Chapter 17

Defining Block Attributes

Learning Objectives

After completing this chapter, you will be able to:

- *Understand block attributes*
- *Create annotative block attributes*
- *Edit attribute tag names*
- *Insert blocks with attributes and assign values to attributes*
- *Extract attribute values from inserted blocks*
- *Control attribute visibility*
- *Perform editing of attributes*
- *Insert a text file in a drawing to create Bill Of Materials*

Key Terms

- *Attribute Definition*
- *Block Attribute Manager*
- *Data Extraction*
- *ATTDISP*
- *EATTEDIT*
- *-ATTEDIT*
- *REFEDIT*

UNDERSTANDING ATTRIBUTES

AutoCAD 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 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 attribute. AutoCAD references the attributes with a block through tag names.

Before assigning attributes to a block, you must create an attribute definition by using the **Define Attributes** tool. 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 a block, AutoCAD 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 the **Extract Data** tool. 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 tools in the **Edit Attribute** drop-down.

DEFINING ATTRIBUTES

Ribbon: Insert > Block Definition > Define Attributes **Command:** ATTDEF/ATT



When you invoke the **Define Attributes** tool, the **Attribute Definition** dialog box will be displayed, as shown in Figure 17-1. The block attributes can be defined using this dialog box. When creating an attribute definition, you must define the Mode, Attribute, Insertion Point, and Text Settings for each attribute. All these information can be entered in the dialog box. The following is the description of each area of the **Attribute Definition** dialog box.

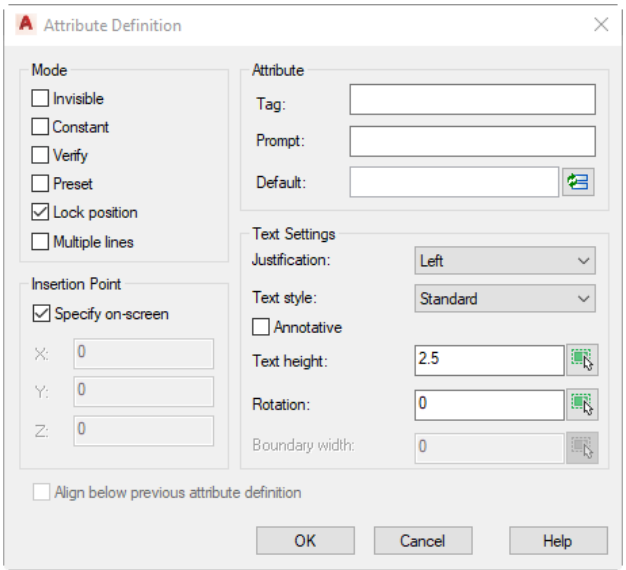

Figure 17-1 *The* ***Attribute Definition*** *dialog box*

Mode Area

The **Mode** area of the **Attribute Definition** dialog box has six check boxes: **Invisible**, **Constant**, **Verify**, **Preset**, **Lock position**, and **Multiple lines**. These check boxes determine the display and edit features of the block attributes. For example, if you select the **Invisible** check box, the attribute becomes invisible which means, 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 allows you to create an attribute that is not visible on the screen. Select this check box if you want the attribute to be invisible.

Constant

This option allows you to create an attribute that has a fixed value and cannot be changed after block insertion.

Verify

This option allows you to verify the attribute value you have entered when inserting a block by asking you twice for the data.

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.

Lock position

This check box is selected by default. As a result, the position of the attributes with respect to the block object is locked. In case of non-dynamic blocks, if the attributes are created with the **Lock position** check box cleared, then a separate grip will be displayed on that attribute. This grip can be used to modify the location of the attribute. When the dynamic blocks are created with the **Lock position** check box cleared, the separate grip does not get displayed on the attribute. Besides this, the attribute cannot be manipulated by the parameters and action.

Multiple lines

Select this check box to specify that the default value of the attribute may be a single-line or multiple line text.

Attribute Area

The **Attribute** area, refer to Figure 17-2, of the **Attribute Definition** dialog box has three edit boxes: **Tag**, **Prompt**, and **Default**, where you can enter values. The three edit boxes have been discussed next.

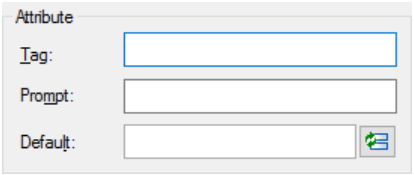

Figure 17-2 *The **Attribute** area of the **Attribute Definition** dialog box*

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. The tag names specified in lowercase letters are automatically converted into uppercase when displayed. The tag name cannot be null. Also, it must not contain any blank spaces.

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 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.

Default

The entry in the **Default** edit box defines the default value of the specified attribute. If you do not enter a value, the attribute takes the value entered in the **Tag** edit box. The entry of a value is optional.

Insert Field



This button is used 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.

Multiline Editor



This button is displayed in place of the **Insert Field** button when the **Multiple lines** check box is selected from the **Mode** area. Choose this button and click in the drawing area to invoke the **Text Formatting** toolbar on the screen. You can enter the text that you want to assign as the default value of the attribute in the text window.



Note

*The **Text Formatting** toolbar displayed here has some commonly used options only. The full **Text Formatting** toolbar can be displayed by changing the value of the **ATTIPE** system variable to 1.*

Insertion Point Area

The **Insertion Point** area, refer to Figure 17-3, of the **Attribute Definition** dialog box allows you to 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 if not selected by default. Now, set the parameters in all the other fields and areas and then choose the **OK** button; you will be prompted to select the start point of the attribute.

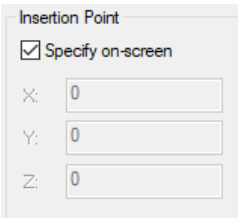

***Figure 17-3** The **Insertion Point** area*

Just below the **Insertion Point** area of the dialog box, their 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 Settings** areas of the dialog box are not available and AutoCAD 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 Settings Area

The **Text Settings** area of the **Attribute Definition** dialog box allows you to control the justification, text style, annotation property, height, rotation, and boundary width of the attribute text, refer to Figure 17-4. To set the text justification, select a justification type from the **Justification** drop-down list. The default option is **Left** for the single line text and **Top left** for the multiline text. 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**. To write annotative attributes, select the **Annotative** text style from the **Text style** drop-down list. Select the **Annotative** check box to assign the annotative property to the attribute text. The annotative attributes can be attached to both the annotative or non-annotative blocks. You can specify the text height and text rotation in the **Text height** and **Rotation** edit boxes, respectively. You can also define the text height by choosing the **Text Height** button. When you choose this button, AutoCAD temporarily exits the **Attribute Definition** dialog box and allows you to enter the height value by selecting points on the screen or from the command prompt. Once the height on the screen is defined, the **Attribute Definition** dialog box will reappear and the defined text height will be 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 prompt. The **Boundary width** edit box is used to specify the maximum width for the multiline text. If the width of the text line is more than this value, the text will automatically go to the second line. This edit box will be available only when you select the **Multiple lines** check box from the **Mode** area.

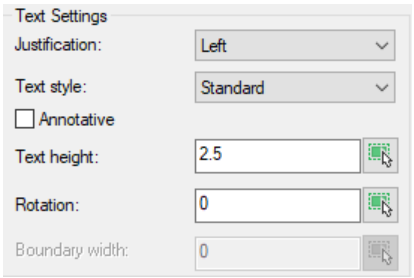

Figure 17-4 *The **Text Settings** area of the **Attribute Definition** dialog box*



Note

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

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

*3. If you have selected the **Align** option from the **Justification** drop-down list, the **Insertion Point** area, the **Text height**, and the **Rotation** edit boxes in the **Text Settings** area will be disabled.*

*4. If you have selected the **Fit** option from the **Justification** drop-down list, the **Insertion Point** area and the **Rotation** edit box are disabled. To specify the location of the attribute, choose the **OK** button from the dialog box; you will be prompted to specify the first and second endpoints of the text baseline. Specify the point; the text will fit between the two endpoints that you specify.*

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



Note

*You can use the **-ATTDEF** command to display the dynamic prompt in the drawing area. The options in the **Attribute Definition** dialog box are available through the dynamic prompt too, refer to Figure 17-5.*

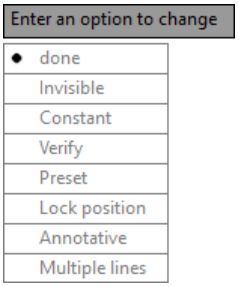

***Figure 17-5** Dynamic prompt for the **-ATTDEF** command*

Example 1 — Attribute Definition

In this example, you will define the following attributes for a computer and then create a block using the **Create Block** tool. 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 sketch of a computer and convert it into a block, as shown in Figure 17-6. Assume the dimensions, or measure the dimensions of the computer you are using for AutoCAD.

2. Choose the **Define Attributes** tool from the **Block Definition** panel in the **Insert** tab; the **Attribute Definition** dialog box is displayed.

3. Define the first attribute shown in the preceding table. Select the **Constant** check box in the **Mode** area. In the **Tag** edit box, enter the tag name, **ITEM**. Similarly, enter **COMPUTER** in the **Default** edit box. Note that the **Prompt** edit box is not available because the mode is constant.

4. In the **Insertion Point** area, select the **Specify on-screen** check box to define the text insertion point, if not selected by default.

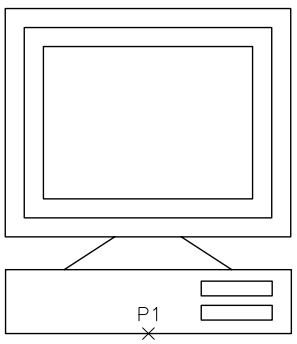

***Figure 17-6** Drawing for Example 1*

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

6. Choose the **OK** button once you have entered information in the **Attribute Definition** dialog box. Select a point below the insertion base point (P1) of the computer to place the text.

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. Now, choose the **OK** button. AutoCAD places the attribute text just below the previous attribute text.

8. Similarly, define the remaining attributes also, refer to Figure 17-7.

9. Now, use the **Create Block** tool to create a block. Make sure that the **Retain** radio button is selected in the **Objects** area. The name of the block is **COMP**, and the insertion point of the block is P1, midpoint of the base. When you select objects for the block, make sure you also select attributes.

10. Insert the block created. You will notice that the order of prompts is the same as the order of attributes selection.

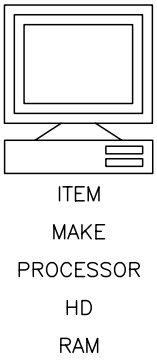

Figure 17-7 *Attributes defined below the computer drawing*

EDITING ATTRIBUTE DEFINITION

Menu Bar: Modify > Object > Text > Edit **Command:** TEXTEDIT



You can edit the text and attribute definitions before you define a block, using the **Edit** tool from the **Modify > Object > Text** menu. After invoking this tool, AutoCAD will prompt you to **Select an annotation object or [Undo Mode]**. If you select an attribute, then the **Enhanced Attribute Editor** dialog box is displayed, as shown in Figure 17-8.

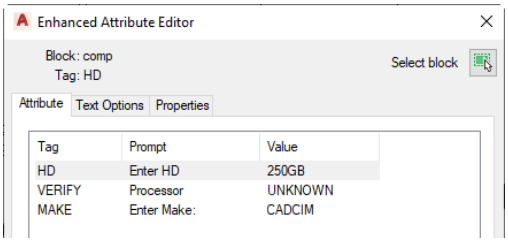

Figure 17-8 *The **Enhanced Attribute Editor** dialog box*

You can also invoke the **Enhanced Attribute Editor** dialog box by double-clicking on the attribute definition. 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. If you have selected the **Multiple** option in the mode of **TEXTEDIT** command, AutoCAD will continue to prompt you to select an annotation object. If you have finished editing and do not want to select another attribute object to edit, press ENTER to exit the command.

Using the PROPERTIES Palette

The **Properties** tool has already been discussed in Chapter 4. It can also be used to edit defined attributes. Select the attribute to be modified and right-click to display a shortcut menu. Next, choose the **Properties** option from the shortcut menu; the **PROPERTIES** palette will be displayed, as shown in Figure 17-9. You will notice that **Attribute Definition** is displayed in the drop-down located at the top of the palette. Also, you will find that all properties of the selected attribute are displayed under four headings. They are **General**, **3D Visualization**, **Text**, 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, annotation scale, text height, rotation angle, width factor, and make the angle values of the selected attribute oblique.

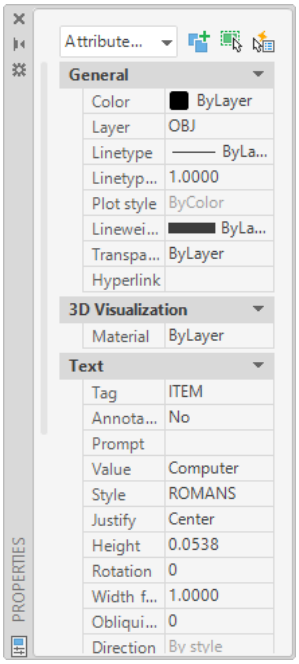

***Figure 17-9** The **PROPERTIES** palette to modify attributes*

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, for example **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 becomes invisible and if you select **No**, the attribute becomes visible. To select a particular mode, you should choose **Yes** from their corresponding drop-down lists.

INSERTING BLOCKS WITH ATTRIBUTES

The value of the attributes can be specified during block insertion, either at the command prompt or in the **Edit Attributes** dialog box, if the system variable **ATTDIA** is set to **1**. When you use the **Insert** tool or the **-INSERT** command to insert a block in a drawing (discussed earlier in Chapter 16), and after you have specified the insertion point, scale factors, and rotation angle, the **Edit Attributes** dialog box will be displayed, as shown in Figure 17-10.

