

47

BATCH PLOT UTILITY AND EXPRESS TOOLS

Chapter Objectives

After completing this chapter you should:

1. be able to use the Batch Plot Utility to select and plot several drawings unattended;
2. be able to specify which layers and what drawing areas to plot with the batch utility;
3. be able to install the Express Tools from the AutoCAD 2004 CD-ROM, load the Express menus, and invoke the Express toolbars;
4. be able to use the Express Text commands;
5. be able to use the Express Layer commands;
6. be able to use the Express Standard toolbar commands;
7. be able to use the Express Block commands;
8. be able to use miscellaneous Express commands from the pull-down menus and the command line.

CONCEPTS

AutoCAD 2004 ships with many useful bonus (Express) tools and a batch plotting feature. Either or both of these utilities can be installed during the initial AutoCAD installation by ensuring these features are selected. Alternately, you can install these features at a later time by inserting the CD, selecting *Install*, then selecting *Add or Remove Features* from the setup utility.

The Batch Plot Utility is very useful in an office or laboratory where many drawings must be printed or plotted. This utility allows you to specify multiple drawings to be printed or plotted unattended. For example, a set of drawings can be set up to be printed at night so they are ready for you when you return in the morning.

The Express Tools include several utility commands that have a specialized or streamlined usefulness. Many very powerful and useful features are available, including layer utilities and a Layer Manager, file utilities, modify commands, draw commands, and commands for blocks, text, layouts, dimensioning, hyperlinks, and miscellaneous tools.

BATCH PLOT UTILITY

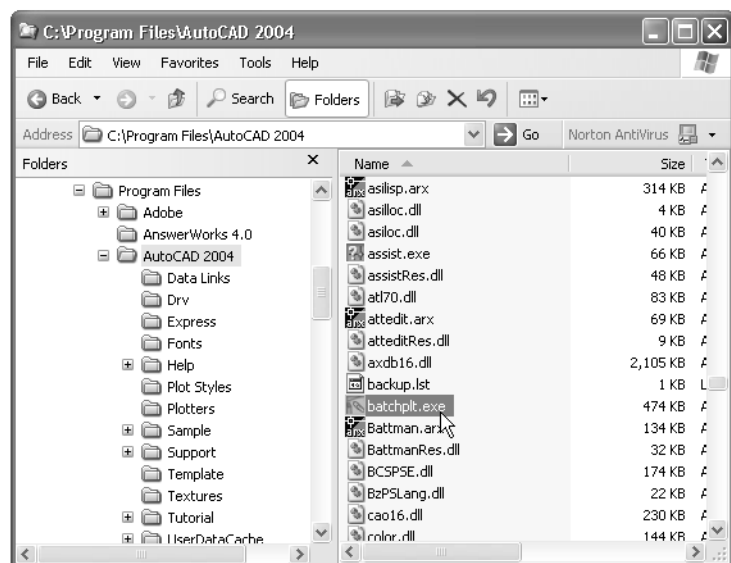
AutoCAD's Batch Plot Utility is useful in offices or laboratories where many prints or plots are made because a set of drawings can be assigned for plotting as a "batch" without human intervention. For other applications, such as when a number of plots are required as a complete project set, the entire set of drawings can be saved to a list to simplify plotting the same set again at a later time. Other specifications about the drawings to be plotted can be saved, such as which layers should be included in the plot, drawing areas to plot, and the print/plot devices to use. A plot test can be used as a "dry run" to locate any potential problems before running the batch. If errors occur during an unattended plot run, an error log is kept so problems can be tracked.

The Batch Plot Utility requires AutoCAD for operation. Invoking the Batch Plot Utility in turn starts its own dedicated AutoCAD session used just for plotting.

Follow these steps to start the Batch Plot Utility:

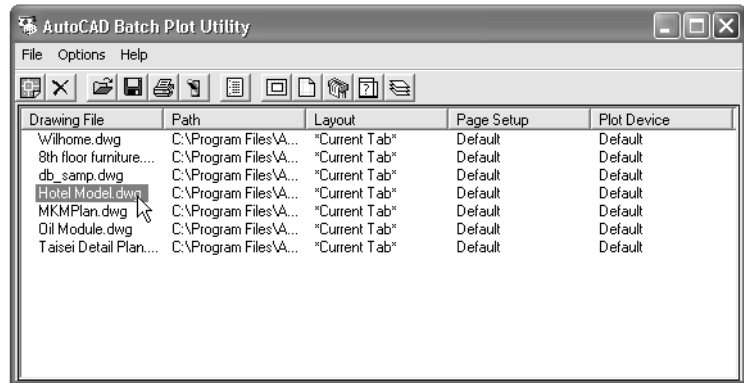
1. Start **My Computer** or **Windows Explorer** (Fig. 47-1).
2. Locate the **C:\Program Files\AutoCAD 2004** folder and highlight it.
3. Locate the **batchplt.exe** file and highlight it.
4. To launch the Batch Plot Utility, double-click on the file or right-click and select **Open** from the shortcut menu.

Figure 47-1



Once the Batch Plot Utility is started (with its own AutoCAD session), the *AutoCAD Batch Plot Utility* dialog box appears (Fig. 47-2). Two pull-down menus in this dialog box (*File* and *Options*) allow you to specify which drawings you want to plot and with what parameters. The choices in each of the pull-down menus are explained next.

Figure 47-2



File Menu

With this menu you manage the list of drawings (or "batch" of drawings) you want to plot. Drawings selected for plotting appear in the list displayed in the central area of the *AutoCAD Batch Plot Utility* dialog box (see Figure 47-2).

Add Drawing...



Use this option to select drawings to include in the list. The *Add Drawing File* dialog box appears (not shown) for you to locate and select the desired drawing(s).

Remove



Use this option to remove selected drawings from the list. First highlight the desired drawing name(s) in the list, and then select *Remove*.

New List...

Use this menu item to clear the current list. This action is necessary in preparation for creating a new list or selecting a saved list for plotting.

Open List...



Use this button to select a list of drawings (previously saved) to include in the list of drawings to plot.

Save List...



When a group of drawings has been selected for plotting, you can save the list for future use with this option. The file is saved as an ASCII file with a .BP3 file extension. The file also records the options you select for plotting each drawing.

Append List...

This option appends (adds) a previously saved list (.BP3 file contents) to the current list of drawings to plot. Using this method, several "sets" of drawings can be plotted in batch.

Plot



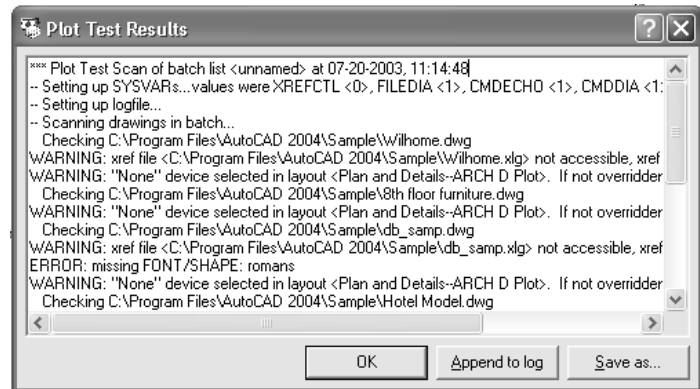
When you are ready to make the plots, select the *Plot* button. As AutoCAD progresses through the plot batch, the current status for each drawing is indicated by a check mark placed just before the drawing name (if the drawing is plotted correctly) or an "X" mark (if the drawing is not plotted). Before plotting, select the options you want for each drawing using the options buttons or *Options* pull-down menu.

Plot Test



Selecting the *Plot Test* option causes AutoCAD to make a “dry run” of all the drawings listed for batch plotting. In this test, each drawing is checked to ensure all component parts are located (Xrefs, linetypes, fonts, etc.) and to ensure that the selected settings are operable. Warning messages appear if a problem (or potential problem) exists. For example, if an attached Xref, raster file, font, etc., is missing, AutoCAD reports it. A sample *Plot Test Results* dialog box is displayed in Figure 47-3.

Figure 47-3



AutoCAD also creates a log file for each drawing in the batch for the plot test. These log files are automatically assigned a name beginning with the drawing name, followed by a set of numbers and a .LOG file extension (such as 8TH FLOOR_1_1_5436.LOG). Since each drawing is opened in AutoCAD for batch plotting and for plot testing, the drawing log file gives the time the drawing was opened and other activity that occurs such as resolving Xrefs. The sample log is shown below.

```
[ AutoCAD - Sun Jul 20 11:20:37 2003 ] _____
Opening an AutoCAD 2004 format file.
Resolve Xref "8th floor plan": 8th floor plan.dwg
Resolve Xref "8th floor furniture": 8th floor furniture.dwg
Resolve Xref "8th floor hvac": 8th floor hvac.dwg
Resolve Xref "8th floor lighting": 8th floor lighting.dwg
Regenerating layout.
Regenerating model.
[ AutoCAD - Sun Jul 20 11:20:39 2003 ] _____
AutoCAD Express Tools Copyright © 2002-2003 Autodesk, Inc. AutoCAD menu utilities loaded.
Command:
```

Logging...

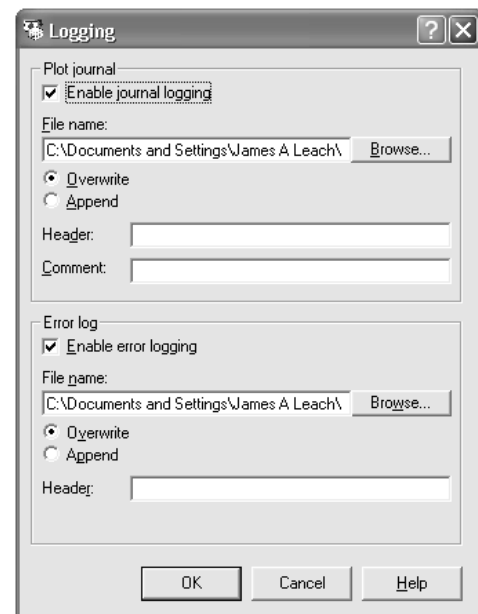


To locate problems that may have occurred during an unattended batch plot, journal log file and error log files could be checked. For example, if you set up a batch of drawings to plot at night and an error occurred while you were out, the log files would record if the drawings plotted correctly or not. Two types of logs can be created while plotting: plot journals and error logs. First, you must select *Enable journal logging* and *Enable error logging* in the *Logging* dialog box (Fig. 47-4) before running the batch to plot.

Enable journal logging

The plot journal is saved in the My Documents folder under the name BPJOURNAL.LOG. This log keeps a list of all drawings that plot correctly during the batch plot session. The list gives information about each plot such as the drawing location and time for each plot, etc. An example plot journal is shown on the next page.

Figure 47-4



```

*** Plot Test Scan of batch list <unnamed> at 07-20-2003, 11:23:24
— Setting up SYSVARs...values were XREFCTL <0>, FILEDIA <1>, CMDECHO <1>, CMDDIA <1>,
SHPNAME <>
— Setting up logfile...
— Scanning drawings in batch...
  Checking C:\Program Files\AutoCAD 2004\Sample\8th floor furniture.dwg
— Restoring logfile...<C:\documents and settings\james a leach\local settings\application
data\autodesk\autocad 2004\r16.0\enu\>
— Restoring SYSVARs...XREFCTL, FILEDIA, CMDECHO, CMDDIA, SHPNAME

```

Enable error logging

When *Enable error logging* is checked in the *Logging* dialog box, a log called BPERERROR.LOG is created listing the drawings that did not plot during the batch plot session. An example plot journal is shown below.

```

ERROR: Could not open drawing: C:\Program Files\AutoCAD 2004\Sample\8th floor.dwg
ERROR: Failed to plot drawing: C:\Program Files\AutoCAD 2004\Sample\8th floor.dwg
ERROR: Could not open drawing: C:\Program Files\AutoCAD 2004\Sample\db_samp.dwg
ERROR: Failed to plot drawing: C:\Program Files\AutoCAD 2004\Sample\db_samp.dwg

```

Options Menu

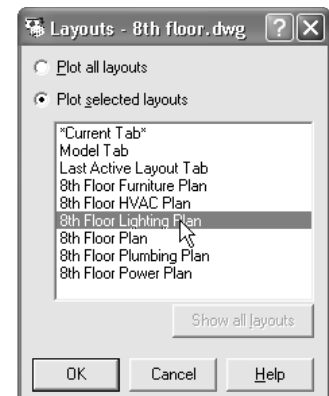
Your selections from this group of options determines how each drawing is plotted during the batch plot session. To specify options for a particular drawing, first select the drawing from the list in the *AutoCAD Batch Plot* dialog box (see Figure 47-2), then select the desired option(s) described below. Remember, an entire set of options can be set for each drawing in the batch.

Layouts



The *Layouts* option produces the *Layouts* dialog box (Fig. 47-5). Here you select the layouts contained in the drawing that you want to plot. The *Show all layouts* button produces a list of the named layouts contained in the drawing. Multiple layouts can be selected by holding down the Shift key (to specify a range) or Ctrl key (to select individual names) when you pick the layout names.

Figure 47-5

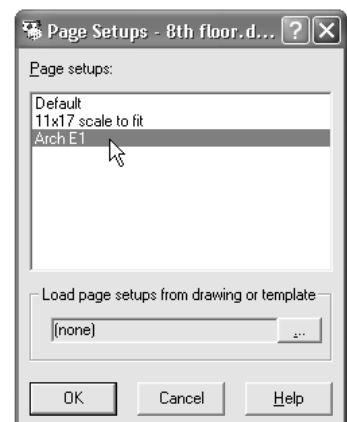


Page Setups



This option produces the *Page Setups* dialog box shown in Figure 47-6 where you specify page setups to use during the batch plot session. You can select from the page setups contained in the drawing or load other page setups from other drawings by selecting the browse button in the lower-right corner.

Figure 47-6

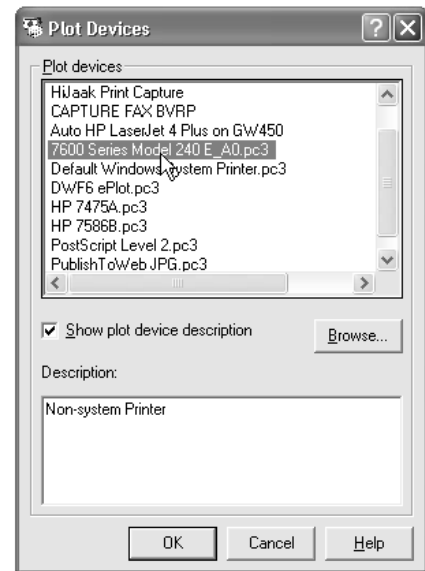


Plot Devices



This feature is especially powerful because you can specify that different drawings in the batch plot session be plotted with different devices. First, highlight the desired drawing from the *AutoCAD Batch Plot Utility* dialog box list (see Figure 47-2), then use the *Plot Devices* option to produce the *Plot Devices* dialog box (Fig. 47-7). All AutoCAD-configured devices for your system are available to choose from.

Figure 47-7

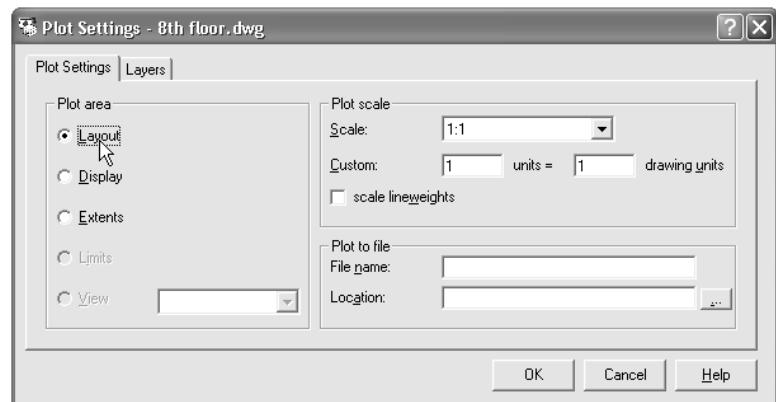


Plot Settings



The *Plot Settings* tab (Fig. 47-8) of the *Plot Settings* dialog box allows you to specify what area of the selected drawing you want to appear in the plot. You can also specify the plot scale for the selected drawing. Selecting *Plot to file* does not create a paper plot but produces a .PLT file.

Figure 47-8

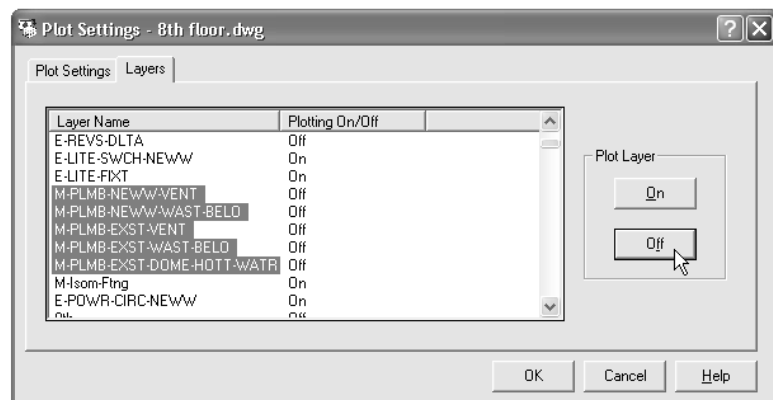


Layers



The *Layers* option also produces the *Plot Settings* dialog box, but opens the *Layers* tab. (This tab is not available when several drawings are highlighted from the list in the *AutoCAD Batch Plot Utility* dialog box.) This tab allows you to indicate which layers from the selected drawing should be included in the plot (Fig. 47-9). The selected drawing's layer settings in the *Layer Properties Manager Plot* column are read and displayed as *On* or *Off* in the *Plotting On/Off* column in this dialog box.

