Chapter 15

Defining Block Attributes

Learning Objectives

After completing this chapter, you will be able to:

- *Understand what attributes are and how to define them with a block.*
- *Edit attribute tag names.*
- *Insert blocks with attributes and assign values to attributes.*
- *Extract attribute values from the inserted blocks.*
- *Control attribute visibility.*
- *Perform global and individual editing of attributes.*
- *Insert a text file in a drawing to create bill of material.*

UNDERSTANDING ATTRIBUTES

AutoCAD LT has provided a facility that allows the user to attach information to blocks. This information can then be retrieved and processed by other programs for various purposes. For example, you can use this information to create a bill of material for a project, find the total number of computers in a building, or determine the location of each block in a drawing. Attributes can also be used to create blocks (such as title blocks) with prompted or preformatted text, to control text placement. The information associated with a block is known as **attribute value** or simply **attribute**. AutoCAD LT references the attributes with a block through tag names.

Before assigning attributes to a block, you must create an attribute definition by using the **ATTDEF** command. The attribute definition describes the characteristics of the attribute. You can define several attribute definitions (tags) and include them in the block definition. Each time you insert the block, AutoCAD LT will prompt you to enter the value of the attribute. The attribute value automatically replaces the attribute tag name. The information (attribute values) assigned to a block can be extracted and written to a file by using AutoCAD LT's **ATTEXT** command. This file can then be inserted in the drawing as a table or processed by other programs to analyze the data. The attribute values can be edited by using the **ATTEDIT** command. The display of attributes can be controlled with the **ATTDISP** command.

DEFINING ATTRIBUTES

Menu:	Draw > Block > Define Attributes
Command:	ATTDEF

When you invoke the **ATTDEF** command, the **Attribute Definition** dialog box is displayed, see Figure 15-1. The block attributes can be defined through this dialog box. When creating an attribute definition, you must define the mode, attributes, insertion point, and text information for each attribute. All this information can be entered in the dialog box. The following is the description of each area of the **Attribute Definition** dialog box.

Mode Area

The **Mode** area of the **Attribute Definition** dialog box has four check boxes: **Invisible**, **Constant**, **Verify** and, **Preset**. These determine the display and edit features of the block attributes. For example, if you select the **Invisible** check box, the attribute becomes invisible; that is, it is not displayed on the screen. Similarly, if the **Constant** check box is selected, the attribute becomes constant. This means that its value is predefined and cannot be changed. These options are described next.

Invisible

This option lets you create an attribute that is not visible on the screen, by default. Clear this check box if you want the attribute to be visible.

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Figure 15-1 *The* ***Attribute Definition*** *dialog box*

Tip

The Invisible mode is especially useful when you do not want the attribute values to be displayed on the screen to avoid cluttering the drawing. Also, if the attributes are invisible, it takes less time to regenerate the drawing.

You can make the invisible attribute visible by using the **ATTDISP** command discussed later in this chapter, in the section "Controlling Attribute Visibility".

Constant

This option lets you create an attribute that has a fixed value and cannot be changed after block insertion. When you select this mode, the **Prompt** edit box and the **Verify** and **Preset** check boxes are disabled. Since the value is constant, there is no need to be prompted for new values. This check box is cleared by default and you can use different attribute values for the blocks.

Verify

This option allows you to verify the attribute value you have entered when inserting a block by asking you twice for the data. If the value is incorrect, you can correct it by entering the new value. If this check box is cleared, you are not prompted for verification of the attribute values.

Preset

This option allows you to create an attribute that is automatically set to the default value. The attribute values are not requested when you insert a block and the default values are used. But unlike a constant attribute, the preset attribute value can be edited later.

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Note

*Not selecting any of the check boxes in the **Mode** area displays all the prompts at the command line and the values will be visible on the screen. This is also referred to as the **Normal** mode.*

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Attribute Area

The **Attribute** area (Figure 15-2) of the **Attribute Definition** dialog box has three edit boxes: **Tag**, **Prompt**, and **Value**, where you can enter values. You can enter up to 256 characters in these edit boxes. If the first character to be entered in any one of these edit boxes is a space, you should start with a backslash (\). But if the first character is a backslash (\), you should start the value to be entered with two backslashes (\\). The three edit boxes have been described next.

Figure 15-2 *The **Attribute** area of the **Attribute Definition** dialog box*

Tag

This is like a label that is used to identify an attribute. For example, the tag name COMPUTER can be used to identify an item. Here you can enter the tag names as uppercase, lowercase, or both, but all lowercase letters are automatically converted into uppercase when displayed. The tag name cannot be null. Also, it must not contain any blank spaces.

Tip

It is advisable to specify a tag name that reflects the contents of the item being tagged. For example, the tag name COMP or COMPUTER is an appropriate name for labeling computers.

Prompt

The text that you enter in the **Prompt** edit box is used as a prompt when you insert a block that contains the defined attribute. For example, if COMPUTER is the tag, you can enter WHAT IS THE MEMORY? or ENTER MEMORY: in the **Prompt** edit box. AutoCAD LT will then prompt you with this same statement when you insert the block with which the attribute is defined. If you have selected the **Constant** check box in the **Mode** area, the **Prompt** edit box is not available because no prompt is required if the attribute is constant. If you do not enter anything in the **Prompt** edit box, the entry made in the **Tag** edit box is used as the prompt.

Value

The entry in the **Value** edit box defines the default value of the specified attribute. If you do not enter a value, it is used as the value for the attribute. The entry of a value is optional.

Insert field

This button is chosen to insert a field as the value of the attribute. When you choose this button, the **Field** dialog box is displayed that can be used to insert the required field.

Insertion Point Area

The **Insertion Point** area of the **Attribute Definition** dialog box (Figure 15-3) lets you define the

insertion point of the block attribute text. You can define the insertion point by entering the values in the **X**, **Y**, and **Z** edit boxes or by specifying it on the screen. To specify the insertion point on the screen, select the **Specify On-Screen** check box. Now, set the parameters in all the other fields and areas and then choose **OK**. You will be prompted to select the **Start Point** of the attribute.

Figure 15-3 *The* ***Insertion Point*** *area of the* ***Attribute Definition*** *dialog box*

Just below the **Insertion Point** area of the dialog box is a check box labeled **Align below previous attribute definition**. This check box is not available when you use the **Attribute Definition** dialog box for the first time. After you have defined an attribute and when you press ENTER to display the **Attribute Definition** dialog box again, this check box is available. You can select this check box to place the subsequent attribute text just below the previously defined attribute automatically. When you select this check box, the **Insertion Point** area and the **Text Options** areas of the dialog box are not available and AutoCAD LT assumes previously defined values for text such as text height, text style, text justification, and text rotation. The text is automatically placed on the following line.

