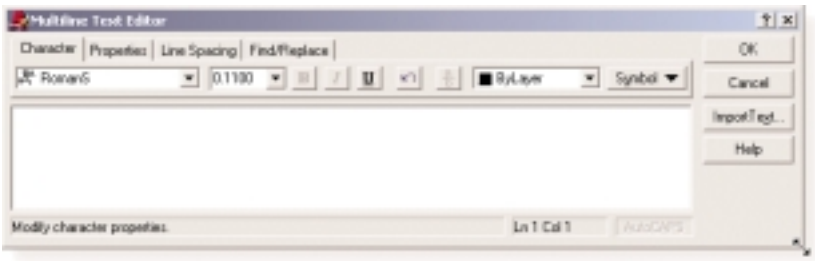
15. Press Esc on the keyboard twice to de-select the leaders.

You drawing should look like the After example shown earlier in this chapter.



This completes Exercise 3. Close Chap05Ex03.dwg, discard changes.

Text



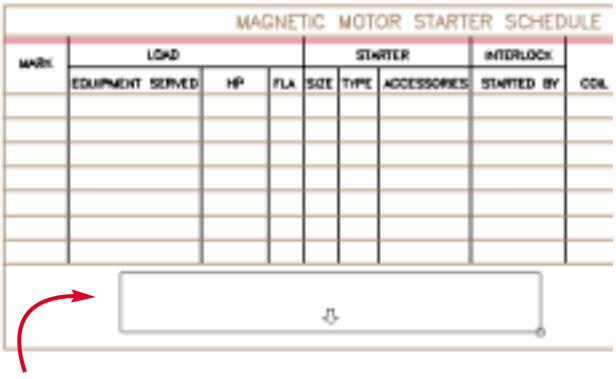
Multiline Text (Mtext) enhancements to AutoCAD 2000 include:

- AutoCAPS option always uses uppercase, regardless of your keyboard caps lock setting
- Resizable Mtext editor dialog
- Fraction features include automatic stacked, architectural-style, alignment control (top, middle, bottom), scale relative to text height
- Case override, and new Combine Paragraph option available from shortcut menu
- Most recently used fonts displayed in pull-down list
- Adjustable Line spacing
- Faster: display of Mtext dialog, object snap detection, text importing, generation of Mtext

Complete the following exercise to learn about the new Mtext features.

Exercise 4 - Mtext - Resize, AutoCAPS and Stacked Fractions

1. Open Chap05Ex04.dwg.
2. Make sure keyboard Caps Lock is OFF.
3. Click Draw ➔ Text ➔ Multiline Text from the menu bar, or click the Multiline Text button from the Draw toolbar.



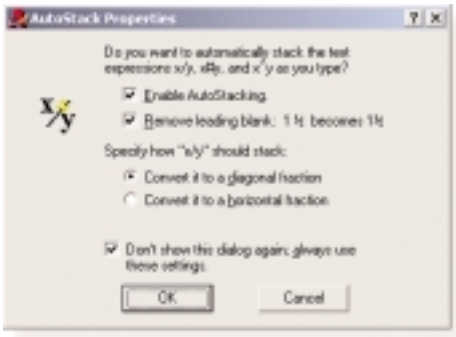
4. Pick two corners in the empty area at the bottom of the schedule (shown above).
5. As you move your mouse pointer over any corner of the Mtext dialog, notice the pointer changes shape - it appears as a double-sided arrow. Drag the corner of the dialog by clicking and holding the left mouse button.

   The ability to resize the Mtext dialog is new to Release 2000

6. In step 2, you turned your keyboard caps lock off. Type some text into the Mtext editor - it should be lowercase.
7. Double-click on AutoCAPS near the lower right corner of the Mtext dialog. With AutoCAPS now on, type more text into the editor. It should be uppercase.

   When you exit the Mtext editor (don't exit now), your keyboard caps lock will be off, just like before you used Mtext. When you return to the Mtext command, since AutoCAPS is on, new text you enter will be uppercase.