Figure 47-9

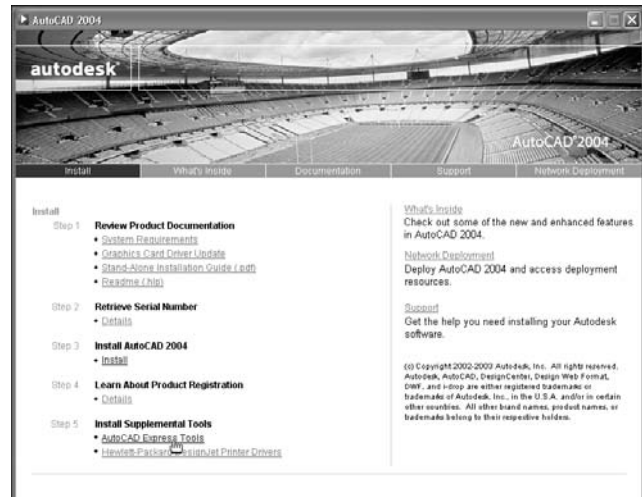


EXPRESS TOOLS

Installing the Express Tools from the CD-ROM

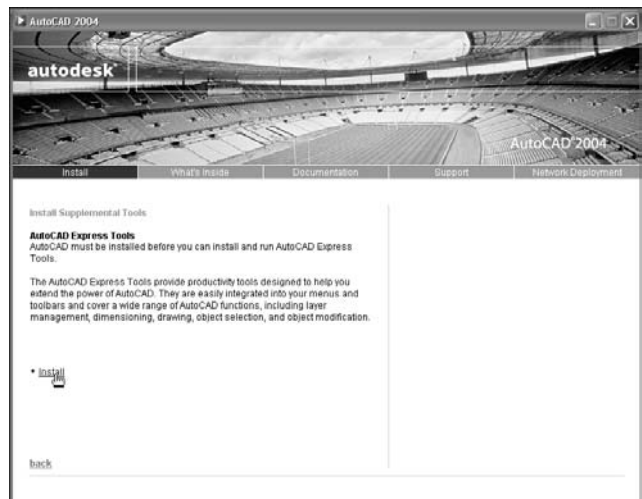
1. Insert the AutoCAD 2004 CD-ROM.
2. The Windows Setup utility should start automatically. If it does not start automatically, use Windows Explorer to find the Setup.exe file on the CD-ROM and double-click on **Setup.exe**.
3. From the *AutoCAD 2004* installation wizard, look for Step 5, Install Supplemental Tools, and select *AutoCAD Express Tools* (Fig. 47-10, near the bottom left).

Figure 47-10



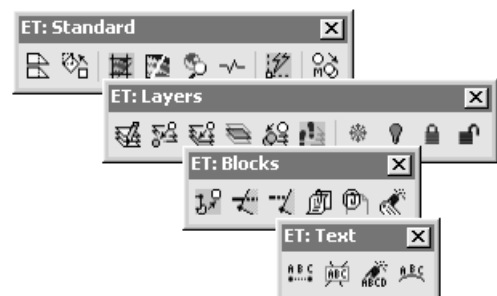
4. In the next screen, select *Install* to launch the *AutoCAD Express Tools Volumes 1-9 Installation Wizard* (Fig. 47-11).
5. Select *Next* to proceed through the dialog boxes to accept the license agreement and install the Express Tools. After the AutoCAD Express Tools Volumes 1-9 have been successfully installed, select *Finish* to exit the installation.

Figure 47-11



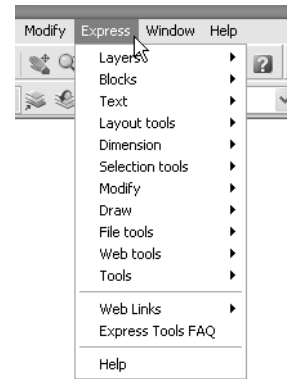
When you launch AutoCAD after completing the Express Tools installation, the four Express toolbars (*ET: Blocks*, *ET: Layers*, *ET: Text*, *ET: Standard*) should automatically be displayed in the Drawing Editor (Fig. 47-12).

Figure 47-12



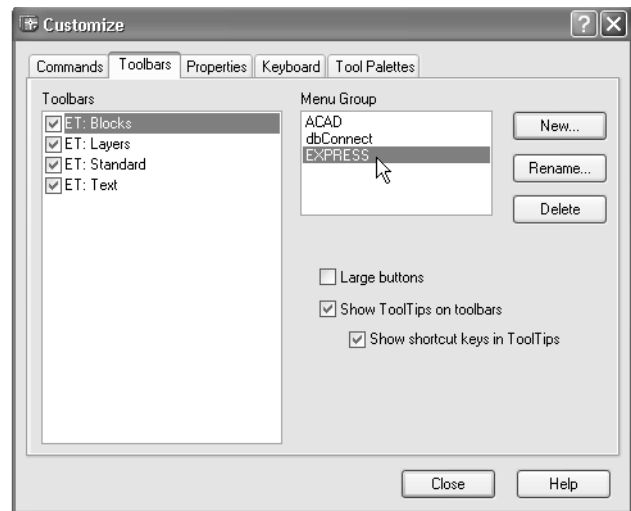
In addition, an *Express* pull-down menu should appear between the *Modify* and *Windows* menus (Fig. 47-13).

Figure 47-13



If the you close one or more Express Tool toolbars, you can reactivate the toolbars by using the *Toolbar* command to invoke the *Customize* dialog box (Fig. 47-14). From the *Menu Group* listing on the right, select *EXPRESS* to display the available toolbars on the left. Select the toolbar(s) you want to display, then select *Close*.

Figure 47-14



Express Layer Tools Commands

Several useful layer utilities are available in this set. The productivity you can gain from these express commands is well worth the trouble of loading these menus and toolbars. These commands operate as an extension to the standard layer commands, allowing you to accomplish in one command what might otherwise take two or three commands or what might require multiple object or dialog box selections.

LMAN



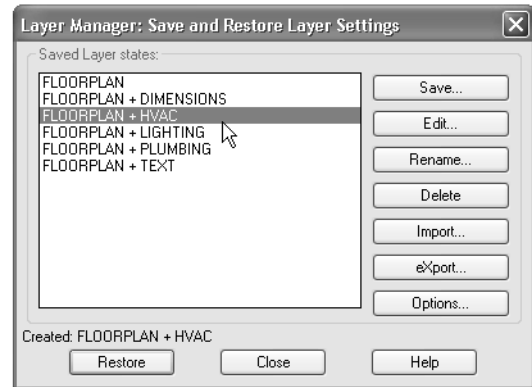
Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
Express Layers> Layer Manager...	LMAN

Lman is short for *Layer Manager*. *Layer Manager* allows you to save, restore, and edit layer settings. With many drawings you may have certain combinations of layers that are frozen, thawed, on, or off for particular operations. For example, you may require a certain combination of layers to be *On* for making a plot, while a different set of layers may be *On* for working with construction lines (*Xlines*, *Rays*) or for creating dimensions and text. Instead of using the *Layer* command each time you make a plot and each time you create and edit *Xlines*, dimensions, and text, you can *Save* these settings to be *Restored* at any later time.

Layer Manager can also save the current color of the layers. For example, this feature enables you to quickly switch between a colored layer set and a black-and-white layer set. A black-and-white layer set may be useful if you need to frequently copy and paste drawing objects into Word documents.

Lman produces the *Layer Manager: Save and Restore Layer Settings* dialog box (Fig. 47-15). The central area lists the *Saved Layer states* (empty when you first begin). The options operate as described here.

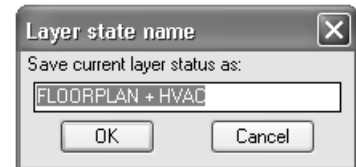
Figure 47-15



Save...

Using *Save...* invokes the *Layer State Name* dialog box (Fig. 47-16). Here you can assign a name to the current layer settings. Normally, you would want to specify the desired *On*, *Off*, *Freeze*, *Thaw* or *Color* state for each layer before using this option. However, once a layer state is created, you can use *Edit...* to change the *On*, *Off*, *Freeze*, *Thaw* or *Color* state of each layer. The layer states are saved in the drawing file, so you can *Restore* the layer states whenever needed. Spaces and other characters are allowed in the name.

Figure 47-16



Restore

Highlight a name from the list, then use this option to restore the selected *Saved Layer state*. AutoCAD automatically sets the *On*, *Off*, *Freeze*, *Thaw* and *Color* settings to those saved under the assigned name.

Edit...

This option allows you to change the layer settings for any named state. First, select the desired name from the list, then press *Edit...*. The *Layer* tab of the *Layer Properties Manager* dialog box (not shown) appears for you to make the desired changes.

Delete

This button deletes a highlighted *saved layer state* from the list.

Import...

You can import previously saved layer states from a file. Layer states are saved (with *Export...*) with a *.LAY* file extension. Importing works well when you have several drawings with the same layer names and functions (as when similar drawings are created from the same template drawing). Layer states can be saved in one drawing, then *Exported* to a file and *Imported* to the other drawings.

Export...

Once you have created the desired layer states, you can save them to a file (with a *.LAY* extension). The layer states can be imported into other drawings with *Import...*

Options...

Selecting this button produces the *Layer Manager: Restore Options* dialog box (not shown). This feature gives additional power for using and restoring layer states. Normally, when a layer state is restored, all the properties of the layers (when the layer state was created) are restored. However, with this dialog box you can specify which properties of the layers are to be restored when the layer state you select is

made current. For example, you may have a layer state set specifically for displaying objects in different lineweights, but want to view the drawing with all lines having the same lineweight. Simply restore the desired layer state leaving *Lineweight status* unchecked. The *Options* feature is also especially helpful for creating new layer states similar to existing layer states by restoring the layer state you want to “clone” but without restoring the properties you want to change.

LAYMCH

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Layers> Layer Match</i>	LAYMCH

Laymch allows you to change the layer of selected objects to the layer of the selected destination object. This command performs the same function as the *Matchprop* command (see Chapter 12), but only with respect to layers (*Matchprop* can change other properties such as color, linetype, or linetype scale).

Using *Laymch* produces the following prompt:

```
Command: laymch
Select objects to be changed:
Select objects: PICK
Select objects: PICK
Select objects: Enter
2 found.
Select object on destination layer or [Type-it]: PICK
2 objects changed to layer 0.
Command:
```

You can use the *Type-it* option to enter the name of the destination layer:

```
Select object on destination layer or [Type-it]: t
Enter layer name: text
2 objects changed to layer text.
```

LAYCUR

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Layers> Change to Current Layer</i>	LAYCUR

Laycur changes the layer of selected objects to the current layer. The following prompt is issued:

```
Command: laycur
Select objects to be changed to the current layer:
Select objects: PICK
Select objects: PICK
Select objects: Enter
2 found.
2 objects changed to layer TEXT (the current layer).
Command:
```

LAYISO

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Layers> Layer Isolate</i>	LAYISO

Use this express command if you want to turn *Off* all layers except the layer(s) of the selected object(s). Several layers can be selected to be “isolated.” If more than one layer is isolated, the last one selected becomes the current layer. The following prompt is used:

Command: **layiso**
 Select object(s) on the layer(s) to be isolated:
 Select objects: **PICK**
 Select objects: **PICK**
 Select objects: **Enter**
 2 layers have been isolated. Layer TEXT is current.
 Command:

COPYTOLAYER

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Layers> Copy Objects to New Layer</i>	COPYTOLAYER...

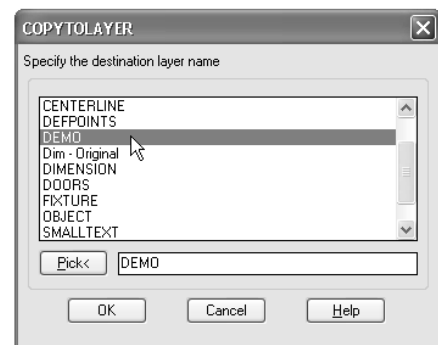
When you need two different layers that contain similar (or duplicate) objects, such as “Existing Doors” and “Demo Doors,” you can use this express feature to select existing object(s), then choose the destination layer for copied objects from a dialog box. The destination layer must exist prior to launching this command.

Command: **copytolayer**
 Select objects: **PICK**
 Select objects: **PICK**
 Select objects: **Enter**

After selecting the object(s) to copy to the new layer, the **COPYTOLAYER** dialog box opens (Fig. 47-17). Here you select the destination layer name from the list or use the *Pick* button to designate the destination layer by selecting an object on the desired layer.

When you choose *OK* to exit the dialog box, prompts appear confirming your selection (see prompt below). You can select a new location for the new objects or press *Enter* to copy the new objects to the same coordinates as the original set of objects.

2 objects copied and placed on layer “DEMO”.
 Base point or <return> to finish: **PICK or Enter**
 Specify second point of displacement or <use first point as displacement>: **PICK or Enter**
 Command:

Figure 47-17

LAYWALK



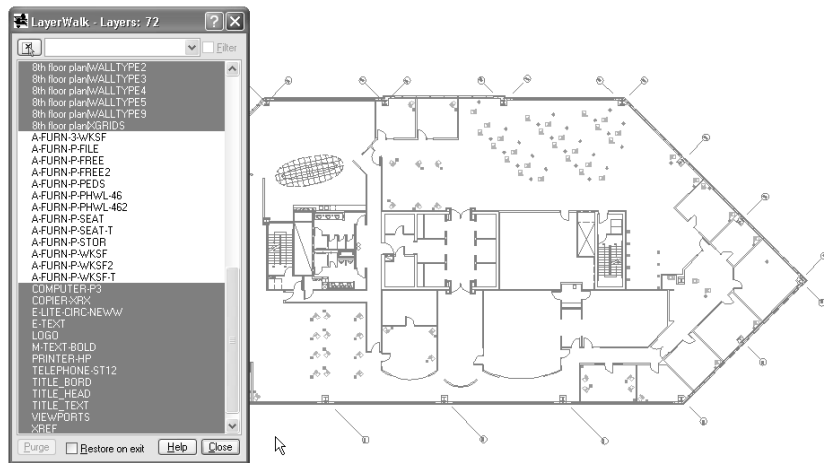
Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
Express Layers> Layer Walk	LAYWALK

The main feature of this utility enables you to view the objects on any layer(s) in a drawing even if the layers are *Off* or *Frozen*. *Laywalk* enables you to view objects on any specific layer or any set of layers, one layer at a time if desired. *Laywalk* is best utilized to examine the layer contents of unfamiliar drawings, such as those created by an outside source. Although *Laywalk* dynamically changes the *Freeze/Thaw* or *Off/On* state of any layers you select to view, you can choose not to keep the changes made using *Laywalk* and restore the original layer visibility settings when you exit this utility. The *LayerWalk* dialog box provides several options and a shortcut menu with additional features to manipulate the visibility of layers. You also have access to the *Layer Manager* utility from the shortcut menu, enabling you to manipulate and save layer states.

NOTE: Since *Laywalk* shows the contents of each layer, you may want to use *Zoom Extents* just prior to launching *Laywalk* with an unfamiliar drawing.

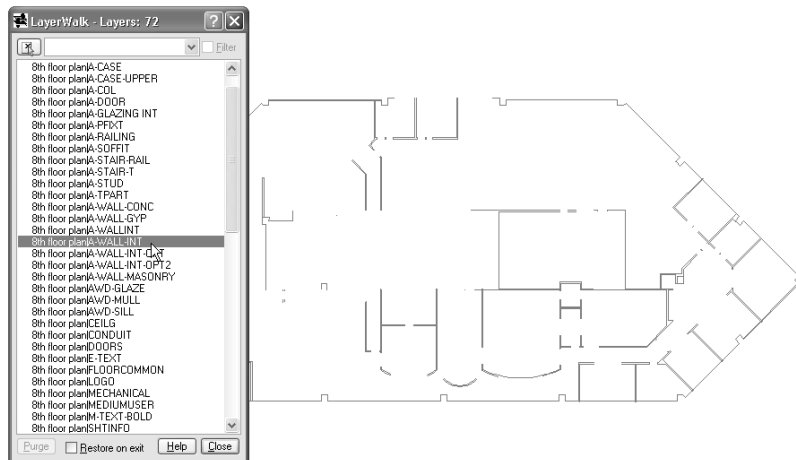
Invoking the *Laywalk* command by any method produces the *LayerWalk* dialog box. When the *LayerWalk* dialog box first appears, layers that are visible in the drawing are highlighted in the list. For example in Figure 47-18, the *LayerWalk* dialog box list indicates (highlights) the names of all visible layers in the drawing, but the A-FURN-* layers are not visible in the drawing and are not highlighted in the list. (The drawing used in this and the following illustrations is 8th Floor Furniture.dwg located in the ... \Sample folder.)

Figure 47-18



NOTE: The *LayerWalk* dialog box is sizable, so enlarge the dialog box if needed to see more of the layer names in the list.

Figure 47-19



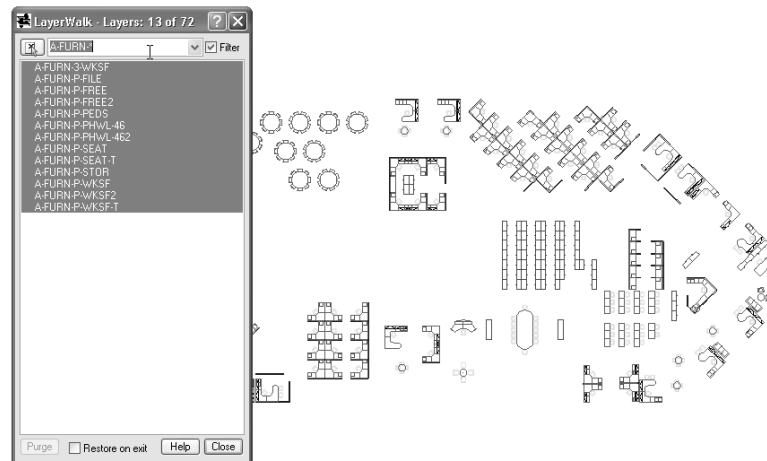
You can also easily view objects on each layer individually, one layer at a time. To view objects on a particular layer, select any layer name from the list. For example, Figure 47-19 displays only the geometry drawn on the Xref layer "8th floor plan | A-WALL-INT."

TIP To “walk” through the layers, highlight the first layer name from the list, then use the down arrow key on your keyboard to go through the list and display the geometry on each layer in the drawing, one layer at a time.

You can also select more than one layer by holding down the left button and dragging your cursor across the layers to view (or use Ctrl to select single layers or Shift to select a range).