Text Options Area

The **Text Options** area of the **Attribute Definition** dialog box (Figure 15-4) lets you define the justification, text style, height, and rotation of the attribute text. To set the text justification, select a justification type from the **Justification** drop-down list. The default option is **Left**. Similarly, you can use the **Text Style** drop-down list to select a text style. All the text styles defined in the current drawing are displayed in the **Text Style** drop-down list. The default text style is **Standard**. You can specify the text height and text rotation in the **Height** and **Rotation** edit boxes. You can also define the text height by choosing the **Height** button. When you choose this button, AutoCAD LT temporarily exits the dialog box and lets you enter the height value by selecting points on the screen or from the command line. Once you have defined the height on the screen, the dialog box reappears and the defined text height is displayed in the edit box. Similarly, you can define the text rotation by choosing the **Rotation** button and then selecting points on the screen or by entering the rotation angle at the command line.

Figure 15-4 *The* ***Text Options*** *area of the* ***Attribute Definition*** *dialog box*

Lock position in block Check box

This check box is selected to lock the position of the attribute, in case of blocks, and to consider the attribute while adding an action, in case of dynamic blocks. In case blocks are created using attributes with this check box cleared, a separate grip will be displayed on this attribute using which you can edit it. In case dynamic blocks are created using attributes, the attributes that were defined with the **Lock position in block** check box cleared, will not be considered to be manipulated with the parameters and actions.

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Note

The text style must be defined before it can be used to specify the text style.

If you select a style that has the height predefined, AutoCAD LT automatically disables the ***Height*** *edit box.*

If you have selected the ***Align*** *option from the* ***Justification*** *drop-down list, the* ***Insertion Point*** *area and the* ***Height*** *and* ***Rotation*** *edit boxes in the* ***Text Options*** *area are disabled.*

If you have selected the ***Fit*** *option from the* ***Justification*** *drop-down list, the* ***Insertion Point*** *area and the* ***Rotation*** *edit box is disabled. To specify the location of the attribute, choose* ***OK*** *from the dialog box. AutoCAD LT prompts you to specify the first and second endpoint of the text baseline. The text will be fit between the two endpoints that you specify.*

After you complete the settings in the **Attribute Definition** dialog box and choose **OK**, the attribute tag text is inserted in the drawing at the specified insertion point. Now, you can use the **BLOCK** or **WBLOCK** commands to select all the objects and attributes to define a block.

Figure 15-5 *Dynamic prompt for the -ATTDEF command*

Note

You can use the ***-ATTDEF*** *command to display dynamic prompt on the drawing area. The options in the* ***Attribute Definition*** *dialog box are available through the dynamic prompt too. See Figure 15-5.*

Example 1 — *General*

In this example, you will define the following attributes for a computer and then create a block using the **BLOCK** command. The name of the block is COMP.

Mode	Tag name	Prompt	Default value
Constant	ITEM		Computer
Preset, Verify	MAKE	Enter make:	CAD-CIM
Verify	PROCESSOR	Enter processor type:	Unknown
Verify	HD	Enter Hard-Drive size:	40 GB
Invisible, Verify	RAM	Enter RAM:	256 MB

1. Draw the computer, as shown in Figure 15-6. Assume the dimensions, or measure the dimensions of the computer you are using for AutoCAD LT.

2. Invoke the **ATTDEF** command. The **Attribute Definition** dialog box is displayed.

3. Define the first attribute as shown in the preceding table. Select **Constant** check box in the **Mode** area because the mode of the first attribute is constant. In the **Tag** edit box, enter the

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tag name, ITEM. Similarly, enter COMPUTER in the **Value** edit box. Note that the **Prompt** edit box is not available because the mode is constant.

4. In the **Insertion Point** area, choose the **Pick Point** button to define the text insertion point. Select a point below the insertion base point (P1) of the computer to place the text.

5. In the **Text Options** area, specify the justification, style, height, and rotation of the text.

6. Choose the **OK** button once you have entered information in the **Attribute Definition** dialog box.

7. Press ENTER to invoke the **Attribute Definition** dialog box again. Enter the mode and attribute information for the second attribute as shown in the table at the beginning of Example 1. You need not define the insertion point and text options again. Select the **Align below previous attribute definition** check box that is located just below the **Insertion Point** area. You will notice that when you select this check box, the **Insertion Point** and **Text Options** areas are not available. Now, choose the **OK** button. AutoCAD LT places the attribute text just below the previous attribute text.

8. Similarly, define the remaining attributes also (Figure 15-7).

Figure 15-6 *Drawing for Example 1*

Figure 15-7 *Define attributes below the computer drawing*

9. Now, use the **BLOCK** command to create a block. The name of the block is COMP, and the insertion point of the block is P1, midpoint of the base. When you select the objects for the block, make sure you also select the attributes.

Note

The order of prompts is the same as the order of attributes selection.

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EDITING ATTRIBUTE DEFINITION

Toolbar:	Text > Edit
Menu:	Modify > Object > Text > Edit
Command:	DDEDIT

Using the **DDEDIT** command, you can edit text and attribute definitions, before you define the block. After invoking this command, AutoCAD LT will prompt you to **Select an annotation object or [Undo]**. If you select an attribute created using the **Attribute Definition** dialog box, the **Edit Attribute Definition** dialog box is displayed and lists the tag name, prompt, and default value of the attribute (Figure 15-8). You can also invoke the **Edit Attribute Definition** dialog box by double-clicking on the attribute definition.

Figure 15-8 *The* ***Edit Attribute Definition*** *dialog box*

You can enter the new values in the respective edit boxes. Once you have entered the changed values, choose the **OK** button in the dialog box. After you exit the dialog box, AutoCAD LT will continue to prompt you to select another text or attribute object (Attribute tag). If you have finished editing and do not want to select another attribute object to edit, press ENTER to return to the Command prompt.

Using the PROPERTIES Palette

The **PROPERTIES** command has been already discussed in Chapter 4, Working with Drawing Aids. It can also be used to edit defined attributes. Select the attribute to be modified and right-click to display a shortcut menu. Choose **Properties** here and the **PROPERTIES** palette is displayed, see Figure 15-9. You will notice that **Attribute** is displayed in the text box located at the top of the palette. Also, under the **Categorized** tab, you will find that all the properties of the selected attribute are displayed under four headings. They are **General**, **Text**, **Geometry**, and **Misc**. You can change these values in their corresponding fields. For example, you can modify the color, layer, linetype, thickness, linetype scale, and so on, of the selected attribute under the **General** head. Similarly, you can modify the tag name, prompt, and value of the selected attribute in the **Tag**, **Prompt**, and **Value** fields under the **Text** heading. Under the **Text** head, you can also modify the text style, justification, text height, rotation angle, width factor, and obliquing angle values of the selected attribute.