If the value of the system variable **ATTDIA** is set to **0**,the **Edit Attributes** dialog box will be disabled and the prompts and their default values, which you had specified with the attribute definition, are then displayed at the command prompt after you have specified the insertion point, scale, and rotation angle for the block to be inserted.

In the **Edit 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. If there are more attributes, they can be accessed by using the **Next** or **Previous** button. The block name is

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

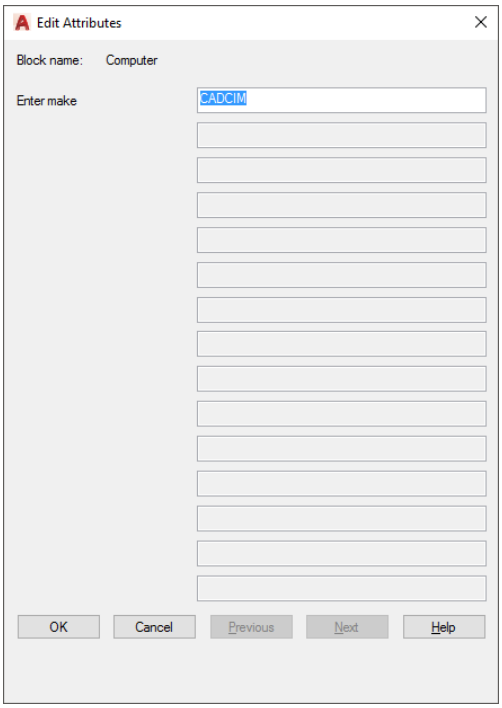

Figure 17-10 *The* ***Edit Attributes*** *dialog box*



Tip

1. It is 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 (if not already set as 1), before you insert a block with attributes.*

2. Attributes can also be defined from the command prompt by setting the system variable ***ATTDIA*** *to 0. Now, when you choose the* ***Insert*** *tool, AutoCAD does not display the* ***Edit Attributes*** *dialog box. Instead, AutoCAD prompts you to enter the attribute values for various attributes that have been defined in the block at the pointer input.*

Example 2

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	Dell	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 shown in Figure 17-11 (assume the dimensions).

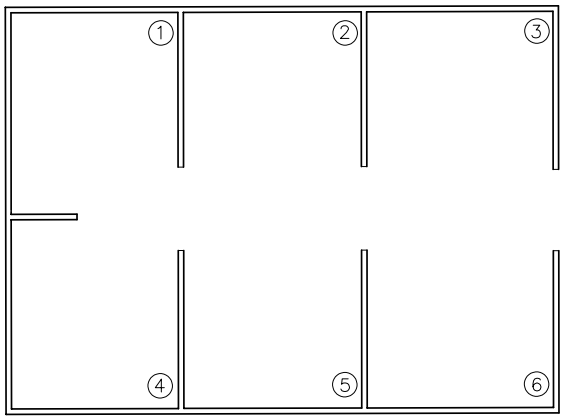

Figure 17-11 *Floor plan drawing for Example 2*

2. Type INSERT in command line to invoke the BLOCKS palette. Enter **COMP** in the **Filter** area. Now select block and choose an insertion point on the screen to insert the block. After you specify the insertion point, the **Edit Attributes** dialog box is displayed. In this dialog box, you can specify the attribute values, if you need to, in the different edit boxes.

3. Similarly you can insert other blocks and define their attribute values, as shown in Figure 17-12.

4. Save the drawing for further use.

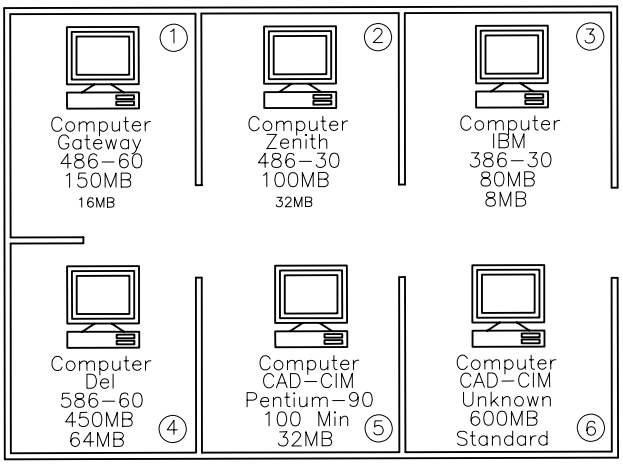

Figure 17-12 *The floor plan after inserting blocks and defining their attributes*

MANAGING ATTRIBUTES

Ribbon: Insert > Block Definition > Manage Attributes **Command:** BATTMAN
Menu Bar: Modify > Object > Attribute > Block Attribute Manager

The **Manage Attributes** tool allows you to manage attribute definitions for blocks in the current drawing. Choose this tool; the **Block Attribute Manager** dialog box will be displayed, as shown in Figure 17-13. If the attribute is edited in the **Constant** mode, then changes in the default value of the attribute in the existing drawing can be seen. If the mode is other than the **Constant** mode, the changes in the attribute can only be seen for new block insertions.

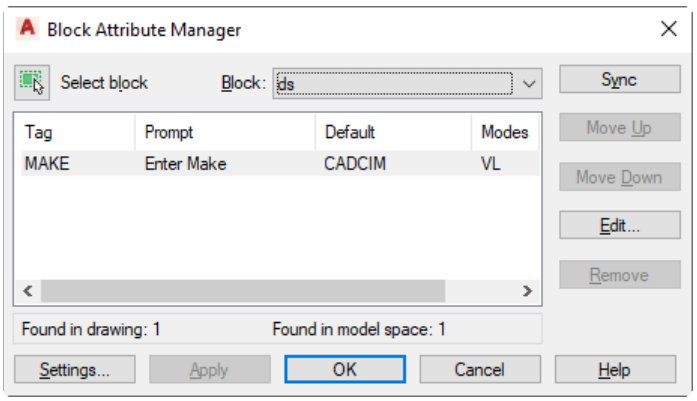

Figure 17-13 *The* ***Block Attribute Manager*** *dialog box*

When you invoke this tool, AutoCAD displays the number of blocks that exist in the current drawing below the list box in the dialog box. The block can be in the model space or in the layout. The names of the existing blocks in the current drawing are displayed in the **Block** drop-down list. Attributes of the selected block are displayed in the attribute list. By default, the **Tag**, **Prompt**, **Default**, **Modes**, and **Annotative** attribute properties are displayed in the attribute list. You can edit the attribute definitions in blocks, remove attributes from blocks, and change the order in which you were prompted for attribute values while inserting the block.

Select block

This button allows you to select a block from the drawing area whose attributes you want to edit. When you choose the **Select block** button, the dialog box closes temporarily until you select block from the drawing or cancel by pressing ESC. Once you have selected the block, the dialog box is displayed again. The name of the block you have selected is displayed in the **Block** drop-down list. If you modify the attributes of a block and then select a new block before saving the attribute changes, you will be prompted to save the changes before selecting another block through an alert message box, as shown in Figure 17-14.

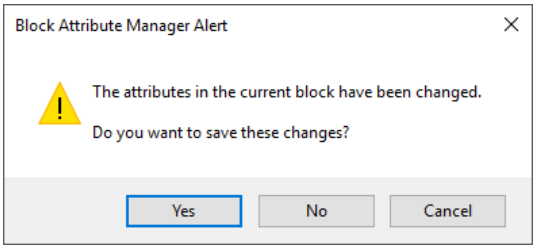

***Figure 17-14** The **Block Attribute Manager Alert** message box*

Block Drop-down List

This drop-down list in the **Block Attribute Manager** dialog box allows you to choose the block whose attributes you want to modify. It lists all the blocks in the current drawing that have attributes.

Sync

This button of the **Block Attribute Manager** dialog box allows you to update all instances of the selected block with the attribute properties currently defined. This does not affect any of the values assigned to attributes in each block when attributes were defined. On choosing this button, you can update attributes in all block references in the current drawing with the changes you made to the block definition. For example, you may have used the **Block Attribute Manager** dialog box to modify attribute properties of several block definitions in your drawing. But you do not want it to automatically update the existing block references when you made the changes. Now, when you are satisfied with the attribute changes you made, you can apply those changes to all blocks in the current drawing by using the **Sync** button.

You can also use the **Synchronize** tool in the **Block Definition** panel to update attribute properties in block references to match their block definition. This tool can also be invoked by entering **ATTSYNC** at the command prompt or by choosing the **Synchronize Attributes** tool from the **Modify II** toolbar.

Note

Changing the attribute properties of an existing block does not affect the values assigned to the block. For example, in a block containing an attribute whose Tag is Cost and default value is 200, the default value 200 remains unaffected, if the Tag is changed to UnitCost.

Move Up

This button moves the selected attribute tag up in the prompt sequence. Now, when you insert the block, the attribute you have just moved up will appear earlier in the prompt sequence. The **Move Up** button is not available when a constant attribute is selected and if the attribute is at the top most position in the attribute list.

Move Down

This button moves the selected attribute tag down in the prompt sequence. The **Move Down** button is not available when a constant attribute is selected and when the attribute is at the bottom most position in the attribute list.

Edit

This button allows you to modify the attribute properties. To edit the attribute definition in blocks, select the attribute and then choose **Edit** from the dialog box; the **Edit Attribute** dialog box will be displayed.



Note

*To edit an attribute, you can also double-click on the attribute or choose the **Single** tool from **Home > Block > Edit Attributes** drop-down. Then, select the attribute to display the **Enhanced Attribute Editor** dialog box.*

The options in the **Edit Attribute** dialog box are discussed next.

Attribute Tab

The options under the **Attribute** tab of the **Edit Attribute** dialog box are used to modify the mode and data of attributes, refer to Figure 17-15. The options under the **Mode** area can be selected for the block. The **Data** area of the dialog box has three edit boxes: **Tag**, **Prompt**, and **Default**, where you can enter the data to be changed.

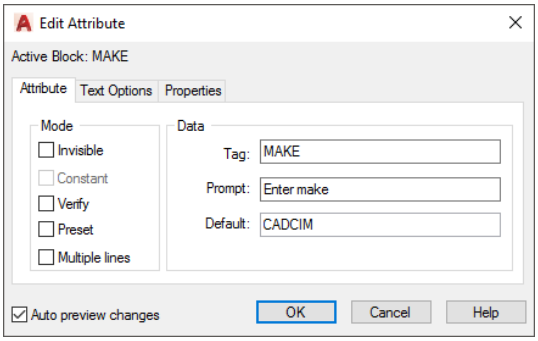

***Figure 17-15** The **Attribute** tab of the **Edit Attribute** dialog box*

Text Options Tab

The options under this tab are used to set the properties that define the way the text of an attribute is displayed in the drawing, refer to Figure 17-16. The options in this tab are discussed next.

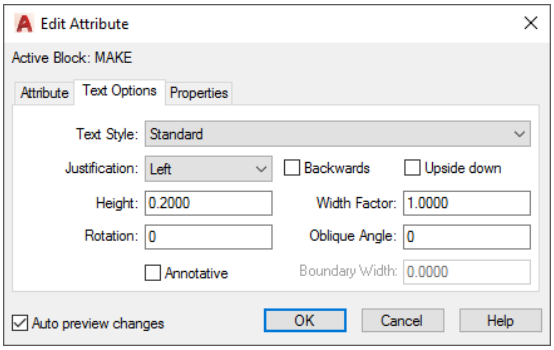

Figure 17-16 *The* ***Text Options*** *tab of the* ***Edit Attribute*** *dialog box*

Text Style: The options in this drop-down list are used to specify the text style for attribute text.

Justification: The options in this drop-down list are used to specify how attribute text is justified.

Height: This edit box is used to specify the height of the attribute text. If the height is defined in the current text style, this edit box is not available.

Rotation: This edit box is used to specify the rotation angle of the attribute text.

Backwards: This check box is used to specify whether or not the text is displayed backwards. Select this box to display the text backwards.

Upside down: This check box is used to specify whether or not the text is displayed upside down. Select this check box to display the text upside down.

Width Factor: This edit box is used to set the character spacing for attribute text. Entering a value less than 1.0 condenses the text and a value greater than 1.0 expands it.

Oblique Angle: This edit box is used to specify the angle at which the attribute text is slanted away from its vertical axis.

Annotative: This check box is used to convert the non-annotative attribute to the annotative attribute and vice-versa.

Boundary Width: This edit box sets the width of the boundary that will be displayed while writing the multiline text for the attributes' default value. The value specified here is the relative scale value. If you specify a value less than 1, it will decrease the boundary width. Whereas, if you specify a value more than 1, it will increase the boundary width. Accordingly, the paragraph width for the multiline text will increase or decrease. You can modify the value of the boundary width only when you select the **Multiple lines** check box from the **Mode** area of the **Attribute Definition** dialog box.

Properties Tab

The options under this tab of the **Edit Attribute** dialog box are used to define the layer, color, lineweight, and linetype of the attributes, refer to Figure 17-17. Also, if the drawing uses plot styles, you can assign a plot style to the attribute. The options of the **Properties** tab are discussed next.

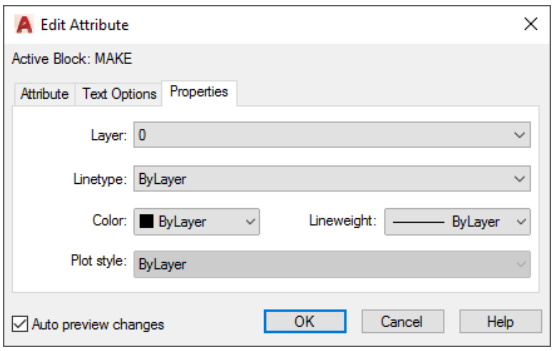

Figure 17-17 *The **Properties** tab of the **Edit Attribute** dialog box*

Layer: The layer(s) on which the attribute is defined is displayed in this drop-down list. You can also change the layer of the selected attribute using this drop-down list.

