Chapter Objectives

After completing this chapter you should:

1. understand the concept of creating and inserting symbols in AutoCAD drawings;

2. be able to use the Block command to transform a group of objects into one object that is stored in the current drawing's block definition table;

3. be able to use the Insert and Minsert commands to bring Blocks into drawings;

4. know that color, linetype and lineweight of Blocks are based on conditions when the Block is made;

5. be able to convert Blocks to individual objects with Explode;

6. be able to use Wblock to prepare .DWG files for insertion into other drawings;

7. be able to redefine and globally change previously inserted Blocks;

8. be able to use AutoCAD DesignCenter™ to drag and drop Blocks from other drawings into the current drawing.
CONCEPTS

A Block is a group of objects that are combined into one object with the Block command. The typical application for Blocks is in the use of symbols. Many drawings contain symbols, such as doors and windows for architectural drawings, capacitors and resistors for electrical schematics, or pumps and valves for piping and instrumentation drawings. In AutoCAD, symbols are created first by constructing the desired geometry with objects like Line, Arc, and Circle, then transforming the set of objects comprising the symbol into a Block. A description of the objects comprising the Block is then stored in the drawing’s “block definition table.” The Blocks can then each be Inserted into a drawing many times and treated as a single object. Text can be attached to Blocks (called Attributes) and the text can be modified for each Block when inserted.

Figure 21-1 compares a shape composed of a set of objects and the same shape after it has been made into a Block and Inserted back into the drawing. Notice that the original set of objects is selected (highlighted) individually for editing, whereas, the Block is only one object.

Since an inserted Block is one object, it uses less file space than a set of objects that is copied with Copy. The Copy command creates a duplicate set of objects, so that if the original symbol were created with 10 objects, 3 copies would yield a total of 40 objects. If instead the original set of 10 were made into a Block and then Inserted 3 times, the total objects would be 13 (the original 10 + 3).

Upon Inserting a Block, its scale can be changed and rotational orientation specified without having to use the Scale or Rotate commands (Fig. 21-2). If a design change is desired in the Blocks that have already been Inserted, the original Block can be redefined and the previously inserted Blocks are automatically updated. Blocks can be made to have explicit Linetype, Lineweight and Color, regardless of the layer they are inserted onto, or they can be made to assume the Color, Linetype, and Lineweight of the layer onto which they are Inserted.
Blocks can be nested; that is, one Block can reference another Block. Practically, this means that the definition of Block “C” can contain Block “A” so that when Block “C” is inserted, Block “A” is also inserted as part of Block “C” (Fig. 21-3).

**Figure 21-3**

![Diagram of nested blocks](image)

**Blocks** created within the current drawing can be copied to disk as complete and separate drawing files (.DWG file) by using the Wblock command (Write Block). This action allows you to Insert the Blocks into other drawings. Specifically, when you use the Insert command, AutoCAD first searches for the supplied Block name in the current drawing’s block definition table. If the designated Block is not located there, AutoCAD searches the directories for a .DWG file with the designated name. Commands related to using Blocks are:

- **Block** Creates a Block from individual objects
- **Insert** Inserts a Block into a drawing
- **Minsert** Permits a multiple insert in a rectangular pattern
- **Explode** Breaks a Block into its original set of multiple objects
- **Wblock** Writes an existing Block or a set of objects to a file on disk
- **Base** Allows specification of an insertion base point
- **Purge** Deletes uninserted Blocks from the block definition table
- **Rename** Allows renaming Blocks
- **Adcenter** Invokes AutoCAD DesignCenter, which allows you to drag and drop Blocks, Dimension Styles, Layers, Layouts, Linetypes, Text Styles, and Xrefs into the current drawing (separate window)

AutoCAD DesignCenter, a new feature in AutoCAD 2000, provides an easy method for inserting Blocks into a drawing. The unique characteristic of DesignCenter is that you can drag and drop any named objects (including Blocks) from other drawings into the current drawing. AutoCAD DesignCenter is fully explained near the end of this chapter.
COMMANDS

BLOCK

<table>
<thead>
<tr>
<th>Pull-down Menu</th>
<th>COMMAND (TYPE)</th>
<th>ALIAS (TYPE)</th>
<th>Short-cut</th>
<th>Screen (side) Menu</th>
<th>Tablet Menu</th>
</tr>
</thead>
<tbody>
<tr>
<td>Draw</td>
<td>BLOCK, or -BLOCK</td>
<td>B or -B</td>
<td>...</td>
<td>DRAW2</td>
<td>N,9</td>
</tr>
</tbody>
</table>

Selecting the icon button, using the pull-down or screen menu, or typing Block or Bmake, produces the Block Definition dialog box shown in Figure 21-4. This dialog box provides the same functions as using the -Block command (a hyphen prefix produces the Command line equivalent).

To make a Block, first create the Lines, Circles, Arcs, or other objects comprising the shapes to be combined into the Block. Next, use the Block command to transform the objects into one object—a Block.

In the Block Definition dialog box, enter the desired Block name in the Name edit box. Then use the Select Objects button (upper right of the Objects section) to return to the drawing temporarily to select the objects you wish to comprise the Block. After selection of objects, the dialog box reappears. Use the Specify Insertion Base Point button (in the Base Point section of the dialog box) if you want to use a point other than the default 0,0,0 as the “insertion point” when the Block is later inserted. Usually select a point in the corner or center of the set of objects as the base point. When you select OK, the new Block is defined and stored in the drawing’s block definition table awaiting future insertions.

If Delete is selected in the Objects section of the dialog box, the original set of “template” objects comprising the Block disappear even though the definition of the Block remains in the table. Checking Retain forces AutoCAD to retain the original objects (similar to using Oops after the Block command), or selecting Convert to Block keeps the original set of objects visible in the drawing but transforms them into a Block.

The bottom half of the dialog box is used to specify how Blocks are described when AutoCAD DesignCenter is used to drag and drop the Blocks into a drawing instead of using the Insert command. In DesignCenter, you can preview the Blocks and read the description of the Blocks. Generally the Create Icon from Block Geometry option is used to create the preview thumbnail sketch unless the Block geometry is very complex, in which case another icon file could be selected. The Block Units options are described later in this chapter.

