

#### IN THIS CHAPTER

- » Creating and inserting blocks
- » Counting blocks
- » Replacing and managing blocks
- » Using attributes to store information

## Chapter **1**

# Playing with Blocks

This book looks at creating and managing reusable content, which is content that needs to be created only once and then can be used in many different drawings. The ability to reuse content was a huge advantage for those early drafters that dared to venture into the unknown world of CAD so many years ago. If you've ever had the pleasure of drafting on a board, you may have used plastic templates for things such as circles, furniture in a house plan, and mechanical fasteners. Those plastic templates allowed you to draw the same objects over and over without drawing them completely from scratch each time. As you will find out, blocks are AutoCAD's equivalent to plastic drafting templates, except they are much more powerful.

## Working with Reusable Content

AutoCAD allows you to create blocks that act like geometry templates. A *block* is an object that is made up of many different graphical objects and is assigned a name to reference when you want to insert the block into a drawing. Blocks are great for providing a consistent look to your drawings. You may have already used a block in your drawings for a title block that contains a project name, date, and other

information that relates to a particular drawing. Blocks can be defined with static or dynamic geometry and contain attributes to store static text or allow users to enter their own text value. Attributes allow you to assign a varying text value that identifies a feature of the physical object that the block represents or to store information about the drawing. An attribute can have a different value for each instance of a block with the same name inserted into a drawing. Attributes are often used to store costing, descriptions, room numbers, and other information that you might want to extract out of a drawing. The extracted attribute information can be imported into an external application, such as Microsoft Excel, or even placed in a drawing as a table object that can represent a bill of materials (BOM) or door and window schedule.

Blocks are not the only form of reusable content in AutoCAD. AutoCAD can reference entire drawings and other files into a drawing, called *external referencing*. When a drawing file is referenced in another drawing, it is referred to as an *xref*. Xrefs allow you to share drawings with others who may need to ensure that their drawings match up with yours. For example, you can share a floor plan with the building systems contractor and the landscaper at one time. If the floor plan is used as an xref, the next time the drawing is opened by the contractors, the updates are displayed (of course, the contractors need the updated drawing file). AutoCAD is also capable of referencing raster images and PDF, DWF, DWFx, and DGN files.

Drawings often display schedules. Sometimes drafters create this information in a word processor or spreadsheet program. AutoCAD allows you to copy and paste content from Windows-based applications through the use of the Windows Clipboard, but doing so causes the information to become static. AutoCAD can keep some of this information dynamic through the use of Object Linking and Embedding (OLE) and DataLinks.

Being able to reuse content has advantages, but being able to access that content quickly makes it even more valuable. AutoCAD provides three interfaces that allow you to organize and access content quickly; these interfaces are the Blocks palette, the DesignCenter, and the Tool Palettes window. I cover the Blocks palette in this chapter, and DesignCenter and the Tool Palettes window in Chapter 4 of this book.

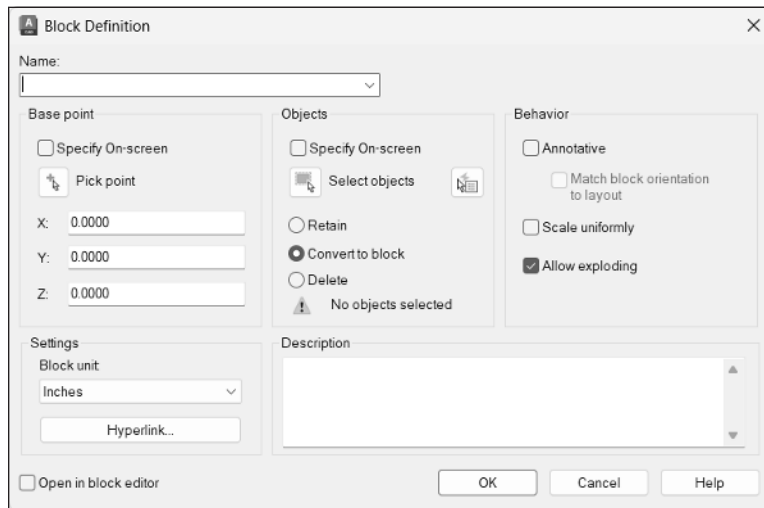
## Creating Blocks



Blocks can contain the same geometry and annotation objects that you add to a drawing file, such as lines, circles, arcs, dimensions, and multiline text. Objects that are part of a block look just like individual objects drawn in a drawing.