Linetype: The options in this drop-down list are used to specify the linetype of attribute text.

Color: The options in this drop-down list are used to specify the attribute's text color.

Lineweight: The options in this drop-down list are used to specify the lineweight of attribute text. The changes you make to this option do not come into effect if the **LWDISPLAY** system variable is off.

Plot Style: The options in this drop-down list are used to specify the plot style of the attribute. If the current drawing uses color-dependent plot styles, the **Plot style** list in this dialog box is not available.

Auto preview changes

This check box controls whether or not the drawing area is immediately updated to display any visible attribute changes you make. If the **Auto preview changes** check box is selected, changes are immediately visible. If this check box is cleared, changes are not immediately visible.



Tip

*The buttons such as **Move Up**, **Move Down**, **Edit**, **Remove**, and **Settings** can also be invoked by using the shortcut menu which appears by right-clicking on the attributes in the attribute list in the **Block Attribute Manager** dialog box.*

Settings

Choosing this button from the **Block Attribute Manager** opens the **Block Attribute Settings** dialog box, as shown in Figure 17-18, where you can customize how attribute information is listed in the **Block Attribute Manager**.

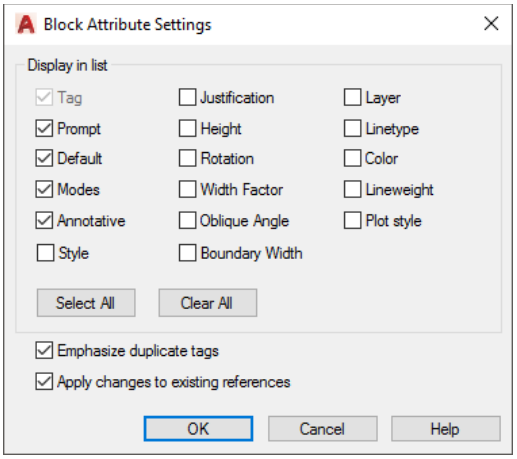

Figure 17-18 *The* ***Block Attribute Settings*** *dialog box*

Display in list

You can specify the attribute properties you want to be displayed in the list by selecting the check boxes available in this area of the **Block Attribute Settings** dialog box. This area shows the check boxes of all the properties that are displayed in the attribute list. Only the check boxes of the properties selected are displayed in the attribute list. These properties can be viewed in the attribute list by scrolling it. The **Tag** check box is always selected by default.

This area of the dialog box has two buttons, **Select All** and **Clear All**, which select and clear all the check boxes, respectively.

Emphasize duplicate tags

This check box turns duplicate tag emphasis on and off. If this check box is selected, duplicate attribute tags are displayed in red type in the attribute list. For example, if a block has PRICE as a tag more than once, the attribute PRICE will be displayed in red in the attribute list. If this check box is cleared, duplicate tags are not emphasized in the attribute list.

Apply changes to existing references

This check box is used to specify whether or not to update all existing instances of the block whose attributes you are modifying. Therefore, if this check box is selected, all instances of the block with the new attribute definitions are updated. If this check box is cleared, only new insertions of the block with the new attribute definitions are displayed.

Use **Sync** in the **Block Attribute Manager** to apply changes immediately to existing block instances. This temporarily overrides the **Apply changes to existing references** option.



Tip

If constant attributes or nested attribute blocks are affected by your changes, use the ***REGEN*** *command to update the display of those blocks in the drawing area.*

Remove

This button removes the selected attribute from the block definition. If the **Apply changes to existing references** check box is selected in the **Block Attribute Settings** dialog box before you choose the **Remove** button, the attribute is removed from all instances of the block in the current drawing. If this check box is cleared before choosing the **Remove** button, the attribute is removed only from the attribute list and the blocks that will be inserted henceforth. You can remove attributes from block definitions and from all existing block references in the current drawing. Attributes removed from existing block references do not disappear in the drawing area until you regenerate the drawing using the **REGEN** command. You cannot remove all attributes from the block, at least one attribute should remain. However, if you want to remove all the attributes, you have to redefine the block.

Note that the **Remove** button is not available for blocks with only one attribute.

EXTRACTING ATTRIBUTES

Ribbon: Insert > Linking & Extraction > Extract Data
Command: EATTEXT or DATAEXTRACTION



The **Extract Data** tool allows you to extract the block attribute information and property information such as drawing summary from single or multiple drawings. When this tool is invoked, the **Data Extraction** wizard is displayed, as shown in Figure 17-19. This wizard includes the following pages:

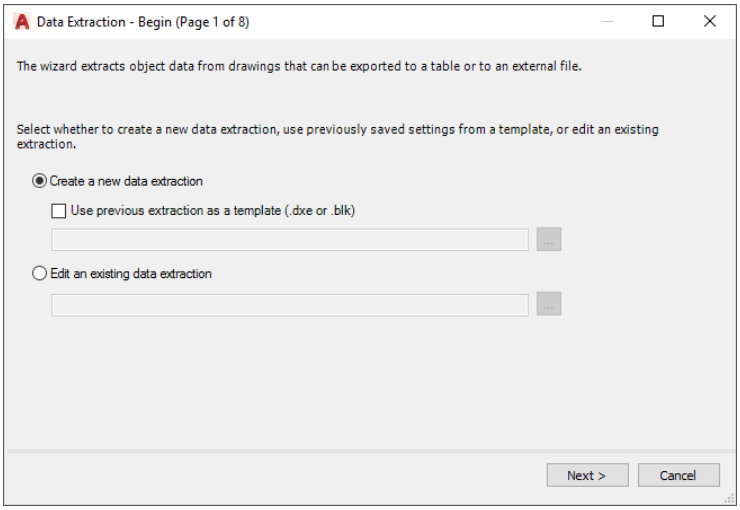

***Figure 17-19** The **Begin** page of the **Data Extraction** wizard*

Begin

The **Begin** page, refer to Figure 17-19, allows you to save extracted data to a new *.dxe* file or to modify an existing *.dxe* file. You can also use the saved *.dxe* or *.blk* file as the template for extracting data.

This wizard page has two radio buttons: **Create a new data extraction** and **Edit an existing data extraction**. If you select the **Use previous extraction as a template (.dxe or .blk)** check box, you will be allowed to select an existing template file. To select a template file, choose the Browse[...] button available on the right of the template name edit box. A .blk template file is a text file that allows you to specify the attribute and property values that you want to extract and the information about the drawing that you want to retrieve. The file can be created by using any text editor such as Notepad or WordPad. While using a *.blk* template file, you will be prompted to save the data extraction to a *.dxe* file to extract the information. Select the **Edit an existing data extraction** check box to modify an existing *.dxe* file. Now, you will be allowed to select the existing data file. To select a template file, choose the Browse[...] button available on the right of the template name edit box.

Choose the **Next** button from the **Begin** page of the **Data Extraction** wizard; the **Save Data Extraction As** dialog box will be invoked. Next, specify the desired name and location in this dialog box and choose the **Save** button; the **Define Data Source** page will be displayed, as shown in Figure 17-20.

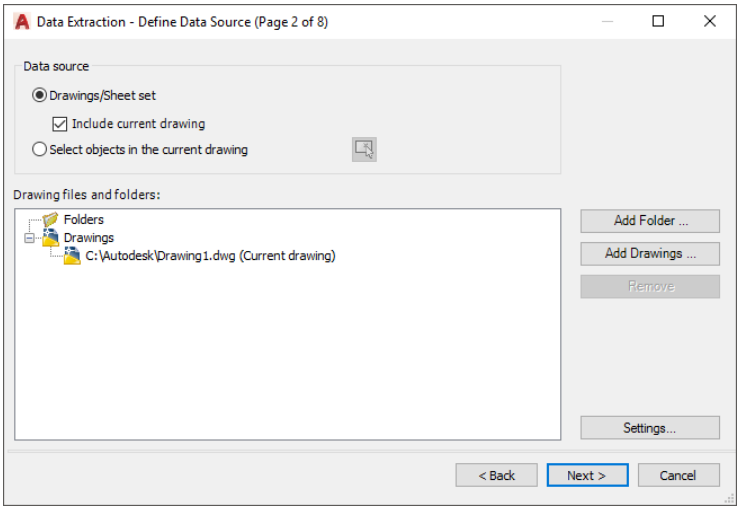

***Figure 17-20** The **Define Data Source** page of the **Data Extraction** wizard*

Define Data Source

The **Define Data Source** page is used to select an object either from a single drawing or from multiple drawings. The **Data source** area of this dialog box has two radio buttons: **Drawings/Sheet set** and **Select objects in the current drawing**.

Drawings/Sheet set

This radio button enables you to specify the drawings and folder from which you can extract the data. If this radio button is selected, the **Add Folder** and **Add Drawings** buttons will be enabled. The name of the drawings selected will be displayed with full path in the **Drawing files and folders** area.

Next, choose the **Add Folder** button; the **Add Folder Options** dialog box will be displayed, as shown in Figure 17-21. All drawings within the added folder will be considered for extracting

data. To add a folder, choose the Browse[...] button available on the right of the **Folder** edit box. Select the **Automatically include new drawings added in this folder to the data extraction** check box in the **Options** area to automatically consider the new drawing added to this folder for data extraction. Select the **Include subfolders** check box to include the drawings within the sub-folders of the folder selected to be added. Select the **Utilize wild-card characters to select drawings** check box to include only the files matching the specified wild-card characters for data extraction. By default, *.dwg is used as the wild-card character and that is why, all the files with extension *.dwg* (all the drawing files) are considered for the data extraction. For example, if you want AutoCAD to consider all the drawing files having "computer" in their name, enter the wild-card character ***computer*.dwg**. Next, specify the desired settings and choose the **OK** button.

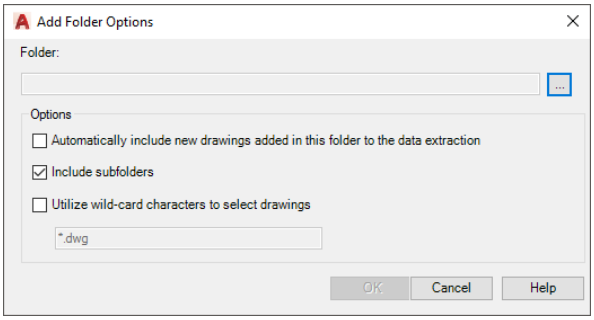

Figure 17-21 *The **Add Folder Options** dialog box*

Similarly, you can add a single drawing for the data extraction by choosing the **Add Drawings** button of the **Define Data Source** page of the **Data Extraction** wizard.

Select the **Include current drawing** check box in the **Define Data Source** page to add the current drawing for the data extraction including the folders and drawings.

Select objects in the current drawing

This radio button enables you to add the selected object from the current drawing for the data extraction. If you select this radio button, the **Select objects in the current drawing** button will be enabled. Choose this button to select the objects from the drawing.

Drawing files and folders

This area of the dialog box displays the tree view of all the drawing and folder names with their locations that are selected for the data extraction. All the drawing and folders are displayed under the **Folders** and **Drawings** heads respectively. Only the checked drawings under the folder are included for the data extraction. You can also remove the selected drawing or folder from the list of drawing to be used for data extraction. To remove the file or folder from the list of files and folders to be used for the data extraction, select them from the **Drawing files and folders** area and choose the **Remove** button.

Settings

Choose the **Settings** button from the **Define Data Source** page to invoke the **Data Extraction - Additional Settings** dialog box, as shown in Figure 17-22. This dialog box has two areas: **Extraction settings** and **Extract from**. By default, the **Extract objects from blocks** and **Extract objects from xrefs** check boxes are selected in the **Extraction settings** area. The first

two check boxes allow you to specify whether you want to extract the block attribute information from the external reference drawings or the nested blocks, or both. If you select the **Include xrefs in block counts** check box, then the xrefs will automatically be included while counting the blocks. The two radio buttons available in the **Extract from** area allow you to control the settings for counting the blocks. If you select the **Objects in model space** radio button, only the blocks currently inserted into the model space will be counted. If you select the **All objects in drawing** radio button, all the blocks from the current drawing will be counted. Choose the **OK** button to exit the **Data Extraction - Additional Settings** dialog box. Choose the **Next** button from the **Define Data Source** page; the **Select Objects** page of the **Data Extraction** wizard will be displayed.

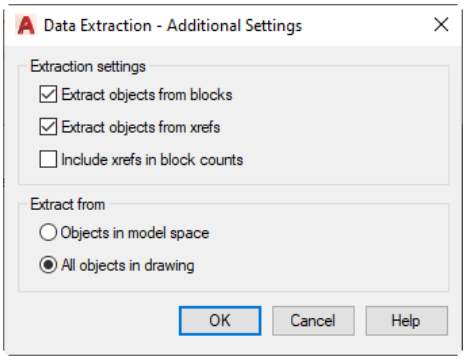

***Figure 17-22** The **Data Extraction - Additional Settings** dialog box*

Select Objects

The **Select Objects** page, as shown in Figure 17-23, allows you to filter out the objects to be included for data extraction. The objects that are checked in the **Objects** list box will only be considered for data extraction. The objects that are present in the current drawing are checked by default. Apart from the check box column, three more columns namely, **Object**, **Display Name**, and **Type** are displayed in the **Objects** list box. These columns display the information about the objects that are available to be selected and their type based on blocks and non-blocks.