TIP As an alternative, you can enter individual layer names in the edit box at the top of the *LayerWalk* dialog box to display that layer. Wildcards can be used. For example entering “A-FURN-*” in the edit box would display those layers in the drawing whose names began with “A-FURN-” (Fig. 47-20).

Figure 47-20



Layer name edit box and Filter check box

Use a filter to reduce the number of layers appearing in the list and in the drawing. Create a filter by entering layer names with a common prefix, then an asterisk, such as “A-FURN-*” (see Figure 47-20), or enter layer names of an Xref drawing, such as “8th floor plan*.” This action creates a temporary filter for the list. When the *Filter* box is checked, only those layers listed in the edit box appear in the list and in the drawing. With a filter in effect, you can still select individual names from the list to view the geometry, etc. Uncheck the *Filter* box to make all layers in the drawing appear in the list. A filter remains in effect until cleared by unchecking the box. Remember that filters are temporary (useful only during the current *LayerWalk* session) unless saved. See “Shortcut Menu,” *Save Filter*, to save filters for use in subsequent *LayerWalk* sessions.

Select Objects button

This button (upper-left corner of the dialog box) enables you to select one or multiple objects in the drawing, then returns to the *LayerWalk* dialog box with the objects layer(s) highlighted. This is an excellent way to examine an unfamiliar drawing and find out which layers specific objects were drawn on, especially when selected objects are on more than one layer.

Purge

Purge deletes layers from the drawing (even if *Restore on Exit* is checked). The *Purge* button is enabled only if there are one or more unreferenced layers (layers containing no objects) in the drawing and those layers are highlighted in the list. Use the *Select Unreferenced* option in the shortcut menu to locate and highlight any unused layers, then the *Purge* button to delete those layer names from the drawing and from the layer list.

Restore on Exit check box

Restore on Exit means to disregard any changes made to the layer status using *LayerWalk* and restore the drawing to the visibility state before using *LayerWalk*. Use this option if you want to examine the geometry on specific layers of a drawing but then return to work on the drawing without saving any of those layer visibility changes. In other words, if *Restore on Exit* is checked when you *Close* the *LayerWalk* dialog box, AutoCAD returns to the layers displayed prior to using *LayerWalk*. If you want to keep the layer display attained using *LayerWalk*, but return to work on the drawing, make sure *Restore on Exit* is not checked before you close the dialog box.

Shortcut menu

Right-click in the *LayerWalk* dialog to display the shortcut menu (Fig. 47-21).

Hold Selection, Release Selection, Release All

If you want to turn on the display of a set of layers, then select other layers from the list to examine the geometry in the drawing, use *Hold Selection*. For example, in an architectural setting you could highlight the floorplan layer names, use *Hold Selection* to keep those layers visible, then select other layer names to display that geometry within the context of the floorplan. When you “hold” a list of layers, highlighting is removed from the names, so “held” selections appear in the list with an asterisk before the name. You can “hold” multiple selections. Use *Release Selection* to remove one selection from “hold” status and use *Release All* to remove multiple layer selections.



Select All, Clear All

Use *Select All* to select all names in the list or *Clear All* to clear the list. Use *LayerWalk's Select All* feature to quickly turn on all layers in a drawing, even if the layers selected are *Frozen*!

Invert Selection

The *Invert Selection* option highlights (and displays in the drawing) all layers from the list previously not highlighted and clears (and does not display in the drawing) those names previously shown.

Select Unreferenced

The *Select Unreferenced* feature highlights any unreferenced layers in the drawing (layers that contain no geometry). Using *Select Unreferenced* is a quick way to locate unused layers so you can then use the *Purge* button to remove those layer names from the drawing.

Save Layer State

The *Save Layer State* feature enables you to give a name to the highlighted layer group for future use. A *Layer State* is a particular layer visibility setting saved with a name. For example, if you use *LayerWalk* to specify a set of layers for viewing, but expect to view this specific layer set again sometime in the future, *Save* the layer state. In this way you can *Restore* the layer state by name in the future rather than having to select that specific set of layers one by one in the *LayerWalk* dialog box. Named layer states are available in the Express tool *Layer Manager* (but are not available in the *State Manager* from the *Layer Properties Manager*). See previous discussion, “*Layer Manager*,” for more information. Express tool layer states are similar to layer states available in the *Layer Properties Manager* discussed in Chapter 11.

Figure 47-21

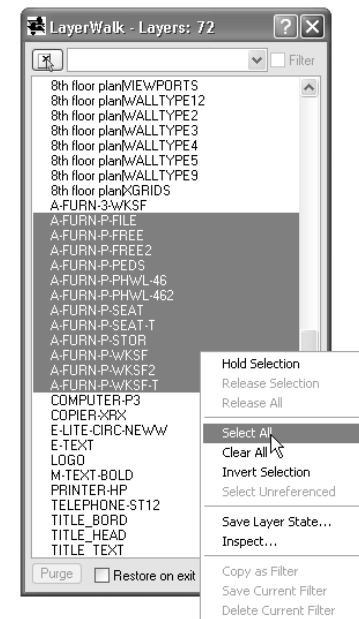
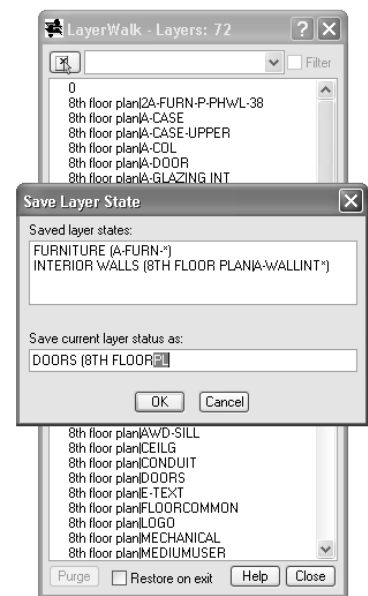


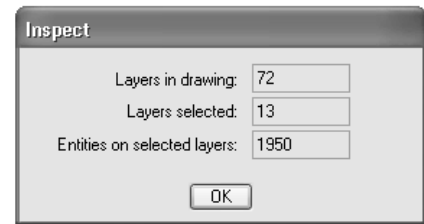
Figure 47-22



Inspect

Select *Inspect* to open the *Inspect* dialog box (Fig. 47-23). This utility displays the number of layers in the drawing, the number of layers selected (if any), and the number of objects on the selected layers.

Figure 47-23

*Copy Filter, Save Current Filter, Delete Current Filter*

Use the *Copy Filter, Save Current Filter, Delete Current Filter* functions to capture and discard layer filter names. Remember that filters are temporary (useful only during the current *LayerWalk* session) unless saved (see previous discussion, “Layer name edit box and *Filter* checkbox”). To save a filter for use in subsequent *LayerWalk* sessions (after closing the *LayerWalk* dialog box), create a filter by entering a layer name string in the edit box (see previous Figure 47-20), then select *Save Current Filter* from the shortcut menu. Or, use *Copy Filter* if a single layer name has been highlighted and you want to save the name as a filter. The named filter then appears in the drop-down list whenever *LayerWalk* is used again for the drawing. The filter is saved regardless of the setting for *Restore on Exit*. The *LayerWalk* filters are not available as named layer filters from the *Layer Properties Manager*.

LAYFRZ

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Layers> Layer Freeze</i>	LAYFRZ

This utility *Freezes* the layer(s) of the selected object(s). More than one object (layer) can be selected for *Freezing* with this command, but you select only one object at a time, then the prompt repeats. You cannot *Freeze* the current layer. AutoCAD gives the following prompt:

Command: **layfrz**

Select an object on the layer to be frozen or [Options/Undo]: **PICK**

Layer TEXT has been frozen.

Select an object on the layer to be frozen or [Options/Undo]: **PICK**

Layer GEOMETRY has been frozen.

Select an object on the layer to be frozen or [Options/Undo]: **Enter**

Command:

You can use *Undo* to undo the last layer (object) selected. Type *O* for *Options*, which issues the following prompt:

Select an object on the layer to be frozen or [Options/Undo]: **o**

Enter an option [Entity level nesting/No nesting]<Block level nesting>:

With the *No Nesting* option, only layers in the current drawing (not Xref layers or nested Block layers) are *Frozen*. *Entity level nesting* and *Block level nesting* automatically *Freeze Xrefs* or *Blocks* down to the level of the selected object (*Block* or *Xref*).



Use *Layfrz* to freeze one viewport of a multi-viewport layout.

LAYTHW

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Layers> Thaw All Layers</i>	<i>LAYTHW</i>

Use this command to *Thaw* all layers in the drawing. Layers that are *Frozen* and *Off* are only *Thawed* by this command so they still remain invisible until turned *On*:

Command: **laythw**

All layers have been thawed.

Command:

LAYOFF

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Layers> Layer Off</i>	<i>LAYOFF</i>

Layoff is used to turn *Off* the layer(s) of the selected object(s). Multiple objects can be selected (one at a time) to turn off their layers. The *Options* are the same as those for *Layfrz*:

Command: **layoff**

Select an object on the layer to be frozen or [Options/Undo]: **PICK**

Layer ASESMP}1-WALL has been turned off.

Select an object on the layer to be frozen or [Options/Undo]: **Enter**

Command:

LAYON

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Layers> Turn All Layers On</i>	<i>LAYON</i>

This command is a quick way to turn all layers on. Layers that are *Frozen* are not affected; only layers that are *Off* can be turned *On* with this command:

Command: **layon**

Warning: layer 0 is frozen. Will not display until thawed.

All layers have been turned on.

Command:

LAYLCK

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Layers> Layer Lock</i>	<i>LAYLCK</i>

Laylck locks the layers of selected objects. There are no options for this command. If objects are selected that are part of an *Xref* or *Block*, the layer that was current when the *Xref* or *Block* was *Attached* or *Inserted* is locked.

Command: **laylck**

Select an object on the layer to be locked: **PICK**

Layer TEXT has been locked.

LAYULK

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Layers> Layer Unlock</i>	<i>LAYULK</i>

Layulk has the opposite function of *Laylck*; that is, it unlocks the layers of selected objects. If the selected objects are part of an *Xref* or *Block*, the layer that was current when the *Xref* or *Block* was *Attached* or *Inserted* is unlocked. There are no options for this command.

Command: **layulk**

Select an object on the layer to be unlocked: **PICK**

Layer TEXT has been unlocked.

Command:

LAYVPI

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Layers> Isolate Layer to Current Viewport</i>	<i>LAYVPI</i>

This feature is intended to be used to freeze layers within viewports. *Layvpi* freezes specific layers in all viewports but the current one. You identify the layer(s) you want to freeze by selecting any object on the desired layer. Make sure you select the objects from within the layout in which you do not want the layer(s) frozen.

Command: **layvpi**

Select an object on the layer to be Isolated in viewport or [Options/Undo]: **PICK**

Layer A-FURN-P-SEAT has been frozen in all viewports but the current one.

When you select an object, its layer is immediately frozen in all other viewports. The prompt repeats so you can continually select objects (on layers to freeze). If you type *O* for *Options*, the following prompt appears:

Select an option [Layouts/Selection]:

The *Selection* option here is the same as that found in *Layfrz* which allows you to determine the level of freezing for nested *Blocks* or *Xrefs* (see previous discussion under “*Layfrz*”).

Normally using *Layopi* affects viewports only in the current layout. If you want to have layers frozen for all layouts, enter *L* for *Layouts* at the previous prompt to choose from the following:

Isolate layers in [All layouts/Current layout] <Current>:

The *All layouts* option specifies that the selected layer is frozen in all layouts. In other words, with this option, selecting an object on a layer in a viewport not only freezes that layer for all other viewports in that layout, but it also freezes the layer in all other viewports in all the layouts in that drawing.

LAYMRG

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Layers> Layer Merge</i>	<i>LAYMRG</i>

Laymrg combines selected layers with any other layer. Objects on the layers to merge are moved to the target layer, then the merged layer names are deleted. For example, merging layers LAYER1 and LAYER2 to layer 0 would move the related objects to layer 0 and delete layers LAYER1 and LAYER2.

Command: *laymrg*

Select object on layer to merge or [Type-it/Undo]: **PICK**

Selected layers: Layer2

Select object on layer to merge or [Type-it/Undo] <done>: **PICK**

Selected layers: Layer1,Layer2

Select object on layer to merge or [Type-it/Undo] <done>: **Enter**

Select object on target layer or [Type-it]: **PICK**

***** WARNING *****

You are about to permanently merge the following layers into layer 0:

Layer1

Layer2

Do you wish to continue? [Yes/No] <No>: **y**

Merging layer Layer1 into layer 0.

Merging layer Layer2 into layer 0.

All entities which were on layer Layer1 have been moved to layer 0.

All entities which were on layer Layer2 have been moved to layer 0.

Deleting layer “Layer1”.

Deleting layer “Layer2”.

2 layers deleted.

Command:

LAYDEL

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Layers> Layer Delete</i>	LAYDEL

Laydel deletes all objects on a selected layer, then deletes the layer (name). For example, selecting an object on LAYER1 would cause *Laydel* to delete LAYER1 and all objects residing on that layer.

Command: **laydel**

Select object on layer to delete or [Type-it/Undo]: **PICK**

Selected layers: Layer1

Select object on layer to delete or [Type-it/Undo] <done>: **Enter**

***** WARNING *****

You are about to permanently delete layer Layer1 from this drawing.

Do you wish to continue? [Yes/No] <No>: **y**

Deleting layer "Layer1".

1 layer deleted.

Command:

Express Block Tools Commands

In AutoCAD 2004, these commands are grouped in the *ET: Blocks* toolbar and in the *Blocks* cascading menu (under the *Express* pull-down menu).

NCOPY

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Blocks> Copy Nested Entities</i>	NCOPY

Ncopy allows you to copy objects that are nested within *Xrefs* or *Blocks*. With built-in AutoCAD commands this is possible to do for *Xrefs* (using *Xbind*), but the process is more complex than *Ncopy*. *Ncopy* allows you to select any object within an *Xref* or *Block* to use for copying. *Ncopy* operates similarly to the *Copy* command:

Command: **ncopy**

Select nested objects to copy: **PICK**

Select nested objects to copy: **Enter**

Select objects: Specify base point or displacement, or [Multiple]: **PICK**

Specify second point of displacement or <use first point as displacement>: **Enter**

Command:

BTRIM

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Blocks> Trim to Block Entities</i>	BTRIM

Btrim allows you to trim objects using nested *Block* or *Xref* objects as cutting edges. The command is simple to use since it operates identically to the *Trim* command. If you select an object within a clipped *Xref*, the entire object (outside the clipping boundary) is highlighted.

Command: **btrim**

Select cutting edges: **PICK**

Select cutting edges: **Enter**

Select objects:

Select object to trim or shift-select to extend or [Project/Edge/Undo]: **PICK**

Select object to trim or shift-select to extend or [Project/Edge/Undo]: **Enter**

Command:

Btrim has the same options as *Trim*, and uses the same method to implement *Extend* or *Trim* by holding down the Shift key when selecting objects (see *PROJMODE* and *EDGEMODE*, Chapter 9).

BEXTEND

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Blocks> Extend to Block Entities</i>	BEXTEND

Bextend allows you to extend objects using nested *Block* or *Xref* objects as boundary edges. This utility (like the *Btrim* utility) is similar to the built-in AutoCAD counterpart but with the added capability of extending objects to *Block* and *Xref* objects. *Bextend* operates identically to the *Extend* command:

Command: **bextend**

Select edges for extend: **PICK**

Select edges for extend: **Enter**

Select objects:

Select object to extend or shift-select to trim or [Project/Edge/Undo]: **PICK**

Select object to extend or shift-select to trim or [Project/Edge/Undo]: **Enter**

Command:

XLIST

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Blocks> List Xref/Block Entities</i>	XLIST

The *Xlist* command displays properties of objects that are nested in *Xrefs* or *Blocks*. This is helpful since the *List* command reports information about only the top level object (*Xref* or *Block*), not information on nested objects. Using *Xlist* produces the following prompt:

Command: **xlist**
 Select nested xref or block object to list: **PICK**

Figure 47-24



After selecting an object within an *Xref* or *Block*, *Xlist* produces the *Xref/Block Nested Object List* dialog box (Fig. 47-24). This dialog box lists properties of the selected *Block* or *Xref*.

CLIPIT



Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Blocks> Extended Clip</i>	CLIPIT

AutoCAD has the unique feature of clipping *Xrefed* drawings by specifying a polygonal or rectangular *Pline* (no arc segments) as the clipping boundary. *Clipit* extends that capability by allowing you to use an existing *Circle*, *Arc*, *Pline*, *Ellipse*, or text as a clipping edge for an *Xref*, *Block*, or *Wipeout* (see *Wipeout*):

Command: **clipit**
 Pick a POLYLINE, CIRCLE, ARC, ELLIPSE, or TEXT object for clipping edge...
 Select objects: **PICK**
 Command:
 Pick an IMAGE, a WIPEOUT, or an XREF/BLOCK to clip...
 Select objects: **PICK**
 Command:
 Enter maximum allowable error distance for resolution of arc segments <0.0200>: **Enter** or (value)
 Command:

BURST



Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Modify> Explode Attributes to text</i>	BURST

Use *Burst* with attributed text (text combined with a *Block*) like you would use *Explode* with an unattributed *Block*. *Burst* actually explodes text attributes but converts the attributes to *Text* objects. Normally, if you select attributes to *Explode*, the *Block* is exploded and the text is changed back to text, indicating the original attribute definitions—that is, tags, values, and prompts. (See Chapter 22, Block Attributes.) Using *Burst* may cause the newly created *Text* objects to be misaligned from their previous orientation.

Command: **burst**
 Select objects: **PICK**
 Select objects: **Enter**
 Command:

SHP2BLK

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express></i> <i>Blocks></i> <i>Convert Shape to Block</i>	SHP2BLK

AutoCAD “shapes” are files containing alpha-numeric code that defines “strokes.” The “strokes” can “draw” shapes that can be geometry or characters of an alphabet. *Shp2blk* converts AutoCAD “shapes” into *Blocks*.