AutoCAD allows you to create a block directly from the objects in a drawing without needing to do much more. To create a block, you use the **BLOCK** command to display the Block Definition dialog box (see Figure 1-1). When you create a block, you need to know three things:

- » **Name:** The text string that will be used to identify the block when inserting it into the drawing.
- » **Base point:** The point on a block that helps place it into the drawing when you are inserting it. This point is usually established on an object that's selected when creating the block, but you don't have to use the same point to insert the block.
- » **Selected objects:** The objects to define the way the block looks and behaves when inserted into a drawing.



**FIGURE 1-1:**  
The Block  
Definition  
dialog box.

## Accessing the Block Definition dialog box

To start the **BLOCK** command and display the Block Definition dialog box, use one of the following methods:

- » On the ribbon, click the Insert tab → Block Definition panel → Create/Write Block drop-down menu → Create Block button.
- » At the command prompt, type **BLOCK** or **B** and press Enter.

Follow these steps to create a block:

**1. Add the geometry to your drawing that you want to use to define a block.**

If you want to control the visibility of the objects within the block later, be sure to place the objects on specific layers so that they can be turned on and off or frozen and thawed.

**2. Use one of the methods previously listed to initiate the BLOCK command.**

The Block Definition dialog box (see Figure 1-1) is displayed.

**3. In the Block Definition dialog box, enter a meaningful name in the Name combo box.**

The name entered must be unique from other blocks that exist in the drawing, or you end up redefining an existing block with the new geometry, which might not be what you want to do. The name entered must be fewer than 255 characters.

**4. In the Base point area, either click the Pick Point button and specify a point in the drawing or enter a coordinate value for the base point in the X, Y, and Z text boxes.**

The point that you specify will be used as the base point of the block or its insertion point. Usually this point is on the geometry that's selected to define the block.

If you clicked the Pick Point button, you are returned to the drawing window. Specify a point in the drawing to use as the base point for the block. After you specify the base point, you are returned to the Block Definition dialog box.

**5. In the Objects area, click the Select Objects button.**

You're returned to the drawing window once again so you can select the objects that you want to add to the block.

**6. In the drawing window, select the objects you want to add to the block. Press Enter to complete selecting objects and return to the Block Definition dialog box.**

The objects you select can be text, lines, circles, and even other blocks. Any object you can select can be added to the block. You can use the Quick Select button to the right of the Select Objects button to filter objects by specific property values. (See bonus Chapter 1 on the web, [www.dummies.com/go/autocad&autocad1taiofd](http://www.dummies.com/go/autocad&autocad1taiofd), to learn more about Quick Select.)

**7. In the Objects area, click one of the Select Objects modes.**

The different object modes affect what happens to the selected objects after the block is created:

- **Retain:** The selected objects are used to define the block but are retained as is in the drawing.
- **Convert to block:** The selected objects are removed from the drawing and the new block is inserted in the same location as the original selected objects.
- **Delete:** The block is defined using the selected objects and the selected objects are then removed from the drawing.

## 8. Deselect the Open in Block Editor option (in the lower-left corner of the dialog box) for now; I cover that option in a little bit.

If the Open in Block Editor is selected, the block is opened in the Block Editor when the Block Definition dialog box is closed. See Chapter 2 of this book for more information on the Block Editor and on adding dynamic properties and actions to a block.

## 9. Click OK.

The Block Definition dialog box closes. Results vary based on the objects selected, the selected objects mode, and the other settings you choose.



TIP

Creating geometry on Layer 0 (zero) is usually a “no-no” in AutoCAD, but when geometry is added to a block that’s on Layer 0, it inherits the properties of the layer on which the block is inserted. This is a great way to verify that a block is on the correct layer.

## Converting repetitive geometry into blocks



NEW

Along with manually creating a block with the BLOCK command, starting with AutoCAD 2025, there are two new features that allow you to convert repetitive geometry into blocks. These two features are as follows:



- » **Search and Convert:** Allows you to select objects and search the drawing for identical instances of the selected geometry and then convert the selected objects into a block and replace all found instances with a block. You start the Search and Convert feature with the BSEARCH (or BCONVERT) command, which displays the Convert to Block dialog box (see Figure 1-2).



- » **Detect and Convert (or Object Detection):** Utilizes Artificial Intelligence (AI) to identify identical instances of geometry in a drawing and then converts the matches into a block and replaces all found instances with a block. You start the Detect and Convert feature with the BDETECT (or DETECT) command, which displays the Detect palette.



AUTOCAD LT

AutoCAD LT doesn't support the DETECT command or the Detect palette.