The **Display options** area is used to control the objects to be displayed in the **Objects** list box. You can sort out the display from the blocks and non-blocks, blocks having attributes, and objects used in the current drawing. After specifying the objects to be displayed, choose the **Next** button from the **Select Objects** page to move to the **Select Properties** page.



Tip

*You can reverse the display order of the entities by clicking on the respective column head in the **Objects** list box. The direction of the arrow displayed at the column head of the selected column represents whether the data is arranged in the ascending or descending order.*

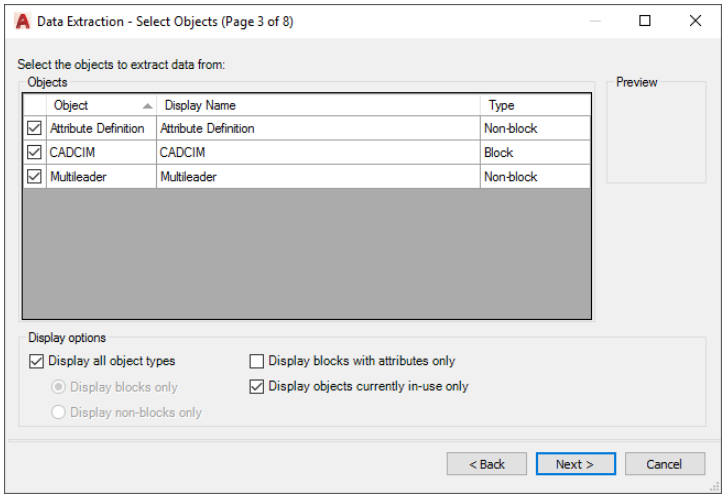

Figure 17-23 *The* ***Select Objects*** *page of the* ***Data Extraction*** *wizard*

Select Properties

The **Select Properties** page, as shown in Figure 17-24, allows you to select the properties of the objects to be included for data extraction. Only the properties that are checked in the **Properties** list box are considered for data extraction. Apart from the check box column, three more columns namely, **Property**, **Display Name**, and **Category** are displayed in the **Properties** list box. These columns display the information about the available properties and their category.

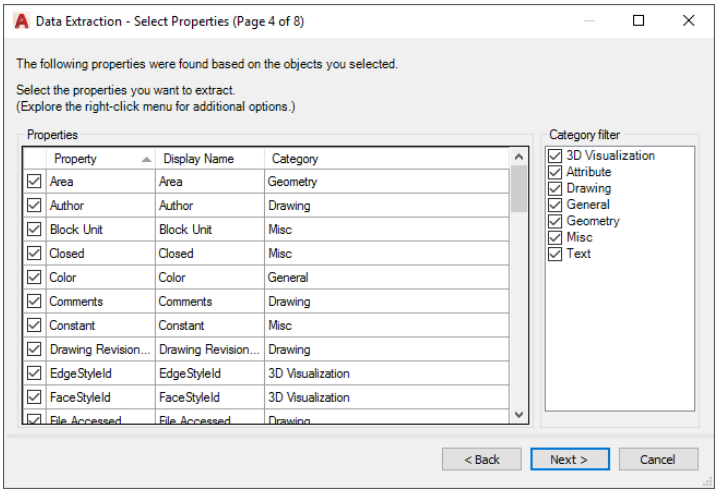

Figure 17-24 *The* ***Select Properties*** *page of the* ***Data Extraction*** *wizard*

The **Category filter** area is used to control the further display of properties in the **Properties** list box. To do so, select the categories of the properties to be included in the **Properties** list box. For example, if you select the **Attribute** check box in the **Category filter** area, only the information about the attributes defined for the drawing will be extracted and displayed in the **Properties** list box. All the properties and categories displayed here are similar to the one displayed in the **PROPERTIES** palette.

You can also edit the names below the **Display Name** column. Select the row for which you want to change the display name and double-click on the **Display Name** column or right-click to display the shortcut menu. Next, choose the **Edit Display Name** option from the shortcut menu and enter a new name. After specifying the properties, choose the **Next** button from the **Select Properties** page to display the **Refine Data** page.

Refine Data

The **Refine Data** page, as shown in Figure 17-25, allows you to format the data extracted from a drawing. The properties selected from the **Select Properties** page of the **Data Extraction** wizard are displayed as the column head on the **Refine Data** page and the extracted data is displayed under these column heads of the table.

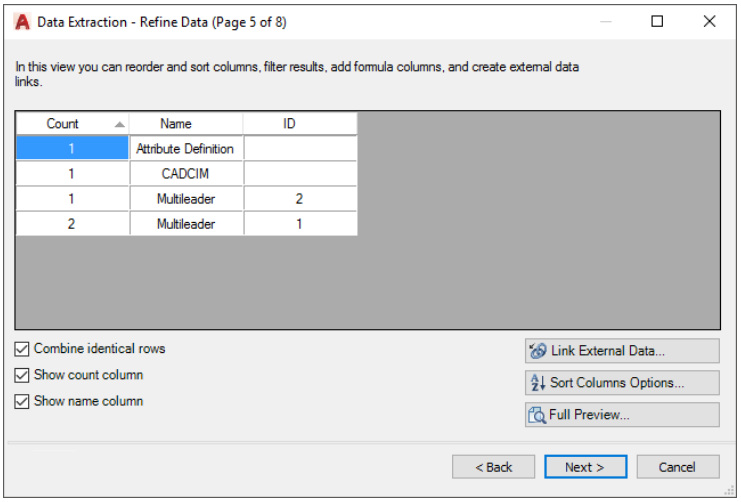

Figure 17-25 *The* ***Refine Data*** *page of the* ***Data Extraction*** *wizard*

The **Combine identical rows** check box is used to merge the rows having the same data. The sum of the merged rows having the same data will be displayed below the **Count** column head. The **Show count column** check box is used to display the **Count** column head on the extracted data table. The **Show name column** check box is used to display the **Name** column head on the extracted data table. All the check boxes explained above are selected by default.

You can also add columns from an excel sheet to the data extracted from a drawing. Choose the **Link External Data** button; the **Link External Data** dialog box will be displayed, as shown in Figure 17-26. The options in the **Link External Data** dialog box are discussed next.

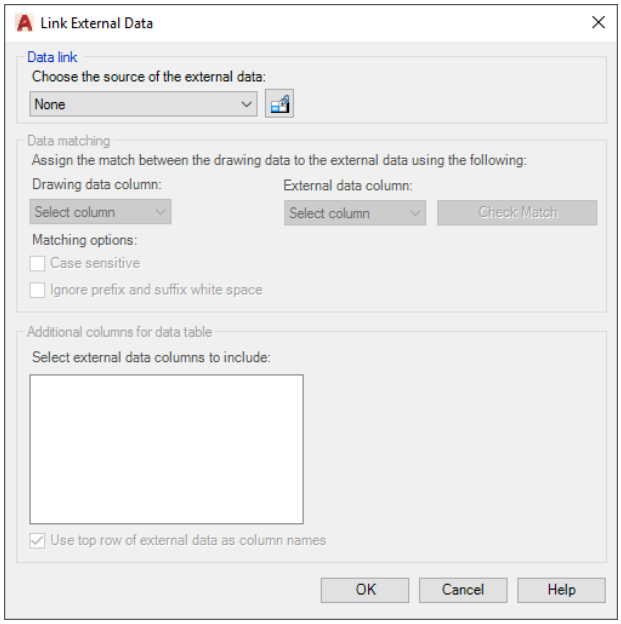

Figure 17-26 *The* ***Link External Data*** *dialog box*

Data link Area

This area is used to specify the Excel sheet to be attached with the drawing. The **Choose the source of the external data** drop-down list displays all the external data links that are attached to the current drawing. Select the external data link from which you want to include the columns. To attach a new Excel file to the drawing, choose the **Launch Data Link Manager** button; the **Data Link Manager** dialog box will be displayed on the screen. The other options in the **Data Link Manager** dialog box have been discussed earlier in Chapter 7 (Creating Texts and Tables). The rest of the options in the **Link External Data** dialog box will be available as soon as you specify the external data to be linked with the drawing.

Data matching Area

The options in this area are used to establish a relationship between the data extracted from the drawing and the attached external Excel sheet. There must be at least one cell with matching data in the selected column of the drawing data and the external data column respectively. This matching data in the external and internal data acts as a key cell to specify the proper insertion of the external data into the extracted data.

The **Drawing data column** drop-down list displays all the property column heads of the data extracted by the drawing. Select the column head from this drop-down list that is to be used for matching. The **External data column** displays the column names from the external data. Select the column name from where you want to insert data into the extracted data. Select the **Case sensitive** check box to check the case of the alphabets while matching the data of the selected columns from the extracted and external data. Select the **Ignore prefix and suffix white space** check box to ignore the blank spaces in the front and back of the entries in the columns that are used for matching.

Specify preferences for matching the data and choose the **Check Match** button to compare the data of the specified columns from the extracted and external data. If the data matching is successful, the **Valid Key** message box will be displayed, as shown in Figure 17-27. Otherwise, the **Check Key** message box will be displayed suggesting the possible reasons of conflict between the two datums, as shown in Figure 17-28. Unless there is a proper matching between the two datums, the merging of the external data into the extracted data will not take place.

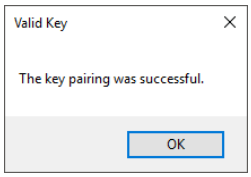

Figure 17-27 *The* ***Valid Key*** *message box*

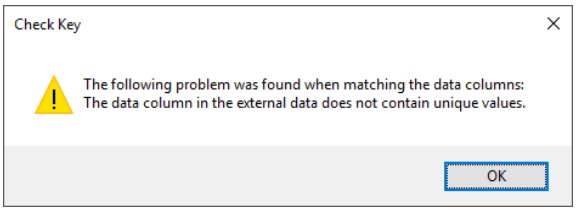

Figure 17-28 *The* ***Check Key*** *message box*

Additional columns for data table Area

This area displays all columns of the external data. Select the check box on the left of each column entry to include that column into data extraction. The **Use top row of external data as column names** check box will be used to specify the first row of the selected columns as the column head for the extracted data. Otherwise, the column heads are displayed as **Column 1**, **Column 2**, and so on. By default, this check box is selected.

Specify the preferences for the external data to be included in the current data extraction in the **Link External Data** dialog box and choose the **OK** button. The specified columns of the external data will be added to the extracted data and displayed in the **Refine Data** page of the **Data Extraction** wizard. The drawing data column used for matching and the external data column included in the extracted data will be represented with an external data link symbol. This symbol will be displayed on the side of the column head in the **Refine Data** page, as shown in Figure 17-29.

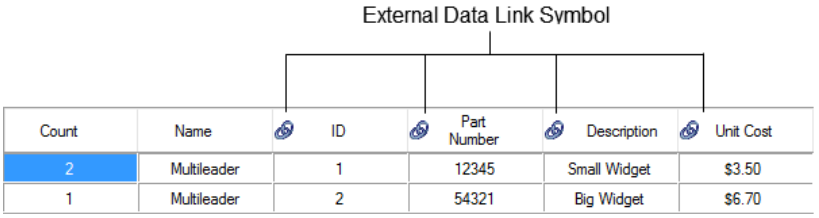

Figure 17-29 *The external data link symbol*

You can also change the display order of rows in the extracted data. To do so, choose the **Sort Columns Options** button from the **Refine Data** page; the **Sort Columns** dialog box will be displayed, refer to Figure 17-30. You can define the sort order for a single column or a group of columns at a time with this dialog box. The options in the **Sort Columns** dialog box are discussed next.

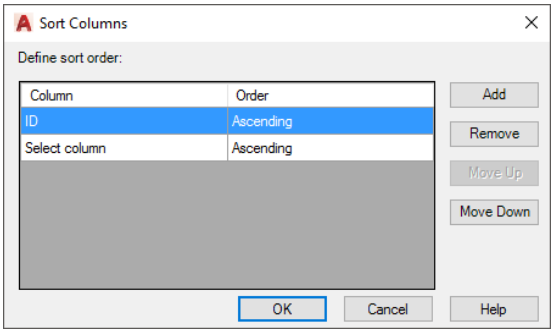

Figure 17-30 *The* ***Sort Columns*** *dialog box*

The **Define sort order** list box has two columns with the **Column** and **Order** column heads. Click on the drop-down arrow below the **Column** head to display all the column head entries for the extracted data. Select the column for which you want to change the display order of the rows. Choose the **Ascending** or **Descending** option from the **Order** drop-down list to specify the display order. Choose the **Add** button to add more rows in the **Define sort order** list box so as to specify the display order for other columns of the extracted data. The **Remove** button enables you to remove the selected sort order from the **Define sort order** list box. Choose the **Move Up** or **Move Down** button to move the selected row in the sort order, one level up or one level down respectively in the **Define sort order** list box.

Choose the **Full Preview** button; a preview will be displayed in the **Full Preview** window. Partial view of the **Full Preview** window is shown in Figure 17-31.

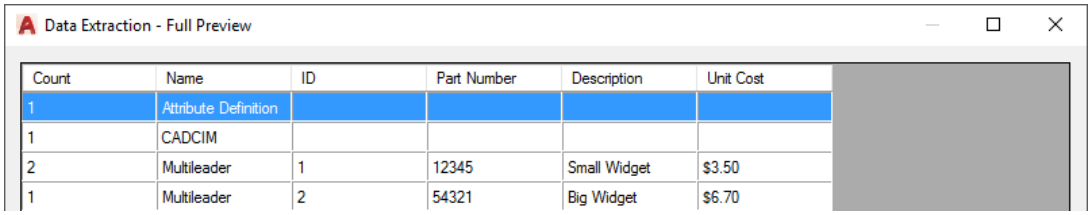

Figure 17-31 *Partial view of the* ***Full Preview*** *window*

When you right-click on the table displayed on the **Refine Data** page, a shortcut menu will be displayed, as shown in Figure 17-32. The options in the shortcut menu are discussed next.