Before using *Shp2blk*, you must have a shape object placed in the drawing. Once you have placed a shape in a drawing (using the *Shape* command), use *Shp2blk* to convert the shape into a *Block*. Follow this prompt sequence:

Command: **shp2blk**
 Select shape entity to convert: **PICK**
 Select objects: **Enter**
 Enter the name of the block to create <CHKARO>: **Enter**
 The shape CHKARO has been replaced with block CHKARO.

Keep in mind with the Express Tools you can easily create shape files from existing geometry. Use *Makeshp* (*Express/Tools/Make Shape*) to create a shape file from existing geometry you select. (See “Express Tools, Tools Commands.”)

ATTOUT

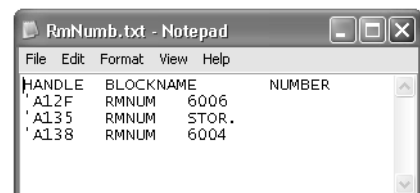
Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express></i> <i>Blocks></i> <i>Export Attribute Information</i>	ATTOUT

This feature extracts attributes from one or more *Blocks*. *Attout* creates an editable, tab-delimited .TXT file from any attributed *Blocks* you select. The file that is created contains the block-specific handle information (unique to the drawing in which it was created). This procedure is similar to AutoCAD’s *Atttext* command but does not require the use of an attribute template file (.TXT).

Command: **attout**
 Initializing...
 Select objects: **PICK**
 Select objects: **PICK**
 Select objects: **PICK**
 Select objects: **Enter**
 Output file: “C:\Dwgs\RmNumb.txt” created.

Note that you can select more than one *Block* during the command. You can select either the block geometry or the attributes.

Figure 47-25 is an example of the resulting .TXT file, viewed using Windows Notepad. The attribute tag headers and the values for each block selected are shown. The resulting .TXT file can be

Figure 47-25

edited, then re-assigned to the drawing using the *Attin* Express Tool (see next).

NOTE: Do not change the column alignment of the .TXT file created from *Attout*. Since this file is a tab-delimited file, edits to the column alignment may produce unexpected results if you use *Attin*.

ATTIN

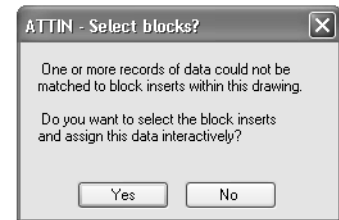
Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express></i> <i>Blocks></i> <i>Import Attribute Information</i>	<i>ATTIN</i>

Attin converts an attribute extract file back to attributes in a drawing. This feature works well with extract files created with *Attout* or extract files in the same format (.TXT files that contain block handle-specific information). This feature is useful if you use *Attout*, then edit the resulting .TXT file, then use *Attin* in order to reassign new attribute data to the same blocks in the same drawing.

Invoke the command, then select the desired .TXT file in the *Enter input filename* dialog box, then select *Open*. If AutoCAD finds the block name and handle information in the drawing that is referenced in the .TXT file, the new attribute information is inserted and the command ends.

If the edits made to the .TXT file do not result in a match (do not find the specific *Blocks* based on the handle information), an *ATTIN* warning box appears (Fig. 47-26). If you choose *Yes* to assign the data interactively, you are prompted to select the blocks for the attribute assignment.

Figure 47-26



BLOCKTOXREF

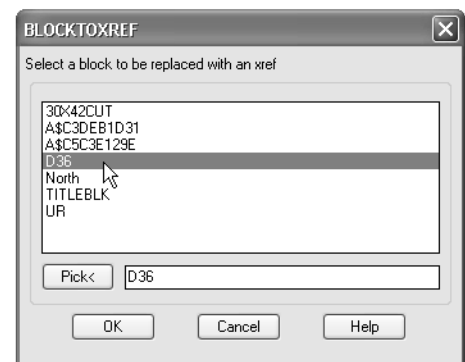
Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express></i> <i>Blocks></i> <i>Convert Block to Xref</i>	<i>BLOCKTOXREF</i>

The *Blocktoxref* utility does not convert *Blocks* in the current drawing to *Xrefs*, but it allows you to replace *Blocks* defined in the drawing with existing drawings that can be selected as *Xref* attachments.

If changes to a *Block* in a drawing are frequent, you may want to consider converting all *Block* references to *Xref* attachments. This enables you to make one change in the externally referenced file, then all instances of the *Xref* will reflect the change in the drawings to which it is attached.

Prior to beginning this routine, you must have a .DWG file available to select as the *Xref*. When you invoke *Blocktoxref*, the *BLOCKTOXREF* dialog box appears (Fig. 47-27) listing the existing blocks in the drawing. Choose the *Block* name you want to replace, then select *OK*. From the *Select an xref file* dialog box (not shown), choose the *Xref* you want to swap for the *Block*, then choose *Open*.

Figure 47-27



Next, the following prompt appears regarding purging the block being converted.

```
Purge unreferenced items when finished? <Y>:
10 block inserts replaced with xref: C:\Dwgs\D36-Tag.dwg
Results may not be apparent until next regen.
```

If you will no longer need that *Block*, press Enter to accept the default of <Y> and that block will no longer be defined in the drawing's block definition table or available from the *Insert* dialog box.

BLOCKREPLACE

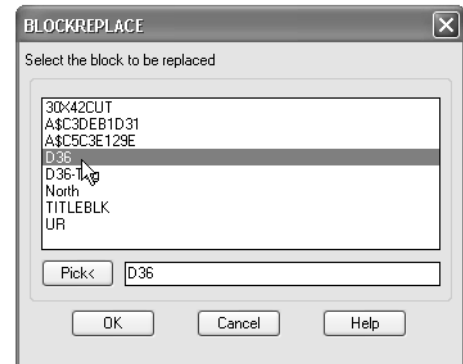
Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express></i> <i>Blocks></i> <i>Replace block with another block</i>	BLOCKTOREPLACE

Blockreplace allows you to replace all insertions of a *Block* in a drawing with another *Block* from the drawing. The new *Block* must be contained in the current drawing before you launch *Blockreplace*.

When you begin *Blockreplace*, the **BLOCKREPLACE** dialog box appears (Fig. 47-28). The list includes all the existing blocks in the drawing. Notice the prompt above the list of block names stating, "Select the block to be replaced." Select the block to replace, then select *OK*. In the next **BLOCKREPLACE** dialog box (not shown), the prompt asks you to "Select a block to replace ..." and displays the same list of available blocks in the drawing. Select the desired *Block* to use as the replacement, then *OK*. Before making the replacement, the following prompt appears asking if you want to *Purge* the *Block(s)* that are no longer used.

```
Purge unreferenced items when finished? <Y>:
Purging...
No unreferenced linetypes found.
No unreferenced linetypes found.
done.
10 blocks replaced.
Results may not be apparent until next regen.
```

Figure 47-28



Express Text Tools Commands

Several outstanding text utilities are included in this set. For applications that insert and manipulate text, these commands are truly an express. Some of the functions of these commands can be accomplished otherwise only by writing custom AutoLISP routines, not by using standard AutoCAD commands.

RTEXT

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Text> Remote Text</i>	<i>RTEXT</i>

Rtext is a streamlined method for importing external text. The text to import can be from an external file or can be created “on the fly” using the *Diesel* option. The *File* option produces the *Select File* dialog box for you to select a .TXT file for importation. The text is drawn in AutoCAD based on the current settings for *Style*, *Height*, and *Rotation*.

Command: **rtext**

Initializing...

Current settings: Style=Standard Height=0.5000 Rotation=0

Enter an option [Style/Height/Rotation/File/Diesel] <File>:

When the text is imported into AutoCAD, it is listed as an *Rtext* object and cannot be edited unless exploded. Using *Explode* on *Rtext* converts the text to an *Mtext* object.

TEXTFIT

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Text> Text Fit</i>	<i>TEXTFIT</i>

Textfit fits text between two specified points. Using this command gives the same results as using the *Fit* option of text alignment; however, this Express command acts on existing text:

Command: **textfit**

Select Text to stretch or shrink: **PICK**

Specify endpoint or [Start point]: **PICK**

Command:

Using the default option you need to PICK only the new ending point and the line of text is changed using the existing start point but specifying a new endpoint. You can also change the start point and endpoint, as shown below:

Command: **textfit**

Select Text to stretch or shrink: **PICK**

Specify endpoint or [Start point]: **s**

Specify new starting point: **PICK**

ending point: **PICK**

Command:

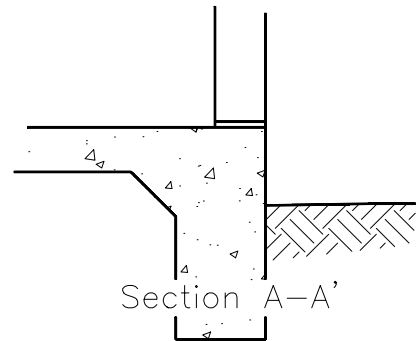
TEXTMASK



Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
Express Text> Text Mask	TEXTMASK

When text and other objects are drawn “on top of” each other (so they occupy the same space), you can use *Textmask* to make the text appear “in front of” other objects by “masking” the other objects. For example, using *Textmask* creates text that masks other drawing objects as shown in Figure 47-29. Use the following command syntax:

Figure 47-29



Command: **textmask**
 Initializing...
 Current settings: Offset factor = 0.3500, Mask type = Wipeout
 Select text objects to mask or [Masktype/Offset]: **PICK**
 1 found
 Current settings: Offset factor = 0.3500, Mask type = Wipeout
 Select text objects to mask or [Masktype/Offset]: **Enter**
 Masking text with a Wipeout Wipeout created.
 1 text items have been masked with a Wipeout.
 Command:

Offset factor is the distance around the text for the imaginary box that is used for “masking” (trimming) the objects behind the text.

Select text objects to mask or [Masktype/Offset]: **o**
 Mask offset currently set to 0.3500
 Enter offset factor relative to text height <0.3500>: (**value**)

The *Masktype* option allows for three types of masking. A *Wipeout* obscures the background objects completely. A *3dface* mask does not mask the background objects until a *Hide* is performed. The *Solid* option allows you to select a solid color from the standard *Color* dialog box to use for obscuring the background objects.

Select text objects to mask or [Masktype/Offset]: **m**
 Mask type currently set to Wipeout
 Specify entity type to use for mask [Wipeout/3dface/Solid] <Wipeout>: (**option**)

TEXTUNMASK

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
Express Text> Unmask Text	TEXTUNMASK

Use *Textunmask* to remove the mask (wipeout, 3dface, or solid color) from text that a *Textmask* was previously applied to.

Command: **textunmask**
 Select text or MText object from which mask is to be removed.
 Select objects: **PICK**
 Select objects: **Enter**
 Removed mask from one text object.
 Command:

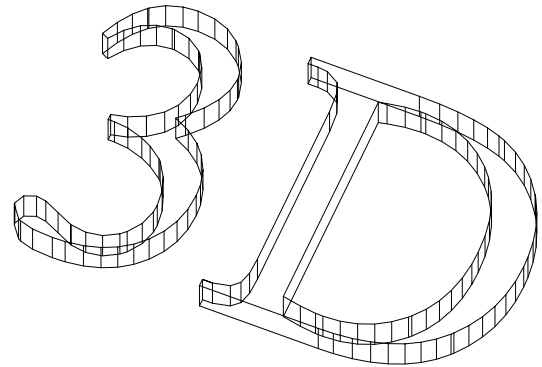
TXTEXP



Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
Express Text> Explode Text	TXTEXP

Textexp explodes text. The resulting objects are *Plines*. As you probably know, a line or paragraph of text is one object; therefore, without this command you cannot explode text. Exploded text is helpful if you need to alter or manipulate the individual text characters. For example, to convert text into 3D solids, use *Textexp*, convert each letter to a *Region*, then *Extrude* the text (Fig. 47-30):

Figure 47-30



Command: **txtexp**
 Select text to be EXPLODED:
 Select objects: **PICK**
 Select objects: **Enter**
 1 found.
 1 text object(s) have been exploded to lines.
 The line objects have been placed on layer 0.
 Command:

TXT2MTXT

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
Express Text> Convert Text to Mtext	TXT2MTXT

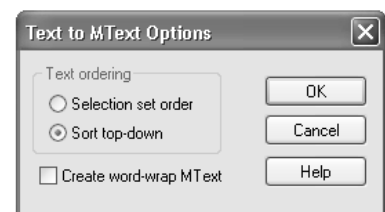
Use *Txt2mtxt* to convert text created with the *Text* or *Dtext* command to an *Mtext* object.

Command: **txt2mtxt**
 Initializing..._TXT2MTXT
 Select text objects, or [Options]<Options>: **PICK**
 Select objects: **PICK**
 Select objects: **PICK**
 Select objects: **Enter**
 2 Text objects removed, 1 MText object added.
 Command:

When the conversion is complete, no visible change may be apparent depending on the options you used; however, the new *Mtext* object can be edited using the typical *Mtext* editing methods (see Chapter 18 for more information on editing *Mtext*).

Figure 47-31

Pressing Enter or entering *O* for *Options* at the first prompt (before selecting text to convert) produces the *Text to Mtext Options* dialog box (Fig. 47-31). Here you can select from the following options.



Text Ordering

The *Text Ordering* option you select in this dialog box is important only when converting multiple text objects at one time with the *Txt2mtxt* command. Only one of the two methods can be used at a time.

Sort top-down

The *Sort top-down* option keeps the lines of *Text* or *Dtext* in the same order after conversion. You can select the desired lines of *Text* or *Dtext* by any method or in any order. Figure 47-32 displays how four lines of *Dtext* would be converted with the *Sort top-down* option. Note that before and after conversion the lines of text appear the same.

Figure 47-32

Sample text
 1. First line
 2. Second line
 3. Third line

Selection set order

With this option, you must select the text objects to convert in the order you want the text to appear after conversion. Therefore, you should select individual lines of text using the pickbox rather than using a window method. Figure 47-33 displays how the four lines of *Dtext* would appear after conversion using this option and having selected the lines of text to convert (in Figure 47-32) from bottom to top.

Figure 47-33

3. Third line
 2. Second line
 1. First line
 Sample text

Create word-wrap MText

If the *Create word-wrap MText* box is checked, *Txt2mtxt* creates a single paragraph of *Mtext* from the selected text objects and word-wraps the resulting *Mtext* to the width of the longest (original) text string in the selection. Figure 47-34 shows the four lines of text converted (from Figure 47-32) using the word-wrap option but not changing the order. Note that the resulting paragraph, in this case, is confusing to read after word-wrapping since the next character (“1.”) from the second line wraps up to the first, and so on, based on the width of the paragraph (text boundary is shown highlighted). To maintain the same line breaks as the original *Dtext* objects, don’t check the *Create word-wrap Mtext* box.

Figure 47-34

Sample text 1.
 First line 2.
 Second line 3.
 Third line

ARCTEXT



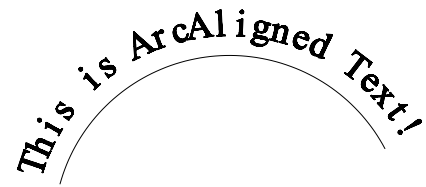
Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
Express Text> Arc Aligned Text	ARCTEXT

There is no other easy method to use in AutoCAD to create text along an arc. The built-in AutoCAD commands allow creation of a line of text only along a straight line. With this command, you can “attach” text to an existing *Arc* (Fig. 47-35, on the next page). You can use the same command to edit existing *ArcAlignedText* objects, as noted by the prompt:

Command: **arctext**

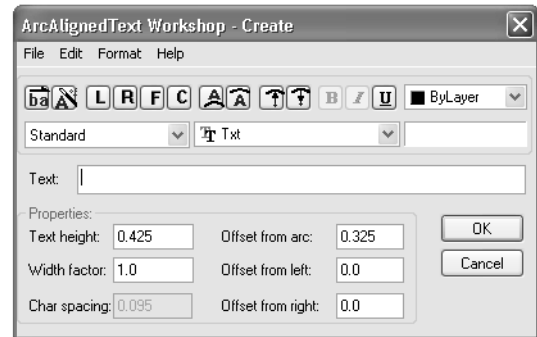
Select an Arc or an ArcAlignedText: **PICK** (existing *Arc* or existing *ArcAlignedText*)

Figure 47-35



After selecting an arc to use for text alignment or selecting an existing *ArcAlignedText* object, AutoCAD produces the *ArcAlignedText Workshop* dialog box (Fig. 47-36). Here you can control the typical text features (style, font, height, width factor, etc.) as well as many special features.

Figure 47-36



The following options are available in the *ArcAlignedText Workshop* dialog box.

Reverse Text

This option reverses the text so it reads backwards.

Alignment

Left, *Right*, *Fit*, and *Center* methods can be used. If *Fit* is not selected, you can specify the *Offset from left* and *Offset from right*.

Position

The text can be created on the *Convex* or *Concave* side of the arc. You can also set a value for *Offset from arc*.

Outward from the center/Inward to the center

This controls which direction the “top” of the letters point. Using Figure 47-35 as an example, if *Inward to the center* were selected, the top of the letters would point downward resulting in upside-down text.

Typeface

Bold, *Italic*, and *Underline* can be selected.

Other Options

Other typical text options are available, such as text *Height*, *Width factor*, *Style*, and font file.

Existing *ArcAlignedText* objects can be modified with grips but keep the concentric orientation with the aligned arc.

TJUST

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express> Text> Justify Text</i>	<i>TJUST</i>

Tjust changes the insertion point for existing text created with *Text*, *Dtext*, or *Mtext*. *Tjust* does not change the location of the text, it only moves the insertion point to a new location on the existing text.

Command: **tjust**
 Select objects: **PICK**
 Select objects: **Enter**
 Enter new justification...
 [Start/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR] <Start>: Enter option
 1 objects modified.

For example, assume you created a line of *Dtext* using the *Center* justification option (Fig. 47-37). Note that *Dtext* has one grip at the lower-left corner and one at the justification point (center in this case). The grips can be used to edit the text or the *INSertion Osnap* mode can be used to select the text's insertion point. For a review of the various justification options for text and information about editing text, see Chapter 18, *Creating and Editing Text*.