To search for repetitive objects and convert all instances into a block, follow these steps:

- 1. On the ribbon, click the Insert tab⇨Block panel⇨Detect/Search and Convert drop-down menu⇨Search and Convert button.**

The BSEARCH (or BCONVERT) command starts and the following prompt is displayed:

```
Select object to find matching instances to convert to blocks:
```

- 2. At the Select object prompt, select the objects you want to search for identical instances and convert into a block. Press Enter to end object selection.**

The objects you select are used to define the pattern of objects you want to search for within the drawing and then convert to a new block or even replace with an existing block. The Search and Convert toolbar is displayed and the matching instances are identified in the drawing window.

- 3. In the drawing window, use the Search and Convert toolbar along the top of the drawing window to review the instances found. Then click Convert on the toolbar.**

The Convert to Block dialog box (refer to Figure 1-2) is displayed.

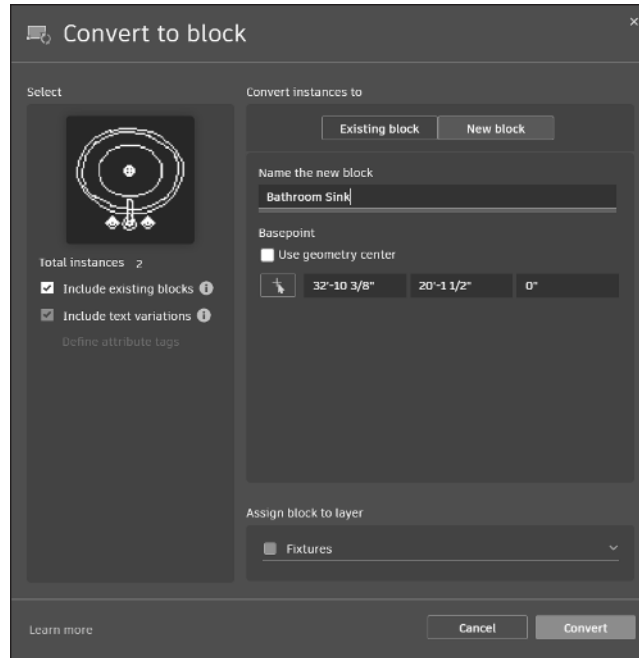
- 4. In the Convert to Block dialog box, click New Block under the Convert Instances To area.**
- 5. In the Name the New Block text box, enter a meaningful name for the new block.**
- 6. Under the Basepoint area, click Pick Basepoint (which has a plus and arrow cursor on it) and specify an insertion point for the new block in the drawing window.**

You are returned to the drawing where you can specify a base point for the new block. After you specify a point, you are returned to the Convert to Block dialog box.

- 7. Under the Assign Block to Layer area, click the drop-down list and select a layer on which the instances of the new block should be placed.**
- 8. Click Convert and then press Esc to deselect the newly inserted block instances.**

The originally selected objects are converted to a new block, and all other identical instances of the objects are replaced with the new block and placed on the specified layer.

**FIGURE 1-2:**  
Converting  
identical  
instances of  
objects to a  
block with the  
Convert to Block  
dialog box.



## Inserting Blocks



AutoCAD stores blocks in what is called the *block definition table*. The block definition table holds information and settings about all the block definitions in a drawing. To display a block in the drawing after you create it, you need to insert a reference of the block definition. An inserted block in the drawing is known as a *block reference* because it refers to its definition in the block table.

To insert a block into the current drawing file, you use the `BLOCKSPALETTE (INSERT)` command, which displays the Blocks palette (see Figure 1-3), or the Insert dialog box if you are using AutoCAD 2019 or earlier. The Blocks palette can insert a block defined in the current drawing, an external drawing file as a block from the Favorites list, or even a block library. It just depends on where the information you want to insert is located. You use the four tabs along the side of the Blocks palette to access blocks from the different sources and locations.

To start the `BLOCKSPALETTE` command and display the Blocks palette, use one of the following methods:

- »» On the ribbon, click the View tab → Palettes panel → Blocks button.
- »» At the command prompt, type **BLOCKSPALETTE**, **INSERT**, or **I** and press Enter.



TIP

If you are using AutoCAD 2019 or earlier, use the INSERT command to insert a block with the Insert dialog box. The options in the Insert dialog box are similar to the Blocks palette, but the following steps vary slightly. If you need information on inserting a block, see AutoCAD's online Help system.