The Names drop-down list (Fig. 21-5) is used to select existing Blocks if you want to redefine a Block (see “Redefining Blocks” later in this chapter).
If you prefer to type, use -Block to produce the Command line equivalent of the Block Definition dialog box. The command syntax is as follows:

**Command:** -Block  
**Block name (or ?):** (name) (Enter a descriptive name for the Block up to 31 characters.)  
**Insertion base point:** PICK or (coordinates) (Select a point to be used later for insertion.)  
**Select objects:** PICK  
**Select objects:** PICK (Continue selecting all desired objects.)  
**Select objects:** Enter

The Block then disappears as it is stored in the current drawing’s “block definition table.” The Oops command can be used to restore the original set of “template” objects (they reappear), but the definition of the Block remains in the table. Using the ? option of the Block command lists the Blocks stored in the block definition table.

**Block Color, Linetype, and Lineweight Settings**

The color, linetype, and lineweight of the Block are determined by one of the following settings when the Block is created:

1. When a Block is inserted, it is drawn on its original layer with its original color, linetype and lineweight (when the objects were created), regardless of the layer or color, linetype and lineweight settings that are current when the Block is inserted (unless conditions 2 or 3 exist).

2. If a Block is created on Layer 0 (Layer 0 is current when the original objects comprising the Block are created), then the Block assumes the color, linetype and lineweight of any layer that is current when it is inserted (Fig. 21-6).

3. If the Block is created with the special BYBLOCK color, linetype and lineweight setting, the Block is inserted with the color, linetype and lineweight settings that are current during insertion, whether the BYLAYER or explicit object color, linetype and lineweight settings are current.
Once the Block has been created, it is inserted back into the drawing at the desired location(s) with the Insert command. The Insert command produces the Insert dialog box (Fig. 21-7) which allows you to select which Block to insert and to specify the Insertion Point, Scale, and Rotation, either interactively (On-screen) or by specifying values.

First, select the Block you want to insert. All Blocks located in the drawing's block definition table are listed in the Name drop-down list. Next, determine the parameters for Insertion Point, Scale, and Rotation. You can enter values in the edit boxes if you have specific parameters in mind or check Specify On-screen to interactively supply the parameters. For example, with the settings shown in Figure 21-7, AutoCAD would allow you to preview the Block as you dragged it about the screen and picked the Insertion Point. You would not be prompted for a Scale or Rotation angle since they are specified in the dialog box as 1.0000 and 0 degrees, respectively. Entering any other values in the Scale or Rotation edit boxes causes AutoCAD to preview the Block at the specified scale and rotation angle as you drag it about the drawing to pick the insertion point. Remember that Osnapss can be used when specifying the parameters interactively. Check Uniform Scale to ensure the X, Y, and Z values are scaled proportionally. Explode can also be toggled, which would insert the Block as multiple objects (see “Explode”).

Inserting Other Drawings as Blocks
Selecting the Browse tile in the Insert dialog box produces the Select Drawing dialog box (Fig. 21-8). Here you can select any AutoCAD drawing (.DWG file) for insertion. When one drawing is Inserted into another, the entire drawing comes into the current drawing as a Block, or as one object. If you want to edit individual objects in the inserted drawing, you must Explode the object.

If you prefer the command line equivalent, type –insert. The following prompts are issued.

Command: -insert
Enter block name or [?] : name (Type the name of an existing block or .DWG file to insert.)
Specify insertion point or [Scale/X/Y/Z/Rotate/PScale/PX/PY/PZ/PRotate] : PICK or option
Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1> : value, PICK or option
Enter Y scale factor <use X scale factor> : value or PICK
Specify rotation angle <0> : value or PICK
Command:
Sometimes it is desirable to see the Block in the intended scale factor or rotation angle before you choose the insertion point. Presets (“PScale/PC/PY/PZ/PRotate”) allow you to specify a rotation angle or scale factor before you dynamically drag the Block to pick the insertion point. (Normally, you would have to select the insertion point before the prompts for scale factor and rotation angle appear.)

**MINsert**

<table>
<thead>
<tr>
<th>Pull-down Menu</th>
<th>COMMAND (TYPE)</th>
<th>ALIAS (TYPE)</th>
<th>Shortcut</th>
<th>Screen (side) Menu</th>
<th>Tablet Menu</th>
</tr>
</thead>
<tbody>
<tr>
<td>...</td>
<td>MININSERT</td>
<td>...</td>
<td>...</td>
<td>...</td>
<td>...</td>
</tr>
</tbody>
</table>

This command allows a multiple insert in a rectangular pattern (Fig. 21-9). **Minsert** is actually a combination of the Insert and the Array Rectangular commands. The Blocks inserted with **Minsert** are associated (the group is treated as one object) and cannot be edited independently.

Examining the command syntax yields the similarity to a Rectangular Array.

**Figure 21-9**

Command: **Minsert**

Enter block name [or ?] <current>: \(\text{name}\)

Specify insertion point or [Scale/X/Y/Z/Rotate/PScale/PX/PY/PZ/PRotate]: \(\text{value}\), PICK or option

Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: \(\text{value}\) or Enter

Enter Y scale factor <use X scale factor>: \(\text{value}\) or Enter

Specify rotation angle <0>: \(\text{value}\) or Enter

Enter number of rows (—-) <1>: \(\text{value}\)

Enter number of columns (|||) <1>: \(\text{value}\)

Enter distance between rows or specify unit cell (—-): \(\text{value}\) or PICK (Value specifies Y distance from Block corner to Block corner; PICK allows drawing a unit cell rectangle.)

Distance between columns: \(\text{value}\) or PICK (Specifies X distance between Block corners.)