8. Enter the following into the Mtext editor: 1 1/2"
9. As you press the " key, the AutoStack Properties dialog is presented.

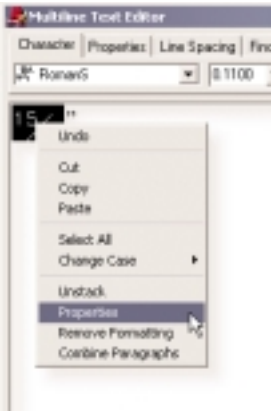


   If you check the "Don't show this dialog again..." option, AutoCAD will stack fractions without asking. Also, you can choose horizontal or diagonal format.

10. Change your settings to match those shown above, then click OK to exit the AutoStack Properties dialog.
11. Enter another fraction into the Mtext dialog, for example 15/16" - it should automatically stack.

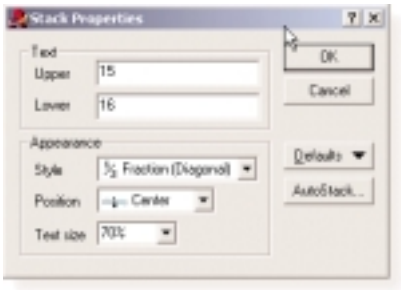
12. Highlight 15/16. (To highlight text, drag your mouse across it while holding down your left mouse button.)

13. Right-click over the highlighted text and select Properties.



14. The Stack Properties allows you to change text, style, position and size.

15. Click OK to exit Stack Properties.



16. Click OK to exit the Mtext editor.

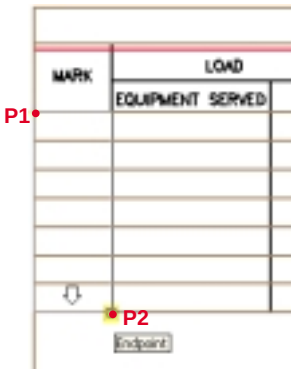
This completes Exercise 4. Leave open Chap05Ex04.dwg for the next exercise.

Exercise 5 - Mtext - A Schedule Using the New Line Spacing Feature

This exercise uses the new Mtext Line Spacing feature to fill in a schedule.

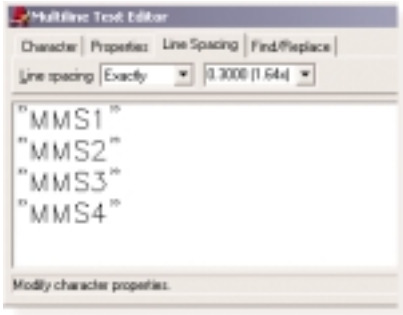
1. Use the drawing from the last exercise, Chap05Ex04.dwg, start the Mtext command by clicking Draw ➔ Text ➔ Multiline Text from the menu bar, or click the Multiline Text button A from the Draw toolbar.

2. Pick two corners in the MARK column, indicated here by P1 and P2. Use endpoint or intersection osnaps to make the picked points accurate.



3. In the Line Spacing tab of the Mtext editor, change the Line spacing settings to **Exactly 0.300**.

(We know each line in the schedule is 0.300 inches apart.)

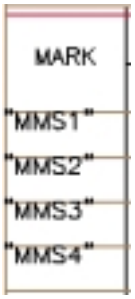


4. Enter the following text:

"MMS1"
"MMS2"
"MMS3"
"MMS4"

5. Click OK to exit the Mtext editor.

6. Since the text is top/left justified, it's too high and up against the left side of the schedule. Move the text down and to the right to better center it in the schedule lines.