To insert a block into a drawing, follow these steps:

1. **Start the BLOCKSPALETTE command by using one of the previously listed methods.**

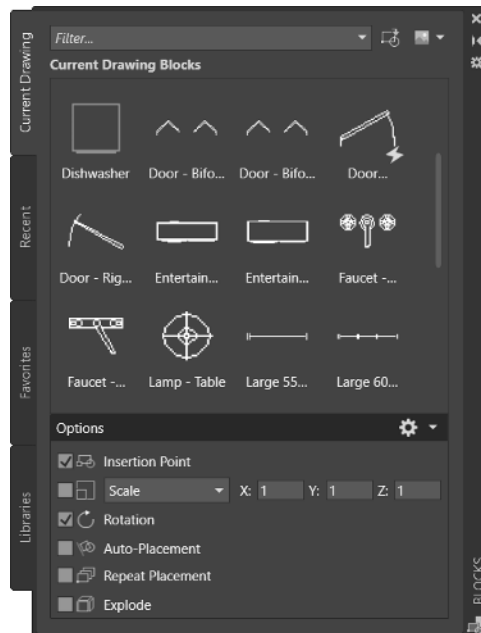
The Blocks palette (refer to Figure 1-3) is displayed.

2. **On the Blocks palette, click the Current Drawing tab.**

The Current Drawing tab is set as current. From this tab, you can access the blocks defined in the current drawing or insert a drawing file. Before clicking a block preview or selecting a drawing file, you want to specify the insertion options.

3. **Under the Options area, select or deselect the Insertion Point option. If it's deselected, enter the coordinate value for the insertion point of the block in the X, Y, and Z text boxes.**

If Insertion Point is selected, you will be prompted for an insertion point during the block insertion process after a block preview is clicked or drawing file is selected.



**FIGURE 1-3:** The Blocks palette is used to insert a reference into a block definition.

4. **Select or deselect the Scale option. If it's deselected, enter the scale for the X, Y, and Z axes in the text boxes. From the drop-down list, select Uniform Scale to scale the block equally in all three axes based on the entered value.**

If Scale is selected, you will be prompted for the scale of the block during the block insertion process. If the block was created with the Scale Uniformly option selected, the scaling method is restricted to Uniform Scale.

5. **Select or deselect the Rotation option. If it's deselected, enter the rotation angle in the Angle text box.**

If Rotation is selected, you are prompted for the rotation angle of the block during the block insertion process.

6. **Deselect the Auto-Placement, Repeat Placement, and Explode options.**

When it's selected, the Auto-Placement uses suggestive placement logic to help insert the block into the drawing, while Repeat Placement allows you to keep inserting the block until the Esc key is pressed. Finally, if Explode is selected, the block is exploded after being inserted. The Explode option is disabled if you deselected Allow Exploding option in the Block Definition dialog box when the block was created.

7. **In the Blocks area, click the preview of the block you want to insert, or click the Insert File as Block button to select and insert a drawing as a block.**

If you clicked a block preview, you are returned to the drawing window and the block insertion process starts. Follow the prompts displayed at the command prompt to insert the block. If you clicked the Insert DWG as Block button, the Select File to Insert dialog box appears.

8. **If you clicked the Insert File as Block button, in the Select File to Insert dialog box, browse and select the drawing file that you want to insert as a block. Click Open.**

You are returned to the drawing window and the block insertion process starts for the selected drawing file.



TIP

If you selected the Auto-Placement from the Options area before clicking a block preview, you can temporarily disable Auto-Placement while dragging the block by pressing and holding Shift+W or Shift+[.



TIP

When inserting a block, you can specify its rotation and scale by using the Properties palette if it's displayed before you specify the insertion point of the block. Using the Properties palette this way also allows you to change the attribute values and dynamic properties of a block before it's inserted. I cover attributes later in this chapter and dynamic properties of blocks in Chapter 2 of this book.

# Counting Blocks

Blocks inserted into a drawing can be counted. You might want to do this so that you know the number of instances of a block in a drawing in order to create a bill of materials or even a door and window schedule. AutoCAD provides several ways to count blocks in a drawing, and which method you choose depends on what you plan on doing with the block count.



AUTOCAD LT

The first and simplest way to count blocks in a drawing is with the BCOUNT command, which is part of Express Tools. This command displays a basic list of the blocks inserted into the drawing and the number of instances of each block in the Command Line window. Press F2 to expand the Command Line history or press Ctrl+F2 to open the Text window to better view the list of blocks. AutoCAD LT doesn't support the BCOUNT command, as it is part of Express Tools.