Under the **Geometry** heading, you can redefine the insertion point of the selected attribute by choosing the button with an arrow icon, which is displayed when you select the **Position X**, **Position Y**, or **Position Z** fields. When you choose the button with the arrow icon, you are

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allowed to reposition an existing insertion point by selecting on the screen. You can also determine if you want the attribute text to appear upside down or backwards or not under the **Misc** heading. Here, you can also modify the attribute modes, which have been already defined. When you select a particular mode, say **Invisible**, a drop-down list is available in the corresponding field. This list displays two options, **Yes** and **No**. If you select **Yes**, the attribute is made invisible and if you select **No**, the attribute is not made invisible. To select a particular mode, you should choose **Yes** from their corresponding drop-down lists.

Figure 15-9 *The* ***PROPERTIES*** *palette to modify attributes*

Note

Remember that if you change the justification of the first attribute below which you have aligned the remaining attributes, the remaining attributes are not updated automatically. You will have to change the justifications of the remaining attributes individually to align them below the first attribute.

If you change the justification of the attributes from align to some other, the size of the text is also changed.

You can also use the ***CHANGE*** *command to edit the attribute objects using the Command line.*

INSERTING BLOCKS WITH ATTRIBUTES

The value of the attributes can be specified during block insertion, either at the command line or in the **Enter Attributes** dialog box. When you use the **INSERT** command or the **-INSERT** command to insert a block in a drawing (discussed earlier in Chapter 14, Working with Blocks), and after you have specified the insertion point, scale factors, and rotation angle, the **Enter Attributes** dialog box (Figure 15-10) is displayed, if the system variable **ATTDIA** is set to **1**.

The default value for **ATTDIA** is **0**, which disables the dialog box. The prompts and their default values, which you had specified with the attribute definition, are then displayed on the command line after you have specified the insertion point, scale, and rotation angle for the block to be inserted.

In the **Enter Attributes** dialog box, the prompts that were entered at the time of attribute definition in the dialog box are displayed with their default values in the corresponding fields. If an attribute has been defined with the **Constant** mode, it is not displayed in the dialog box because a constant attribute value cannot be edited. You can enter the attribute values in the fields located next to the attribute prompt. If no new values are specified, the default values are displayed. Eight attribute values are displayed at a time in the dialog box. If there are more attributes, they can be accessed by using the **Next** or **Previous** buttons. The block name is

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displayed at the top of the dialog box. After entering the new attribute values, choose the **OK** button. AutoCAD LT will place these attribute values at the specified location.

Figure 15-10 *The* ***Enter Attributes*** *dialog box*

Tip

It is more convenient to use the ***Edit Attributes*** *dialog box because you can view all the attribute values at a glance and can correct them before placing. Therefore, it is a good idea to set the* ***ATTDIA*** *value to 1, before you insert a block with attributes.*

Attributes can also be defined from the command line by setting the system variable ***ATTDIA*** *to 0 (default value). Now, when you use* ***the INSERT*** *command, AutoCAD LT does not display the* ***Enter Attributes*** *dialog box. Instead, AutoCAD LT prompts you to enter the attribute values for various attributes that have been defined in the block at the pointer input.*

Example 2 — *General*

In this example, you will insert the block (COMP) that was defined in Example 1. The following is the list of the attribute values for computers.

ITEM	MAKE	PROCESSOR	HD	RAM
Computer	Gateway	486-60	150 MB	16 MB
Computer	Zenith	486-30	100 MB	32 MB
Computer	IBM	386-30	80 MB	8 MB
Computer	Del	586-60	450 MB	64 MB
Computer	CAD-CIM	Pentium-90	100 Min	32 MB
Computer	CAD-CIM	Unknown	600 MB	Standard

1. Draw the floor plan drawing, as shown in Figure 15-11 (assume the dimensions).

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Figure 15-11 *Floor plan drawing for Example 2*

2. Set the system variable **ATTDIA** to 1. Use the **INSERT** command to insert the blocks. When you invoke the **INSERT** command, the **Insert** dialog box is displayed. Enter COMP in the **Name** edit box and choose **OK** to exit the dialog box. Select an insertion point on screen to insert the block. After you specify the insertion point, the **Enter Attributes** dialog box is displayed, where you can change the attribute values, if you need to, in the different edit boxes.

3. Repeat the **INSERT** command to insert other blocks, and define their attribute values, as shown in Figure 15-12.

4. Save the drawing for further use.

Figure 15-12 *The floor plan after inserting blocks and defining their attributes*

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ATTEXT COMMAND FOR ATTRIBUTE EXTRACTION

Command: ATTEXT

The **ATTEXT** command allows you to use the **Attribute Extraction** dialog box (Figure 15-13) for extracting the attributes. The information about the **File format**, **Template file**, and **Output** file must be entered in the dialog box to extract the defined attribute. Also, you must select the blocks whose attribute values you want to extract. If you do not specify a particular block, all the blocks in the drawing are used.

Figure 15-13 *The* ***Attribute Extraction*** *dialog box*

File Format Area

This area of the dialog box lets you select the file format. You can select either of the three radio buttons available in this area. They are **Comma Delimited File (CDF)**, **Space Delimited File (SDF)**, and **DXF Format Extract File (DXX)**. The format selection is determined by the application that you plan to use to process data. Both CDF and SDF formats can be used with the database software. All of these formats are printable.

Comma Delimited File (CDF). When you select this radio button, the extracted attribute information is displayed in a CDF format. Here, each character field is enclosed in single quotes, and the records are separated by a delimiter (comma by default). A CDF file is a text file with the extension *.txt*. Of the three formats available, this is the most cumbersome.

Space Delimited File (SDF). In SDF format, the records are of fixed width as specified in the template file. The records are separated by spaces and the character fields are not enclosed in single quotes. The SDF file is a text file with the extension *.txt*. This file format is the most convenient and easy to use.

DXF Format Extract File (DXX). If you select this file format, you will notice that the **Template File** button and edit box in the **Attribute Extraction** dialog box are not available. This is because extraction in this file format does not require any template. The file created by this

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option contains only block references, attribute values, and end-of-sequence objects. This is the most complex of the three file formats available and is related to programming. The extension of these files is *.dxx*.

Select Objects

Select the blocks with attributes whose attribute information you want to extract. You can use any object selection method. Once you have selected the blocks that you want to use for attribute extraction, right-click or press ENTER. The **Attribute Extraction** dialog box is redisplayed and the number of objects you have selected is displayed adjacent to **Number found**.