Sort Descending/Sort Ascending

These options sort the data of the selected column in the descending/ ascending order.

Sort Columns Options

This option is used to invoke the **Sort Columns** dialog box which has been discussed earlier.

Rename Column

This option is used to rename the column head of the selected column.

Hide Column

This option is used to hide the selected column from the table of the extracted data. The selected column is only removed from the view, it is not deleted.

Show Hidden Columns

This option is used to display the hidden columns. This option becomes available only when atleast one column is hidden from the view. You can choose to display all the hidden columns or only the selected one.

Set Column Data Format

This option is used to invoke the **Set Cell Format** dialog box. You can set the data format for the selected row through this dialog box.

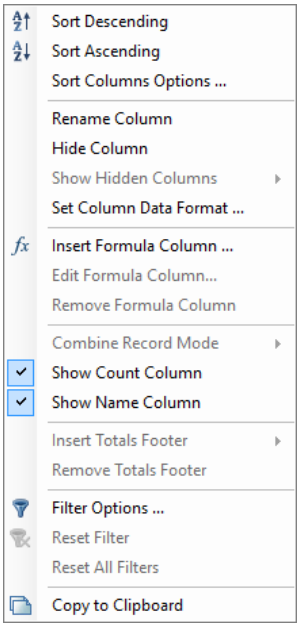

Figure 17-32 *The shortcut menu for the* ***Refine Data*** *page*

Insert Formula Column

This option is used to insert a formula column in the extracted data. Choose the **Insert Formula Column** option to invoke the **Insert Formula Column** dialog box, as shown in Figure 17-33. You can define and insert a formula column using this dialog box. The entries in the **Formula** column are governed by the mathematical formula defined by using one or more columns. The entry in a particular cell of the formula column can be calculated by using the formula and taking the value from the same row under the column that has been used to define the formula. Enter the column head name to be displayed on the formula column in the **Column name** edit box. The **Formula** area is used to define the formula. To select a column to define the formula, double click on the column in the **Columns** list box, or drag and drop the selected column into the **Formula** area. To use an operator such as +,-,*,or / in the formula, choose the desired operator button from the **Formula** area.

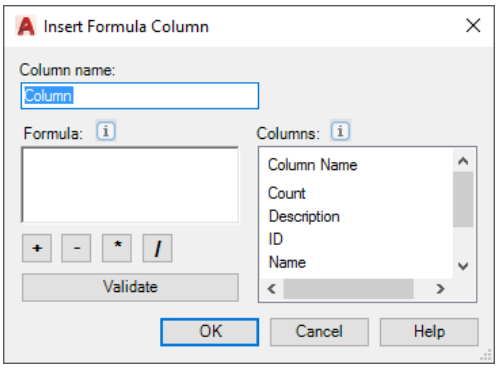

Figure 17-33 *The* ***Insert Formula Column*** *dialog box*

You can also make entry in the **Formula** area through the keyboard. Choose the **Validate** button to check the validity of the formula. If the defined formula is valid, AutoCAD will display the **Valid Equation** message box, as shown in Figure 17-34, and if the formula defined is invalid,

the **Invalid Equation** message box will be displayed, as shown in Figure 17-35. The **Formula** column is always inserted on the right of the selected column and is indicated by a formula symbol, refer to Figure 17-36.

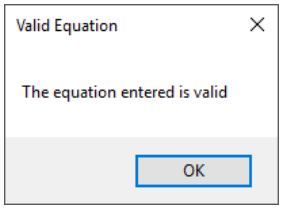

Figure 17-34 *The* ***Valid Equation*** *message box*

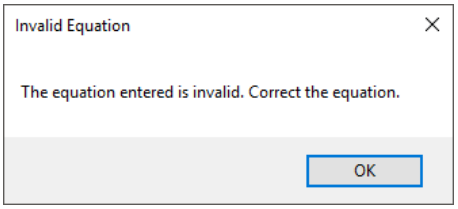

Figure 17-35 *The* ***Invalid Equation*** *message box*

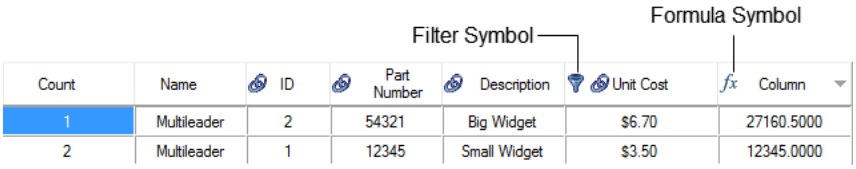

Figure 17-36 *The* ***Formula*** *and* ***Filter*** *symbols*

Edit Formula Column

This option is used to edit the pre-defined formula. Choose this option to invoke the **Edit Formula Column** dialog box.

Remove Formula Column

This option is used to delete the formula column from the extracted data. The **Edit Formula Column** and the **Remove Formula Column** options are available in the shortcut menu only when you right-click on the formula column.

Combine Record Mode

This option is used to control the display of the numeric data into the selected columns by displaying them either as separate values or collapsing the identical property rows into one and displaying their total sum. If any formula column is added and you apply the **Combine Record Mode** option to the formula column, then in case of the **Separate Values** sub-option, the formula will be applied after combining the identical rows. But in case of **Sum Values** sub-option, the formula will be applied to the individual rows and then the identical rows will be combined together and the resultant value will be displayed as the sum of those individual columns. This option will be available only when the selected columns have numerical data and the **Combine identical rows** check box is selected.

Show Count Column/Show Name Column

These options are used to display the **Count** column and the **Name** column, respectively. These options are selected by default.

Insert Totals Footer

This option is used to add a footer row under the selected column. This row can display the sum, maximum or minimum value in the selected column, and the average of all values above the footer row in that column. This option is only available for the columns that have numerical data.

Remove Totals Footer

This option is used to remove the totals footer from the selected row.

Filter Options

This option is used to decide the filter criterion for the selected column. Choose this option to invoke the **Filter <column name>** dialog box, refer to Figure 17-37. Only the entries that fulfill the specified filter criterion will be displayed on the table. The column on which the filter is applied is represented by a Filter symbol on the side of the column name, refer to Figure 17-36.

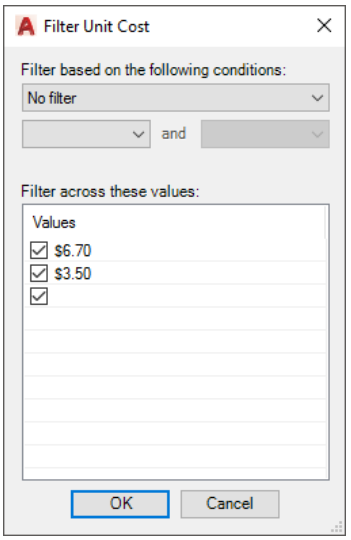

***Figure 17-37** The **Filter <column name>** dialog box*

Reset Filter/Reset All Filters

These options are used to remove the applied filters from the selected columns or from all the columns of the table at a time respectively.

Copy to Clipboard

This option is used to copy the selected table to the clipboard.

After formatting the data from the extracted files, choose the **Next** button to display the **Choose Output** page.

Choose Output

The **Choose Output** page, as shown in Figure 17-38, allows you to specify the format of the output file and the location to save it. Select the **Insert data extraction table into drawing** check box to insert the extracted data into the drawing as a table. Select the **Output data to external file (.xls .csv .mdb .txt)** check box to save the extracted data to a separate file at the end of the wizard. Select the Browse [...] button to specify the location to save this file. Choose the **Next** button from the **Choose Output** page to move to the **Table Style** page.

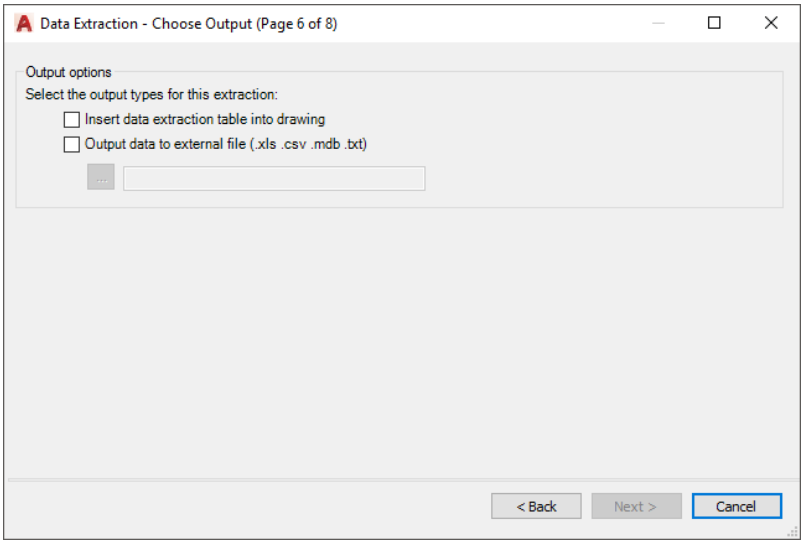

Figure 17-38 *The* ***Choose Output*** *page of the* ***Data Extraction*** *wizard*

Table Style

The **Table Style** page, as shown in Figure 17-39, will be available only if the **Insert data extraction table into drawing** check box is selected in the **Choose Output** page. The options in the **Table Style** page are discussed next.

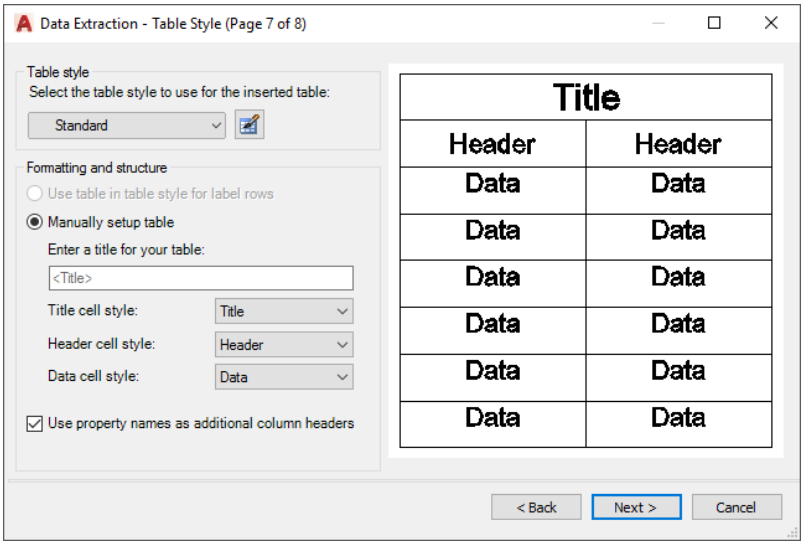

Figure 17-39 *The* ***Table Style*** *page of the* ***Data Extraction*** *wizard*

Table style Area

You can select the table style from the **Select the table style to use for the inserted table** drop-down list. By default, only the **Standard** table style is available. To create a new table style, choose the **Table Style** button on the right of the **Select the table style to use for the inserted table** drop-down list; the **Table Style** dialog box will be displayed. You can use this dialog box to create a new table style or modify an existing table style to be used for the extracted attributes.

Formatting and structure Area

The **Use table in table style for label rows** radio button will be available only when the table style that you have specified contains a table template. Select this radio button to insert the extracted data into the pre-defined tables with the pre-defined header and footers. Select the **Manually setup table** radio button to enter the title, header, and data cell styles manually. Enter the title for the table in the **Enter a title for your table** text box. Select the title style defined in the specified table styles in the **Title cell style** drop-down list. Similarly, you can specify the header and data cell style in their respective drop-down lists. The **Use property names as additional column headers** check box is selected by default. This option enables you to include the column headers into the table and use the displayed name as the header row. The preview of the table style is displayed on the right of the **Table Style** page. After specifying the **Table Style** formatting and structure, choose the **Next** button to move to the **Finish** page.

Finish

The **Finish** page, as shown in Figure 17-40, completes the extraction of data. Choose the **Finish** button and specify the insertion point in the drawing area to insert the extracted data. Note that this is possible only if the **Insert data extraction table into drawing** check box is selected on the **Choose Output** page of the **Data Extraction** wizard.

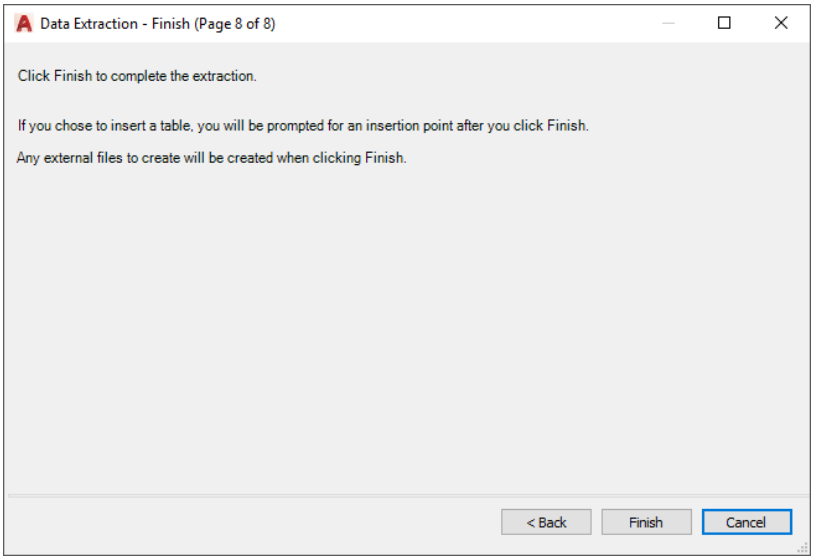

Figure 17-40 *The* ***Finish*** *page of the* ***Data Extraction*** *wizard*

The ATTEXT Command for Attribute Extraction

Command: ATTEXT

The **ATTEXT** command allows you to use the **Attribute Extraction** dialog box, as shown in Figure 17-41, for extracting 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.