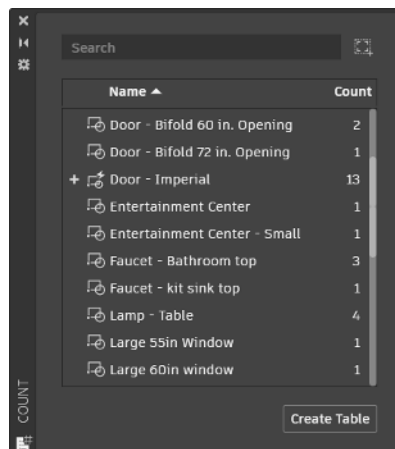


Another way to count blocks is using the Quick Select dialog box. The Quick Select dialog box is displayed using the QSELECT command. Click the Select Objects button to select the objects in the area you want to count. Use the Object Type drop-down list to filter out all objects except blocks by selecting Block Reference. From the Properties list, choose Name and then select the name of the block to count from the Value drop-down list and click OK. The count of the block references selected is displayed in the Command Line window. (See Bonus Chapter 1 on the web at [dummies.com/go/autocad&autocadltaiofd](http://dummies.com/go/autocad&autocadltaiofd) for more information on Quick Select.)



NEW

Yet another way to count block instances is with the Count feature, which was introduced in AutoCAD 2022. The Count feature quickly counts block instances in a drawing. The blocks inserted into the drawing and the number of instances of each block is displayed on the Count palette (see Figure 1-4).



**FIGURE 1-4:** Counting blocks can be as easy as 1, 2, 3 with the Count palette.

| Item                         | Count |
|------------------------------|-------|
| Door - Bifold 60 in. Opening | 2     |
| Door - Bifold 72 in. Opening | 1     |
| Door - Imperial:Open 30°     | 13    |
| Large 55in Window            | 1     |
| Large 60in window            | 1     |
| Window - 36 in.              | 6     |



You can count all block instances in the entire drawing or within a selected area. With the blocks counted, you can choose to simply view the count on the Count palette or create a table with the name of the blocks and the number of instances of each block. As blocks are added or removed from a drawing, the count of block instances is updated in the table. You can display the Count palette with the COUNTLIST command.

As if there weren't enough ways to count blocks already, you can also use the Data Extraction wizard to count blocks and other geometry in a drawing. I mention the Data Extraction wizard later in this chapter under the "Extracting attribute data from blocks" section.

## Replacing Blocks



As you work on a drawing, chances are you will have to replace one block with another. For example, perhaps the block you initially created representing a valve of one type now needs to be changed to a different type of valve. While this might be a simple task if you only need to replace one or two blocks in a drawing, it's a very different task if you must replace tens or hundreds of instances of the same block throughout a drawing.



NEW

Starting with AutoCAD 2024, you can replace selected block references with another block defined in the drawing. To replace blocks in a drawing, you use the BREPLACE command, which prompts you for the blocks to replace and then displays the Block Replacement dialog box (see Figure 1-5). From the Block Replacement dialog box, choose the block you want to use as the replacement for the selected blocks.



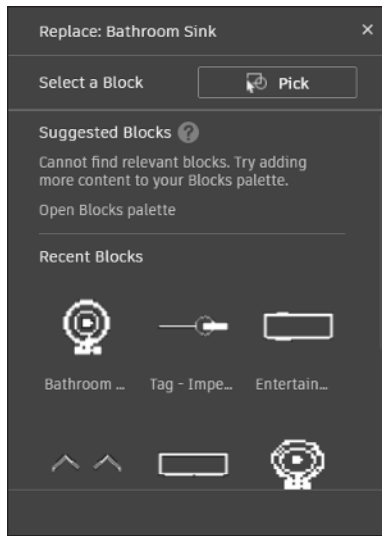
AUTOCAD LT

AutoCAD LT doesn't support the BREPLACE command.



TIP

Prior to AutoCAD 2024, you could use the BLOCKREPLACE command from Express Tools to replace all instances of one block with another block, not individual selected blocks like BREPLACE allows. Recall that Express Tools are not available in AutoCAD LT.



**FIGURE 1-5:**  
Replacing  
selected block  
references with a  
different block.

## Managing Blocks

Blocks are a powerful feature of AutoCAD and no doubt you will be using them as often as you can. AutoCAD offers several tools and commands for managing blocks. Some of the ways to manage blocks in a drawing are as follows:

- » Rename a block definition
- » Redefine a block definition
- » Purge a block definition
- » Export a block definition as a drawing file

### Renaming a block definition



You can rename blocks with the `RENAME` command, which displays the Rename dialog box. In the Rename dialog box, select Block from the Named Objects list and then select the block to rename. Next, enter the new name for the block and click the Rename To button.

