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AutoCAD 2006 VBA

A Programmer's Reference

Joe Sutphin

Apress®

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*This book is dedicated to my wife, Grace.
Without her, I would not be able to
accomplish the task of writing a book.*

Contents at a Glance

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About the Author

JOE SUTPHIN's background includes more than 25 years in the machinery manufacturing industry. He has more than 18 years of CAD experience with 12+ years of AutoCAD-specific experience. Joe is an Autodesk-registered developer, and his work has appeared in the pages of *Cadence* and *Cadalyst* magazines. He has been programming for nearly 20 years, with the last 15 years being Visual Basic–specific experience. In 1998 he collaborated with Microsoft on a Visual Basic application case study. He is the author of the best-selling book *AutoCAD 2000 VBA Programmer's Reference*.

About the Technical Reviewers

PHILLIP ASH began programming in the early 80s when public schools had either Apple IIs or TRS-80s. DOS was loaded from a cassette player, and BASIC was the *lingua franca* of the tyro programming set. He got involved in computer-aided design as a hobby in the late 80s, and after a few semesters in an architectural design and drafting curriculum, he took a job as an AutoCAD drafter/designer at a naval architecture firm in Portsmouth, Virginia. It was there he was introduced to AutoLISP.

For the next few years, Phill wrote code exclusively in AutoLISP. He began dabbling with Visual Lisp shortly after its inception and, at about the same time, started writing macros for Excel in Visual Basic for Applications. When Autodesk included VBA in AutoCAD, everything came together.

Eight years later, Phill is a senior developer of ShipWorks, the only shipbuilding AutoCAD add-on used by Northrop Grumman Newport News in the design of next-generation aircraft carriers.

Phill lives in Virginia Beach, Virginia, with his wife, Amy; sons, Alex and Rowan; and daughter, Morgan.

STEVE JOHNSON was born in England but has lived in Perth, Western Australia, since 1984. He is married with two children and has a bachelor's degree in applied science (information science). Steve has been an AutoCAD specialist since 1985. His company, cad nauseam, provides AutoCAD-based consulting, development, support, training, and technical writing services. His software for AutoCAD is used by hundreds of clients around the world.

Steve is a contributing editor of *Cadalyst* magazine and has written the monthly column *Bug Watch* since 1995. He is also the vice president, Asia Pacific, of Cadlock and a past president of the Western Australian AutoCAD User Group (WAAUG). As a former committee member on the Applications Programming Special Interest Group of the North American Autodesk User Group, he contributed to the NAAUG newsletter. NAAUG is now known as Autodesk User Group International (AUGI). Steve was part of the AUGI Benchmark Committee, which assisted in the development of the AUGI Gauge, a comprehensive AutoCAD benchmark.

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Second, a special thanks to my technical reviewers, Phillip Ash and Steve Johnson.

A special thank you to my children—Stephen, Clair, David, and Emily—for understanding when Daddy had to work.

Lastly, thanks to those who helped and whose names I forgot to include here. To everyone in the AutoCAD community with whom I've had the pleasure of crossing paths, from the bottom of my heart, thanks!

Introduction

This book provides a concise guide to the kind of customization programmers can achieve with AutoCAD 2006. It demonstrates how to use AutoCAD through short code examples written in Visual Basic for Applications (VBA). It also includes a complete quick reference that lists all the events, methods, and properties available with AutoCAD. Finally, it describes all the constants and system variables.

What Is AutoCAD?

So, what is AutoCAD? First released in 1982 under the name MicroCAD, AutoCAD has become a powerful tool for drafting and design purposes. AutoCAD 2006 incorporates many new features to enhance flexibility and drawing control. To reflect this extra functionality, many new ActiveX objects, properties, methods, and events have been included for improved programmability.

What Is This Book About?

This book is about AutoCAD 2006 and how to use AutoCAD VBA in your applications to handle all your drawing tasks more efficiently. It shows you how to programmatically control the creation and editing of individual drawing objects, manipulate linetypes and layers, control text and dimension styles, and do much more. As you encounter each of these topics, you'll learn all about the associated objects, including their properties, methods, and events.

By interfacing with AutoCAD, you can exploit all of AutoCAD's functionality that would have taken you a long time to write yourself. This book will first help you learn how to use this functionality. Then it will become a handy reference later, when you have a question that you just can't answer.

This book splits topics into neat and intuitive segments and makes it easy to find specific information when you need it (that is, when you're coding real-world applications).

This book is divided into three main parts:

- Chapters 1 through 3 provide a rapid introduction to Visual Basic and explain the notation and commands particular to AutoCAD VBA projects.

- Chapters 4 through 22 supply a detailed breakdown of most of the AutoCAD object model, covering common tasks complete with several varied code examples demonstrating how to use the relevant objects' methods and properties.

- Finally, the quick-reference appendixes describe all the members of all the AutoCAD objects at a glance. Appendix D, on Object Model Cross-Reference, can be found on the Apress website (`www.apress.com`).

Who Is This Book For?

The book is a reference guide for AutoCAD programmers, and it's primarily designed to explain and demonstrate the features of AutoCAD 2006. As such, this isn't a beginner's guide; however, if you've programmed in any language that can interface with other COM objects, you should be able to easily understand and use this book.

In particular, the book is aimed at programmers who use AutoCAD for daily tasks and can see the benefits of customizing and automating these tasks. I present programming techniques needed to create and modify AutoCAD drawings, customize preferences, query and set system variables, and so on, using the built-in VBA.

You can customize AutoCAD to any degree of sophistication. If you can think it up, then I bet you can use AutoCAD VBA and this book to help you achieve your goal.

Tell Us What You Think

I've worked hard on this book to make it enjoyable and useful. My best reward would be to hear from you that you liked it and that it was worth the money you paid for it. I've done my best to try to understand and match your expectations.

Apress and I would like to know what you think about it. Tell us what you liked best, and what we could have done better. If you think this is just a marketing gimmick, then test us—drop us a line! We'll answer, and we'll take whatever you say under consideration for future editions. The easiest way to do so is to send e-mail to `feedback@apress.com`.

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CHAPTER 1

The VBA Integrated Development Environment (VBAIDE)

Within AutoCAD, you develop VBA programs in the Visual Basic for Applications (VBA) Integrated Development Environment (IDE). As it does with the Visual LISP IDE, Autodesk provides the VBAIDE as an integral part of many of its products, including AutoCAD. Unlike the Visual LISP IDE, however, Microsoft licenses the VBAIDE to Autodesk for inclusion in its products. Therefore, its features are from Microsoft, not Autodesk.

This chapter explores the VBAIDE environment's facets and shows you how to take advantage of its tools. It covers these topics:

- Visual Basic concepts
- Starting the editor
- Exploring the user interface
- Managing projects
- Using the text editor
- The Object Browser

Visual Basic Concepts

Since VBA is a Microsoft Windows development environment, you'll find developing in VBA easiest if you have some knowledge of Windows. If you are new to Windows, you'll find fundamental differences between programming in Windows and programming in other environments, such as Visual LISP. The next sections outline concepts of Windows programming that may be new to you.

Windows, Events, and Messages

Explaining the inner workings of Windows requires much more space than is available in this book. But you don't need an extensive knowledge of the Windows workings to create useful applications. The Windows operating system can be simplified to three basic concepts: windows, events, and messages.

A window is a rectangular region on the screen that has its own border. The AutoCAD drawing window, Notepad, a Word document, and the place where you compose an e-mail are all windows.

Windows can have hierarchy. A dialog form, which is a window within a distinct application, contains relevant ActiveX components and code. A drawing window is the parent of a dialog form window, and AutoCAD is the parent of the drawing windows within it. The operating system is the parent "window" of all applications running in it, including AutoCAD.

Each window recognizes and controls the programs that execute inside them or their subordinate windows. To manage windows, Windows assigns a unique ID known as a *handle*, or *hWnd* in programming jargon, to each window. Windows uses *events* to constantly monitor each window for signs of activity. Events change the application environment or *system state*. They occur when a user acts, such as by clicking the mouse or pressing a key; programmatically; or by another window through system processes.

Each time an event is triggered, Windows sends a message to the hosting application. Windows processes the message and broadcasts it to the windows. Then, based on its own instructions, each window can take appropriate action such as repainting itself when uncovered by another window. In the case of VBA in AutoCAD, the VBA work space intercepts event messages. VBA programs can then respond directly or pass the event up to AutoCAD or to Windows if necessary. VBA provides a controlling environment within AutoCAD in which to execute and respond to events, either directly or by allowing AutoCAD or Windows to respond.

Although this seems like a lot of work, VBA hides most of the low-level details from you and exposes *event procedures*, which are routines that execute when a particular event occurs, for your convenience. You can quickly create very powerful applications without being concerned with low-level details.

Event-Driven vs. Procedural Programming

When a traditional procedural application runs, it follows a predetermined path that controls the portions and sequence of code executed. It starts with the first line of code and progresses from the top down, calling each procedure when needed, until reaching the end of the code. This predetermined path is the major difference between procedural and event-driven applications.

Event-driven applications do not have a predetermined destiny. Different sections of code are executed based upon the events triggered in whatever order they occur. Depending on what events occur when the application runs, some sections of code may not get executed at all.

You can't predict the sequence of events, so you must make assumptions about the application's "state" at any moment. This seems like it would be difficult, but it's really not. Typically, you have a set of possibilities to work with, such as the `Click`, `DblClick`, `KeyPress`, and `LostFocus` events. For example, you might require the user to type a value in a `TextBox` before enabling a `CommandButton` that allows further processing. The `TextBox` control's `Change` event would contain code that enables the `CommandButton` control, as shown in this sample code:

```
Private Sub TextBox1_Change()
  If Len(TextBox1.Text) > 0 Then
    CommandButton1.Enabled = True

    Else
      CommandButton1.Enabled = False
  End If
End Sub
```

Each time you add or delete text from the `TextBox` control, the program executes this event procedure. The code checks the length of the text and if it is greater than zero, meaning there is something in the `TextBox`, it enables the `CommandButton`. Otherwise, it disables the `CommandButton`.

Programmatically changing the text in the `TextBox` triggers the `Change` event. If you allow for this occurrence, you might get unexpected results. Using events is very powerful, but you must know what each event might trigger elsewhere in your application.

Developing Your Applications Interactively

In more-traditional development environments, most developers follow a distinct three-step process: writing, compiling, and testing. However, VBA uses a more interactive approach to development that makes it easier for both beginning and experienced developers.

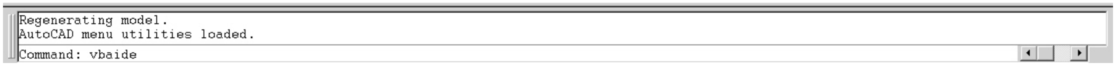
Languages such as C++ require you to write all the code then compile it. During the compile, you may uncover numerous errors, from simple typing errors to more-complex syntax errors. The Visual Basic programming environment, on the other hand, interprets your code every line of the way, alerting you to potential problems now instead of during a lengthy compile cycle.

Because Visual Basic is partially compiling your code as you type it, it takes very little time to finish compiling the code and execute your application. Unlike with other languages, you will find that you are constantly writing, executing, and refining your application. In addition, the Visual Basic environment employs a graphical environment. You most often work in the graphical interface first and on the code second. This lets you spend more time creating and less time compiling and recompiling.

Starting the Editor

One of the first questions that you will face is "How do I enter source code (structured commands) or develop a user interface (forms or dialog boxes)?" The answer is the IDE, a graphical user interface you use to develop applications. It is similar to development environments provided in other applications, such as Microsoft Access and Microsoft Excel.

To display the VBAIDE, choose Tools ➤ Macro ➤ Visual Basic Editor or press Alt+F11. Alternatively, you can start the editor by typing VBAIDE at the AutoCAD command prompt, as shown in Figure 1-1.



```
Regenerating model.
AutoCAD menu utilities loaded.
Command: vbaide
```

Figure 1-1. *The AutoCAD command prompt*

■ **Note** AutoCAD includes a Visual Basic menu and toolbar. You can load it by copying `ACAD.DVB` from \Sample\VBA\VBAIDEMenu (in the AutoCAD folder) to a directory in your support path. The toolbar will then autoload when the first VB command is launched. If you already have an `ACAD.DVB`, you can cut and paste from the provided `ACAD.DVB` to your own. This provides an AutoCAD toolbar for the following commands: VBAIDE, VBAPREF, VBAMAN, VBALOAD, and VBARUN. This also adds New, Open, and Close to the File pull-down on the VBAIDE toolbar.

Exploring the User Interface

The editor is composed of several different windows. The first time you open it, it looks like Figure 1-2. Use the View menu to control which windows are visible. To get context-sensitive help on any window, click in it and press F1.

The rest of this section discusses the most frequently used windows.

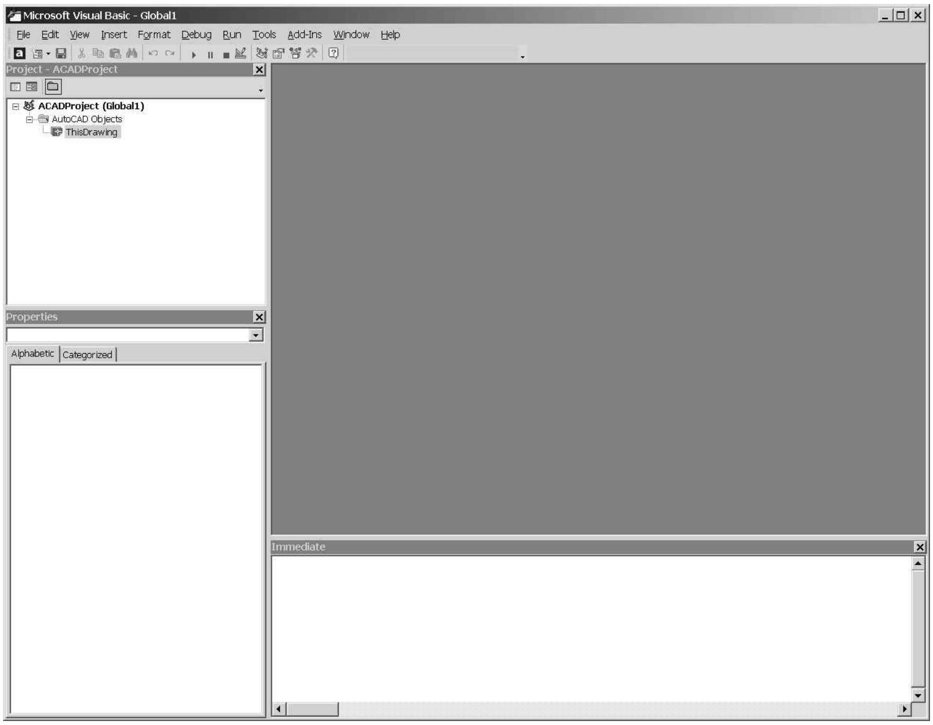


Figure 1-2. *The AutoCAD VBAIDE*

The Project Explorer

The Project Explorer window displays a hierarchical view called a *treeview* of your project's components that are loaded in the IDE. The view lists all of the `UserForms`, code, and class modules associated with each project. Use this window to keep track of and move components between projects. Figure 1-3 shows what a typical Project Explorer window looks like.

As you add, create, and remove files, the Project Explorer reflects those changes. You can save each project with a `.dvb` extension or you can embed it directly into a drawing file. You can't delete the required `ThisDrawing` object, which represents the active drawing, but nothing requires you to place code in this module.

You can close the Project Explorer window to gain more space in the IDE. To reopen it, choose View ➤ Project Explorer or press Ctrl+R.

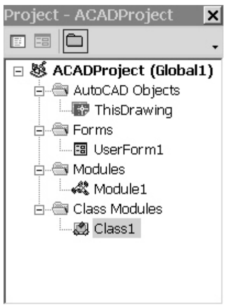
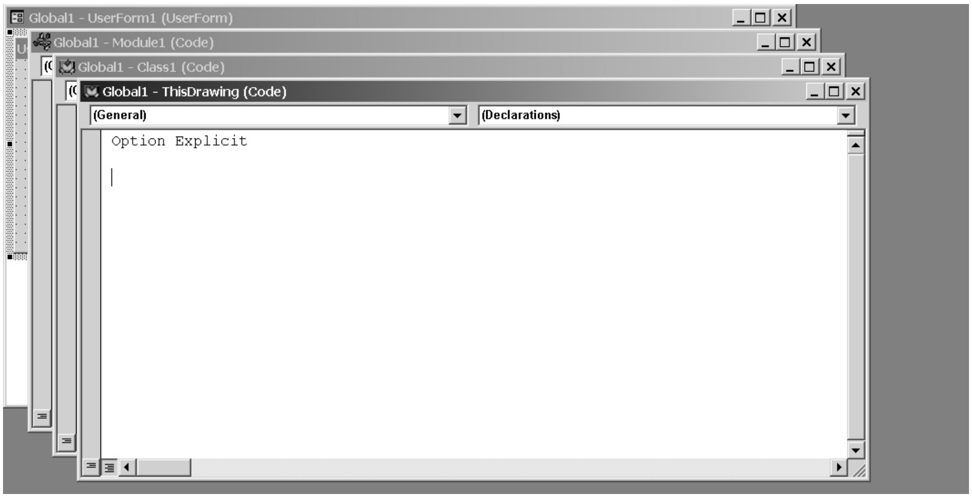


Figure 1-3. *Project properties*

The Code Window

The Code window is where you'll do most of your development work to manipulate AutoCAD. You can have a window open for each module in the project, as shown in Figure 1-4.



```
Option Explicit
```

Figure 1-4. *Cascaded VBAIDE windows*

Alternatively, you can see one window at a time by clicking the window's maximize button, shown in Figure 1-5, which is in its upper-right corner.

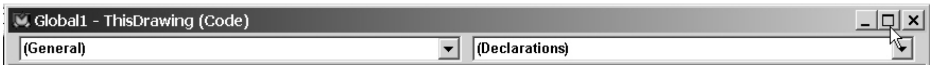


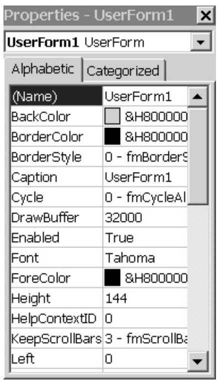
Figure 1-5. *Click the maximize button to see one window at a time.*

Later, this chapter discusses features such as Auto List Members and Auto Quick Info that make writing code easier and less prone to common typing errors.

The Properties Window

Properties control an object's behavior. You can query or control properties of AutoCAD entities, such as the color or linetype of a `Line` object, using the `Color` or `Linetype` properties.

The Properties window displays an object's design-time properties. You can change design-time properties while you develop your application. Changing properties during development has no effect on what you do in the IDE. However, any of these property changes could affect how well (and sometimes whether) your application will run. Figure 1-6 shows the properties of a typical `UserForm` object.



(Name)	UserForm1
BackColor	☐ &H800000
BorderColor	■ &H800000
BorderStyle	0 - fmBorderS
Caption	UserForm1
Cycle	0 - fmCycleAl
DrawBuffer	32000
Enabled	True
Font	Tahoma
ForeColor	■ &H800000
Height	144
HelpContextID	0
KeepScrollBars	3 - fmScrollBa
Left	0

Figure 1-6. *The Properties dialog box for a typical* `UserForm` *object*

■ **Note** Design time is when you develop your application in the IDE, writing code and adding forms and controls.

If you need to set a particular property once, set it at design time in the Properties window. But you will sometimes need to set a property's value at run time. Additionally, some properties are read-only and some are available only at run time. Each situation requires that you set the property value programmatically (using code).

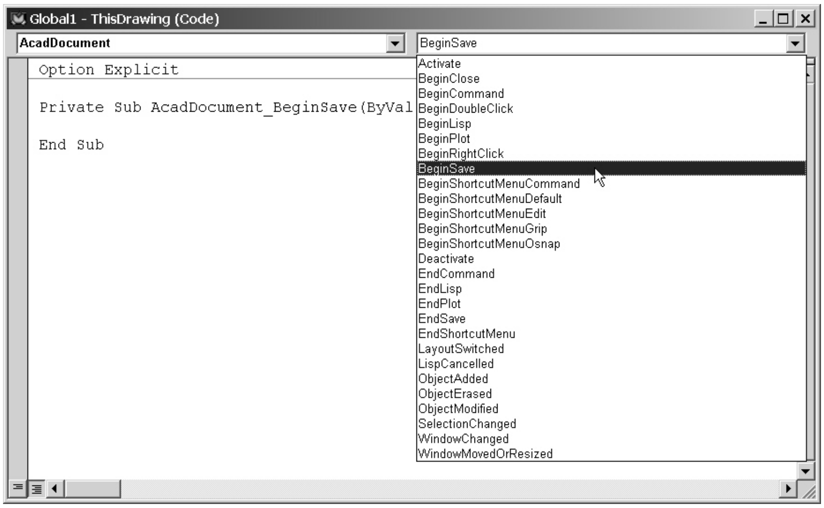
You can close the Properties window to gain more space in the IDE. To reopen the Properties window, choose View ➤ Properties Window or press F4.

The Object and Procedure Boxes

The Object box (the left half of Figure 1-7) lists all the objects and controls in your project. The Procedure box (the right half of Figure 1-7) lists all the procedures and events implemented in your application.

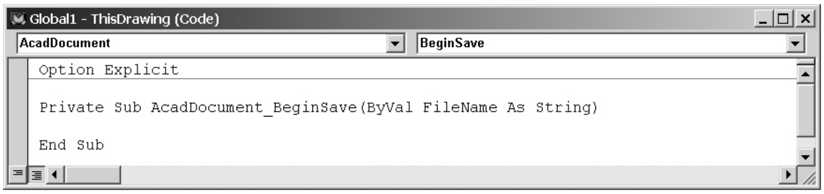
When you select an object from the Object box, the procedures associated with that object appear in the Procedure box. Figure 1-8 shows that after you choose a procedure, the IDE adds its basic framework to your code window, where you write your source code.

When a procedure or event does not appear in bold type in the Procedure box, it contains no code. Incidentally, when you choose an object, the IDE adds its default event procedure to the Code window. You may or may not choose to use the default procedure.



```
Option Explicit

Private Sub AcadDocument_BeginSave(ByVal

End Sub
```

Figure 1-7. *Object and Procedure boxes*

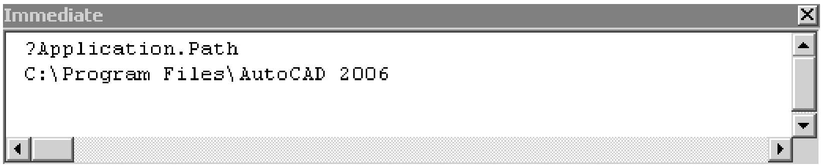


```
Option Explicit

Private Sub AcadDocument_BeginSave(ByVal FileName As String)

End Sub
```

Figure 1-8. *A VBAIDE procedure example*

The Immediate Window

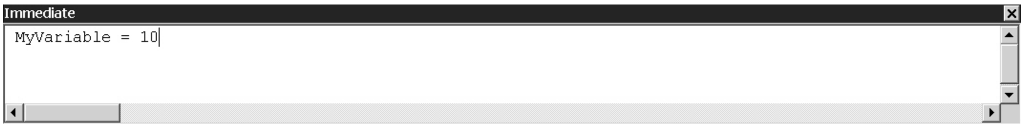
The Immediate window serves as a convenient place to send output from a running application during development. Alternatively, you can use it to set variables when you run an application, or to check a variable's value, such as the path to the AutoCAD executable file as shown in Figure 1-9.



```
?Application.Path
C:\Program Files\AutoCAD 2006
```

Figure 1-9. *Executing in the Immediate window*

In this example, use a question mark (?) to ask the VBAIDE to tell you a variable's value. To set a variable's value in the Immediate window, type the statement just as you would in the Code window. Figure 1-10 shows how it's done.



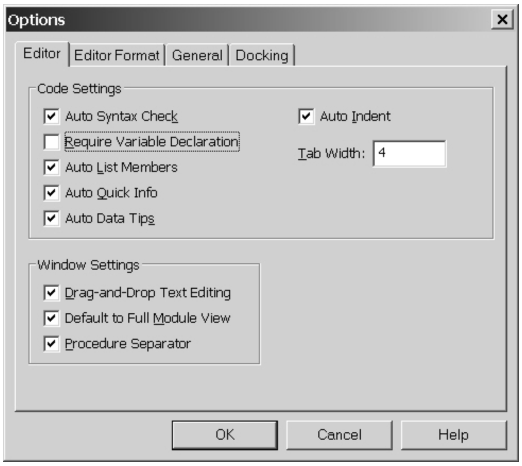
```
MyVariable = 10
```

Figure 1-10. *Setting a variable in Immediate window*

Note You can't declare variables in the Immediate window. Also, a variable's scope and valid VBA commands in the Immediate window are equally important. Chapter 2 covers variable scope.

The Options Dialog Box

The Options dialog box, shown in Figure 1-11, lets you customize the IDE's look and feel, including syntax color scheme, source code font, and tab spacing. To open the Options dialog box, choose Tools ➤ Options. This section covers the most commonly used options.



Code Settings
- [✓] Auto Syntax Check
- [ ] Require Variable Declaration
- [✓] Auto List Members
- [✓] Auto Quick Info
- [✓] Auto Data Tips
- [✓] Auto Indent
- Tab Width: 4

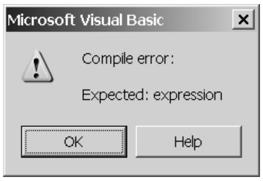
Window Settings
- [✓] Drag-and-Drop Text Editing
- [✓] Default to Full Module View
- [✓] Procedure Separator

Figure 1-11. *The Options dialog box*

For most developers, the default settings are fine. However, you should consider changing two settings on the Editor tab: Auto Syntax Check and Require Variable Declaration.

When the Auto Syntax Check option is checked, syntax errors in your code generate an error message similar to Figure 1-12.

As you begin to develop more complex applications and reuse lines of code from other places in your application, these error messages will become a nuisance. Any time you move the cursor off the offending line of code, you get one of these error messages. But if you uncheck Auto Syntax Check, the VBAIDE notifies you of errors by changing the color of the offending line of code to red.

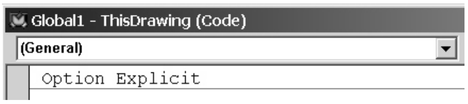


Compile error:

Expected: expression

Figure 1-12. *The error-message dialog box*

The Require Variable Declaration option is unchecked by default, meaning that the VBAIDE does not require that you properly declare your variables before you use them. This isn't much of a concern when you write a simple macro, but when you start developing larger and more complex applications, you'll find this option indispensable. Checking this option forces you to think about each variable and its data type. When you check this option, the VBAIDE adds a line of code to the start of each module, as shown in Figure 1-13.



```
Option Explicit
```

Figure 1-13. *An example of the checked Require Variable Declaration*

After you check this option, exit and restart AutoCAD to make it take effect.

By declaring variables to be a specific data type, you save memory resources. Undeclared variables are, by default, assigned the variant data type. This data type uses the most memory and could lead to memory resource problems when users run your application. As a rule of thumb, always declare each variable you use in your application, and choose the data type that uses the least possible memory. Chapter 2 discusses data types and memory in more detail.

Managing Projects

Managing your code components is critical to successfully developing applications. This section discusses adding components to your project, saving your project, and loading and executing an application.

Project Structure

A VBA project contains several different types of files, including the following:

- `UserForm` module
- `Standard` module
- `Class` module
- Reference `.dvb` file

UserForm Module

UserForm modules (files with a `.frm` extension) contain a text description of your form, controls placement, and property settings. They also include `UserForm`-level declarations of constants, variables, and procedures; and event procedures.

Standard Module

Standard modules (files with a `.bas` extension) contain module-level declarations of programmer-defined data types, constants, variables, and public procedures. A standard module typically contains routines that don't fit nicely into a class definition.

Class Module

Use class modules (files with a `.cls` extension) to create your own objects, including methods, properties, and events. `Class` modules are similar to `UserForm` modules except that they have a visible user interface. `Class` modules are very versatile and vital to VBA and AutoCAD. As you progress through this book, you'll see that classes and objects are everywhere.

Reference `.dvb` File

You can reference the code of another `.dvb` file in your current project. This feature lets you easily reuse code among several projects. You can't create a circular reference, which is a reference to one project and a reference in that project to the current project. If you accidentally create a circular reference, AutoCAD tells you of the error. You have to undo the reference before you can continue.

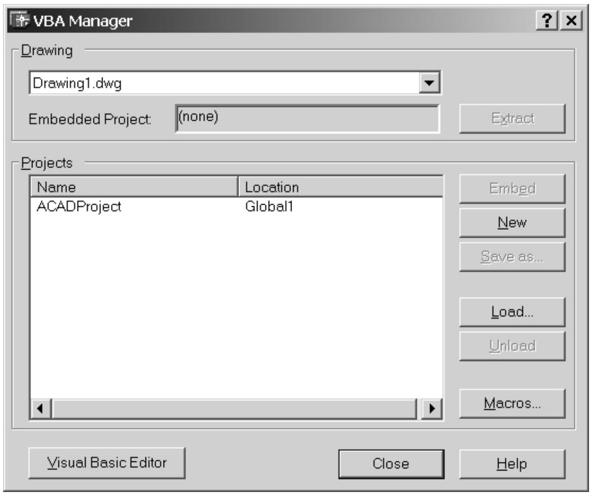
Creating, Opening, and Saving Projects

To extract, embed, create, save, load, and unload VBA projects, open the VBA Manager dialog box, shown in Figure 1-14. To open it, either type VBAMAN at the AutoCAD command prompt or choose Tools ➤ Macros ➤ VBA Manager.

You must explicitly load all `.dvb` projects. AutoCAD loads embedded projects automatically when the drawing containing them is opened, depending upon how you configure AutoCAD's security options. Clicking New creates a new project in the VBAIDE that you can access by clicking the Visual Basic Editor button. To load an existing project, click the Load button. The Open VBA Project dialog box in Figure 1-15 appears, letting you choose the project to load.

■ **Tip** Embedding VBA macros within drawings is fine for drawings that remain within your organization. Avoid embedding macros when you'll deliver the drawings to outside users or customers as it imposes a security risk on their part to trust your macros in their environment.

This dialog box is similar to the standard File Open dialog box in Windows. Similar to creating a new project, when you choose the `.dvb` project you want, click the Visual Basic Editor button to start working on your project.



Drawing

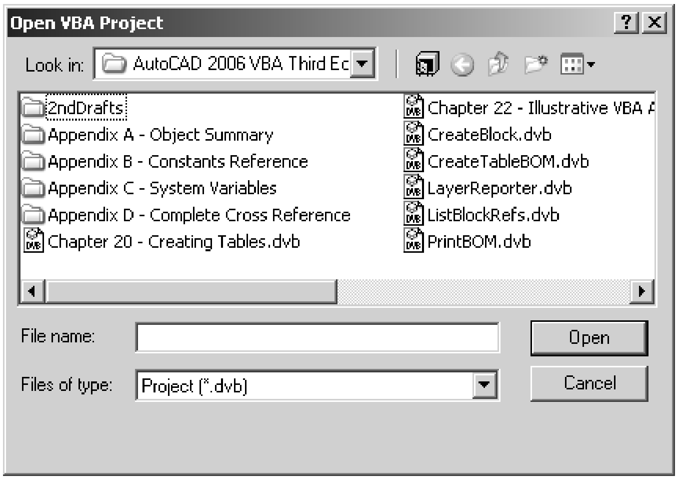
Drawing1.dwg

Embedded Project: (none)

Projects

Name	Location
ACADProject	Global1

Figure 1-14. *The VBA Manager dialog box*



Look in: AutoCAD 2006 VBA Third Ec

2ndDrafts
Appendix A - Object Summary
Appendix B - Constants Reference
Appendix C - System Variables
Appendix D - Complete Cross Reference
Chapter 20 - Creating Tables.dvb
Chapter 22 - Illustrative VBA A
CreateBlock.dvb
CreateTableBOM.dvb
LayerReporter.dvb
ListBlockRefs.dvb
PrintBOM.dvb

File name:

Files of type: Project (*.dvb)

Figure 1-15. *The Open VBA Project dialog box*

There are two other ways to create and load `DVB` project files:

- Type **VBAIDE** at the AutoCAD command prompt or press Alt+F11 to open or create a DVB project file.
- Type **VBALOAD** at the AutoCAD command prompt to open the Open VBA Project dialog box so you can choose a project to load.

To save your project, choose File ➤ Save or press Ctrl+S. If you have not previously saved your project, the standard Save As dialog box appears, as shown in Figure 1-16.

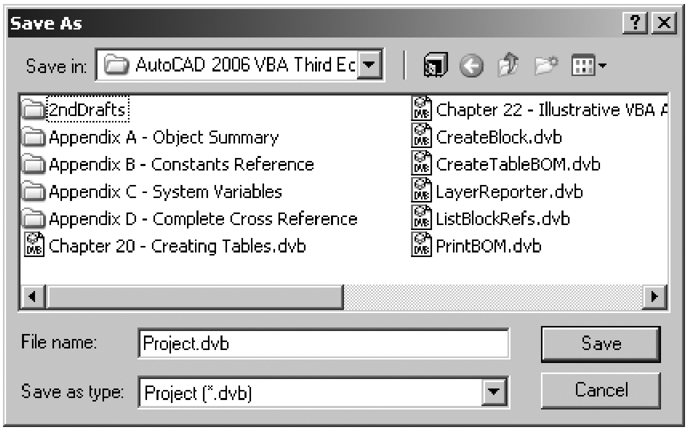

Save As
Save in: AutoCAD 2006 VBA Third Ec
2ndDrafts
Appendix A - Object Summary
Appendix B - Constants Reference
Appendix C - System Variables
Appendix D - Complete Cross Reference
Chapter 20 - Creating Tables.dvb
Chapter 22 - Illustrative VBA A
CreateBlock.dvb
CreateTableBOM.dvb
LayerReporter.dvb
ListBlockRefs.dvb
PrintBOM.dvb
File name: Project.dvb
Save as type: Project (*.dvb)
Save
Cancel

Figure 1-16. *Save As dialog box*

Unlike in Visual Basic, you don't need to save each project module separately. AutoCAD saves them all in a `.dvb` file. However, as the next section illustrates, you can export each module to a separate file.

■ **Tip** Avoid saving your custom program files under the AutoCAD installation folder tree. Instead, create a separate folder tree for them. This prevents AutoCAD installations and updates from affecting your program files.

Adding, Saving, and Removing Files

You will sometimes want to add to your VBA project a file such as a common `UserForm` module or a collection of routines in a standard module. To do this, choose File ➤ Insert File or press Ctrl+M. The Import File dialog box appears, as shown in Figure 1-17.

To export a module to a separate file, highlight the module name in the Project Explorer window, and then either choose File ➤ Export File or press Ctrl+E. A Save As dialog box appears for the type of file to export. Alternatively, you can highlight the module name in the Project Explorer window, right-click to invoke the pop-up menu, and choose Export File.

To remove a file from your project, highlight the module name in the Project Explorer dialog. Then choose File ➤ Remove (you will be offered the option of exporting the file prior to removal). You can also highlight the module name, right-click to invoke the pop-up menu in Figure 1-18, and then choose Remove.

Notice that this menu also includes the Export File option.

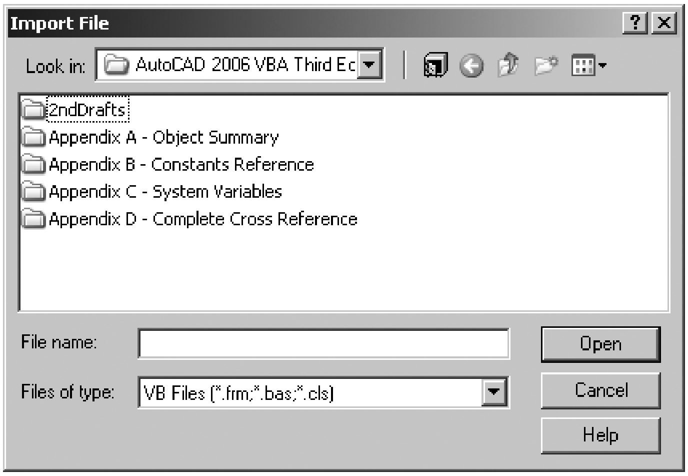

Import File
Look in: AutoCAD 2006 VBA Third Ec
2ndDrafts
Appendix A - Object Summary
Appendix B - Constants Reference
Appendix C - System Variables
Appendix D - Complete Cross Reference
File name:
Files of type: VB Files (*.frm;*.bas;*.cls)
Open
Cancel
Help

Figure 1-17. *The Import File dialog box*

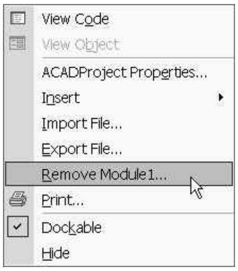


Figure 1-18. *The Remove Module pop-up menu*

Adding ActiveX Controls and Code Components

When you start a project and add a `UserForm` module, a common Toolbox appears. It contains a standard collection of ActiveX controls called *intrinsic controls*. Chapter 3 covers intrinsic controls in more detail. If you want to insert an ActiveX control that is not in the Toolbox, choose Tools ➤ Additional Controls or right-click in the Toolbox window and choose Additional Controls. The Additional Controls dialog box shown in Figure 1-19 appears.

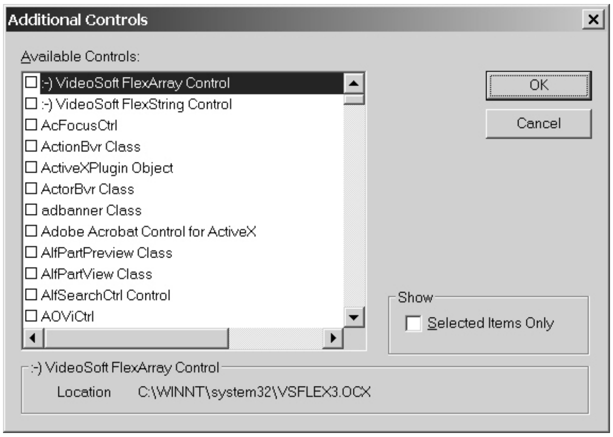


Figure 1-19. *The Additional Controls dialog box*

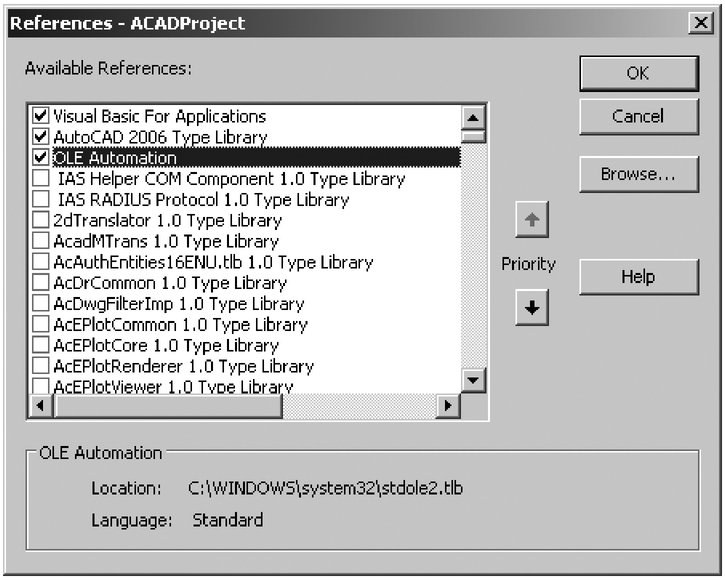
This dialog box lists all the ActiveX controls that are properly registered on your machine. However, if you want to use a particular ActiveX control, check its End User License Agreement (EULA) to determine whether you have a license to use it in a VBA host application.

Many ActiveX controls installed with Microsoft Visual Basic, including `TreeView`, `ListView`, `File`, `Directory`, and `Drive`, are not licensed for use in a VBA host application. However, Chapter 19 explains some easy ways you can use the Windows application programming interface (API) to get around this dilemma.

You can add more than ActiveX controls—code components are perhaps even more common. This is the means by which your application can gain access to other ActiveX automation applications such as Microsoft Excel, Access, and Word. ActiveX automation is your key to building powerful applications that take advantage of objects exposed by other applications.

To use the objects, methods, properties, and events that other applications expose, first add a reference to that application. Choose Tools ➤ References to open the References dialog box, shown in Figure 1-20.

Check the reference you want to add. The dialog box contains an alphabetical list of the references your application doesn't use. Later, when you write your own objects using `Class` modules, this is where you'll add them to your project.



- [✓] Visual Basic For Applications
- [✓] AutoCAD 2006 Type Library
- [✓] OLE Automation
- [ ] IAS Helper COM Component 1.0 Type Library
- [ ] IAS RADIUS Protocol 1.0 Type Library
- [ ] 2dTranslator 1.0 Type Library
- [ ] AcadMTrans 1.0 Type Library
- [ ] AcAuthEntities16ENU.tlb 1.0 Type Library
- [ ] AcDrCommon 1.0 Type Library
- [ ] AcDwgFilterImp 1.0 Type Library
- [ ] AcEPlotCommon 1.0 Type Library
- [ ] AcEPlotCore 1.0 Type Library
- [ ] AcEPlotRenderer 1.0 Type Library
- [ ] AcEPlotViewer 1.0 Type Library

OLE Automation
Location: C:\WINDOWS\system32\stdole2.tlb
Language: Standard

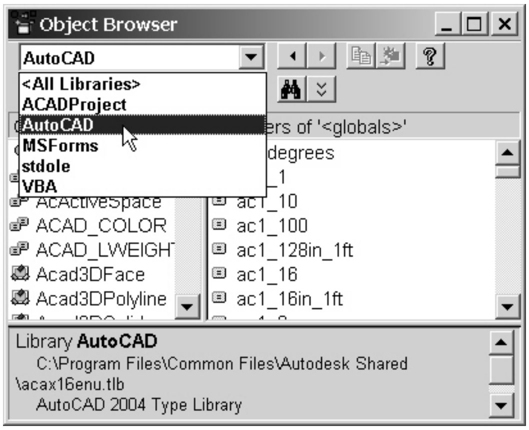
Figure 1-20. *The References dialog box*

To improve performance, deselect any references your application doesn't use. Each reference must be resolved before your project loads. Depending on your project's size, this could greatly decrease how long your user waits for the application to load and run.

■ **Note** If your application uses an object of another application, you can't remove the reference to it without first removing the object.

The Object Browser

When you set a reference to an application's object library, the Object Browser lists all the objects, methods, properties, constants, and events that application exposes. To open the Object Browser, choose View ➤ Object Browser or press F2. By default, the Object Browser lists all libraries your project currently references. To view just the AutoCAD library, for example, click AutoCAD in the drop-down list in the Object Browser, as shown in Figure 1-21. The Object Browser gives you a perspective of the objects an application exposes and the methods, properties, events, and constants those objects expose for your application to manipulate.



Library **AutoCAD**
C:\Program Files\Common Files\Autodesk Shared \acax16enu.tlb
AutoCAD 2004 Type Library

Figure 1-21. *The Object Browser dialog box*

Loading and Running Applications

You can load and subsequently execute your VBA applications in many different ways. This section explains the most common ways.

Acad.dvb

AutoCAD searches the support file search path for the file `acad.dvb`. If AutoCAD finds this file, it loads it into the current session. The following example illustrates how to implement this feature.

```
Public Sub Start()
  Application.ActiveDocument.SetVariable "OSMODE", 35
End Sub
```

Place this code in the `ThisDrawing` module and save the file as `acad.dvb`. Save this file in any subdirectory specified in the support file search path. Now each time you start an AutoCAD session, AutoCAD loads this file. Also, if you include a routine called `AcadStartup`, AutoCAD executes it when it loads `acad.dvb`.