7. Copy the text you just created to the next column (Equipment Served). Use object snaps so it's accurately placed.

8. Right-click on the new text (the copy), then select **Mtext Edit** from the shortcut menu.

9. Change the text to match the Equipment Served column in the completed schedule shown below.

MAGNETIC MOTOR STARTER SCHE

MARK	LOAD			STARTER			INTERLOCK
	EQUIPMENT SERVED	HP	FLA	SIZE	TYPE	ACCESSORIES	STARTED B
"MMS1"	"CH#P1"	1½	6.0	0	FVNR	HOA,RP,AC	ATC
"MMS2"	"CH#P2"	1½	6.0	0	FVNR	HOA,RP,AC	ATC
"MMS3"	"HWP1"	1	4.2	0	FVNR	HOA,RP,AC	ATC
"MMS4"	"HWP2"	1	4.2	0	FVNR	HOA,RP,AC	ATC

10. If time permits, complete the schedule as shown.

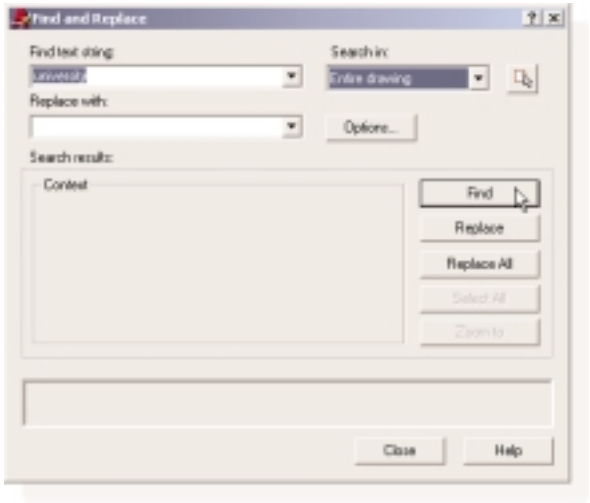
This completes Exercise 5. Close Chap05Ex04.dwg, discard changes.

Find & Replace Text

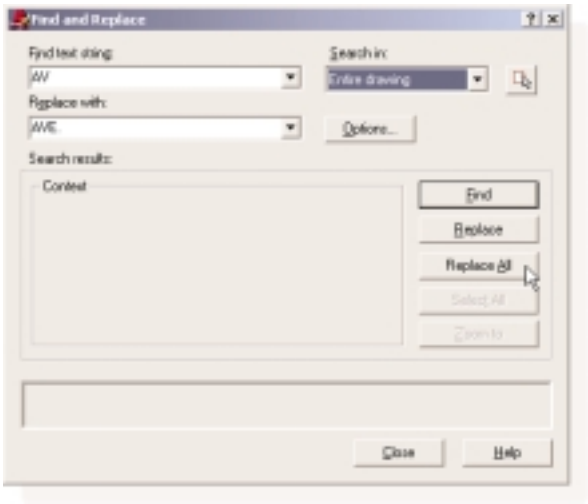
AutoCAD 2000 introduces the Find feature, which provides you with a quick and easy method for finding and replacing text within a drawing file. You can use the Find tool to locate single line text, multiline text, text in block attributes, dimension annotations and hyperlink/URL names. The Zoom option enables you to locate and zoom to the text in the drawing file. Options include match case and find whole words only, as well as search the entire drawing or a selection set.

Exercise 6 - Find & Replace

1. Open Chap05Ex06.dwg.
2. Right-click over the AutoCAD graphic screen (where you draw objects), then choose **Find** from the shortcut menu.



3. In the **Find text string** box, type **university**, then click the **Find** button.
4. In the Context window, see the word University. If it were part of a longer sentence, that entire sentence would show up here.
5. Click Zoom to, which causes AutoCAD to zoom to the text it found. You may have to move the Find dialog around the screen to uncover the text.
6. Click Fint Next, then Zoom to. AutoCAD zooms to a second occurence of text with university.



7. In the **Find text string** box, type **AV**
8. In the **Replace with**, type **AVE**
9. Notice **Search in** is set to **Entire drawing**. If you select objects before starting Find, you can change the **Search in** option to **Current selection**, or simply press the Select objects button . Leave **Search in** set to **Entire drawing** for this exercise.
10. Click the **Replace All** button.