### Redefining a block definition



You can redefine (or edit) a block definition. In earlier releases, you would insert a block into the drawing and then explode it. After exploding the block, you make the

necessary changes to the geometry and then reblock it using the BLOCK command again. Or you might use the REFEDIT command, which allows you to open a block or an xref so the geometry can be modified. Some issues with using the REFEDIT command can be overcome with the BEDIT command. I cover the REFEDIT command in Chapter 3 of this book.



Redefining a block with the Block Editor is much simpler than exploding and reblocking a block after making changes to its geometry. The Block Editor allows you to edit a block in its own drawing window. You use the BEDIT command to start the Block Editor. Follow these steps to edit a block definition with the Block Editor:

1. **In the drawing window, select the block that you want to redefine.**
2. **Right-click and choose Block Editor from the shortcut menu.**  
The block definition is opened in the Block Editor.
3. **In the Block Editor, make your changes to the existing geometry or add geometry as you normally do when working in the drawing window.**
4. **On the ribbon, click the Editor contextual tab → Close panel → Close Block Editor button to exit the Block Editor.**
5. **When prompted to save the changes, click Yes.**

You're returned to the drawing window, and the references to the edited block are updated.

## Purging a block definition



You can remove block definitions that are not being used and will never be used by using the PURGE command and Purge dialog box. In the Purge dialog box, click the Purgeable Items tab and expand the All Items node. Then select the Blocks node to purge all unused blocks or expand the Blocks node to select the individual block definitions you want to remove. After selecting the blocks to remove, click Purge Checked Items. (See Bonus Chapter 1 on the web at [dummies.com/go/autocad&autocad1taiofd](http://dummies.com/go/autocad&autocad1taiofd) or the PURGE command topic in AutoCAD's online Help system for more information on the purging unused blocks and named objects.)



TIP

To quickly purge all unused named objects from a drawing, select the Purge Nested Items option and deselect the Confirm Each Item to Be Purged option in the Purge dialog box. Then click Purge All. This purges all nested named objects out of a drawing, without stopping to prompt you.

## Exporting a block definition



Blocks created by using the `BLOCK` or `BEDIT` commands are defined only in the current drawing, but many times you will want to be able to access the block in other drawings. The Write Block dialog box allows you to export a block from the current drawing to a new drawing file. The Write Block dialog box is displayed with the `WBLOCK` command. Once a block has been written to a drawing file, you can click the Insert DWG as Block button on the Current Drawing of the Blocks palette or use DesignCenter to insert blocks stored in a drawing; I cover DesignCenter in Chapter 4 of this book. For more information on the `WBLOCK` command, see AutoCAD's online Help system.



REMEMBER



TIP

Drawings created with the `WBLOCK` command are saved in the latest drawing file format.

When you open a drawing, you can change the insertion base point of the drawing file that is used when inserting it into another drawing. To redefine the insertion base point of a drawing, you use the `BASE` command or the `INSBASE` system variable.

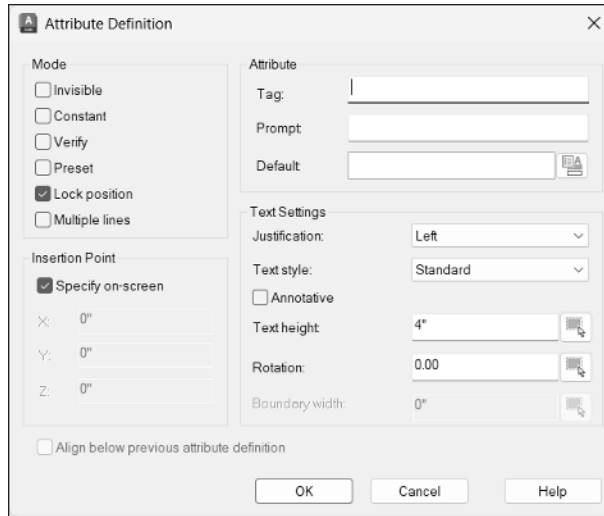
## Enhancing Blocks with Attributes

Blocks can store custom information, and that information can later be extracted to an external file or displayed in the drawing as a table object (not available in AutoCAD LT). To store custom information with a block, you use an attribute. You use attributes to hold information in a block such as a part number, a description or even project-related information. You will often find that title blocks contain attributes, which allow the user to edit all the attributes at one time instead of individually, as you have to do with text or multiline text. Attributes may appear like standard text objects, but they're much more than that. You can change the text value of attributes when they're part of a block.