The example in Figure 1-22 shows how to use the macros that you have stored in the `acad.dvb` file.

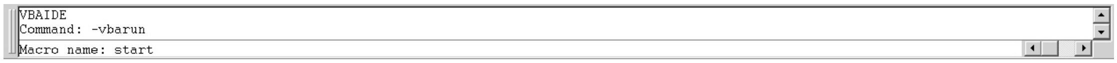

```
VBAIDE
Command: -vbarun
Macro name: start
```

Figure 1-22. *The AutoCAD command prompt*

Type **–vbarun** then the name of the macro to run (`Start` in this instance). In this example, the value of the system variable `OSMODE` is set to 35. This is a convenient way to store macros that will invoke AutoCAD with a particular setup.

Acaddoc.lsp

AutoCAD automatically searches the default search path (a combination of the support/file path list, the current working folder, and the shortcut's startup folder) for a file named `acaddoc.lsp`. If AutoCAD finds this file, it loads it into the current drawing. Unlike with `acad.dvb`, which loads only when you start a new AutoCAD session, AutoCAD loads `acaddoc.lsp` each time you open or create a drawing.

■ **Tip** To verify the actual search-path list, go to the command prompt, type *-insert*, enter a meaningless string of characters such as *sdfsdfsdf* (keep it less than 31 characters, though), and press Enter. AutoCAD displays a list of folders in which it tried to find your file when it failed.

AutoCAD provides a special programmer-defined function called `S::STARTUP` that, if included in `acaddoc.lsp` or any default startup LISP file, is automatically executed when you open or create a drawing. You can define `S::STARTUP` in `acaddoc.lsp` to perform setup operations for each drawing.

You can define an `S::STARTUP` function in several different places, including `acad.lsp`, `acaddoc.lsp`, a `.mnl` file, or any AutoLISP file loaded from any of these files. You can overwrite a previously defined `S::STARTUP` function—which means that another definition can overwrite your `S::STARTUP` routine.

■ **Note** Never modify or replace the files `acad2006.lsp` or `acad2006doc.lsp`, as they may be overwritten by a service pack installation without warning. It is also a good idea to append the `S::STARTUP` function instead of defining it, as many third-party products rely upon `S::STARTUP` to initialize their environments. If you define a new `S::STARTUP`, you could disable some or all of another loaded product in the process.

The following example shows how to ensure that your startup function works with other functions.

```
 (defun-q Startup ()
  (command "-vbarun" "Start")
)
(setq S::STARTUP (append S::STARTUP Startup))
```

This code appends your startup function to any existing `S::STARTUP` function, and then redefines the `S::STARTUP` function to include your startup code. This works regardless of any other existence of an `S::STARTUP` function.

■**Note** In AutoLISP, you must use `defun-q` as opposed to `defun` for this example to work properly. Visual LISP constructs functions differently from AutoLISP between `defun` and `defun-q`.

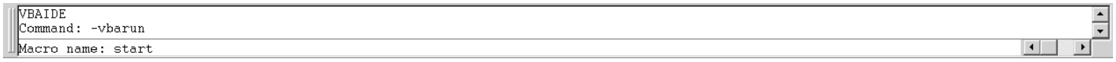
Embedded Projects

AutoCAD lets you embed a VBA project into a drawing. Each time the drawing is loaded, AutoCAD also loads the VBA project embedded in the drawing. Of all the options to load VBA projects automatically, this is the worst one. It stores the VBA project with the drawing, making your drawing file that much bigger. If you copy the drawing file to create a new drawing, you also copy its VBA project. If you want to change the VBA project, you need to change each VBA project in every drawing file that you created from the original. In addition, delivering drawings with embedded macros to customers imposes a serious potential security risk on their part. Well, you get the picture of why this is the worst option to choose!

`VBARUN` and the Macros Dialog Box

The dialog-box version of the `VBARUN` command features several options that extend how you create and execute a VBA macro.

You can execute a VBA macro at the AutoCAD command prompt. To execute a macro from the AutoCAD command prompt, type **–VBARUN**, press Enter, and type the macro name, similar to the example in Figure 1-23.



```
VBAIDE
Command: -vbarun
Macro name: start
```

Figure 1-23. *The AutoCAD command prompt*

In this example, AutoCAD would execute the Start macro.

If the macro you wish to execute is unique among all the VBA projects loaded, then just specify the macro name after the `-VBARUN` command. However, if you have multiple macros loaded with the same name, specify the macro to execute using either of these syntaxes:

```
ProjectName.ModuleName.MacroName
```

or

```
ModuleName.MacroName
```

How far down inside your project and modules your macro is placed, and whether you have multiple projects loaded determine which syntax is appropriate.

You can also execute a macro through the Macros dialog box, shown in Figure 1-24. To open this dialog box, type **VBARUN** at the AutoCAD command prompt. You can also choose Tools ➤ Macro ➤ Macros or press Alt+F8. For the beginner or end user, this is an excellent way to execute macros.

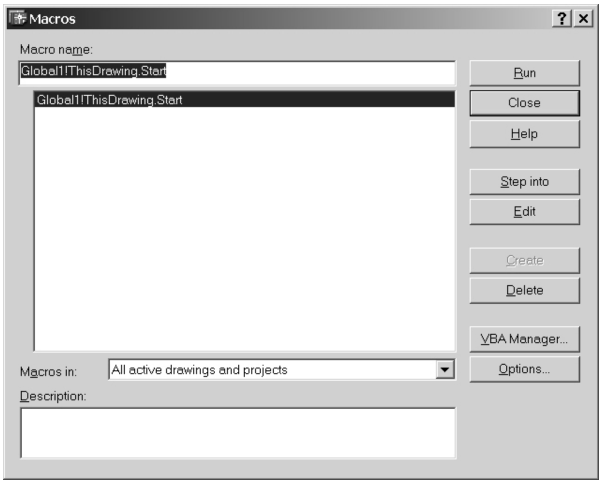


Figure 1-24. *The Macros dialog box*

Macros

Tasks that occur frequently or are of a complex nature are ideal candidates for a macro. Using a macro gives you a more consistent and convenient way to accomplish these tasks. Macros let you create this process once and then reuse it multiple times.

You can create macros in one of two places, in either the `ThisDrawing` module or in a standard module. In either case, always declare them as `Public` subprocedures and not functions that return a value and can't receive arguments. Despite these limitations, macros are a powerful tool for increasing AutoCAD end-user productivity.

An example of where you might want to use a macro is in initializing user preferences whenever a drawing is created or opened, as in the following code:

```
Public Sub Setup()
  Application.Preferences.OpenSave.AutoSaveInterval = 15
End Sub
```

In this example, AutoCAD will save the user's work every 15 minutes. You might consider using a macro similar to this to set various system variables for each drawing that you work on or perhaps for when each AutoCAD session begins. The previous section explained how to make VBA routines execute automatically for the start of either an AutoCAD session or a drawing.

The Macros Dialog Box

Use the Macros dialog box to create, run, debug, edit, and delete macros. You can also control how a macro runs using several options. This section explains each button in this dialog box.

Run

This button executes your macro. You can also execute your macro from within the VBAIDE by choosing Run ➤ Run or by pressing F5.

Step Into

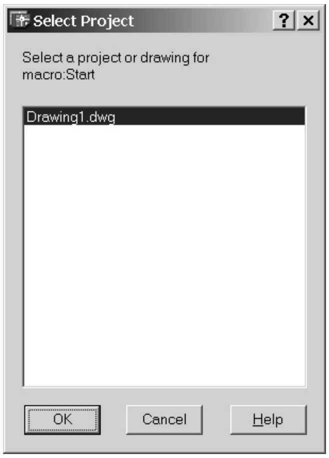
This button lets you step line by line through the code of the macro you select. It enters the VBAIDE in step (or debug) mode at the beginning of the selected macro. You can also start Step Into from within the VBAIDE by choosing Debug ➤ Step Into or by pressing F8.

Edit

This button lets you edit your macro's code. It opens the VBAIDE in edit mode at the beginning of the selected macro. This is identical to typing VBAIDE at the AutoCAD command prompt and navigating to the appropriate project, module, and macro.

Create

To create a macro, enter a name in the Macros dialog box and press the Create button. This opens a dialog box that lets you choose where you want to create your macro. You can also specify the project in which to create the macro. If you do not specify a project name, the Select Project dialog box, shown in Figure 1-25, appears. Choose from the list of loaded projects.



Select a project or drawing for macro:Start

Drawing1.dwg

Figure 1-25. *The Select Project dialog box*

Delete

To delete a macro from the project file, select the Delete button. If you accidentally delete a macro, retrieve it by *immediately* switching to the VBA editor, bring the module that contained your source code into focus, and either choose Edit ➤ Undo or press Ctrl+Z.

Caution This command deletes the source code associated with the macro. Be careful!

VBA Manager

This button opens the VBA Manager dialog box, which lets you create, edit, load, unload, close, embed, and extract your VBA projects.

■ **Note** For more about this dialog box, see "Creating, Opening, and Saving Projects," earlier in this chapter.

Options

AutoCAD VBA provides three options for controlling how AutoCAD handles VBA projects by default. Table 1-1 explains these options.

Table 1-1. *VBA Options in AutoCAD*

Option	Meaning	Default Value
Enable Auto Embedding	Determines whether a VBA project will be embedded in the current drawing	`False` or unchecked
Allow Break on Errors	Determines whether VBA is instructed to enter Break mode when an error occurs in your application	`True` or checked
Enable Macro Virus Protection	Determines whether macro virus protection is enabled	`True` or checked

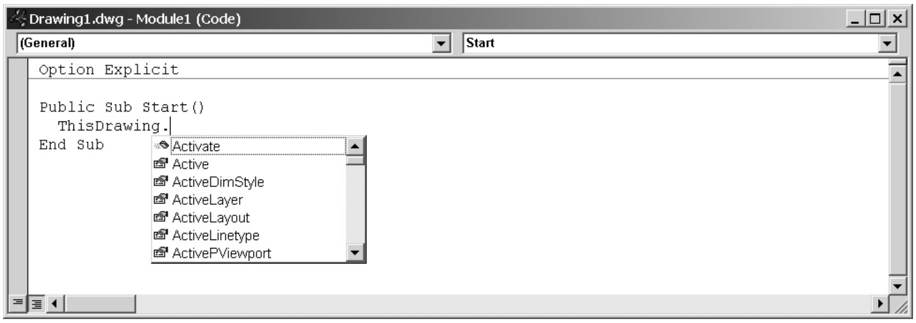
Using the Text Editor

Fortunately, the VBA text editor is not just a plain old text editor. It has some pretty nice features that make your application-development efforts more efficient. This section tells you how to take advantage of their usefulness.

Auto List Members

When you type a valid object name in your code and then type a period (**.**), VBA drops down a list of properties and methods available for that object, as shown in Figure 1-26. Typing the first few letters of the property or method name moves you to that selection in the list. Pressing the Tab key completes the typing for you. You can also press the Complete Word button on the Edit toolbar. This option is also helpful when you aren't sure which properties are available for an object.

Visual Basic 6 and VB.NET programmers know this feature by its other name: IntelliSense.



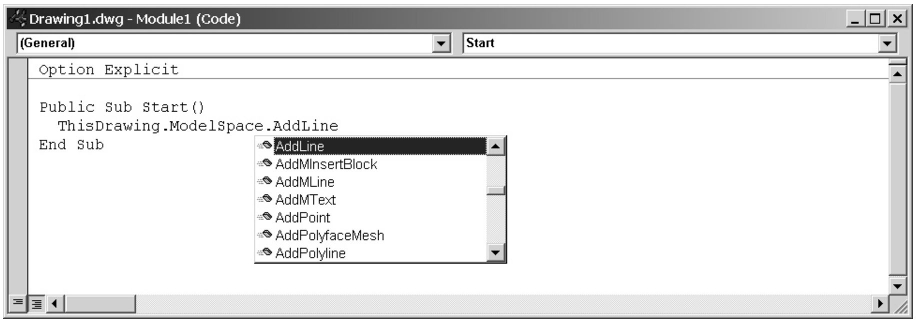
```
Option Explicit

Public Sub Start()
  ThisDrawing.
End Sub
```

Figure 1-26. *An Auto List Members example*

Tip If you disable the Auto List feature, you can still access it by pressing Ctrl+J.

Auto Quick Info

Auto Quick Info displays the syntax of statements, procedures, and functions that are *early-bound* (there is a reference set to the object library you're using). This feature is not available for *late-bound* objects (no reference set to the object library). When you type the name of a valid Visual Basic statement, procedure, or function, the IDE shows the syntax immediately below the current line, with the first argument in boldface, as shown in Figure 1-27. After entering the first argument value, the second argument appears in boldface, and so on.



```
Option Explicit

Public Sub Start()
  ThisDrawing.ModelSpace.AddLine
End Sub
```

Figure 1-27. *An Auto Quick Info example*

Tip If you disable the Auto Quick Info feature, you can still access it by pressing Ctrl+I.

You may have noticed that this feature also provides you with a function's return value, which can be very helpful when you use an unfamiliar function for the first time.

Overview of AutoCAD VBA Commands

AutoCAD provides a number of commands to access the VBA editor and execute VBA code. Table 1-2 summarizes the commands that you can enter at the AutoCAD command prompt.

Table 1-2. *VBA Editor Commands*

Command	Description
`VBALOAD`	Loads a VBA (`.dvb`) project file into AutoCAD memory. If you are going to edit your project, you may want to click the Visual Basic Editor option. This puts you inside the VBA editor for immediate editing.
`VBAIDE`	Opens the VBAIDE. If you have loaded a project, this opens the VBA editor so you can begin editing.
`VBAUNLOAD`	Unloads the current project. If you made changes without saving them, AutoCAD asks you to save your VBA project first.
`VBARUN`	Opens a dialog box from which you can choose a VBA macro to run. This option has a number of features you may choose.
`-VBARUN`	Runs a VBA macro from the AutoCAD command prompt. If you have macros with the same name in more than one module file, then you must use the <*modulename.macroname*> syntax.
`VBAMAN`	Invokes the VBA Manager dialog box, which lets you view, create, load, close, embed, and extract projects.
`VBASTMT`	Executes a VBA expression from the AutoCAD command prompt.

Summary

This chapter covered a tremendous amount of information about the Visual Basic for Applications Integrated Development Environment, or VBAIDE. While this is sufficient for most of your AutoCAD VBA development needs, you can get more help from the AutoCAD help files. The next chapter introduces you to the Visual Basic programming language and its variables, constants, control structures, and more. It also introduces you to the basics of object-oriented programming.

CHAPTER 2

Introduction to Visual Basic Programming

A Visual Basic application contains several components:

- `UserForm` modules, which contain an application's visual interface, including ActiveX controls and the form's Visual Basic code
- `Standard` (`.bas`) and `Class` (`.cls`) modules, which contain customized routines and classes
- Intrinsic and custom ActiveX controls (`.ocx`) and code components in ActiveX DLLs (`.dll`)

By default, an AutoCAD VBA project contains a single module called `ThisDrawing`. You can then add `UserForm`, `Standard` and `Class` modules, and ActiveX controls as needed. Chapter 3 describes how to add these elements to an application. If you are an experienced VBA developer in AutoCAD 2000 or 2002, read the documentation in AutoCAD 2006 for new, changed, and removed features for ActiveX, VBA, system variables, and drawing data classes. Each new release brings considerable changes that often affect program development in good and bad ways.

Variables

Variables let you retain values for use locally in a procedure or globally in your entire application. For example, you might retrieve user input for drawing a `Line` entity, saving the coordinates the user selected. You might also ask the user to select a color for an entity using a drop-down combo box or a list box.

Visual Basic refers to variables by name, such as `InsertionPoint` or `LayerName`. It's best to avoid using variable names that coincide with predefined property or event names. Variables are known by their data type, that is, the kind of data that they can store. The data type determines the amount of memory required to store the variable's value.

Declaring Variables

Use the `Dim` statement in a procedure to declare a variable. Here's the syntax:

```
Dim <VariableName> [As DataType]
```

Declare the variable's data type in the optional `As DataType` clause in the `Dim` statement. A variable's data type determines the kind of information a variable holds, such as *String, Integer*, or *Object*. See the "Data Types" section later in this chapter for more information.

Implicit Declaration

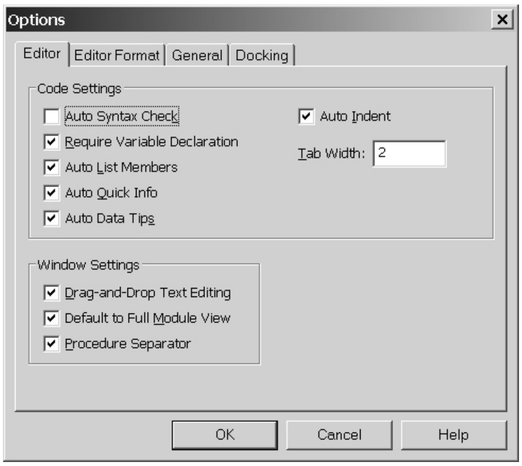
You don't have to declare variables as a specific data type before you use them. If you use a variable without explicitly declaring it, Visual Basic creates a variable with that name and assigns it the Variant data type. This is the largest data type available in terms of memory usage and could lead to memory resource problems. When you use an undeclared variable, accidentally misspelling the variable name creates another undeclared variable. These kinds of errors can lead to poor performance and a lot of wasted time debugging your code.

Explicit Declaration

To avoid the hazards mentioned, force Visual Basic to require that you declare all variables before you use them. To do so, place the following line at the top of your code module:

```
Option Explicit
```

Alternatively, you can turn on explicit data type checking by choosing Tools ➤ Options and checking the Require Variable Declaration box, as shown in Figure 2-1.


Options

Editor | Editor Format | General | Docking

Code Settings
- [ ] Auto Syntax Check
- [✓] Require Variable Declaration
- [✓] Auto List Members
- [✓] Auto Quick Info
- [✓] Auto Data Tips
- [✓] Auto Indent

Tab Width: 2

Window Settings
- [✓] Drag-and-Drop Text Editing
- [✓] Default to Full Module View
- [✓] Procedure Separator

Figure 2-1. *Setting explicit data type checking in the Options dialog box*

Variable Names

You can name variables almost any way you want, but some restrictions exist. Follow these rules in variable names:

- You must use a letter as the first character.
- You can't use a space; a period (`.`); an exclamation mark (`!`); or the characters `@`, `&`, `$`, or `#`

- The name can't be longer than 255 characters.
- Generally, don't use names that are the same as Visual Basic's functions, statements, and methods. To use an intrinsic language function, statement, or method that conflicts with an assigned name, explicitly identify it. Precede the intrinsic function, statement, or method name with the name of the associated type library. For example, if you have a variable called `Left`, you can invoke the `Left` function only by using `VBA.Left`.
- You can't repeat names in the same scope. For example, you can't declare two variables named `age` in the same procedure. However, you can declare a private variable named `age` and a procedure-level variable named `age` in the same module.

■**Note** Visual Basic isn't case-sensitive, but it preserves a variable's capitalization in the statement where you declare it.

In addition to restrictions and requirements, there are widely accepted best practices for naming variables to ensure consistency and ease of reading. Begin each variable's name with a prefix that indicates its data type. For example, a String variable named `FirstName` should be instead named `strFirstName`. Table 2-1 lists some commonly used prefixes. Not only does this book use this convention but so does most sample code elsewhere.

Table 2-1. *Commonly Used Variable Prefixes*

Data Type	Prefix	Example
String	str	`strFirstName`
Label	lbl	`lblProperties`
Double	dbl	`dblXcoordinate`
Integer	int	`intFloorLevel`
ListBox	lst	`lstSystemNames`
ComboBox	cbo	`cboElecSystems`
Button	btn	`btnChoice`
CheckBox	chk	`chkSaveBackup`
OptionButton	opt	`optSaveFormat`
Frame	frm	`frmSaveOptions`

Variable Scope and Lifetime

If you declare a variable in a procedure, only code in that procedure can access it. This is known as *local scope*. But sometimes you'll want procedures throughout a module or throughout the application to access one of your variables. Visual Basic lets you specify this broader scope when you declare your variables.

All procedures, variables, constants, and so on, have a scope, also called a *lifetime*. Where you declare them determines when and from where you can access them. You can declare variables as procedure (local) or module variables, as Table 2-2 explains.

Table 2-2. *Variable Scope*

Scope	Private	Public
Procedure	The variable is accessible only in the procedure where you declared it.	You cannot declare variables as Public in a procedure.
Module	The variable is accessible only in the module where you declared it.	The variable is accessible by all procedures in the module.

Procedure Variables

Procedure variables, also called *local variables*, are available only in the procedure in which you declare them. Declare them using either the `Dim` keyword, as in this example:

```
Dim Line As AcadLine
```

or the `Static` keyword, as in this example:

```
Static Count As Integer
```

Variables that you declare in a procedure using the `Dim` keyword have scope only while you're inside that procedure. When you leave the procedure, the variable's value is gone. On the other hand, a variable declared in a procedure using the `Static` keyword lives as long as your application runs, much like a global variable but with local scope (that is, you can access it only when you are in that procedure).

Local variables are used for calculations pertinent to the procedure. Declare a local variable as `Static` only when you need to keep a running count, such as in a procedure, and don't want other procedures in the application to have access to the variable.

Module Variables

By default, variables that you create in a `UserForm`, `Standard`, or `Class` module are available only to procedures in that module. At the module level, the `Private` and `Dim` keywords are equal. However, it's better to use the `Private` keyword to make your code more understandable and easily distinguishable from variables declared using the `Public` keyword.

In the declarations section at the top of your module, declare `Private` module variables like this:

```
Private LayerOn As Boolean
```

Global Variables

If you want to access module variables anywhere in your application, use the `Public` keyword as follows:

```
Public LayerOn As Boolean
```

When you do this, the variable is available to every module and procedure in your application. You can't declare `Public` variables in a procedure.

■ **Tip** When you declare a variable, declare it as locally as possible. Then as you find that particular variable needs to be accessible more globally, move its declaration out one level at a time.

Constants

Constants are values that never change. If you will use a value frequently in your application and its value will never change, declare it as a constant. For example:

```
'declare any constants used throughout applications
Public Const SAVETIME = 15      'public constant for default automatic save time
```

The following scope rules apply to constants:

- If you want a constant to be accessible in a procedure, you must declare it in that procedure.
- To make a constant accessible only in a module in the module's declaration section, be sure to declare the constant as `Private`.
- You can declare constants as `Private` or `Public`, but they are most often `Public`.
- To make constants accessible to any module or procedure in your application, declare them in the module's declaration section using the `Public` keyword.

■ **Tip** Typically, constant names are all capitals to distinguish them from variables.

Data Types

If you don't specify a variable's data type using the `As` keyword, as the following example shows, Visual Basic makes it a Variant data type:

```
Dim Variable
```

Although you can create variables this way, you should instead assign a data type to every variable you create. This makes your code more readable and removes ambiguity about what your intentions are in your application.

```
Dim InsertionPoint As Variant
Dim Counter As Integer
```

VBA provides a number of data types. Each data type has a specific purpose that you should consider when you choose which one to use. Table 2-3 explains VBA's data types.

■**Note** Data type properties have been modified in Visual Basic .NET and Visual Studio for Applications. If you plan to work in a mixed environment where your VBA programs may interface with .NET programs, be sure to research the differences carefully!

Table 2-3. *Visual Basic Data Types*

Data Type	Range of Values	Purpose
Integer	–32768 to +32767	Relatively small positive and negative whole numbers
Long	–2,147,483,648 to +2,147,483,647	Relatively large positive and negative whole numbers
Single	Approximately 1.4E-45 to 3.4E+48	Single-precision floating-point numbers that use 4 bytes of data
Double	Approximately 4.94E-324 to 1.8E+308	Double-precision floating-point numbers that use 8 bytes of data
Currency	–922,337,203,685,477.5808 to +922,337,203,685,477.5807	Money calculations where accuracy is important, giving 15 digits to the left of the decimal and 4 digits to the right of the decimal
String	Approximately 0 to 2 billion characters	Text characters
Boolean	1 or 0	`True`/`False`, `Yes`/`No`, `On`/`Off` values
Byte	0 to 255	A finite group of positive integers, usually used for character or color code values
Date	1/1/100 to 12/31/9999 and 0:00:00 to 23:59:59	General date and time
Variant	Anything	The default data type if you don't specify one

Exchanging Numbers and Strings

You can assign string values that represent a numeric value to a numeric variable. You can also assign a numeric value to a String variable. The following code illustrates this:

```
Public Sub MyMacro()
Dim Count As Integer
Dim NumericString As String

  Count = 100
  NumericString = "555"

  MsgBox "Integer: " & Count & vbCrLf & _
         "String: " & NumericString
  Count = NumericString

  MsgBox "Integer: " & Count
End Sub
```

Place this code in a `Standard` module file, and run it from the AutoCAD command prompt. Caution should be the rule when using this strategy. Assigning a String data type that does not represent a valid number to a numeric variable causes a run-time error. The next section outlines the Visual Basic functions that check the values being converted and give you feedback about their validity.

Converting Data Types

To help you convert one data type to another, Visual Basic provides the conversion functions shown in Table 2-4.

Table 2-4. *Data Type Conversion Functions*

Function Name	Converts To
`CBool`	Boolean
`CByte`	Byte
`CCur`	Currency
`CDate`	Date
`CDbl`	Double
`CInt`	Integer
`CLng`	Long
`CSng`	Single
`CStr`	String
`CVar`	Variant
`CVErr`	Error

Using these conversion functions ensures that you assign appropriate values to the different variables. The value that you convert must fit in the target data type's value range. If you try to convert to a smaller data type, you may encounter errors or your program may behave strangely.

Introduction to Arrays

An *array* is a variable that can hold multiple values of the same data type, each value placed into a separate compartment. In contrast, a normal variable can hold only one value.

Tip If you need to store an array of different kinds of values, create an array of Variants.

You can work with arrays as a whole or by using an index value to access each element. AutoCAD uses arrays for 3-D WCS coordinates. Arrays are nothing more than a collection of values treated as a single unit, analogous to a `List` in Visual LISP.

Fixed-Length Arrays

If you know how many elements your array will contain, then declare it with that many elements. The following example, typical of an AutoCAD VBA application, creates a three-element array:

```
Dim StartPoint(0 to 2) As Double
```

This example populates each element of the array with a value:

```
StartPoint(0) = 0
StartPoint(1) = 0
StartPoint(2) = 0
```

Detecting an Array's Bounds

When you use an array to control a looping operation, you need to know the array's lower and upper bounds, or dimension. Going beyond an array's lower and upper bounds generates a run-time error. Visual Basic provides two functions, `LBound` and `UBound`, that tell you these two values. The following example uses an array of unknown size to control a `For ... Next` loop:

```
Dim Index As Integer

For Index = LBound(GivenArray) To UBound(GivenArray)
   .
   .
   .
Next Index
```

Dynamic Arrays

Sometimes you don't know how many elements will be needed, and sometimes you just don't want to create more elements than you need. Visual Basic lets you create an array whose size you can change as your application runs. To do so, declare your array variable without specifying its dimensions in the parentheses, as in this example:

```
Dim Index() As Integer
```

When you define the array in your source code, use the `ReDim` statement:

```
ReDim Index(10)
```

`ReDim` can appear only in a procedure because it executes an action at run time, unlike the `Dim` and `Static` statements.

Whenever you redimension an array, the array's contents are erased. You can keep this from happening by using the `Preserve` keyword, as follows:

```
ReDim Preserve Index(UBound(Index) + 1)
```

This example uses UBound to find the current array's size and then adds one more element to it. Using ReDim imposes some performance issues (such as speed) because it copies the array in the background. Each time you change the array's dimensions, Visual Basic makes a new array, copies the contents into it, and then replaces the original array with the new array. The performance issues become evident when manipulating large arrays that contain large elements.

Modules

VBA applications may consist of any number of UserForms and Standard and/or Class modules. This section briefly explains each component you can use in a project.

UserForm

Although not contained in separate files as in a stand-alone Visual Basic application, the UserForm module is an integral part of your project. The UserForm module contains the procedures you write to react to the UserForm's events and the controls you place on your UserForm. The UserForm module contains UserForm-level procedures, variables, and constants specific to the project. Also, if you were to export the UserForm module to a file and look at it with a text editor, you would see descriptions of the UserForm and all the controls and their property settings.

To import a UserForm module file (.frm), choose File ➤ Import File or press Ctrl+M. The UserForm module file must have been created by AutoCAD or a similar VBA-enabled application, such as Microsoft Word. If you create the UserForm module file in Visual Basic using the Microsoft Forms 2.0 form add-in, you can import them too. You can't, however, import a .frm file created by Visual Basic.

■**Note** UserForm modules in earlier versions of VBA are modal, meaning that you can't click away from them to other applications that might be running. To let the user access the AutoCAD window, you must first hide your application window. In AutoCAD 2006 with VBA 6.3, set the form window's ShowModal property at design time to make UserForm modules modal (True) or modeless (False).

Standard

The Standard module file (.bas) generally contains procedures and declarations that other modules in your application can access. The Standard module can contain global (available to the entire application) or module-level declarations of procedures, variables, constants, and programmer-defined data types. Your application does not have to use the code that you write in a Standard module. The Standard module gives you a place to put code that may be used frequently and that you would like to have handy when the need arises. Code modules may also include subroutines or functions that you will use frequently throughout your programs.

Class

The foundation of object-oriented programming in Visual Basic is the `Class` module. You can create new objects by writing code in `Class` modules that implement their own properties, methods, and events. They support the same features as the `UserForm` module but allow a new level of functionality such as creating ActiveX controls, ActiveX documents, and so on. AutoCAD VBA can create only code components. To create components such as ActiveX controls and ActiveX documents, you need Visual Basic.

`Class` modules hold the key to creating new objects. These objects may contain their own properties and methods and may be instantiated more than once. `Class` modules let you combine functionality into a single source. Tasks such as API function calls and common database activities are prime candidates for `Class` modules.

Procedures

Procedures are blocks of code that have a specific purpose. You will use three kinds of procedures in most of your applications: `Sub`, `Function`, and `Event`. Each of these has a specific purpose.

Sub

`Sub` procedures are blocks of code that another procedure in your application explicitly calls. They may contain parameters when called, but they do not return a value.

The syntax of a `Sub` procedure is

```
 [Private|Public|Static] Sub ProcedureName([argumentlist])
     <... block of statements ...>
End Sub
```

The statements between `Sub` and `End Sub` are executed each time the `Sub` procedure is called. By default, `Sub` procedures are `Public` and therefore may be called from a `UserForm`, `Class`, or `Standard` module unless specified otherwise using the `Private` keyword.

The argument list is similar to the variables declared in a `Sub` procedure. These are variables that the calling procedure passes to the `Sub` procedure when it is executed. For more details on passing arguments to procedures, see the "Passing Arguments to Procedures" section later in this chapter.

The following example illustrates a typical `Sub` procedure. In this example, the `Layers` collection displays the name of each layer in the currently active document.

```
Public Sub IterateLayers()
Dim Layer As AcadLayer
  For Each Layer In ThisDrawing.Layers
    Debug.Print Layer.Name
  Next Layer
End Sub
```

Function

`Function` procedures are similar to `Sub` procedures, except that they can return a value to the calling procedure.

The syntax of a `Function` procedure is

```
 [Private|Public|Static] Function ProcedureName([argumentlist]) [As VariableType]
     <... block of statements ...>
End Function
```

This example illustrates calling a `Function` procedure and assigning the return value to a variable of the appropriate type:

```
ReturnValue = FunctionName([argumentlist])
```

The `Function` procedure's return value has a data type. If the data type is not specified, it is Variant by default.

`FunctionName` is used to return the `Function` procedure's value. The value becomes part of the function expression that called the `Function` procedure:

```
Public FunctionName(arguments) As DataType
     <... statements ...>

     FunctionName = ReturnValue
End Function
```

Event

`Event` procedures are executed when events occur in an object. You can declare `Event` procedures in a `Class` module or, more typically, as part of a `UserForm` module either attached to the `UserForm` itself or on ActiveX controls placed on the `UserForm`.

`Event` procedures use a unique naming convention that distinguishes them from other procedures:

```
ObjectVariable_EventName
```

This example shows the default event procedure for a typical `UserForm`:

```
Private Sub UserForm_Click()

End Sub
```

VBA passes arguments to some events when they occur, such as

```
Private Sub AcadDocument_BeginSave(ByVal FileName As String)

End Sub
```

This event is called just prior to AutoCAD saving the current drawing. VBA passes the current drawing's name as a string to the `Event` procedure. This may be useful if you have several drawings open and want to save only a particular drawing by checking the `FileName` parameter for that name.

Use `Event` procedures only when you must since they impose performance overhead by executing every time the event occurs. Overusing them can bring AutoCAD to a crawl or even crash it entirely.

Calling Procedures

You can't call a `Sub` procedure in a Visual Basic expression, which means you can't use one in an `If ... Then` statement in an assignment or in a comparative statement. You call a `Sub` procedure as a stand-alone statement, similar to the following:

```
CreateLayer Name
```

However, a `Function` procedure is typically part of an expression, as in the following:

```
If CheckForLayer("Layer1") Then
```

A `Sub` or `Function` procedure can modify the values of the arguments passed.

Passing Arguments to Procedures

There are two methods of passing arguments to procedures: by value and by reference.

By value: When you pass arguments by value (`ByVal`), only a copy of the variable is passed to the procedure. If the procedure modifies the variable's value, only the copy is changed. The original variable still contains its original value. Calling a procedure using the `ByVal` keyword lets you pass a value to the procedure as follows:

```
Public Sub ProcedureName(ByVal Variable)
.
.
.
End Sub
```

By reference: Passing arguments by reference (`ByRef`) is the Visual Basic default. Variables passed by reference let you change the original variable's value. Since `ByRef` is the default, you don't need to explicitly declare it as such when you call procedures.

■ **Note** Newer programming languages, such as VB .NET, C#, and VSA, have made `ByVal` the default.

Control Structures

Control structures control your application's flow and execution. Without control structures, your application would run from top to bottom. This may be suitable for the simplest of applications, but the power of Visual Basic lets you control the flow and execution and effectively change its order.

Decision Structures

Decision structures let you test the condition, value, or state of a variable and perform operations based on the test's result.

If ... Then

Use an `If ... Then` statement to conditionally execute one or more other statements. The `If ... Then` statement is used either in this way:

```
If <condition> Then <statement>
```

or in this way:

```
If <condition> Then
     <statements>
End If
```

The `<condition>` can be an expression that evaluates to a numeric value. Visual Basic interprets the numeric value as either `True` (nonzero) or `False` (zero).

When the `<condition>` evaluates to `True`, then Visual Basic executes the `<statement(s)>` following the `Then` keyword, as in the following single-line `If ... Then` statement:

```
If Count < 20 Then Count = Count + 1
```

To execute more than one statement after the conditional test, write it like this:

```
If Count < 20 Then
  Count = Count + 1
  Application.Visible = False
End If
```

The multiple-line version of the `If ... Then` statement requires that you end the statement with the `End If` keywords.

If ... Then ... Else

To test several conditions, define several blocks of statements where a statement is executed only when its condition is met. For example:

```
If <condition1> Then
  <statement block 1>

  Else If
    <statement block 2>

  Else
    <statement block 3>
End If
```

If `<condition1>` returns `True`, Visual Basic executes the statements following the `Then` keyword. Visual Basic ignores the statements in `<statement block 2>` and `<statement block 3>`, continuing execution with the statements following the `End If` keywords.

If `<condition1>` returns `False`, Visual Basic executes the statements following the `Else If` keywords, if supplied. Visual Basic ignores the statements in `<statement block 1>` and `<statement block 3>`, continuing execution with the statements following the `End If` keywords.

You may use any number of `Else If` statements, but only one `Else` statement, in an `If ... Then ... Else` statement.

Select Case

When you evaluate more than two conditions, it often makes more sense to use the `Select Case` control structure.

When you want to selectively execute a block of statements from a choice of many, use the `Select Case` structure, as in this example:

```
Select Case <TestExpression>
  Case <ExpressionList>
    <statements>
.
.
.
  Case Else
    <statements>
End Select
```

The `Select Case` structure begins with a single test expression. Visual Basic evaluates it once and compares it with each `Case` statement in the structure. When Visual Basic finds a match, it executes the associated block of statements.

The `<ExpressionList>` may contain one or more values separated by commas. The statement may contain zero or more statements. Visual Basic executes only `<statements>` associated with the first match to the `<TestExpression>`. After those `<statements>` are completed, execution continues with any statements following the `End Select` keywords.

You can include an optional `Case Else` statement that Visual Basic executes if it finds no `Case` matches.

This example shows how the `Select Case` structure works:

```
Select Case UCase(ColorName)
  Case "RED"
    Layer.Color = acRed

  Case "YELLOW"
    Layer.Color = acYellow

  Case "GREEN"
    Layer.Color = acGreen

  Case "CYAN"
    Layer.Color = acCyan

  Case "BLUE"
    Layer.Color = acBlue

  Case "MAGENTA"
    Layer.Color = acMagenta

  Case "WHITE"
    Layer.Color = acWhite
  Case Else
    If CInt(ColorName) > 0 And CInt(ColorName) < 256 Then
      Layer.Color = CInt(ColorName)

      Else
        MsgBox UCase(ColorName) & " is an invalid color name", _
               vbCritical, "Invalid Color Selected"
    End If
End Select
```

Another way to format `Select Case` is to put the return statements on the same line as the `Case` statements. This keeps the code format simpler when you have a small set of cases. Type a colon (`:`) after the `Case` statement to separate it from the return statement code:

```
Select Case Ucase(ColorName)
   Case RED: Layer.Color = acRed
   Case BLUE: Layer.Color = acBlue
 End Select
```

While the `Select Case` evaluates a single test expression at the top of the structure, the `If ... Then ... Else` structure evaluates a different expression for each `Else ... If` statement. Therefore, you replace an `If ... Then ... Else` structure with a `Select Case` statement if each `Else If` statement evaluates the same expression but looks for different values.

Loop Structures

Use loop structures to execute a series of statements repeatedly. Loops execute based on conditions. Depending on their structure, they can stop executing at their beginning or end.

Do While ... Loop

You can use a `Do While ... Loop` to execute a block of statements an infinite number of times. This loop first evaluates a numeric value. If it's `True` (nonzero), Visual Basic executes each statement in the loop. At the end of the loop, Visual Basic then reevaluates the condition and keeps executing the loop's statements until the condition is `False` (zero). When the condition is `False`, the loop skips over all its statements. Visual Basic executes the statements after the `Loop` keyword. Here's a `Do While ... Loop`'s syntax:

```
Do While <condition>
  <statements>
Loop
```

The following example first tests the `Index` value against a string's `Length`, then executes each statement, and finally increments the `Index` variable at the end of the loop. Execution continues until it evaluates the entire string or detects an illegal value.

```
Do While Index - 1 < Length
  Character = Asc(Mid(Name, Index, 1))
  Select Case Character
    Case 36, 45, 48 To 57, 65 To 90, 95
      IsOK = True
    Case Else
      IsOK = False
      Exit Sub
  End Select
  Index = Index + 1
Loop
```

In the `Do While ... Loop`, evaluation takes place at the top of the loop. If the condition is `False` (zero), then Visual Basic executes none of the loop's statements.

`Do ... Loop While`

Use the `Do ... Loop While` to execute the loop's statements at least one time before evaluating the condition. If the condition evaluates to `True` (nonzero), execution continues at the top of the loop. Otherwise, the condition is `False` (zero), and the loop stops. This loop has the following syntax:

```
Do
  <statements>
Loop While <condition>
```

`Do Until ... Loop` and `Do Loop ... Until`

There are two other loop structures: `Do Until ... Loop` and `Do Loop ... Until`. These loops run while the condition is `False` (zero) rather than `True` (nonzero). `Do Until ... Loop` has the following syntax:

```
Do Until <condition>
  <statements>
Loop
```

`Do Loop ... Until` has the following syntax:

```
Do
  <statements>
Loop Until <condition>
```

This example compares `Do While ... Loop` to `Do Until ... Loop`, iterating an ADO database recordset collection until they reach the recordset's end of file (EOF) marker. This is the result for `Do While ... Loop`:

```
Do While Not Recordset.EOF
     Debug.Print Recordset.Fields("layername").Value
Recordset.MoveNext
Loop
```

This is the result for `Do Until ... Loop`:

```
Do Until Recordset.EOF
     Debug.Print Recordset.Fields("layername").Value
     Recordset.MoveNext
Loop
```

For ... Next

Use a `For ... Next` loop when you know how many times you want to execute a block of statements. It uses a counter variable that you can either increase or decrease to control the number of cycles or times Visual Basic executes the loop. The `For ... Next` loop has this syntax:

```
For <counter> = <StartingValue> To <EndingValue> [Step <increment>]
  <statements>
Next  [<counter>]
```

Note The optional `<increment>` value can be positive or negative. If it's positive, then `<StartingValue>` must be less than or equal to `<EndingValue>`. Otherwise, the statements in the loop never execute. If it's negative, then `<StartingValue>` must be greater than or equal to `<EndingValue>`. If you don't set a step, it defaults to 1.

When a `For ... Next` loop executes, the following sequence of events occurs:

1. The `<counter>` variable is set to `<StartingValue>`.
2. If `<counter>` is greater than `<EndingValue>`, Visual Basic exits the loop. If the `<increment>` variable is negative, Visual Basic tests whether `<counter>` is less than `<EndingValue>`.
3. Visual Basic executes each statement in the loop.
4. Visual Basic increments or decrements the `<counter>` variable by 1 or by the `[Step <increment>]` value, if you specified one.
5. Visual Basic repeats steps 2 through 4 until the `<condition>` in step 2 is met, when it exits the loop.

The following example illustrates using the `For ... Next` loop:

```
Dim Point(0 To 2) As Double
Dim Index As Integer

  For Index = 0 To 2
    Point(Index) = 0
  Next Index
```

`For ... Each ... Next`

Use a `For ... Each ... Next` structure to iterate through an object collection's elements or through an array, executing a block of statements for each element. This structure iterates through each element of the collection or array regardless of how many elements the collection or array contains. `For ... Each ... Next` has this syntax:

```
For Each <element> In <collection>
  <statements>
Next <element>
```

The following restrictions apply when using `For ... Each ... Next:`

- For collection objects, the element data type can be Variant, generic (late-bound object), or a specific (early-bound) object type.
- Arrays must contain only Variant data types.
- You can't use `For ... Each ... Next` with an array of programmer-defined data types. This is because the Variant data type can't contain user-defined data types.

The following example illustrates using this structure:

```
Public Sub DisplayLayers()
     Dim Layer As AcadLayer

       For Each Layer In ThisDrawing.Layers
         Debug.Print Layer.Name
       Next Layer
End Sub
```

Nested Control Structures

Placing one control structure in another is called nesting. You can nest control structures to any level you want, but indent each level for readability. The following example demonstrates nesting control structures:

```
Public Sub WhatColor()
Dim Layer As AcadLayer
Dim Answer As String

  For Each Layer In ThisDrawing.Layers
    Answer = InputBox("Enter color name: ")

    'user pressed cancel
    If Answer = "" Then Exit Sub

    Select Case UCase(Answer)
      Case "RED"
        Layer.Color = acRed
      Case "YELLOW"
        Layer.Color = acYellow

      Case "GREEN"
        Layer.Color = acGreen

      Case "CYAN"
        Layer.Color = acCyan

      Case "BLUE"
        Layer.Color = acBlue

      Case "MAGENTA"
        Layer.Color = acMagenta

      Case "WHITE"
        Layer.Color = acWhite

      Case Else
        If CInt(Answer) > 0 And CInt(Answer) < 256 Then
          Layer.Color = CInt(Answer)

          Else
            MsgBox UCase(Answer) & " is an invalid color name", _
                 vbCritical, "Invalid Color Selected"
        End If
    End Select
  Next Layer
End Sub
```

Exiting a Control Structure

Using the Exit statement immediately exits a For or Do loop. The syntax to exit a loop is as follows:

```
Do While <condition>
  <statements>

  Exit Do
Loop
```

or as follows:

```
For Count = 0 To 5
  <statements>

  Exit For
Next Count
```

Exiting a `Sub` or `Function` Procedure

Using the `Exit` statement immediately exits a `Sub` or `Function` procedure. The syntax is simple:

```
Public Sub SubName()
.
  Exit Sub
.
End Sub
```

The syntax can also be as follows:

```
Public Function FunctionName() As Object
.
  Exit Function
.
End Function
```

It is useful to exit a `Sub` or `Function` procedure when a condition has been met and you don't want to execute any remaining code. This provides a means of exiting the procedure as soon as the condition is met.