    A result message at the bottom of the Find dialog should tell you that 26 replacements were made.

11. Click Close to exit Find.

This completes Exercise 6. Close Chap05Ex06.dwg, discard changes.

Quick Select

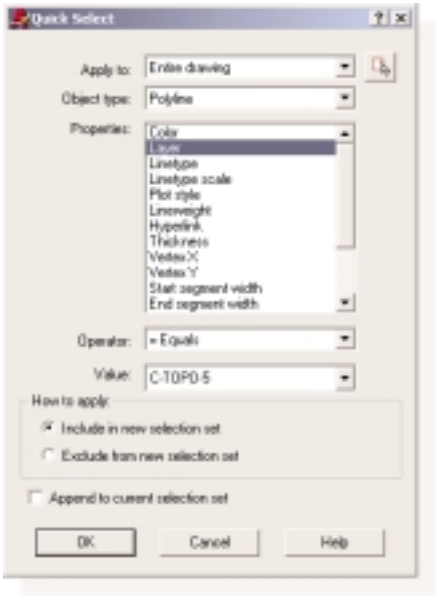
If you've ever used the AutoCAD Filter command to find and select objects, you should find the new Quick Select easier and faster to use, though it can be a bit confusing.

Using Quick Select, you can find and select objects based on type and/or property (color, layer, etc.). You can combine searches, for instance select all lines on layer Topo, and all Polylines on layer Topo, by completing one search, then using the Append to current selection option.

Exercise 7 - Quick Select

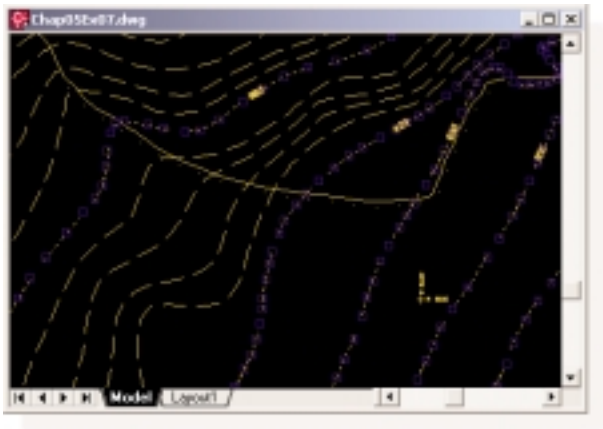
In this exercise, you'll use Quick Select to find and select all Polyline objects on the C-TOPO-5 layer, so that we can change linetype scale of those objects.

1. Open Chap05Ex07.dwg.
2. Right-click over the AutoCAD graphic screen (where you draw objects), then choose **Quick Select** from the shortcut menu.



3. Use the following settings in the Quick Select dialog:

   Apply to: Entire drawing
   Object type: Polyline
   Properties: Layer
   Operator: = Equals
   Value: C-TOPO-5
   How to apply: Include in new selection set
   Append to current selection set: Unchecked
4. Click **OK**.



5. Your drawing should look like the example above.
6. Use the Properties window to change the linetype scale of the select polylines to a value of 1.2. (Remember Chapter 4, Exercise 2?!)

This completes Exercise 7. Close Chap05Ex07.dwg, discard changes.

In Place Block/Xref Editing

AutoCAD's new Refedit command allows you to edit reference geometry and the current drawing at the same time. Use it to select block or external reference objects and edit them from within the context of the parent drawing, i.e., in place. You also have the capability to copy objects in the reference drawing into the parent drawing as well as copy objects from the parent drawing into the external reference.

An additional benefit of this feature is the ability to edit the definition of a block without exploding it. With Refedit, you can select a block within a drawing, edit its definition, and save it. Once you have saved your changes, all instances of that block in the drawing file will automatically be updates with your change.