Template File

When you choose the **Template File** button, the **Template File** dialog box is displayed, where you are allowed to select a template file that has been defined already. After you have selected a template file, choose the **Open** button to return to the **Attribute Extraction** dialog box. The name of the selected file is displayed in the **Template File** edit box. The template file is saved with the extension of the file as *.txt*. The following are the fields that you can specify in a template file (the comments given on the right are for explanation only; they must not be entered with the field description):

BL:LEVEL	Nwww000	(Block nesting level)
BL:NAME	Cwww000	(Block name)
BL:X	Nwwwddd	(X coordinate of block insertion point)
BL:Y	Nwwwddd	(Y coordinate of block insertion point)
BL:Z	Nwwwddd	(Z coordinate of block insertion point)
BL:NUMBER	Nwww000	(Block counter)
BL:HANDLE	Cwww000	(Block's handle)
BL:LAYER	Cwww000	(Block insertion layer name)
BL:ORIENT	Nwwwddd	(Block rotation angle)
BL:XSCALE	Nwwwddd	(X scale factor of block)
BL:YSCALE	Nwwwddd	(Y scale factor of block)
BL:ZSCALE	Nwwwddd	(Z scale factor of block)
BL:XEXTRUDE	Nwwwddd	(X component of block's extrusion direction)
BL:YEXTRUDE	Nwwwddd	(Y component of block's extrusion direction)
BL:ZEXTRUDE	Nwwwddd	(Z component of block's extrusion direction)
Attribute tag		(The tag name of the block attribute)

The extract file may contain several fields. For example, the first field might be the item name and the second field might be the price of the item. Each line in the template file specifies one field in the extract file. Any line in a template file consists of the name of the field, the width of the field in characters, and its numerical precision (if applicable). For example:

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ITEM N015002
BL:NAME C015000

Where **BL:NAME** ------ Field name
Blankspaces ---- Blank spaces (must not include the tab character)
C ---------- Designates a character field
N ---------- Designates a numerical field
015 ---------- Width of field in characters
002 ---------- Numerical precision

BL:NAME or **ITEM** — Indicates the field names; can be of any length.

C — Designates a character field; that is, the field contains characters or it starts with characters. If the file contains numbers or starts with numbers, then C will be replaced by N. For example, **N015002**.

015 — Designates a field that is fifteen characters long.

002 — Designates the numerical precision. In this example, the numerical precision is 2, or two places following the decimal. The decimal point and the two digits following the decimal are **included in the field width**. In the next example, (000), the numerical precision, is not applicable because the field does not have any numerical value (the field contains letters only).

After creating a template file, when you choose the **Template File** button, the **Template File** dialog box is displayed, where you can browse and select a template file (Figure 15-14).

Note

You can put any number of spaces between the field name and the character C or N (ITEM N015002). However, you must not use the tab characters. Any alignment in the fields must be done by inserting spaces after the field name.

In the template file, a field name must not appear more than once. The template file name and the output file name must be different.

The template file must contain at least one field with an attribute tag name because the tag names determine which attribute values are to be extracted and from which blocks. If several blocks have different block names but the same attribute tag, AutoCAD LT will extract attribute values from all the selected blocks. For example, if there are two blocks in the drawing with the attribute tag PRICE, then when you extract the attribute values, AutoCAD LT will extract the value from both blocks (if both blocks were selected). To extract the value of an attribute, the tag name must match the field name specified in the template file. AutoCAD LT automatically converts the tag names and the field names to uppercase letters before making a comparison.

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Figure 15-14 *The* ***Template File*** *dialog box to select a template file*

Output File

When you choose the **Output File** button, the **Output File** dialog box is displayed. You can select an existing file here, if you want the extracted or output file to be saved as an existing file. You can enter a name in the **File Name** edit box in the **Output File** dialog box (Figure 15-15) and then choose the **Save** button, if you want to save the output file as a new file. By default, the output file has the same name as the drawing name. For example, a drawing named *Drawing1.dwg* will have an output file by the name *Drawing1.txt* by default. Once a name for the output file is specified, it is displayed in the **Output File** edit box in the **Attribute Extraction** dialog box. You can also enter the file name in this edit box. As discussed earlier, AutoCAD LT appends *.txt* file extension for CDF or SDF files and *.dxx* file extension for DXF files.

Note

You can also use the ***-ATTEXT*** *command to extract attributes using the command line. Here, you are prompted to specify a file format for the extract information and you can also select specific blocks to extract their attribute information. You can specify a template file using the* ***Select Template File*** *dialog box and an extract file using the* ***Create extract file*** *dialog box.*

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Figure 15-15 *The* ***Output File*** *dialog box to save an output file*

Example 3 — *General*

In this example, you will extract the attribute values that were defined in Example 2. Extract the values of MAKE, PROCESSOR, HD, and RAM. These attribute values must be saved in a **Tab Delimited File** named TEST and arranged, as shown in the following table.

COMP	1	Gateway	486-60	150 MB	16 MB	Computer
COMP	1	Zenith	486-30	100 MB	32 MB	Computer
COMP	1	IBM	386-30	80 MB	8 MB	Computer
COMP	1	Del	586-60	450 MB	64 MB	Computer
COMP	1	CAD-CIM	Pentium-90	100 Min	32 MB	Computer
COMP	1	CAD-CIM	Unknown	600 MB	Standard	Computer

1. Open the drawing you saved in Example 2.

2. Use the **ATTEXT** command to invoke the **Attribute Extraction** wizard. Select the **Select Objects** radio button from the **Select Drawing** page. Choose **Select Objects** button and select the objects in the current drawing whose attributes have to be extracted. After selecting, choose the **Next** button.

3. In the **Settings** page, clear both the check boxes of Xrefs and Nested blocks since there are no external references and nested blocks. Choose the **Next** button.

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4. In the **Use Template** page, select the **No template** radio button and choose **Next**.

5. In the **Select Attributes** page, select the check box named **COMP** from the **Blocks** area. In the attribute list, clear all the check boxes and then select the **MAKE**, **PROCESSOR**, **HD**, **RAM**, and **ITEM** check boxes. Choose the **Next** button.

6. The **View output** page shows the result of your query. Here, choose the view you want for the data to be displayed in an exported file. Choose the **Next** button.

7. In the **Save template** page, choose the **Next** button.

8. In the **Export** page, enter the name of the output file, **TEST**, and save it in **Tab Delimited File** format. Choose the **Finish** button.