`With ... End With`

The `With ... End With` statement lets you shorten your coding work by telling the compiler to repeatedly use an implicit reference to a named object without having to restate the object each time you use it. Use this feature anywhere in a procedure that you repeatedly refer to an object. Consider the following example:

```
Dim mylayer As AcadLayer
mylayer = ThisDrawing.ActiveLayer
mylayer.Color = acBlue
mylayer.Linetype = "continuous"
mylayer.Lineweight = acLnWtByLwDefault
mylayer.Freeze = False
mylayer.LayerOn = True
mylayer.Lock = False
```

Using the `With ... End With` statement, you do less typing:

```
Dim mylayer As AcadLayer
mylayer = ThisDrawing.ActiveLayer
With mylayer
.Color = acBlue
.Linetype = "continuous"
.Lineweight = acLnWtByLwDefault
.Freeze = False
.LayerOn = True
.Lock = False
End With
```

Application Writing Techniques

This section presents techniques for writing readable code. Failing to implement any of these techniques has no effect on your code's execution.

Writing Statements on Multiple Lines

To write a long statement on more than one line, use the line continuation character, which is a space followed by an underscore (_). Writing your code this way makes it more readable both in print and on the screen. This example shows the line continuation character in action:

```
'create the solid
Set AcadSolid = ThisDrawing.ModelSpace.AddExtrudedSolidAlongPath _
                (AcadRegion(0), AcadSpline)
'concatenate a long string value
Mypath = Application.Path & "\My Custom Folders\VBA Programs" & _
                "\Program 1\DVB Files"
```

■**Note** Don't place comments after the line continuation character on the same line of code. There are also some restrictions as to where you can place a line continuation character, such as in the middle of an object or procedure name.

Using this feature lets you format your code for easier reading. However, do not overuse this feature because too much of it can make your code unreadable.

Combining Statements on a Single Line

Typically, you place only one statement on a line of source code. But sometimes combining more than one statement on a single line can make your code more readable, especially in AutoCAD. Consider these six lines of code:

```
StartPoint(0) = 0
StartPoint(1) = 0
StartPoint(2) = 0
EndPoint(0) = 1
EndPoint(1) = 1
EndPoint(2) = 0
```

You can make them more readable by placing similar statements on the same line:

```
StartPoint(0) = 0: StartPoint(1) = 0: StartPoint(2) = 0
EndPoint(0) = 1: EndPoint(1) = 1: EndPoint(2) = 0
```

Each statement is separated by a colon (:). The colon appears only between statements and not at the end of the line.

Adding Comments to Your Code

Making notes about how your code sections function is more a necessity than an option. Comments explain the purpose of the code not only to you but to other developers. When you return to a section of code months later, if you have properly commented it, you won't have to guess or waste time trying to determine what is being executed.

Just as important as commenting functions and code blocks, always provide a comment for files themselves, usually at the top. This comment block explains what the file is for, who wrote it, and when it was written. You can also include notes about related files and dependent and specific requirements for using the program. The following example is just one way you could comment your program files:

```
"*********************************************************
" Filename: myProgramFile.dvb
" Author:   Joe Sutphin
" Date:     September 12, 2003
" Purpose:  This program inspires awe in my customers!
"*********************************************************
```

All comments begin with an apostrophe ('), called the comment character. Visual Basic ignores all text you type after the comment character to the end of the line.

The following example shows how you can use comments to document your program statements:

```
'define the Start and End point
StartPoint(0) = 0: StartPoint(1) = 0: StartPoint(2) = 0
EndPoint(0) = 1: EndPoint(1) = 1: EndPoint(2) = 0
```

Overview of Object-Oriented Programming

When you create an application in VBA, you use objects whether you realize it or not. Both AutoCAD and VBA are objects. This section examines the fundamentals of object-oriented programming.

Objects and Classes

An object is the creation, or instantiation, of a class. A class is a template, or definition, of a potential object. This template is used to create as many objects as needed based on a single class definition. Each object is an instance of the class. The action of creating a new object instance is referred to as instantiation.

The word *class* refers to classifying objects. For example, the entity objects `Line` and `Circle` are instances of the `AcadLine` and `AcadCircle` classes, respectively.

Visual Basic and VBA use `Class` modules to define classes. An object, which is an instance of the class, is created based on the `Class` module definition. `Class` modules consist of a declaration section followed by a series of `Sub`, `Function`, and `Property` subroutines similar to the `UserForm` and `Standard` modules. Variables and code in the `Class` module can be used only by creating an object, which is an instance of the class.

Object Data

An object contains data about itself. For example, an `AcadLine` object contains data about its starting and ending points, color, and thickness. The object also knows its linetype and the layer it sits on. These values are properties of the object. There may be many `AcadLine` objects, each having its own set of similar property values but each created from the same `AcadLine` class.

Instance variables store data values for each object instance. Each object instance contains its own set of variables, which were created from the same class. This is possible because each instance of the class, although created from the same class, must have a unique name. The `Name` property accesses each object and its properties. Each instance of a particular class has a unique name identifier.

Variables used in a `Class` module are declared at the module level, as the following pseudocode shows:

```
Private StartPoint(0 To 2) As Double
Private EndPoint(0 To 2) As Double
```

You can declare variables in a `Class` module as either `Private` or `Public` similar to `UserForm` and `Standard` modules.

Private Variables

You can access variables you declare as `Private` only in the module in which you declare them. Their main use in the `Class` module is to manage code specific to the object. Client programs can access class code only through the class interface. This adds a layer of security around your objects so that external applications can't circumvent the rules, haphazardly changing values without using the interface that you provide for accessing your object.

Public Variables

You can access variables you declare as `Public` from within your `Class` module and from a client application. It's generally bad practice to use `Public` variables in `Class` modules. When you declare a variable as `Public`, you grant unrestricted access to it, trusting that the client application will not change them inappropriately or provide incorrect data. Using `Public` variables also means that there is no code to force control over how the client application manipulates the object.

One primary reason for using object-oriented programming techniques is that they hide or encapsulate the class's inner workings, exposing a client application's interface only for changing your objects.

Variables declared as `Private` force the client application to use the object's code to change the object's data, letting you change how the code is implemented with little or no change on the client application's part. Variables declared as `Public` directly break the concept of encapsulation, providing a client with direct and uncontrolled access to the object's data.

An Object's Behavior

The concept behind objects is to let applications interact with them. If the object's data is encapsulated in the object, then you must provide a means for a client application to access the object's data.

Note Autodesk makes a clear distinction between an entity and an object in terms of an entity being a graphical instance. For example, a `Line` object is the database representation of the `Line` entity, which is the selectable item in the drawing editor window. In general, Autodesk calls a graphic an *entity* and calls the entity's programmatic aspects an *object*.

An object provides an interface that allows client applications to access its data. Methods, properties, and events provide a client application with this access to the object.

The object's interface contains `Property`, `Sub`, and `Function` routines and any events implemented in the object. Declaring any of these as `Public` makes them part of the object's interface. An object can contain multiple interfaces, just as in the real world, letting the client application access the object's data.

Properties

Properties generally describe an object's characteristics such as height, width, length, and color or the object's state such as On/Off or Thaw/Freeze. An object typically contains many properties that provide the client application with access to most, if not all, of its characteristics.

Methods

Methods are `Sub` procedures or functions that act on the object, such as rotating, offsetting, or moving it. Most AutoCAD objects contain a small number of methods. Also, many of the methods are the same for similar objects, such as all the graphical objects or all the non-graphical objects.

Events

Events are actions that the end user can perform while the application runs, including clicking the mouse button or pressing a key. No preset actions happen—you must provide the code to perform whatever tasks you want when an event occurs. This is perhaps the trickiest part of programming—several events can occur rapidly, and it is challenging to decide which event to supply code to in order to respond to the event.

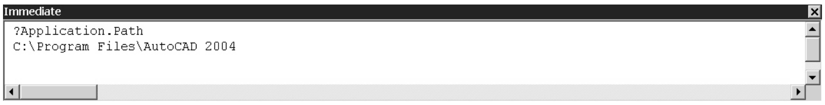
Debugging Basics

This section discusses the basics of debugging your application. Debugging is more an art form than a science, and as such you'll need to decide what works best for you. These techniques give you a good understanding of the available tools and how to use them.

The Immediate Window

Use the Immediate window to display the results of statements in your code. Sometimes referred to as the *Debug window*, its main purpose is to display debugging information. To open the Immediate window, choose View ➤ Immediate Window or press Ctrl+G.

To execute code in the Immediate window, type the statement into it and press the Enter key. To see the statement's return value, precede the statement with a question mark (?), as shown in Figure 2-2.



```
?Application.Path
C:\Program Files\AutoCAD 2004
```

Figure 2-2. *Viewing the* `Application.Path` *statement's return value*

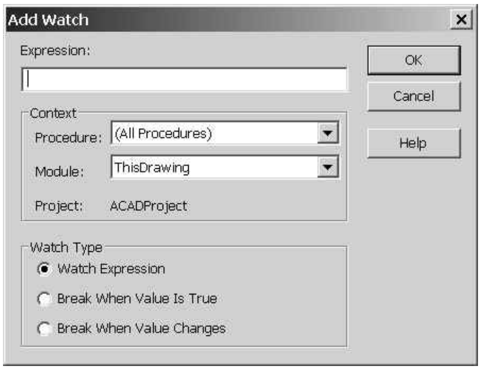
You typically use the Immediate window for these tasks:

- Testing new code
- Querying and/or changing a variable's value while your application is running
- Querying and/or changing a property value while your application is running
- Calling procedures similarly to calling them in code
- Viewing your program's debugging information while it runs

To can get help on syntax for functions, statements, properties, or methods in the Immediate window, select the keyword, property name, or method name and press F1.

Adding a Watch

Adding a watch lets you observe the value of any valid Visual Basic expression. To create a watch, choose Debug ➤ Add Watch. The dialog box in Figure 2-3 appears.



Procedure: (All Procedures)
Module: ThisDrawing
Project: ACADProject
Watch Type: [✓ Watch Expression]

Figure 2-3. *Creating a watch using the Add Watch dialog box*

If you have watches already defined in your code, the Watch window appears when the program's execution hits that watch. If a watch expression is out of scope, the Watch window contains no value. To display the Watch window, choose View ➤ Watch Window.

The Call Stack

The Call Stack window, shown in Figure 2-4, lists started procedures that have not completed. This feature is available only in break mode, which is when program execution pauses. When the current procedure's code is executed, VBA adds the procedure to a list of active procedures. VBA also adds each `Sub`, `Function`, or `Property` procedure that the procedure calls. As execution returns to the calling procedure, each procedure is removed from the list. The Immediate window procedures you execute are added to the Call Stack as well. To open the Call Stack window, when in break mode, click View ➤ Call Stack or press Ctrl+L.

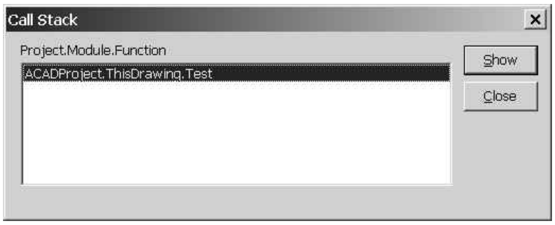


Figure 2-4. *Viewing unfinished procedures in the Call Stack window*

The Locals Window

The Locals window, shown in Figure 2-5, is similar to the Watch window, except that it displays local values—that is, variables declared in the current procedure. To open the Locals window, choose View ➤ Locals Window.

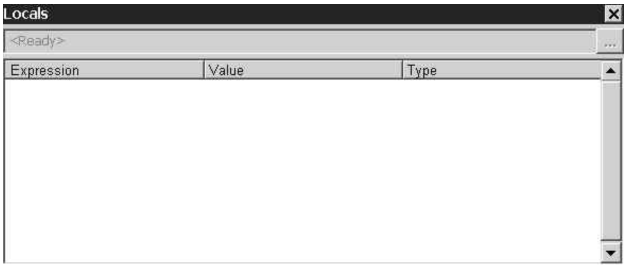


Figure 2-5. *Viewing local values in the Locals window*

Breakpoints

You use breakpoints to stop program execution at a statement in a procedure where you think a problem may exist. Breakpoints are cleared when they are no longer needed.

To set a breakpoint, follow these steps:

1. Place the insertion point somewhere on the line of code where you want to stop execution.

2. Choose Debug ➤ Toggle Breakpoint, press F9, or click next to the statement in the margin indicator bar if it's visible.

VBA adds the breakpoint and highlights the line using the breakpoint color defined on the Editor tab in the VBAIDE's Options dialog box.

Note Setting a breakpoint on a line that contains several statements separated by colons (`:`) breaks at the first statement on the line.

To clear a breakpoint, follow these steps:

1. Place the insertion point somewhere on the line of code containing the breakpoint.
2. Choose Debug ➤ Toggle Breakpoint, press F9, or click next to the statement in the margin indicator bar if it's visible.

VBA clears the breakpoint and removes the highlighting.

To clear all the breakpoints in your application, choose Debug ➤ Clear All Breakpoints or press Ctrl+Shift+F9.

Note Breakpoints are not saved when you save your code.

Stepping Through Your Code

You can step through your code, which executes it one line at a time. Using this feature lets you monitor the effects of code on system and application variables.

VBA gives you four methods to step through your code: `Step Into`, `Step Over`, `Step Out`, and `Run to Cursor` (see Table 2-5).

Table 2-5. *Methods for Stepping Through Code*

Method	Function
`Step Into`	Executes your code one statement at a time. To use this method, choose Debug ➤ Step Into or press F8.
`Step Over`	Executes a procedure as a whole unit, and then returns and steps to the next statement in the current procedure. To use this method, choose Debug ➤ Step Over or press Shift+F8.
`Step Out`	Executes the remaining code of a `Sub` or `Function`, and then displays the statement following the procedure call. This feature is available in break mode only. To use this method, choose Debug ➤ Step Out or press Ctrl+Shift+F8.
`Run to Cursor`	Executes from the current statement in design mode to where you place the cursor. To use this method, click Debug ➤ Run to Cursor or press Ctrl+F8.

The On Error Statements

If you don't use an `On Error` statement, any run-time error is fatal—that is, an error message appears and execution stops.

■**Note** An error-handling routine is a section of code marked by a line label such as `HandleError:` or a line number.

`On Error` initiates error handling, which lets the application decide whether an error is fatal. It provides a structured means of controlling how execution will continue for nonfatal errors and stops execution for fatal errors.

Error-handling routines typically rely on the value in the `Err` object's `Number` property. The error-handling routine should test or save relevant property values in the `Err` object before any other error can occur or before the program calls a procedure that might cause an error. The `Err` object's property values reflect only the most recent error. `Err.Description` contains the error message associated with `Err.Number`.

■**Note** VBA, VBScript, Visual Basic, and Visual Basic .NET are significantly different from each other in their error-handling features and capabilities.

On Error Resume Next

The `On Error Resume Next` statement allows execution to continue despite a run-time error. Execution can continue either with the statement right after the statement that caused the run-time error or with the statement right after the most recent call out of the procedure that contains the `On Error Resume Next` statement. You can place the error-handling routine inline with where the error would occur, rather than transferring control to another location in the procedure:

```
Dim oLayer As AcadLayer
  On Error Resume Next
  Set oLayer = Application.ActiveDocument.Layers.Add(Name)
  If Err = 0 Then
    Set CreateLayer = oLayer
  End If
```

The `On Error Resume Next` statement becomes inactive when another procedure is called, so use an `On Error Resume Next` statement in each called routine if you want inline error handling in that routine.

Note The `On Error Resume Next` structure may be preferable to `On Error GoTo` when handling errors generated during access to other objects. Checking `Err` after each interaction with an object removes ambiguity about which object the code accessed, which object placed the error code in `Err.Number`, and which object originally generated the error (specified in `Err.Source`).

On Error GoTo 0

`On Error GoTo 0` disables the current procedure's error handling. It doesn't specify line 0 as the start of the error-handling code, even if the procedure contains a line numbered 0. Without an `On Error GoTo 0` statement, Visual Basic disables error handling when it exits the procedure.

Exit Sub, Exit Function, and Exit Property

To prevent error-handling code from running when no error has occurred, place an `Exit Sub`, `Exit Function`, or `Exit Property` statement immediately before the error-handling routine, as in this example:

```
Public Function ConnectToAutoCAD(WindowState As AcWindowState) As AcadApplication
  On Error GoTo HandleError
  . . .
  Exit Function

ExitHere:
  Exit Function

HandleError:
  . . .
  Resume ExitHere
End Function
```

In this example, the error-handling code follows the `Exit Function` statement and precedes the `End Function` statement to separate it from the procedure flow. You can place error-handling code anywhere in a procedure, but you typically place it at the end for easy maintenance.

The Err Object

The `Err` object contains information about run-time errors. You will use the `Err` object quite a bit to keep your applications from crashing on the end user at run time. Typically, you'll most frequently use the `Err` object's `Description` and `Number` properties and its `Clear` method.

The Description Property

The `Description` property is a string describing the error. Users will appreciate you displaying this property in a `MsgBox` control instead of crashing the program.

The `Number` Property

The `Number` property is the error's number. Some errors give you only a number to use. All of this book's examples list both the `Description` and `Number` properties.

To use the `Err` object, include the following statement in your procedure, which executes if an error should occur:

```
On Error GoTo HandleError
```

If an error occurs, execution goes directly to the error handler, as shown in this example:

```
ExitHere:
  Exit Sub

HandleError:
  MsgBox Error:  & Err.Description &  ( & Err.Number & )
  Resume ExitHere
```

If an error should happen, this example successfully traps the error and lets your application exit gracefully. Include this construct where the user is likely to make an error. The `Resume` statement tells Visual Basic where to continue execution after handling the error.

■**Note** For a more detailed list of `Number` property values and their meanings, search for the phrase *trappable errors* in AutoCAD VBA's online help. Most OLE objects return errors whose numbers are in a unique range; you usually have to convert them to get their descriptions.

The `Clear` Property

The `Clear` property clears any error's `Err` object. The most common use of this property is when VB programmers try to connect or start an instance of AutoCAD, as in this example:

```
Dim Application As AcadApplication

On Error Resume Next

Set Application = GetObject(Class:="AutoCAD.Application")

If Err Then
  Err.Clear

  Set Application = CreateObject(Class:="AutoCAD.Application")

  If Err Then
    MsgBox "Error connecting to AutoCAD", vbCritical, "AutoCAD Fatal Error"
    Exit Sub
  End If
End If
```

In this example, the `On Error Resume Next` statement tells Visual Basic to keep executing code should an error occur. This lets you gracefully handle the error. This example first tries to connect to an already running instance of AutoCAD. If that attempt fails, then the `Err` object holds the error code. The `Err` object's `Clear` method then clears the error. It then tries to start AutoCAD. If that attempt fails, then the application can't continue.

Summary

This chapter presented the essence of the Visual Basic programming language. It also discussed common techniques for debugging your applications. The next chapter covers your application's user interface, including graphical elements such as using ActiveX controls and designing and handling your application's visual aspects.

CHAPTER 3

Application Elements

AutoCAD VBA provides support for creating complete applications, including dialog box interfaces. This chapter examines how to create and manipulate these dialog boxes and how to place and manipulate ActiveX control components such as text boxes and list boxes.

Designing a `UserForm`

The `UserForm` object is the canvas upon which you visually design your application, and it provides the windows your users interact with when they run the application. `UserForms` have their own properties, methods, and events you can use to control their appearance and behavior.

The first step in designing a `UserForm` is to set its properties, such as width and height. You may set a `UserForm`'s properties at design time or at run time using code.

Adding a `UserForm` to Your Application

If your application requires a user interface or dialog with the user, you need to add a `UserForm`. A `UserForm`, or dialog box, is a window in which several bits of information can be gathered or displayed at once. A login dialog box is a simple example of a form, whereas the AutoCAD Select File dialog box in Figure 3-1 represents a more complex application example.

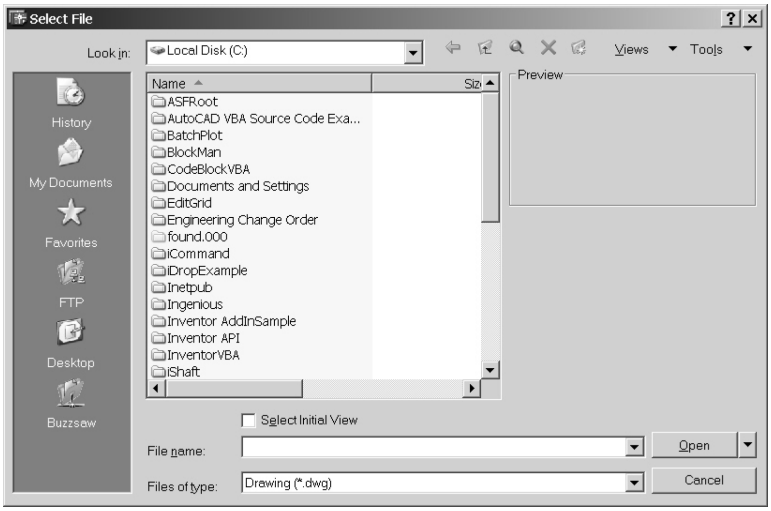


Figure 3-1. *The Select File dialog box*

To add a UserForm to your project, select Insert UserForm. Alternatively, you can use a drop-down toolbar button, shown in Figure 3-2, that enables you to add these four basic components: UserForm, Module, Class Module, and Procedure.

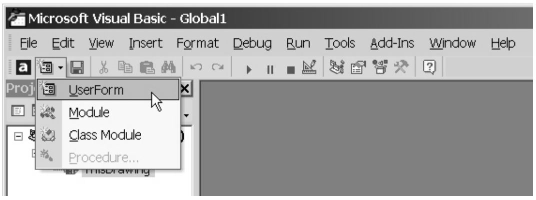

Microsoft Visual Basic - Global1
File Edit View Insert Format Debug Run Tools Add-Ins Window Help
UserForm
Module
Class Module
Procedure...

Figure 3-2. *The drop-down toolbar button showing various modules*

Once you've chosen to add a UserForm to your project, your project should look something like Figure 3-3.

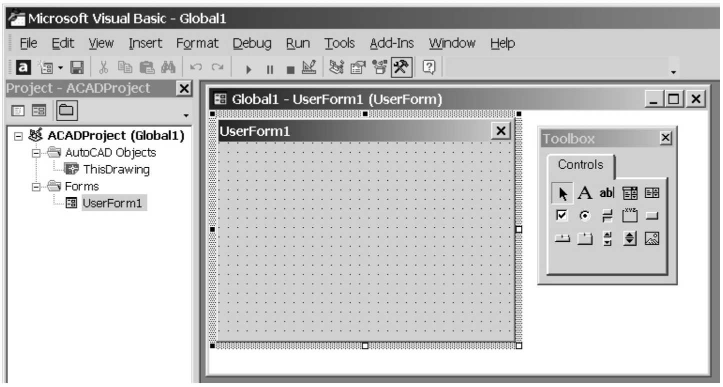

Microsoft Visual Basic - Global1
File Edit View Insert Format Debug Run Tools Add-Ins Window Help
Project - ACADProject
ACADProject (Global1)
AutoCAD Objects
ThisDrawing
Forms
UserForm1
Global1 - UserForm1 (UserForm)
UserForm1
Toolbox
Controls

Figure 3-3. *The AutoCAD VBAIDE*

Setting UserForm Properties

UserForm properties control a form's physical appearance. The UserForm.Caption property defines the text that appears on the left-hand side of the UserForm title bar, as shown in Figure 3-4. UserForms in AutoCAD 2000 VBA are modal and therefore don't contain a minimize or maximize button on the right side of the UserForm title bar. UserForms in AutoCAD 2002 through 2006 VBA (VBA 6.3), however, can be either modal or modeless.

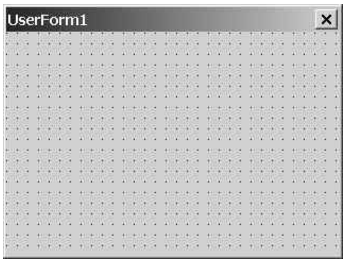


Figure 3-4. *The* Caption *property of* UserForm

Note When the ShowModal property is set to False, you may work outside the UserForm at run time. Normally, the UserForm focus remains off even while you work in the UserForm. By setting a Reference to the AutoCAD Focus Control for VBA Type Library (acFocusCtrl16.dll in the AutoCAD folder) and then adding acFocusCtrl (a transparent Control that will appear on the Control Toolbox) to the UserForm, focus will remain on the UserForm until you click outside of it.

The UserForm.Height and UserForm.Width properties control the initial UserForm height and width, respectively, as shown in Figure 3-5.

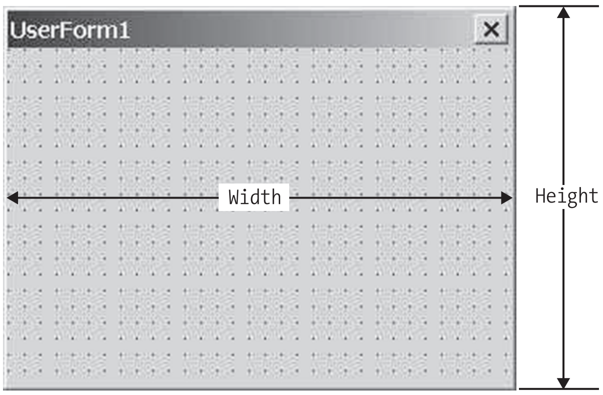


Figure 3-5. *The* Width *and* Height *properties of* UserForm

However, you may change these properties at run time as demonstrated in the following example:

```
Private Sub UserForm_Activate()
    UserForm1.Width = 200
    UserForm1.Height = 150
End Sub
```

In this example, the UserForm's Width and Height properties change when the UserForm is activated through the Activate method.

As you can see in Figure 3-6, the UserForm.Left and UserForm.Top properties control the initial UserForm position relative to the upper-left corner of the screen.

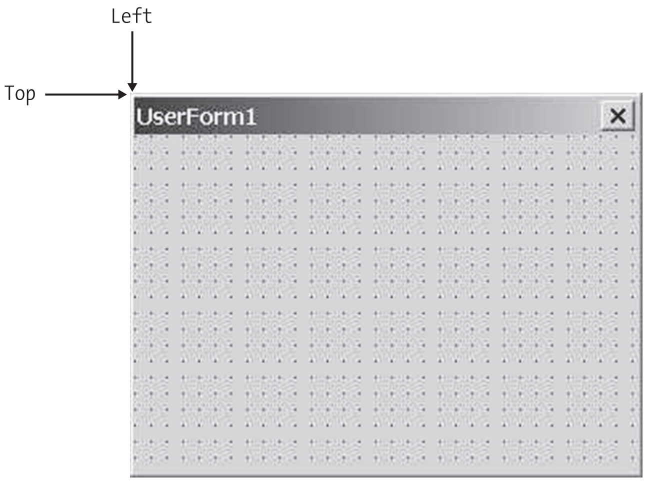


Figure 3-6. *The* `Top` *and* `Left` *properties of* `UserForm`

Adding a Control to a Form

When you add a `UserForm` to your project, another window immediately appears. This window, called the Toolbox, is where you'll find the intrinsic, or default, ActiveX controls that are available for you to design your user interface (see Figure 3-7).

Placing an ActiveX control on a form is simply a matter of clicking the control in the Toolbox and then clicking the area of the `UserForm` where you want to place the control. Each time you drop a new control onto a form, VBA automatically provides a default name and index number for the control. For example, the first `TextBox` control will be named `TextBox1` by default. You can rename controls by modifying their `Name` property.

When you click the ActiveX control you want, the cursor becomes a plus (+) sign, and the icon of the control appears to the lower right of the plus sign when the cursor is moved over the `UserForm`. The middle of the plus sign signifies the upper-left corner (the `Left` and `Top` properties) of the control. By default, the control will snap to the closest grid point displayed on your `UserForm`. Grid density can be controlled in the General tab of the Tools/Options dialog.

For example, pick the `TextBox` control, as shown in Figure 3-8. Place the control in the upper-left corner of the `UserForm`, similar to what you see in Figure 3-9.

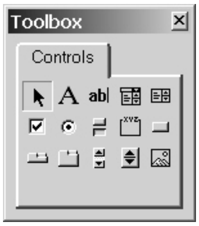


Figure 3-7. *The ActiveX Toolbox*

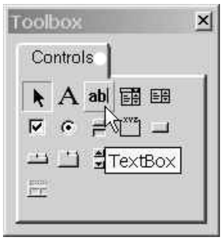


Figure 3-8. *The Toolbox with IntelliSense displayed*

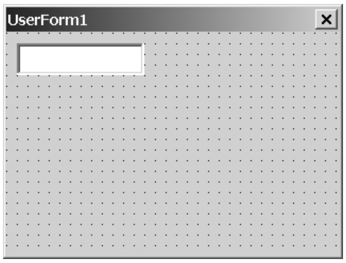


Figure 3-9. *A* `UserForm` *with a* `TextBox` *control*

Now create a dialog box interface that looks something like the example depicted in Figure 3-10. (Don't worry about making it look exactly like the figure; you just need to have the same controls on a form for this example.) Table 3-1 lists the controls and their respective property settings.

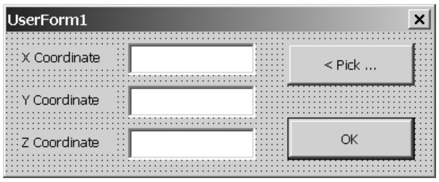


Figure 3-10. *A completed* `UserForm`

Table 3-1. *Class Controls and Their Property Settings*

Control Class	Name	Caption
`Label`	`Label1`	X Coordinate
`Label`	`Label2`	Y Coordinate
`Label`	`Label3`	Z Coordinate
`CommandButton`	`cmdPick`	< Pick …
`CommandButton`	`cmdOK`	OK
`TextBox`	`txtX`	
`TextBox`	`txtY`	
`TextBox`	`txtZ`	

Next you'll enter some code and see how to make this dialog box interface work. First, you need to open the `UserForm` code module by selecting View ➤ Code or pressing F7. Or you may start by double-clicking the < Pick … command button and entering the code listed here:

```
Private Sub cmdPick_Click()
Dim Point As Variant
  On Error Resume Next
  'hide the UserForm
  UserForm1.Hide
  'ask user to select a point
  Point = ThisDrawing.Utility.GetPoint(, "Select a point")
  If Err Then Exit Sub
  'assign values to appropriate textbox
  txtX = Point(0): txtY = Point(1): txtZ = Point(2)
  'redisplay the UserForm
  UserForm1.Show
End Sub
```

This is all the code you need to make this example work. Execute this example by selecting Run ➤ Run Sub/UserForm or by pressing F5. Alternatively, you can run the example by clicking the Run toolbar button as shown in Figure 3-11.

This final bit of code allows you to exit your application gracefully:

```
Private Sub cmdOK_Click()
  Unload Me
End Sub
```

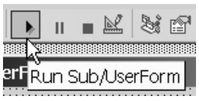


Figure 3-11. *The VBAIDE Run, Pause, and Stop buttons*

Visual Basic ActiveX Controls

In this section, you'll be presented with each of the ActiveX controls that are available for use by default. You'll take a look at code examples demonstrating how you might use each of these controls within your own application.

The AutoCAD VBA environment provides you with 14 ActiveX controls. In this section I briefly explain what the most commonly used ActiveX controls do and how you may use them. Each of the controls appears in the Toolbox, shown in Figure 3-12, that is displayed whenever you have a `UserForm` active.

You can close the Toolbox window if needed. To redisplay it, select View ➤ Toolbox.

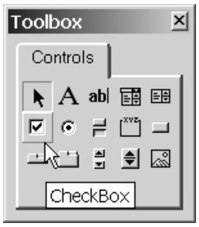


Figure 3-12. *The Toolbox window*

`Label`

Used to convey information back to the user, the `Label` control looks similar to what you see in Figure 3-13 when you place it on a `UserForm`.

Typically, you use this type of control to display error messages, entity counts, etc. `Labels` prove most useful when you control them at run time by changing the `Caption` property, as in the following snippet:

```
Private Sub UserForm_Activate()
  Label1.Caption = "# of Blocks = " & ThisDrawing.Blocks.Count
End Sub
```

Executing this bit of code produces the dialog box output shown in Figure 3-14.

When you use a `Label` control, you may need to adjust the `Label1.Width` and/or `Label1.Height` properties to compensate for the amount of text to be displayed.

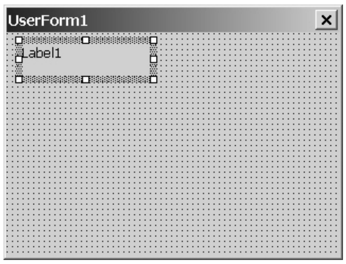


Figure 3-13. *A `UserForm` with a `Label` control*

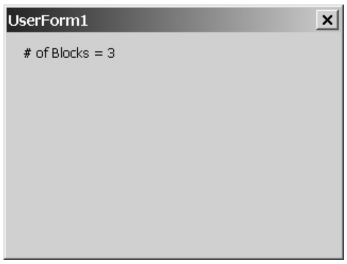


Figure 3-14. *A `UserForm` at run time with a `Label` control*

TextBox

You use TextBox controls for data entry by the user (see Figure 3-15). Usually these controls enable the user to enter a single line of text, but you can change them to allow multiline text entry.

Typically, you'll trap the KeyDown, KeyUp, and KeyPress events for a TextBox control. The KeyDown and KeyUp events occur in sequence when any key is pressed.

A KeyPress event may occur when any of the following keys or key combinations are pressed:

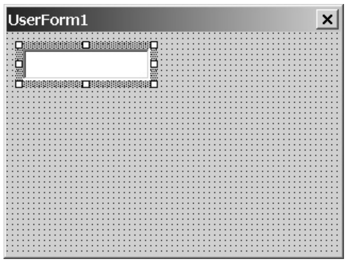


Figure 3-15. *A* UserForm *with a* TextBox *control*

- Any printable keyboard character
- Ctrl combined with a character from the standard alphabet
- Ctrl combined with any special character
- Backspace
- Esc

A KeyPress event won't occur under any of the following situations:

- Pressing the Tab key
- Pressing the Enter key
- Pressing an arrow key
- When a keystroke causes the focus to move from one control to another

Note Backspace is part of the ANSI character set, but Delete isn't. Deleting a character in a control using Backspace causes a KeyPress event; deleting a character using Delete doesn't trigger a KeyPress event.

The following example waits until the user presses the Enter key, and then prints the text entered in a standard Visual Basic MsgBox dialog box:

```
Private Sub TextBox1_KeyUp(ByVal KeyCode As MSForms.ReturnInteger, _
                       ByVal Shift As Integer)
  If KeyCode = 13 Then
    MsgBox "You entered: " & TextBox1.Text
  End If
End Sub
```

ComboBox

The ComboBox control, an example of which appears in Figure 3-16, allows a selection from a standard group of possible responses presented in a drop-down list. The DropDown List style is very useful in controlling what the user enters. This style of ComboBox doesn't allow the user to enter any response; the user may only pick from the list. This means you don't need to check for invalid values in your code. Conversely, the DropDown Combo style allows users to enter a value if what they want isn't in the list. As the programmer, you'll have to decide which style to use.

Here's an example of populating a drop-down list with values and then responding to the user's choice by displaying what was picked in a standard Visual Basic MsgBox dialog box:

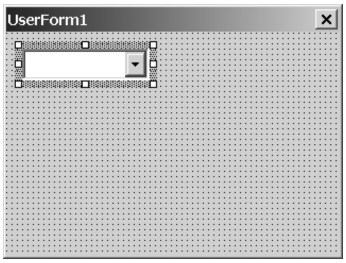


Figure 3-16. *A* UserForm *with a* ComboBox *control*

```
Private Sub UserForm_Activate()
  With ComboBox1
    .AddItem "Item 1"
    .AddItem "Item 2"
    .AddItem "Item 3"
  End With
End Sub

Private Sub ComboBox1_Click()
  MsgBox "You choose: " & ComboBox1.List(ComboBox1.ListIndex)
End Sub
```

ListBox

The ListBox control, shown in Figure 3-17, allows a selection from a standard group of possible responses. The difference between the ListBox control and the ComboBox control is the ListBox control displays more than a single choice at a time in a list format. Also, you can't type values into a ListBox control.

The following example illustrates how to populate a ListBox control and then respond to the Click event:

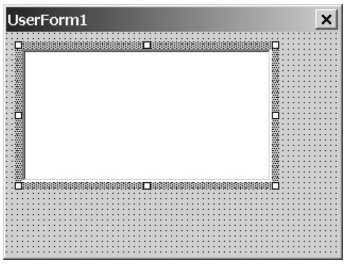


Figure 3-17. *A* UserForm *with a* ListBox *control*

```
Private Sub UserForm_Activate()
  With ListBox1
    .AddItem "Item 1"
    .AddItem "Item 2"
    .AddItem "Item 3"
  End With
End Sub

Private Sub ListBox1_Click()
  MsgBox "You clicked on: " & ListBox1.List(ListBox1.ListIndex)
End Sub
```

CheckBox

Use the CheckBox control to determine if an item is selected (see Figure 3-18). If the box is unchecked, then the item isn't selected. If the box is checked, then the user has selected that item. You may have any number of CheckBox controls on a UserForm; each CheckBox control's checked state is independent of any other CheckBox control.

The following example uses the `Click` event of the `CheckBox` control to determine the current state of the control:

```
Private Sub CheckBox1_Click()
  If CheckBox1.Value Then
    MsgBox "Checked"

    Else
      MsgBox "Unchecked"
  End If
End Sub
```

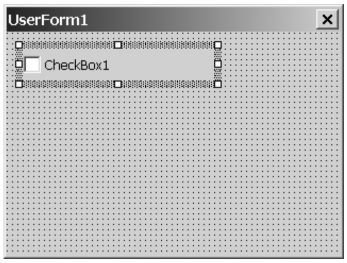


Figure 3-18. *A* `UserForm` *with a* `CheckBox` *control*

OptionButton

The `OptionButton` control is ideal for situations in which you want your user to choose just one item. Most AutoCAD users know these as radio buttons. The `OptionButton` control is usually placed inside a `Frame` control for grouping, as Figure 3-19 demonstrates. On a `UserForm` or within a `Frame` control, only one `OptionButton` may be selected at a time.

In the following example, the `Click` event of each of the `OptionButton` controls contains code that is executed depending on which `OptionButton` is clicked:

```
Private Sub OptionButton1_Click()
  MsgBox "OptionButton1"
End Sub

Private Sub OptionButton2_Click()
  MsgBox "OptionButton2"
End Sub
```

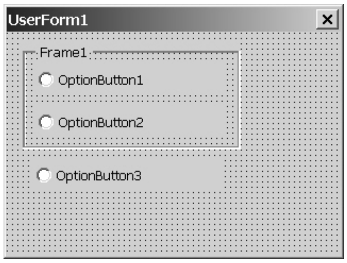


Figure 3-19. *A* `UserForm` *with* `OptionButton`*s*

ToggleButton

Use a `ToggleButton` control to enable an option and leave it that way until the user depresses the button again (see Figure 3-20). This control presents an on/off switch whose appearance changes depending on whether or not it the user has depressed it. The `ToggleButton` control has a `Picture` property you could change depending on the state of the button.

In the following example, the condition of the `ToggleButton` control is tested using a `Select ... Case` statement and an appropriate message is displayed:

```
Private Sub ToggleButton1_Click()
  Select Case ToggleButton1.Value
    Case False
      MsgBox "ToggleButton1 is Off"
    Case True
      MsgBox "ToggleButton1 is On"
  End Select
```

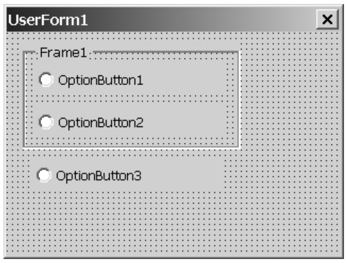


Figure 3-20. *A* `UserForm` *with* `ToggleButton`*s*

Frame

The Frame control, shown in Figure 3-21, is a container for other controls, similar to the UserForm. Controls placed within a Frame control will move when you move the Frame control. Also, if you set the Frame control's Enabled property to False, then all the controls within the Frame control are disabled. Coordinate placement within the Frame is based on the Frame and not the UserForm. The Frame control provides a means of grouping related controls together for easier user selection.

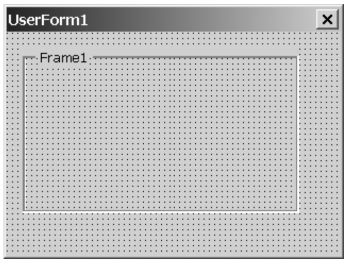


Figure 3-21. *A* UserForm *with a* Frame *control*

CommandButton

Use CommandButton controls, shown in Figure 3-22, to allow users to signify that they have made all the selections and text entries they want to make, and either want to continue on with the application or want to cancel the operation. The most common use of this control is for OK and Cancel button operations.

The following is the most common method of ending a running application:

```
Private Sub cmdOK_Click()
  Unload Me
End Sub
```

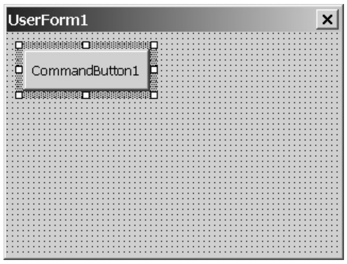


Figure 3-22. *A* UserForm *with a* CommandButton *control*

Additional ActiveX Controls

This section briefly explains the remaining five ActiveX controls available by default. Because these controls aren't commonly used, I don't cover them in as much detail as the previous controls. You can get further details about each of these ActiveX controls by placing one on a UserForm, highlighting it, and pressing F1.

TabStrip

Use TabStrip controls when the data format between each tab is the same, but each tab represents a different entity (see Figure 3-23). A good analogy would be a custom control. All TextBox controls have the same property definitions (the data that appears on every tab selection). However, each instance of the control must have a different name (the tab name). Controls added to a TabStrip are actually added to the UserForm. When you move the TabStrip, the other controls do not move with it.

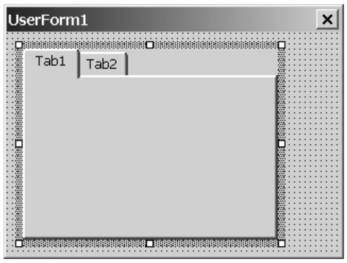


Figure 3-23. *A* UserForm *with a* TabStrip *control*

MultiPage

The MultiPage control, shown in Figure 3-24, is similar to a UserForm. Each tab or page is separate from the others. For example, if you add a control to Page1, the same control doesn't appear on any other page. Nor is it a part of the UserForm. The MultiPage control is similar to a Frame control in that when you move the MultiPage control, all the controls contained within it move as well.