Exercise 8 - Refedit

In this exercise, you'll use Refedit to solve a common problem. You'll open a sub-sheet, E2, which uses an external reference, X-Topo. In E2, the topo lines need to be displayed in color #42, which is a 50% screened 0 pen. The problem is, the topo lines drawn in X-Topo where created with a color of White, instead of Bylayer. Therefore, in E2, when we assign color #42 to all X-Topo layers, those lines stay white. Rather than open X-Topo, we'll use Refedit to modify the topo lines, changing their color to Bylayer.

1. Open Chap05Ex08.dwg.

   As you can see, the topo lines are white in color. If you check the layer settings, all X-Topo layers are color 42.

2. Type GLR<Enter>. Select one of the white topo lines. The GLR command give us the following information:

   Select object:
   Lwpolyline: color=White, linetype=Bylayer
   Layer: "X_topo|s-topo-1"

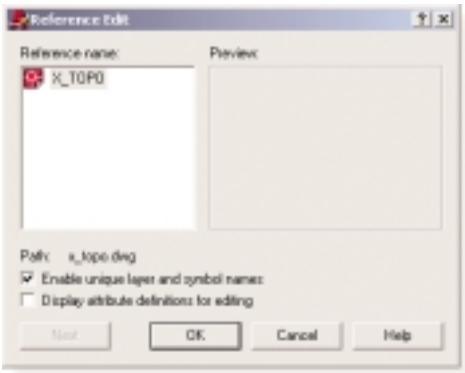
   Notice the color is White, not Bylayer like it should be.

3. Turn on the Refedit toolbar: right-click over any AutoCAD toolbar button, then select Refedit from the shortcut menu.

4. Click the **Edit block or xref** button on the Refedit toolbar.

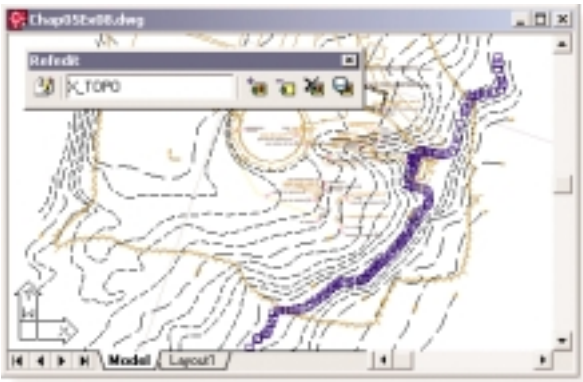
5. At the **Select reference:** prompt, pick on one of the topo lines to select the X-Topo external reference.

6. In the Reference Edit dialog, click **OK** to accept the defaults.



Having checked the **Enable unique layer and symbol names** simply means that the X-Topo layers will begin with a $#$ while you're editing this xref. For example, the X-Topo layer Topo-1 will appear as 0S-TOPO-1, until we're finished editing it. If there were a TOPO-1 layer in this drawing (Chap05Ex08.dwg), it would remain separate from the 0S-TOPO-1 layer of the xref.

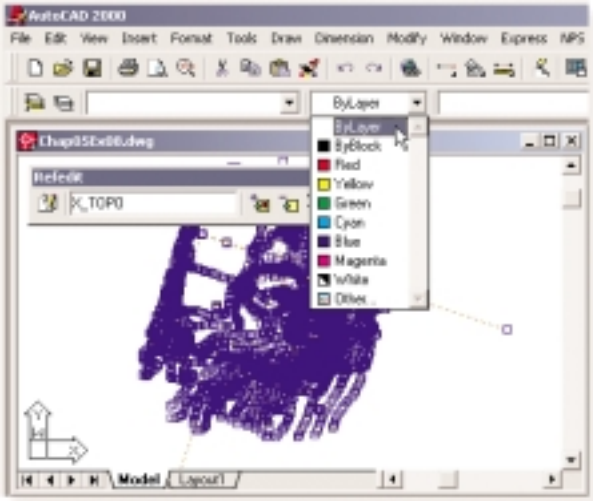
7. At the **Select nested objects:** prompt, select all objects using a window. As an alternative, you could select only those objects you need to edit, but for this exercise, select all objects.