**EXPLODE**

<table>
<thead>
<tr>
<th>Pull-down Menu</th>
<th>COMMAND (TYPE)</th>
<th>ALIAS (TYPE)</th>
<th>Shortcut</th>
<th>Screen (side) Menu</th>
<th>Tablet Menu</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modify</td>
<td>EXPLODE</td>
<td>X</td>
<td>...</td>
<td>MODIFY2 EXPLODE</td>
<td>Y, 22</td>
</tr>
</tbody>
</table>

**EXPLODE** breaks a previously inserted Block back into its original set of objects (Fig. 21-10, on the next page), which allows you to edit individual objects comprising the shape. Blocks that have been **Minserted** cannot be **Exploded** in AutoCAD 2000. There are no options for this command.
-INSERT with *

Using the -Insert command with the asterisk (*) allows you to insert a Block, not as one object, but as the original set of objects comprising the Block. In this way, you can edit individual objects in the Block, otherwise impossible if the Block is only one object (Fig. 21-10).

The normal -Insert command is used (type -Insert with a hyphen); however, when the desired Block name is entered, it is prefaced by the asterisk (*) symbol:

Command: -insert
Enter Block name (or ?): * (name) (Type the * symbol, then the name of an existing block or .DWG file to insert.)
Specify insertion point for block:
Scale factor <1>:
Specify rotation angle <0>:
Command:

This action accomplishes the same goal as using the Insert dialog box; then Explode.

**XPLODE**

<table>
<thead>
<tr>
<th>Pull-down Menu</th>
<th>COMMAND (TYPE)</th>
<th>ALIAS (TYPE)</th>
<th>Shortcut</th>
<th>Screen (side) Menu</th>
<th>Tablet Menu</th>
</tr>
</thead>
<tbody>
<tr>
<td>...</td>
<td>XPLODE</td>
<td>...</td>
<td>...</td>
<td>...</td>
<td>...</td>
</tr>
</tbody>
</table>

The Xplode command is an expanded version of Explode. The Xplode command has been included in AutoCAD for many past releases as a Bonus command. In AutoCAD 2000, Xplode has been moved into the core product, so you do not have to load the Bonus commands to use it.

When you Insert a Block into a drawing, all layers, linetypes, and colors contained in the Block are also inserted into the parent drawing if they do not already exist. If you use Explode to break down the Block into its component entities, those entities retain their native properties of layer, linetype and color. For example, you may insert a Block that also inserts its layer named FLOORPLAN. If that Block is Exploded, its objects remain on layer FLOORPLAN. Often, however, you may want those objects to take on properties of the parent drawing. You may instead want to Explode the Block and have the objects reside on a layer existing in the parent drawing, AR-WALL, for example.

The Xplode command allows you to specify new properties for the objects that are exploded. When you use Xplode, you can also specify the new Layer, Linetype, Color, Lineweight, or choose to Explode the Block normally.

Command: xplode
Select objects to Xplode.
Select objects: PICK
Select objects: Enter
Enter an option [All/Color/Layer/LType/Inherit from parent block/Explode] <Explode>: 
The options are as follows.

**Color**
Use this option to specify a color for the exploded objects. Entering *ByLayer* causes the component objects to inherit the color of the exploded object’s layer. Entering *ByBlock* causes the component objects to inherit the object-specific color of the exploded object.

```
Enter an option [All/Color/Layer/LType/Inherit from parent block/Explode] <Explode>: c
Enter new color for exploded objects.
[Red/Yellow/Green/Cyan/Blue/Magenta/White/ByLayer/ByBlock] <BYLAYER>:
```

**Layer**
With this option you can specify the layer of the component objects after you explode them. The default option is to inherit the current layer rather than the layer of the exploded object.

```
Enter an option [All/Color/Layer/LType/Inherit from parent block/Explode] <Explode>: la
Enter new layer name for exploded objects <0>:
```

**Linetype**
You can enter the name of any linetype that is loaded in the drawing. The exploded objects assume the specified linetype. Entering *ByLayer* causes the component objects to inherit the linetype of the exploded object’s layer. Entering *ByBlock* causes the component objects to inherit the object-specific linetype of the exploded object.

```
Enter an option [All/Color/Layer/LType/Inherit from parent block/Explode] <Explode>: lt
Enter new linetype name for exploded objects <BYLAYER>:
```

**Inherit from parent block**
This option sets the color, linetype, lineweight, and layer of the exploded objects to that of the *Block* if the *Block* was created using *ByBlock* color, linetype, and lineweight and the objects were drawn on layer 0.

**All**
Use this option to specify a layer, color, linetype, and lineweight for the exploded objects. This is the only method for specifying a distinct lineweight.

```
Enter an option [All/Color/Layer/LType/Inherit from parent block/Explode] <Explode>: all
Enter new color for exploded objects.
[Red/Yellow/Green/Cyan/Blue/Magenta/White/ByLayer/ByBlock] <BYLAYER>:
Enter new linetype name for exploded objects <BYLAYER>:
Enter new lineweight <BYLayer>:
Enter new layer name for exploded objects <0>:
```

**WBLOCK**

<table>
<thead>
<tr>
<th>Pull-down Menu</th>
<th>COMMAND (TYPE)</th>
<th>ALIAS (TYPE)</th>
<th>Short-cut</th>
<th>Screen (side) Menu</th>
<th>Tablet Menu</th>
</tr>
</thead>
<tbody>
<tr>
<td><em>File</em></td>
<td><em>WBLOCK</em></td>
<td><em>W</em></td>
<td>...</td>
<td><em>FILE</em></td>
<td><em>W,24</em></td>
</tr>
</tbody>
</table>

The *Wblock* command writes a *Block* out to disk as a separate and complete drawing (.DWG) file. The *Block* used for writing to disk can exist in the current drawing’s *Block* definition table or can be created by the *Wblock* command. Remember that the *Insert* command inserts *Blocks* (from the current drawing’s block definition table) or finds and accepts .DWG files and treats them as *Blocks* upon insertion.
There are two ways to create a Wblock from the current drawing, (1) using an existing Block and (2) using a set of objects not previously defined in the current drawing as a Block. If you are using an existing Block, a copy of the Block is essentially transformed by the Wblock command to create a complete AutoCAD drawing (.DWG) file. The original block definition remains in the current drawing’s block definition table. In this way, Blocks that were originally intended for insertion into the current drawing can be inserted into other drawings.

Figure 21-11 illustrates the relationship among a Block, the current drawing, and a WBlock. In the figure, SCHEM1.DWG contains several Blocks. The RES block is written out to a .DWG file using Wblock and named RESISTOR. RESISTOR is then Inserted into the SCHEM2 drawing.