File Format Area

This area of the dialog box allows you to select the file format. You can select either of the three radio buttons available in this area, as per your requirement. 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.

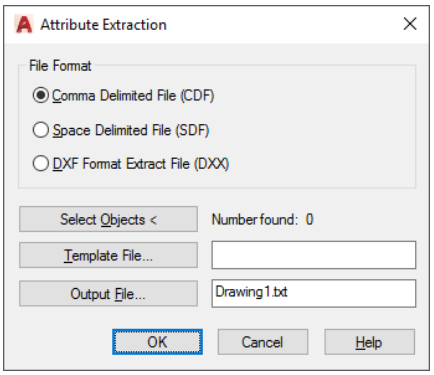

Figure 17-41 *The* ***Attribute Extraction*** *dialog box*

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 option contains only block references, attribute values, and end-of-sequence objects. 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. In this dialog box, you can select an already defined template file. 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. A 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, consider the case given next.

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. You can browse and select a template file from this dialog box, as shown in Figure 17-42.

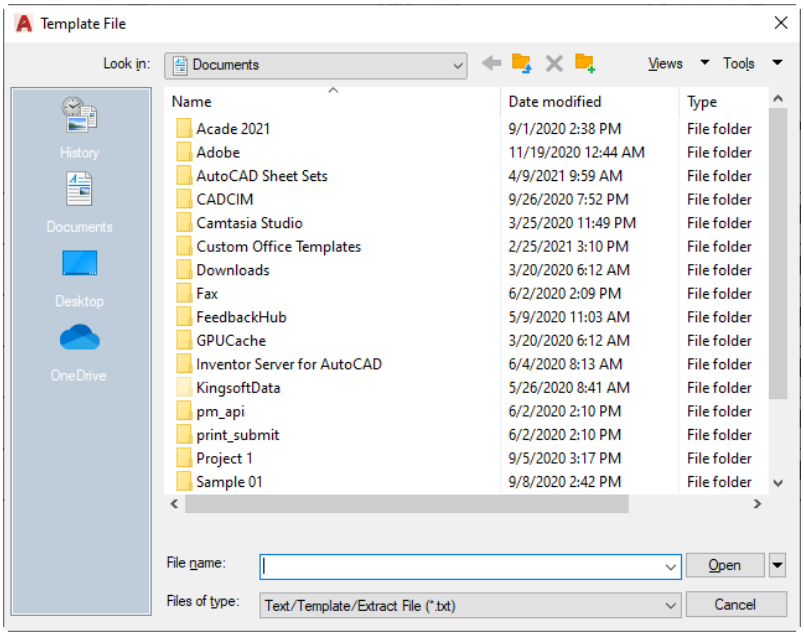

Figure 17-42 *The* ***Template File*** *dialog box to select a template file*



Note

1. 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.

2. 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 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 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 automatically converts the tag names and the field names to uppercase letters before making a comparison.

Output File

When you choose the **Output File** button, the **Output File** dialog box is displayed. You can select an existing file from the dialog box, 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, as shown in Figure 17-43 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 appends *.txt* file extension for CDF or SDF files and *.dxx* file extension for DXF files.

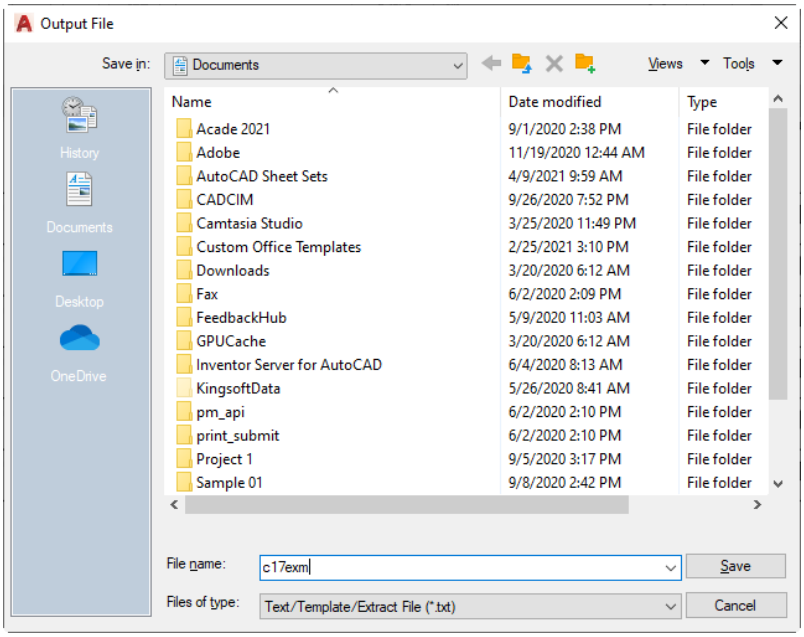

Figure 17-43 *The* ***Output File*** *dialog box to save an output file*



Note

You can also use the ***-ATTEXT*** *command to extract attributes using the command prompt. Here, you are prompted to specify a file format for the extracted 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.*

Example 3

In this example, you will extract the attribute values defined in Example 2. Extract the values of NAME, COUNT, MAKE, PROCESSOR, HD, RAM, and ITEM. These attribute values must be saved in a *.txt* format named TEST and arranged, as shown in the table given next.

COMP	1	Dell	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
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

1. Open the drawing that you saved in Example 2.

2. Choose the **Extract Data** tool from the **Link & Extraction** panel of the **Insert** tab; the **Data Extraction** wizard is displayed. Select the **Create a new data extraction** radio button from the **Begin** page and choose the **Next** button; the **Save Data Extraction As** dialog box is displayed.

3. Enter **TEST** in the **File name** edit box of the **Save Data Extraction As** dialog box and choose the **Save** button; the **Define Data Source** page is displayed.

4. Select the **Drawings/Sheet set** radio button and make sure that the **Include current drawing** check box is selected on the **Define Data Source** page. Choose the **Next** button; the **Select Objects** page is displayed.

5. Clear all check boxes in the **Object** column except the **COMP** object from the **Select Objects** page and choose the **Next** button; the **Select Properties** page is displayed.

6. Clear all check boxes except the **Attribute** check box from the **Category filter** area of the **Select Properties** page of the **Data Extraction** wizard. Also, make sure that the check box on the left of the **HD**, **ITEM**, **MAKE**, **PROCESSOR**, and **RAM** properties under the **Property** column are selected. Choose the **Next** button; the **Refine Data** page is displayed.

7. Arrange the display order of the columns according to the table displayed above, by dragging and dropping the column heads to the desired position in the **Refine Data** page of the **Data Extraction** wizard. Make sure that the **Show count column** and **Show name column** check boxes are selected. Choose the **Full Preview** button and check the data displayed in the **Data Extraction - Full Preview** window and match it with the table displayed above. Close the window and choose the **Next** button from the **Refine Data** page of the **Data Extraction** wizard; the **Choose Output** page is displayed.

8. Select the **Output data to external file (.xls .csv .mdb .txt)** check box and choose the Browse [...] button; the **Save As** dialog box is displayed. Specify the location to save the file, select the **Text/Template/Extract File (*.txt)** option from the **Files of type** drop-down list, enter **TEST** in the **File name** edit box, and choose the **Save** button. Choose the **Next** button from the **Choose Output** page; the **Finish** page is displayed.

9. Choose the **Finish** button from the **Finish** page of the **Data Extraction** wizard.

CONTROLLING THE ATTRIBUTE VISIBILITY

Ribbon: Home > Block > Retain Attribute Display drop-down **Command:** ATTDISP

The options to change the display of attributes are available in the **Retain Attribute Display** drop-down of the **Block** panel in the **Home** tab of the **Ribbon**. Generally, the attributes are visible unless they are defined invisible by using the **Invisible** mode. The invisible attributes are not displayed, but they are the part of the block definition. The tools to toggle the display of attributes **Retain Attribute Display**, **Display All Attributes**, and **Hide All Attributes** are available in the **Retain Attribute Display** drop-down.

The **Retain Attribute Display** tool is used to display the attributes as they were set while they were created. For example, the attributes created using the **Invisible** mode will not be displayed.

If you choose the **Hide All Attributes** tool, all attributes will become invisible.

If you choose the **Display All Attributes** tool, all attributes will be displayed in the drawing area.

In Example 2 of this chapter, the RAM attribute was defined with the **Invisible** mode. If you choose the **Retain Attribute Display** tool, the RAM values will not be displayed with the block, refer to Figure 17-44. If you want to make the RAM attribute values visible, choose the **Display All Attributes** tool from the **Block** panel.

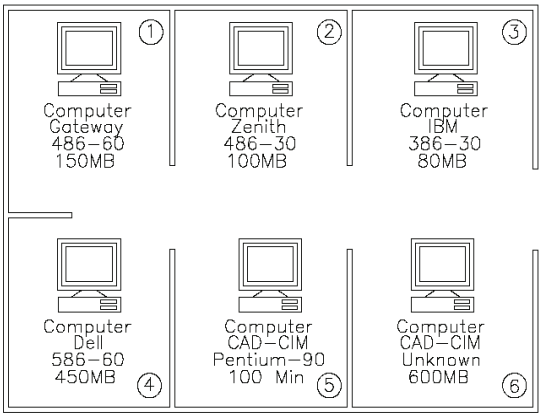

Figure 17-44 *The drawing of Example 2 after choosing the* ***Retain Attribute Display*** *tool*



Tip

After you have defined the attribute values and saved them with the block definition, it may be a good idea to hide all the attributes. By doing this, the drawing is simplified and also the regeneration time is reduced.

EDITING BLOCK ATTRIBUTES

The block attributes can be edited by using several methods. These methods are discussed next.

Editing Attributes Using the Enhanced Attribute Editor

Ribbon: Insert > Block > Edit Attribute drop-down > Single
Command: EATTEDIT **Toolbar:** Modify II > Edit Attribute



Choose the **Single** tool from the **Edit Attribute** drop-down from the **Block** panel to edit the block attribute values. When you invoke this tool, AutoCAD prompts you to select the block whose values you want to edit. After selecting the block, the **Enhanced Attribute Editor** dialog box is displayed, as shown in Figure 17-45. This dialog box allows you to change the **Attribute**, **Text Options**, and **Properties** of the block attribute. This dialog box can also be invoked by double-clicking on the block or the attribute in the current drawing. The **Enhanced Attribute Editor** dialog box displays all the attributes of the selected block and their properties. You can change the properties of the attribute you want. 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. The dialog box displays the following tabs.

Attribute Tab

The options under this tab are used to change the value of the attribute. Also, here the **Mode** of the attribute is not displayed. The other attributes of the block such as **Tag**, **Prompt**, and **Value** are displayed, as shown in Figure 17-45.

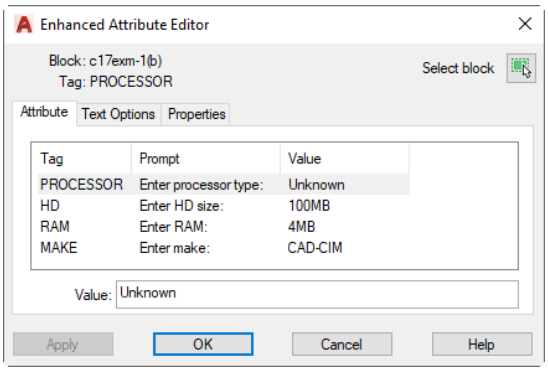

Figure 17-45 *The* ***Attribute*** *tab of the* ***Enhanced Attribute Editor*** *dialog box*

Text Options Tab

Under this tab, the options related to the text of attributes can be modified. This includes **Text Style**, **Justification**, **Height**, **Rotation**, **Backwards**, **Upside down**, **Width Factor**, **Oblique Angle**, **Annotative**, and **Boundary width**, as shown in Figure 17-46.

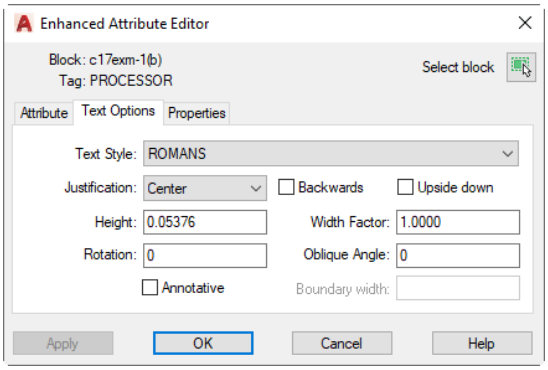

Figure 17-46 *The* ***Text Options*** *tab of the* ***Enhanced Attribute Editor*** *dialog box*

Properties Tab

This tab is used to edit the properties of attributes such as **Layer**, **Linetype**, **Color**, **Lineweight**, and **Plot style**. Choose the **Properties** tab of the **Enhanced Attribute Editor** dialog box to display the options in this tab, as shown in Figure 17-47.