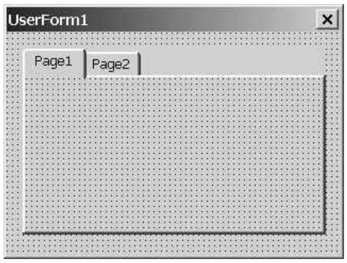


Figure 3-24. *A* UserForm *with a* MultiPage *control*

ScrollBar

Use ScrollBar controls to allow the user to easily change values based on clicking the up and down arrows of the control (see Figure 3-25). Typically a ScrollBar increases or decreases counting values within a TextBox control. This control contains a slider bar indicating its relative position to the minimum and maximum values. The slider also allows the user to quickly make very large changes to the value.

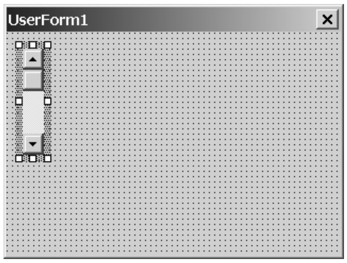


Figure 3-25. *A* UserForm *with a* ScrollBar *control*

SpinButton

A SpinButton control allows the user to change values easily based on clicking the up and down arrows of the control (see Figure 3-26). Typically, you use this control to increase or decrease counting values with a TextBox control. Values may be changed only by clicking the up and down arrows. Unlike ScrollBar controls, SpinButton controls have no slider bar.

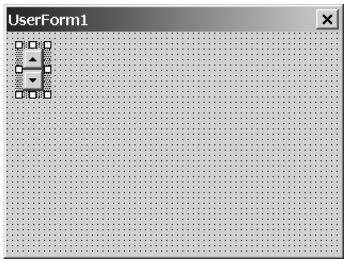


Figure 3-26. *A* UserForm *with a* SpinButton *control*

Image

The Image control, shown in Figure 3-27, enables you to display pictures on your UserForm. The following types of images are supported: *.bmp, *.cur, *.gif, *.ico, *.jpg, and *.wmf. With the Image control's default event being Click, you could also use it as a fancy CommandButton control with a picture. In addition to specifying the picture to be displayed, you can define the Image control's display properties such as clip, stretch, and zoom, as well as apply 3-D border effects on the control itself, such as Flat, Raised, and Sunken.

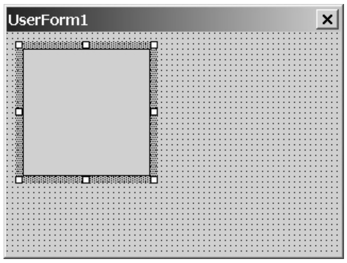


Figure 3-27. *A* UserForm *with an* Image *control*

Summary

This chapter presented the essence of the visual portion of your application. The combinations of the different ActiveX controls provided by AutoCAD VBA are practically endless, and the right combination for you will depend on the needs of your application. You're somewhat forced to find your own way when creating your user interface. In this chapter I discussed `UserForms`, how to add controls to a `UserForm`, and the various ActiveX controls available to you. One thought you may want to bear in mind when designing your user interface: keep it simple.

CHAPTER 4

AutoCAD Events

Events occur as a result of actions happening while your program is running, such as opening or saving a drawing. They allow you to write source code that will execute whenever that event occurs. Messages such as "Would you like to save changes?" are the common results of a user action that has triggered an event.

AutoCAD 2006 supports three levels of events: application, document, and object. These event levels correspond to the three major areas of AutoCAD. Event handlers are `Sub` procedures that are executed automatically every time their associated event occurs. Some AutoCAD events allow information to be passed to the event handlers through parameters.

Application-Level Events

Changes to the AutoCAD application environment result in application-level events. These include the opening and saving of drawings, running of AutoCAD commands, changes to system variables, and changes to the AutoCAD application window.

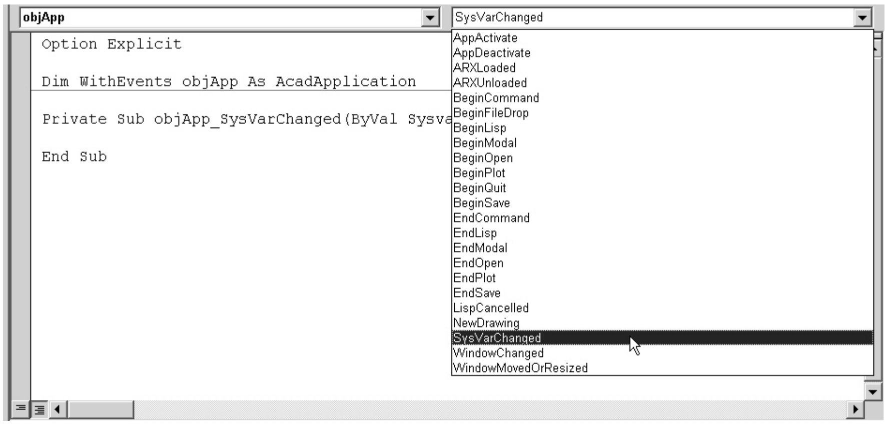
Application-level events aren't enabled when you load a VBA project. The following example illustrates the steps you need to take to enable application-level events.

First, insert a new class module by selecting Insert ➤ Class Module, and name the new class module appropriately, for example `clsApplicationEvents`. Then declare an object of type `AcadApplication` using the `WithEvents` keyword.

```
Public WithEvents objApp As AcadApplication
```

You should now see the new object `objApp` appear in the Object list box of the class module and all its event procedures are available in the Procedure list box, as shown in Figure 4-1.

You can then go ahead and write the code within these procedures that you want to be executed each time the events occur. However, these event handlers won't be triggered unless you've set the reference to correspond to the `Application` object.



```
Option Explicit

Dim WithEvents objApp As AcadApplication

Private Sub objApp_SysVarChanged(ByVal Sysva

End Sub
```

Figure 4-1. *The Object and Procedure boxes*

You can do this as follows. In the ThisDrawing or any code module, declare a variable to be a new instance of the class module you just created, and in a subroutine set this variable to hold a reference to the Application object.

```
Option Explicit
Public objApp As New clsApplicationEvents

Public Sub InitializeEvents()
  Set objApp.objApp = ThisDrawing.Application
End Sub
```

As soon as the InitializeEvents subroutine is called, in this case by running the App_StartMacro macro shown next, the application-level events are enabled.

```
Public Sub App_StartMacro()
  InitializeEvents
End Sub
```

The following examples illustrate writing code within the event procedures of the class module to execute when those events occur. The first informs the user when a system variable changes, and the second ensures that the AutoSave interval for a new drawing is always set to 30 minutes.

```
Private Sub objApp_SysVarChanged(ByVal SysvarName As String, _
                                 ByVal newVal As Variant)
  MsgBox "The System Variable: " & SysvarName & " has changed to " & newVal
End Sub

Private Sub objApp_NewDrawing()
    ThisDrawing.SetVariable "SAVETIME", 30
    MsgBox "The autosave interval is currently set to 30 mins"
End Sub
```

The following list summarizes the events available at the application level:

`AppActivate`

`AppDeactivate`

`ARXLoaded`

`ARXUnloaded`

`BeginCommand`

`BeginFileDrop`

`BeginLisp`

`BeginModal`

`BeginOpen`

`BeginPlot`

`BeginQuit`

`BeginSave`

`EndCommand`

`EndLisp`

`EndModal`

`EndOpen`

`EndPlot`

`EndSave`

`LispCancelled`

`NewDrawing`

`SysVarChanged`

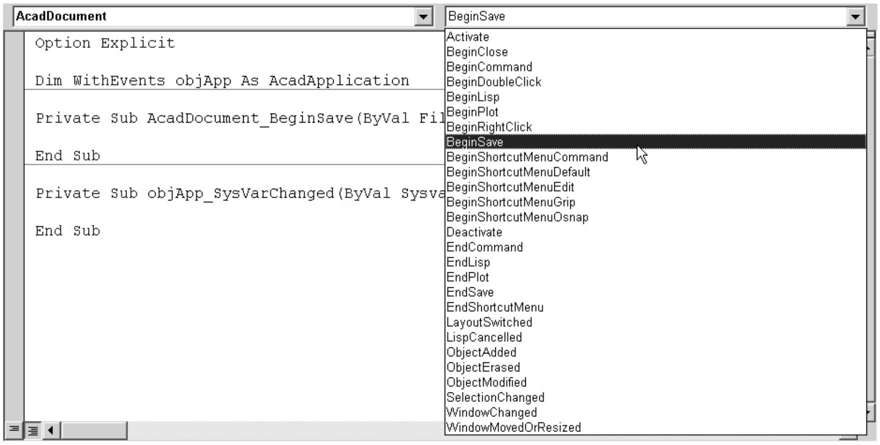
`WindowChanged`

`WindowMovedOrResized`

■**Note** Unlike Visual LISP, VBA provides no `CommandCancelled` or `CancelledCommand` event to respond to. When a user presses the Esc key, in most cases it won't fire the `EndCommand` event.

Document-Level Events

Changes to a document or its contents result in document-level events. Adding or editing objects and regeneration of the drawing are just some examples of document-level events. Unlike application-level events, document-level events are available by default in the ThisDrawing module of an AutoCAD project. If you choose the AcadDocument object in the Object list box of the ThisDrawing module, the document-level events are listed in the Procedure list box, as shown in Figure 4-2.



```
Option Explicit

Dim WithEvents objApp As AcadApplication

Private Sub AcadDocument_BeginSave(ByVal Fil

End Sub

Private Sub objApp_SysVarChanged(ByVal Sysva

End Sub
```

Figure 4-2. *The Procedure box showing document-level events*

The following is a summary of events available at the document level:

Activate

BeginClose

BeginCommand

BeginDocClose

BeginDoubleClick

BeginLisp

BeginPlot

BeginOpen

BeginRightClick

BeginSave

BeginShortcutMenuCommand

BeginShortcutMenuDefault

```
BeginShortcutMenuEdit

BeginShortcutMenuGrip

BeginShortcutMenuOsnap

Deactivate

EndCommand

EndLisp

EndOpen

EndPlot

EndSave

EndShortcutMenu

LayoutSwitched

LispCancelled

ObjectAdded

ObjectErased

ObjectModified

SelectionChanged

WindowChanged

WindowMovedOrResized
```

Next you'll look at some of the document events and why you might want to add code to execute when they occur.

The `BeginCommand` and `EndCommand` Events

When you issue an AutoCAD command such as `LINE` or `DIM`, the `BeginCommand` event is triggered. Any code that you've written inside the `BeginCommand` event procedure is then executed. Once the code associated with the `BeginCommand` event has finished executing and after the command itself has finished, the `EndCommand` event is triggered. Now any code that you've written inside the `EndCommand` event procedure executes. You may have code associated with either, both, or neither of the events. If a command is canceled prior to completion, such as when a user presses the Esc key, it doesn't fire the `EndCommand` event.

The following `BeginCommand` event procedure illustrates creating a layer called Objects (if it doesn't exist) and making the layer active based on the user starting the `LINE` command:

```
Option Explicit
Public objCurrentLayer As AcadLayer
Public objPreviousLayer As AcadLayer
```

```
Private Sub AcadDocument_BeginCommand(ByVal CommandName As String)
  Set objPreviousLayer = ThisDrawing.ActiveLayer
  Select Case CommandName
    Case "LINE"
      If Not ThisDrawing.ActiveLayer.Name = "OBJECTS" Then
        Set objCurrentLayer = ThisDrawing.Layers.Add("OBJECTS")
        ThisDrawing.ActiveLayer = objCurrentLayer
      End If
  End Select
End Sub
```

The corresponding `EndCommand` event procedure puts the user back to the layer he or she was on if you had to change it in order to draw a line:

```
Private Sub AcadDocument_EndCommand(ByVal CommandName As String)
  Select Case CommandName
    Case "LINE"
        ThisDrawing.ActiveLayer = objPreviousLayer
    End Select
  Set objCurrentLayer = Nothing
  Set objPreviousLayer = Nothing
End Sub
```

The reason for using such code is that it brings continuity to your drawings. For example, all lines will be on the Objects layer, and by adding further code, all text could be added automatically to a Text layer.

The `BeginOpen` and `EndOpen` Events

When AutoCAD receives a request to open an existing drawing file, the `BeginOpen` event is triggered. Once AutoCAD has finished loading the drawing file and it's visible, an `EndOpen` event occurs. One possible use of this event procedure would be to store all the current system variables before you open a drawing. Then once the drawing is opened, you restore the system variables back to their previous values.

When AutoCAD receives a request to create a new drawing file, a slightly different sequence of events occurs. The `BeginOpen` event occurs as before, and then the `BeginSave` event occurs (see the section "The `BeginSave` and `EndSave` Events" for details). This is followed by an `EndSave` event and finally an `EndOpen` event.

The `BeginClose` and `BeginDocClose` Events

The `BeginClose` event is triggered upon the closing of a drawing session within AutoCAD. Be careful when you use this event! If you attempt to perform a lengthy task, it can frustrate users due to slowness, or it could even result in serious problems with AutoCAD, causing it to become unstable, lock up, or crash entirely. The `BeginDocClose` event is similar to the `BeginClose` event. However, the `BeginDocClose` event allows you to cancel the Close command.

The Activate and Deactivate Events

The Activate event is triggered when a drawing window gains focus. When only one drawing is opened in AutoCAD, it will always have focus. When multiple drawings are opened, this event is triggered when switching between drawing windows. The drawing window that loses focus triggers the Deactivate event. Normally, the Deactivate event is triggered just before the Activate event as drawing window focus is switched.

Keep in mind that the Deactivate event indicates the drawing has lost focus. Firing off a procedure as a result of this event might not be a good idea, as it may not complete its task until the drawing regains focus (indicated by another Activate event). You can develop programs to work with what is called a zero document state, meaning there are no drawings opened. Consult the Autodesk developer guide for more information on this topic.

The BeginSave and EndSave Events

Immediately before AutoCAD begins to save the current drawing, the BeginSave event is triggered. Once AutoCAD has completed saving the drawing file, the EndSave event occurs. You might use the BeginSave event to query whether or not the user wants to purge his or her drawing before saving it. You could use the EndSave event to reinitialize standard layers, linetypes, and text styles that may have been purged because they weren't currently being used.

As you can see, AutoCAD has provided some very useful events that greatly enhance its controllability.

Object-Level Events

Object-level events occur when changes are made to a specific entity that you've declared as having events. Modified is the only object-level event and, as you would expect, it occurs when the specified object is modified.

To use object-level events, you must first create a new class module and declare a variable to hold a reference to the object whose Modified event you want to code. You might call the new class module something like clsObjectEvent. The new class module contains the declaration of the object using the VBA keyword WithEvents, as in the following example:

```
Public WithEvents objLine As AcadLine
```

The new object then appears in the Object list box of the class module and the event procedure for the new object may now be written within the class module in the same way as for other subroutines. For your event procedures to be triggered, you must associate the declared object in the class module with the object of interest. For this Line object example, you could do this by placing the following code in the ThisDrawing module or any code module:

```
Dim objLine As New clsObjectEvent
Public Sub InitializeEvent()
Dim dblStart(2) As Double
Dim dblEnd(2) As Double
    dblEnd(0) = 1: dblEnd(1) = 1: dblEnd(2) = 0
    Set objLine.objLine = ThisDrawing.ModelSpace.AddLine(dblStart, dblEnd)
End Sub
```

You first declare your object to be a new instance of the class `clsObjectEvent`, and in the initial event procedure set the `objLine` variable to hold a reference to a newly created `Line` object. Now, as soon as this procedure is called, a new `Line` object is created that responds to any changes made to the line in question by executing the code in the `Modified` event procedure.

So if you put the following code in the `clsObjectEvent` class module, the new coordinates of the `Line` object will be displayed to the user, whenever the line is moved, rotated, scaled, etc.:

```
Private Sub objLine_Modified(ByVal pObject As AutoCAD.IAcadObject)
Dim varStartPoint As Variant
Dim varEndPoint As Variant

    varStartPoint = pObject.StartPoint
    varEndPoint = pObject.EndPoint
    MsgBox "New line runs from (" & varStartPoint(0) & ", " & _
        varStartPoint(1) & ", " & varStartPoint(2) & " ) to (" & _
        varEndPoint(0) & ", " & varEndPoint(1) & ", " & varEndPoint(2) & ")."

End Sub
```

Summary

In this chapter you've seen how AutoCAD events can greatly increase programming flexibility. The steps involved in initializing events may seem a little complex to those first encountering events, but hopefully this won't deter you from taking full advantage of the power and flexibility they provide.

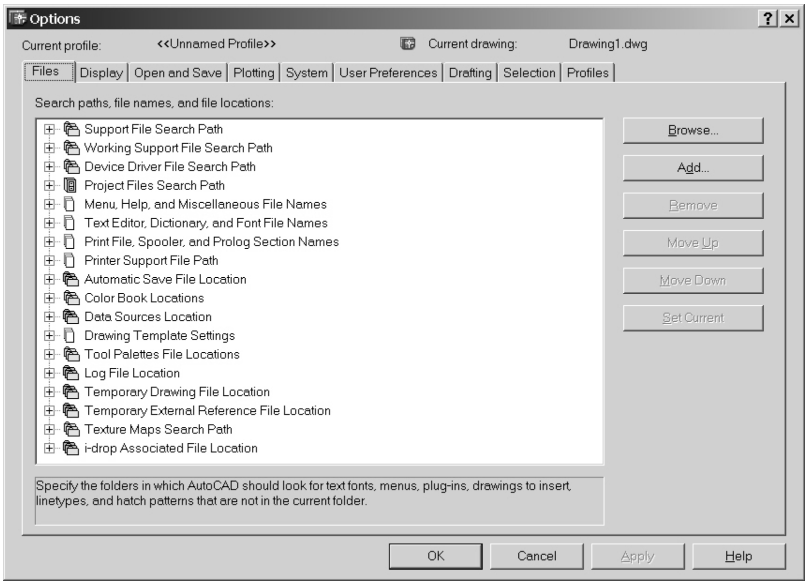
CHAPTER 5

User Preferences

There are actually two Preferences objects within AutoCAD: AcadPreferences and DatabasePreferences. The AcadPreferences object is stored by AutoCAD and applies to all drawing sessions. The DatabasePreferences object (also called the Document Preferences) is stored with each drawing file and applies only to the drawing in which it was saved.

This chapter covers a few aspects of the AcadPreferences object. A quick way to discern the differences between the AcadPreferences and DatabasePreferences objects is to look through the Options dialog box tabs and note the features with a drawing icon beside them. These indicate Document Preferences. All others are AcadPreferences features.

Users can set many different properties that affect the way they work with AutoCAD. For example, users can set the paths that are searched to find support files and programs, the properties affecting performance, and how the display is presented. You can view and set these properties through the Options dialog box, which you can access through the Tools ➤ Options menu, as shown in Figure 5-1, or by typing **Options** at the command prompt.


Options
Current profile: <<Unnamed Profile>>
Current drawing: Drawing1.dwg
Files
Display
Open and Save
Plotting
System
User Preferences
Drafting
Selection
Profiles
Search paths, file names, and file locations:
Support File Search Path
Working Support File Search Path
Device Driver File Search Path
Project Files Search Path
Menu, Help, and Miscellaneous File Names
Text Editor, Dictionary, and Font File Names
Print File, Spooler, and Prolog Section Names
Printer Support File Path
Automatic Save File Location
Color Book Locations
Data Sources Location
Drawing Template Settings
Tool Palettes File Locations
Log File Location
Temporary Drawing File Location
Temporary External Reference File Location
Texture Maps Search Path
i-drop Associated File Location
Browse...
Add...
Remove
Move Up
Move Down
Set Current
OK
Cancel
Apply
Help

Specify the folders in which AutoCAD should look for text fonts, menus, plug-ins, drawings to insert, linetypes, and hatch patterns that are not in the current folder.

Figure 5-1. *The Options dialog box*

The `Preferences` collection object contains nine objects representing the nine tabs on the Options dialog box. These tabs roughly correspond to the objects contained within the `AcadPreferences` collection object. From left to right, they relate to the following `AcadPreferences` objects:

`PreferencesFiles`

`PreferencesDisplay`

`PreferencesOpenSave`

`PreferencesOutput`

`PreferencesSystem`

`PreferencesUser`

`PreferencesDrafting`

`PreferencesSelection`

`PreferencesProfiles`

AutoCAD 2000 and 2002 users will see a few new additions in AutoCAD 2004 and later, such as color book locations, tool palette file settings, i-drop settings, hover grip colors, security options, right-click options, hidden line options, and more. Accessing these objects may seem a bit odd, as they're called PreferencesFiles, but you request them from the `Preferences` object using their short name, such as Files.

For example, the following code returns the `PreferencesProfiles` object used to save a custom profile file for later use with each session:

```
Dim PrefProfiles As AcadPreferencesProfiles

Set PrefProfiles = ThisDrawing.Application.Preferences.Profiles
```

In this chapter, I cover the following aspects of the `Preferences` object:

- Getting and setting the support path(s)
- Controlling the cursor size
- Getting and setting the `AutoSaveInterval` property
- Getting and setting the drawing template file path
- Getting and setting the printer support path
- Getting and setting the file `SaveAs` type
- Enabling and disabling the Startup dialog box
- Saving and retrieving personal preferences

Getting and Setting Support Path(s)

Controlling support paths can be important if you're using custom applications during your AutoCAD session. Generally speaking, the default paths set by AutoCAD are probably not adequate because they don't include any paths for customs applications, for example. AutoCAD's saving grace is that you can change the default paths to accommodate your program needs. If you need to know the current path for your support files, you can read the `SupportPath` property of the `PreferencesFiles` object:

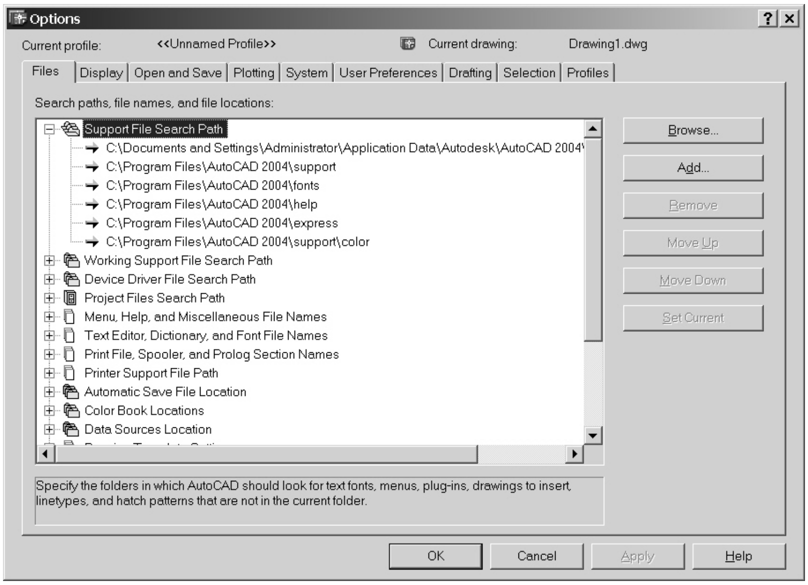
```
strSetPaths = ThisDrawing.Application.Preferences.Files.SupportPath
```

You can also append the `SupportPath` property as follows:

```
Dim strNewPath As String, strSetPath As String
strNewPath = ;c:\cadfiles\apress\dvb
strSetPath = ThisDrawing.Application.Preferences.Files.SupportPath
If Len(strSetPath & strNewPath) < 256 Then
  strNewPath = strSetPath & strNewPath
  ThisDrawing.Application.Preferences.Files.SupportPath = strNewPath
End If
```

Note that there is a 255-character limit on the cumulative length of the support path collection. This includes the semicolon delimiter used to store the value internally (it's actually stored as a single string and shown as a list only through the Options dialog form). If you exceed the limit, AutoCAD will truncate the paths, which may yield odd or unpredictable results.

In AutoCAD 2004 and later, the support path list changed, as did other default path behaviors. This is due to Microsoft Windows XP logo certification requirements that Autodesk sought for its latest product releases. For example, you'll see two new default path entries under support paths that refer to the user profile path, as shown in Figure 5-2.



Support File Search Path
- C:\Documents and Settings\Administrator\Application Data\Autodesk\AutoCAD 2004\
- C:\Program Files\AutoCAD 2004\support
- C:\Program Files\AutoCAD 2004\fonts
- C:\Program Files\AutoCAD 2004\help
- C:\Program Files\AutoCAD 2004\express
- C:\Program Files\AutoCAD 2004\support\color

Working Support File Search Path
Device Driver File Search Path
Project Files Search Path
Menu, Help, and Miscellaneous File Names
Text Editor, Dictionary, and Font File Names
Print File, Spooler, and Prolog Section Names
Printer Support File Path
Automatic Save File Location
Color Book Locations
Data Sources Location

Specify the folders in which AutoCAD should look for text fonts, menus, plug-ins, drawings to insert, linetypes, and hatch patterns that are not in the current folder.

Figure 5-2. *Support File Search Path entries in the Options dialog box*

Controlling Cursor Size

You may want to change the default cursor size to your particular preference. You can achieve this by using the CursorSize property of the PreferencesDisplay object:

```
intCursorSize = ThisDrawing.Application.Preferences.Display.CursorSize
```

The value of the CursorSize property is a positive integer that represents the percentage of the cursor size to the screen size. The default value is 5, the minimum is 1, and the maximum is 100. All other values will generate an error. The following example prompts the user to enter a value for the cursor size, and if the returned value lies between 1 and 100, the size is changed. Otherwise, the user is informed that he or she has entered an invalid value.

```
Dim intCursorSize As Integer

intCursorSize = ThisDrawing.Utility.GetInteger(vbCrLf & _
    "Enter number for size of cursor proportional to screen size" & vbCrLf)

If intCursorSize < 1 Or intCursorSize > 100 Then
    MsgBox "Cursor Size value must be between 1 and 100"
Else
    ThisDrawing.Application.Preferences.Display.CursorSize = intCursorSize
End If
```

Getting and Setting the AutoSaveInterval Property

The AutoSaveInterval property is a positive integer value representing the number of whole minutes between automatic saves. The timer for automatic saves starts as soon as you make a change to the current drawing. Setting the AutoSaveInterval property to 0 (zero) means that you'll never get an automatic save operation. The maximum value allowed for the AutoSaveInterval property is 600 minutes.

The default value that AutoCAD uses for the time between automatic saves is 120 minutes. You can do a lot of work in two hours, all of which could potentially be lost if you don't initiate a save operation. Most users, then, will want to set the AutoSaveInterval property to a much shorter interval.

The following example retrieves the current setting, and if the autosave interval isn't set to 15 minutes, it's altered and the user is informed of the change:

```
If ThisDrawing.Application.Preferences.OpenSave.AutoSaveInterval <> 15 Then
    ThisDrawing.Application.Preferences.OpenSave.AutoSaveInterval = 15
    MsgBox "The autosave interval has been changed to 15 minutes."
End If
```

■**Tip** Autodesk recommends setting `IncrementalSavePercent` to 0 if you experience problems saving files across a network connection. This forces a full save at every automatic save, which may require a few more milliseconds but ensures a complete disk-write at every save. This can also help you avoid the dreaded automatic-save backup files when you lose your sessions prematurely.

Getting and Setting the Drawing Template File Path

The `TemplateDWGPath` property is a string value specifying the path of template files used by the start-up wizards. The following example illustrates how to retrieve the current template files path:

```
strTemplatePath = ThisDrawing.Application.Preferences.Files.TemplateDwgPath
```

Setting the path to your specific value requires a fully qualified path, as shown here:

```
ThisDrawing.Application.Preferences.Files.TemplateDwgPath = _
    "C:\Program " &  "Files\AutoCAD 2006\Templates"
```

Manipulating the path settings can be helpful if you need to use various sets of templates from different customers or vendors, or if your internal departments must use different standards, such as the metric system and English system of measurement or the ANSI and ISO industry standards.

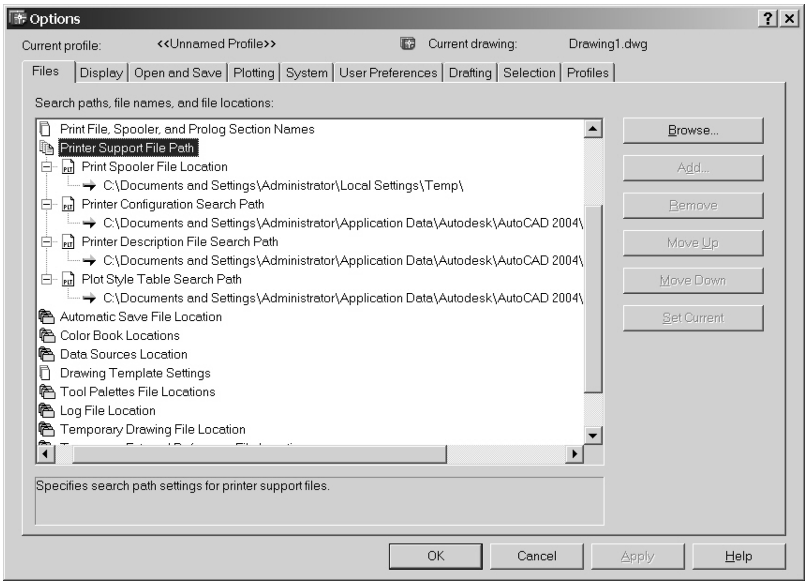
Getting and Setting the Printer Support Path

One of the most common uses of the `Preferences` object is to deal with printer or plotter configuration aspects, as shown in Figure 5-3. Among these are the plotter configuration support paths. These are displayed one way in the Options dialog form and somewhat differently in the `AcadPreferences.Files` object.

Printer Configuration Search Path is where the PC3 configuration files are stored and accessed. This is linked to the `PrinterConfigPath` property. This is normally the Plotters folder in AutoCAD 2000 and 2002, but it has been moved in AutoCAD 2006.

Printer Description File Search Path is where the PMP paper size and plotter calibration files are stored. This is normally the Drv folder in AutoCAD 2000 and 2002; however, it has been moved in AutoCAD 2006 to PMP Files under the Plotters folder. It's linked to the `PrinterDescPath` property.

Plot Style Table Search Path is where the CTB and STB plot style files are stored and accessed. This is normally the Plot Styles folder in AutoCAD 2000 and 2002, but it has also been moved in AutoCAD 2004 and later. This is linked to the `PrinterStyleSheetPath` property.



Search paths, file names, and file locations:

```
Print File, Spooler, and Prolog Section Names
Printer Support File Path
  Print Spooler File Location
    C:\Documents and Settings\Administrator\Local Settings\Temp\
  Printer Configuration Search Path
    C:\Documents and Settings\Administrator\Application Data\Autodesk\AutoCAD 2004\
  Printer Description File Search Path
    C:\Documents and Settings\Administrator\Application Data\Autodesk\AutoCAD 2004\
  Plot Style Table Search Path
    C:\Documents and Settings\Administrator\Application Data\Autodesk\AutoCAD 2004\
Automatic Save File Location
Color Book Locations
Data Sources Location
Drawing Template Settings
Tool Palettes File Locations
Log File Location
Temporary Drawing File Location
```

Specifies search path settings for printer support files.

Figure 5-3. *Printer Support File Path options in the Options dialog box*

The following example shows how to modify the Printer Configuration Search Path using VBA and the `AcadPreferences.File` object:

```
Dim strPC3Path As String
strPC3Path = c:\cadfiles\plotconfigs
ThisDrawing.Application.Preferences.Files.PrinterConfigPath = strPC3Path
```

Getting and Setting the File `SaveAs` Type

The `SaveAsType` property controls the format used when AutoCAD saves the current drawing. Table 5-1 shows the applicable values available for different formats supported by AutoCAD 2006 by using the `acSaveAs` enumeration. You can always explicitly override this setting either from the `SAVEAS` command or programmatically from the `SaveAs` method of the `Document` object in VBA.

Table 5-1. `acSaveAs` *Enumerations*

Name	Value	Description
`acR12_dxf`	1	AutoCAD Release12/LT2 DXF (*`.dxf`).
`ac2000_dwg`	12	AutoCAD 2000 DWG (*`.dwg`).
`ac2000_dxf`	13	AutoCAD 2000 DXF (*`.dxf`).
`ac2000_Template`	14	AutoCAD 2000 Drawing Template File (*`.dwt`).
`ac2004_dwg`	24	AutoCAD 2004 DWG (*`.dwg`).

Name	Value	Description
ac2004_dxf	25	AutoCAD 2004 DXF (*.dxf).
ac2004_Template	26	AutoCAD 2004 Drawing Template File (*.dwt).
acNative	24	A synonym for the current drawing release format. If you want your application to save the drawing in the format of whatever version of AutoCAD the application is running on, then use the acNative format.

Note The values shown in Table 5-1 are the only valid values that you can use. Valid values from earlier versions of AutoCAD are not shown in the table. Invalid values will generate an error.

The following example determines the current file format in which drawings will be saved:

```
Public Sub SaveAsType()
Dim iSaveAsType As Integer
  iSaveAsType = ThisDrawing.Application.Preferences.OpenSave.SaveAsType
  Select Case iSaveAsType
    Case acR12_dxf
      MsgBox "Current save as format is R12_DXF", vbInformation
    Case ac2000_dwg
      MsgBox "Current save as format is 2000_DWG", vbInformation
    Case ac2000_dxf
      MsgBox "Current save as format is 2000_DXF", vbInformation
    Case ac2000_Template
      MsgBox "Current save as format is 2000_Template", vbInformation
    Case ac2004_dwg, acNative
      MsgBox "Current save as format is 2004_DWG", vbInformation
    Case ac2004_dxf
      MsgBox "Current save as format is 2004_DXF", vbInformation
    Case ac2004_Template
      MsgBox "Current save as format is 2004_Template", vbInformation
    Case acUnknown
      MsgBox "Current save as format is Unknown or Read-Only", vbInformation
  End Select
End Sub
```

You set the SaveAsType property by using the same enumerated types as shown in the following example:

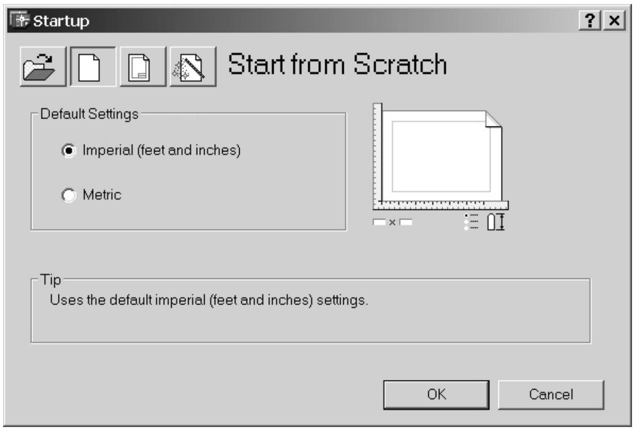
```
ThisDrawing.Application.Preferences.OpenSave.SaveAsType = ac2000_dwg
```

The default value for the SaveAsType property for AutoCAD 2004 is ac2004_dwg.

Tip In general, it's best (and safest) to leave the default `SaveAsType` setting as the current version and convert drawing files only at the end of the design phase, when you're preparing to deliver them to whomever requires the prior format. This eliminates the constant conversion process AutoCAD must do each time you open a drawing and save it. The repetitive conversion processes can impact performance and increase the risk of drawing database corruption. You can also batch-convert drawings using the Migration Assistance Batch Drawing Converter to save time at the end of a major project.

Enabling and Disabling the Startup Dialog Box

Users may or may not want the Startup dialog box as shown in Figure 5-4 when they open an existing drawing or start a new drawing. In AutoCAD 2004 and later, the Startup dialog box is disabled by default.



Uses the default imperial (feet and inches) settings.

Figure 5-4. *The Startup dialog box*

You can control whether this dialog box is displayed using the `EnableStartupDialog` property of the `PreferencesSystem` object, as in the following example:

```
ThisDrawing.Application.Preferences.System.EnableStartupDialog = False
```

The initial value of the `EnableStartupDialog` property is `True` for versions up to AutoCAD 2002 only. It is `False` for AutoCAD 2004 and later products.

An AutoCAD 2004 and later quirk is that although the command line setting has been removed from the Display tab of the Options dialog box, the corresponding property is still exposed under the `Display` object (`DockedVisibleLines`). However, modifying its value has no effect whatsoever due to the command prompt being converted to a tool palette object.

Saving and Retrieving Personal Preferences

Profiles are named environment configurations that allow you to load and change your entire desktop and support configurations at will. For example, you may create one for each project you work on, you may create one for each user on a shared workstation, or you may create one for each add-on application you use in AutoCAD.

Profiles are saved and recalled from the Windows Registry. You can save them and export them to `.arg` files, but they are actually Registry export files, having the same internal content and structure as a Windows `.reg` file. In fact, you can rename them as `.reg` files and double-click them to import them directly into the registry. AutoCAD, however, provides API access into the `Profiles` object to allow you to manage profiles from within AutoCAD.

Once you've settled on a standard for your preferences, you may want to save these settings in a file that you can later import for your current session. You should make all necessary changes to the preferences and then use the following code to export them to a file:

```
Dim strActiveProfile as String

strActiveProfile = ThisDrawing.Application.Preferences.Profiles.ActiveProfile
ThisDrawing.Application.Preferences.Profiles.ExportProfile _
    strActiveProfile, "C:\MYPROFILE.ARG"
```

Now that you've exported your preferences to a file, the following code will allow you to import these preferences into each of your AutoCAD sessions:

```
Dim strMyProfile As String

'name of profile
strMyProfile = "My Personal Profile"

ThisDrawing.Application.Preferences.Profiles.ImportProfile _
    strMyProfile, "C:\MYPROFILE.ARG", True
ThisDrawing.Application.Preferences.Profiles.ActiveProfile = strMyProfile
```

The `True` argument at the end of the `ImportProfile` method tells AutoCAD to preserve the path information from the `.arg` file and save this information into the Windows Registry. Also, notice that at the end of the preceding code I set the newly imported profile to be active. You must make your personal profile active for your settings to take effect.

If you attempt to import a profile from a `.arg` file, AutoCAD will always check to see if the profile name already exists within AutoCAD. If it exists, AutoCAD ignores the `.arg` file entirely. To force an import to take effect, you must either delete or rename the existing profile prior to importing. You can't delete the active profile, so it's usually easiest to rename it, import the new one, set it active, and delete the renamed profile, as shown in the following code:

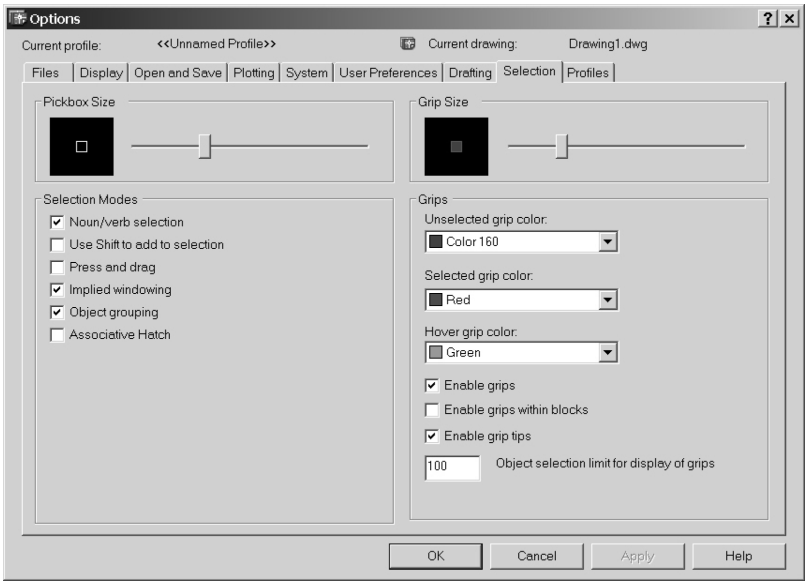
```
Dim strActiveProfile As String
Dim strMyProfile As String
strMyProfile = "My Personal Profile"
strActiveProfile = ThisDrawing.Application.Preferences.Profiles.ActiveProfile
With ThisDrawing.Application.Preferences.Profiles
```

```
.RenameProfile strActiveProfile " MyBackupProfile"
.ImportProfile strMyProfile, "C:\MYPROFILE.ARG ",True
.ActiveProfile = strMyProfile
.DeleteProfile "MyBackupProfile"
End With
```

Obviously, the preceding example doesn't verify whether the active profile is the one you want to replace, so that's something else you should check if you use this method. It's interesting to note that whereas the `Profiles` object contains a logical collection of profiles, there is no `Item` or `Count` property. The `ActiveProfile` property returns a string name, not an object. The methods also are string-based, not object-based. This means that you can't access and manipulate profiles as objects. However, you can copy, delete, rename, import, export, and reset them.

User Preferences Changes in AutoCAD 2004

AutoCAD 2004 (and hence, 2006) adds, removes, and modifies quite a few Options dialog box features, as well as many `AcadPreferences` objects. Be sure to read the online development documentation for changes in AutoCAD 2004. There are some interesting things to note, in addition to my comment earlier about changing the `DockedVisibleLines` property having no effect on the number of lines displayed for the command prompt. For example, although AutoCAD 2004 adds the new "hover grip" feature to the Options dialog box, the three color options are not represented cohesively under the hood. Figure 5-5 shows the Selection tab.


Options
Files | Display | Open and Save | Plotting | System | User Preferences | Drafting | Selection | Profiles
OK
Cancel
Apply
Help

Current profile: <<Unnamed Profile>> Current drawing: Drawing1.dwg

Pickbox Size

Selection Modes
- ✓ Noun/verb selection
- ☐ Use Shift to add to selection
- ☐ Press and drag
- ✓ Implied windowing
- ✓ Object grouping
- ☐ Associative Hatch

Grip Size

Grips
- Unselected grip color: Color 160
- Selected grip color: Red
- Hover grip color: Green
- ✓ Enable grips
- ☐ Enable grips within blocks
- ✓ Enable grip tips
- 100 Object selection limit for display of grips

Figure 5-5. *The Selection tab of the Options dialog box*

The `AcadPreferences.Selection` object contains only `GripColorSelected` and `GripColorUnselected`. The `GRIPHOVER` system variable is the only means to access and modify the hover grip color. Another curious issue is that of tool palettes, which are new to AutoCAD 2004. These aren't exposed through the ActiveX model anywhere; they're actually stored as XML documents under the `ToolPalettesPath` property of the `File` object. There currently is no means to configure their display properties from VBA without invoking XML services.

Summary

There's no way to cover every possible combination of preferences a typical user may use, but I've covered a few of the most useful ones in this chapter. Please refer to Appendix A for more details on preferences, including exact names and syntax.

CHAPTER 6

Controlling Layers and Linetypes

Effective use of layers and linetypes is the key to creating structured drawings and manipulating AutoCAD in an efficient manner. In this chapter, you'll see how to access the `Layers` and `Linetypes` collections and their respective `Layer` and `Linetype` objects, and learn about their methods and properties. The code samples throughout this chapter demonstrate how to control these objects through VBA.

Layers

A layer is a property of every AutoCAD drawing object. By using multiple layers, you may organize drawing data into logical categories. For example, when you design an office layout, you could use one layer to display the walls and other fixed structural objects, and you could use other layers to show the potential furniture or electrical arrangements. Alternatively, you could use a layer to hold dimensions (measurement annotations) or hidden lines. A `Layer` object represents one of these logical groupings.

Manipulating the state of layers makes it easier to manage complex drawings. For example, by making a layer visible or hidden, the user can choose to work with specific entity categories without being overwhelmed by all the other drawing entities.

Creating layers and controlling their state is the subject of the first part of this chapter. Later I discuss how to control linetypes.

The following list outlines the various actions that I cover with regard to layers:

- Accessing the `Layers` collection and `Layer` objects
- Checking for the existence of a specific layer
- Creating a new layer and making it the active layer
- Setting or returning the `On/Off`, `Thawed/Frozen`, `Locked/Unlocked` properties
- Renaming and deleting a layer
- Setting or returning a layer's `Color` and `Linetype` properties

Accessing Layers

AutoCAD has a Layers collection that contains all the Layer objects in the drawing. You can create as many layers as you want by adding new Layer objects to the Layers collection.