As shown here, using Refedit, you can directly edit the objects of an Xref without opening the Xref in a separate drawing window.

Click on one of the objects of the xref. Notice you can select object one at a time.

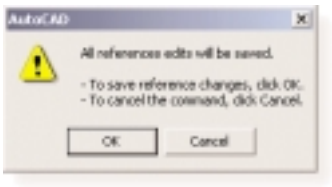
8. Select all objects again using a window.



9. Use the Color Control pull-down (show above) to change the color of the selected object to Bylayer - or use the Properties window if you're more comfortable with that method.

10. Click the **Save back changes to reference** button on the Refedit toolbar.

11. Click OK to save the reference changes.



The topo lines should now appear in color 42.

This completes Exercise 8. Close Chap05Ex08.dwg, discard changes.

Bonus Exercise 9 - Use Refedit to modify a block

You can also use Refedit to modify a block definition. For instance, if you had a drawing with a block depicting a tree in plan view, and this drawing had 100 instances of this tree block inserted, you could use Refedit to modify the tree block. You might want to change the tree block, so that instead of using many lines, it could be drawn using two or three simple circles. When you save the changes, all 100 instances of the tree block would be updated.

1. Open Chap05Ex09.dwg.

2. Use the Refedit command to modify the tree shown here. Make the new tree look like anything you want - maybe a simple circle or two.

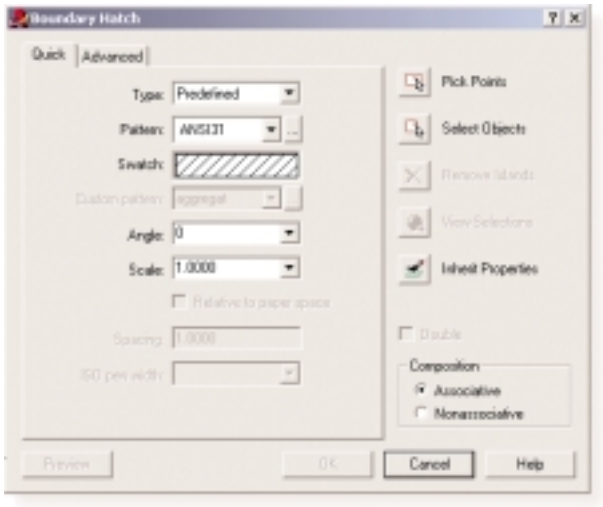
Use the same procedures as in steps 3 - 11 of exercise 8.

3. When you save your changes, all instances of that tree should have changed.

This completes Exercise 9. Close Chap05Ex09.dwg, discard changes.

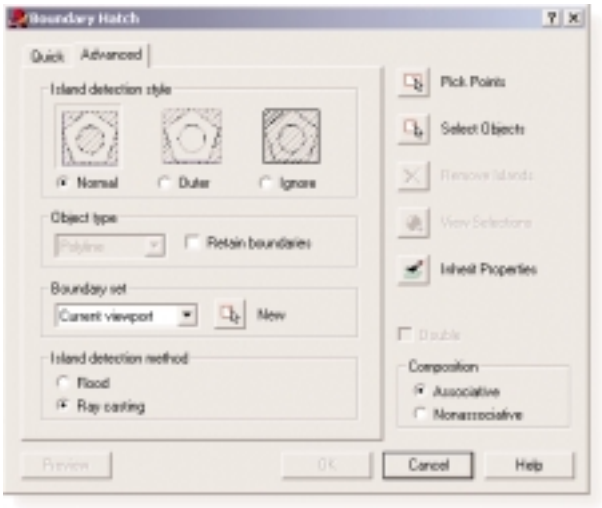
Hatch

The Boundary Hatch dialog in AutoCAD 2000 has undergone a redesign to enhance productivity. It now presents the hatching parameters in two tabs: Quick and Advanced. A hatch preview is available from a right-click shortcut menu enabling you to check settings while picking hatch points or objects. In addition, you can now preview custom hatch patterns using the Swatch feature.