If you want to transform a set of objects into a Block to be used in other drawings but not in the current one, you can use Wblock to transform (a copy of) the objects in the current drawing into a separate .DWG file. This action does not create a Block in the current drawing. As an alternative, if you want to create symbols specifically to be inserted into other drawings, each symbol could be created initially as a separate .DWG file.
In AutoCAD 2000, the Wblock command produces the Write Block dialog box (Fig. 21-12). You should notice similarities to the Block Definition dialog box. Under Source, select Block if you want to write out an existing Block and select the Block name from the list, or select Objects if you want to transform a set of objects (not a previously defined Block) into a separate .DWG file.

The Base Point section allows you to specify a base point to use upon insertion of the Block. Enter coordinate values or use the Specify Insert Base Point button to pick a location in the drawing. The Objects section allows you to specify how you want to treat selected objects if you create a new .DWG file from objects (not from an existing Block). You can Retain the objects in their current state, Convert to Block, or Delete from Drawing.

The Destination section defines the desired File Name, Location, and Insert Units. Your choice for Insert Units is applicable only when you drag and drop a Block, as with AutoCAD DesignCenter. See “AutoCAD DesignCenter” later in this chapter for an explanation of this subject.

If you prefer the Command line equivalent to the Write Block dialog box, type -Wblock and follow the prompt sequence shown here to create Wblocks (.DWG files) from existing Blocks.

Command: -wblock
(At this point, the Create Drawing File dialog box appears, prompting you to supply a name for the .DWG file to be created. Typically, a new descriptive name would be typed in the edit box rather than selecting from the existing names.)

Enter name of existing block or [= (block=output file)* (whole drawing)] <define new drawing>: (Enter the name of the desired existing Block. If the file name given in the previous step is the same as the existing Block name, an “=” symbol can be entered, or enter an asterisk to write out the entire drawing.)

Command:

A copy of the existing Block is then created in the selected directory as a Wblock (.DWG file).

An alternative method for creating a Wblock is using the Export Data dialog box accessed from the File pull-down menu. Make sure you select Drawing (*.DWG) as the type of file to export (bottom left of the dialog box). You can create a Wblock (.DWG file) from an existing Block or from objects that you select from the screen. After specifying the .DWG name you want to create and the dialog box disappears, you are presented with the same command syntax as shown previously for the -Wblock command.

When Wblocks are Inserted, the Color, Linetype, and Lineweight settings of the Wblock are determined by the settings current when the original objects comprising the Wblock were created. The three possible settings are the same as those for Blocks (see “Block,” Color, Linetype, and Lineweight Settings).

When a Wblock is Inserted, its parent (original) layer is also inserted into the current drawing. Freezing either the parent layer or the layer that was current during the insertion causes the Wblock to be frozen.

Figure 21-12
Redefining Blocks

If you want to change the configuration of a Block, even after it has been inserted, it can be accomplished by redefining the Block. In doing so, all of the previous Block insertions are automatically and globally updated (Fig. 21-13). AutoCAD stores two fundamental pieces of information for each Block insertion—the insertion point and the Block name. The actual block definition is stored in the block definition table. Redefining the Block involves changing that definition.

To redefine a Block, use the Block command. First, draw the new geometry or change the original “template” set of objects. (The change cannot be made using an inserted Block unless it is Exploded because a Block cannot reference itself.) Next, use the Block command and select the new or changed geometry. The old Block is redefined with the new geometry as long as the original Block name is used.

The Block command can also be used to redefine Wblocks that have been inserted. In this case, enter the same Wblock name (.DWG filename) in the “File name:” edit box of the Write Block dialog box to redefine (actually replace) a previously inserted Wblock.

BASE

<table>
<thead>
<tr>
<th>Pull-down Menu</th>
<th>COMMAND (TYPE)</th>
<th>ALIAS (TYPE)</th>
<th>Shortcut</th>
<th>Screen (side) Menu</th>
<th>Tablet Menu</th>
</tr>
</thead>
<tbody>
<tr>
<td>Draw Block &gt; Base</td>
<td>BASE</td>
<td>...</td>
<td>...</td>
<td>DRAW2 Base</td>
<td>...</td>
</tr>
</tbody>
</table>

The Base command allows you to specify an “insertion base point” (see the Block command) in the current drawing for subsequent insertions. If the Insert command is used to bring a .DWG file into another drawing, the insertion base point of the .DWG is 0,0 by default. The Base command permits you to specify another location as the insertion base point. The Base command is used in the symbol drawing, that is, used in the drawing to be inserted. For example, while creating separate symbol drawings (.DWGs) for subsequent insertion into other drawings, the Base command is used to specify an appropriate point on the symbol geometry for the Insert command to use as a “handle” other than point 0,0 (Fig. 21-14).
**Blocks Search Path**

In AutoCAD 2000 you can use the Options dialog box to specify the search path used when the Insert command attempts to locate .DWG files. As you remember, when Insert is used and a Block name is entered, AutoCAD searches the drawing’s block definition table and, if no Block by the specified name is found, searches the specified path for a .DWG file by the name. (If you use the Insert dialog box, you can select the Browse… button to locate a .DWG file.) If the paths of the intended files are specified in the “Support File Search Path,” you can type Insert and the name of the file, and AutoCAD will locate and insert the file. Otherwise, you have to locate the file using the Browse button each time you Insert a new file.

The Files tab of the Options dialog box allows you to list and modify the path that AutoCAD searches for drawings to Insert (Fig. 21-15). The search path is also used for finding font files, menus, plug-ins, line-types, and hatch patterns.

Suppose all of your symbols were stored in a folder called D:\Dwg\Symbols. Access the Options dialog box by typing Options or selecting from the Tools pull-down menu. In the Files tab, open the Support File Search Path section.

Select Add… or Browse… to enter the location of your symbols as shown in the figure. Use Move Up or Move Down to specify an order (priority) for searching. Then each time you type “Insert,” you could enter the name of a file without the path and AutoCAD can locate it.