### Adding an attribute to a block definition



Attributes are added to a block in the same way you add other types of geometry, such as lines or arcs. First you add an attribute definition to the drawing and then you add the attribute definition to the block. The attribute definition is stored in the block definition and describes what the actual attribute of the block reference should be like. When a block is inserted into the drawing, attributes are added to a block reference based on each attribute definition of the block. To create an attribute definition, you use the `ATTDEF` command to display the Attribute Definition dialog box (see Figure 1-6).



**FIGURE 1-6:** Creating an attribute definition.

To start the `ATTDEF` command and display the Attribute Definition dialog box, follow one of the following methods:

- » On the ribbon, click the Insert tab → Block Definition panel → Define Attributes button.
- » When the Block Editor is active, on the ribbon, click the Block Editor tab → Action Parameters panel → Attribute Definition button.
- » At the command prompt, type **ATTDEF** or **ATT** and press Enter.

The following steps explain how to create an attribute definition and add it to a block:

1. Add the geometry to your drawing that you want to use to define a block.
2. Start the `ATTDEF` command by using one of the methods previously listed.

The Attribute Definition dialog box appears (refer to Figure 1-6).

3. In the Attribute Definition dialog box, under the Mode area, select the options that you want to use for the attribute definition.

Attributes can have six modes:

- **Invisible:** By default, attributes are visible. When this option is selected, the attribute is invisible. The attribute's value is still editable and readable in the Properties palette, but it will not be displayed in the drawing and will not plot. If an attribute is invisible, you can set the `ATTMODE` system variable to a value of 2 to display all invisible attributes in a drawing.

- **Constant:** When selected, the attribute has a fixed value that is assigned to all references of the block inserted into the drawing and can't be changed unless you use a command such as BEDIT or BATTMAN (Block Attribute Manager).
- **Verify:** When selected, you're prompted to verify that the value entered is correct for the block being inserted into a drawing.
- **Preset:** When selected, the default value is automatically assigned to the block when it's inserted into a drawing.
- **Lock position:** When selected, the attribute can't be moved by using grips. You use this mode also when you want to control the placement of the attribute with dynamic properties. I cover working with dynamic properties in Chapter 2 of this book.
- **Multiple lines:** When selected, the attribute supports multiple lines of text instead of a single line of text.

#### 4. In the Attribute area, specify the attribute's tag, prompt, and default value.

The tag is the text you see onscreen before the attribute is added to a block. The prompt is what the user sees at the command prompt, or in the Edit Attributes dialog box or the Enhanced Attribute Editor (AutoCAD only), when inserting or editing the attribute values of a block. The default value defines the value that the attribute has when it's inserted in a drawing. The default value can be a plain text string that contains field values or it can be defined with special multiline formatting. I cover how to use fields and the BlockPlaceholder field in the "Using block placeholder fields" section later in this chapter.

#### 5. In the Insertion Point area, select or deselect the Specify On-Screen option. If it's deselected, enter the coordinate value for the insertion point of the block in the X, Y, and Z text boxes.

If Specify On-Screen is selected, you will be prompted for the insertion point of the attribute after you click OK to close the Attribute Definition dialog box.

#### 6. In the Text Settings area, specify the attribute's justification, text style, height, rotation, and boundary width (if the Multiple Lines box in the Mode area is selected).

Justification of the attribute controls how the value of the attribute appears at its insertion point. The text style controls the font and other text characteristics used for the tag and value of the attribute. The Annotative checkbox controls whether the attribute scales based on the current annotation scale or matches the orientation of the block. Text height specifies the height of the text, and rotation controls the rotation angle after the attribute is created. The text height and rotation options both have buttons that can be used to specify a distance or angle by using points in the drawing. Boundary width defines the

width of the bounding box for the attribute text when the attribute is set to Multiple Lines; you can enter a number or click Boundary Width to pick a distance in the drawing.

- 7. Initially, the Align below Previous Attribute Definition checkbox is deselected. Select the option if you have inserted an attribute in your drawing and want the next one placed below the one previously created.**

When selected, the attribute is positioned below the previously created attribute.