Quick tab of the redesigned Boundary Hatch dialog

Double-click on the Swatch image to view a hatch pattern palette, which allows you to visually select predefined and custom patterns.

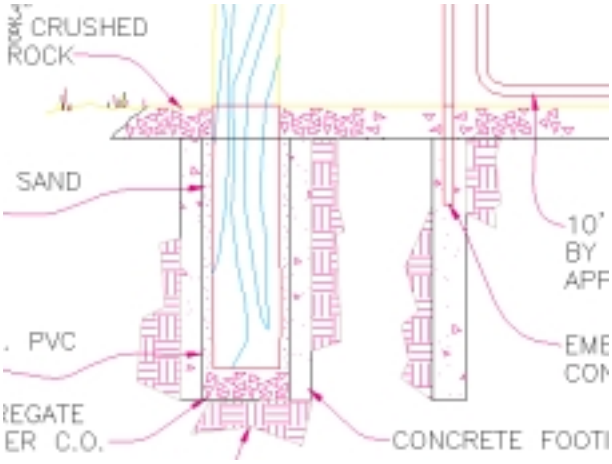


Advanced tab of the redesigned Boundary Hatch dialog

In the Advanced tab, you can select island detection style, object boundary settings, boundary set and island detection method (flood works well for most hatching, and includes islands as boundary objects).

Exercise 10 - Boundary Hatch

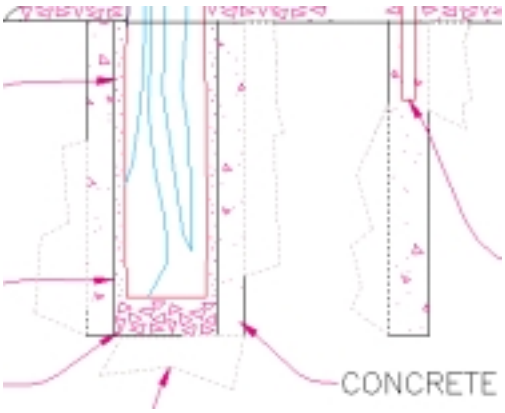
The goal of this exercise is to use boundary hatch to complete the earth hatching as shown below:



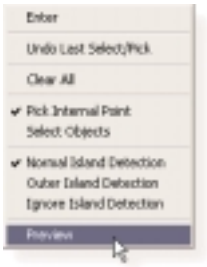
1. Open Chap05Ex10.dwg.
2. Click Draw ➔ Hatch from the menu bar.
3. In the quick tab of the Boundary Hatch dialog, click on the swatch image.

4. Select the Earth pattern in the Other Predefined tab, then click OK to exit the palette.

5. Enter 16 for the scale.
6. Click Pick Points in the Boundary Hatch dialog.



7. Pick a point in each of the 5 highlighted areas shown above.
8. Right-click, then select Preview. Notice this shortcut menu also includes an undo option and island detection modes.
9. After confirming the preview, right-click to return to the Hatch dialog, then click OK to create the hatch.



This completes Exercise 10. Close Chap05Ex10.dwg, discard changes.