You access the Layers collection via a Document object. In the following code, ThisDrawing is used as the active document:

```
Dim objLayers As AcadLayers
Set objLayers = ThisDrawing.Layers
```

To set a reference to an existing Layer object, use the Item method of the Layers collection as follows:

```
Dim objLayer As AcadLayer

Set objLayer = objLayers.Item(2)
Set objLayer = objLayers.Item("My Layer")
```

The parameter of this method is either an integer representing the position of the desired Layer object within the Layers collection or a string representing the name of the desired Layer object. If you use an index number, it must be between 0 and the value of the Layers.Count property minus 1.

Like in other AutoCAD collections, Item is the default method for Layers. This means that the method name may be omitted, and the parameter passed straight to the Layers reference. Some programmers prefer this, as it's simpler to type and read. The following code does the same thing as the prior example using the default method to specify the Layer object:

```
Dim objLayer As AcadLayer

Set objLayer = objLayers(2)
Set objLayer = objLayers("My Layer")
```

However, I recommend that you avoid using default methods and instead strive to use explicit properties and methods. Microsoft is making a strong commitment to explicit coding in all of its current and future programming technologies, such as .NET. In fact, .NET doesn't support default methods or implicit data types by default.

Iterating Layers

In some situations you'll want your program to step through each item in a collection—perhaps to check or alter some property of every element. This is termed *iteration.* Like all collections in VBA, Layers has built-in support for iteration using a For ... Each loop. The following example iterates the Layers collection, printing each Layer name to the Immediate or Debug window:

```
Public Sub ListLayers()
    Dim objLayer As AcadLayer

    For Each objLayer In ThisDrawing.Layers
        Debug.Print objLayer.Name
    Next
End Sub
```

You may also iterate the collection manually using the Count property in conjunction with the Item method. The Count property contains the total number of elements in the collection.

Remember, when you access a collection element using a number, the index must be between 0 and the value of the Layers.Count property minus 1. This is because all collections are by default zero-based. The following example prints each Layer name to the Debug window as before, but this time iterates the collection manually:

```
Public Sub ListLayersManually()
    Dim objLayers As AcadLayers
    Dim objLayer As AcadLayer
    Dim intI As Integer

    Set objLayers = ThisDrawing.Layers

    For intI = 0 To objLayers.Count - 1
        Set objLayer = objLayers(intI)
        Debug.Print objLayer.name
    Next
End Sub
```

You can see that the manual version is a bit more complex, with several variables involved. For most purposes, the For ... Each version is preferred, but because the manual version gives you full control over the iteration, it may be needed in some situations. For example, the following sample prints the Layer names to the Debug window by iterating the collection in *reverse* order:

```
Public Sub ListLayersBackwards()
    Dim objLayers As AcadLayers
    Dim objLayer As AcadLayer
    Dim intI As Integer

   Set objLayers = ThisDrawing.Layers

   For intI = objLayers.Count - 1 To 0 Step -1
       Set objLayer = objLayers(intI)
       Debug.Print objLayer.name
   Next
End Sub
```

Checking for Existing Layers

At times your program may need to determine if an element is present in the collection. One way to do this is to search for the element while iterating the collection as described previously. The following example gets a layer name from the user and then checks for its existence by iterating the Layers collection:

```
Public Sub CheckForLayerByIteration()
    Dim objLayer As AcadLayer
    Dim strLayerName As String
    strLayername = InputBox("Enter a Layer name to search for: ")
    If "" = strLayername Then Exit Sub    ' exit if no name entered

    For Each objLayer In ThisDrawing.Layers    ' iterate layers
        If 0 = StrComp(objLayer.name, strLayername, vbTextCompare) Then
            MsgBox "Layer '" & strLayername & "' exists"
            Exit Sub                           ' exit after finding layer
        End If
    Next objLayer
    MsgBox "Layer '" & strLayername & "' does not exist"
End Sub
```

Unlike prior releases in which layer names were converted to uppercase, AutoCAD 2000 and higher allows the names to be mixed case. However, you can't have two layers with the same name but with a different combination of upper- and lowercase letters. For example, AutoCAD will treat "Objects" and "objects" as the same layer. This sample uses a case-insensitive string comparison of `strLayerName` and each `Layer` name to allow for capitalization differences.

A second technique is to let AutoCAD perform the search for you. This is quite a bit more efficient because AutoCAD can internally determine if the element is present, and it eliminates sending unneeded elements to your program. AutoCAD will also handle the case-insensitive name comparison for you. To use this technique you must employ the VBA error handler.

Like many AutoCAD VBA objects, the `Layers` collection uses run-time errors to signal unexpected conditions. These run-time errors, also called *exceptions,* must be handled by the calling program. If the exception isn't handled, the calling program halts and displays an error message to the user. In this case, `Layers` will raise an exception if you attempt to access an unknown `Layer` name. By detecting this exception, or more correctly the lack of the exception, your program is notified of the existence of the `Layer`. The following example gets a layer name from the user and uses AutoCAD to check for its existence:

```
Public Sub CheckForLayerByException()
    Dim strLayerName As String
    Dim objLayer As AcadLayer

    strLayerName = InputBox("Enter a Layer name to search for: ")
    If "" = strLayerName Then Exit Sub      ' exit if no name entered

    On Error Resume Next               ' handle exceptions inline
    Set objLayer = ThisDrawing.Layers(strLayerName)

    If objLayer Is Nothing Then        ' check if obj has been set
        MsgBox "Layer '" & strLayerName & "' does not exist"
    Else
        MsgBox "Layer '" & objLayer.Name & "' exists"
    End If
End Sub
```

Creating a New Layer

You can create as many layers as needed using the criterion illustrated in Table 6-1. The `Add` method is used to create a `Layer` object and to add it to the `Layers` collection.

```
Set LayerObject = LayerCollection.Add(LayerName)
```

Table 6-1. *The* `LayerName` *Property*

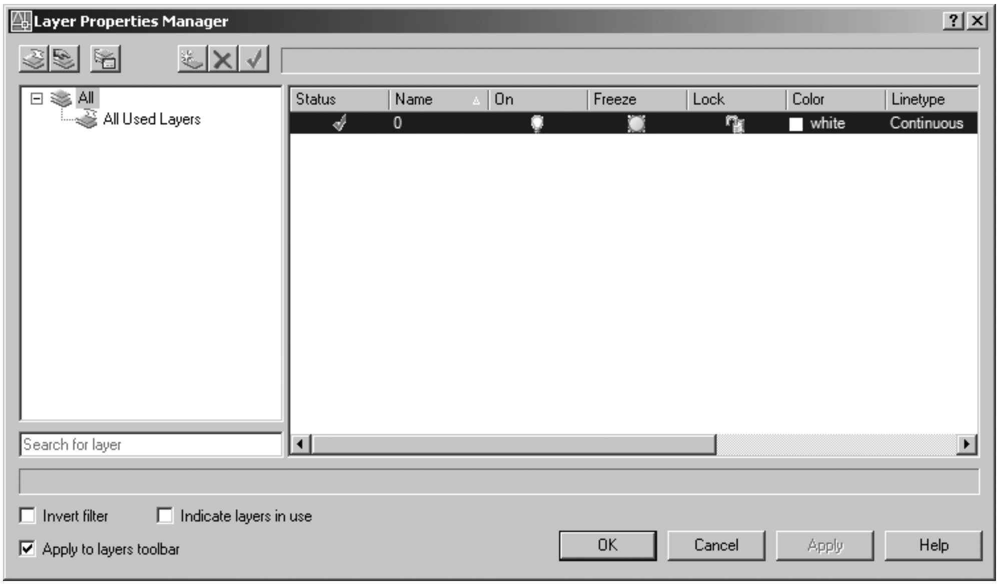
Name	Data Type	Description
`LayerName`	String	The name for the new layer. If the parameter isn't a valid layer name, an exception will be raised. If you attempt to add a new layer with the same name as an existing layer, a reference to the existing `Layer` is returned.

The following example retrieves a layer name from the user and attempts to add it to the `Layers` collection:

```
Public Sub AddLayer()
    Dim strLayerName As String
    Dim objLayer As AcadLayer

    strLayerName = InputBox("Name of Layer to add: ")
    If "" = strLayerName Then Exit Sub      ' exit if no name entered

    On Error Resume Next               ' handle exceptions inline
    'check to see if layer already exists
    Set objLayer = ThisDrawing.Layers(strLayerName)

    If objLayer Is Nothing Then
        Set objLayer = ThisDrawing.Layers.Add(strLayerName)
        If objLayer Is Nothing Then ' check if obj has been set
            MsgBox "Unable to Add '" & strLayerName & "'"
        Else
            MsgBox "Added Layer '" & objLayer.Name & "'"
        End If
    Else
        MsgBox "Layer already existed"
    End If
End Sub
```

In normal programming situations, it's usually best to define entity- or object-creation routines as functions rather than subroutines. This way, you can return the object to other functions or subroutines, making it possible to further manipulate the new object outside the function that creates it.

When a layer is first created, its properties are set to certain default values. Figure 6-1 shows what you would see in the Layer Properties Manager dialog box when a new layer called Room Shell is added with the preceding code.



Status	Name	On	Freeze	Lock	Color	Linetype
✓	0	●	●	🔓	⬜ white	Continuous

Figure 6-1. *The Layer Properties Manager dialog box*

Here you can see two layers. The first, layer 0, is created automatically by AutoCAD and may not be deleted or renamed. The second is the layer that was just created through code. The default attribute settings are shown.

We next examine how to change and retrieve the settings for some of these attributes and how to rename and delete a `Layer` object.

Making a Layer Active

When you create new entities in AutoCAD, they're placed on the current or active layer. Therefore, to draw entities on a specific layer, you must first make that layer active.

■ **Note** You can't make a layer active if it's frozen. However, you can check for this condition before you attempt to make a layer active. For more details, see the section "Setting a Layer to Be Frozen or Thawed."

The `ActiveLayer` property is a member of the `Document` object. To make a specific layer active, assign the `Layer` object to the `ActiveLayer`.

```
DocumentObject.ActiveLayer = LayerObject
```

The following code makes the layer named "Walls" the active layer for the current `Document`:

```
ThisDrawing.ActiveLayer = ThisDrawing.Layers("Walls")
```

New entities will now be placed on the “Walls” layer until another layer is made active. You can change the layer of existing entities by altering the `Layer` property of that particular entity. The `Layer` property is a string that corresponds to the name of a `Layer` object. The following example lets the user pick a drawing entity and specify a new layer name for that entity:

```
Public Sub ChangeEntityLayer()
    On Error Resume Next                  ' handle exceptions inline
    Dim objEntity As AcadEntity
    Dim varPick As Variant
    Dim strLayerName As String
    Dim objLayer As AcadLayer

    ThisDrawing.Utility.GetEntity objEntity, varPick, "Select an entity"
    If objEntity Is Nothing Then
        MsgBox "No entity was selected"
        Exit Sub  ' exit if no entity picked
    End If

    strLayerName = InputBox("Enter a new Layer name: ")
    If "" = strLayerName Then Exit Sub          ' exit if no name entered

    Set objLayer = ThisDrawing.Layers(strLayerName)
    If objLayer Is Nothing Then
        MsgBox "Layer was not recognized"
        Exit Sub   ' exit if layer not found
    End If
    objEntity.Layer = strLayerName              ' else change entity layer
End Sub
```

You can determine if a specific layer is active by comparing the string to the `ActiveLayer.Name` property:

```
If ThisDrawing.ActiveLayer.Name = "Walls" Then ...
```

This is a pretty common operation, so make it a function. You’ll also want to allow for various combinations of upper- and lowercase letters, as AutoCAD does:

```
Public Function IsLayerActive(strLayerName As String) As Boolean
    IsLayerActive = False          'assume failure
    If 0 = StrComp(ThisDrawing.ActiveLayer.Name, strLayerName, _
                   vbTextCompare) Then
        IsLayerActive = True
    End If
End Function
```

This `IsLayerActive` function checks whether a particular string corresponds to the active layer name. `True` will be returned if the layer is active, otherwise `False` will be returned. The following gets a layer name from the user and employs the new function to see if it’s active:

```
Public Sub LayerActive()
    Dim strLayerName As String
    strLayerName = InputBox("Name of the Layer to check: ")

    If IsLayerActive(strLayerName) Then
        MsgBox "'" & strLayerName & "' is active"
    Else
        MsgBox "'" & strLayerName & "' is not active"
    End If
End Sub
```

Turning a Layer On/Off

Turning a layer on or off allows you to control its visibility on the screen. This feature affects only the layer being turned on/off and not the entire drawing. This can be useful if you want to work on some aspect of a complex drawing. By making certain layers invisible, you can ensure that the drawing entities on these layers don't obscure your current work.

You can also hide a layer by freezing it. This is more suitable if you want to hide a layer for a considerable length of time and is discussed later in this chapter in the section "Setting a Layer to Be Frozen or Thawed." Turning a layer on/off is more suited to situations in which the visibility will be changed frequently.

Layers that are turned off are regenerated with the rest of the drawing but won't be displayed or plotted. When a layer that's currently off is turned back on, the entities on the layer are redrawn but don't require regeneration of the entire drawing. Setting the `LayerOn` property will turn a layer on or off.

```
LayerObject.LayerOn = blnLayerOn
```

A value of `True` for this property will turn the layer on. Conversely, a value of `False` will turn the layer off.

The following code turns off every layer except the one specified by the user:

```
Public Sub ShowOnlyLayer()
    On Error Resume Next                 ' handle exceptions inline
    Dim strLayerName As String
    Dim objLayer As AcadLayer

    strLayerName = InputBox("Enter a Layer name to show: ")
    If "" = strLayerName Then Exit Sub          ' exit if no name entered

    For Each objLayer In ThisDrawing.Layers
        objLayer.LayerOn = False           ' turn off all the layers
    Next objLayer

    Set objLayer = ThisDrawing.Layers(strLayerName)
    If objLayer Is Nothing Then
        MsgBox "Layer does not exist"
        Exit Sub   ' exit if layer not found
    End If

    objLayer.LayerOn = True                ' turn on the desired layer
```

You may sometimes want to execute a particular chunk of code depending on the value of the `LayerOn` property, for example:

```
If objLayer.LayerOn Then ... 'executes if layer is On
```

Setting a Layer to Be Frozen or Thawed

Freezing a layer makes the entities on that layer invisible. This improves the speed of display and reduces the amount of time needed to regenerate a drawing, because entities on frozen layers aren't regenerated with the rest of the drawing. Entities on frozen layers aren't considered when AutoCAD calculates the drawing extents, either. In AutoCAD 2004 and earlier versions, when a frozen layer is thawed, AutoCAD forces a regeneration of the entire drawing.

■ **Note** AutoCAD 2006 does *not* force a regeneration when the various states, such as `Freeze` and `LayerOn`, of a layer change.

The amount of performance improvement in 2004 and earlier versions depends on the number of entities on the frozen layers. Also, if your drawing is complex with a lot of detail on different layers, freezing a layer that isn't frequently used will make your drawing less cluttered and improve entity selection.

Setting the `Freeze` property to `True` will freeze a layer, whereas setting it to `False` will thaw the layer. For example, the following freezes a specific `Layer` object:

```
objLayer.Freeze = True
```

Check the value of the `Freeze` property to determine the current state of a `Layer` object:

```
If objLayer.Freeze Then ... 'executes if layer is Frozen
```

Because of the resulting drawing regeneration in 2004 and earlier versions, frequently resetting the `Freeze` property can greatly decrease performance. Thus, you should use `Freeze` only if you intend to hide a layer for an extended period. If you need to change the visibility of a layer often, `LayerOn`, as described earlier, is a better alternative.

Locking/Unlocking a Layer

You can't select or edit entities on locked layers. However, the objects are still visible as long as the layer is on and thawed, and you can still use them in object snap selection, acting as an aid to the creation and modification of entities on other layers. Although you can't edit or select entities on a locked layer, you may add new entities to it. This feature can be very useful when you create an "overlay" or for reference data. Locking a layer doesn't preclude you from altering its `on/off`, `freeze/thaw`, `plot/noplot`, `color`, `linetype`, or `lineweight` properties.

The `Lock` property holds a Boolean value, and setting it to `True` will lock a layer. A value of `False` will unlock the layer, for example:

```
objLayer.Lock = False
```

Check the value of the `Lock` property to determine the current state of a `Layer` object:

```
If objLayer.Lock Then ...   'executes if layer is locked
```

Making Layers Plottable

You can control the ability to plot layers by manipulating the `Plottable` property of the `Layer` object. Setting it to `True` enables it to be processed for plotting. Setting it to `False` treats it as though it has been frozen with respect to plot output.

```
objlayer.Plottable = False
```

Note that some layers cannot be forced to plot even if you set their `Plottable` property to `True`. Certain layers created by the ACIS or ShapeManager solid-modeling engine as well as the layer DEFPOINTS are not plottable.

Renaming a Layer

To rename a layer, simply assign a new name to the `Layer.Name` property. Take care to address the possible exceptions that may be raised: invalid name, existing name, etc.

■ **Note** You can't rename the AutoCAD-defined layer 0.

The following example renames a `Layer` object based on user input:

```
Public Sub RenameLayer()
    On Error Resume Next                  ' handle exceptions inline
    Dim strLayerName As String
    Dim objLayer As AcadLayer

    strLayerName = InputBox("Original Layer name: ")
    If "" = strLayerName Then Exit Sub         ' exit if no old name

    Set objLayer = ThisDrawing.Layers(strLayerName)
    If objLayer Is Nothing Then           ' exit if not found
        MsgBox "Layer '" & strLayerName & "' not found"
        Exit Sub
    End If

    strLayerName = InputBox("New Layer name: ")
    If "" = strLayerName Then Exit Sub         ' exit if no new name

    objLayer.Name = strLayerName               ' try and change name
    If Err Then                           ' check if it worked
        MsgBox "Unable to rename layer: " & vbCr & Err.Description
    Else
        MsgBox "Layer renamed to '" & strLayerName & "'"
    End If
End Sub
```

Deleting a Layer

The `Layer.Delete` method removes a `Layer` object from the `Layers` collection. This method takes no parameters and has no return value:

```
LayerObject.Delete
```

Within certain restrictions appearing in the following list, it's possible to delete a layer whenever you choose. A layer may not be deleted if

- It is the active layer.
- It is layer 0 (zero).
- It contains entities.
- It is an Xref-dependent layer.

If a layer contains entities in model space, any paper space layout, or any block definition, it can't be deleted. There's no definitive way to tell which entities are referenced by a specific layer short of doing an exhaustive search of each of these collections. If all entities are moved to another layer, you can then delete the layer.

Xref-dependent layers are created when an external reference file is attached and activated. Entities in the current drawing can't reside on Xref-dependent layers. Furthermore, these layers are simply duplications of the layers in the external drawing and aren't saved with the current drawing, so there's no need to delete them.

Remember, AutoCAD creates certain layers automatically for its own purposes. An example of this is the special DEFPOINTS layer, which is created during dimensioning. I discuss dimensions in Chapter 11.

The following example deletes a layer based on user input:

```
Public Sub DeleteLayer()
    On Error Resume Next                    ' handle exceptions inline
    Dim strLayerName As String
    Dim objLayer As AcadLayer

    strLayerName = InputBox("Layer name to delete: ")
    If "" = strLayerName Then Exit Sub          ' exit if no old name

    Set objLayer = ThisDrawing.Layers(strLayerName)
    If objLayer Is Nothing Then             ' exit if not found
        MsgBox "Layer '" & strLayerName & "' not found"
        Exit Sub
    End If

    objLayer.Delete                         ' try to delete it
    If Err Then                             ' check if it worked
        MsgBox "Unable to delete layer: " & vbCr & Err.Description
    Else
        MsgBox "Layer '" & strLayerName & "' deleted"
    End If
End Sub
```

■Tip Whenever you delete a collection member such as a layer, linetype, dimstyle, or text-style object, you should consider executing a `Purge` method against the drawing object to clean up the database.

Getting a Layer's Handle

AutoCAD assigns every object a unique handle or ID that remains constant for as long as the object exists. You access object handles are using the `Handle` property:

```
Dim objLayer As AcadLayer
Dim strLayerHandle As String

Set objLayer = ThisDrawing.Layers("0")
strLayerHandle = objLayer.Handle
```

Handles are used extensively when working with extended entity data. Extended entity data, or Xdata, is nongraphical information that can be attached to objects by application programs.

Layer Colors

Each layer has a `Color` property that provides the color for all entities drawn on the layer whose `Color` property is set to `ByLayer`. Unless an entity is set to a specific color, it will take on the color of the layer where it is drawn. This is very efficient—changing the layer color automatically changes the color of all entities whose `Color` property is set to `ByLayer`.

By default, a new layer's color will be white or black depending upon the drawing background color, and is represented by the number 7. You can change the layer color at any time, and likewise you can assign individual entities their own color at any time.

A layer may take any one of 257 different color values; nine of these have associated AutoCAD VBA constants, which are listed in Table 6-2. You set the `Color` property by specifying either the constant or the AutoCAD color index number, ranging between 0 and 256. The following statements are equivalent:

```
objLayer.Color = acRed
```

or

```
objLayer.Color = 1
```

Table 6-2. *AutoCAD-Defined Color Constants*

Constant	Color Index	Color
`acByBlock`	0	`ByBlock`
`acRed`	1	Red
`acYellow`	2	Yellow
`acGreen`	3	Green
`acCyan`	4	Cyan
`acBlue`	5	Blue
`acMagenta`	6	Magenta
`acWhite`	7	White/Black (depending on the screen background color)
`acByLayer`	256	`ByLayer`

■ **Caution** You shouldn't set the `Color` property of a layer to `ByBlock` or `ByLayer`. The results are unpredictable, and furthermore it doesn't make sense to do so.

When you retrieve the value of the `Color` property, it's returned as an integer:

```
intColor = objLayer.Color
```

Therefore, if the color were red, for example, the value of the variable `intColor` would be equal to 1.

AutoCAD 2004 introduces support for RGB-based `TrueColor` properties as well as Pantone color palettes and color books. `Layer` objects therefore expose a `TrueColor` property, which employs a new enumeration class called `AcadAcCmColor`.

Layer Linetypes

Much like the color, each layer has a `Linetype` property that provides the default linetype of all entities drawn on the layer whose `Linetype` property is set to `ByLayer`. Unless an entity is set to a specific linetype, it will take on the linetype of the layer where it is drawn. Again, this is very efficient—changing the layer linetype automatically changes all entities whose `Linetype` property is set to `ByLayer`.

By default, a new layer's linetype will be `Continuous`, which is a solid line. You can change the layer's linetype at any time, and in addition you can assign individual entities their own linetypes at any time. The `Layer.Linetype` property is a string that you can access as follows:

```
Public Sub Layer0Linetype()
Dim objLayer As AcadLayer
Dim strLayerLinetype As String

    Set objLayer = ThisDrawing.Layers("0")
    objLayer.Linetype = "Continuous"
    strLayerLinetype = objLayer.Linetype
```

Linetypes are generally loaded from files containing linetype definitions, which is demonstrated later in the section "Loading a Linetype."

Layer Lineweights

Lineweights control the visible and plottable widths of entities. You can apply them to individual objects, groups, block insertions, Xrefs, layers, and more. You can toggle their display on or off, and you can scale them to suit viewport scales in paper space.

Lineweights themselves are somewhat of a unique animal. They're a list of varying widths, from thinnest to thickest; however, they're actually constructed from a mapping of an index list to a corresponding collection of explicit metric (millimeter) values. The index values are enumerations with integer values beginning with `acLnWt`, but the index simply points to an explicit value, such as 0.04 millimeters. This works very well, unless the default values are edited. So although `acLnWt040` by default refers to a 0.04-millimeter thickness, it can't be guaranteed to be so in all cases. Refer to Appendix B for more information on lineweight values.

The initial value for this property is `acLnWtByBlock`.

Lineweight values consist of standard enumeration including `ByLayer` (`acLnWtByLayer`), `ByBlock` (`acLnWtByBLock`), and `DEFAULT` (`acLnWtByLwDefault`). The `DEFAULT` value is set by the `LWDEFAULT` system variable and defaults to a value of 0.01 in. or 0.25 mm. All new objects and layers have a default setting of `DEFAULT`. The lineweight value of 0 plots at the thinnest lineweight available on the specified plotting device and is displayed at 1 pixel wide in model space.

```
Public Sub GetLwt()
    Dim objLayer As AcadLayer
    Dim lwtLweight As Integer
    Set objLayer = ThisDrawing.ActiveLayer
    lwtLweight = objLayer.Lineweight
    Debug.Print "Lineweight is " & lwtLweight
End Sub
```

Linetypes

As you would expect, linetypes characterize the different appearances that a line within a drawing may adopt. Linetypes are arranged in alternate patterns of dashes, dots, etc. to represent something specific to the user, such as process pipelines or railroad tracks, as shown in Figure 6-2.

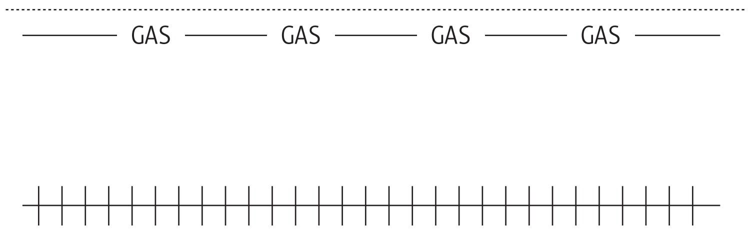


Figure 6-2. *Linetype examples*

You have a great deal of flexibility when it comes to linetypes—in fact, your imagination is virtually the only limit.

Although it's possible to create new `Linetype` objects through VBA, it isn't possible to control the line pattern directly through code. However, VBA provides the `Load` method to load existing linetypes into your drawing. I discuss most of the methods and properties that AutoCAD provides for working with the `Linetypes` collection and `Linetype` objects.

You'll learn how to

- Access the `Linetypes` collection and `Linetype` objects.
- Check for the existence of a specific linetype.
- Load linetypes and make a specific linetype active.
- Rename and delete a linetype.
- Set or return a linetype's scale properties and description.

Accessing Linetypes

Much like layers, AutoCAD has a `Linetypes` collection that contains all of the `Linetype` objects in the drawing. You can create as many `Linetype` objects as you want by adding to the `Linetypes` collection, but they'll be created with the default properties only.

The `Linetypes` collection is accessed via a `Document` object. In the following code, `ThisDrawing` is used as the active document:

```
Dim objLinetypes As AcadLineTypes
Set objLinetypes = ThisDrawing.Linetypes
```

To set a reference to an existing `Linetype` object, use the `Item` method of the `Linetypes` collection as follows:

```
Dim objLinetype As AcadLineType
Set objLinetype = objLinetypes.Item(2)
Set objLinetype = objLinetypes.Item("Dashed")
```

The parameter of this method is either an integer representing the position of the desired `Linetype` object within the `Linetypes` collection or a string representing the name of the desired `Linetype` object. If you use an index number, it must be between 0 and `Linetypes.Count` minus 1.

Like other AutoCAD collections, `Item` is the default method for `Linetypes`. This means that the method name may be omitted, and the parameter passed straight to the `Linetypes` reference. Some programmers prefer this, as it's simpler to type and read. The following code does the same thing as the prior example using the default method to specify the `Linetype` object:

```
Dim objLinetype As AcadLinetype
Set objLinetype = objLinetypes(2)
Set objLinetype = objLinetypes("Dashed")
```

However, I recommended in the "Accessing Layers" section that you avoid using default methods and instead try to use explicit properties and methods. Microsoft is committed to explicit coding in programming technologies. For instance, .NET doesn't support default

Checking for Existing Linetypes

In some situations your program must determine if an element is present in the collection. One way to do this is to search for the element while iterating the collection, as described earlier in the chapter. The following example gets a linetype name from the user and then checks for its existence by iterating the `Linetypes` collection:

```
Public Sub CheckForLinetypeByIteration()
    Dim objLinetype As AcadLineType
    Dim strLinetypeName As String

    strLinetypeName = InputBox("Enter a Linetype name to search for: ")
    If "" = strLinetypeName Then Exit Sub         ' exit if no name entered
    For Each objLinetype In ThisDrawing.Linetypes
        If 0 = StrComp(objLinetype.Name, strLinetypeName, vbTextCompare) Then
            MsgBox "Linetype '" & strLinetypeName & "' exists"
            Exit Sub                           ' exit after finding linetype
        End If
    Next objLinetype

    MsgBox "Linetype '" & strLinetypeName & "' does not exist"
End Sub
```

Unlike prior releases, in which linetype names were converted to uppercase, AutoCAD 2000 and later allow the names to be a mix of upper- and lowercase, although “Linetype1” and “LINETYPE1” will be treated as the same name. This sample uses a case-insensitive string comparison of `strLinetypeName` and each `Linetype` name to allow for capitalization differences.

A second technique is to let AutoCAD perform the search for you. This is quite a bit more efficient because AutoCAD can internally determine if the element is present, and it eliminates sending unneeded elements to your program. AutoCAD will also handle the case-insensitive name comparison for you. To use this technique you must employ the VBA error handler.

Like many AutoCAD ActiveX objects, the `Linetypes` collection uses run-time errors or exceptions to signal unexpected conditions. If an exception isn’t handled, the calling program halts and displays an error message to the user. In this case, `Linetypes` will raise an exception if you attempt to access an unknown `Linetype` name. By detecting this exception, or more correctly the lack of the exception, your program is notified of the existence of the `Linetype`. The following example gets a linetype name from the user and employs AutoCAD to check for its existence:

```
Public Sub CheckForLinetypeByException()
    Dim strLinetypeName As String
    Dim objLinetype As AcadLineType

    strLinetypeName = InputBox("Enter a Linetype name to search for: ")
    If "" = strLinetypeName Then Exit Sub      ' exit if no name entered
    On Error Resume Next              ' handle exceptions inline
    Set objLinetype = ThisDrawing.Linetypes(strLinetypeName)

    If objLinetype Is Nothing Then     ' check if obj has been set
        MsgBox "Linetype '" & strLinetypeName & "' does not exist"
    Else
        MsgBox "Linetype '" & objLinetype.Name & "' exists"
    End If
End Sub
```

Loading a Linetype

Linetype definitions are stored in linetype library files, which are external to drawing files and have the extension `.lin`. Linetypes are loaded by name from library files into similarly named `Linetype` objects in the drawing. Once loaded into a drawing, a `Linetype` object has no more connection to the library file. I don't cover the creation and definition of linetypes here, because no methods exist for customizing linetypes programmatically, but you can take a look at the AutoCAD Customization Guide in the online help files for details on creating custom linetypes. It's interesting to note that you can create custom linetypes by writing a linetype definition file programmatically and then loading it.

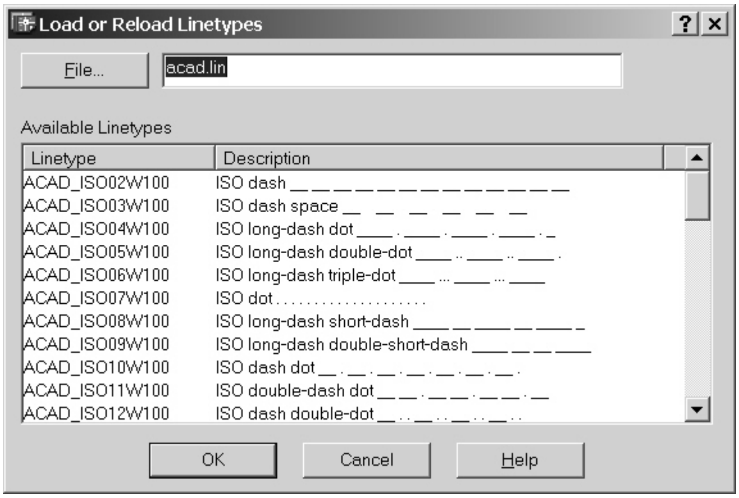
You can load a `Linetype` object into your drawing using the `Load` method shown in the following code and detailed in Table 6-3.

```
Set LinetypeObject = LinetypesCollection.Load(LinetypeName, _
                       LinetypeFilename)
```

Table 6-3. *Linetype* `Load` *Parameter Specifications*

Name	Data Type	Description
`LineTypeName`	String	The name of the linetype
`LinetypeFilename`	String	The path and file name of the linetype library file

The AutoCAD default `Linetype` object definition file for standard linetypes is `acad.lin`. You can view the contents of this via the Linetype dialog box (click Format ➤ Linetype to bring it up) by clicking Load and browsing to the file (see Figure 6-3). You can also open a linetype definition file in any ASCII text editor such as Windows Notepad.



acad.lin

Available Linetypes

Linetype	Description
ACAD_ISO02W100	ISO dash __ __ __ __ __ __ __ __ __ __ __ __ __
ACAD_ISO03W100	ISO dash space __ __ __ __ __ __ __
ACAD_ISO04W100	ISO long-dash dot ____ . ____ . ____ . ____ . _
ACAD_ISO05W100	ISO long-dash double-dot ____ .. ____ .. ____ .
ACAD_ISO06W100	ISO long-dash triple-dot ____ ... ____ ... ____
ACAD_ISO07W100	ISO dot . . . .
ACAD_ISO08W100	ISO long-dash short-dash ____ __ ____ __ ____ _
ACAD_ISO09W100	ISO long-dash double-short-dash ____ __ __ ____
ACAD_ISO10W100	ISO dash dot __ . __ . __ . __ . __ . __ .
ACAD_ISO11W100	ISO double-dash dot __ __ . __ __ . __ __ . __
ACAD_ISO12W100	ISO dash double-dot __ . . __ . . __ . . __ . .

Figure 6-3. *The Linetype dialog box*

Once you determine the linetype library and the name of a specific linetype you'd like to load, you can use the following code to load it:

```
Public Sub LoadLinetype()
    Dim strLinetypeName As String
    Dim objLinetype As AcadLineType

    strLinetypeName = InputBox("Enter a Linetype name" & _
                           " to load from ACAD.LIN: ")
    If "" = strLinetypeName Then Exit Sub      ' exit if no name entered

    On Error Resume Next               ' handle exceptions inline
    ThisDrawing.Linetypes.Load strLinetypeName, "acad.lin"

    If Err Then                        ' check if err was thrown
        MsgBox "Error loading '" & strLinetypeName & "'" & vbCr & _
                Err.Description
    Else
        MsgBox "Loaded Linetype '" & strLinetypeName & "'"
    End If
End Sub
```

■ **Note** Depending on the setting of the drawing's MEASUREMENT system variable, the linetype library could be `acadiso.lin`.

Making a Linetype Active

The `Linetype` property for new entities is determined by the drawing linetype setting for the drawing. By default, this is set so that new entities are displayed using the linetype assigned to the layer that they're drawn on. By changing this setting, you may assign specific linetypes directly to entities rather than the layer setting, which normally takes precedence. You can change the linetype of an existing entity at any time by altering its `Linetype` property.

■**Note** You can't make a linetype active if it's Xref-dependent.

You can use two special linetypes: `ByLayer` and `ByBlock`. The `ByLayer` option was described earlier in this chapter. This is the most commonly used method when creating drawing entities. If you use the `ByBlock` option, all new entities that you create will use the linetype associated with the block containing the entity. `Block` objects are considered in more detail in Chapter 13.

The `ActiveLinetype` property is a member of the `Document` object. To make a specific linetype active, assign the `Linetype` object to the `ActiveLinetype` property:

```
DocumentObject.ActiveLinetype = LinetypeObject
```

The following example makes the linetype named TRACKS the active linetype for the current `Document`:

```
ThisDrawing.ActiveLinetype = ThisDrawing.Linetypes("TRACKS")
```

Now, all the new entities will be created with this linetype until another linetype is activated. You can change the linetype of existing entities by altering the `Linetype` property of that particular entity. The `Linetype` property is a string that corresponds to the name of a `Linetype` object. The following example lets the user pick a drawing entity and specify a new linetype name for that entity:

```
Public Sub ChangeEntityLinetype()
    On Error Resume Next                      ' handle exceptions inline
    Dim objEntity As AcadEntity
    Dim varPick As Variant
    Dim strLinetypeName As String
    Dim objLinetype As AcadLineType

    ThisDrawing.Utility.GetEntity objEntity, varPick, "Select an entity"
    If objEntity Is Nothing Then Exit Sub     ' exit if no entity picked

    strLinetypeName = InputBox("Enter a new Linetype name: ")
    If "" = strLinetypeName Then Exit Sub      ' exit if no name entered
    Set objLinetype = ThisDrawing.Linetypes(strLinetypeName)
    If objLinetype Is Nothing Then
        MsgBox "Linetype is not loaded"
        Exit Sub   ' exit if linetype not found
    End If

    objEntity.Linetype = strLinetypeName        ' else change entity layer
End Sub
```

Renaming a Linetype

To rename a linetype, simply assign a new name to the `Linetype.Name` property. Take care to address the possible exceptions that may be thrown up: invalid name, existing name, etc.

■ **Note** You may not rename or delete `ByLayer`, `ByBlock`, `Continuous`, and Xref-dependent linetypes.

The following example renames a `Linetype` object based on user input:

```
Sub RenameLinetype()
    On Error Resume Next                    ' handle exceptions inline
    On Error Resume Next                    ' handle exceptions inline
    Dim strLinetypeName As String
    Dim objLinetype As AcadLineType

    strLinetypeName = InputBox("Original Linetype name: ")
    If "" = strLinetypeName Then Exit Sub          ' exit if no old name

    Set objLinetype = ThisDrawing.Linetypes(strLinetypeName)
    If objLinetype Is Nothing Then          ' exit if not found
        MsgBox "Linetype '" & strLinetypeName & "' not found"
        Exit Sub
    End If

    strLinetypeName = InputBox("New Linetype name: ")
    If "" = strLinetypeName Then Exit Sub          ' exit if no new name

    objLinetype.Name = strLinetypeName             ' try and change name
    If Err Then                             ' check if it worked
        MsgBox "Unable to rename Linetype: " & vbCr & Err.Description
    Else
        MsgBox "Linetype renamed to '" & strLinetypeName & "'"
    End If
End Sub
```

Deleting a Linetype

The `Linetype.Delete` method removes a `Linetype` object from the `Linetypes` collection. This method takes no parameters and has no return value:

```
LinetypeObject.Delete
```

It's possible to delete a linetype practically whenever you choose; however, certain linetypes may not be deleted. A linetype may not be deleted if

- It is the active linetype.
- It is a `ByLayer`, `ByBlock`, or `Continuous` linetype.
- It is an Xref-dependent linetype.

Just as with layers, if a linetype is used by entities in model space, any paper space layout or any block definition, it can't be deleted. There's no definitive way to tell which entities are using a specific linetype short of doing an exhaustive search of each of these collections. If all entities using the linetype are assigned to another, you can then delete the linetype.

Xref-dependent linetypes are created when an external reference file is attached and activated. Entities in the current drawing can't use Xref-dependent linetypes. Furthermore, these linetypes are simply duplications of the linetypes in the external drawing and aren't saved with the current drawing, so you don't need to delete them.

The following example deletes a linetype based on user input:

```
Public Sub DeleteLinetype()
    On Error Resume Next                  ' handle exceptions inline
    Dim strLinetypeName As String
    Dim objLinetype As AcadLineType

    strLinetypeName = InputBox("Linetype name to delete: ")
    If "" = strLinetypeName Then Exit Sub         ' exit if no old name

    Set objLinetype = ThisDrawing.Linetypes(strLinetypeName)
    If objLinetype Is Nothing Then           ' exit if not found
        MsgBox "Linetype '" & strLinetypeName & "' not found"
        Exit Sub
    End If

    objLinetype.Delete                       ' try to delete it
    If Err Then                           ' check if it worked
        MsgBox "Unable to delete linetype: " & vbCr & Err.Description
    Else
        MsgBox "Linetype '" & strLinetypeName & "' deleted"
    End If
End Sub
```

Getting a Linetype's Handle

AutoCAD assigns to every object a unique handle or ID that remains constant for as long as the object exists. The handle of an object takes the form of a string and may be accessed via the read-only `Handle` property. For example:

```
Dim objLinetype As AcadLinetype
Dim strLinetypeHandle As String

Set objLinetype = ThisDrawing.Linetypes("Center")
strLinetypeHandle = objLinetype.Handle
```

You'll use handles extensively when working with extended entity data. Extended entity data is specific information that can be attached to an object or entity by an external application.

Changing a Linetype's Description

AutoCAD allows you to read, add, or modify a linetype's description using the `Description` property of the `Linetype` object. This property is a string that typically describes the linetype, often with a simplified symbolized version of the actual linetype. The following code snippet shows the `Description` property being set and retrieved:

```
Dim objLineType As AcadLineType
Dim strLineTypeDescription As String

objLineType.Description = "Linetype Description: -.-.-."
strLineTypeDescription = objLineType.Description
```

The following example changes a `Linetype` description based on user input:

```
Public Sub DescribeLinetype()
    On Error Resume Next                        ' handle exceptions inline
    Dim strLinetypeName As String
    Dim strLinetypeDescription As String
    Dim objLinetype As AcadLineType

    strLinetypeName = InputBox("Enter the Linetype name: ")
    If "" = strLinetypeName Then Exit Sub             ' exit if no old name

    Set objLinetype = ThisDrawing.Linetypes(strLinetypeName)
    If objLinetype Is Nothing Then              ' exit if not found
        MsgBox "Linetype '" & strLinetypeName & "' not found"
        Exit Sub
    End If

    strLinetypeDescription = InputBox("Enter the Linetype description: ")
    If "" = strLinetypeDescription Then Exit Sub      ' exit if no new name
    objLinetype.Description = strLinetypeDescription    ' try and change name
    If Err Then                                        ' check if it worked
        MsgBox "Unable to alter Linetype: " & vbCr & Err.Description
    Else
        MsgBox "Linetype '" & strLinetypeName & "' description changed"
    End If
```

Scaling Linetypes

The scale of a linetype determines how many times the pattern of a linetype is repeated over a given distance: the smaller the scale, the finer the pattern. You need to adjust the scale of linetypes to suitable values so that your drawings will display and plot sensibly.

Here I cover two types of scaling factors for linetypes: the global linetype scale or `LTSCALE` and the individual linetype scale or `CELTSCALE`. `LTSCALE` affects all objects within a drawing, and when it is changed, the lines of all existing objects are updated. `CELTSCALE`, on the other hand, affects only the objects that are created after it has been set. If `CELTSCALE` is 2.0 when you create a line object, that object will have a `LinetypeScale` property of 2.0. Note that you can not assume a link between the current setting of `CELTSCALE` and the `LinetypeScale` property of any objects in the drawing. The scaling of a linetype depends on a combination of `LTSCALE` and `each object's LinetypeScale` property, as shown in Figure 6-4.

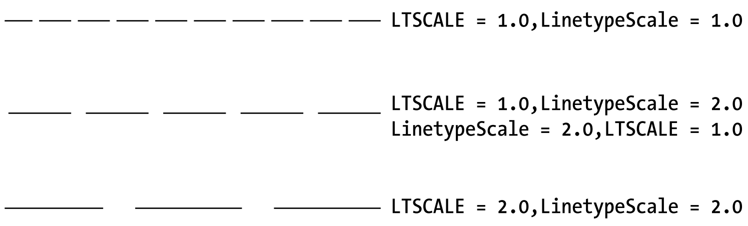

LTSCALE = 1.0,LinetypeScale = 1.0
LTSCALE = 1.0,LinetypeScale = 2.0
LinetypeScale = 2.0,LTSCALE = 1.0
LTSCALE = 2.0,LinetypeScale = 2.0

Figure 6-4. *An example of linetype scaling*

Because you're likely to assign the linetype scale a number such as 0.5 or 2.0, you might be tempted to use a single precision floating-point variable when reading `LTSCALE` or `CELTSCALE` for a drawing. However, you might well find that the number returned isn't so simple and that you need more precision. Be proactive and use a double-precision variable type. Additionally, viewport scaling affects global and individual linetype scaling, as can plot-style configurations.