9. You can view the resultant output file in the Notepad. Open the Notepad using the **Start** menu and then specify the location of the *Test.txt* file. The output file will be similar to the file shown at the beginning of Example 3.

CONTROLLING ATTRIBUTE VISIBILITY

Menu:	View > Display > Attribute Display
Command:	ATTDISP

The **ATTDISP** command allows you to change the visibility of all attribute values. Normally, the attributes are visible unless they are defined invisible by using the **Invisible** mode. The invisible attributes are not displayed, but they are a part of the block definition. You can select any one of the options of the **ATTDISP** command to turn the display of the attributes completely on or off. You can also select the **Normal** option, where the attributes that were created using the **Invisible** mode continue to be invisible. The options can be selected from the **View** > **Display** > **Attribute Display** cascading menu. You can also enter **ATTDISP** at the pointer input or at the command line. The prompt sequence for the command is given next.

Command: **ATTDISP**
Enter attribute visibility setting [Normal/ON/OFF] <Normal>: *Specify an option and press ENTER.*

When you select **ON**, all attribute values will be displayed, including the attributes that are defined with the **Invisible** mode. If you select **OFF**, all attribute values will become invisible. Similarly, if you select **N** (**Normal**), AutoCAD LT will display the attribute values the way they are defined, that is, the attributes that were defined invisible will stay invisible and the attributes that were defined visible are visible.

In Example 2, the RAM attribute was defined with the Invisible mode. Therefore, the RAM values are not displayed with the block. If you want to make the RAM attribute values visible (Figure 15-16), choose **On** from the **View > Display > Attribute Display** menu.

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Tip

*After you have defined the attribute values and saved them with the block definition, it may be a good idea to use the **Off** option of the **ATTDISP** command. By doing this, the drawing is simplified and also regeneration time is reduced.*

Figure 15-16 *Using the **ATTDISP** command to make the RAM attribute values visible*

EDITING BLOCK ATTRIBUTES

You can edit single block at a time using the dialog box or can edit more than one blocks at a time. Both the methods of editing the attribute value are discussed next.

Editing Attributes Using the Edit Attributes Dialog Box

Toolbar:	Modify II > Edit Attribute
Menu:	Modify > Object > Attribute > Single
Command:	ATTEDIT

The **ATTEDIT** command allows you to edit the block attribute values through the **Edit Attributes** dialog box. Invoke this command; AutoCAD LT prompts you to select the block whose values you want to edit. After selecting the block, the **Edit Attributes** dialog box (Figure 15-17) is displayed.

The dialog box is similar to the **Enter Attributes** dialog box and shows the prompts and the attribute values of the selected block. If an attribute has been defined with the **Constant** mode, it is not displayed in the dialog box because a constant attribute value cannot be edited. To make any changes, select the existing value and enter a new value in the corresponding edit box. After you have made the modifications, choose the **OK** button. The attribute values are updated in the selected block.

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Figure 15-17 *Editing attribute values using the* ***Edit Attributes*** *dialog box*

If a selected block has no attributes, AutoCAD LT will display the alert message **That block has no editable attributes**. Similarly, if the selected object is not a block, AutoCAD LT again displays the alert message **That object is not a block**.

Note

You cannot use the ***ATTEDIT*** *command to perform global editing of attribute values, or to modify position, height, or style of the attribute value. The global editing can be performed using the* ***-ATTEDIT*** *command, which is discussed in the next section.*

In this example, you will use the **ATTEDIT** command to change the attribute of the first computer

Example 4 — *General*

(150 MB to 2.1 GB), which is located in Room-1.

1. Open the drawing that was created in Example 2. The drawing has six blocks with attributes. The name of the block is COMP, and it has five defined attributes, one of them invisible. Zoom in so that the first computer is displayed on the screen (Figure 15-18).

2. Invoke the **ATTEDIT** command. AutoCAD LT will prompt you to select a block. Select the computer located in Room-1.

Figure 15-18 *Zoomed view of the first computer*

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AutoCAD LT will display the **Edit Attribute** dialog box (Figure 15-31) that shows the attribute prompts and the attribute values.

3. Edit the values, select **Apply**, and choose the **OK** button in the dialog box. When you exit the dialog box, the attribute values are updated.

Tip

*You can also edit attributes with the **FIND** command. This command can be invoked by entering **FIND** at the pointer input. When you invoke this command, the **Find and Replace** dialog box is displayed.*

*When you know that a set of attributes in a drawing may have to be changed in the future, you can make a group out of them by using the **GROUP** command. Then later you can choose the **Select objects** button in the **Find and Replace** dialog box and enter G at the **Select objects** prompt and enter the name of the group. All the objects in the group will get selected.*

Global Editing of Attributes

Menu:	Modify > Object > Attribute >Global
Command:	-ATTEDIT

The **-ATTEDIT** command allows you to edit the attribute values independently of the blocks that contain the attribute reference. For example, if there are two blocks, COMPUTER and TABLE, with the attribute value PRICE, you can globally edit this value (PRICE) independently of the block that references these values. You can also edit the attribute values one at a time. For example, you can edit the attribute value (PRICE) of the block TABLE without affecting the value of the other block, COMPUTER. When you enter the **-ATTEDIT** command, AutoCAD LT displays the following prompt:

Command: **-ATTEDIT**
Edit attributes one at a time? [Yes/No]: N
Performing global editing of attribute values

If you enter **N** at this prompt, it means that you want to do the global editing of the attributes. However, you can restrict the editing of attributes by block names, tag names, attribute values, and visibility of attributes on the screen.

Editing Visible Attributes Only

After you select global editing, AutoCAD LT will display the following prompt:

Edit only attributes visible on screen? [Yes/No] <Y>: **Y**

If you enter Y at this prompt, AutoCAD LT will edit only those attributes that are visible and displayed on the screen. The attributes might not have been defined with the Invisible mode, but if they are not displayed on the screen, they are not visible for editing. For example, if you zoom in, some of the attributes may not be displayed on the screen. Since the attributes are not displayed on the screen, they are invisible and cannot be selected for editing.

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Editing All Attributes

If you enter N at the previously mentioned prompt, AutoCAD LT flips from graphics to text screen and displays the following message on the screen:

Drawing must be regenerated afterwards.

Now, AutoCAD LT will edit all attributes even if they are not visible or displayed on the screen. Also, changes that you make in the attribute values are not reflected immediately. Instead, the attribute values are updated and the drawing is regenerated after you are done with the command.

Editing Specific Blocks

Although you have selected global editing, you can confine the editing of attributes to specific blocks by entering the block name at the prompt. For example

Enter block name specification <*>: **COMP**

When you enter the name of the block, AutoCAD LT will edit the attributes that have the given block (COMP) reference. You can also use the wild-card characters to specify the block names. If you want to edit attributes in all blocks that have attributes defined, press ENTER.