Figure 47-37



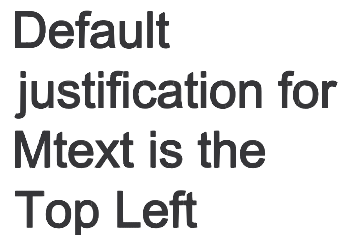
Figure 47-38 displays the same line of *Dtext* after using *Tjust* to change the justification to *BR* (bottom right). Note the new grip at the bottom right instead of at the center.

Figure 47-38



Although *Tjust* can be used on both *Dtext* and *Mtext* objects, its advantage may be more evident when applied to *Mtext*. If you use *Tjust*, for example, to change an *Mtext* object's justification from the default location of *TL* (top left) (see Figure 47-39, left) to *BR* (bottom right), the text automatically adjusts to the new justification (see Figure 47-39, right). Keep in mind that the text boundary does not change—the text justification point is changed within the existing text boundary.

Figure 47-39




TORIENT

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express></i> <i>Text></i> <i>Rotate Text</i>	<i>TORIENT</i>

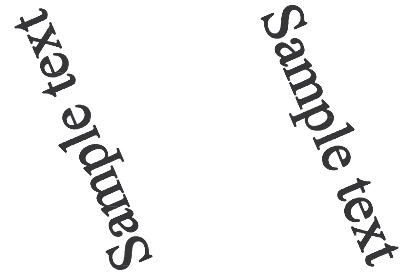
Use the *Torient* command to adjust the orientation of existing text (including *Dtext*, *Mtext*, or attribute text within *Blocks*) to a readable position. *Torient* cannot be used to edit dimension text or text within tolerance dimension feature control frames. The routine enables you to either specify an absolute rotation angle for the selected text or use the default option of *Most Readable*.

Command: **torient**
 Select TEXT, MTEXT, ATTDEF, or BLOCK inserts w/attributes...
 Select objects: **PICK**
 Select objects: **Enter**
 New absolute rotation <Most Readable>: **0**
 5 objects modified.

The *New absolute rotation* option prompts for a rotation angle value, then rotates the selected text about its middle justification point, not its default insertion points. Enter a value representing the desired angle for the new orientation.

The *Most Readable* option does not rotate the line of text. This option only ensures that the text is read from left to right. For example, if the text appears somewhat upside down and reads from right to left (see Figure 47-40, left), the *Most Readable* option “flips” the text about its center (see Figure 47-40, right) to read mostly right to left.

Figure 47-40



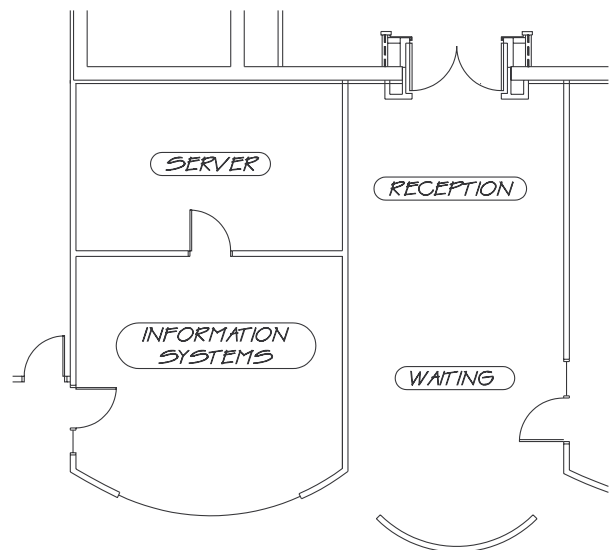
TCIRCLE

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express></i> <i>Text></i> <i>Enclose Text with Object</i>	TCIRCLE

Tcircle allows you to create a circle, slot, or box around existing text objects. This is a great command to use for emphasizing specific text such as part numbers, room names or numbers, and column or data callouts. This routine operates with *Dtext*, *Mtext* or *Attdef* text objects (before creating the *Block*) and encloses the selected objects with your choice of circles, slots, or rectangles. Figure 47-41 displays slots created with *Tcircle*.

Command: **tcircle**
 Select TEXT, MTEXT or ATTDEF objects...
 Select objects: **PICK**
 Select objects: **Enter**
 Enter distance offset factor <0.3500000>: **Enter**
 Enclose text with [Circles/Slots/Rectangles]
 <Slots>: **Enter** or option
 Create slots of constant or variable size
 [Constant/Variable] <Variable>: **Enter** or option
 Creating Slots...Done.
 4 Slots created.

Figure 47-41



The new objects are drawn on the current layer. Slots and boxes that are created are actually *Plines*.

Distance offset factor

The prompt to *Enter distance offset factor* enables you to specify a percentage of offset distance for the new object from the body of the text. For example, the default of 0.35 makes the offset distance 35% of the largest text character. This option is similar to the that found in the Express tools *Textmask* command.

Constant/Variable

The *Constant/Variable* option allows you to specify whether you want the same box, circle, or rectangle size for all the selected text objects or if the new objects should adjust to accommodate the size of the individual text objects. With the default option of *Variable*, the routine automatically adjusts the object based on the text size and your offset factor (see Figure 47-41). If you opt for *Constant* the following prompt appears:

```
Create slots of constant or variable size [Constant/Variable] <Variable>: C
Maintain constant slot [Width/Height/Both] <Both>: Enter
Determining best size...Done.
Creating Slots...Done.
4 Slots created.
```

Since *Tcircle* creates new objects around individual text objects, you may want to first convert multiple lines of *Dtext* that are to be enclosed to a single *Mtext* object using *Txt2mtxt*.

TCOUNT

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express></i> <i>Text></i> <i>Automatic Text Numbering</i>	TCOUNT

Use *Tcount* to add a numeric prefix or suffix to existing text. *Tcount* does not create new, sequentially numbered text. You can, however, overwrite existing text with user-specified sequential numbers.

```
Command: tcounT
Select objects: PICK
Select objects: PICK
Select objects: PICK
Select objects: PICK
Select objects: Enter
Sort selected objects by [X/Y/Select-order] <Select-order>: Enter
Specify starting number and increment (Start,increment) <1,1>: 100,2
Placement of numbers in text [Overwrite/Prefix/Suffix/Find&replace..] <Prefix>: Enter
4 objects modified.
```

Select-order

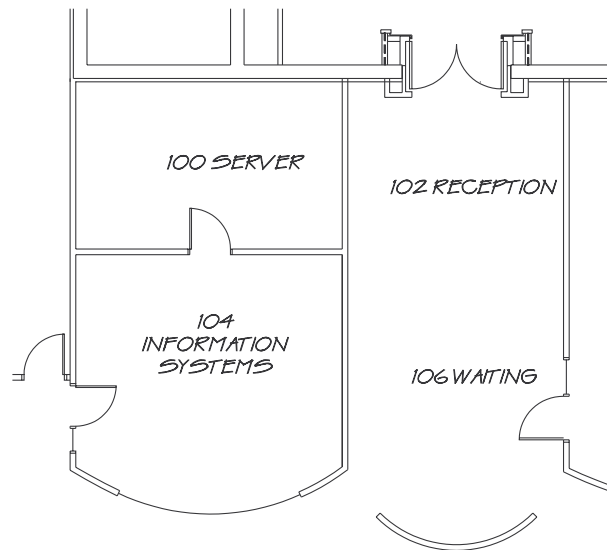
When adding a prefix or suffix to existing text, you can select the text objects in the order you want the numbers applied, then accept the default *Select-order* option. Numbers are applied in the order that objects were selected.

X/Y

If you are numbering text objects in rows or columns, you can use the *X/Y* methods. Using the sort method of *X*, the numeric sequencing goes from left to right. Use the *Y* sort method to automate the sequence from top to bottom (actually, negative *Y* direction).

Starting number and increment

Accept the “starting number and increment” of 1,1 if you want to begin with number 1 and increase numbering by 1 with each text object. You can instead specify that sequencing begin at any other number and specify another increment for counting. Leading zeros are ignored. Figure 47-42 displays four existing text objects after a numerical prefix has been added to each text object using the command sequence shown previously. Note the values entered for *Start* and *increment* and the resulting numbers.

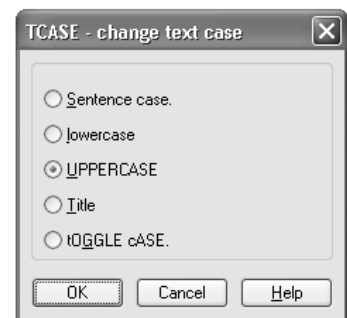
Figure 47-42**Find&replace**

The *Find&replace* feature operates in the same manner as other similar routines in which you are prompted to enter a search string and all instances are replaced with the specified method of numbering.

TCASE

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express></i> <i>Text></i> <i>Change Text Case</i>	TCASE

Tcase provides a utility for changing the case (upper case, lower case, etc.) of existing text objects. When you select existing text for this command, the *TCASE – change text case* dialog box appears (Fig. 47-43). Here you can choose from five different text case formatting options (including the current case). The feature selected in the dialog box will apply to all text selected. The following options are available.

Figure 47-43**Sentence case**

Use this to capitalize only the first word in a multi-word text string.

lowercase

This option changes all selected text to lower case.

UPPERCASE

Use this option to change all the selected text to upper case.

Title

This Option Capitalizes The First Letter Of Each Word In The Selected Text String As Shown Here.

tOGGLE cASE.

This feature reverses the current case of each letter of the selected text. This option is useful if the Caps Lock key on your keyboard is on and you enter lines of text before noticing.

Express Standard Tools Commands

The Standard Express toolbar contains a number of commands useful principally for modifying objects. A few commands are included for drawing and for other utilities. If you use pull-down menus, these commands are spread through the *Selection Tools*, *Modify*, *Draw*, and *Tools* sections of the *Express* pull-down menu.

MSTRETCH



Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Modify> Multiple Entity Stretch</i>	MSTRETCH

The typical *Stretch* command requires that you select objects to stretch with one *Crossing Window* or *Window Polygon*. Any objects that are not within the selection window must be stretched with another use of the command. In contrast, *Mstretch* allows you to create multiple selection windows before stretching the objects:

Command: **mstretch**

Define crossing windows or crossing polygons...

Options: Crossing Polygon or Crossing first point

Specify an option [CP/C] <Crossing first point>: **PICK**

Specify other corner: **PICK**

Options: Crossing Polygon, Crossing first point or Undo

Specify an option [CP/C/Undo] <Crossing first point>: **PICK**

Specify other corner: **PICK**

Options: Crossing Polygon, Crossing first point or Undo

Specify an option [CP/C/Undo] <Crossing first point>: **Enter**

Done defining windows for stretch...

Specify an option [Remove objects] <Base point>: **PICK**

Second base point: **PICK**

Command:

You can use either a crossing window (C option) or a crossing polygon (CP option) to select the object(s) to stretch.

MOCORO



Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Modify> Move Copy Rotate</i>	MOCORO

Mocoro allows you to perform a *Move*, *Copy*, *Rotate*, or *Scale*, or any combination of these operations in one command. This utility is especially useful for those who set *GRIPS* to 0 (off), but occasionally need these capabilities without using Grips:

Command: **mocoro**
 Select objects: **PICK**
 Select objects: **Enter**
 Base point: **PICK**
 [Move/Copy/Rotate/Scale/Base /Undo]<eXit>: **m**
 Second point of displacement: **PICK**
 [Move/Copy/Rotate/Scale/Base /Undo]<eXit>: **r**
 Second point or rotation angle: **PICK**
 [Move/Copy/Rotate/Scale/Baset/Undo]<eXit>: **c**
 Second point of displacement/Undo]<eXit>: **PICK**
 Second point of displacement/Undo]<eXit>: **Enter**
 [Move/Copy/Rotate/Scale/Base /Undo]<eXit>: **Enter**
 Command:

SUPERHATCH



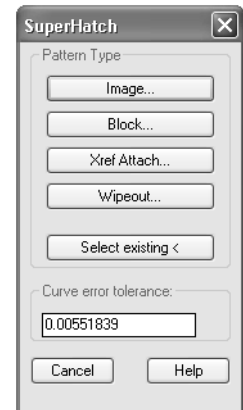
Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Draw> Super Hatch...</i>	SUPERHATCH

The *Superhatch* command allows you to hatch an enclosed area, similar to the *Hatch* command. However, with *Superhatch* you can select an image (.TIF, .TGA, .BMP, etc.), *Block*, *Xref*, or *Wipeout* object to use as the hatch pattern. Using the *Superhatch* command produces the *SuperHatch* dialog box (Fig. 47-44).

After you select the appropriate choice to use as a hatch pattern object (*Image*, *Block*, *Xref*, or *Wipeout*), another dialog box appears for you to select the desired file or object, followed by a series of prompts. For example, after selecting an image file as the hatch pattern object, the following prompts appear:

Command: **superhatch**
 Insertion point <0,0>: **PICK**
 Base image size: Width: 1.000000, Height: 0.920792, Inches
 Specify scale factor <1>: **Enter** or (value)
 Command:
 Is the placement of this IMAGE acceptable? [Yes/No] <Yes>: **Enter**
 Selecting visible objects for boundary detection...Done.
 Specify an option [Advanced options] <Internal point>: **PICK**
 Specify an option [Advanced options] <Internal point>: **Enter**
 Preparing hatch objects for display...
 Done.
 Use TFRAMES to toggle object frames on and off.
 Command:

Figure 47-44



Typically you would prefer that the hatch object is smaller than the internal area you select for hatching. In that case, the hatch object is replicated similar to any other hatch pattern.

TFRAMES

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
...	TFRAMES

Tframes is short for “toggle frames.” Type this command or use the button shown to toggle a “frame” or outline of the hatched area (hatched with *Superhatch*) on or off. Keep in mind that the object(s) used to define the hatch boundary may obscure the *Superhatch* frame, so when *Tframes* is toggled on, it may appear as if there are two outlines. This feature can also be used to toggle the visibility of the frame of an inserted raster image or masked text.

SHOWURLS

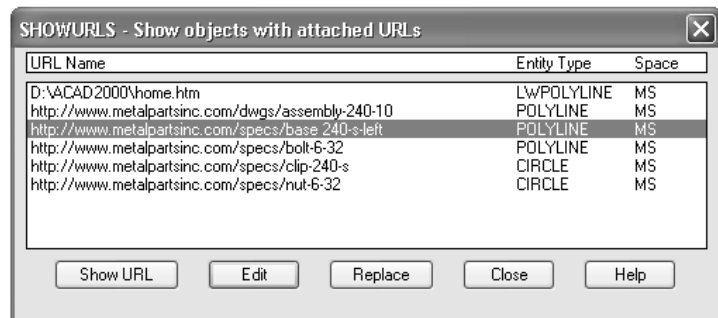
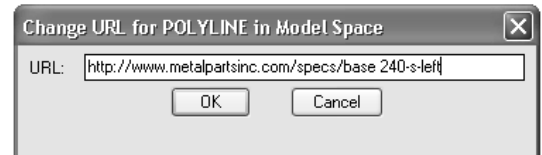
Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Tools</i> > <i>Show URLs...</i>	SHOWURLS

The *Showurls* utility is helpful for displaying and editing URL links (created with the *Hyperlink* or *Attachurl* command) in the current drawing. *Showurls* produces the *SHOWURLS* dialog box that lists all the URLs in the drawing (Fig. 47-45).

You can easily locate an object with an attached URL by selecting the *Show URL* button. AutoCAD dismisses the dialog box temporarily and highlights the drawing object, then displays the dialog box again automatically after a second or two.

Use the *Edit* option to produce the dialog box shown in Figure 47-46. Here you can edit the contents of any selected URL. Use this option to change URLs one at a time.

Selecting the *Replace* option produces the *Replace URL Text* dialog box where you specify a text string to *Find What* and *Replace What*. This option globally searches and replaces specified text strings contained in URLs in the drawing. This is helpful when a series of URLs in a drawing reference a web site that has undergone a name change.

Figure 47-45**Figure 47-46**

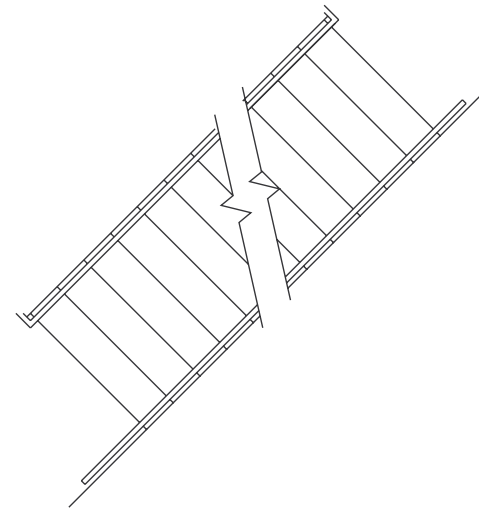
BREAKLINE



Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
Express> Draw> Break-line Symbol	BREAKLINE

This command adds a line to a drawing with a break symbol at the center of the line or other location you specify. Break lines are used for many applications where you do not need to draw the entire object or length of a series of lines. For example, Figure 47-47 shows two break lines added to a drawing of a stair case, each break line was created using the values given in the following prompts (*Trim* was used to trim lines between the two break lines).

Figure 47-47



Command: **breakline**
 Block= BRKLINE.DWG, Size= 8'-0", Extension= 3'-0"
 Specify first point for breakline or [Block/Size/Extension]: **PICK**
 Specify second point for breakline: **PICK**
 Specify location for break symbol <Midpoint>: **PICK**

Use the *Size* option to specify the width of the break symbol along the line. The *Extension* option specifies the distance beyond the points you pick as the "first point" and "second point." Typically, you would use an *Osnap* option to select these two points and *Breakline* draws a line between the selected points. Next, you can specify a point on the line to place the break symbol or use the default *Midpoint* option. If you pick a point, the *Nearest* and *Midpoint Osnap* options are turned on.

FS

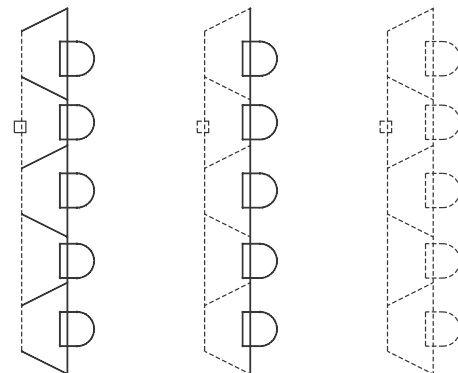


Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
Express> Selection Tools> Fast Select	FS

The *Fast Select* command is actually a selection method used to select multiple objects with one pick rather than having to select individual objects. *Fast Select* allows you to select a group of objects that are connected (physically touch) by selecting only one of the objects.