Global Scale

The default for the global linetype `LTSCALE` system variable is 1.0, which means one linetype unit per drawing unit. A `LTSCALE` factor of 2.0 would mean two linetype units per drawing unit. Changing this value will change the linetype scale for all entities in the drawing. System variables are retrieved and set using the `GetVariable` and `SetVariable` methods of the `Document` object.

The following example demonstrates setting and reading the `LTSCALE` system variable, which holds a real number:

```
Dim dblNewLTScale As Double
ThisDrawing.SetVariable "LTSCALE", 2#
dblNewLTScale = ThisDrawing.GetVariable("LTSCALE")
```

You should also note that the global PSLTSCALE system variable appears to override LTSCALE. This variable is a simple toggle, and can be either 0 or 1. If PSLTSCALE is set to 1, AutoCAD automatically scales the linetypes in proportion to each viewport's zoom scale, ensuring that a layout with viewports with differing scale factors will display its linetypes at a consistent scale.

The following example demonstrates setting the and reading the PSLTSCALE system variable, which holds the integers 0 and 1:

```
ThisDrawing.SetVariable "PSLTSCALE", 0
ThisDrawing.Regen acAllViewports
MsgBox "PSLTSCALE has been set to 0"
ThisDrawing.SetVariable "PSLTSCALE", 1
ThisDrawing.Regen acAllViewports
MsgBox "PSLTSCALE has been set to 1"
```

Individual Scale

The `CELTSCALE` system variable is set in a similar way to `LTSCALE` and also holds a real number.

```
Dim dblNewCELTScale As Double
ThisDrawing.SetVariable "CELTSCALE", 2#
dblNewCELTScale = ThisDrawing.GetVariable("CELTSCALE")
```

Summary

In this chapter, I showed you the advantages of effective use of layers and linetypes. I discussed and gave code examples for many of the methods and properties of the `Layers` collection and `Layer` object and the `Linetypes` collection and `Linetype` object. For complete details of each of these AutoCAD objects, please refer to Appendix A.

CHAPTER 7

User Interaction and the Utility Object

To program truly powerful AutoCAD applications and macros, you need to be able to interact with the user. The AutoCAD Automation model provides the `Utility` and `SelectionSet` objects to do just that. Chapter 12 discusses the `SelectionSet` object. This chapter discusses the `Utility` object, which contains a number of methods for obtaining user input via the AutoCAD command line and graphics screen, such as picking points, entering distances and angles, and picking a single entity.

In addition to these AutoCAD objects, you can, of course, interface with the user through standard VBA functions and objects. The VBA `InputBox` and `MsgBox` functions allow simple user input, while VBA `UserForms` and graphical controls allow a level of sophistication limited only by your imagination.

This chapter covers the following:

- Controlling user keyword input
- Using `GetXXX` methods for all types of input
- Using other interactive or interesting methods of the `Utility` object

Interface Methods

Interface methods present and retrieve information from the user. This section discusses the various methods made available through AutoCAD's `Utility` object.

Input Methods and Dialogs

All the AutoCAD input methods require that the user interact with the AutoCAD drawing or command window. To use these methods from a VBA `UserForm`, either hide the form before you call the input methods or set the `UserForm` to modeless operation. If you don't, an error will occur. This is because modal forms maintain the application focus while they're visible. As long as a modal VBA form is visible, there's no way for the user to get to the AutoCAD drawing or command window. `UserForms` can be modal or modeless only with AutoCAD 2002 and later versions.

The following example places the command button cmdGetReal on a UserForm, which is then hidden while the GetReal user input method executes:

```
Private Sub cmdGetReal_Click()
Dim dblInput As Double

    Me.Hide
    dblInput = ThisDrawing.Utility.GetReal("Enter a real value: ")
    Me.Show

End Sub
```

■Note Remember to Show the form after you Hide it for the GetReal user input method, or your program will exit without completing any of the form's remaining code.

The code in this chapter runs from the Macros dialog box. To work from a UserForm, you need to modify each sample. For example, change code from this:

```
Public Sub TestUserInput()
Dim strInput As String

    With ThisDrawing.Utility
        .InitializeUserInput 1, "Line Arc Circle laSt"
        strInput = .GetKeyword(vbCr & "Option [Line/Arc/Circle/laSt]: ")
        .Prompt "You selected '" & strInput & "'"
    End With
End Sub
```

to this:

```
Private Sub CommandButton1_Click()
Dim strInput As String

    Me.Hide
    With ThisDrawing.Utility
        .InitializeUserInput 1, "Line Arc Circle laSt"
        strInput = .GetKeyword(vbCr & "Option [Line/Arc/Circle/laSt]: ")
        MsgBox "You selected '" & strInput & "'"
    End With
    Me.Show

End Sub
```

Programs written using the full version of Visual Basic 6 or VBA 6.3 (AutoCAD 2002 and later) are not limited to modal dialog forms. But sometimes it may make sense to hide the form while interacting with the user at the AutoCAD command prompt. Try to think like your users when you design the interaction process and your forms' behavior.

The Prompt Method

The Prompt method displays messages on the AutoCAD command line. It returns nothing.

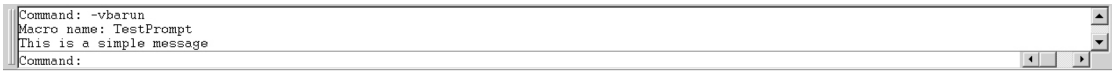
```
UtilityObject.Prompt Message
```

The Message parameter is a string that contains the message to display at the command prompt.

■Note Remember to include carriage returns or linefeeds in the message string, or your message will just be appended to whatever text is on the command prompt.

The following code displays a simple message at the command prompt, as shown in Figure 7-1:

```
Public Sub TestPrompt()
    ThisDrawing.Utility.Prompt vbCrLf & "This is a simple message"
End Sub
```



```
Command: -vbarun
Macro name: TestPrompt
This is a simple message
Command:
```

Figure 7-1. *Displaying a message at the command prompt*

The remaining examples in this chapter extensively use the Prompt method to give the user feedback.

The InitializeUserInput Method

The InitializeUserInput method establishes which keywords the program will accept during command line input. It also sets input criteria such as whether null values are allowed. It affects the very next call to an input function, after which the keywords and other input criteria are cleared.

The InitializeUserInput method has this syntax:

```
UtilityObject.InitializeUserInput OptionBits[, KeywordList]
```

Table 7-1 explains this method's parameters.

Table 7-1. *The* InitializeUserInput *Method's Parameters*

Name	Type	Description
OptionBits	Long	Restricts what the user can input, as Table 7-2 explains
KeywordList	String	Displays a list of the keywords delimited by spaces

Table 7-2 explains valid `OptionBits` options.

Table 7-2. *The Acceptable Keyword List Values*

Bit Value	Description
0 or not set	No control conditions are applied.
1	Prevents null input such as pressing Enter or entering a space.
2	Prevents the user from entering a zero value.
4	Prevents the user from entering a negative value.
8	Lets the user enter a point outside the current drawing limits. AutoCAD won't check the drawing limits even if the `LIMCHECK` system variable is on. `LIMCHECK` controls whether entities may be created outside the drawing limits.
16	Not currently used.
32	For methods that allow input via the graphics screen, AutoCAD uses a dashed rather than solid line when it displays a rubber-band line or box. AutoCAD ignores this bit if the `POPUPS` system variable is set to zero. `POPUPS` indicates whether the currently configured display driver supports dialog boxes, the menu bar, and icon menus.
64	For the `GetDistance` method, ignores the Z-coordinate of 3-D points, thus returning a 2-D distance.
128	Accepts any user input.

Keywords

Command line keywords give the user various options. For instance, if you use the AutoCAD `Draw` toolbar's rectangle command, you may either pick a point or use one of the five keywords to choose another command:

```
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:
```

The `KeywordList` parameter specifies a set of keywords available at the input prompt. It is a string that contains the keywords, each separated by a space. The capital letter in each keyword is the shortcut key the user can type.

It is common practice to give the user a list of valid keywords when prompting for input. You must supply this list in your input prompt, since AutoCAD does not indicate the active keywords in any other way. This example demonstrates these ideas by getting a keyword from the user and then printing it on the command line:

```
Public Sub TestUserInput()
Dim strInput As String

    With ThisDrawing.Utility
        .InitializeUserInput 1, "Line Arc Circle laSt"
        strInput = .GetKeyword(vbCr & "Option [Line/Arc/Circle/laSt]: ")
        .Prompt vbCr & "You selected '" & strInput & "'"
    End With
End Sub
```

In this example, you can type **L** for a line, **A** for an arc, **C** for a circle, or **S** for the last command. AutoCAD rejects any other single letter, whether in a keyword or otherwise. However, you can type a partial keyword, or even the entire keyword, if you want. When you type partial keywords, you must supply enough of the keyword to uniquely distinguish it from all other keywords. The partial entry must also include at least up to the capital letter in the keyword. For example, AutoCAD rejects typing **la**, but typing **las** correctly returns laSt.

Regardless of how the user types a keyword, AutoCAD returns a string that exactly matches the capitalization used when you initialized the keyword. In the previous example, typing **S**, **LAS**, **Las**, or any other variation returns laSt.

You can use keywords in conjunction with these `Utility` input methods: `GetKeyword`, `GetInteger`, `GetReal`, `GetDistance`, `GetAngle`, `GetOrientation`, `GetPoint`, and `GetCorner`. Each of these methods can obtain keywords set using the `InitializeUserInput` method. See "The `GetInput` Method" later in this chapter to learn how to retrieve keywords from them.

■**Note** AutoCAD 2006 provides a new feature known as *Dynamic Input*. This feature provides a command line–like interface near the cursor to help you keep your focus in the drawing area. Figure 7-2 illustrates this new feature.

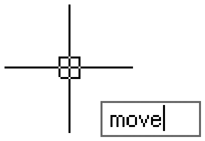


Figure 7-2. *Using Dynamic Input*

The `GetXXX` Methods

The `GetXXX` methods get specific types of data from the user. These methods make AutoCAD pause until the user supplies a value at the command prompt or picks a point in the drawing window. If the user supplies the wrong type of data, such as typing a string when a number is needed, AutoCAD displays a message at the command prompt that tells them to reenter the data.

The `GetKeyword` Method

The `GetKeyword` method gets a command line option from the user. You must call `InitializeUserInput` to establish the list of keywords before you use this method. It returns the keyword the user entered exactly as the keyword list specified it. If you want, you can include a prompt to display on the command line while the function waits for user input. This method has the following syntax:

```
strUserKeyWordInput = Object.GetKeyword([Prompt])
```

This method's `Prompt` parameter is a string that contains the prompt to display on the command line.

If the user tries to enter a string that's not in the keyword list, AutoCAD displays the error message `Invalid option keyword` in the command window. AutoCAD then tries to get valid user input by redisplaying the prompt if you specified one or by displaying a blank command line if you didn't. If you allow null input, the user can press Enter to return an empty string.

This example asks the user for an option and then starts the specified command, using the `SendCommand` method outlined in Appendix A:

```
Sub TestGetKeyword()
Dim strInput As String

    With ThisDrawing.Utility
        .InitializeUserInput 0, "Line Arc Circle"
        strInput = .GetKeyword(vbCr & "Command [Line/Arc/Circle]: ")
    End With

    Select Case strInput

        Case "Line": ThisDrawing.SendCommand "_Line" & vbCr
        Case "Arc": ThisDrawing.SendCommand "_Arc" & vbCr
        Case "Circle": ThisDrawing.SendCommand "_Circle" & vbCr
        Case Else:  MsgBox "You pressed Enter."

    End Select

End Sub
```

The `GetString` Method

The `GetString` method gets string values from the user. AutoCAD pauses until the user enters a value. This method has the following syntax:

```
dblUserStringInput = UtilityObject.GetString(HasSpaces[ ,Prompt])
```

Table 7-3 lists this method's parameters.

Table 7-3. *The* `GetString` *Method's Parameters*

Name	Type	Description
`HasSpaces`	Boolean	Specifies whether the user input may contain spaces. If set to `True`, spaces are valid, and the user must press Enter to terminate the input. If set to `False`, spaces are not allowed, and the user can terminate the input either by pressing Enter or by entering a space.
`Prompt`	String	An optional parameter used to display a prompt for input.

This example gets a string from the user, including spaces, and then displays it:

```
Public Sub TestGetString()
Dim strInput As String

    With ThisDrawing.Utility
        strInput = .GetString(True, vbCr & "Enter a string: ")
        .Prompt vbCr & "You entered '" & strInput & "' "
    End With
End Sub
```

This method lets the user enter up to 132 characters. Entering more than 132 characters generates the error `Method 'GetString' of object 'IAcadUtility' failed`, which is unfortunately the same description reported when a user issues a Cancel or Esc. If you need to distinguish between the overflow and the Cancel for this method, consider using the unique exception number instead of the description.

The `GetInteger` Method

The `GetInteger` method gets an integer from the user. AutoCAD waits for the user to input an integer and returns the entered value. This method has the following syntax:

```
intUserIntegerInput = UtilityObject.GetInteger([Prompt])
```

This method has one parameter, `Prompt`, a string. Optionally use it to specify a prompt for input.

The user may enter either an integer in the range –32,768 to +32,767 or a keyword (see "The `GetInput` Method" later in the chapter for more information). If the user tries to enter any other value, AutoCAD returns the error message `Requires an integer value or option keyword` and asks the user to enter another value. Here's an example of `GetInteger`:

```
Public Sub TestGetInteger()
Dim intInput As Integer

    With ThisDrawing.Utility
        intInput = .GetInteger(vbCr & "Enter an integer: ")
        .Prompt vbCr & "You entered " & intInput
    End With
End Sub
```

This method throws exceptions for null input, keyword entry, and canceled input.

The `GetReal` Method

The `GetReal` method is similar to `GetInteger` but gets floating-point numbers. It returns a value of data type Double.

```
dblUserRealInput = UtilityObject.GetReal([Prompt])
```

This method's `Prompt` parameter, a string, is optional. Use it to specify a prompt for input.

This method accepts any real (double-precision floating-point) value or any previously set keyword. For more about these keywords, see “The `GetInput` Method” later in this chapter. If the user enters any other value, AutoCAD returns the error message `Requires numeric value` and asks the user to enter another value. This example retrieves and displays a real value from the user:

```
Public Sub TestGetReal()
Dim dblInput As Double

    With ThisDrawing.Utility
        dblInput = .GetReal(vbCrLf & "Enter an real: ")
        .Prompt vbCr & "You entered " & dblInput
    End With
End Sub
```

This method raises an exception for null input, keyword entry, and canceled input.

The `GetPoint` Method

The `GetPoint` method gets a point from the user, either by typing coordinates at the command prompt or by picking points in the drawing area. The return value is a Variant data type and contains a three-element array of doubles holding the point’s World Coordinate System (WCS) coordinates. This method has the following syntax:

```
varUserPointInput = UtilityObject.GetPoint([BasePoint] [,Prompt])
```

Table 7-4 explains this method’s parameters.

Table 7-4. *The* `GetPoint` *Method’s Parameters*

Name	Type	Description
`BasePoint`	Variant	Optional. A three-element array of doubles that specifies an angle vector’s first point in WCS.
`Prompt`	String	Optional. A prompt for input.

The optional `BasePoint` parameter sets a rubber-band line’s start point. This line, which extends to the current crosshair position, can be a useful visual aid to the user during input.

This example gets a point from the user and displays its coordinate values:

```
Public Sub TestGetPoint()
Dim varPick As Variant

    With ThisDrawing.Utility
        varPick = .GetPoint(, vbCr & "Pick a point: ")
        .Prompt vbCr & varPick(0) & "," & varPick(1)
    End With
End Sub
```

This method raises exceptions for null input, keyword entry, and canceled input.

The `GetCorner` Method

Given a base point in a rectangle, the `GetCorner` method gets the diagonally opposing corner point. It returns a Variant data type and contains a three-element array of doubles showing the corner point's WCS coordinates. This method has the following syntax:

```
varUserCornerInput = UtilityObject.GetCorner(BasePoint [,Prompt])
```

Table 7-5 explains this method's parameters.

Table 7-5. *The* `GetCorner` *Method's Parameters*

Name	Type	Description
`BasePoint`	Variant	A three-element array of doubles specifying the rectangle's first corner in WCS.
`Prompt`	String	Optional. A prompt for input.

If the user picks a point on the graphic screen, the `GetCorner` method ignores the point's Z-coordinate and sets it to the current elevation. This example gets a point and then a corner from the user and displays the rectangle's values:

```
Public Sub TestGetCorner()
Dim varBase As Variant
Dim varPick As Variant

    With ThisDrawing.Utility
        varBase = .GetPoint(, vbCr & "Pick the first corner: ")
        .Prompt vbCrLf & varBase(0) & "," & varBase(1)
        varPick = .GetCorner(varBase, vbLf & "Pick the second: ")
        .Prompt vbCr & varPick(0) & "," & varPick(1)
    End With
End Sub
```

This method throws exceptions for null input, keyword entry, and canceled input.

The `GetDistance` Method

The `GetDistance` method gets a double from the user. It differs from `GetReal` in that the user can either type a distance in the current units format or pick the point(s) on the graphics screen. These two methods are otherwise similar, and most people prefer `GetDistance` because it's more flexible. This method has the following syntax:

```
dblUserDistanceInput = UtilityObject.GetDistance([BasePoint] [,Prompt])
```

Table 7-6 explains this method's parameters.

Table 7-6. *The* `GetDistance` *Method's Parameters*

Name	Type	Description
`BasePoint`	Variant	Optional. A three-element array of doubles specifying a start point from which to begin measuring (in WCS). If you don't provide this parameter, the user must specify two points.
`Prompt`	String	Optional. A prompt for input.

Note This method lets you enter a negative number at the command prompt and returns this negative number. But it calculates the absolute distance between points if you enter it on the graphics screen.

If the user chooses to pick points from the screen, AutoCAD draws a rubber-band line as a visual aid from the *base point*, or first pick point, to the current crosshair position. By default, the points are 3-D. You may force AutoCAD to calculate a planar distance by first calling `InitializeUserInput` with a bit code of 16 in `OptionBits`. This makes AutoCAD ignore the Z-coordinates.

This example code sets the base point to the origin of WCS, prompts the user for input, and then displays the value:

```
Public Sub TestGetDistance()
Dim dblInput As Double
Dim dblBase(2) As Double

    dblBase(0) = 0:  dblBase(1) = 0:  dblBase(2) = 0

    With ThisDrawing.Utility
        dblInput = .GetDistance(dblBase, vbCr & "Enter a distance: ")
        .Prompt vbCr & "You entered " & dblInput
    End With
End Sub
```

This method raises an exception for null input, keyword entry, and canceled input.

The `GetAngle` Method

Use the `GetAngle` to get an angle, in radians, from the user. The user may either type the angle at the command prompt or pick point(s) on the screen. VBA ignores the points' Z-coordinates. It measures the angle counterclockwise with respect to the `ANGBASE` system variable's current value. This method has the following syntax:

```
dblUserAngleInput = UtilityObject.GetAngle([BasePoint] [,Prompt])
```

Table 7-7 explains this method's parameters.

Table 7-7. *The* `GetAngle` *Method's Parameters*

Name	Type	Description
`BasePoint`	Variant	Optional. A three-element array of doubles specifying an angle vector's first point in WCS. If not provided, the user must specify two points to specify the angle on the graphics screen.
`Prompt`	String	Optional. A prompt for input.

This method returns the angle in radians regardless of the current setting of the `DIMAUNIT` angular units system variable or the angular unit type the user entered. In this way, it acts in a similar manner to the `Utility` object's `AngleToReal` conversion method.

This example sets the angular units to degrees and then retrieves and displays an angle from the user:

```
Public Sub TestGetAngle()
Dim dblInput As Double

    ThisDrawing.SetVariable "DIMAUNIT", acDegrees

    With ThisDrawing.Utility
        dblInput = .GetAngle(, vbCr & "Enter an angle: ")
        .Prompt vbCr & "Angle in radians: " & dblInput
    End With
End Sub
```

This method throws exceptions for null input, keyword entry, and canceled input.

The `GetOrientation` Method

The `GetOrientation` method is similar to `GetAngle`, except that the angle returned is always measured from the east, or three o'clock, regardless of the `ANGBASE` system variable setting. This method has the following syntax:

```
dblUserOrientationInput = UtilityObject.GetOrientation([BasePoint] [,Prompt])
```

Table 7-8 explains this method's parameters.

Table 7-8. *The* `GetOrientation` *Method's Parameters*

Name	Type	Description
`BasePoint`	Variant	Optional. A three-element array of doubles specifying an angle vector's first point in WCS. If not provided, the user must specify two points if they want to specify the angle on the graphics screen.
`Prompt`	String	Optional. A prompt for input.

This method throws an exception for both null input and keyword entry if allowed and supplied.

The GetInput Method

As mentioned in the discussion of GetKeyword, you can use seven other input methods in conjunction with the keywords set using InitializeUserInput:

- GetInteger
- GetReal
- GetDistance
- GetAngle
- GetOrientation
- GetPoint
- GetCorner

When these methods have keywords and they are executed, the user can enter the data type requested or choose one of the available keywords. Because each of these input methods returns a specific data type, each can't also return the keyword string. Instead, the input methods use an exception with the description User input is a keyword to signal the presence of a keyword. Unless null input is disabled, VBA uses this same exception to indicate null input, such as when the user simply presses Enter at the input prompt.

Your code must handle the exception, or the program will stop and display an error message. After detecting this exception, use GetInput to retrieve the keyword before calling any other GetXXX method. The GetInput method takes no parameters. It returns either a string containing the keyword in InitializeUserInput or an empty string in the case of null input.

```
StrUserKeywordInput = UtilityObject.GetInput()
```

As shown in the previous examples, it is not necessary to call the GetInput method after the GetXXX methods when no keywords are offered to the user. If called independently, this method returns an empty string.

The following code retrieves either an integer or a keyword from the user. It uses GetInteger to demonstrate the GetInput method, but the technique is identical for each of the six other input methods:

```
Public Sub TestGetInput()
Dim intInput As Integer
Dim strInput As String

On Error Resume Next                    ' handle exceptions inline
    With ThisDrawing.Utility
        strInput = .GetInput()
        .InitializeUserInput 0, "Line Arc Circle"
        intInput = .GetInteger(vbCr & "Integer or [Line/Arc/Circle]: ")
        If Err.Description Like "*error*" Then
            .Prompt vbCr & "Input Cancelled"
        ElseIf Err.Description Like "*keyword*" Then
            strInput = .GetInput()
            Select Case strInput
                Case "Line": ThisDrawing.SendCommand "_Line" & vbCr
                Case "Arc": ThisDrawing.SendCommand "_Arc" & vbCr
                Case "Circle": ThisDrawing.SendCommand "_Circle" & vbCr
                Case Else: .Prompt vbCr & "Null Input entered"
            End Select
        Else
            .Prompt vbCr & "You entered " & intInput
        End If
    End With

End Sub
```

This example first sets the keywords using `InitializeUserInput` and then uses an input method to get the user's input. Because the code sets keywords, the input method either returns an integer or throws an exception when the user enters a keyword. If there was an exception and it contains the word *error*, this code displays a cancel message. Otherwise, if the exception contains the word *keyword*, the code uses `GetInput` to retrieve the keyword and takes an appropriate action based on the keyword. If no exception was thrown, this code uses the `Integer` value the `GetInteger` method returned.

A bug causes `GetInput` to return the keyword from earlier calls to `InitializeUserInput` when the user enters null input at a later input method that takes keywords. The following code demonstrates the problem. Choose any option at the first input prompt and then press Enter for the second input. In case the user enters a keyword, the second call to `GetInput` should return a null—but instead it returns the keyword you selected in the previous input.

```
Public Sub TestGetInputBug()

On Error Resume Next                    ' handle exceptions inline

    With ThisDrawing.Utility

        '' first keyword input
        .InitializeUserInput 1, "Alpha Beta Ship"
        .GetInteger vbCr & "Option [Alpha/Beta/Ship]: "
        MsgBox "You entered: " & .GetInput()

        '' second keyword input - hit Enter here
        .InitializeUserInput 0, "Bug May Slip"
        .GetInteger vbCr & "Hit enter [Bug/May/Slip]: "
        MsgBox "GetInput still returns: " & .GetInput()
    End With
End Sub
```

Because null and keyword input throw the same exceptions, `GetInput`'s return value is the only way to determine which the user entered. But because of this odd persistence of the previous keyword entry, no sure way exists to determine whether the user entered a keyword or nothing at all.

The following code demonstrates a partial workaround for this problem. The technique is to grab the existing keyword from `GetInput` before calling `InitializeUserInput` for the second `GetXXX` method. If, after getting input from the user, the newly entered keyword matches the previous keyword, it's possible that the user entered a null.

```
Public Sub TestGetInputWorkaround()
Dim strBeforeKeyword As String
Dim strKeyword As String

On Error Resume Next                    ' handle exceptions inline
    With ThisDrawing.Utility

        '' first keyword input
        .InitializeUserInput 1, "This Bug Stuff"
        .GetInteger vbCrLf & "Option [This/Bug/Stuff]: "
        MsgBox "You entered: " & .GetInput()

        '' get lingering keyword
        strBeforeKeyword = .GetInput()

        '' second keyword input - press Enter
        .InitializeUserInput 0, "Make Life Rough"
        .GetInteger vbCrLf & "Hit enter [Make/Life/Rough]: "
        strKeyword = .GetInput()

        '' if input = lingering, it might be null input
        If strKeyword = strBeforeKeyword Then
            MsgBox "Looks like null input: " & strKeyword
        Else
            MsgBox "This time you entered: " & strKeyword
        End If
    End With

End Sub
```

The `GetEntity` Method

Use the `GetEntity` method to select an AutoCAD object by letting the user pick an entity from the graphics screen.

■**Note** Because the user may type **L** for the last entity in the drawing window, this method can return an invisible entity or an entity that's on a frozen layer. However, in 2005 and 2006, AutoCAD will select the last visible entity.

This method has the following syntax:

```
UtilityObject.GetEntity PickedEntity, PickedPoint[, Prompt]
```

Table 7-9 explains this method's parameters.

Table 7-9. *The* `GetEntity` *Method's Parameters*

Name	Type	Description
`PickedEntity`	`AcadEntity` object	Output. Returns a reference to the drawing object that the user picked.
`PickPoint`	Variant	Output. A three-element array of doubles that specifies the point by which the entity was picked in WCS.
`Prompt`	String	Optional. A prompt for input.

This example gets an entity from the user and displays the point's object type and coordinates:

```
Public Sub TestGetEntity()
Dim objEnt As AcadEntity
Dim varPick As Variant

On Error Resume Next
    With ThisDrawing.Utility
        .GetEntity objEnt, varPick, vbCr & "Pick an entity: "
        If objEnt Is Nothing Then 'check if object was picked.
            .Prompt vbCrLf & "You did not pick as entity"
            Exit Sub
        End If
        .Prompt vbCr & "You picked a " & objEnt.ObjectName
        .Prompt vbCrLf & "At " & varPick(0) & "," & varPick(1)
    End With
End Sub
```

`GetEntity` raises an error if the input is null, such as when there is no entity at the picked point, or if the user presses Enter without selecting an entity. The example checks for this condition to avoid the error.

The `GetSubEntity` Method

Use the `GetSubEntity` method in place of `GetEntity` when you need to obtain subentity information based on the user's selection. A *subentity* is an entity that another entity contains, such as an entity in a block or a vertex entity contained in polylines. This method lets the user pick an entity or subentity from the graphics screen and returns details about that entity and any object that contains it.

■**Note** Like `GetEntity`, this method can return an entity even if it is not currently visible or is on a frozen layer because the user can type **L** to select the last entity in the drawing window. However, in 2005 and 2006, AutoCAD will select the last visible entity.

This method has the following syntax:

```
UtilityObject.GetSubEntity PickedEntity, PickPoint, Matrix, Context[, Prompt]
```

Table 7-10 explains this method's parameters.

Table 7-10. *The* `GetSubEntity` *Method's Parameters*

Name	Type	Description
`PickedEntity`	`AcadEntity` object	Output. Returns a reference to the drawing object that the user picked.
`PickPoint`	Variant	Output. A three-element array of doubles that specifies an angle vector's first point in WCS.
`Matrix`	Variant	Output. Returns a four-by-four element array of doubles that holds the selected entity's translation matrix.
`Context`	Variant	Output. Returns an array of long integers holding the `ObjectIds` for each parent block containing the selected entity, if the entity is in a block.
`Prompt`	String	Optional. A prompt for input.

The `Matrix` output parameter is the selected entity's Model to World Transformation Matrix. It is a composite of all the transformations involved in the entity's visible representation. Imagine a line stored in a block, which is also stored in a block. The `Matrix` output parameter encapsulates each scale, rotation, and translation involved in the nested line's display. You can use it to translate points from the internal Model Coordinate System (MCS) to WCS.

The `Context` output parameter is an array of the `ObjectIds` for any objects that contain the selected entity. For the example line, this would be an array of two `ObjectIds`, one for each nesting level of the containing blocks. To get information about each containing entity, use the `ObjectIdToObject` method of the `Document` object to convert the `ObjectId` to an object reference.

The following example uses the `Prompt` method to display information about the selected entity and any containing entities. Try it on a variety of entities, including those nested in blocks.

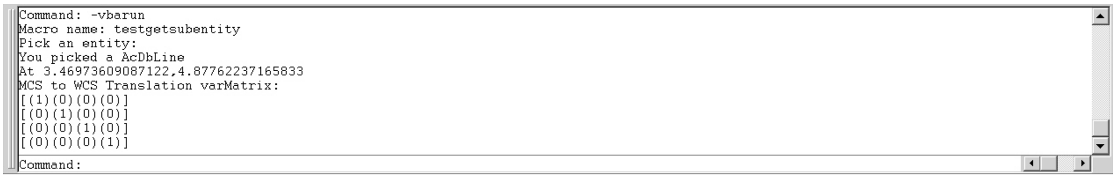
```
Public Sub TestGetSubEntity()
Dim objEnt As AcadEntity
Dim varPick As Variant
Dim varMatrix As Variant
Dim varParents As Variant
Dim intI As Integer
Dim intJ As Integer
Dim varID As Variant
```

```
With ThisDrawing.Utility

    '' get the subentity from the user
    .GetSubEntity objEnt, varPick, varMatrix, varParents, _
        vbCr & "Pick an entity: "

    '' print some information about the entity
    .Prompt vbCr & "You picked a " & objEnt.ObjectName
    .Prompt vbCrLf & "At " & varPick(0) & "," & varPick(1)

    '' dump the varMatrix
    If Not IsEmpty(varMatrix) Then
        .Prompt vbLf & "MCS to WCS Translation varMatrix:"

        '' format varMatrix row
        For intI = 0 To 3
            .Prompt vbLf & "["

            '' format varMatrix column
            For intJ = 0 To 3
                .Prompt "(" & varMatrix(intI, intJ) & ")"
            Next intJ

            .Prompt "]"
        Next intI
        .Prompt vbLf
    End If

    '' if it has a parent nest
    If Not IsEmpty(varParents) Then

        .Prompt vbLf & "Block nesting:"

        '' depth counter
        intI = -1

        '' traverse most to least deep (reverse order)
        For intJ = UBound(varParents) To LBound(varParents) Step -1

            '' increment depth
            intI = intI + 1

            '' indent output
            .Prompt vbLf & Space(intI * 2)

            '' parent object ID
            varID = varParents(intJ)

            '' parent entity
            Set objEnt = ThisDrawing.ObjectIdToObject(varID)
```

```
                '' print info about parent
                .Prompt objEnt.ObjectName & " : " & objEnt.Name
            Next intJ
            .Prompt vbLf
        End If
        .Prompt vbCr
    End With
End Sub
```

`GetSubEntity` throws an exception if the input is null, such as when there is no entity at the picked point or when the user presses Enter without selecting an entity.

Figure 7-3 shows the result of picking a line that forms part of a block reference named Window.



```
Command: -vbarun
Macro name: testgetsubentity
Pick an entity:
You picked a AcDbLine
At 3.46973609087122,4.87762237165833
MCS to WCS Translation varMatrix:
[(1)(0)(0)(0)]
[(0)(1)(0)(0)]
[(0)(0)(1)(0)]
[(0)(0)(0)(1)]
Command:
```

Figure 7-3. *The* `TestGetSubEntity` *command line output*

Handling Errors in User Input

Each AutoCAD command line user input method this chapter discusses uses run-time errors to signal input conditions such as null input, keyword input, or the user canceling the command line input. Your code must handle these run-time errors, known as *exceptions*, or the program halts and displays an error message. VBA gives you two basic strategies for dealing with these exceptions.

`On Error Goto`: The first strategy is to use `On Error Goto` to jump to labeled sections of code called *error handlers*. This strategy has the advantage of keeping the main program logic very clean—all the error handler code is separate from the program body. It has the disadvantage of becoming unwieldy as the program's size or complexity increases. As the complexity of errors increases, so too does the error handler's complexity. At a certain level of complexity, you reach a point of diminishing returns with labeled error handlers. They begin to take on a life of their own, with all their necessary conditions, loops, and jumps back and forth from the main program body.

`On Error Resume Next`: The second strategy is to use `On Error Resume Next` to handle the errors in the main program body. This is called *inline error handling*. It has the advantage of keeping the exception handling close to the method or function that generated the condition. This encourages, and in some ways forces, a more direct cause-and-effect framework for dealing with errors. Because inline error handling is woven into the main program execution, it has the disadvantage of clouding the pure algorithmic logic. Because of this interaction with the main program logic, it is also more difficult to retrofit existing code that lacks error handling with code that uses inline handling.

This chapter's examples minimize error handling to focus all attention on the methods being described. However, because each input method generates exceptions for normal and expected conditions such as null input and keyword entry, be prepared to handle these errors in your program. See the sample code for the GetInput method for an example of handling these kinds of exceptions. You can apply this same framework to each input method.

Conversion Methods

Several Utility methods do not involve direct user interaction but are often used in conjunction with user input. You can categorize these methods as conversion methods. Use them to convert between common unit types or among AutoCAD's various coordinate systems.

The AngleToReal Method

Use the AngleToReal method to convert a string that represents an angle into the equivalent radian value. This is useful, for example, for converting angles the user inputs as a string into a form compatible with most of AutoCAD's geometric methods. This method returns a double, giving the angle in radians. It has the following syntax:

```
dblAngle = UtilityObject.AngleToReal(Angle, Unit)
```

Table 7-11 explains this method's parameters.

Table 7-11. *The* AngleToReal *Method's Parameters*

Name	Type	Description
Angle	String	A string containing the angle to convert
Unit	Long	The input angle's default unit format as one of AcAngleUnits' constants: acDegreeMinuteSeconds, acDegrees, acGrads, or acRadians

The Unit parameter specifies the conversion's default unit type. This default unit is assigned when the input specifies no units. For instance, the string 2.5 can be interpreted as degrees, grads, or radians. If the input contains an explicit unit type, AutoCAD uses it and ignores the Unit parameter. For instance, 2.5d, 2.5g, and 2.5r are each explicit, and AutoCAD reads them as degrees, grads, and radians, respectively.

This example gets an angle from the user, converts it to a double using AngleToReal with a default of degrees, and then displays the results of the conversion to radians:

```
Public Sub TestAngleToReal()
Dim strInput As String
Dim dblAngle As Double

    With ThisDrawing.Utility
        strInput = .GetString(True, vbCr & "Enter an angle: ")
        dblAngle = .AngleToReal(strInput, acDegrees)
        .Prompt vbCr & "Radians: " & dblAngle
    End With
End Sub
```

An exception is thrown if AngleToReal is unable to convert the input string.

The `AngleToString` Method

Use the `AngleToString` method to convert an angle in radians to a string representing the angle in angular units. This is useful for converting radian angles, used by most geometric properties and methods, into a form that the user can work with. This method returns a string that gives the angle in a specified unit type and precision. This method has the following syntax:

```
strAngle = UtilityObject.AngleToString(Angle, Unit, Precision)
```

Table 7-12 explains this method's parameters.

Table 7-12. *The* `AngleToString` *Method's Parameters*

Name	Type	Description
`Angle`	Double	The angle, in radians, to be converted
`Unit`	Long	The angular unit for the output string as one of `AcAngleUnits`' constants: `acDegreeMinuteSeconds`, `acDegrees`, `acGrads`, or `acRadians`
`Precision`	Long	Specifies the output's precision from zero to eight decimal places

This example gets an angle from the user, converts it to a double using `AngleToReal` with a default of degrees, and then converts and displays the results using `AngleToString` with a unit of degrees and a precision of four decimal places:

```
Public Sub TestAngleToString()
Dim strInput As String
Dim strOutput As String
Dim dblAngle As Double

    With ThisDrawing.Utility
        strInput = .GetString(True, vbCr & "Enter an angle: ")
        dblAngle = .AngleToReal(strInput, acDegrees)
        .Prompt vbCr & "Radians: " & dblAngle

        strOutput = .AngleToString(dblAngle, acDegrees, 4)
        .Prompt vbCrLf & "Degrees: " & strOutput
    End With
End Sub
Sub TestAngleToString()
```

The `DistanceToReal` Method

Use the `DistanceToReal` method to convert a string that represents a linear distance into the equivalent double. This is useful, for example, for converting distances the user inputs as a string into their numeric equivalents. This method returns a double. It has the following syntax:

```
dblDistance = UtilityObject.DistanceToReal(Distance, Unit)
```

Table 7-13 explains this method's parameters.

Table 7-13. *The* `DistanceToReal` *Method's Parameters*

Name	Type	Description
`Distance`	String	A string that contains the angle to convert
`Unit`	Long	The input's default unit format, as one of `AcUnits`' constants: `acArchitectural`, `acDecimal`, `acDefaultUnits`, `acEngineering`, `acFractional`, or `acScientific`

The `Unit` parameter specifies the conversion's default unit type. VBA assigns this default unit when the input specifies no units. If the input contains an explicit unit type, AutoCAD uses it and ignores the `Unit` parameter. For instance, `1'6-1/2"`, `1.54'`, `18 1/2`, and `0.185e+2` are each explicit, and AutoCAD reads them as architectural, engineering, fractional, and scientific notation, respectively. The `acDefaultUnits` value uses whichever unit type the `LUNITS` system variable currently specifies.

This example gets a distance from the user, converts it using `DistanceToReal` with a default of architectural units, and then displays the results of the conversion to a double:

```
Public Sub TestDistanceToReal()
Dim strInput As String
Dim dblDist As Double

    With ThisDrawing.Utility
        strInput = .GetString(True, vbCr & "Enter a distance: ")
        dblDist = .DistanceToReal(strInput, acArchitectural)
        .Prompt vbCr & "Distance: " & dblDist
    End With
End Sub
```

An exception is thrown if `DistanceToReal` is unable to convert the input string.

The `RealToString` Method

Use the `RealToString` method to convert a double to a string representing a distance in linear units. This is useful for converting the lengths, sizes, and locations that most AutoCAD geometric properties and methods use into a form that the user can work with. This method returns a string containing the distance in a specified unit type and precision. It has the following syntax:

```
strDistance = UtilityObject.RealToString(Distance, Unit, Precision)
```

Table 7-14 explains this method's parameters.

Table 7-14. *The* `RealToString` *Method's Parameters*

Name	Type	Description
`Distance`	Double	A double containing the distance
`Unit`	Long	The linear unit format for the output string, as one of `AcUnits`' constants: `acArchitectural`, `acDecimal`, `acDefaultUnits`, `acEngineering`, `acFractional`, or `acScientific`
`Precision`	Long	The output's precision, from zero to eight decimal places

This example gets a distance from the user, converts it to a double using `DistanceToReal` with a default of architectural, and then converts and displays the results using `RealToString` with a unit of architectural and a precision of four decimal places:

```
Public Sub TestRealToString()
Dim strInput As String
Dim strOutput As String
Dim dblDist As Double

    With ThisDrawing.Utility
        strInput = .GetString(True, vbCr & "Enter a distance: ")
        dblDist = .DistanceToReal(strInput, acArchitectural)
        .Prompt vbCr & "Double: " & dblDist

        strOutput = .RealToString(dblDist, acArchitectural, 4)
        .Prompt vbCrLf & "Distance: " & strOutput
    End With
End Sub
```

The `AngleFromXAxis` Method

Use the `AngleFromXAxis` method to measure the angle, in radians, between an imaginary line formed by two points and the WCS X-axis. This method returns a double. It has the following syntax:

```
dblAngle = UtilityObject.AngleFromXAxis(StartPoint, EndPoint)
```

Table 7-15 explains this method's parameters.

Table 7-15. *The* `AngleFromXAxis` *Method's Parameters*

Name	Type	Description
`StartPoint`	Variant	A three-element array of doubles that specifies the first point of an imaginary line in WCS
`EndPoint`	Variant	A three-element array of doubles that specifies the second point of an imaginary line in WCS

This example gets two points from the user and then uses the `AngleFromXAxis` method to calculate the angle of a line formed by those points and the WCS X-axis:

```
Public Sub TestAngleFromXAxis()
Dim varStart As Variant
Dim varEnd As Variant
Dim dblAngle As Double

    With ThisDrawing.Utility
        varStart = .GetPoint(, vbCr & "Pick the start point: ")
        varEnd = .GetPoint(varStart, vbCr & "Pick the end point: ")
        dblAngle = .AngleFromXAxis(varStart, varEnd)
        .Prompt vbCr & "The angle from the X-axis is " _
           & .AngleToString(dblAngle, acDegrees, 2) & " degrees"
    End With
End Sub
```

The `PolarPoint` Method

Use the `PolarPoint` method to get a new point that's a specified distance and angle from another point. This method returns a Variant that holds a three-element array of doubles. It has the following syntax:

```
varPolarPoint = UtilityObject.PolarPoint(OriginalPoint, Angle, Distance)
```

Table 7-16 explains this method's parameters.

Table 7-16. *The* `PolarPoint` *Method's Parameters*

Name	Type	Description
`OriginalPoint`	Variant	A three-element array of doubles that specifies the point from which to begin the polar calculation
`Angle`	Double	The angle in radians measured from WCS X-axis
`Distance`	Double	The distance between the original and new point measured in current drawing units

In this example, the user specifies a start point and a rectangle's length, height, and angle. It uses the `PolarPoint` method and user input to calculate the rest of the points:

```
Public Sub TestPolarPoint()
Dim varpnt1 As Variant
Dim varpnt2 As Variant
Dim varpnt3 As Variant
Dim varpnt4 As Variant
Dim dblAngle As Double
Dim dblLength As Double
Dim dblHeight As Double
Dim dbl90Deg As Double
```

```
    '' get the point, length, height, and angle from user
    With ThisDrawing.Utility

        '' get point, length, height, and angle from user
        varpnt1 = .GetPoint(, vbCr & "Pick the start point: ")
        dblLength = .GetDistance(varpnt1, vbCr & "Enter the length: ")
        dblHeight = .GetDistance(varpnt1, vbCr & "Enter the height: ")
        dblAngle = .GetAngle(varpnt1, vbCr & "Enter the angle: ")

        '' calculate remaining rectangle points
        dbl90Deg = .AngleToReal("90d", acDegrees)
        varpnt2 = .PolarPoint(varpnt1, dblAngle, dblLength)
        varpnt3 = .PolarPoint(varpnt2, dblAngle + dbl90Deg, dblHeight)
        varpnt4 = .PolarPoint(varpnt3, dblAngle + (dbl90Deg * 2), dblLength)
    End With

    '' draw the rectangle
    With ThisDrawing
        .ModelSpace.AddLine varpnt1, varpnt2
        .ModelSpace.AddLine varpnt2, varpnt3
        .ModelSpace.AddLine varpnt3, varpnt4
        .ModelSpace.AddLine varpnt4, varpnt1
    End With

End Sub
```

Chapter 8 covers creating drawing objects.