Editing Attributes with Specific Attribute Tag Names

Like blocks, you can confine attribute editing to those attribute values that have the specified tag name. For example, if you want to edit the attribute values that have the tag name MAKE, enter the tag name at the following AutoCAD LT prompt.

Enter attribute tag specification <*>: **MAKE**

When you specify the tag name, AutoCAD LT will not edit attributes that have a different tag name, even if the values being edited are the same. You can also use the wild-card characters to specify the tag names. If you want to edit attributes with any tag name, press ENTER.

Editing Attributes with a Specific Attribute Value

Like blocks and attribute tag names, you can confine attribute editing to a specified attribute value. For example, if you want to edit the attribute values that have the value 100 MB, enter the value at the following AutoCAD LT prompt.

Enter attribute value specification <*>: **100MB**

When you specify the attribute value, AutoCAD LT will not edit attributes that have a different value, even if the tag name and block specification are the same. You can also use the wild-card characters to specify the attribute value. If you want to edit attributes with any value, press ENTER.

Sometimes the value of an attribute is null, and these values are not visible. If you want to select the null values for editing, make sure you have not restricted the global editing to visible attributes.

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To edit the null attributes, enter \ at the following prompt.

Enter attribute value specification <*>: \

After you enter this information, AutoCAD LT will prompt you to select the attributes. You can select the attributes by selecting individual attributes or by using one of the object selection options (Window, Crossing, or individually).

Select Attributes: *Select the attribute values parallel to the current UCS only.*

After you select the attributes, AutoCAD LT will prompt you to enter the string you want to change and the new string. A string is a sequence of consecutive characters. It could also be a portion of the text. AutoCAD LT will retrieve the attribute information, edit it, and then update the attribute values.

Enter string to change: *Enter the value which is to be modified.*
Enter new string: *Enter the new value.*

The following is the complete command prompt sequence of the **-ATTEDIT** command. It is assumed that the editing is global and for visible attributes only.

Command: -**ATTEDIT**
Edit attributes one at a time? [Yes/No] <Y>: N
Performing global editing of attribute values.
Edit only attributes visible on screen? [Yes/No] <Y>: N
Drawing must be regenerated afterwards.
Enter block name specification <*>: Enter
Enter attribute tag specification <*>: Enter
Enter attribute value specification <*>: Enter
Enter string to change: *Enter the value to be modified.*
Enter new string: *Enter the new value.*

Note

*AutoCAD LT regenerates the drawing at the end of the command automatically unless system variable **REGENAUTO** is off, which controls automatic regeneration of the drawing.*

*If you select an attribute defined with the **Constant** mode using the **-ATTEDIT** command, prompt displays 0 found, since attributes with constant mode is uneditable.*

Example 5 *General*

In this example, you will use the drawing from Example 2 to edit the attribute values that are **highlighted** in the following table. The tag names are given at the top of the table (ITEM, MAKE, PROCESSOR, HD, RAM). The RAM values are invisible in the drawing.

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	ITEM	MAKE	PROCESSOR	HD	RAM
COMP	Computer	Gateway	486-60	150 MB	**16 MB**
COMP	Computer	Zenith	486-30	100 MB	**32 MB**
COMP	Computer	IBM	386-30	80 MB	**8 MB**
COMP	Computer	Del	586-60	450 MB	**64 MB**
COMP	Computer	**CAD-CIM**	Pentium-90	100 Min	**32 MB**
COMP	Computer	**CAD-CIM**	**Unknown**	600 MB	Standard

Make the following changes in the **highlighted** attribute values (Figure 15-19).

Figure 15-19 *Using* ***-ATTEDIT*** *to change the attribute values*

1. Change Unknown to Pentium.

2. Change CAD-CIM to Compaq.

3. Change MB to Meg for all attribute values that have the tag name RAM. (No changes should be made to the values that have the tag name HD.)

The following is the prompt sequence to change the attribute value from **Unknown** to **Pentium**.

1. Enter the **-ATTEDIT** command at the Command prompt. At the next prompt, enter **N** and press ENTER.

   Command: **-ATTEDIT**
   Edit attributes one at a time? [Yes/No] <Y>: **N**
   Performing global editing of attribute values.

2. We want to edit only those attributes that are visible on the screen, so press ENTER at the following prompt:

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Edit only attributes visible on screen? [Yes/No] <Y>: Enter

3. As shown in the table, the attributes belong to a single block, COMP. In a drawing, there could be more blocks. To confine the attribute editing to the COMP block only, enter the name of the block (COMP) at the next prompt.

   Enter block name specification <*>: **COMP**

4. At the next two prompts, enter the attribute tag name and the attribute value specification. When you enter these two values, only those attributes that have the specified tag name and attribute value will be edited.

   Enter attribute tag specification<*>: **Processor**
   Enter attribute value specification<*>: **Unknown**

5. Next, AutoCAD LT will prompt you to select attributes. Use any object selection option to select all the blocks. AutoCAD LT will search for the attributes that satisfy the given criteria (attributes belong to the block COMP, the attributes have the tag name Processor, and the attribute value is Unknown). Once AutoCAD LT locates such attributes, they will be highlighted.

6. At the next two prompts, enter the string you want to change, and then enter the new string.

   Enter string to change: **Unknown**
   Enter new string: **Pentium**

7. The following is the Command prompt sequence to change the make of the computers from **CAD-CIM** to **Compaq**.

   Command: **-ATTEDIT**
   Edit attributes one at a time? [Yes/No] <Y>: **N**
   Performing global editing of attribute values.
   Edit only attributes visible on screen? [Yes/No] <Y>: Enter
   Enter block name specification <*>: **COMP**
   Enter attribute tag specification <*>: **MAKE**
   Enter attribute value specification <*>: Enter
   Select Attributes: *Use any selection method to select the attributes.*
   n attributes selected.
   Select Attributes: Enter
   Enter string to change: **CAD-CIM**
   Enter new string: **Compaq**

8. The following is the Command prompt sequence to change **MB** to **Meg**.

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Command: **-ATTEDIT**
Edit attributes one at a time? [Yes/No] <Y>: **N**
Performing global editing of attribute values.

At the next prompt, you must enter **N** because the attributes you want to edit (tag name, RAM) are not visible on the screen.

Edit only attributes visible on screen? [Yes/No] <Y>: **N**
Drawing must be regenerated afterwards.
Enter block name specification <*>: **COMP**

At the next prompt, about the tag specification, you must specify the tag name because the text string MB also appears in the hard drive size (tag name, HD). If you do not enter the tag name, AutoCAD LT will change all MB attribute values to Meg.