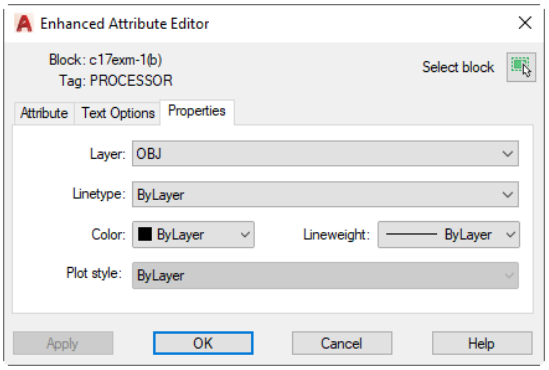

Figure 17-47 *The* ***Properties*** *tab of the* ***Enhanced Attribute Editor*** *dialog box*

Editing Attributes Using the Edit Attributes Dialog Box

The **ATTEDIT** command is used to edit the block attribute values through the **Edit Attributes** dialog box. Invoke this command; AutoCAD prompts you to select the block whose values are to be edited. After selecting the block, the **Edit Attributes** dialog box is displayed, as shown in Figure 17-48.

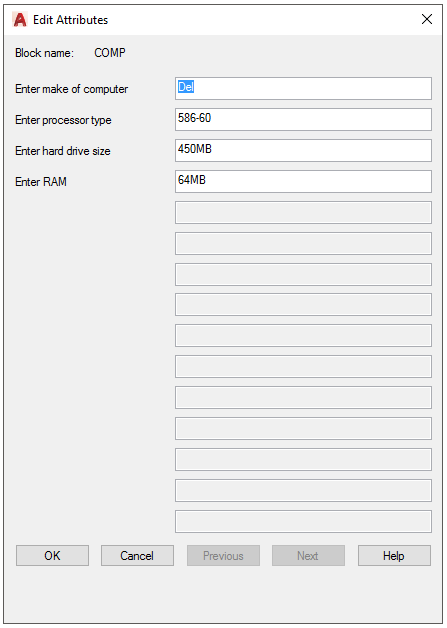

Figure 17-48 *The* ***Edit Attributes*** *dialog box*

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.

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

Note

*You cannot use the **ATTEDIT** command to perform the global editing of attribute values or to modify the 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.*

Example 4

In this example, you will choose the **Single** tool from the **Edit Attribute** drop-down in the **Attributes** panel to change the attribute of the first computer (150 MB to 2.1 GB) which is located in Room-1.

1. Open the drawing created in Example 2. The drawing has six blocks with attributes. The name of the block is **COMP** and has five defined attributes. One of the attributes is invisible. Zoom in the drawing so that the first computer is displayed on the screen, refer to Figure 17-49.

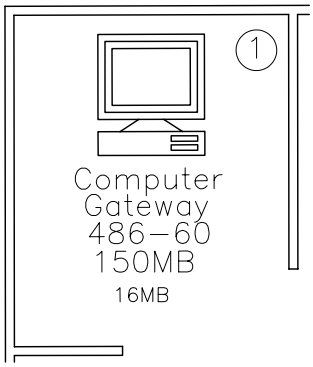

Figure 17-49 *Zoomed view of the first computer*

2. Choose the **Single** tool from **Insert > Block > Edit Attribute** drop-down; you are prompted to select a block. Select the computer located in Room-1. The **Enhanced Attribute Editor** dialog box showing the attribute prompts and values will be displayed, refer to Figure 17-45.

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

Tip

*You can also replace the 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.*

Global Editing of Attributes

Ribbon: Insert > Block > Edit Attribute drop-down > Multiple **Command:** -ATTEDIT

Choose the **Multiple** tool from the **Edit Attribute** drop-down in the **Block** panel 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. On choosing the **Multiple** tool from the **Edit Attribute** drop-down, AutoCAD displays the following prompt:

Edit attributes one at a time? [Yes/No]<Y>: **N** Enter

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 will display the following prompt:

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

If you enter **Y** at this prompt, AutoCAD 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.

Editing All Attributes

If you enter **N** at the previously mentioned prompt, AutoCAD flips from graphics to text screen and displays the message stating: '**Drawing must be regenerated afterwards**'.

Now, AutoCAD 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** Enter

When you enter the name of the block, AutoCAD 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 prompt:

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

When you specify the tag name, AutoCAD 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 prompt:

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

When you specify the attribute value, AutoCAD 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. To edit the null attributes, enter \ at the following prompt:

Enter attribute value specification <*>: \ Enter

After you enter this information, AutoCAD 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* Enter.

After you select the attributes, AutoCAD 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 will retrieve the attribute information, edit it, and then update the attribute values.

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

The following is the complete command prompt sequence after choosing the **Multiple** tool. It is assumed that the editing is global and for visible attributes only.

Edit attributes one at a time? [Yes/No] <Y>: **N** Enter
Edit only attributes visible on screen? [Yes/No] <Y>: **N** Enter
Drawing must be regenerated afterwards.

Enter block name specification <*>: Enter
Enter attribute tag specification <*>: Enter
Enter attribute value specification <*>: Enter
n attributes selected.
Enter string to change: *Enter the value to be modified* Enter.
Enter new string: *Enter the new value* Enter.



Note

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

2. If you select an attribute defined with the ***Constant*** *mode using the* ***Multiple*** *tool, the prompt displays* ***0 found*** *since attributes with the* ***Constant*** *mode are not editable.*

Example 5

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.

	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	Dell	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 shown in the above table.

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 steps are required to change the attribute value from **Unknown** to **Pentium**.

1. Choose the **Multiple** tool from **Insert > Block > Edit Attribute** drop-down and enter **N** at the next prompt and press ENTER as given below.

   Edit attributes one at a time? [Yes/No] <Y>: **N** Enter

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

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

3. As shown in the table, the attributes belong to a single block, COMP. A drawing may have 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** Enter

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

   On entering these values, AutoCAD will prompt you to select attributes. Use any object selection option to select all the blocks. AutoCAD 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 locates such attributes, they are highlighted.

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

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

6. The following is the command prompt sequence to change the make of the computer from **CAD-CIM** to **Compaq**:

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

7. The following is the command prompt sequence to change **MB** to **Meg**:

   *Choose the **Multiple** tool.*
   Edit attributes one at a time? [Yes/No] <Y>: **N** Enter
   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** Enter
Drawing must be regenerated afterwards.
Enter block name specification <*>: **COMP** Enter

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 will change all MB attribute values to Meg.

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

8. Choose the **Display All Attributes** tool from the **Block** panel to display the invisible attributes.



Note

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

Example 6

In this example, you will use the drawing of Example 5 to edit attributes individually, refer to Figure 17-50. 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.

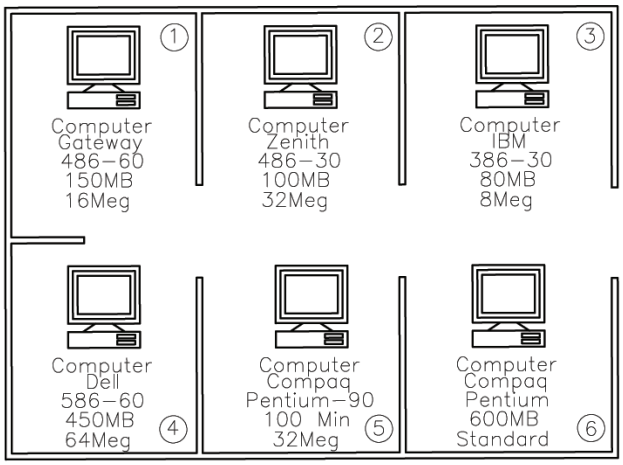

Figure 17-50 *The attribute values to be changed individually*

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

2. Double-click on the **100Min** attribute value in the drawing; the **Enhanced Attribute Editor** dialog box is displayed.

3. In the **Value** edit box, change **100 Min** to **100MB**. Choose **Apply** and then choose **OK**.

4. To change the height of attribute text, double-click on the value of **RAM** in the drawing; the **Enhanced Attribute Editor** dialog box is displayed.

5. In the dialog box, choose the **Text Options** tab and change the height to **0.075** in the **Height** edit box.

6. Choose **Apply** and then **OK**.

7. Change the height of the other attribute values using the process followed in Steps 5 and 6.

REDEFINING A BLOCK WITH ATTRIBUTES

If a block with attributes has been inserted a number of times in a drawing and you wish to edit the block geometry, or add or delete attributes from them, it may take a lot of time. You can use the **ATTREDEF** command to redefine an existing block with attributes, and once redefined, all the copies of the block with attributes get updated automatically. But before using this command, you need to explode a copy of the block containing the attributes and make the changes or you may use the new geometry. If you do not explode a block and try to use this command, an error message will be displayed: **New block has no attributes**. The prompt sequence is as follows:

Command: **ATTREDEF**
Enter name of the block you wish to redefine: *Enter the name of the block to be redefined and press ENTER.*
Select objects: *Select the new block geometry with attributes and press ENTER.*
Specify insertion base point of new Block: *Specify an insertion base point.*

After you select an insertion base point, the block is redefined and all the copies of the block in the drawing are updated. If you do not include an existing attribute when selecting objects for the new block, it will not be a part of the new block definition.



Note

*The **ATTREDEF** command removes all the formatting changes incorporated into the property by using the **ATTEDIT** or **EATTEDIT** command.*

In-place Editing of Blocks with Attributes

Ribbon: Insert > Reference > Edit Reference **Command:** REFEDIT
Toolbar: Refedit > Edit Reference In-Place



The **Edit Reference** tool has already been discussed in Chapter 16, Working with Blocks. This tool can also be used for in-place editing of blocks with attributes. After invoking the **Edit Reference** tool, select a block with attributes that you want to edit. The **Reference Edit** dialog box is displayed with the name of the block in the **Reference name** list box. To be able

to edit attributes, you should select the **Display attribute definitions for editing** check box from the **Settings** tab of the dialog box and then choose the **OK** button to exit. When you select this check box, it displays the attributes and makes them available for editing. Next, you can select any part of the block with attributes that need to be edited and press ENTER. If you want to edit some other objects that are part of the block, you can select them too. All the selected objects become a part of the working set. You will notice that when you select the block with attributes, the tags replace the assigned attribute values and objects in the drawing that were not selected appear faded.

You can now edit the block geometry or edit only the attributes. To edit an attribute, double-click on it; the respective **Edit Attribute** dialog box will be displayed, refer to Figure 17-51. Using the options in this dialog box, you can change the value of the attribute.

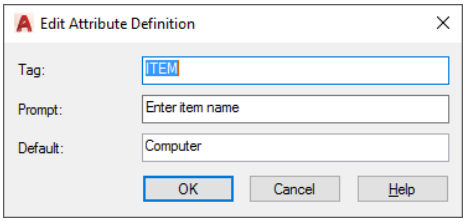

Figure 17-51 *The* ***Edit Attribute Definition*** *dialog box*

INSERTING TEXT FILES IN THE DRAWING

Ribbon: Annotate > Text > Text drop-down > Multiline Text **Command:** MTEXT
Menu Bar: Draw > Text > Multiline Text

A You can insert a text file in a drawing by using the **Import Text** option that is displayed when you right-click in the text window.

On invoking the **Multiline Text** tool, AutoCAD prompts you to enter the insertion point and the other corner of the paragraph text box within which the text file will be placed. After you specify these points, the text window is displayed. To insert the text file *TEST.txt* (created in Example 3), right-click in the text window to display the shortcut menu. In this shortcut menu, choose the **Import Text** option. AutoCAD displays the **Select File** dialog box.

In the **Select File** dialog box, you can select the **TEST** text file from the list box and then choose the **Open** button. The imported text is displayed in the text window, refer to Figure 17-52. Note that only the ASCII files are properly interpreted. Now, click on the screen outside the text window to get the imported text in the selected area on the screen, refer to Figure 17-53.

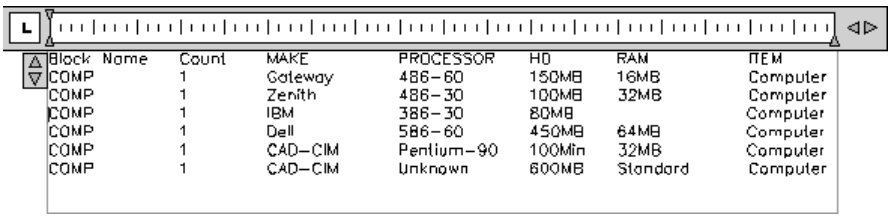

Figure 17-52 *The* ***Text Editor*** *displaying the imported text*

You can also use the **Multiline Text** tool to change the text style, height, direction, width, rotation, line spacing, and attachment.

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	80MB8MB		Computer
COMP	1	Dell	586–60	450MB	64MB	Computer
COMP	1	CAD–CIM	Pentium–90	100Min	32MB	Computer
COMP	1	CAD–CIM	Unknown	600MB	Standard	Computer

Figure 17-53 *The imported text file on the screen*

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. If you have selected the **Align** option from the **Justification** drop-down list in the **Text Options** tab of the **Enhanced Attribute Editor** dialog box, the **Height** and **Rotation** edit boxes will be _____.

2. You can use the _____ tool of the **Edit Attribute** drop-down in the _____ panel of the **Insert** tab to edit attribute definitions.

3. If the value of the **ATTDIA** variable is _____, it disables the **Edit Attribute** dialog box.

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

5. The _____ tool is used to redefine an existing block with attributes in a drawing.

6. Like the **Constant** attribute, the **Preset** attribute cannot be edited. (T/F)

7. In case of tag names, any lowercase letter is automatically converted into an uppercase letter. (T/F)

8. You can choose the **Single** tool from the **Edit Attribute** drop-down in the **Block** panel to modify the justification, height, or style of an attribute value. (T/F)

9. If you select the **Display All Attributes** option in the **Attribute Display** drop-down list of the **Block** panel, then even the attributes defined with the **Invisible** mode are displayed. (T/F)

Review Questions

Answer the following questions:

1. Which of the following buttons of the **Block Attribute Manager** is used to update all instances of the selected block?