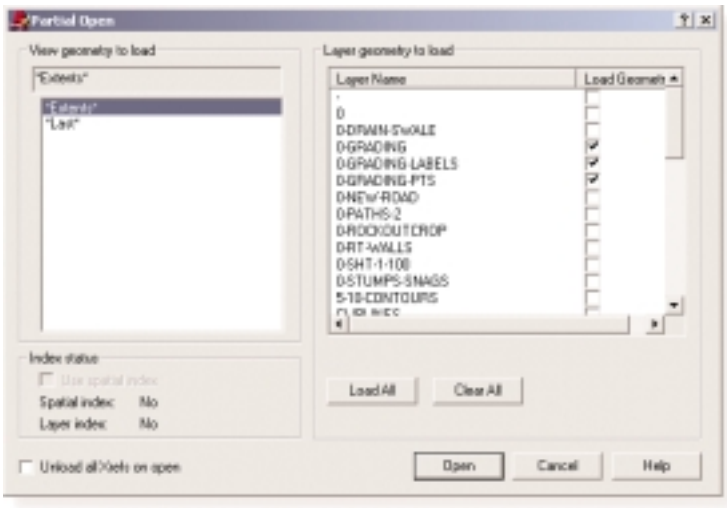
Partial Open & Load

Using AutoCAD 2000's new Partial Open feature, you can open a portion of a drawing and external reference file, based on saved views, or based on layers. Since only the needed portion of a drawing is loaded, memory requirements and opening times are reduced. Once you have partially loaded a drawing, additional portions of the file can be loaded with the Partial Load feature.

Note: You cannot use Partial Open with a release 14 or earlier drawing file.

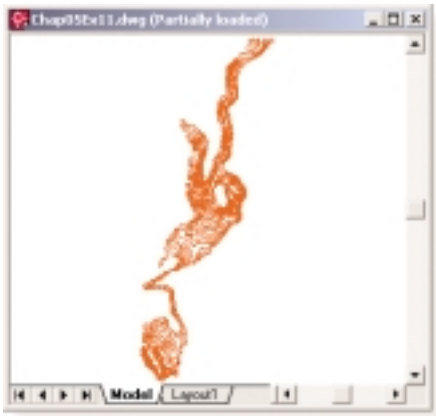
Exercise 11 - Partial Open & Load

1. Click File ➔ Open, or click the Open button on the Standard toolbar.
2. In the Select File dialog, highlight Chap05Ex11.dwg, but <u>don't</u> double-click or press the Open button.
3. Click Partial Open.



In the View geometry to load area, you can select Extents (all objects not on frozen layers), Last (the last display of the drawing), or user created views. For this exercise, you'll use select geometry to load based on layer.

5. Imagine you need to view only the contour geometry in this drawing - click the check box beside the 5-10-CONTOURS layer, then click Open.
6. A zoom extents may be necessary, but the partially loaded drawing should contain only objects on layer 5-10-COUNTOURS. This 16 megabyte file would take much longer to open it its entirety.
7. To partially load additional geometry, click File ➔ Partial Load.
8. Check the Flora layer, then click OK.
9. Zoom in if necessary to verify that objects from both layers are now available.



This completes Exercise 11. Close Chap05Ex11.dwg, discard changes.

If you print a partially opened drawing using AutoCAD's print command, only the loaded geometry will print. If you print a partially opened drawing using CADNET PrfGenerator, ALL objects will print as if the drawing was fully opened.

More New Stuff

Whohas

Have you ever tried to open a drawing file from the network, only to receive a message from AutoCAD stating that the drawing was currently in use by another user?

Using the new WHOHAS command, you can sometimes tell who has a file open (if that user is working in AutoCAD 2000, and has WHOHAS installed).

Exercise 12 - Whohas

1. Open ElectricalPrototype.dwg.
2. Type WHOHAS<Enter> at the AutoCAD command prompt.
3. In the Select drawing to query dialog, select ElectricalPrototype.dwg, then click OK.
4. Something similar to the following should be displayed:

   Command: whohas
   Owner: Bruce Littlehorn
   Computer's Name : NP2000010313
   Time Acessed : Thursday, February 24, 2000 9:25:44 PM

   Beware - I have found that Whohas doesn't always work - it sometimes reports that no owner information is available when there should be.

This completes Exercise 12. Close ElectricalPrototype.dwg.

Revert

Using AutoCAD 2000's new Revert command, you can open a drawing file, make modifications, then change your mind and use the Revert command to re-open the drawing file as it was before your changes. To use Revert, type REVERT<Enter> at the command prompt. AutoCAD will ask you to confirm that you wish to abandon changes. Remember, once you make changes to a drawing file and **save those changes**, you cannot use Revert to undo those changes.

This completes Chapter 5.