Enter attribute tag specification <*>: **RAM**
Enter attribute value specification <*>:
n Attributes selected
Enter string to change: **MB**
Enter new string: **Meg**

9. Choose **On** from the **View > Display > Attribute Display** menu to display the invisible attributes on the screen.

Note
*You can also use the **-ATTEDIT** command to edit the attribute values individually. Attribute value, string, position, height, angle, layer, and color can be changed using this command.*

Example 6 *General*

In this example, you will use the drawing in Example 5 to edit the attributes individually (Figure 15-20). Make the following changes in the attribute values.

a. Change the attribute value 100 Min to 100 MB.
b. Change the height of all attributes with the tag name RAM to 0.075 units.

1. Load the drawing that you had saved in Example 5.

2. At the AutoCAD LT Command prompt, enter the **-ATTEDIT** command. The following is the command prompt sequence to change the value of 100 Min to 100 MB.

Command: **-ATTEDIT**
Edit attributes one at a time? [Yes/No] <Y>: Enter
Enter block name specification <*>: **COMP**
Enter attribute tag specification <*>: Enter
Enter attribute value specification <*>: Enter

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Figure 15-20 *Using* ***-ATTEDIT*** *to change the attribute values individually*

Select Attributes: *Select the attribute.*
Enter an option [Value/Position/Height/Angle/Style/Layer/Color/Next] <N>: **V**
Enter type of value modification [Change/Replace] <R>: **C**
Enter string to change: \ **Min**
Enter new string: **100MB**

When AutoCAD LT prompts **Enter string to change:**, enter the characters you want to change. In this example, the characters **Min** are preceded by a space. If you enter a space, AutoCAD LT displays the next prompt, **Enter new string**. If you need a leading blank space, the character string must start with a backslash (\), followed by the desired number of blank spaces.

3. To change the height of the attribute text, enter the **-ATTEDIT** command as just shown. When AutoCAD LT displays the following prompt, enter **H** for height.

   Enter an option [Value/Position/Height/Angle/Style/Layer/Color/Next] <N>: **H**
   Specify new height <current>: **0.075**

   After you enter the new height and press ENTER, AutoCAD LT will change the height of the text string that has the **X** mark. AutoCAD LT will then repeat the last prompt. Use the **Next** option to move the **X** mark to the next attribute. To change the height of other attribute values, repeat these steps. The drawing after editing the attributes is shown in Figure 15-20.

4. Repeat Steps 2 and 3 to change the height of the other attribute values.

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Tip

When you are defining attributes and have certain attributes whose values are not known at that time, you should enter AAAA or something similar. Later on you can replace such text with the values that you have obtained, using the ***ATTEDIT*** *or* ***-ATTEDIT*** *commands. This is easier than adding an attribute later after the block has already been defined.*

INSERTING TEXT FILES IN THE DRAWING

Toolbar:	Draw > Multiline Text
	Text > Multiline Text
Menu:	Draw > Text > Multiline Text
Command:	MTEXT

After you have extracted attribute information into a file, you may want to insert this text into a drawing. You can insert this text file in a drawing by using the **Import Text** option that is displayed when you right-click in the text window of the **In-Place Text Editor**. The **In-Place Text Editor** is displayed using the **MTEXT** command.

After you have invoked the **MTEXT** command, AutoCAD LT prompts you to enter the insertion point and other corner of the paragraph text box, within which the text file will be placed. After you specify these points, the **In-Place Text Editor** is displayed. To insert the text file *Test.txt* (created in Example 3), right-click in the text window of the **In-Place Text Editor** to display the shortcut menu. In this shortcut menu, choose **Import Text**. AutoCAD LT displays the **Select File** dialog box.

In the **Select File** dialog box, you can select the text file **Test** from the list box and then choose the **Open** button. The imported text is displayed in the text window of the **In-Place Text Editor** (Figure 15-21). Note that only the ASCII files are properly interpreted. Now choose the **OK** button to get the imported text in the selected area on the screen (Figure 15-22).

You can also use the **In-Place Text Editor** to change the text style, height, direction, width, rotation, line spacing, and attachment.

Figure 15-21 *The* ***In-Place Text Editor*** *displaying the imported text*

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Block Name	Count	MAKE	PROCESSOR	HD	RAM	ITEM
COMP	1	Gateway	486–60	150MB	16MB	Computer
COMP	1	Zenith	486–30	100MB	32MB	Computer
COMP	1	IBM	386–30	80MB	8MB	Computer
COMP	1	Del	586–60	450MB	64MB	Computer
COMP	1	CAD–CIM	Pentium–90	100Min	32MB	Computer
COMP	1	CAD–CIM	Unknown	600MB	Standard	Computer

Figure 15-22 *Imported text file on the screen*

Self-Evaluation Test

Answer the following questions, and then compare your answers to those given at the end of this chapter.

1. Like a constant attribute, the **Preset** attribute cannot be edited. (T/F)

2. For tag names, any lowercase letters are automatically converted to uppercase. (T/F)

3. You can use the **ATTEDIT** command to modify the justification, height, or style of the attribute value. (T/F)

4. If you select the **On** option of the **ATTDISP** command, even the attributes defined with the **Invisible** mode are displayed. (T/F)

5. The entry in the **Value** edit box of the **Attribute Definition** dialog box defines the _____ of the specified attribute.

6. If you have selected the **Align** option from the **Justification** drop-down list, in the **Text Options** area of the **Attribute Definition** dialog box, the **Height** and **Rotation** edit boxes are _____.

7. You can use the _____, and _____ commands or the **PROPERTIES** palette to edit text or attribute definitions.

8. The default value of the **ATTDIA** variable is _____, which disables the dialog box.

9. In the _____ File, the records are not separated by a comma and the character fields are not enclosed in single quotes.

10. The _____ command is ued to redefine an existing block with attributes in a drawing and once redefined, all the copies of the block in the drawing are updated.

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Review Questions

Answer the following questions.

1. If you do not enter anything in the **Prompt** edit box, the entry made in the **Tag** edit box is used as the prompt. (T/F)

2. You can also use the **Find and Replace** dialog box to modify the attribute values. (T/F)

3. The **Constant** mode lets you define an attribute that has a constant value and cannot be edited later. (T/F)

4. You can use ? and * in the string value. When these characters are used in string values, AutoCAD LT does not interpret them as wild-card characters. (T/F)

5. The template file name and the output file name can be the same. (T/F)

6. Not selecting any of the check boxes in the **Mode** area of the **Attribute Definition** dialog box displays all the prompts at the command line and the values will be visible on the screen. This is also referred to as which mode?