   (a) **Settings** (b) **Edit**
   (c) **Sync** (d) None of these

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

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

3. Which of the following tools is used to invoke the **Block Attribute Manager** dialog box?

   (a) **Define Attributes** (b) **Manage Attributes**
   (c) **Block Editor** (d) **Edit Reference**

4. Which of the following tools is used to update the selected block after making changes in the block?

   (a) **Create Block** (b) **Edit Reference**
   (c) **Synchronize** (d) **Edit Attribute**

5. You can insert the text file in the drawing by choosing the _____ tool in the **Tools** panel of the **Text Editor** contextual tab displayed when using the **Multiline Text** tool.

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

7. If you select the _____ check box in the **Mode** area of the **Attribute Definition** dialog box, the **Prompt** edit box will be disabled.

8. 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.

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

10. If you do not enter anything in the **Prompt** edit box, the entry made in the **Tag** edit box will be displayed as prompt. (T/F)

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

12. When using the **Edit Reference** tool for in-place editing of attributes, the existing blocks in a drawing do not get updated. Only the subsequent insertions display the modified block. (T/F)

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

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

Exercise 1 *Attribute Definition*

In this exercise, you will define the following attributes for a resistor and then create a block using the **Create Block** tool. The name of the block is **RESIS**. The distance between the dotted lines is 0.5 units.

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 17-54.

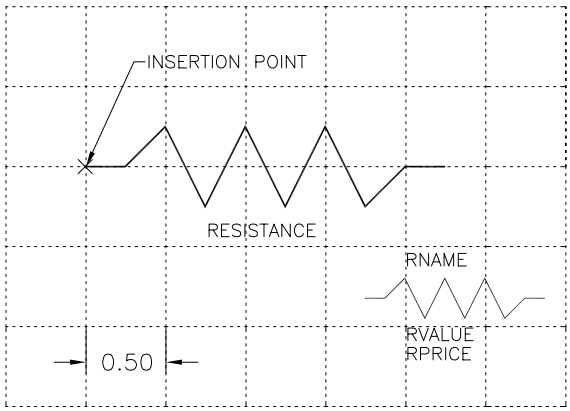

Figure 17-54 *Drawing of a resistor for Exercise 1*

2. Choose the **Define Attributes** tool 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 17-54.

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.

Exercise 2

In this exercise, you will use the **Insert** tool 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	.32
R2	27	.25
R3	52	.40

R4	8	.21
RX	10	.21

1. Draw the electric circuit diagram, as shown in Figure 17-55 (assume the dimensions).

2. Set the system variable **ATTDIA** to **1**. Use the **Insert** tool to insert the blocks and define the attribute values in the **Attribute Definition** dialog box.

3. Repeat the **Insert** tool to insert other blocks, and define their attribute values as given in the table. Save the drawing as *attexr2.dwg*, refer to Figure 17-56.

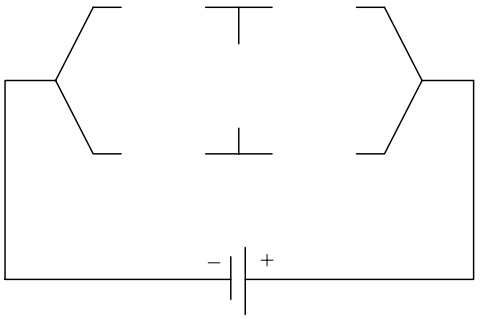

Figure 17-55 *Electric circuit diagram without resistors for Exercise 2*

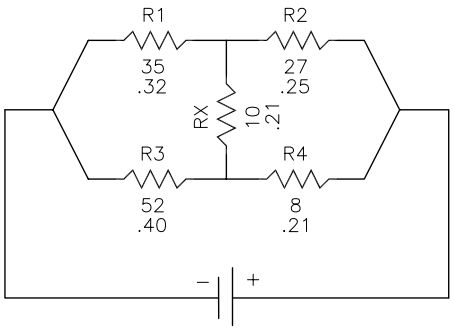

Figure 17-56 *Electric circuit diagram with resistors for Exercise 2*

Exercise 3 — *Data Extraction*

In this exercise, 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 *.txt* file format named **RESISLST** and arranged, as shown in the following table.

RESIS	1	R1	35	.32
RESIS	1	R2	27	.25
RESIS	1	R3	52	.40
RESIS	1	R4	8	.21
RESIS	1	RX	10	.21

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

2. Use the **Extract Data** tool to invoke the **Data Extraction** wizard and create a new data extraction. Next, select the **Select objects in the current drawing** radio button. Now, select the objects in the current drawing, whose attributes have to be extracted.

3. Choose the **Settings** button to invoke the **Additional Settings** dialog box. On the **Additional Settings** page, clear both the check boxes, of the **Extract objects from blocks** and **Extract objects from xrefs** and choose **OK** and then the **Next** button.

4. In the **Select Objects** page, under the object name heading, select the check box named **RESIS** if not selected by default. Choose **Next**; the **Select Properties** page is displayed.

5. In the **Select Properties** page, the list of properties that you want to extract is displayed. Clear all the check boxes, in the **Category filter** area except the **Attribute**. In the **Properties** area, clear all the check boxes and select the **RNAME**, **RVALUE**, and **RPRICE** check boxes. Choose **Next**; the **Refine Data** page is displayed, showing the preview of the data to be extracted.

6. In the **Refine Data** page you can reorder and sort columns, filter results, add formula columns, and create external data links. Choose the **Next** button; the **Choose Output** page is displayed.

7. Select **Insert data extraction table into drawing** and **Output data to external file .xls .csv .mdb .txt** check box and click on the browse button to invoke the **Save As** dialog box. Select **Text/Template/Extract file (* .txt)** in the **Files of type** drop-down. Enter *RESISLST* in the **File name** edit box and click **Save** and then the **Next** button.

8. On the **Table style** page, click **Next** and then choose the **Finish** button. You can use Notepad to open the output file, *RESISLST*. The table in the output file will be similar to the table shown at the beginning of Exercise 3.

Exercise 4 *Edit Attribute*

In this exercise, you will use the **EATTEDIT** 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 is invisible.

2. Double click on the block to edit the values that are highlighted in the following table.

RESIS	R1	**40**	.32
RESIS	R2	**29**	.25
RESIS	R3	52	**.45**
RESIS	R4	8	**.25**
RESIS	**R5**	10	.21

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

4. Use the **Multiline Text** tool to insert the text file in the drawing.

Exercise 5 *Attribute Data Extraction*

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** or **-ATTEXT** commands to extract the attribute values.

Exercise 6 *Attribute Data Extraction*

In this exercise, you will create the blocks with the required attributes. Next, draw the circuit diagram shown in Figure 17-57 and then extract the attributes to create the Bill of Materials.

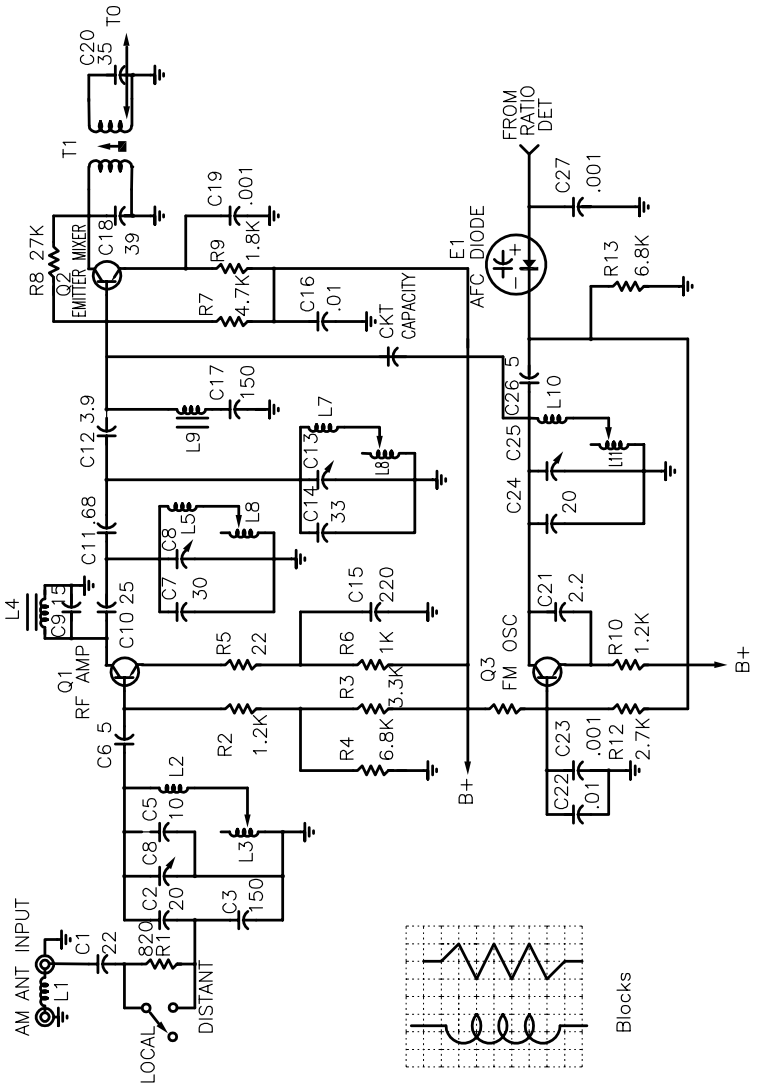

Figure 17-57 Drawing of the circuit diagram for Exercise 6

Exercise 7 *Data Extraction*

In this exercise, you will create the blocks with the required attributes. Next, draw the circuit diagram shown in Figure 17-58 and then extract the attributes to create the Bill Of Materials.

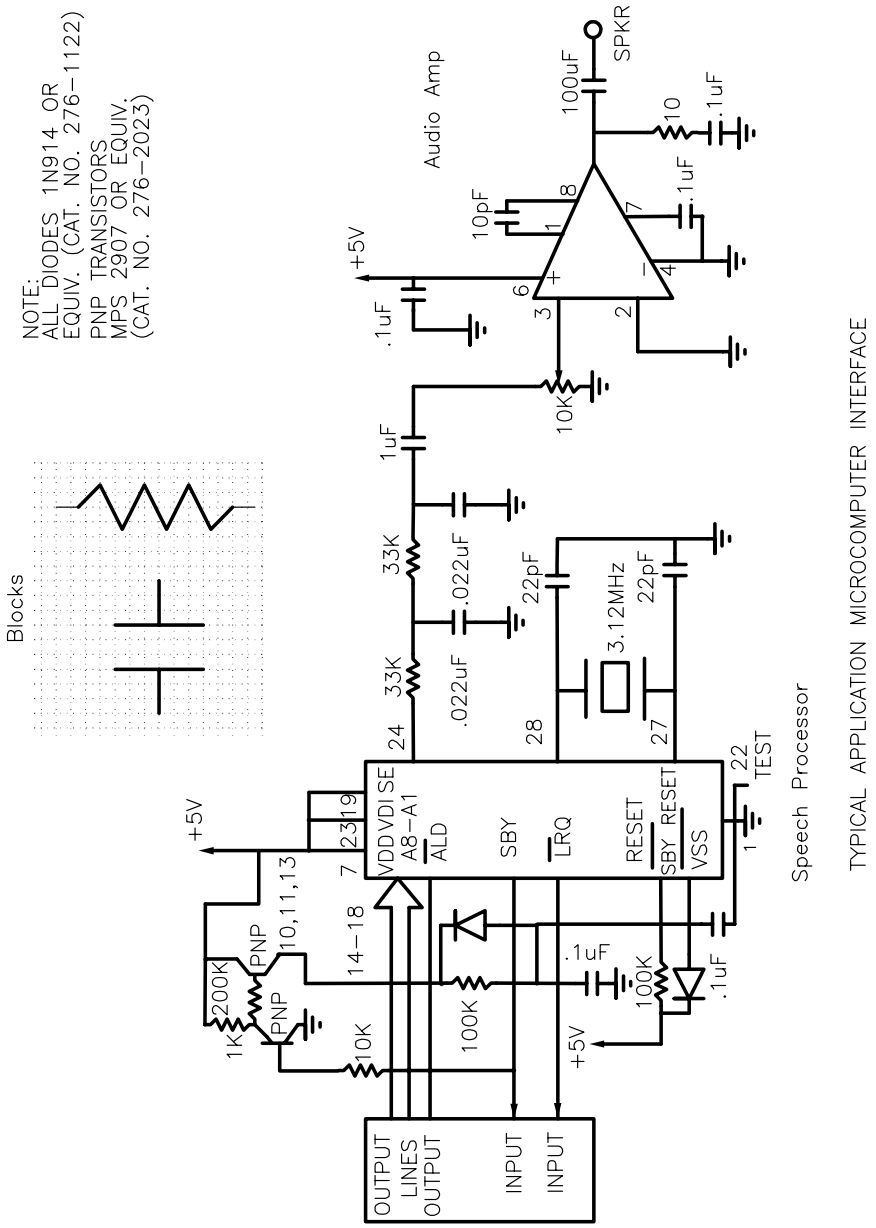

Figure 17-58 *Drawing for Exercise 7*

Answers to Self-Evaluation Test

1. disabled, **2**. **Single**, **Block**, **3**. 0, **4**. Space Delimited, **5**. **Edit Reference**, **6**. F, **7**. T, **8**. T, **9**. T