The `TranslateCoordinates` Method

Use the `TranslateCoordinates` method to translate a point from one coordinate system to another. This is often necessary when the user is working in a construction plane, for example. Most AutoCAD input and object-creation methods work exclusively in WCS, so you need to translate user input to correctly work in the construction plane.

This method returns an array of doubles. It has the following syntax:

```
dblAngle = UtilityObject.AngleFromXAxis(Point, From, To, Displacement [, Norm])
```

Table 7-17 explains this method's parameters.

Table 7-17. *The* `TranslateCoordinates` *Method's Parameters*

Name	Type	Description
`Point`	Variant	A three-element array of doubles that specifies the point to translate.
`From`	Long	The source coordinate system. One of the `AcCoordinateSystem` constants: `acDisplayDCS`, `acOCS`, `acPaperSpaceDCS`, `acUCS`, or `acWorld`.
`To`	Long	The destination coordinate system. One of the `AcCoordinateSystem` constants: `acDisplayDCS`, `acOCS`, `acPaperSpaceDCS`, `acUCS`, or `acWorld`.
`Displacement`	Boolean	If `True`, the input is treated as a displacement vector. If `False`, the input is treated as a point.
`Norm`	Variant	Optional. A three-element array of doubles specifying the Object Coordinate System (OCS) normal. Use with an `acOCS` value in either the `From` parameter or the `To` parameter.

The `From` and `To` parameters specify the point translation's source and destination coordinate systems. For instance, to convert the `Point` parameter from WCS to the current construction plane, set `From` to `acWorld` and set `To` to `acUCS`. If either `From` or `To` is set to `acOCS`, you must pass the `Norm` to indicate the OCS normal vector.

■ **Note** Because you can pass only a single normal vector into the method, you can't translate points from one OCS directly to another. Instead, translate the source OCS to an intermediate coordinate system and then translate from the intermediate coordinate system to the destination OCS.

When the `Displacement` parameter is `True`, the `Point` parameter is treated like a displacement vector instead of like a point. Use this setting when you need to calculate offset values instead of a new point, such as when performing relative movement. For example, to move an entity by a specified amount, compose the X, Y, and Z displacements and pass them to `TranslateCoordinates` as a displacement vector.

The following example gets two points from the user and then uses the `AngleFromXAxis` method to calculate the angle of the line those points form and the WCS X-axis. It uses `TranslateCoordinates` to translate the first input point to User Coordinate System (UCS) for use in the base point of the second point acquisition.

```
Public Sub TestTranslateCoordinates()
Dim varpnt1 As Variant
Dim varpnt1Ucs As Variant
Dim varpnt2 As Variant

    '' get the point, length, height, and angle from user
    With ThisDrawing.Utility

        '' get start point
        varpnt1 = .GetPoint(, vbCr & "Pick the start point: ")
        '' convert to UCS for use in the base point rubber-band line
        varpnt1Ucs = .TranslateCoordinates(varpnt1, acWorld, acUCS, False)

        '' get end point
        varpnt2 = .GetPoint(varpnt1Ucs, vbCr & "Pick the end point: ")
    End With

    '' draw the line
    With ThisDrawing
        .ModelSpace.AddLine varpnt1, varpnt2
    End With
End Sub
```

Try `TestTranslateCoordinates` from various coordinate systems, and you can see that the rubber-band line in the second point input always starts in the correct position. Without the coordinate translation, the point is always expressed in WCS coordinates and appears incorrectly.

Internet Methods

AutoCAD 2000 and higher has the ability to open and save drawing files from the Internet. The following methods let your program accept, return, or validate Uniform Resource Locators (URLs), which are Internet addresses that uniquely identify remote files. The methods include a Web file browser, Internet file upload and downloading capabilities, and functions for validating URLs and determining if a file originated from the Internet.

The `IsURL` Method

Use the `IsURL` method to determine if a string is a properly formed URL. It does not ensure that the URL is actually accessible, just that it is syntactically correct. It returns a Boolean. Its syntax is as follows:

```
blnStatus = UtilityObject.IsURL(InputURL)
```

This method has one parameter, `InputURL`, a string. Use it to specify the URL to check for validity.

This example gets a string from the user and then, using `IsURL`, displays whether it is a valid URL:

```
Public Sub TestIsURL()
Dim strInput As String

    With ThisDrawing.Utility
        strInput = .GetString(True, vbCr & "Enter a URL: ")
        If .IsURL(strInput) Then
            MsgBox "You entered a valid URL"
        Else
            MsgBox "That was not a URL"
        End If
    End With
End Sub
```

The LaunchBrowserDialog Method

Use the `LaunchBrowserDialog` method to perform a Hypertext Transfer Protocol (HTTP) request, displaying a document using the default Web browser. It can't verify or validate URLs; it simply opens the browser and sends the request, exactly the way launching a favorite works in Internet Explorer or launching a bookmark works in Netscape. This method sets `OutputURL` to the URL the user selects. It returns `True` if the Web browser successfully launched and the user selected a file, or it returns `False` to indicate failure or that the user didn't select a file. This method has the following syntax:

```
blnStatus = UtilityObject.LaunchBrowserDialog(OutputURL, _
    Title, SelectCaption, StartingURL, PreferencesKey, SelectLinks)
```

Table 7-18 explains this method's parameters.

Table 7-18. *The* `LaunchBrowserDialog` *Method's Parameters*

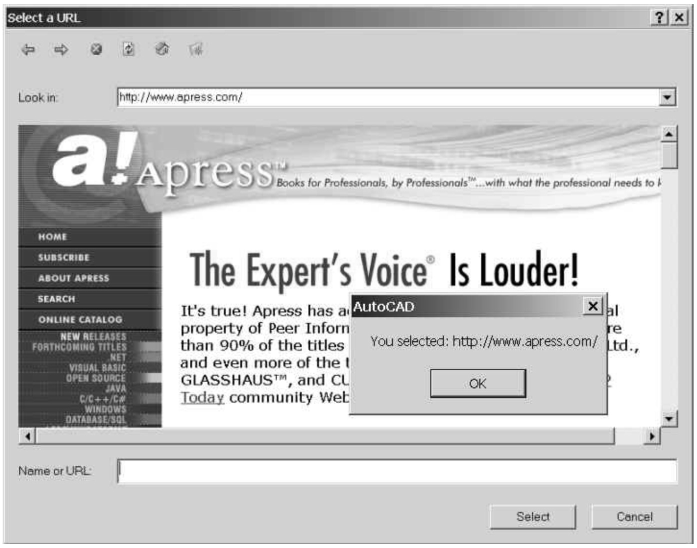
Name	Type	Description
`SelectedURL`	String	Output. The URL of the remote file the user selected.
`DialogTitle`	String	A caption for the Web browser dialog.
`OpenButtonCaption`	String	A caption for the OK/Open button.
`StartPageURL`	String	The starting URL for the Web browser.
`RegistryRootKey`	String	The product root key for storing persistent Web browser dialog information. This key specifies where to store information about the size, position, and other preferences information of the dialog and can be stored across sessions. Leave empty to disregard this functionality.
`OpenButtonAlwaysEnabled`	Boolean	Input Only. If `True`, the Open button is enabled, allowing a file or link to be selected. If `False`, the Open button is disabled and is enabled only when the user selects a file for download.

The `RegistryRootKey` parameter stores the browser window's size and location in the Windows Registry. This lets your application make the Web browser appear with the same size and location each time it is called. The information is stored in `HKEY_CURRENT_USER` and so is specific to each user. If you don't want to use this capability, pass a null (empty) string in this parameter.

The `OpenButtonAlwaysEnabled` parameter enables URL link selection. If `True`, the Select button is enabled at all times, and the browser can return any URL, whether it's an actual file or just a link. Use this if your program just needs to obtain a URL from the user and doesn't need to actually open or work with the contents of a remote file.

The following example uses `LaunchBrowserDialog` to get a URL from the user and displays the selection in a message box. Figure 7-4 illustrates this code in action.

```
Public Sub TestLaunchBrowserDialog()
Dim strStartUrl As String
Dim strInput As String
Dim blnStatus As Boolean

    strStartUrl = InputBox("Enter a URL", , "http://www.apress.com")
    With ThisDrawing.Utility
        If .IsURL(strStartUrl) = False Then
            MsgBox "You did not enter a valid URL"
            Exit Sub
        End If

        blnStatus = .LaunchBrowserDialog(strInput, _
                                         "Select a URL", _
                                         "Select", _
                                         strStartUrl, _
                                         "ContractCADDgroup", _
                                         True)
        If Not blnStatus Then
            MsgBox "You cancelled without selecting anything"
            Exit Sub
        End If

        If strStartUrl = strInput Then
            MsgBox "You selected the original URL"
        Else
            MsgBox "You selected: " & strInput
        End If
    End With
End Sub
```



You selected: http://www.apress.com/

Figure 7-4. *Getting output from* `TestLaunchBrowserDialog`

The GetRemoteFile Method

Use the GetRemoteFile method to retrieve a remote file given a URL. The file is downloaded to a temporary local file with the name being passed back in the LocalFile output parameter. This method returns nothing. It has the following syntax:

```
UtilityObject.GetRemoteFile InputURL, LocalFile, IgnoreCache
```

Table 7-19 explains this method's parameters.

Table 7-19. *The* GetRemoteFile *Method's Parameters*

Name	Type	Description
InputURL	String	The URL from which to download the file.
LocalFile	String	Output. The temporary local copy's file name.
IgnoreCache	Boolean	Whether to use the browser's file cache to retrieve the file.

If IgnoreCache is True, the file gets downloaded from the URL even if it is present in the local browser file cache. Do this to ensure you have the most current copy of the file.

If InputURL is a secure URL, a dialog box asks the user for access information such as username and password.

This example gets a file from a user-specified URL and displays the temporary file name in a message box:

```
Public Sub TestGetRemoteFile()
Dim strUrl As String
Dim strLocalName As String
Dim blnStatus As Boolean

 strUrl = InputBox("Enter a URL of a drawing file")

    With ThisDrawing.Utility
        If .IsURL(strUrl) = False Then
            MsgBox "You did not enter a valid URL"
            Exit Sub
        End If

        .GetRemoteFile strUrl, strLocalName, True
        If Err Then
            MsgBox "Failed to download: " & strUrl & vbCr & Err.Description
        Else
            MsgBox "The file was downloaded to: " & strLocalName
        End If
    End With
End Sub
```

Note When you exit AutoCAD, the temporary file this method creates is deleted from your system.

The IsRemoteFile Method

Use the IsRemoteFile method to determine whether a local file was retrieved from a URL. The OutputURL parameter contains the file's original URL. This method returns True if the file was downloaded from a URL. This method has the following syntax:

```
blnStatus = UtilityObject.IsRemoteFile(LocalFile, OutputURL)
```

Table 7-20 explains this method's parameters.

Table 7-20. *The* IsRemoteFile *Method's Parameters*

Name	Type	Description
LocalFile	String	The file name of a local file to check.
OutputURL	String	Output. The original URL of the local file.

This example gets a file name from the user and then displays a message telling whether the file was retrieved from a URL:

```
Public Sub TestIsRemoteFile()
Dim strOutputURL As String
Dim strLocalName As String

    strLocalName = InputBox("Enter the file and path name to check")
    If strLocalName = "" Then Exit Sub
    With ThisDrawing.Utility
        '' check if the local file is from a URL
        If .IsRemoteFile(strLocalName, strOutputURL) Then
            MsgBox "This file was downloaded from: " & strOutputURL
        Else
            MsgBox "This file was not downloaded from a URL"
        End If
    End With
End Sub
```

The PutRemoteFile Method

Use the PutRemoteFile method to upload a local file to a URL. This method returns nothing. It has the following syntax:

```
UtilityObject.PutRemoteFile UploadURL, LocalFile
```

Table 7-21 explains this method's parameters.

Table 7-21. The PutRemoteFile *Method's Parameters*

Name	Type	Description
UploadURL	String	The URL to which to upload the file
LocalFile	String	The file name to upload

If `UploadURL` is a secure URL, the method displays a dialog box to ask the user for their access information such as username and password.

For more extensive Web-enabled capabilities, look at the Autodesk i-drop technology and related API tools. The i-drop technology lets you publish content to Web sites and drag and drop selected items (and associated metadata) directly into drawings.

Summary

Although this chapter didn't cover every conceivable user interaction scenario, it did cover the most commonly used methods. The user interface is unique to each situation, and as a programmer you have to decide how best to use AutoCAD VBA's methods and properties to create the ideal user interface.

CHAPTER 8

Drawing Objects

AutoCAD provides you with a number of methods to create the drawing entities users normally access through the application window. Nearly every drawing entity that you can create and manipulate through the AutoCAD GUI has an equivalent object with associated properties and methods in the AutoCAD object model.

In this chapter I cover the majority of drawing entities that you can create as single AutoCAD objects, including the following:

- The `Arc`, `Circle`, and `Ellipse` objects
- Lines of finite and infinite length, such as the `Line`, `Ray`, and `Polyline` objects
- `Solid`, `Hatch`, and `Region` compound objects

The code samples in this chapter demonstrate how to create these objects through VBA. Realistically, though, before you create drawing entities you need to understand how to control the drawing space where you place and view these objects. You'll start in the next section by examining how to toggle between the model space and paper space.

Controlling the Drawing Space

AutoCAD segregates entities that users create into one of several collections. The two primary collections separate objects that make up the model geometry and objects used to describe the geometry for printing or plotting purposes. The model is stored in the `ModelSpace` collection, and the printing/plotting layout is stored in the `PaperSpace` collection.

Model space is the drawing area where users create the objects that they're designing or rendering. It typically contains everything that would exist in reality—each of the components in an assembly, or all of the physical elements of a building, for instance. Scaled views (or *viewports*) of specific areas of the model are then placed in *paper space*. Each of the views is dynamic, and changes to the model are automatically reflected in the paper space views.

Users will typically place items that describe the views of the model, such as dimensions, annotation, labels, and textual tables, into paper space. Title blocks and sheet borders are also placed here, and the complete page layout is formatted for output to a printer or plotter.

AutoCAD does nothing to enforce the division between the model space and the paper space—in fact, users may draw anything they want in either space. The spaces are simply provided to assist in the format and page layout for presentation and output purposes.

In AutoCAD 2000 and higher, users may create any number of paper space pages, which are called *layouts*. The layouts, each of which generally represents a single formatted page, are given unique names and are shown as a series of tabs at the bottom of each drawing window. AutoCAD automatically generates a model space layout named `Model` and a paper space layout named `Layout1`, but users may create as many as they'd like. Every layout except `Model` is a paper space layout, and the last of these accessed is always referred to by the `PaperSpace` property of the `Document` object.

■ **Note** You can't rename or delete the `Model` layout. If you delete the last paper space layout, AutoCAD automatically creates a new `Layout1`. However, you may rename any of the paper space layouts.

Internally, AutoCAD stores model space and each of the paper space layouts as special block definitions. Therefore, you may use all the operations noted in this chapter equally well with `ModelSpace`, `PaperSpace`, or `Block` objects with only occasional, minor differences.

The ModelSpace and PaperSpace Collections

One of the first things you need to know to use the `ModelSpace` and `PaperSpace` collections is which space—model or paper—is currently active. This information is held in the `ActiveSpace` property of the `Document` object. This is an integer property, and it may hold one of the two values shown in Table 8-1, along with that value's associated AutoCAD constants.

Table 8-1. `ModelSpace` *and* `PaperSpace` *Constants*

Constant	Value
acModelSpace	1
acPaperSpace	0

The following code snippet uses a message box to inform you of the active space for the current document:

```
If ThisDrawing.ActiveSpace = acModelSpace Then
    MsgBox "The active space is model space"
Else
    MsgBox "The active space is paper space"
End If
```

To change the active space, you need to set the value of the `ActiveSpace` property. The following code contains a macro you can use to toggle between the two spaces:

```
Public Sub ToggleSpace()
    With ThisDrawing
        If .ActiveSpace = acModelSpace Then
            .ActiveSpace = acPaperSpace
        Else
            .ActiveSpace = acModelSpace
        End If
    End With
End Sub
```

Alternatively, you can simply use the following line of code:

```
ThisDrawing.ActiveSpace = (ThisDrawing.ActiveSpace + 1) Mod 2
```

Creating Objects

As mentioned previously, the `AddXXX` methods apply equally to the `PaperSpace`, `ModelSpace`, and `Block` objects. Therefore, when I present the syntax of each method in this section, the word `Object` denotes any one of these collections. For example:

```
Set CircleObject = Object.AddCircle(CenterPoint, Radius)
```

Each of the `AddXXX` methods returns an object reference to the newly created entity, so you must use the VBA `Set` operator if you want to assign this return to a variable.

When you create or modify an entity, the changes to the drawing don't display until the `Update` method of that object, the `Update` method of the `Application` object, or the `Regen` method of the `Document` object is called. In some cases, AutoCAD will update the display once your macro or program is complete; however, it's always safest to ensure the update takes place in your code. In the examples presented here, I use the `Update` method of the newly created entity to update the drawing display.

Circular Objects

This section demonstrates how to create the various circular objects.

The `Arc` Object

You create an `Arc` object by using the `AddArc` method. This method takes four arguments that determine the position and size of the arc:

```
Set ArcObject = Object.AddArc(CenterPoint, Radius, StartAngle, EndAngle)
```

Table 8-2 provides a brief description of each argument.

Table 8-2. `AddArc` *Method Parameters*

Name	Data Type	Description
`CenterPoint`	Variant	A three-element array of doubles specifying the center of the arc in the WCS
`Radius`	Double	The radius of the arc
`StartAngle`	Double	The start angle of the arc given in radians with respect to the X-axis of the WCS
`EndAngle`	Double	The end angle of the arc given in radians with respect to the X-axis of the WCS

Arcs are drawn counterclockwise from the start angle to the end angle, as shown in Figure 8-1.

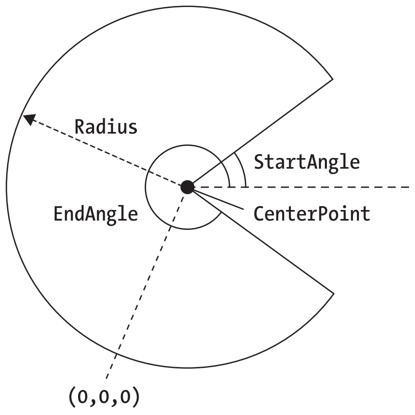


Figure 8-1. *The* `Arc` *object*

The following example creates an arc in the active drawing space (model or paper space) utilizing user input:

```
Public Sub TestAddArc()
Dim varCenter As Variant
Dim dblRadius As Double
Dim dblStart As Double
Dim dblEnd As Double
Dim objEnt As AcadArc

    On Error Resume Next
    '' get input from user
    With ThisDrawing.Utility
        varCenter = .GetPoint(, vbCr & "Pick the center point: ")
        dblRadius = .GetDistance(varCenter, vbCr & "Enter the radius: ")
        dblStart = .GetAngle(varCenter, vbCr & "Enter the start angle: ")
        dblEnd = .GetAngle(varCenter, vbCr & "Enter the end angle: ")
    End With

    '' draw the arc
If ThisDrawing.ActiveSpace = acModelSpace Then
    Set objEnt = ThisDrawing.ModelSpace.AddArc(varCenter, dblRadius, _
                                              dblStart, dblEnd)
Else
Set objEnt = ThisDrawing.PaperSpace.AddArc(varCenter, dblRadius, dblStart, dblEnd)
End If
    objEnt.Update
End Sub
```

The Circle Object

You can create a Circle object by specifying the position of the center and the radius, and using the AddCircle method:

```
Set CircleObject = Object.AddCircle(CenterPoint, Radius)
```

Table 8-3 provides a brief description of each argument.

Table 8-3. AddCircle *Method Parameters*

Name	Data Type	Description
CenterPoint	Variant	A three-element array of doubles specifying the center of the circle in the WCS
Radius	Double	The radius of the circle

Figure 8-2 illustrates the Circle object.

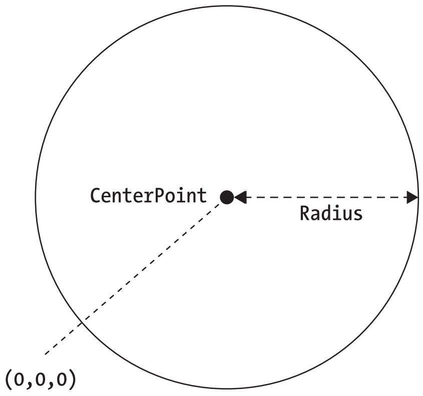


Figure 8-2. *The* Circle *object*

The following example creates a circle based on user input:

```
Public Sub TestAddCircle()
Dim varCenter As Variant
Dim dblRadius As Double
Dim objEnt As AcadCircle

On Error Resume Next
    '' get input from user
    With ThisDrawing.Utility
        varCenter = .GetPoint(, vbCr & "Pick the centerpoint: ")
        dblRadius = .GetDistance(varCenter, vbCr & "Enter the radius: ")
    End With
    '' draw the entity
If ThisDrawing.ActiveSpace = acModelSpace Then
  Set objEnt = ThisDrawing.ModelSpace.AddCircle(varCenter, dblRadius)
Else
  Set objEnt = ThisDrawing.PaperSpace.AddCircle(varCenter, dblRadius)
End If
    objEnt.Update
End Sub
```

The `Ellipse` Object

You use the `AddEllipse` method to create a fully closed `Ellipse` object. This method takes three parameters:

```
Set EllipseObject = Object.AddEllipse(CenterPoint, MajorAxis, RadiusRatio)
```

Table 8-4 describes the `AddEllipse` method's parameters.

Table 8-4. `AddEllipse` *Method Parameters*

Name	Data Type	Description
`CenterPoint`	Variant	A three-element array of doubles specifying the center of the ellipse in the WCS.
`MajorAxis`	Variant	A three-element array of doubles specifying the vector of the major axis of the ellipse from the `CenterPoint`.
`RadiusRatio`	Double	The ratio of the lengths of the minor to major axis vectors: 0 < `RadiusRatio` ≤ 1. A value of 1 generates an ellipse object that looks like a circle.

If you want to create an elliptical arc, set the `StartAngle` and `EndAngle` properties after you create the full `Ellipse` object, as shown in Figure 8-3.

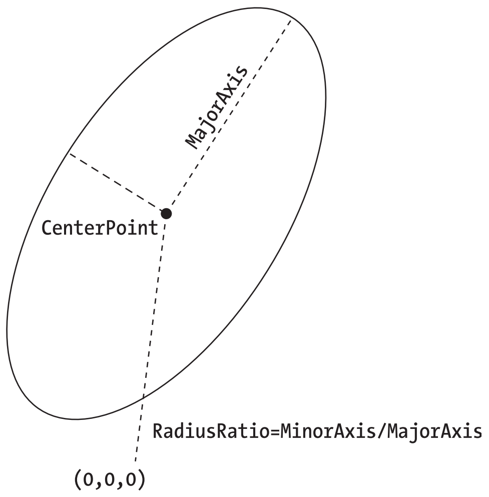


Figure 8-3. *The* `Ellipse` *object*

The following example creates a new `Ellipse` object by first setting the center axis, major axis, and radius ratio. It then gets a start and an end angle from the user to convert the closed `Ellipse` into an elliptical arc.

```
Public Sub TestAddEllipse()
Dim dblCenter(0 To 2) As Double
Dim dblMajor(0 To 2) As Double
Dim dblRatio As Double
Dim dblStart As Double
Dim dblEnd As Double
Dim objEnt As AcadEllipse

On Error Resume Next

    '' setup the ellipse parameters
    dblCenter(0) = 0: dblCenter(1) = 0: dblCenter(2) = 0
    dblMajor(0) = 10: dblMajor(1) = 0: dblMajor(2) = 0
    dblRatio = 0.5

    '' draw the ellipse
If ThisDrawing.ActiveSpace = acModelSpace Then
  Set objEnt = ThisDrawing.ModelSpace.AddEllipse(dblCenter, dblMajor, dblRatio)
Else
  Set objEnt = ThisDrawing.PaperSpace.AddEllipse(dblCenter, dblMajor, dblRatio)
End If
    objEnt.Update
    '' get angular input from user
    With ThisDrawing.Utility
        dblStart = .GetAngle(dblCenter, vbCr & "Enter the start angle: ")
        dblEnd = .GetAngle(dblCenter, vbCr & "Enter the end angle: ")
    End With

    '' convert the ellipse into elliptical arc
With objEnt
    .StartAngle = dblStart
    .EndAngle = dblEnd
    .Update
End With
End Sub
```

Line Objects

This section demonstrates how to create the various line-shaped objects.

The `Line` Object

The `AddLine` method creates a `Line` object, which is a single, straight line running between two points:

```
Set LineObject = Object.AddLine(StartPoint, EndPoint)
```

Table 8-5 provides a brief description of each argument.

Table 8-5. `AddLine` *Method Parameters*

Name	Data Type	Description
`StartPoint`	Variant	A three-element array of doubles specifying the line's start in the WCS
`EndPoint`	Variant	A three-element array of doubles specifying the line's end in the WCS

Figure 8-4 illustrates the `Line` object.

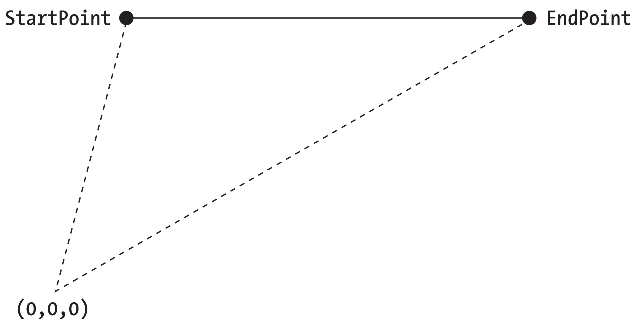


Figure 8-4. *The* `Line` *object*

This example gets a start and end point from the user and then creates a new line:

```
Public Sub TestAddLine()
Dim varStart As Variant
Dim varEnd As Variant
Dim objEnt As AcadLine

On Error Resume Next
    '' get input from user
    With ThisDrawing.Utility
        varStart = .GetPoint(, vbCr & "Pick the start point: ")
        varEnd = .GetPoint(varStart, vbCr & "Pick the end point: ")
    End With

    '' draw the entity
If ThisDrawing.ActiveSpace = acModelSpace Then
  Set objEnt = ThisDrawing.ModelSpace.AddLine(varStart, varEnd)
Else
  Set objEnt = ThisDrawing.PaperSpace.AddLine(varStart, varEnd)
End If
    objEnt.Update
End Sub
```

The `LWPolyline` Object

A *lightweight polyline* is a 2-D line consisting of straight and arced segments. It's functionally similar to the legacy `Polyline` entity, but it's internally represented in a "flat" data structure as opposed to the tree-structured `Polyline` entity. This results in a more compact data size per entity and faster graphics regeneration; hence the "lightweight" name.

You create a new `LWPolyline` object using the `AddLightWeightPolyline` method, as follows:

```
Set LWPolylineObject = Object.AddLightWeightPolyline(Vertices)
```

Table 8-6 provides a brief description of the `Vertices` argument.

Table 8-6. *The* `AddLightWeightPolyline` *Method Parameter*

Name	Data Type	Description
`Vertices`	Variant	An array of doubles specifying a list of 2-D vertex points in WCS coordinates. It's a simple array with a single dimension composed of alternating X and Y values (i.e., `p1x`, `p1y`, `p2x`, `p2y`, etc.). Because you must supply at least two points to create a polyline, this array must have a minimum of four elements.

■**Note** Lightweight polylines don't store Z-axis or elevation information with each vertex point. Instead, the `Polyline` object has an `Elevation` property that specifies the Z elevation relative to the coordinate system for all vertices.

Figure 8-5 illustrates the `LWPolyline` object.

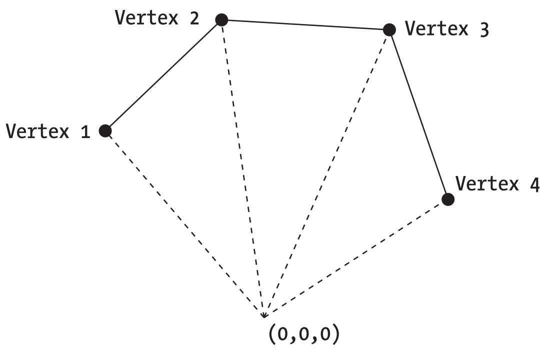


Figure 8-5. *The* `LWPolyline` *object*

When you create an `LWPolyline` object, it's open as shown in Figure 8-5. To close the polyline (i.e., to join the first vertex to the last), set its `Closed` property to `True`. All the segments are straight lines by default. See the section titled "Polyline Arc Segments" later in this chapter for details on how to optionally create arcs.

The following example creates a new lightweight polyline:

```
Public Sub TestAddLWPolyline()
Dim objEnt As AcadLWPolyline
Dim dblVertices() As Double
```

```
    '' setup initial points
    ReDim dblVertices(11)
    dblVertices(0) = 0#: dblVertices(1) = 0#
    dblVertices(2) = 10#: dblVertices(3) = 0#
    dblVertices(4) = 10#: dblVertices(5) = 10#
    dblVertices(6) = 5#: dblVertices(7) = 5#
    dblVertices(8) = 2#: dblVertices(9) = 2#
    dblVertices(10) = 0#: dblVertices(11) = 10#

    '' draw the entity
If ThisDrawing.ActiveSpace = acModelSpace Then
  Set objEnt = ThisDrawing.ModelSpace.AddLightWeightPolyline(dblVertices)
Else
  Set objEnt = ThisDrawing.PaperSpace.AddLightWeightPolyline(dblVertices)
End If
    objEnt.Closed = True
    objEnt.Update
End Sub
```

To add vertices to an `LWPolyline` object one at a time instead of all at once, use the `AddVertex` method. The following example uses this method in a loop, adding vertices to a selected polyline until the user is satisfied. Try using it on the polyline you created in the previous example:

```
Public Sub TestAddVertex()
    On Error Resume Next
    Dim objEnt As AcadLWPolyline
    Dim dblNew(0 To 1) As Double
    Dim lngLastVertex As Long
    Dim varPick As Variant
    Dim varWCS As Variant

    With ThisDrawing.Utility

        '' get entity from user
        .GetEntity objEnt, varPick, vbCr & "Pick a polyline <exit>: "

        '' exit if no pick
        If objEnt Is Nothing Then Exit Sub

        '' exit if not a lwpolyline
        If objEnt.ObjectName <> "AcDbPolyline" Then
            MsgBox "You did not pick a polyline"
            Exit Sub
        End If
        '' copy last vertex of pline into pickpoint to begin loop
        ReDim varPick(2)
        varPick(0) = objEnt.Coordinates(UBound(objEnt.Coordinates) - 1)
        varPick(1) = objEnt.Coordinates(UBound(objEnt.Coordinates))
        varPick(2) = 0

        '' append vertexes in a loop
        Do
            '' translate picked point to UCS for basepoint below
            varWCS = .TranslateCoordinates(varPick, acWorld, acUCS, True)

            '' get user point for new vertex, use last pick as basepoint
            varPick = .GetPoint(varWCS, vbCr & "Pick another point <exit>: ")

            '' exit loop if no point picked
            If Err Then Exit Do

            '' copy picked point X and Y into new 2d point
            dblNew(0) = varPick(0):  dblNew(1) = varPick(1)

            '' get last vertex offset.  it is one half the array size
            lngLastVertex = (UBound(objEnt.Coordinates) + 1) / 2

            '' add new vertex to pline at last offset
            objEnt.AddVertex lngLastVertex, dblNew
        Loop
    End With
    objEnt.Update
End Sub
```

The `MLine` Object

The `MLine` object is a single graphic entity that consists of multiple parallel straight-line segments. The maximum number of parallel lines is 16. You use the `AddMLine` method to create a new `MLine` object:

```
Set MLineObject = Object.AddMLine(Vertices)
```

Table 8-7 provides a brief description of the `Vertices` argument.

Table 8-7. *The* `AddMLine` *Method Parameter*

Name	Data Type	Description
`Vertices`	Variant	An array of doubles specifying a list of 3-D vertex points in WCS coordinates. It's a simple array with a single dimension composed of alternating X, Y, and Z values (i.e., `p1x`, `p1y`, `p1z`, `p2x`, `p2y`, `p2z`, etc.). Because you must supply at least two points to create an `MLine`, this array must have a minimum of six elements.

Although an MLine object consists of multiple parallel lines, the AutoCAD Automation model doesn't expose those lines to VBA. Similarly, MLine styles aren't directly accessible from VBA. However, you have a limited amount of control when you use the CMLJUST, CMLSCALE, and CMLSTYLE system variables to control the justification, scale, and style of newly created MLines.

■**Note** You may create or load additional multiline styles using the MLSTYLE command.

The following example illustrates the AddMLine method:

```
Public Sub TestAddMLine()
Dim objEnt As AcadMLine
Dim dblVertices(17) As Double

    '' setup initial points
    dblVertices(0) = 0: dblVertices(1) = 0: dblVertices(2) = 0
    dblVertices(3) = 10: dblVertices(4) = 0: dblVertices(5) = 0
    dblVertices(6) = 10: dblVertices(7) = 10: dblVertices(8) = 0
    dblVertices(9) = 5: dblVertices(10) = 10: dblVertices(11) = 0
    dblVertices(12) = 5: dblVertices(13) = 5: dblVertices(14) = 0
    dblVertices(15) = 0: dblVertices(16) = 5: dblVertices(17) = 0

    '' draw the entity
If ThisDrawing.ActiveSpace = acModelSpace Then
  Set objEnt = ThisDrawing.ModelSpace.AddMLine(dblVertices)
Else
  Set objEnt = ThisDrawing.PaperSpace.AddMLine(dblVertices)
End If
    objEnt.Update
End Sub
```

The Polyline Object

The Polyline object is similar in function to the LWPolyline object, but it's stored in an alternate format. It's a 2-D or 3-D line consisting of straight and arced segments, but the segments are stored as distinct entities. A number of different polyline types are available; the following list illustrates their AutoCAD command-line equivalent and a description of each type:

```
    PLINE = 2D polyline (AcadPolyline)

    3DPOLY = 3D polyline (Acad3DPolyline)

    3DMESH (also EDGESURF, RULESURF, TABSURF, etc.) = N x M 3D mesh
    (AcadPolygonMesh)

    PFACE = Irregular 3D mesh (AcadPolyfaceMesh)
```

Each segment in an AcadPolyline object has a start point, an end point, and several other unique properties. This object is less efficient than the lightweight polyline object, in which segments are defined as a collection of vertex points.

Use the AddPolyline method to create a Polyline defined by a set of vertices, as follows:

```
Set PolylineObject = Object.AddPolyline(Vertices)
```

Table 8-8 provides a brief description of the Vertices argument.

Table 8-8. *The* AddPolyline *Method Parameter*

Name	Data Type	Description
Vertices	Variant	An array of doubles specifying a list of 3-D vertex points in WCS coordinates. It's a simple array with a single dimension composed of alternating X, Y, and Z values (i.e., p1x, p1y, p1z, p2x, p2y, p2z, etc.). Because you must supply at least two points to create a polyline, this array must have a minimum of six elements.

The following example illustrates the AddPolyline method and shows how to close the polyline and set its type:

```
Public Sub TestAddPolyline()
Dim objEnt As AcadPolyline
Dim dblVertices(17) As Double

    '' setup initial points
    dblVertices(0) = 0: dblVertices(1) = 0: dblVertices(2) = 0
    dblVertices(3) = 10: dblVertices(4) = 0: dblVertices(5) = 0
    dblVertices(6) = 7: dblVertices(7) = 10: dblVertices(8) = 0
    dblVertices(9) = 5: dblVertices(10) = 7: dblVertices(11) = 0
    dblVertices(12) = 6: dblVertices(13) = 2: dblVertices(14) = 0
    dblVertices(15) = 0: dblVertices(16) = 4: dblVertices(17) = 0

    '' draw the entity
If ThisDrawing.ActiveSpace = acModelSpace Then
  Set objEnt = ThisDrawing.ModelSpace.AddPolyline(dblVertices)
Else
  Set objEnt = ThisDrawing.PaperSpace.AddPolyline(dblVertices)
End If
    objEnt.Type = acFitCurvePoly
    objEnt.Closed = True
    objEnt.Update
End Sub
```

AutoCAD is set to draw lightweight polylines by default because of their efficiency; you may change this default setting using the PLINETYPE system variable. You can convert existing polylines from lightweight to heavy and vice versa by using the Convertpoly command.

Although it's less efficient for simple polyline representations, the Polyline object has features not present in the lightweight form, such as the capability to represent splines and other smoothly curved shapes. The following example allows the user to change a specified Polyline curve type from the command line. Try it on the polyline you created in the preceding example:

```
Public Sub TestPolylineType()
Dim objEnt As AcadPolyline
Dim varPick As Variant
Dim strType As String
Dim intType As Integer

On Error Resume Next

    With ThisDrawing.Utility
        .GetEntity objEnt, varPick, vbCr & "Pick a polyline: "
        If Err Then
            MsgBox "That is not a Polyline"
            Exit Sub
        End If

        .InitializeUserInput 1, "Simple Fit Quad Cubic"
        strType = .GetKeyword(vbCr & "Change type [Simple/Fit/Quad/Cubic]: ")

        Select Case strType
            Case "Simple": intType = acSimplePoly
            Case "Fit": intType = acFitCurvePoly
            Case "Quad": intType = acQuadSplinePoly
            Case "Cubic": intType = acCubicSplinePoly
        End Select
    End With

    objEnt.Type = intType
    objEnt.Closed = True
    objEnt.Update
End Sub
```

Polyline Arc Segments

When you first create them, both the LWPolyline and Polyline objects consist of only straight-line segments. Each polyline segment has an arc *bulge factor* that determines the segment's arc radius. By default, all segments have a bulge factor of 0, which is a straight line. A positive bulge factor denotes a counterclockwise arc relative to the start and end points of the polyline segment, and a negative bulge factor denotes a clockwise arc.

You derive the bulge factor from the arc radius by calculating the tangent of one-fourth of the included angle between the polyline segment's start and end points, as shown in Figure 8-6.

Because AutoCAD is already storing the segment points, a bulge factor is a very efficient way to capture the arc radius. Further, because the bulge factor is calculated using the segment's start and end points, the resulting arc is automatically scaled as the distance between these points changes (e.g., when the user scales or grip-edits the polyline).

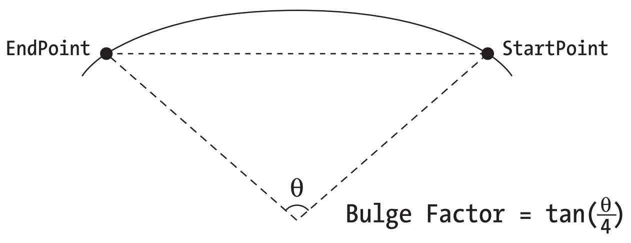


Figure 8-6. *A polyline segment*

To assign a bulge factor to the polyline segments, use the SetBulge method. The following example sets the bulge on a Polyline segment:

```
Public Sub TestAddBulge()
Dim objEnt As AcadPolyline
Dim dblVertices(17) As Double

    '' setup initial points
    dblVertices(0) = 0: dblVertices(1) = 0: dblVertices(2) = 0
    dblVertices(3) = 10: dblVertices(4) = 0: dblVertices(5) = 0
    dblVertices(6) = 7: dblVertices(7) = 10: dblVertices(8) = 0
    dblVertices(9) = 5: dblVertices(10) = 7: dblVertices(11) = 0
    dblVertices(12) = 6: dblVertices(13) = 2: dblVertices(14) = 0
    dblVertices(15) = 0: dblVertices(16) = 4: dblVertices(17) = 0

    '' draw the entity
If ThisDrawing.ActiveSpace = acModelSpace Then
  Set objEnt = ThisDrawing.ModelSpace.AddPolyline(dblVertices)
Else
  Set objEnt = ThisDrawing.PaperSpace.AddPolyline(dblVertices)
End If
    objEnt.Type = acSimplePoly
    'add bulge to the fourth segment
    objEnt.SetBulge 3, 0.5
    objEnt.Update
End Sub
```

The Ray Object

The Ray object represents a line that extends to infinity in one direction from a specified start point. The AddRay method uses the following syntax to create a new ray:

```
Set RayObject = Object.AddRay(StartPoint, SecondPoint)
```

Table 8-9 provides a brief description of each argument.

Table 8-9. `AddRay` *Method Parameters*

Name	Data Type	Description
`StartPoint`	Variant	A three-element array of doubles that specifies the ray's start with respect to the WCS.
`SecondPoint`	Variant	A three-element array of doubles that specifies a second point through which the ray passes. This point simply determines the ray's direction.

Figure 8-7 illustrates the `Ray` object.

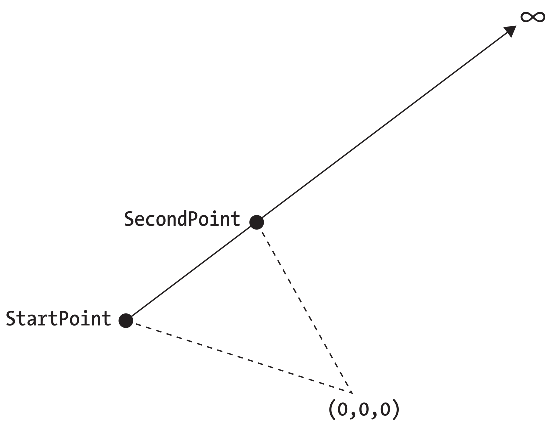


Figure 8-7. *The* `Ray` *object*

The following example uses the `AddRay` method with points obtained from the user:

```
Public Sub TestAddRay()
Dim varStart As Variant
Dim varEnd As Variant
Dim objEnt As AcadRay

On Error Resume Next
    '' get input from user
    With ThisDrawing.Utility
        varStart = .GetPoint(, vbCr & "Pick the start point: ")
        varEnd = .GetPoint(varStart, vbCr & "Indicate a direction: ")
    End With

    '' draw the entity
If ThisDrawing.ActiveSpace = acModelSpace Then
  Set objEnt = ThisDrawing.ModelSpace.AddRay(varStart, varEnd)
Else
  Set objEnt = ThisDrawing.PaperSpace.AddRay(varStart, varEnd)
End If
    objEnt.Update
End Sub
```

The Spline Object

The Spline object represents a nonuniform rational B-spline (NURBS) quadratic or cubic curve. You use the AddSpline method to create a new Spline object:

```
Set SplineObject = Object.AddSpline(FitPoints, StartTangent, EndTangent)
```

Table 8-10 provides a brief description of each argument.

Table 8-10. AddSpline *Method Parameters*

Name	Data Type	Description
FitPoints	Variant	An array of doubles specifying a list of 3-D fit points in WCS coordinates. It's a simple array with a single dimension composed of alternating X, Y, and Z values (i.e., p1x, p1y, p1z, p2x, p2y, p2z, etc.). Because you must supply at least two points to create a spline, this array must have a minimum of six elements.
StartTangent	Variant	A three-element array of doubles that determines the tangent of the spline at its first fit point.
EndTangent	Variant	A three-element array of doubles that determines the tangent of the spline at its last fit point.

Figure 8-8 illustrates the Spline object.