**PURGE**

<table>
<thead>
<tr>
<th>Pull-down Menu</th>
<th>COMMAND TYPE</th>
<th>ALIAS TYPE</th>
<th>Short-cut</th>
<th>Screen (side) Menu</th>
<th>Tablet Menu</th>
</tr>
</thead>
<tbody>
<tr>
<td>File Drawing Utilities &gt; Purge &gt;</td>
<td>PURGE</td>
<td>PU</td>
<td>…</td>
<td>FILE</td>
<td>X,25</td>
</tr>
</tbody>
</table>

Purge allows you to selectively delete a Block that is not referenced in the drawing. In other words, if the drawing has a Block defined but not appearing in the drawing, it can be deleted with Purge. In fact, Purge can selectively delete any named object that is not referenced in the drawing. Examples of unreferenced named objects are:

*Blocks* that have been defined but not *Inserted*;
*Layers* that exist without objects residing on the layers;
*Dimstyles* that have been defined, yet no dimensions are created in the style (see Chapter 29);
*Linetypes* that were loaded but not used;
*Shapes* that were loaded but not used;
*Text Styles* that have been defined, yet no text has been created in the *Style*;
*Mlstyles* (multiline styles) that have been defined, yet no *Mlines* have been drawn in the style.
Since these named objects occupy a small amount of space in the drawing, using Purge to delete unused named objects can reduce the file size. This may be helpful when drawings are created using template drawings that contain many unused named objects or when other drawings are inserted that contain many unused Layers or Blocks, etc.

Purge is especially useful for getting rid of unused Blocks because unused Blocks can occupy a huge amount of file space compared to other named objects (Blocks are the only geometry-based named objects). Although some other named objects can be deleted by other methods (Windows 95/98/NT dialog boxes with right-click options), Purge is the only method for deleting Blocks.

Purge can be invoked by the methods shown in the command table. Using the File pull-down menu as shown in Figure 21-16 reveals all of the Purge options. The command options (when typing Purge) are listed next.

Command: purge
Enter type of unused objects to purge
[Blocks/Dimstyles/Layers/LTypes/SHapes/STyles/Mlinestyles/All]: b
Enter name(s) to purge <*>:
Verify each name to be purged? [Yes/No] <Y>:

You can select one type of object to be Purged or select All to have all named objects listed. AutoCAD lists all named objects matching the type selected. For example, if you specified Blocks to purge, AutoCAD responds with a list of unused Blocks that may look something like this:

Purge block DESK? <N>
Purge block DESK2? <N>
Purge block CHAIR? <N>
Purge block TABLE? <N>

Answering with a "Y" causes the deletion of the Blocks. The other named object options operate similarly. Purge can be used any time in a drawing session.
This utility command allows you to rename a Block or any named object that is part of the current drawing. Rename allows you to rename the named objects listed here:

- Blocks
- Dimension Styles
- Layers
- Linetypes
- Text Styles
- User Coordinate Systems
- Views
- Viewport configurations

If you prefer to use dialog boxes, you can type Rename or select Rename... from the Format pull-down menu to access the dialog box shown in Figure 21-17.

You can select from the Named Objects list to display the related objects existing in the drawing. Then select or type the old name so it appears in the Old Name: edit box. Specify the new name in the Rename To: edit box. You must then PICK the Rename To: tile and the new names will appear in the list. Then select the OK tile to confirm.

The Rename dialog box can be used with wildcard characters (see Chapter 43) to specify a list of named objects for renaming. For example, if you desired to rename all the "DIM-*" layers so that the letters "DIM" were replaced with only the letter "D", the following sequence would be used.

1. In the Old Name edit box, enter "DIM-*" and press Enter. All of the layers beginning with "DIM-" are highlighted.
2. In the Rename To: edit box, enter "D-*." Next, the Rename To: tile must be PICKed. Finally, PICK the OK tile to confirm the change.

The -Rename command (with a hyphen prefix) can be used to rename objects one at a time. For example, the command sequence might be as follows:

```
Command: -rename
Enter object type to rename [Block/Dimstyle/Layer/LType/Plot style/Style/Ucs/View/ViewPort]: b
Enter old block name: chair7
Enter new block name: desk-chair
Command:
```

Wildcard characters are not allowed with the -Rename command but are allowed with the dialog box.

**AutoCAD DesignCenter**

Content Explorer™, one of the features developed for AutoCAD LT97®, greatly simplifies the process of locating and inserting Blocks. Content Explorer allows users to easily locate Blocks within other drawings and Insert the Blocks into the current drawing by dragging and dropping thumbnail sketches of the Blocks.
In addition, Content Explorer provides the same ease of use for locating and opening or importing all kinds of “content,” such as other drawings, dimension styles, layers, linetypes, text styles, Xrefs, and even commonly used web sites. The name “Content Explorer” is derived from its ability to locate a variety of content and its similarity to Windows Explorer.

This same technology is available in AutoCAD 2000 and is called the AutoCAD DesignCenter. The AutoCAD DesignCenter window (Fig. 21-18) allows you to navigate, find, and preview a variety of content, including Blocks, located anywhere accessible to your workstation, then allows you to open or insert the content using drag-and-drop. “Content” that can be viewed and managed includes other drawings, Blocks, Dimstyles, Layers, Layouts, Linetypes, Textstyles, Xrefs, raster images, and URLs (web site addresses). In addition, if you have multiple drawings open, you can streamline your drawing process by copying and pasting content, such as layer definitions, between drawings. (Dimension Styles, Xrefs, Raster Images, and Internet Tools are discussed in Chapters 29, 30, 32, and 43, respectively.)

**Figure 21-18**

Accessing Adcenter by any method produces the AutoCAD DesignCenter window (see Figure 21-19). By default, DesignCenter is docked at the left side of the drawing area (see Figure 21-18). There are two sections to the window. The left side is called the Tree View and displays a Windows Explorer-type hierarchical directory (folder) structure of the local system (Fig. 21-19). The right side is called the Palette and displays lists, icons, or thumbnail sketches of the content selected in the Tree View. The content shown in the Palette can be Blocks, Dimstyles, Layers, or a variety of content.