Figure 47-48

For example, you can use *Fast Select* to select an entire group of tables or both tables and chairs. Figure 47-48 (left group) shows how selecting a line normally (without *Fast Select*) highlights only that line. With *Fast Select*, selecting the same line highlights the line and all other objects touching the line (Fig 47-48, center).



FAST SELECT
FSMODE = 0

FAST SELECT
FSMODE = 1

```

Command: 'fs
Initializing...
Use 'FSMODE to control chain selection.
FSMODE = OFF
Select touching object: PICK
7 object(s) found.
Exiting Fastsel

```

The *FSMODE* variable determines the extent to which the selection is made. With the *FSMODE* variable set to *Off*, only those items touching the selected object are selected, as shown in Figure 47-48, center. With the *FSMODE* variable set to *On*, a “chain” selection is in effect whereby all connected or touching objects are selected (Figure 47-48, right).

The *Fast Select* method can be initialized transparently at any “Select objects” prompt by preceding the entry with an apostrophe, as in *'fs*. When launched directly from the *Express, Selection Tools* menu, the command line indicates the *FSMODE* setting.

NOTE: Using *Fast Select* may result in different selection sets based on which object is the primary object selected, especially when the *FSMODE* variable is set to *Off*.

COPYM



Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express></i> <i>Modify></i> <i>Multiple Copy</i>	<i>COPYM</i>

The *Multiple Copy* command allows you to make multiple copies of selected objects and place the copies using the *Repeat*, *Divide*, and *Measure*, and *Array* options. Using this command and its options is similar to using Grips with the Copy option where the copies are placed using an auxiliary grid (see Chapter 23, Grip Editing, for more information).

Using the *Multiple Copy* express tool with the default option (selecting a “Second point”) is similar to using the normal *Copy* command with the *Multiple* option.

```

Command: copym
Select objects: PICK
Select objects: Enter
Base point: PICK
Second point or [Repeat (last)/Divide/Measure/Array (dynamic)/Undo] <exit>: PICK
Second point or [Repeat (last)/Divide/Measure/Array (dynamic)/Undo] <exit>: PICK
Second point or [Repeat (last)/Divide/Measure/Array (dynamic)/Undo] <exit>: Enter

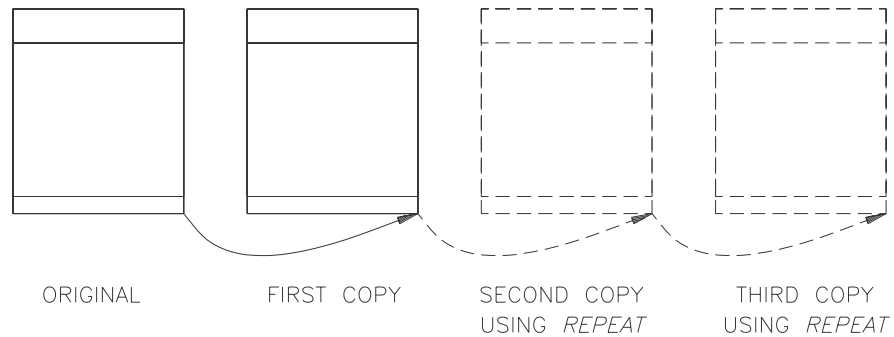
```

Repeat

Once you place the first copy, you can use *Repeat* to create another copy using the same displacement (XY spacing) for the next copy. This is similar to using Grips to make a copy, then holding down the Shift key to set up an auxiliary grid using the same XY spacing.

However, with *Multiple Copy*, simply type R for the

new copy to be made. In Figure 47-49, after making the first copy of the chair, the *Repeat* option was used to make the second copy rather than picking a new "Second point."

Figure 47-49**Divide**

The *Divide* option creates the number of copies you specify along an imaginary line. The first point of the line is the previously selected base point, then you are prompted for the "division ending point" and the number of copies.

Second point or [Repeat (last)/Divide/Measure/Array (dynamic)/Undo] <exit>: **a**
 Select division ending point: **PICK**
 Number of copies: (Enter value)

Measure

The *Measure* option is similar to the *Divide* option, except you are prompted for the "Distance between copies," rather than the number of copies.

Array

Use this feature to set up a rectangular array of copies. You must place each copy individually. The copies can be placed on a grid determined by a point you pick in response to the "Pick a corner point to establish COLUMN and ROW distances" prompt (the first corner is the previously specified base point). This option is also similar to using Grips with the Copy option and the Shift key to set up an auxiliary grid.

Second point or [Repeat (last)/Divide/Measure/Array (dynamic)/Undo] <exit>: **a**
 Pick (dynamic)/Measure/Divide <Pick>: **Enter**
 Specify angle <0>: **Enter**
 Pick a corner point to establish COLUMN and ROW distances: **PICK**
 Pick location for array element or <enter> when done: **PICK** (location for new copy)
 Pick location for array element or <enter> when done: **PICK** (location for new copy)
 Pick location for array element or <enter> when done: **Enter**

Express Layout Tools Commands

This collection of commands is intended for working with layouts. Some of these commands may be particularly helpful when working with older drawings that used paper space (created before multiple layouts were introduced). These commands all require that you are in a *Layout* tab prior to initiating the command.

CHSPACE

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express> Layout tools> Change Space</i>	CHSPACE

The *Chspace* command allows you to transfer selected objects from one space (Paper or Model) to another. You can transfer objects from a Model space viewport to the current Layout or from the current Layout into a specified Model space viewport. For example, if you want to swap objects between Model space and Paper space, such as a sheet border, revision block, or a drawing legend, use this tool. You must invoke this command when in a Layout.

Command: **chspace**

Select objects: **PICK**

Specify opposite corner: **PICK**

Select objects: **Enter**

Set the TARGET viewport active and press ENTER to continue. **Enter**

117 object(s) changed from PAPER space to MODEL space.

Objects were scaled by a factor of 29'-0 7/16" to maintain visual appearance.

When transferred, the objects selected are automatically scaled to appear the same in the destination space. For this reason, when transferring from a Model space viewport to the Layout, it is suggested that you zoom as necessary before invoking *Chspace*.

ALIGNSPACE

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express> Layout tools> Align Space</i>	ALIGNSPACE

Occasionally you may want to change the display of Model space objects visible in a viewport (in a Layout) so they align with objects in the current layout. This command does not move the objects, but rather pans the drawing (Model space) to the desired alignment point.

Command: **alignspace**

FIRST alignment point in MODEL space: **PICK**

SECOND point in MODEL space or <Return> for none: **PICK or Enter**

Alignment point in PAPER space: **PICK**

Activate the desired viewport to align and press ENTER to continue. **Enter**

Paper space = Model space

1" = 3'-6 5/8"

Current zoom factor = 0"xp

After picking the necessary points, *Alignspace* essentially pans the display of the drawing in the viewport so that the points specified in Model space are aligned with the points picked in the layout. Although the drawing is panned, the zoom factor of the viewport remains unchanged.

VPSYNC

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express></i> <i>Layout tools></i> <i>Synchronize Viewports</i>	VPSYNC

This express tool scales (synchronizes) the display of objects in multiple viewports to the scale and display of a selected “master” viewport. *Vpsync* automatically changes the zoom ratio of the viewports to match that of the master viewport, then displays an appropriate area of the drawing (pans the drawing) in each viewport based on the area displayed in the master viewport. Although you are not specifically prompted, you must select the “master” viewport (the viewport object in the layout) in response to the first “Select objects” prompt.

Command: **vpsync**

Select objects: **PICK** (Select the master viewport)

Select viewports to be aligned to master viewport.

Select objects: **PICK**

Select objects: **PICK**

Select objects: **PICK**

Select objects: **Enter**

Paper space = Model space

1" = 29'-7 3/64"

Current zoom factor = 0"xp

Paper space = Model space

1" = 29'-7 3/64"

Current zoom factor = 0"xp

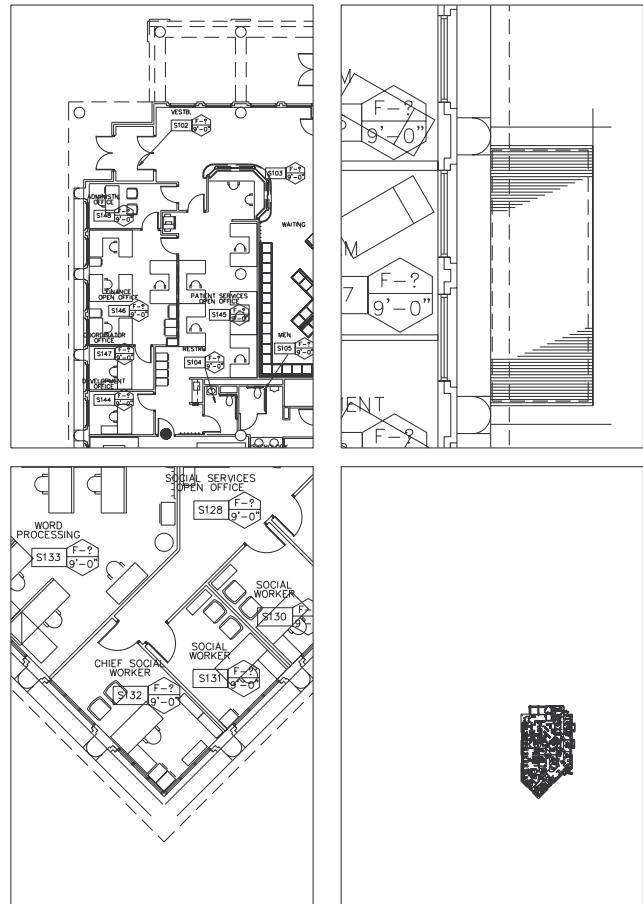
Paper space = Model space

1" = 29'-7 3/64"

Current zoom factor = 0"xp

For example in Figure 47-50, each of four viewports displays the model geometry at a different scale (zoom ratio). Use *Vpsync* to scale the display of the geometry in each viewport to that of the upper-left viewport.

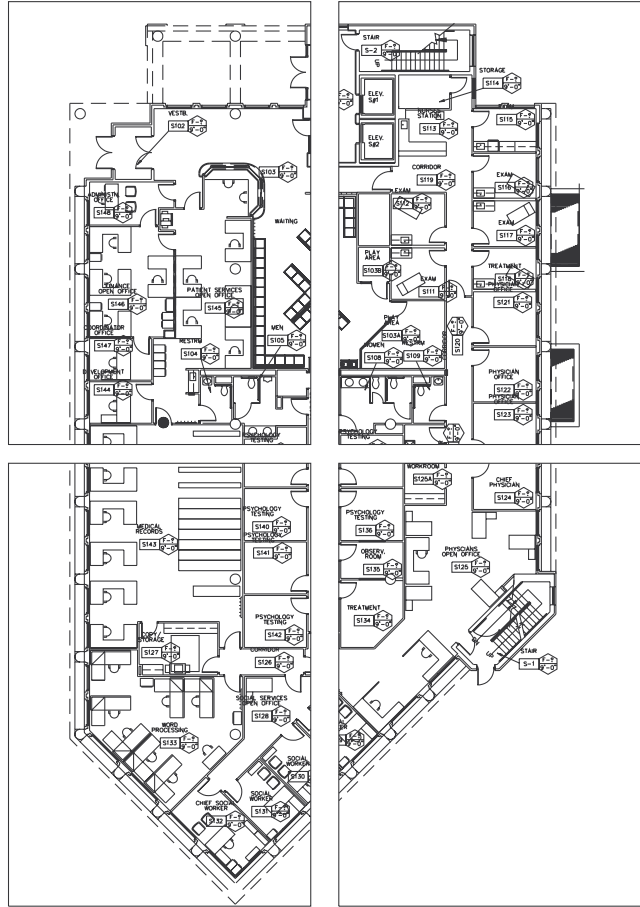
Figure 47-50



First, select the upper-left viewport as the “master” viewport. Next, select the remaining three viewports. Figure 47-51 illustrates how the display (zoom scale and area of the drawing) in those viewports was synchronized with the master viewport.

Note: Do not press Enter after selecting the master viewport as the command automatically moves on to the next prompt for the viewports to be aligned.

Figure 47-51



VPSCALE

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express></i> <i>Layout tools></i> <i>List Viewport Scale</i>	VPSCALE

Vpscale lists the zoom ratio of Paper space to Model space for a particular viewport. This command is helpful when you have zoomed the drawing in a viewport to display the desired area, but you want to ensure you plot the drawing (technically, the viewport scale) to a standard scale. Use *Vpscale* to return the scale of the viewport in ratio format of Paper space to Model space units (PS:MS), then set the viewport scale (using the drop-down list in the *Viewports* toolbar) to standard scale close to the reported ratio.

Command: **vpscale**
 Initializing...
 Select edge of viewport.
 Select objects: **PICK**
 Command: **vpscale**
 PS:MS == 1:353.280001
 Viewport Scale: 9/256" = 1'-0"

In the above example, the ‘Viewport Scale’ value is close to 10/250 or 1/25. The closest common architectural scale would be 1/32"=1'. The closest common engineering scale may be 1:250 or 1:500.

LAYOUTMERGE

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express> Layout tools> Merge Layout</i>	LAYOUTMERGE

This command merges the contents of multiple layouts into a single layout. The entire contents of the selected layouts are “moved,” including the viewport objects, the drawing (model) area displayed in the viewports, and any paper space geometry in the layouts. Specifications for device drivers, sheet sizes, and plotting parameters are not copied to the new layout.

Because this command “sweeps” all layout contents from the selected layouts to the target layout, it should be used with caution. It is recommended to save the drawing to another name before using this command. Also, you should create a new layout which will eventually be used as the destination layout. (No spaces are allowed in the destination layout name.)

Begin the *Layoutmerge* command to produce the *LAYOUTMERGE* dialog box (Fig. 47-52). Here you should select all the layout(s) to be merged to the destination layout. Notice that the text above the layout listing states to “Select layout(s) to merge.” You should use the Shift key or Ctrl key to select more than one layout to merge as shown in Figure 47-52, then select *OK*.

The next dialog box that appears is also named *LAYOUTMERGE* (not shown), but this dialog box prompts you to “Specify (a) destination layout.” Select the layout you created as the layout you want to merge to, then select *OK*. The following prompt appears.

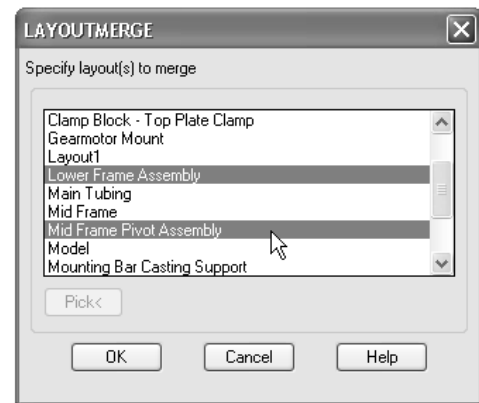
Command: **layoutmerge**
Delete unused layouts? <Y>:

Answering Y causes the selected (merged) layouts to be deleted. If you elect not to “delete unused layouts,” the layouts and their contents will be retained. After entering your response, AutoCAD continues with the merging process. At the completion of the procedure, the layout selected to be the merge layout is set to be the active layout.

Restoring cached viewports - Regenerating layout.
Regenerating layout.
Regenerating model - caching viewports.
Regenerating model.
2 layouts merged.

After using this command, you should use *Page Setup* to specify new plotting or printing parameters for the new layout since the contents of the new layout will most likely require new device or sheet size settings.

Figure 47-52



Express Dimension Commands

Commands in this section are found in the *Dimension* group of the *Express* pull-down menu. The first three, *Qlattach*, *Qldetachset*, and *Qlattacheset*, are in a separate menu of *Leader Tools*. The remaining items are listed in the *Dimension* group.

QLATTACH

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Dimension> Leader Tools> Attach Leader to...</i>	QLATTACH

Use *Qlattach* to attach an existing leader to an annotation (*Mtext*, *Block*, or *Tolerance* objects). *Qleader* automatically changes the endpoint of the leader and/or moves the annotation to align and attach to the top of the top line of text. If the annotation object is later moved, the leader stays attached.

Command: **qlattach**
 Select Leader: **PICK**
 Select Annotation: **PICK**
 Command:

QLDETACHSET

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Tools Dimension> Leader Tools> Detach Leaders from Annotation</i>	QLDETACHSET

The *Qldetachset* command can be used to “break” the connection between a leader and the related text. When *Qldetachset* is used, the leader and related text become independent objects and can be modified separately. In addition, the leader line no longer contains a “hook” (short horizontal segment) aligned with the text object. *Qldetachset* operates for text/leader combinations created with the *Leader*, *Qleader*, or *Qlattach* commands.

QLATTACHSET

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Tools Dimension> Leader Tools> Global Attach Leader to Annotation</i>	QLATTACHSET

This command is intended to attach previously created leaders to existing *Mtext*, *Tolerance*, or *Block* objects. *Qldetachset* is similar to the *Qlattach* command, with the exception that multiple annotation or leader objects can be selected in response to the “Select objects:” prompt.

DIMEX

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Dimension> Dimstyle Export...</i>	DIMEX

This is a very useful utility for exporting dimension styles from the current drawing to a file for importation into another drawing with *Dimim* (dimension import). *Dimex* produces the *Dimension Style Export* dialog box (Fig. 47-53). Select a dimension style to export, then select OK. The dimension style is saved to a file with a .DIM file extension.

Figure 47-53



DIMIM

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Dimension> Dimstyle Import...</i>	DIMIM

Use *Dimim* to import previously exported dimension styles (see *Dimex*). *Dimim* invokes the *Dimension Style Import* dialog box (Fig. 47-54). Use *Browse...* to locate the desired *.DIM file.