- 8. Click OK.**

The Attribute Definition dialog box closes, and you're returned to the drawing window. If you selected the Specify On-Screen options for the insertion point, specify an insertion point for the attribute definition. If you're prompted for an insertion point, the following prompt is displayed:

Specify start point:

- 9. Start the BLOCK command and create the block as you normally do — just don't forget to add the attribute.**



TIP

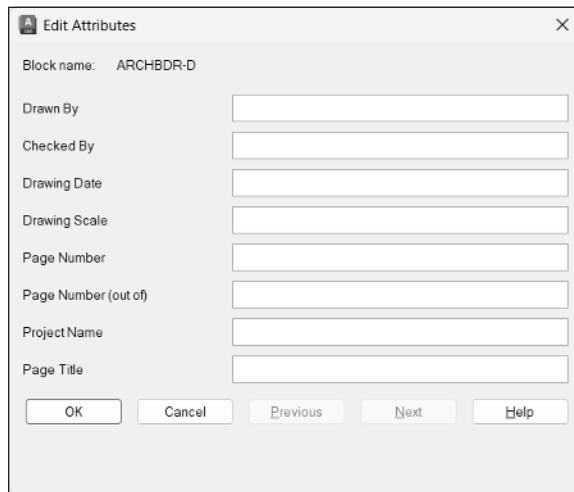
When you insert a block with attributes or edit the attribute values, you're prompted to edit the attribute values in a specific order. The order of the attributes is determined by the order they're selected when the block is created. Select each attribute in the order you want them to be displayed when the block is inserted or edited. You can use the BATTMAN command to change the order of attributes in a block definition.

## Using block placeholder fields

When adding and editing attribute values, you can utilize text and fields. Along with all the standard fields, you can also use a special field named *BlockPlaceholder* in block attributes. The *BlockPlaceholder* field allows you to access information about the values of custom properties of an inserted block reference. To add a field to an attribute's default value, when you create an attribute with the Attribute Definition dialog box (using the ATTDEF command), right-click over the Default Value text box and choose Insert Field. In the Field dialog box, select the field you want to use along with the property and format based on the field selected. Click OK to add the field to the Default Value text box. When editing the attributes of a block, in the Value text box for the attribute, right-click and choose Insert Field or double-click a field value to open the Field dialog box. For more information on fields, see Book 3, Chapter 1.

## Inserting a block with attributes

You insert a block with attributes the same way you insert a block without attributes, except after the block has been inserted into the drawing, you're prompted to change the values for the attributes contained in the block, except for those flagged as constant or preset with the Edit Attributes dialog box (see Figure 1-7) or at the command prompt.



**FIGURE 1-7:**  
Editing attribute  
values after a  
block is inserted.



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The `ATTDIA` and `ATTREQ` system variables affect how a block with attributes is inserted. `ATTDIA` controls prompting for attribute values with the Edit Attributes dialog box at the command prompt when a block with attributes is inserted. `ATTREQ` enables or suppresses the prompting of attributes values when a block with attributes is inserted.

## Editing an attribute's value in a block



You can edit an attribute's value for a block by using the Properties palette, the Edit Attributes dialog box (`ATTEDIT`), or the Enhanced Attribute Editor (`EATTEDIT`) — this works with AutoCAD only. The easiest way to edit an attribute value is to double-click the attribute you want to edit or a block that contains attributes. Doing so launches the default attribute editor. Make the changes to the values of the attributes in the dialog box that is displayed.

## Managing attributes in blocks

When you redefine blocks with attributes, you need to be aware of something. When you use the `BLOCK` or `BEDIT` command to update a block with attributes, only the changes to the objects in the block are reflected in the drawing unless the block contains attributes that are defined as constant. Any redefined block containing nonconstant attributes will take some extra effort to get them to update correctly in the drawing. If you redefine a block and add or change the attributes of a block, you must do one of the following:



- » Delete and reinsert the block so all the attributes and property changes are used as part of the block reference.
- » Use the `ATTSYNC` (Attribute Synchronize) command to ensure that all attributes in the existing blocks inserted into the drawing are updated. (Available only in AutoCAD.)
- » Use the `BATTMAN` (Block Attribute Manager) command to modify the properties of attributes in a block. You can use this command to change the order in which attributes appear when inserted along with other properties such as color, layer, and even text style.

## Extracting attribute data from blocks



You can extract attributes from a drawing by using the Attribute Extraction dialog box (`ATTEXT`), which you can find in both AutoCAD and AutoCAD LT.



You can also use the Data Extraction wizard (`DATAEXTRACTION`), which is available only in AutoCAD. Attributes can be extracted to external data files and brought into your favorite spreadsheet or database program, or in AutoCAD you can extract attribute values and create a table object that summarizes all like values. For more information on extracting attributes out of your drawing, search for “extract attributes” in AutoCAD’s online Help system.