Generally, the Palette is used to drag and drop the icons or thumbnails into the current drawing as shown in Figure 21-19. However, you can also streamline a variety of tasks such as those listed here:

- Browse sources of drawing content including open drawings, other drawings, raster images, content within the drawings (Blocks, Dimstyles, Layers, and so on), content on network drives, or content on a web page.
• Insert, attach, or copy and paste the content (drawings, images, Blocks, Layers, etc.) into the current drawing.
• Create shortcuts to drawings, folders, and Internet locations that you access frequently.
• Use a special search engine to find drawing content on your computer or network drives. You can specify criteria for the search based on key words, names of Blocks, Dimstyles, Layers, etc., or the date a drawing was last saved. Once you have found the content, you can load it into AutoCAD DesignCenter or drag it into the current drawing.
• Open drawings by dragging a drawing (.DWG) file from the Palette into the drawing area.

These features of DesignCenter are explained in the following descriptions of the DesignCenter toolbar icons.

Moving and Resizing the AutoCAD DesignCenter Window
You can resize AutoCAD DesignCenter by moving the bar between the Palette and the Tree View or by moving the lower right corner with your pointing device and dragging the window to the required size. Undock DesignCenter by clicking the top bar, dragging it away from the left side, and dropping it to make it a floating window. You can move the floating window anywhere on the screen and change its width and height.

Tree View Options

In addition to selecting the icons from the toolbar, you can right-click the Palette background to produce the shortcut menu and choose the desired option (Fig. 21-20).

Desktop

Desktop is the default display for the Tree View side of AutoCAD DesignCenter. This choice displays a hierarchical structure of the desktop (local workstation). Because this arrangement is similar to Windows Explorer, you can navigate and locate content anywhere accessible to your system, including network drives. Figure 21-20 displays a typical hierarchical structure on the Tree View side. For example, you may want to import layer definitions (including color and linetype information) from a drawing into the current drawing by dragging and dropping.
Open Drawings
This option changes the Tree View to display all open drawings (Fig. 21-21). This feature is helpful when you have several drawings open and want to locate content from one drawing and import it into the current drawing. As shown in Figure 21-21, you may want to locate Block definitions from one drawing and Insert them into another drawing. Ensure you make the desired “target” drawing current in the drawing area before you drag and drop content from the Palette.

History
This option displays a history (chronological list) of the last 20 file locations accessed through AutoCAD DesignCenter (Fig. 21-22). The purpose of this feature is simply to locate the file and load it into the Palette. Load the file into the Palette by double-clicking on it.

Tree View Toggle
Tree View is helpful for navigating your system for content. Once the desired folder or drawing is found and highlighted in Tree View, you may want to toggle Tree View off so only the Palette is displayed with the desired content. The desired content may be drawings, Blocks, images, or a variety of other content. For example, consider previous Figure 21-21 which displays Tree View with Blocks for EMETFAST.DWG selected. Figure 21-23 illustrates the same selection in DesignCenter with Tree View toggled off and only the Palette displayed. The resulting configuration displays only the Blocks contained in the selected drawing. Keep in mind that you can also change the Views of the Palette to display Large Icons, Small Icons, a List, or Details (see “Views”).
Favorites

This button displays the contents of the AutoCAD Favorites folder in the Palette (Fig. 21-24). The Tree View section displays the highlighted folder in the Desktop view. You can add folders and files to Favorites by highlighting an item in Tree View or the Palette, right-clicking on it, and selecting Add to Favorites from the shortcut menu.

Load

Displays the Load DesignCenter Palette dialog box (not shown), in which you can load the Palette with content from anywhere accessible from your system. The Load DesignCenter Palette dialog box is identical to the Select File dialog box (see “Open,” Chapter 2, Working with Files). After selecting a file, DesignCenter automatically finds the file in Tree View and loads its content (Blocks, Layers, Dimstyles, etc.) into the Palette. Note that the Find File and Locate buttons can be used to locate files to load. You can also load files into the Palette using Windows Explorer (see “Loading the Palette with Windows Explorer” at the end of this section).

Find

This button invokes the Find dialog box (Fig. 21-25), in which you can specify search criteria to locate drawing files, Blocks, Layers, Dimstyles, and other content within drawings. Once the desired content is found in the list at the bottom of the dialog box, double-click on it to load it into the Palette.

This feature is extremely powerful and easy to use. The list of possible items to search for is displayed in the Look For drop-down list and includes the following choices:

- Blocks
- Dimstyles
- Drawings
- Drawings and Blocks
- Layers
- Layouts
- Linetypes
- Text Styles
- Xrefs

If Drawings is selected in the list, three tabs (shown in Figure 21-25) appear to allow you to refine the search criteria.
Drawings
Enter a text string in the Search for words edit box. Wildcards can be used (see Chapter 43 for valid wildcards in AutoCAD). From the In the field(s) drop-down list, select what field you want your text string to apply to:

- File Name
- Title
- Subject
- Author
- Keywords

Title, Subject, Author, and Keywords are searched for only in the Drawing Properties (description), so if you are searching for pre-AutoCAD 2000 drawings or have not specified drawing properties, these fields are not useful.

Date Modified
You can search by Date Modified by specifying modification between two dates or during the previous X number of days or X number of months.

Advanced
The Advanced tab allows you to search for text strings in the Drawing and Block description, Block Name, Attribute Tag, or Attribute Value.

Up
Use the Up tool to display the next higher level in the Tree View hierarchy.

Preview
This button causes a preview image (thumbnail sketch) of the selected item to appear at the bottom of the Palette. If there is no preview image saved with the selected item, the Preview area is empty. You can resize the preview image by dragging the bar between the Palette and the preview pane.

To make a preview image appear, you must select an item (drawing, Block, or image file) from the Palette. Selecting items in the Tree View does not cause a preview to appear. Figure 21-26 displays a drawing preview. Figure 21-19 displays a Block preview.
**Description**

This button displays a text description of the selected item at the bottom of the Palette (Fig. 21-27). The description is displayed only for Blocks and is derived from information saved in the Description field when the Block command is used. The description area can be resized. If you display both Preview and Description panes, the Description pane is displayed below the Preview pane, separated from it by a bar (see Figure 21-19).