Figure 47-54



DIMREASSOC

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express> Dimension> Reset Dim Text Value</i>	DIMREASSOC

This feature does not turn nonassociative dimensions into associative dimensions as the name might imply. Instead, this command is intended to reset the true associative value of an associative dimension whose value has been modified to something other than the original value. For example, assume *Ddedit* was used to change a dimension value from the actual AutoCAD-measured value to other text that was entered (in place of the "<>" symbol). You could then use *Dimreassoc* to reclaim the true AutoCAD-measured value.

- Command: **dimreassoc**
- Select objects: **PICK**
- Select objects: **PICK**
- Select objects: **Enter**
- 2 objects modified.

To learn more about editing dimensions, refer to Chapter 28, Dimensioning.

Express Selection Tools Commands

The commands in this group include two enhanced routines for selecting objects. The *Fast Select* utility is found among the Express Tools *Standard* toolbar group (discussed previously). The *Getsel* feature is command line driven and is similar in operation to Quick Select (*Qselect*).

GETSEL

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Selection Tools> Get Selection Set</i>	GETSEL

Getsel allows you to PICK specific objects by object type or by layer and use them as the current selection set. You can select the desired layer (or all layers) or the desired object type (or all object types). This utility is a fast alternative to using the *Filter* command to locate and find specific object types. *Getsel* issues the following prompt:

```
Command: getsel
Select an object on the Source layer <*>: PICK
Select an object of the Type you want <*>: PICK
Collecting all LWPOLYLINE objects on layer GEOMETRY...
7 objects have been placed in the active selection set.
Command:
```

Express Modify Tools Commands

The *Express Modify* group has a variety of tools that either automate common procedures or enable you to more efficiently edit your drawing. Only the *Exoffset* command is explained here, however, since the other commands in the *Modify* group also appear in the *ET: Standard* toolbar or other locations previously discussed.

EXOFFSET

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Modify> Extended Offset</i>	EXOFFSET

The *Extended Offset* command provides a collection of offset-related features and could, therefore, be considered an “extension” of the *Offset* command. For a review of the *Offset* command, refer to Chapter 9, Modify Commands I.

The *Extended Offset* feature begins with prompt similar to the *Offset* command, but also includes a line of information listing the current settings and access to other options.

```
Command: exoffset
Settings: Distance = Through, Layer = SOURCE, Gaptype = Normal
Specify offset distance or [Through] <Through>: (value) or Enter
Select object(s) to offset or [Options/Undo]:
```

Changes to the *Distance*, *Layer*, and *Gaptype* settings are done using the *Options* selection. When you type *O* for *Options* the following prompt appears.

Specify an option to set [Distance/Layer/Gaptype]:

Distance

The *Distance* option enables you to specify the distance for the offset. This value can also be specified at the opening prompt of the command.

Layer

Type *L* for the *Layer* option and you can choose between the *Source* or *Current* layer as the target location for the offset objects. By default, AutoCAD places the offset objects on the same layer as the selected (*Source*) object.

Gaptype

This option is useful when offsetting *Plines*. *Gaptype* options are *Normal*, *Fillet*, and *Chamfer*. Your selection sets the *OFFSETGAPTYPE* system variable (refer to Appendix A, System Variables, for more information).

Specify an option to set [Distance/Layer/Gaptype]: **g**

Specify offset behavior for plines [Normal/Fillet/Chamfer] <Normal>:

After setting the desired values for the options, press Enter to return to the initial prompt and to select the object to be offset. Despite the indication of the prompt, "Select object(s)," you can select only one object at a time.

Select object(s) to offset or [Options/Undo]: **PICK**

1 object(s) found.

Shift+Pick for multiple; Ctrl+Pick to erase source object.

Side to offset or [Multiple/Options/Undo]:

Multiple, Shift+Pick, Ctrl+Pick

Hold down the Shift key when picking the side to offset to make successive offsets using the original distance for each offset. The same action can also be accomplished by typing *M* for the *Multiple* option. If you hold the Ctrl key down when you pick the side to offset, the highlighted source object is erased after being offset. Type *O* for the previously described collection of options (*Distance*, *Layer*, and *Gaptype*).

Express File Tools Commands

This collection of file-related tools enhances your productivity not only for .DWG files, but also for .BAK, .PLT, and raster files. Also included is a tool that enhances the Drawing Properties utility for working with large projects or implementing consistency among your project files.

MOVEBAK

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express File tools> Move Backup Files</i>	MOVEBAK

Use this command to redirect the .BAK files to a folder you specify (overriding the folder specified in the *Files* tab of the *Options* dialog box). When entering the name of the new folder where the .BAK files are to be saved, be sure to enter the full path name including the drive letter, such as "C:\backup."

Command: **movebak**

New value for MOVEBAK, or . for none <>: (Enter new location for .bak files)

This setting is valid only for the current AutoCAD session. When you save any drawing during that session, you will see the following notice:

Moving BAK file to 'c:\backup\PROJECT 4.bak'.

When you exit AutoCAD, the location specified for *Movebak* is "released." .BAK files in future AutoCAD sessions are saved to the folder in which their related .DWG file is saved as specified in the *Files* tab of the *Options* dialog box. For more information on backup files and settings of the *Options* dialog box, refer to Chapter 43, Miscellaneous Commands and Features.

PLT2DWG

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express File tools></i> <i>Convert PLT to DWG</i>	PLT2DWG

Use this command to convert a .PLT file to a .DWG file. .PLT files are created by configuring an HPGL device, then plotting using the *Plot to File* option in the *Plot* dialog box. Only files created using an HPGL device driver are supported, whereas files created using an HPGL/2 driver are not supported. The Draftpro DXL and Draftpro EXL are also not supported.

When you launch the *Plt2dwg* command, the *Enter the plot file* dialog box (not shown) opens, from which you can choose the .PLT file to convert. Because .PLT files contain information primarily for pen strokes and pen settings, the resulting converted drawing contains only 2D geometry. Some layering and color information is included.

IMAGEEDIT

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express File tools></i> <i>Edit Image</i>	IMAGEEDIT

This command is used to edit an existing raster image inserted in your drawing. The command launches a system-supported image editing program such as Microsoft Paintbrush.

Command: **imageedit**

Use the IMAGEAPP command to specify a non-system-default editor.

Select objects: **PICK**

Once you select the image to be edited, the *IMAGE EDIT* dialog box opens giving the name and location of the inserted image file. Choose *Open* to launch the image editing program. You can also navigate and select another image file you may want to use as a replacement.

After editing and saving the image and closing the image editing application, the newly edited image appears in the drawing. When saving the edited image file, it is suggested to enter a unique name for the edited image, otherwise the existing file is overwritten.

If the desired image editing application does not automatically open, type *IMAGEAPP* at the prompt line and enter the full path, filename, and extension of the application you want AutoCAD to launch as your default image-editing application.

PROPULATE

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express></i> <i>File tools></i> <i>Update Drawing</i> <i>Property Data</i>	PROPULATE

For projects involving many drawings, the *Update Drawing Property Data* command may be useful for maintaining the same project information for all drawings in the project. Using this command, drawing properties information can be populated to multiple drawings using a properties populate template (.PRP). Using this routine avoids having to enter the drawing properties information for each drawing using the *Dwgprops* command. A drawing's properties appear in the *Properties* dialog box that appears if you use the *Dwgprops* command in AutoCAD or if you use Windows Explorer or My Computer to list a drawing file's properties. For more information on the *Dwgprops* command, refer to Chapter 2, Working With Files.

Before populating drawings with the specific information you want, you must have an existing .PRP file containing the information you want to populate. AutoCAD provides a default file, TEMPLATE.PRP, that can be edited and saved under a new name. You can edit the file using the *Propulate* command or using an external text editor. Begin the *Propulate* command to use this series of prompts.

Command: **Propulate**

Default template file: C:\Program Files\AutoCAD 2004\express\template.prp

Enter an option [Active template/Edit template/List/Remove/Update] <Update>: **Enter**

Update Drawing Properties information from template

Enter an option [Current drawing/Other drawings] <Current>: **Enter**

Writing C:\Program Files\AutoCAD 2004\Sample\8th floor.dwg Drawing Properties Info.

Update

The update option allows you to populate drawing properties to the current drawing or directories of drawings. *Update* uses the information supplied from the active template to process the selected drawing(s). (Before using *Update*, specify the active template using the *Active Template* option.) Selecting the *Other drawings* option allows you to specify a search directory and drawing name. For example, entering "C:\projects\proj2*.dwg" causes a search for all drawings whose names start with "proj2" starting in the "C:\projects" directory.

Active template

Choose an existing .PRP drawing properties template file from the *Select Propulate Template File* dialog box. The default file of *template.prp* can be opened, then edited and saved under a new name.

List

To view the properties information for the current or other drawings (rather than using the *Drawing Properties* window), use the *List* option to produce a text screen of the properties information.

Remove

Use this option to strip the drawing properties information from the current drawing or from a directory of drawings.

Edit template

Use this feature to open the *Edit Propulate Template* dialog box (Fig. 47-55). Here you can add to or modify the fields. The template can then be used to *Update* the drawing properties information for the current drawing or multiple other drawings.

Special Insert Variables

You can use special insert commands to extract data (from the drawings that you intend to *Propulate*) and insert the information into the *Drawing Properties* dialog box, therefore listing properties that can be unique for each drawing. Three of these variables appear in the *Comments* section of Figure 47-55. The variables are explained here.

@[block.attributeTag]

This variable extracts the attribute tag value from an attributed block and inserts it in the specified field. For example, inserting @[title.drawnby] in the *Author* field causes *Propulate* to search the drawing being processed for an attributed block named "title," extract the value of its attribute tag "drawnby," and insert that value in the *Author* field of the drawings *Drawing Properties* dialog box. Therefore, if common blocks are used across a project it is relatively simple to update a large number of drawings to reflect their specific attribute values.

@[XREFLIST], @[FONTLIST], @[IMAGELIST]

These variables insert a list of the Xrefs, fonts, or images currently attached to the drawing being processed. You can use these variables only in the *Comment* field.

Two pull-down menus are available at the top of the *Edit Propulate Template* dialog box: *File* and *Template*.

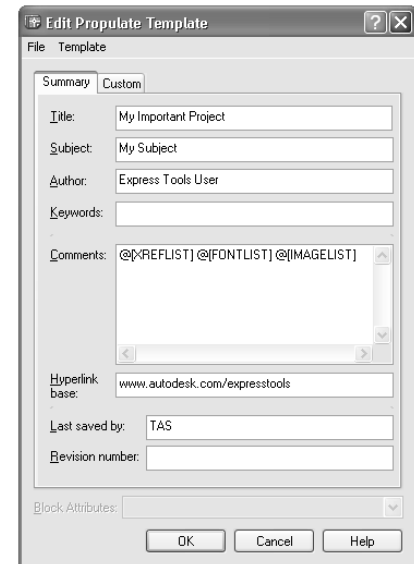
File

The *File* menu enables you to begin with a *New* template with empty fields, *Open* and existing .PRP file, *Save* the current file or use *Save As* and give the file a new name. Unfortunately, the name of the current .PRP file is not listed in the title bar of the dialog box.

Template

Use the *Template* pull-down menu to automate the procedure for including information in the *Comments* section of the template dialog box. You can request *Xref*, *Font*, *Image*, or *Attribute* information from the processed drawings to be included in the drawing properties information (see "Special Insert Variables"). If you want to use the current drawing's information (previously specified using the *Dwgprops* command), select the *Fill from Current drawing* option. The *Make active template* option sets the template currently being edited to be the default template. This option is disabled if the edited template is already the default or if it has not yet been saved to a file name.

Figure 47-55



SAVEALL

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express File tools> Save All Drawings</i>	SAVEALL

Use this feature to save all the open drawings in the current AutoCAD session, but keep the drawings open for further editing. AutoCAD invokes *Qsave* for previously named drawings, and for those drawings that have yet been named, the *Save Drawing As* dialog box appears.

CLOSEALL

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express File tools> Close All Drawings</i>	CLOSEALL

Use this feature to close all open drawings at once. For those drawings that have not yet been named, the typical "Save changes to ...?" alert dialog box appears.

QUIT

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express File tools> Quick Exit</i>	QUIT

Use this feature to exit the current AutoCAD session. If a drawing has not been named, or if you have modified a drawing since it was last saved, the typical alert dialog box regarding saving changes appears. After all drawings have been accounted for, AutoCAD closes.

REVERT

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express File tools> Revert to Original</i>	REVERT

Revert closes and reopens the current drawing. This is useful if you want to disregard any changes and reopen a drawing in its last saved condition. If you select OK from the "Abandon changes...." alert box AutoCAD simply re-opens the last-saved version of the drawing. You can also *Cancel* the process.

REDIR

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Tools> Path Substitution</i>	REDIR		

The *Redir* command allows you to substitute any or all hard coded path names for *Xrefs*, styles, shapes and images. *Redir* operates like a search and replace, so you can specify a new path, substitute a portion of a path, or even remove path references entirely. Use the "?" option to list the current paths.

Command: **redir**

Current REDIRMODE: Styles,Xrefs,Images,Rtext

Find and replace directory names

Enter old directory (use '*' for all), or ? <options>: ?

Enter file references to list <*>: **Enter**

```

Current REDIRMODE: Styles,Xrefs,Images,Rtext
TYPE      NAME      FILE
-----
STYLE     STANDARD  TXT
STYLE     30-FLOR|STANDARD  TXT
STYLE     30-FLOR|ROMANS   ROMANS
STYLE     30-FLOR|COMPLEX  COMPLEX
STYLE     30-FLOR|SIMPLEX  SIMPLEX
STYLE     30-FACIL|STANDARD  TXT
STYLE     30-FACIL|ROMANS   ROMANS
STYLE     30-FACIL|COMPLEX  COMPLEX
STYLE     30-FACIL|SIMPLEX  SIMPLEX
XREF      30-FACIL  D:\Dwg\30-facil.dwg
XREF      30-FLOR  D:\Dwg\30-flor.dwg

```

Enter old directory (use '*' for all), or ? <options>:

To use *Redir* to substitute a new path, follow the example prompt below.

```

Command: redir
Current REDIRMODE: Styles,Xrefs,Images,Rtext
Find and replace directory names
Enter old directory (use '*' for all), or ? <options>: D:\Dwg
Replace "D:\DWG" with: Q:\Dwg\Xrefs
Searching for old dir: D:\DWG
in order to replace it with: Q:\DWG\XREFS
XREF      30-FLOR  D:\Dwg\30-flor.dwg -> Q:\DWG\XREFS\30-FLOR.DWG
XREF      30-FACIL  D:\Dwg\30-facil.dwg -> Q:\DWG\XREFS\30-FACIL.DWG
0 style/shape records modified.
0 image references modified.
2 xrefs modified.
0 rtext objects modified.
Changes to some externally referenced objects may be temporary.
Command:

```

Enter the *Ridmode* command or use the *Options* in the *Redir* command to produce the *REDIRMODE* dialog box (Fig. 47-56). Here you select which objects you want to be affected by the *Redir* search and replace you specify.

Figure 47-56



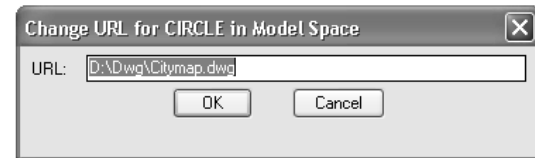
Web Tools Commands

The three commands in this group provide tools for manipulating hyperlinks in the current drawing. The *Show URLs* command is explained previously with the *ET: Standard* toolbar group.

CHURLS

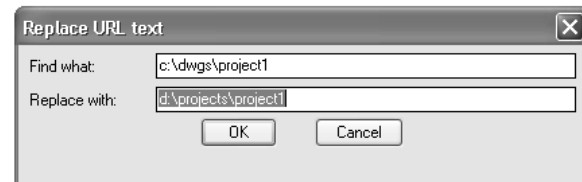
Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Web tools> Change URL</i>	<i>CHURLS</i>		

Use *Churls* to edit the hyperlink (text information) attached to an object. The *Churls* command first prompts you to select objects, then produces the *Change URL* dialog box in which you can edit the contents of the selected hyperlink (Fig. 47-57). This is essentially the same dialog box that is displayed when you select the *Edit* button of the *Showurls* utility.

Figure 47-57**REPURLS**

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Web tools> Find and Replace URLs</i>	<i>REPURLS</i>

This feature is useful for making global changes to multiple hyperlinks in a drawing, such as when an Internet address or a path changes and you must update the text information for all hyperlinks. *Repurls* allows you to change the hyperlink information for the selected objects by searching for a text string and replacing it with another. At the "Select objects:" prompt select multiple objects containing hyperlinks to search and replace. After selecting the desired objects, the *Replace URL text* dialog box opens with fields for you to enter the *Find what* and *Replace with* text strings (Fig. 47-58). For more information on using hyperlinks, refer to Chapter 19, Internet Tools.

Figure 47-58**Tools Commands**

The Tools group contains a collection of miscellaneous utilities, including the automation of creating your own shapes and linetypes to enhanced UCS manipulation.

ALIASEDIT

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express> Tools> Command Alias Editor</i>	<i>ALIASEDIT</i>

Aliasedit is a utility for changing specific command aliases from within AutoCAD, rather than using an external text editor to modify the ACAD.PGP file. For a review of the ACAD.PGP file refer to Chapter 44, Basic Customization.

Aliasedit produces the *AutoCAD Alias Editor* dialog box (Fig. 47-59). Two tabs are available: the *Command* tab for changing command aliases, and the *Shell Commands* tab for modifying shell (external to AutoCAD) commands. Both tabs operate similarly and include the following options.

Add

Select the *Add* button to open the *New Command Alias* dialog box (not shown). Create new command aliases (not previously defined) with this utility. Enter the desired one- or two-letter alias and select the AutoCAD command to which you want it to be associated, then click *OK* to add it to the list.

Remove

To remove an alias from the list, select the alias from the list, then click *Remove*. This action removes the alias only, not the AutoCAD command.