   (a) Formal (b) Normal
   (c) Abnormal (d) None of the above

7. Which of the following system variables when set to 0 will suppress the display of prompts for new values?

   (a) **ATTDIA** (b) **ATTREQ**
   (c) **ATTMODE** (d) **ATTDEF**

8. Selecting the **Off** option of which command turns off the visibility of all attribute values?

   (a) **ATTDISP** (b) **ATTEXT**
   (c) **ATTEDIT** (d) **ATTDEF**

9. AutoCAD LT regenerates the drawing at the end of the **-ATTEDIT** command, unless which of the following is turned off?

   (a) **AUTOSNAP** (b) **REGENAUTO**
   (c) **ATTDIA** (d) **ATTMODE**

10. If you need a leading blank space in the string to be changed, the character string must start with which one of the following characters?

   (a) space ( ) (b) backlash (\)
   (c) asterisk (*) (d) colon (:)

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11. You can insert the text file in the drawing by choosing the _____ button in the **In-Place Text Editor** displayed when using the **MTEXT** command.

12. The function of the **Preset** option is _____.

13. If you select the **Constant** check box in the **Mode** area of the **Attribute Definition** dialog box, the **Prompt** edit box is _____.

14. You should select the _____ check box in the **Attribute Definition** dialog box to automatically place the subsequent attribute text just below the previously defined attribute.

15. The attribute extract file is saved as a _____ file.

Exercises

Exercise 1

Electronics

In this exercise, you will define the following attributes for a resistor and then create a block using the **BLOCK** command. The name of the block is RESIS.

Mode	Tag name	Prompt	Default value
Verify	RNAME	Enter name	RX
Verify	RVALUE	Enter resistance	XX
Verify, Invisible	RPRICE	Enter price	00

1. Draw the resistor, as shown in Figure 15-23.

2. Enter **ATTDEF** at the AutoCAD LT Command prompt to invoke the **Attribute Definition** dialog box.

3. Define the attributes, as shown in the preceding table, and position the attribute text as shown in Figure 15-23.

4. Use the **BLOCK** command to create a block. The name of the block is RESIS, and the insertion point of the block is at the left end of the resistor. When you select the objects for the block, make sure you also select the attributes.

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Figure 15-23 *Drawing of a resistor for Exercise 1*

Exercise 2

Electronics

In this exercise, you will use the **INSERT** command to insert the block that was defined in Exercise 1 (RESIS). The following is the list of the attribute values for the resistances in the electric circuit.

RNAME	RVALUE	RPRICE
R1	35	0.32
R2	27	0.25
R3	52	0.40
R4	8	0.21
RX	10	0.21

1. Draw the electric circuit diagram, as shown in Figure 15-24 (assume the dimensions).

2. Set the system variable **ATTDIA** to 1. Use the **INSERT** command to insert the blocks, and define the attribute values in the **Enter Attributes** dialog box.

3. Repeat the **INSERT** command to insert other blocks, and define their attribute values as given in the table. Save the drawing as *attexr2.dwg* (Figure 15-25).

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Figure 15-24 *Electric circuit diagram without resistors for Exercise 2*

Figure 15-25 *Electric circuit diagram with resistors for Exercise 2*

Exercise 3

Electronics

In this example, you will extract the attribute values that were defined in Exercise 2. Extract the values of RNAME, RVALUE, and RPRICE. These attribute values must be saved in a Tab Delimited File format named **RESISLST** and arranged, as shown in the following table.

RESIS	1	R1	35	0.32
RESIS	1	R2	27	0.25
RESIS	1	R3	52	0.40
RESIS	1	R4	8	0.21
RESIS	1	RX	10	0.21

1. Load the drawing **ATTEXR2** that you saved in Exercise 2.

2. Use the **ATTEXT** command to invoke the **Attribute Extraction** wizard, and select the **Select Objects** radio button. Now, choose **Select Objects** button and select the objects in the current drawing, whose attributes have to be extracted. Now, choose **Next** in the wizard.

3. On the **Settings** page, clear both the check boxes of **Xrefs** and **Nested blocks** and choose **Next**.

4. On the **Use Template** page, select the **No Template** radio button and choose **Next**.

5. On the **Select Attribute** page, under the Block Name heading, select the check box named **RESIS**. In the attribute list, clear all the check boxes and select **RNAME**, **RVALUE**, and **RPRICE** check boxes. Choose **Next**.

6. **View output** page shows the result of your query. Here, choose the view you want the data to be displayed in the exported file. Select the Next button.

7. Ignore the **Save template** page and choose **Next**.

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8. On the **Export** page, enter the name of the output file, **RESISLST**, and save it in **Tab Delimited File** format. Choose **Finish**. Use Notepad to list the output file, **RESISLST**. The output file will be similar to the file shown at the beginning of Exercise 3.

Exercise 4

Electronics

In this exercise, you will use the **ATTEDIT** command to change the attributes of the resistances that are highlighted in the following table. You will also extract the attribute values and insert the text file in the drawing.

1. Load the drawing **ATTEXR2** that was created in Exercise 2. The drawing has five resistances with attributes. The name of the block is RESIS, and it has three defined attributes, one of them invisible.

2. Use the **ATTEDIT** command to edit the values that are **highlighted** in the following table.

RESIS	R1	**40**	0.32
RESIS	R2	**29**	0.25
RESIS	R3	52	**0.45**
RESIS	R4	8	**0.25**
RESIS	**R5**	10	0.21

3. Extract the attribute values, and write the values to a text file.

4. Use the **MTEXT** command to insert the text file in the drawing.

Exercise 5

Electronics

Use the information given in Exercise 3 to extract the attribute values, and write the data to the output file. The data in the output file should be Comma Delimited CDF. Use the **ATTEXT** and **-ATTEXT** commands to extract the attribute values.

Exercise 6

Electronics

In this exercise, you will create the blocks with the required attributes. Next, draw the circuit diagram shown in Figure 15-26 and then extract the attributes to create a bill of materials.

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Figure 15-26 *Drawing of the circuit diagram for Exercise 6*

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Exercise 7

Electronics

In this exercise, you will create the blocks with the required attributes. Next, draw the circuit diagram shown in Figure 15-27 and then extract the attributes to create a bill of materials.

Figure 15-27 *Drawing for Exercise 7*

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Answers to Self-Evaluation Test

1 - F, **2** - T, **3** - T, **4** - T, **5** - default value, **6** - disabled, **7** - **EATTEDIT**, **CHANGE**, **8** - 0, **9** - Space Delimited, **10** - **ATTREDEF**

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