You cannot edit the text description in AutoCAD DesignCenter; however, you can copy the description to the Clipboard. Select the text you want to copy, right-click inside the pane, and then select Copy to Clipboard from the shortcut menu.

**Views**

Four possible Views of the Palette area are possible. Make the choice by selecting the down arrow to display the options (Fig. 21-28) or repeatedly click on the View icon as it cycles through the Views.

- Large Icons
- Small Icons
- List
- Details

It is helpful to reset the configuration of the Palette depending on the job you are performing. For example, if the Preview pane is displayed, you can set the Palette to display only a List or Details (see Figure 21-26). Or if you prefer to “see” all the drawing files or Blocks in the Palette, it would be better to set the View to Large Icons and toggle Preview off (see Figure 21-27).

Generally, items are sorted alphabetically by name in the Palette. If you change the View to Details, you can sort files (not Blocks) by name, size, type, and other properties, depending on the type of content displayed in the Palette (see Figure 21-26). Click on the column header for the column to sort by (one click for ascending order, two clicks for descending order).

**Loading the Palette with Windows Explorer**

You can use Windows Explorer to load content into the Palette. For example, if you are browsing a network drive in Windows Explorer and locate a drawing file, you can drag the selected file directly into the Palette. The selected file must be dropped in to the Palette area, not the Tree View, Preview, or description areas. If you drop the file into any area other than the Palette, the drawing is opened into the AutoCAD session.

**To Open a Drawing from AutoCAD DesignCenter**

In addition to using AutoCAD DesignCenter to view and import content contained within drawing files, it is possible to use AutoCAD DesignCenter to Open drawings. To do this, drag the icon of the drawing file you want to open from the Palette and drop it in the drawing area. Remember, you must drag it from the Palette, not from the Tree View area.
464  Blocks and AutoCAD DesignCenter

Usually, you would want to drop the selected drawing into a blank (New) drawing. Unless the selected drawing is a titleblock and border, drop it into model space, not into a layout (paper space). Make sure the background (drawing area) is visible. You may need to resize the windows displaying any currently open drawings.

A drawing file that is dropped into AutoCAD is actually Inserted as a Block. The typical Command line prompts appear for insertion point, scale factors, and rotation angle. Generally, enter 0,0 as the insertion point and accept the defaults for X and Y scale factors and rotation angle. If you want to edit the geometry, you will have to Explode the drawing.

To Insert a Block Using DesignCenter

One of the primary functions of AutoCAD DesignCenter is to insert Block definitions into a drawing. When you insert a Block into a drawing, the block definition is copied into the drawing database. Any instance of that Block that you Insert into the drawing from that time on references the original Block definition.

You cannot add Blocks to a drawing while another command is active. If you try to drop a Block into AutoCAD while a command is active at the Command line, the icon changes to a slash circle indicating the action is invalid.

There are two methods for inserting Blocks into a drawing using AutoCAD DesignCenter: (1) using drag-and-drop with Autoscaling and (2) using the Insert dialog box with explicit insertion point, scale, and rotation value entry.

Block Insertion with Drag-and-Drop

When you drag-and-drop a Block from DesignCenter into a drawing, you are not prompted for an insertion point, X and Y scale factors, or a rotation angle. Although you specify an insertion point interactively when you “drop” the Block icon, AutoCAD uses Autoscaling to determine the scaling parameters. Autoscaling is a process of comparing the specified units of the Block definition (when the Block was created) and the Drawing Units for Block Inserts set in the Drawing Units dialog box of the target drawing. The value options are Unitless, Inches, Feet, Miles, Millimeters, Centimeters, Kilometers, and many other choices.

The Drawing Units for Block Inserts set in the Drawing Units dialog box (Fig. 21-29) should be set in the target drawing and controls how the Block units are scaled when the Block is dropped (into the target drawing). It is helpful that the setting here can be changed immediately before dropping a Block into the drawing. (Also see “Units” in Chapter 6.)
In addition, you can specify the desired Block units in the Block Definition dialog box (Fig. 21-30). This setting is generally assigned when each Block is created. (See “Block” earlier in this chapter.)

Figure 21-30

Resulting Scale Factors with Autoscale (Drag-and-Drop)
This table gives sample scale factors that Autoscaling uses when Blocks are inserted into other drawings by the drag-and-drop method. The first row (top) of the table lists the Block settings (made in the Block Definition dialog box) and the first column (left) lists the drawing units settings (made in the Drawing Units dialog box of the target drawing).

Note that when the units in either case (defined in the Block or in the target Drawing Units) are set to Unitless, the scale factor is always 1.00. Also, when the two settings match (Inches and Inches, or Feet and Feet), the scale factor is always 1.00. The other cells of the table list a proportion of one unit to the other; for example, .0833 equals 1/12 (feet/inches).

To use Autoscaling without surprises, you should check the units set for the Blocks before you insert them and set the Drawing Units for Block Inserts in the target drawing according to your needs. You can check the settings for each Block by opening the drawing containing the Blocks, invoking the Block command, and selecting each Block from the Name drop-down list.

Although a simple drag-and-drop appears to be a sufficient method for Block insertion, be advised that there can be drawbacks. For example, dimension values inside blocks may not be true if a Block (or another drawing) is scaled automatically when you drag it into the drawing from AutoCAD DesignCenter. If unsure, right-click from the DesignCenter Palette and select Insert Block, then specify the scale factor at the command prompt, as described next.
Block Insertion with Specified Coordinates, Scale, and Rotation

The second method of inserting Blocks from the AutoCAD DesignCenter is to highlight the block name or icon, then right-click on it and select Insert Block from the shortcut menu (Fig. 21-31). Alternately, double-click on the block name or icon. This method produces the Insert dialog box (see “Insert” earlier this chapter). Here you can specify coordinates for insertion point, scale factor, and rotation angle. You have your choice (by using checkboxes) to specify these parameters in edit boxes or at the Command line prompts.

If you prefer this method but want to preview the block before inserting it, you can right-drag the block icon into the drawing area. The block appears in the drawing with the shortcut menu options. Select Insert Block and proceed to specify the parameters.

To Attach an Xref Using DesignCenter
See Chapter 30, Xreferences.