Edit

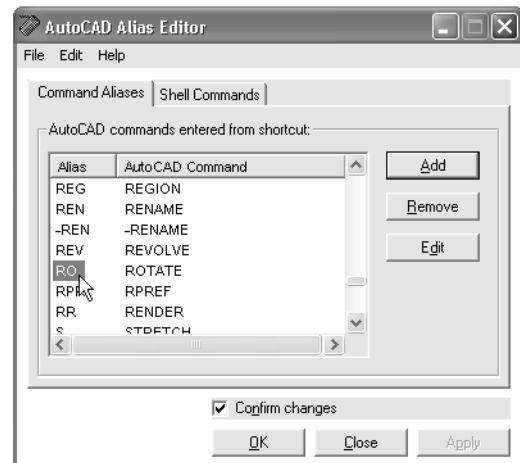
To modify an existing alias, select the alias from the list, then click the *Edit* button to open the *Edit Command Alias* dialog box (not shown). Make the desired (characters) changes to the alias, then click *OK*. If the alias already exists, the *Duplicates found* alert box appears.

Import, Export

Selecting *Import* or *Export* from the *File* pull-down menu allows you to save the current .PGP file to another name (*Export*) or to edit another .PGP file (*Import*).

When you click *OK* to complete the editing session, the aliases are saved to the appropriate .PGP file and the AutoCAD session is updated with the changes (similar to using *Reinit*). Checking the *Confirm changes* check box results in an *Overwrite....?* alert upon selecting *OK* to complete the editing session.

Figure 47-59



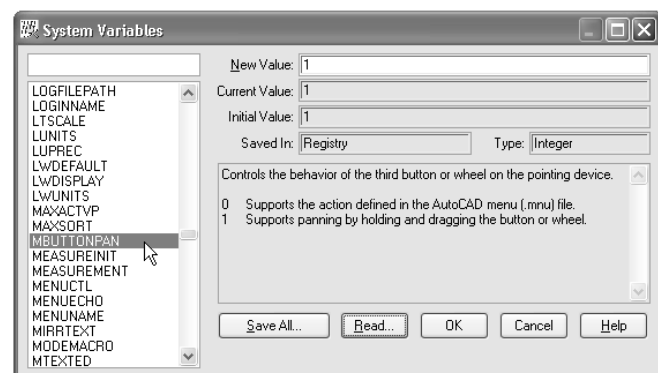
SYSVDLG

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express></i> <i>Tools></i> <i>System Variable Editor</i>	SYSVDLG

This utility is extremely useful for learning about system variables and keeping track of and changing the variable's settings. Complete information for each variable is given, such as a description of each variable, the initial and current settings, and where the variable is saved (registry or drawing file). In addition, you can use this utility to *Save* the current settings for all system variables to a .SVF file, then *Read* the file into another drawing, which resets that drawing's variables to the settings specified in the .SVF file.

SYSVDLG produces the *System Variables* dialog box (Fig. 47-60). The edit field above the list box

Figure 47-60



allows you to enter a variable name or filter the list using a wildcard. For example, entering DIM* in the edit field filters the list to display only system variables that start with DIM. To change the setting for a particular variable, first select the variable from the list. For all variables that are not read-only, you can enter a value or string in the *New Value* edit field at the top of the dialog box. You can specify edits for several variables in this way, then select OK to apply all the changes. No changes are committed until the dialog is closed with the OK button.

Save All

The *Save All* button produces a *Save As* dialog box in which you can save all or selected current variable values to an .SVF (system variable) file or an .SCR (script) file. Use *Read* to import an .SVF file. .SCR files can be read into AutoCAD by using the *Script* command. If only one system variable is selected and the list is not filtered, all the system variables are saved. If more than one system variable is selected, only the highlighted variables are saved.

Read

The *Read* button produces the *Open* dialog box from which a saved .SVF file can be imported into the current AutoCAD session. Doing so resets the system variables for the AutoCAD session and for the current drawing. Be aware that this action overwrites system variable settings in the drawing you are currently editing.

NOTE: The DEFAULTS.SVF and the DEFAULTS.SCR files are provided that contain the default system variable settings (as in a new installation of AutoCAD). The files are located in the Express directory along with the System Variable Editor. You can either read the DEFAULTS.SVF file using *Sysvdlg* or run the DEFAULTS.SCR file from the command line using the *Script* command to return AutoCAD to a “like new” system variable state.

MKLTYPE

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Tools> Make Linetype</i>	MKLTYPE

This linetype maker allows you to create linetypes in two ways. First, you can create linetypes from an existing pattern of lines and polylines and easily turn them into a linetype without going through the typical linetype definition steps. For example, assume you have a series of line segments that you wish to use as your linetype “template.” Use the *Mklttype* command, follow the prompts shown below to select the existing line segments, and AutoCAD automatically creates and loads the new linetype.

Command: **mklttype**

(The *Select File* dialog box appears. Enter the desired name for an .LIN file to store the new linetype.)

Enter linetype name: **longdashes**

Enter linetype description: **Enter**

Specify starting point for line definition: **PICK** (first point to define segment length)

Specify ending point for line definition: **PICK** (second point)

Select objects: **PICK**

Select objects: **PICK**

Select objects: **PICK**

Select objects: **Enter**

Linetype “LONGDASHES” created and loaded.

Command:

Secondly, you can use *Mktype* to define the linetype using the typical linetype definition method. You can even include shape or text objects to create a complex linetype, as shown in the following series of prompts.

```

Command: mktype
Enter linetype name: fence
Enter linetype description: a,1.5,-0.25,["x",standard,s=0.2,r=0.0,x=0.05,y=-0.1],
-0.4,1.5
Specify starting point for line definition: PICK (first point to define segment length)
Specify ending point for line definition: PICK (second point)
Select objects: PICK (to define object type)
Select objects: Enter
Linetype "FENCE" created and loaded.
Command:

```

Refer to Chapter 44, "Creating Custom Linetypes," for information on linetype definitions.

MKSHAPE

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Tools> Make Shape</i>	MKSHAPE

Use this command to create shape files "on the fly" without having to go through the typical steps of shape definition. First, create some geometry composed of AutoCAD objects to use as the "template" shape. Next, use the *Mkshape* command and follow the prompts shown below to create the new shape file.

```

Command: mkshape
(The Select File dialog box appears. Enter the desired name for an .SHX file to store the new shape.)
Enter the name of the shape: square
Enter resolution <128>: Enter or (value)
Specify insertion base point: PICK
Select objects: PICK
Select objects: PICK
Select objects: PICK
Select objects: PICK
Select objects: Enter
Determining geometry extents...Done.
Building coord lists...Done.
Formating coords...|Done.
Writing new shape...Done.
Compiling shape/font description file
Compilation successful. Output file D:\A 2004 Instructor\Dwg\Ch47\newshapes.shx
contains 135 bytes.
Shape "SQUARE" created.
Use the SHAPE command to place shapes in your drawing.
Command:

```

RTUCS

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Tools> Real-Time UCS</i>	RTUCS

The *Real-Time UCS* command allows you to manipulate the orientation of the User Coordinate System (UCS) in real time. The ability to easily manipulate the position and orientation of the UCS is very important for creating and editing complex 3D drawings. For more information, review Chapter 36, User Coordinate Systems.

This command can best be utilized in a 3D view, so prior to using the command use *3Dorbit* or other viewing command to enable you to view the UCS from a 3D viewpoint. When you begin *Real-Time UCS*, the UCS icon displays a highlighted red X-axis, a yellow Y-axis and a green Z-axis, and the following prompt is presented.

Command: **rtucs**
 Press TAB key to change axis or
 [Save/Restore/Delete/Cycle/Angle/Origin/View/World/Undo] <Drag to rotate>

NOTE: Contrary to the operation of other AutoCAD commands, do not press Enter after typing the letter for the option you wish to use. The following is a review of the function of each option.

Drag to Rotate

The default option allows you to dynamically change the orientation of the UCS by dragging your cursor (place your cursor anywhere in the drawing area, then hold down the left button and drag). In doing so, the UCS rotates about one axis, the X axis being the initial default. Use the Tab key to cycle to another axis to rotate about. As you drag your cursor, information regarding the current axis and the angle of rotation is displayed on the left side of the Status bar.

Save

Type *S* to be prompted to save the current UCS, then type a name for the current UCS. The name then appears in the *UCS* dialog box if opened (using the *Dducs* or *Ucsman* command).

Restore

If one or more UCSs have been saved by name in the current drawing using the *Save* feature of either the *UCS* command or *Real-Time UCS*, type *R* to cycle through the named UCS. You cannot select the desired name from a list, but instead typing *R* repeatedly makes each saved USC the current UCS.

Delete

Use this feature to delete a named UCS. You are prompted for the name to delete.

Cycle

Type *C* repeatedly to cycle through the six predefined positions of the UCS: *World (Top)*, *Front*, *Right*, *Back*, *Left*, and *Bottom*. The information for each UCS is displayed on the left side of the status bar as each becomes current.

Angle

By default, when you drag your cursor to rotate the UCS in real-time, rotation occurs in 15-degree increments. Use this option to modify that value or turn it off by typing *0*.

Origin

Similar to the *Origin* option of the *UCS* command, use this option to reposition the origin of the UCS. When using this feature, the orientation (X, Y, Z directions) of the UCS is retained.

View

The *View* option sets the XY plane to be parallel with the current display, with the X-axis set to horizontal. This is similar to the *View* option of the *UCS* command.

World

Returns the UCS position to the World Coordinate System position for the current drawing.

Undo

Each time you type *U* the previous *Real-Time UCS* procedure is undone.

XDATA

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Tools> Xdata Attachment</i>	<i>XDATA</i>

Extended entity data is additional information that can be attached to any AutoCAD object. Extended entity data provides an internal level of information “attached” to each individual object. This AutoCAD feature is extremely powerful for customization applications.

Xdata is an Express utility to simplify attaching extended entity data to an object. Use *Xdata* to produce the following prompt:

```
Command: xdata
Select object: PICK
Enter application name: (name)
(NAME) new application.
3Real/DIR/DISP/DIST/Hand/Int/LAyer/LONG/Pos/Real/SCale/STr/<eXit>: (option)
New xdata appended.
Command:
```

The prompt displays the areas of information that can be attached to an object: *3 Real numbers, 3D world space direction, 3D world space displacement, Distance, Database handle, 16-bit integer, Layer, 32-bit long integer, 3D world space position, Real number, Scale factor, and ASCII string.*

XDLIST

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Tools> List Object Xdata</i>	<i>XDLIST</i>

Xdlist lists extended entity data attached to a nested object. An example is listed on the next page:

Command: **xdlis**
 Select object: **PICK**
 Enter application name <*>: (name)
 * Registered Application Name: (NAME)
 * Code 1002, Starting or ending brace: {
 * Code 1000, ASCII string:
 * Code 1003, Layer name: 0
 * Code 1042, Scale factor: 0.5000
 * Code 1000, ASCII string: sample object
 * Code 1002, Starting or ending brace: }
 Object has 16347 bytes of Xdata space available.

FULLSCREEN

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Tools> Full Screen AutoCAD</i>	FULLSCREEN

Fullscreen re-sizes the AutoCAD window to fill the maximum amount of screen area by hiding the Windows title bar and the AutoCAD pull-down menu bar. *Fullscreen* also hides the Status bar. In full screen mode, you can click at the top of the window to make the pull-down menu bar appear but you cannot make the Status bar appear. Use *Fullscreen* again to return the AutoCAD display to the previous setting (normal screen mode).

EXPLAN

Pull-down Menu	COMMAND (TYPE)	ALIAS (TYPE)	Short-cut	Screen (side) Menu	Tablet Menu
<i>Express Tools> Extended Plan</i>	EXPLAN

Both the (normal) *Plan* command and the *Extended Plan* command align the screen display to be coplanar with the XY plane of the current UCS. Stated another way, both commands change the orientation of your view of the drawing so your line of sight is perpendicular to the XY plane. Remember, these commands change the view, not the orientation of the UCS. (See Chapter 35, 3D Display and Viewing, for the *Plan* command and its options.)

The difference between these commands is that with the (normal) *Plan* command, AutoCAD automatically activates *Zoom* with the *Extents* option, so all of the drawing is visible after changing the view. The *Extended Plan* command, however, does not invoke a *Zoom Extents*, but enables you to specify what objects you want to display after the view is changed.

Command: **explan**
 Initializing...
 Select objects to zoom to or press <enter> to select everything on screen...
 Select objects: **Enter**
 Enter an option [Current ucs/Ucs/World] <Current>: **Enter**
 Regenerating model.

There are two options. First, you can select the objects you want to display after the view is changed to a plan view of the current XY plane. Second, pressing Enter at the "Select objects" prompt causes the view to change, but only those objects displayed previously are displayed after the view changes—AutoCAD does not adjust the magnification of the drawing with *Zoom Extents*. Therefore, you can *Zoom* to the desired area of the drawing before using *Explan*, then press Enter at the "Select objects" prompt.

Additional Express Tools Commands, Command Line Entry Only

Commands in this section are not included in the Express toolbars or *Express* pull-down menus. All of these commands must be typed.

BLOCK?

This utility lists coordinate and other information about objects composing a *Block* definition as shown in the following example:

```
Command: block?
Block name/<Return to select>: PICK
Enter an entity type/<Return for all>: Enter
Press ESC to exit or any key to continue.

((0 . "BLOCK") (2 . "TBLOCK2") (70 . 2) (10 0.0 0.0 0.0) (-2 . <Entity name:
2b90668>))

((-1 . <Entity name: 2b90668>) (0 . "LWPOLYLINE") (5 . "65") (100 .
"AcDbEntity") (67 . 0) (8 . "0") (100 . "AcDbPolyline") (90 . 4) (70 . 0) (43 .
0.02) (38 . 0.0) (39 . 0.0) (10 -4.0 0.0) (40 . 0.02) (41 . 0.02) (42 . 0.0)
(10 -4.0 0.0) (40 . 0.02) (41 . 0.02) (42 . 0.0) (10 -4.0 1.625) (40 . 0.02)
(41 . 0.02) (42 . 0.0) (10 0.0 1.625) (40 . 0.02) (41 . 0.02) (42 . 0.0) (210
0.0 0.0 1.0))

((-1 . <Entity name: 2b90698>) (0 . "LINE") (5 . "6B") (100 . "AcDbEntity") (67
. 0) (8 . "0") (100 . "AcDbLine") (10 -4.0 0.375 0.0) (11 0.0 0.375 0.0) (210
0.0 0.0 1.0))
```

BCOUNT

The *Bcount* Express command counts and lists the number of *Block* insertions in each drawing or selection set. You can count all *Blocks* in a drawing or specify a selection window. The following example is the *Bcount* results for the example schematic drawing used in Chapter 22:

```
Command: count
Press enter to select all or...
Select objects: Enter

Counting block insertions...Block Count
-----
CAP ..... 3
RES ..... 3

Command:
```

EXTRIM

Extrim allows you to use an existing *Circle*, closed *Pline*, *Arc*, *Line*, *Ellipse*, *Image*, or text as a “trimming edge.” Even though the normal *Trim* command allows closed *Plines*, this utility automates the process by prompting only for “Side to Trim” and automatically trims all objects outside or inside the closed boundary that cross the trimming edge object. In other words, *Extrim* allows you to trim like using a “cookie cutter” when closed objects are used as trimming edges.

Command: **extrim**
 Pick a POLYLINE, LINE, CIRCLE, ARC, ELLIPSE, IMAGE or TEXT for cutting edge..
 Select objects: **PICK**
 Command:
 Pick the side to trim on: **PICK**
 Command:

MPEDIT

Mpedit is similar to *Pedit* with the additional capability to select multiple *Plines* for editing. All selected *Plines* are updated as a group. For example, you can change the *Width* of all *Plines* in a drawing globally with this feature rather than having to change one *Pline* at a time:

Command: **mpedit**
 Select objects: **PICK**
 Select objects: **PICK**
 Select objects: **PICK**
 Select objects: **PICK**
 Select objects: **Enter**
 Convert Lines and Arcs to polylines? <Yes>: **Enter**
 Enter an option [Open/Close/Width/Fit/Spline/Decurve/Ltype gen/eXit] <X>:

Although you may select only *Plines*, you will still get the “Convert Lines and Arcs to polylines? <Yes>:” prompt. All typical *Pedit* options are available (note the last prompt line).

TSCALE

This routine is virtually identical to the *Scaletext* command. The only difference is in the order of the options presented for the size change. Refer to Chapter 18, *Creating and Editing Text*, to learn more about the *Scaletext* command. *Tscale* issues the following prompts:

Command: **tscale**
 Select objects: **PICK**
 Select objects: **Enter**
 Specify justification to use as base point for scaling...
 [Existing/Start/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR] <Existing>: **Enter** or option
 Specify size change by [Scale (factor)/Height] <Height>: **Enter**
 Final height <1’>: (value)

SSX

SSX is a powerful utility that allows you to find objects of similar type (*Circle, Arc, Line*, etc.) for the selection set by selecting only one object. You can also specify a certain criteria. This feature is simpler than using *Filter* but can be almost as powerful. Enter *SSX* at the command prompt to initialize the routine, then enter "(SSX)" at any "Select objects:" prompt to use it. For example, you can select all similar objects ("Flatfile" block in this case) by selecting only one:

```
Command: move
Select objects: (ssx)
Select object/<None>: PICK
Filter: ((0 . "INSERT") (2 . "FLATFILE") (8 . "FURNITURE") (210 0.0 0.0 1.0))
>>Block name/Color/Entity/Flag/LAyer/LType/Pick/Style/Thickness/Vector: Enter
10 found. 10 found
Select objects: Enter
Base point or displacement: PICK
Second point of displacement: PICK
Command:
```

EXPRESSTOOLS

You can use the *Expresstools* command to make the AutoCAD *Express Tools* available in the current profile if they are not yet available. *Expresstools* places the Express directory on the search path, and loads and places the *Express* menu on the pull-down menu bar. The *Express Tools* must first be installed for this command to operate (see "Installing the Express Tools from the CD-ROM" earlier this chapter).

Once the *Expresstools* command is used, the *Express Tools* libraries load when AutoCAD is started. This increases the load time for AutoCAD but reduces the delay when you use one of the tools for the first time.

EXPRESSMENU

Expressmenu loads the AutoCAD *Express* pull-down menu specifically. If the *Express Tool* commands are available on the command line but the *Express* pull-down menu is not displayed, use the *Expressmenu* command to display the menu.