CHAPTER EXERCISES

1. Block, Insert

In the next several exercises, you will create an office floor plan, then create pieces of furniture as Blocks and Insert them into the office. All of the block-related commands are used.

A. Start a New drawing. Select Start from Scratch and use the English defaults. Use Save and assign the name OFF-ATT. Set up the drawing as follows:

1. Units
   Architectural
2. Limits
   48' x 36'
(1/4"=1' scale on an A size sheet), drawing scale factor = 48
3. Snap, Grid
   3
4. Grid
   12
5. Layers
   FLOORPLAN continuous .014 colors of your choice
   FURNITURE continuous .060
   ELEC-HDWR continuous .060
   ELEC-LINES hidden .060
   DIM-FLOOR continuous .060
   DIM-ELEC continuous .060
   TEXT continuous .060
   TITLE continuous .060
6. Text Style
   CityBlueprint CityBlueprint (TrueType font)
7. Ltscale
   24
B. Create the floor plan shown in Figure 21-32. Center the geometry in the *Limits*. Draw on layer **FLOORPLAN**. Use any method you want for construction (e.g., *Line, Pline, Xline, Mline, Offset*).

Figure 21-32

C. Create the furniture shown in Figure 21-33. Draw on layer **FURNITURE**. Locate the pieces anywhere for now. Do not make each piece a *Block*. *Save* the drawing as **OFF-ATT**.

Now make each piece a *Block*. Use the *name* as indicated and the *insertion base point* as shown by the “blip.” Next, use the *Block* command again but to list the block definition table. Use *SaveAs* and rename the drawing **OFFICE**.

Figure 21-33
D. Use Insert to insert the furniture into the drawing, as shown in Figure 21-34. You may use your own arrangement for the furniture, but Insert the same number of each piece as shown. Save the drawing.

2. Creating a .DWG file for Insertion, Base

Begin a New drawing. Assign the name CONFTABL. Create the table as shown in Figure 21-35 on Layer 0. Since this drawing is intended for insertion into the OFFICE drawing, use the Base command to assign an insertion base point at the lower-left corner of the table.

When you are finished, Save the drawing.

3. Insert, Explode, Divide

A. Open the OFFICE drawing. Ensure that layer FURNITURE is current. Use Insert to bring the CONFTABL drawing in as a Block in the placement shown in Figure 21-36.

Notice that the CONFTABL assumes the linetype and color of the current layer, since it was created on layer 0.

B. Explode the CONFTABL. The Exploded CONFTABL returns to Layer 0, so use the Properties window to change it back to Layer FURNITURE. Then use the Divide command (with the Block option) to insert the CHAIR block as shown in Figure 21-36. Also Insert a CHAIR at each end of the table. Save the drawing.
4. **Wblock, BYBLOCK setting**

   A. **Open** the EFF-APT2 drawing you worked on in Chapter 18 Exercises. Use the **Wblock** command to transform the plant into a .DWG file (Fig. 21-37). Use the name **PLANT** and specify the **Insertion base point** at the center. Do not save the EFF-APT2 drawing. **Close EFF-APT2**.

   B. **Open** the OFFICE drawing and **Insert** the PLANT into one of the three rooms. The plant probably appears in a different color than the current layer. **Why?** Check the **Layer** listing to see if any new layers came in with the PLANT block. **Erase** the PLANT block.

   C. **Open** the **PLANT** drawing. Change the **Color, Linetype, and Lineweight** setting of the plant objects to **BYBLOCK**. **Save** the drawing.

   D. **Open** the OFFICE drawing again and **Insert** the PLANT onto the FURNITURE layer. It should appear now in the current layer’s color, linetype, and lineweight. **Insert** a **PLANT** into each of the 3 rooms. **Save** the drawing.

5. **Redefining a Block**

   After a successful meeting, the client accepts the proposed office design with one small change. The client requests a slightly larger chair than that specified. **Explode** one of the CHAIR blocks. Use the **Scale** command to increase the size slightly or otherwise redesign the chair in some way. Use the **Block** command to redefine the CHAIR block. All previous insertions of the CHAIR should reflect the design change. **Save** the drawing. Your design should look similar to that shown in Figure 21-38. **Plot** to a standard scale based on your plotting capabilities.

6. **Rename**

   **Open** the OFFICE drawing. Enter **Rename** (or select from the **Format** pull-down menu) to produce the **Rename** dialog box. **Rename** the **Blocks** as follows:

<table>
<thead>
<tr>
<th>New Name</th>
<th>Old Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>DSK</td>
<td>DESK</td>
</tr>
<tr>
<td>CHR</td>
<td>CHAIR</td>
</tr>
<tr>
<td>TBL</td>
<td>TABLE</td>
</tr>
<tr>
<td>FLC</td>
<td>FILECAB</td>
</tr>
</tbody>
</table>
Next, use the *Rename* dialog box again to rename the *Layers* as indicated below:

<table>
<thead>
<tr>
<th>New Name</th>
<th>Old Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>FLOORPLN-DIM</td>
<td>DIM-FLOOR</td>
</tr>
<tr>
<td>ELEC-DIM</td>
<td>DIM-ELEC</td>
</tr>
</tbody>
</table>

Use the *Rename* dialog box again to change the *Text Style* name as follows:

<table>
<thead>
<tr>
<th>New Name</th>
<th>Old Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>ARCH-FONT</td>
<td>CITYBLUEPRINT</td>
</tr>
</tbody>
</table>

Use *SaveAs* to save and rename the drawing to **OFFICE-REN**.

7. **Purge**

Using the Windows Explorer, check the file size of **OFFICE-REN.DWG**. Now use *Purge*. Answer *Yes* to *Purge* any unreferenced named objects. *Exit AutoCAD* and *Save Changes*. Check the file size again. Is the file size slightly smaller?

8. Create the process flow diagram shown in Figure 21-39. Create symbols (*Blocks*) for each of the valves and gates. Use the names indicated (for the *Blocks*) and include the text in your drawing. *Save* the drawing as **PFD**.

---

**Figure 21-39**

[Diagram of process flow shown with numbered components and connections, including nitrogen, reactor A, reactor B, steam, water, and other process elements depicted in a flowchart layout.]