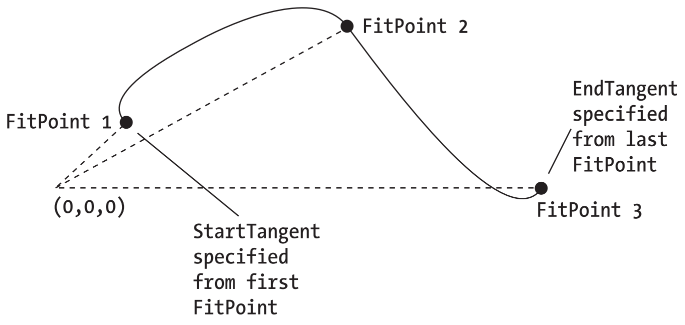

FitPoint 2
EndTangent specified from last FitPoint
FitPoint 1
(0,0,0)
FitPoint 3
StartTangent specified from first FitPoint

Figure 8-8. *The Spline object*

The following code sample demonstrates the AddSpline method:

```
Public Sub TestAddSpline()
Dim objEnt As AcadSpline
Dim dblBegin(0 To 2) As Double
Dim dblEnd(0 To 2) As Double
Dim dblPoints(14) As Double

    '' set tangencies
    dblBegin(0) = 1.5:   dblBegin(1) = 0#:   dblBegin(2) = 0
    dblEnd(0) = 1.5:  dblEnd(1) = 0#:  dblEnd(2) = 0
    '' set the fit dblPoints
    dblPoints(0) = 0: dblPoints(1) = 0: dblPoints(2) = 0
    dblPoints(3) = 3: dblPoints(4) = 5: dblPoints(5) = 0
    dblPoints(6) = 5: dblPoints(7) = 0: dblPoints(8) = 0
    dblPoints(9) = 7: dblPoints(10) = -5: dblPoints(11) = 0
    dblPoints(12) = 10: dblPoints(13) = 0: dblPoints(14) = 0

    '' draw the entity
If ThisDrawing.ActiveSpace = acModelSpace Then
  Set objEnt = ThisDrawing.ModelSpace.AddSpline(dblPoints, dblBegin, dblEnd)
Else
  Set objEnt = ThisDrawing.PaperSpace.AddSpline(dblPoints, dblBegin, dblEnd)
End If
    objEnt.Update
End Sub
```

The XLine Object

The XLine object represents a straight line that passes through two specified points and extends to infinity in both directions. You use the AddXline method to create a new XLine object:

```
Set XLineObject = Object.AddXline(FirstPoint, SecondPoint)
```

Table 8-11 provides a brief description of each argument.

Table 8-11. AddXline *Method Parameters*

Name	Data Type	Description
FirstPoint	Variant	A three-element array of doubles specifying a point through which the XLine object passes in WCS coordinates.
SecondPoint	Variant	A three-element array of doubles specifying a second point through which the XLine passes. This point simply determines the direction of the XLine.

Figure 8-9 illustrates the Xline object.

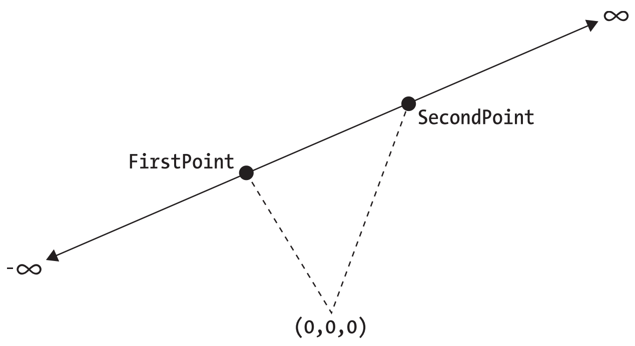


Figure 8-9. *The Xline object*

The following example illustrates how to implement the `AddXline` method:

```
Public Sub TestAddXline()
Dim varStart As Variant
Dim varEnd As Variant
Dim objEnt As AcadXline

On Error Resume Next
    '' get input from user
    With ThisDrawing.Utility
        varStart = .GetPoint(, vbCr & "Pick the start point: ")
        varEnd = .GetPoint(varStart, vbCr & "Indicate an angle: ")
    End With

    '' draw the entity
If ThisDrawing.ActiveSpace = acModelSpace Then
  Set objEnt = ThisDrawing.ModelSpace.AddXline(varStart, varEnd)
Else
  Set objEnt = ThisDrawing.PaperSpace.AddXline(varStart, varEnd)
End If
    objEnt.Update
End Sub
```

Other Objects of Interest

The following sections describe some other objects of interest, starting with the `Hatch` object.

The Hatch Object

The `Hatch` object represents an area filled with a pattern. The pattern may be associated with the area boundary (i.e., if the boundary changes, the pattern also changes appropriately), or the pattern may be independent of the area boundary. The read-only `AssociativeHatch` property determines whether the pattern is associated with the `Hatch` object. You set this property when you create the `Hatch` object, using the `AddHatch` method:

```
Set HatchObject = Object.AddHatch(PatternType, PatternName, Associativity)
```

Table 8-12 provides a brief description of each argument.

Table 8-12. `AddHatch` *Method Parameters*

Name	Data Type	Description
`PatternType`	Long	Specifies the type of pattern to use. It may take one of the three possible values given in Table 8-13.
`PatternName`	String	Specifies which hatch pattern to use from the `.pat` file.
`Associativity`	Boolean	Determines whether the hatch is associative. Set this parameter to `True` to create an associative hatch.

Table 8-13 shows the three possible values for the `PatternType` parameter.

Table 8-13. `PatternType` *Parameter Values*

Constant	Value	Description
`AcHatchPatternTypeUserDefined`	0	Allows you to define a pattern of lines using the current linetype
`AcHatchPatternTypePredefined`	1	Uses a pattern name from those defined in the `acad.pat` file
`AcHatchPatternTypeCustomDefined`	2	Uses a pattern name from those defined in a `.pat` file other than the `acad.pat` file

After you create the `Hatch` object, you must specify the boundary or loop using the `AppendOuterLoop` method. You must close and add this loop before any inner loops. You create any inner loops you may need one at a time using the `AppendInnerLoop` method.

■ **Caution** The documentation for the `AddHatch` method warns that you must call `AppendOuterLoop` immediately after you add the `Hatch` object; otherwise, AutoCAD will enter an unpredictable state.

The following code gets a center point and radius from the user, and then creates some entities using these inputs. The new entities are then used to specify the inner and outer boundaries for a new `Hatch` object. The resulting `Hatch` object is shown in Figure 8-10.

```
Public Sub TestAddHatch()
Dim varCenter As Variant
Dim dblRadius As Double
Dim dblAngle As Double
Dim objEnt As AcadHatch
Dim varOuter() As AcadEntity
Dim varInner() As AcadEntity

On Error Resume Next
    '' get input from user
    With ThisDrawing.Utility
        varCenter = .GetPoint(, vbCr & "Pick the center point: ")
        dblRadius = .GetDistance(varCenter, vbCr & "Indicate the radius: ")
        dblAngle = .AngleToReal("180", acDegrees)
    End With

    '' draw the entities    With ThisDrawing.ModelSpace

        '' draw the Outer loop (circle)
        ReDim varOuter(0)
        Set varOuter(0) = .AddCircle(varCenter, dblRadius)
        '' draw then Inner loop (semicircle)
        ReDim varInner(1)
        Set varInner(0) = .AddArc(varCenter, dblRadius * 0.5, 0, dblAngle)
        Set varInner(1) = .AddLine(varInner(0).StartPoint, _
                                   varInner(0).EndPoint)

        '' create the Hatch object
        Set objEnt = .AddHatch(acHatchPatternTypePreDefined, "ANSI31", True)

        '' append boundaries to the hatch
        objEnt.AppendOuterLoop varOuter
        objEnt.AppendInnerLoop varInner

        '' evaluate and display hatched boundaries
        objEnt.Evaluate
        objEnt.Update
    End With
End Sub
```

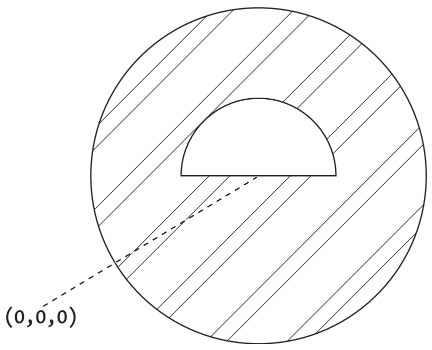


Figure 8-10. *The* `Hatch` *object*

As you can see, the `AppendOuterLoop` and `AppendInnerLoop` methods take as a parameter an array of objects forming a closed loop. The `Evaluate` method of the `Hatch` object evaluates the lines or fill for the hatch pattern.

■ **Note** In AutoCAD 2004, `Hatch` objects support a gradient fill display behavior. This in turn provides several new properties, such as `GradientAngle`, `GradientCentered`, `GradientColor1`, and `GradientColor2`. Consult the AutoCAD 2004 Customization Guide for more information about new `Hatch` object features.

The MText Object

The MText object represents a paragraph of alphanumeric characters that fits within a non-printing bounding box. Each MText object is treated as a single object, regardless of the number of lines of text it contains. The bounding box remains an integral part of the entity, even though it isn't plotted or printed. MText objects use word wrap to break long lines into paragraphs.

You can create an MText entity using the AddMText method:

```
Set MTextObject = Object.AddMText(InsertionPoint, Width, TextString)
```

Table 8-14 provides a brief description of each argument.

Table 8-14. AddMText *Method Parameters*

Name	Data Type	Description
InsertionPoint	Variant	A three-element array of doubles specifying the point at which the MText bounding box will be inserted with respect to the WCS
Width	Double	The width of the text block
TextString	String	The text displayed in the drawing space

Figure 8-11 illustrates the MText object.

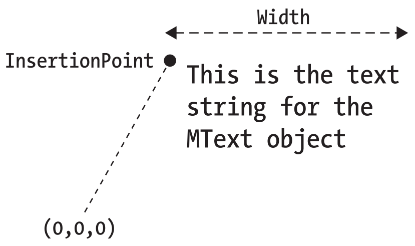


Figure 8-11. *The* MText *object*

The following example demonstrates the AddMText method:

```
Public Sub TestAddMText()
Dim varStart As Variant
Dim dblWidth As Double
Dim strText As String
Dim objEnt As AcadMText

On Error Resume Next
    '' get input from user
    With ThisDrawing.Utility
        varStart = .GetPoint(, vbCr & "Pick the start point: ")
        dblWidth = .GetDistance(varStart, vbCr & "Indicate the width: ")
        strText = .GetString(True, vbCr & "Enter the text: ")
    End With
    '' add font and size formatting
    strText = "\Fromand.shx;\H0.5;" & strText

    '' create the mtext

    Set objEnt = ThisDrawing.ModelSpace.AddMText(varStart, dblWidth, strText)
    objEnt.Update
End Sub
```

Rich text format (RTF) control codes aren't recognized in an MText entity. Text from other programs with embedded formatting will lose that formatting when you import it into an MText-style paragraph if you're working with AutoCAD 2000, 2000i, or 2002. Formatting is well preserved within AutoCAD 2004 and higher, however. You can paste Microsoft Word documents directly into AutoCAD 2002 and higher as MText and maintain their original formatting.

AutoCAD 2000, 2000i, and 2002 exhibit odd behavior with respect to exploding block insertions that contain MText entities. In these versions, the nested MText entities are automatically exploded into individual Text entities when you use the VBA Explode method. This doesn't occur in AutoCAD 2004 and higher, however, and MText is preserved after the Explode method is executed. Text styles may be applied similarly to normal Text objects.

2005 and 2006 allows you to retrieve the field codes used to display special characters within the text string. However, the special codes inserted using an AutoCAD command may not be retrieved. A good rule of thumb is, if you can add a code programmatically, then you can retrieve it as well.

The Point Object

Point objects can act as nodes to which you can snap objects. You can specify a full 3-D WCS location for a point. You use the AddPoint method to create a new Point object, as follows:

```
Set PointObject = Object.AddPoint(PointPosition)
```

Table 8-15 provides a brief description of the PointPosition argument.

Table 8-15. AddPoint *Method Parameter*

Name	Data Type	Description
PointPosition	Variant	A three-element array of doubles specifying the point's WCS coordinates

Controlling How a Point Object Is Displayed

The appearance of Point objects is controlled by the system variables PDMODE and PDSIZE. The PDMODE variable specifies the figure used to represent a point. This variable may take a value from 0 to 4, as Figure 8-12 illustrates. A value of 1 means that nothing is displayed.

You can combine the PDMODE variable value with the number 32, 64, or 96, which represents a surrounding shape. Figure 8-13 shows the possible combined symbols for a Point object.

The PDSIZE variable controls the size of the Point object figure. It has no effect if you set PDMODE to 0 or 1. You may set PDSIZE to a positive, negative, or zero value:

- A positive value specifies an absolute size for the point in drawing units.
- A negative value specifies the point symbol size as a percentage of the viewport area.
- A zero setting will generate the points at five percent of the height of the graphics area.

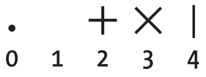


Figure 8-12. *The* `Point` *object*

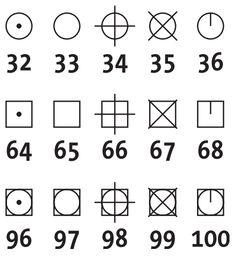


Figure 8-13. *Various point styles*

When you regenerate the drawing, AutoCAD recalculates the size and symbol of all points. So if you change `PDMODE` and `PDSIZE`, the appearance of existing points doesn't change until AutoCAD regenerates the drawing. You set the `PDMODE` and `PDSIZE` variables using the `SetVariable` method, which I discuss in Appendix C.

The following example illustrates how to use the `PDMODE` and `PDSIZE` variables and the `AddPoint` method:

```
Public Sub TestAddPoint()
Dim objEnt As AcadPoint
Dim varPick As Variant
Dim strType As String
Dim intType As Integer
Dim dblSize As Double

On Error Resume Next

    With ThisDrawing.Utility
        '' get the pdmode center type
        .InitializeUserInput 1, "Dot None Cross X Tick"
        strType = .GetKeyword(vbCr & "Center type [Dot/None/Cross/X/Tick]: ")
        If Err Then Exit Sub

        Select Case strType
            Case "Dot": intType = 0
            Case "None": intType = 1
            Case "Cross": intType = 2
            Case "X": intType = 3
            Case "Tick": intType = 4
        End Select

        '' get the pdmode surrounding type
        .InitializeUserInput 1, "Circle Square Both"
        strType = .GetKeyword(vbCr & "Outer type [Circle/Square/Both]: ")
        If Err Then Exit Sub
        Select Case strType
            Case "Circle": intType = intType + 32
            Case "Square": intType = intType + 64
            Case "Both": intType = intType + 96
        End Select

        '' get the pdsize
        .InitializeUserInput 1, ""
        dblSize = .GetDistance(, vbCr & "Enter a point size: ")
        If Err Then Exit Sub

        '' set the system varibles
        With ThisDrawing
        .SetVariable "PDMODE", intType
        .SetVariable "PDSIZE", dblSize
        End With
        '' now add points in a loop
        Do
            '' get user point for new vertex, use last pick as basepoint
            varPick = .GetPoint(, vbCr & "Pick a point <exit>: ")

            '' exit loop if no point picked
            If Err Then Exit Do

            '' add new vertex to pline at last offset

            ThisDrawing.ModelSpace.AddPoint varPick
        Loop
    End With
End Sub
```

The Region Object

The `Region` object represents a bounded plane and may consist of `Line`, `Arc`, `Circle`, `Elliptical Arc`, `LightweightPolyline`, and `Spline` objects. To create a `Region` object from a set of drawing entities, use the `AddRegion` method as follows:

```
RegionArray = Object.AddRegion(ObjectsArray)
```

Table 8-16 provides a brief description of the `ObjectsArray` argument.

Table 8-16. `AddRegion` *Method Parameter*

Name	Data Type	Description
`ObjectsArray`	Array of objects	The objects used to make the region objects. These objects must form a closed area and must be coplanar.

`AddRegion` returns an array of the `Region` objects created from the `ObjectsArray` parameter, as shown in Figure 8-14.

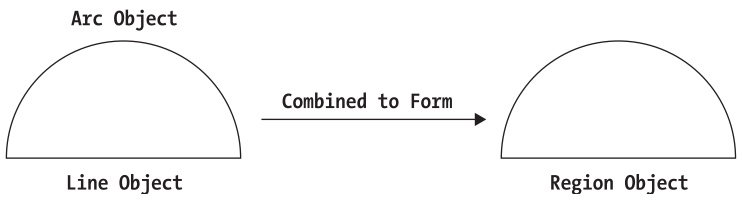


Figure 8-14. Region *objects*

The ObjectsArray may contain drawing objects that form more than one enclosed plane. For each enclosed plane, a Region object is formed and held in an array of Region objects. If two or more curves share end points, the region created may be arbitrary.

The following example demonstrates how to build and manipulate regions. The objEnts variable is an array of objects that forms two closed coplanar faces to be converted into Region objects. One region is a full circle and the other is a closed semicircle. Once the AddRegion method is executed, these two regions are held in the regions variable. Next, the code uses the Boolean method (detailed in Chapter 9) to subtract the semicircle from the larger full circle. Finally, the region is moved to a new location, showing that it's a single composite object.

```
Public Sub TestAddRegion()
Dim varCenter As Variant
Dim varMove As Variant
Dim dblRadius As Double
Dim dblAngle As Double
Dim varRegions As Variant
Dim objEnts() As AcadEntity

    On Error Resume Next
    '' get input from user
    With ThisDrawing.Utility
        varCenter = .GetPoint(, vbCr & "Pick the center point: ")
        dblRadius = .GetDistance(varCenter, vbCr & "Indicate the radius: ")
        dblAngle = .AngleToReal("180", acDegrees)
    End With

    '' draw the entities
    With ThisDrawing.ModelSpace

        '' draw the outer region (circle)
        ReDim objEnts(2)
        Set objEnts(0) = .AddCircle(varCenter, dblRadius)

        '' draw the inner region (semicircle)
        Set objEnts(1) = .AddArc(varCenter, dblRadius * 0.5, 0, dblAngle)
        Set objEnts(2) = .AddLine(objEnts(1).StartPoint, objEnts(1).EndPoint)
        '' create the regions
        varRegions = .AddRegion(objEnts)
    End With

    '' get new position from user
        varMove = ThisDrawing.Utility.GetPoint(varCenter, vbCr & _
                                               "Pick a new location: ")

    '' subtract the inner region from the outer
    varRegions(1).Boolean acSubtraction, varRegions(0)

    '' move the composite region to a new location
    varRegions(1).Move varCenter, varMove
End Sub
```

You may also explode individual regions by using the Explode method. Be aware that invoking the Region entity incurs a performance hit while AutoCAD loads the ACIS or ShapeManager solid-modeling engine, depending upon what version of AutoCAD you're working with.

The Solid Object

The Solid object represents a 2-D polygon, not to be confused with the 3-D solids described in Chapter 9. You can create a polygon by specifying two points to define the first edge. The third point defines an edge back to the start point, not to the second point as you might imagine. Next, you have the option to define a fourth point in the same position as the third, which results in a triangular shape. Alternatively, you can specify a different fourth point to define a variety of shapes (e.g., a rectangle, a parallelogram, a trapezoid, a chevron, or a bow tie). Use the AddSolid method as follows:

```
Set SolidObject = Object.AddSolid(Point1, Point2, Point3, Point4)
```

Table 8-17 provides a brief description of each argument.

Table 8-17. AddSolid *Method Parameters*

Name	Data Type	Description
Point1	Variant	A three-element array of doubles defining the start of the polygon's first side in WCS coordinates.
Point2	Variant	A three-element array of doubles defining the end of the polygon's first side in WCS coordinates.
Point3	Variant	A three-element array of doubles defining the end of the polygon's second side in WCS coordinates. This point is opposite Point1 on the second edge.
Point4	Variant	A three-element array of doubles specifying the last point that defines the polygon's third and optionally fourth edges in WCS coordinates. It may be the same as Point3, in which case only three polygon edges are visible.

If the FILLMODE system variable is set to 1, the shape will be shaded; otherwise, only the boundary lines are visible. The following example uses AddSolid with input from the user:

```
Public Sub TestAddSolid()
Dim varP1 As Variant
Dim varP2 As Variant
Dim varP3 As Variant
Dim varP4 As Variant
Dim objEnt As AcadSolid

On Error Resume Next
    '' ensure that solid fill is enabled
    ThisDrawing.SetVariable "FILLMODE", 1

    '' get input from user
    With ThisDrawing.Utility
        varP1 = .GetPoint(, vbCr & "Pick the start point: ")
        varP2 = .GetPoint(varP1, vbCr & "Pick the second point: ")
        varP3 = .GetPoint(varP1, vbCr & "Pick a point opposite the start: ")
        varP4 = .GetPoint(varP3, vbCr & "Pick the last point: ")
    End With

    '' draw the entity

    Set objEnt = ThisDrawing.ModelSpace.AddSolid(varP1, varP2, varP3, varP4)
    objEnt.Update
End Sub
```

The Text Object

The Text object represents text on a drawing. It differs from the MText object in that it consists of only one line of text. It also lacks the embedded formatting capabilities of MText. The AddText method creates a new Text object:

```
Set TextObject = Object.AddText(TextString, InsertionPoint, Height)
```

Table 8-18 provides a brief description of each argument.

Table 8-18. AddText *Method Parameters*

Name	Data Type	Description
TextString	String	The text to be displayed in the drawing
InsertionPoint	Variant	A three-element array of doubles representing the text's position with respect to the WCS
Height	Double	A positive number indicating the text's height

The following code example shows the AddText method in practice:

```
Public Sub TestAddText()
Dim varStart As Variant
Dim dblHeight As Double
Dim strText As String
Dim objEnt As AcadText

On Error Resume Next
    '' get input from user
    With ThisDrawing.Utility
        varStart = .GetPoint(, vbCr & "Pick the start point: ")
        dblHeight = .GetDistance(varStart, vbCr & "Indicate the height: ")
        strText = .GetString(True, vbCr & "Enter the text: ")
    End With

    '' create the text
If ThisDrawing.ActiveSpace = acModelSpace Then
  Set objEnt = ThisDrawing.ModelSpace.AddText(strText, varStart, dblHeight)
Else
  Set objEnt = ThisDrawing.PaperSpace.AddText(strText, varStart, dblHeight)
End If
    objEnt.Update
End Sub
```

Summary

In this chapter you learned how to create a number of different 2-D and 3-D drawing objects. In the next chapter you'll learn how to create a variety of 3-D solid objects.

CHAPTER 9

Creating 3-D Objects

In Chapter 8 you learned how to create simple 2-D drawing entities, such as circles, lines, and text. AutoCAD also provides you with a number of methods to programmatically create complex 3-D solid entities. Represented by the `3DSolid` object, each solid entity and the creation of this object in its various disguises will form the basis of this chapter.

This chapter covers the following topics:

- 3-D primitives (the box, cone, cylinder, sphere, torus, and wedge objects)
- Elliptical cones and cylinders
- Extruded and revolved solids
- Manipulation of solids (slicing, checking interference, and spatial properties)

Understanding the `3DSolid` Object

The `3DSolid` object represents a solid with a free-form surface. This means the solid may take virtually any shape. You may add a `3DSolid` object to a drawing or block by using one of the `AddXXX` methods I describe in this chapter.

Although you can do amazing things with the ActiveX solids and surfacing features in AutoCAD, they don't provide all the functionality that AutoCAD can harness. Some examples of this functionality include `SOLIDEDIT` operations such as Shell, Imprint, Copy Faces, Taper Faces, Clean, and Check.

This limitation is because of the manner in which solids and surfaces are constructed and managed using an external modeling kernel. Up until AutoCAD 2002, the solid modeling kernel was ACIS by Spatial (`http://www.spatial.com`). With AutoCAD 2004 and later, however, Autodesk has parted ways with ACIS and uses its own modeling technology, ShapeManager, which was adapted from ACIS. Developing applications that work with ACIS objects internally requires a separate developer's license with Spatial. Autodesk is developing an API for ShapeManager, but so far it is built on ObjectARX only.

Methods are exposed by the `ModelSpace`, `PaperSpace`, and `Block` objects to create each of the 3-D solid objects. When I present the syntax for each method, I use the word *Object* to refer to any of these objects. For example:

```
Set 3DSolidObject = Object.AddBox(BoxCenter, Length, Width, Height)
```

Because you may combine solids to form sophisticated geometric shapes, they have no characteristic start or end points. Instead, you specify solids through a control point, the

Centroid, that sits at the center of a bounding box. No matter how complex the solid becomes, the Centroid point can quickly be calculated. Many of the examples in this chapter demonstrate how to convert user input for each primitive, such as the corner of a box, into the Centroid point specification.

In the examples that follow, you'll change the angle at which the drawing is displayed so that you can view the newly created objects unambiguously. You achieve this by changing the Direction property of the active ViewPort object. You use the following function to consolidate this operation:

```
Sub SetViewpoint(Optional Zoom As Boolean = False, _
                 Optional X As Double = 1, _
                 Optional Y As Double = -2, _
                 Optional Z As Double = 1)

    Dim dblDirection(2) As Double
    dblDirection(0) = X:  dblDirection(1) = Y:  dblDirection(2) = Z

    With ThisDrawing
        .Preferences.ContourLinesPerSurface = 10    ' set surface countours
        .ActiveViewport.Direction = dblDirection    ' assign new direction
        .ActiveViewport = .ActiveViewport           ' force a viewport update
        If Zoom Then .Application.ZoomAll           ' zoomall if requested
    End With
End Sub
```

Notice that the ContourLinesPerSurface property of the drawing Preferences object is set to 10. This setting controls the number of surface lines AutoCAD shows on each surface. You can set a higher number with a resulting increase in surface lines.

Creating Simple Solid Objects

This section demonstrates how to create the various 3-D solid objects.

The Box

A box is a 3DSolid object with edges parallel to the axes of the WCS. You create it using the AddBox method:

```
Set 3DSolidObject = Object.AddBox(BoxCenter, Length, Width, Height)
```

Table 9-1 presents the AddBox method's parameters.

Table 9-1. *The* `AddBox` *Method's Parameters*

Name	Data Type	Description
`BoxCenter`	Variant	A three-element array of doubles specifying the center of the box in WCS coordinates
`Length`	Double	A positive value representing the length of the box
`Width`	Double	A positive value representing the width of the box
`Height`	Double	A positive value representing the height of the box

Figure 9-1 shows an example of a box in AutoCAD.

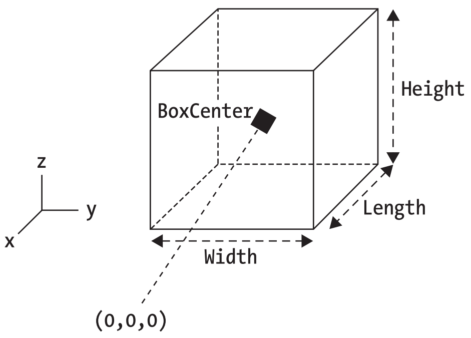


Figure 9-1. *A box*

The following code creates a box based on user input:

```
Public Sub TestAddBox()
      Dim varPick As Variant
      Dim dblLength As Double
      Dim dblWidth As Double
      Dim dblHeight As Double
      Dim dblCenter(2) As Double
      Dim objEnt As Acad3DSolid

    '' set the default viewpoint
    SetViewpoint Zoom:=True

    '' get input from user
    With ThisDrawing.Utility
        .InitializeUserInput 1
        varPick = .GetPoint(, vbCr & "Pick a corner point: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblLength = .GetDistance(varPick, vbCr & "Enter the X length: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblWidth = .GetDistance(varPick, vbCr & "Enter the Y width: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblHeight = .GetDistance(varPick, vbCr & "Enter the Z height: ")
    End With
    '' calculate center point from input
    dblCenter(0) = varPick(0) + (dblLength / 2)
    dblCenter(1) = varPick(1) + (dblWidth / 2)
    dblCenter(2) = varPick(2) + (dblHeight / 2)

    '' draw the entity
    Set objEnt = ThisDrawing.ModelSpace.AddBox(dblCenter, dblLength, _
                                               dblWidth, dblHeight)
    objEnt.Update
    ThisDrawing.SendCommand "_shade" & vbCr
End Sub
```

The Cone

You create a `3DSolid` object in the form of a cone using the `AddCone` method. The base of the cone is parallel with the XY plane of the WCS.

```
Set 3DSolidObject = Object.AddCone(ConeCenter, BaseRadius, Height)
```

Table 9-2 presents the `AddCone` method's parameters, and Figure 9-2 shows an example of a cone in AutoCAD.

Table 9-2. *The* `AddCone` *Method's Parameters*

Name	Data Type	Description
`ConeCenter`	Variant	A three-element array of doubles specifying the cone's center in WCS coordinates
`BaseRadius`	Double	A positive value representing the radius of the cone's circular base
`Height`	Double	A positive value representing the cone's height

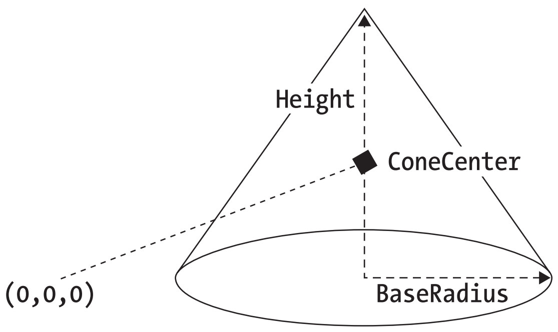


Figure 9-2. *A cone*

This example creates a cone based on user input:

```
Public Sub TestAddCone()
      Dim varPick As Variant
      Dim dblRadius As Double
      Dim dblHeight As Double
      Dim dblCenter(2) As Double
      Dim objEnt As Acad3DSolid

    '' set the default viewpoint
    SetViewpoint

    '' get input from user
    With ThisDrawing.Utility
        .InitializeUserInput 1
        varPick = .GetPoint(, vbCr & "Pick the base center point: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblRadius = .GetDistance(varPick, vbCr & "Enter the radius: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblHeight = .GetDistance(varPick, vbCr & "Enter the Z height: ")
    End With

    '' calculate center point from input
    dblCenter(0) = varPick(0)
    dblCenter(1) = varPick(1)
    dblCenter(2) = varPick(2) + (dblHeight / 2)

    '' draw the entity
    Set objEnt = ThisDrawing.ModelSpace.AddCone(dblCenter, dblRadius, _
                                                dblHeight)
    objEnt.Update

    ThisDrawing.SendCommand "_shade" & vbCr
End Sub
```

The Cylinder

You add a cylinder to a drawing using the `AddCylinder` method. The base of the cylinder lies in a plane parallel to the WCS XY plane.

```
Set 3DSolidObject = Object.AddCylinder(CylinderCenter, Radius, Height)
```

Table 9-3 presents the `AddCylinder` method's parameters, and Figure 9-3 shows an example of a cylinder in AutoCAD.

Table 9-3. *The* `AddCylinder` *Method's Parameters*

Name	Data Type	Description
`CylinderCenter`	Variant	A three-element array of doubles specifying the cylinder's center in WCS coordinates
`Radius`	Double	A positive value representing the radius of the cylinder's base
`Height`	Double	A positive value representing the cylinder's height

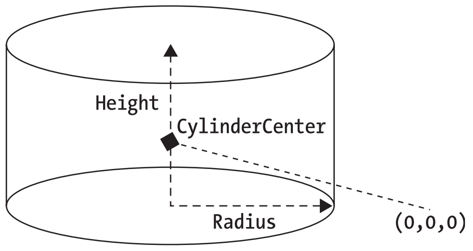


Figure 9-3. *A cylinder*

The following example creates a cylinder based on user input:

```
Public Sub TestAddCylinder()
      Dim varPick As Variant
      Dim dblRadius As Double
      Dim dblHeight As Double
      Dim dblCenter(2) As Double
      Dim objEnt As Acad3DSolid

    '' set the default viewpoint
    SetViewpoint

    '' get input from user
    With ThisDrawing.Utility
        .InitializeUserInput 1
        varPick = .GetPoint(, vbCr & "Pick the base center point: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblRadius = .GetDistance(varPick, vbCr & "Enter the radius: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblHeight = .GetDistance(varPick, vbCr & "Enter the Z height: ")
    End With

    '' calculate center point from input
    dblCenter(0) = varPick(0)
    dblCenter(1) = varPick(1)
    dblCenter(2) = varPick(2) + (dblHeight / 2)
    '' draw the entity
    Set objEnt = ThisDrawing.ModelSpace.AddCylinder(dblCenter, dblRadius, dblHeight)
    objEnt.Update

    ThisDrawing.SendCommand "_shade" & vbCr
End Sub
```

The Sphere

You use the `AddSphere` method to create a `3DSolid` object representing a sphere and to add it to a drawing.

```
Set 3DSolidObject = Object.AddSphere(SphereCenter, Radius)
```

Table 9-4 presents the `AddSphere` method's parameters, and Figure 9-4 shows an example of a sphere in AutoCAD.

Table 9-4. *The* `AddSphere` *Method's Parameters*

Name	Data Type	Description
`SphereCenter`	Variant	A three-element array of doubles specifying the sphere's center in WCS coordinates
`Radius`	Double	A positive value representing the sphere's radius

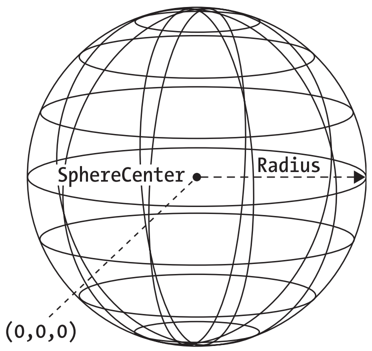


Figure 9-4. *A sphere*

This example creates a sphere based on user input:

```
Public Sub TestAddSphere()
      Dim varPick As Variant
      Dim dblRadius As Double
      Dim objEnt As Acad3DSolid

    '' set the default viewpoint
    SetViewpoint
    '' get input from user
    With ThisDrawing.Utility
        .InitializeUserInput 1
        varPick = .GetPoint(, vbCr & "Pick the center point: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblRadius = .GetDistance(varPick, vbCr & "Enter the radius: ")
    End With

    '' draw the entity
    Set objEnt = ThisDrawing.ModelSpace.AddSphere(varPick, dblRadius)
    objEnt.Update

    ThisDrawing.SendCommand "_shade" & vbCr
End Sub
```

The Torus

Use the `AddTorus` method to create a torus and add it at a given point in a drawing, such that the ring lays flat in the XY plane.

```
Set 3DSolidObject = Object.AddTorus(TorusCenter, TorusRadius, TubeRadius)
```

Table 9-5 presents the `AddTorus` method's parameters, and Figure 9-5 shows an example of a torus in AutoCAD.

Table 9-5. *The* `AddTorus` *Method's Parameters*

Name	Data Type	Description
TorusCenter	Variant	A three-element array of doubles specifying the point about which the torus is centered in WCS coordinates
TorusRadius	Double	A positive value representing the distance from the torus's center to the tube's center
TubeRadius	Double	A positive value representing the tube's radius

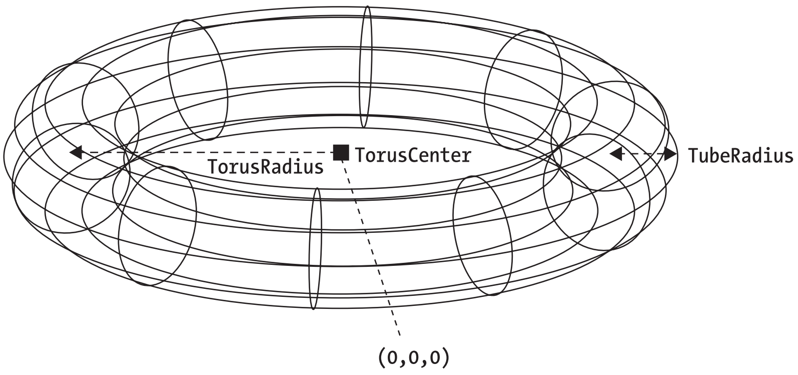


Figure 9-5. *A torus*

The following example creates a torus based on user input:

```
Public Sub TestAddTorus()
      Dim pntPick As Variant
      Dim pntRadius As Variant
      Dim dblRadius As Double
      Dim dblTube As Double
      Dim objEnt As Acad3DSolid
      Dim intI As Integer

    '' set the default viewpoint
    SetViewpoint

    '' get input from user
    With ThisDrawing.Utility
        .InitializeUserInput 1
        pntPick = .GetPoint(, vbCr & "Pick the center point: ")
        .InitializeUserInput 1
        pntRadius = .GetPoint(pntPick, vbCr & "Pick a radius point: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblTube = .GetDistance(pntRadius, vbCr & "Enter the tube radius: ")
    End With

    '' calculate radius from points
    For intI = 0 To 2
        dblRadius = dblRadius + (pntPick(intI) - pntRadius(intI)) ^ 2
    Next
    dblRadius = Sqr(dblRadius)

    '' draw the entity
    Set objEnt = ThisDrawing.ModelSpace.AddTorus(pntPick, dblRadius, dblTube)
    objEnt.Update

    ThisDrawing.SendCommand "_shade" & vbCr
End Sub
```

The Wedge

The `AddWedge` method creates a wedge with edges parallel to the WCS axes.

```
Set 3DSolidObject = Object.AddWedge(FaceCenter, Length, Width, Height)
```

Table 9-6 presents the `AddWedge` method's parameters, and Figure 9-6 shows an example of a wedge in AutoCAD.

Table 9-6. *The* AddWedge *Method's Parameters*

Name	Data Type	Description
FaceCenter4	Variant	A three-element array of doubles specifying the bounding box's center in WCS coordinates. This corresponds to the sloping face's center as well.
Length	Double	A positive value representing the wedge base's length (X-axis).
Width	Double	A positive value representing the wedge base's width (Y-axis).
Height	Double	A positive value representing the wedge base's height (Z-axis).

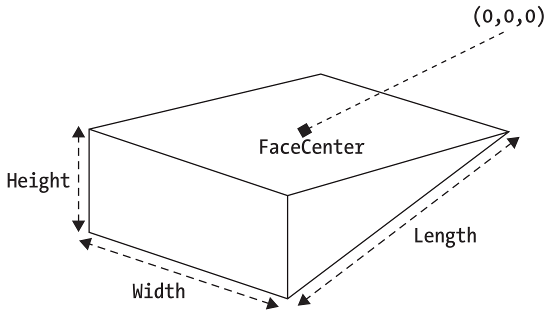


Figure 9-6. *A wedge*

This example creates a wedge based on user input:

```
Public Sub TestAddWedge()
      Dim varPick As Variant
      Dim dblLength As Double
      Dim dblWidth As Double
      Dim dblHeight As Double
      Dim dblCenter(2) As Double
      Dim objEnt As Acad3DSolid

    '' set the default viewpoint
    SetViewpoint

    '' get input from user
    With ThisDrawing.Utility
        .InitializeUserInput 1
        varPick = .GetPoint(, vbCr & "Pick a base corner point: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblLength = .GetDistance(varPick, vbCr & "Enter the base X length: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblWidth = .GetDistance(varPick, vbCr & "Enter the base Y width: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblHeight = .GetDistance(varPick, vbCr & "Enter the base Z height: ")
    End With
    '' calculate center point from input
    dblCenter(0) = varPick(0) + (dblLength / 2)
    dblCenter(1) = varPick(1) + (dblWidth / 2)
    dblCenter(2) = varPick(2) + (dblHeight / 2)

    '' draw the entity
    Set objEnt = ThisDrawing.ModelSpace.AddWedge(dblCenter, dblLength, _
                                             dblWidth, dblHeight)
    objEnt.Update

    ThisDrawing.SendCommand "_shade" & vbCr
End Sub
```

Creating Elliptical 3-D Objects

AutoCAD provides two methods for creating elliptical solids. The first creates an elliptical cone, and the second creates an elliptical cylinder.

The Elliptical Cone

The `AddEllipticalCone` method creates a `3DSolid` object in the form of an elliptical cone whose base lies flat on the WCS XY plane. The major axis of the ellipse may lie in either the X direction or the Y direction.

```
Set 3DSolidObject = Object.AddEllipticalCone(ConeCenter, XLength, YLength, Height)
```

Table 9-7 presents the `AddElliptical` method's parameters, and Figure 9-7 shows an example of an elliptical cone in AutoCAD.

Table 9-7. *The* `AddEllipticalCone` *Method's Parameters*

Name	Data Type	Description
`ConeCenter`	Variant	A three-element array of doubles specifying the center of the bounding box in WCS coordinates
`XLength`	Double	A positive value representing the length of the X-axis of the cone's elliptical base
`YLength`	Double	A positive value representing the length of the Y-axis of the cone's elliptical base
`Height`	Double	A positive value representing the cone's height

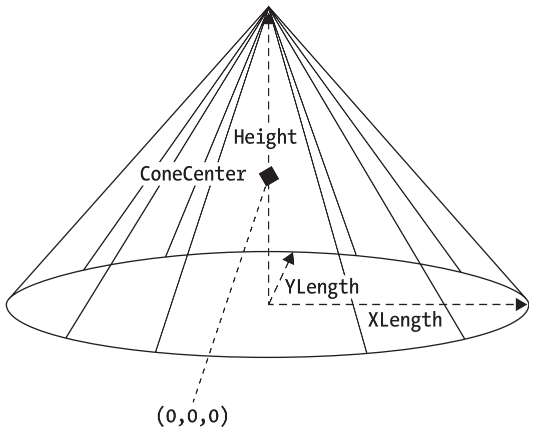


Figure 9-7. *An elliptical cone*

The following example creates an elliptical cone based on user input:

```
Public Sub TestAddEllipticalCone()
      Dim varPick As Variant
      Dim dblXAxis As Double
      Dim dblYAxis As Double
      Dim dblHeight As Double
      Dim dblCenter(2) As Double
      Dim objEnt As Acad3DSolid

    '' set the default viewpoint
    SetViewpoint

    '' get input from user
    With ThisDrawing.Utility
        .InitializeUserInput 1
        varPick = .GetPoint(, vbCr & "Pick a base center point: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblXAxis = .GetDistance(varPick, vbCr & "Enter the X eccentricity: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblYAxis = .GetDistance(varPick, vbCr & "Enter the Y eccentricity: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblHeight = .GetDistance(varPick, vbCr & "Enter the cone Z height: ")
    End With

    '' calculate center point from input
    dblCenter(0) = varPick(0)
    dblCenter(1) = varPick(1)
    dblCenter(2) = varPick(2) + (dblHeight / 2)
    '' draw the entity
    Set objEnt = ThisDrawing.ModelSpace.AddEllipticalCone(dblCenter, _
                                    dblXAxis, dblYAxis, dblHeight)
    objEnt.Update

    ThisDrawing.SendCommand "_shade" & vbCr
End Sub
```

The Elliptical Cylinder

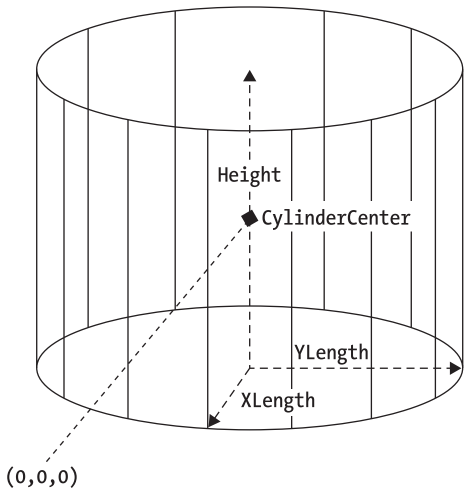
Use the `AddEllipticalCylinder` method to add a `3DSolid` elliptical cylinder whose base lies parallel to the WCS XY plane. The major axis of the elliptical base lies either in the X direction or in the Y direction.

```
Set 3DSolidObject = Object.AddEllipticalCylinder _
        (CylinderCenter, XLength, YLength, Height)
```

Table 9-8 presents the `AddEllipticalCylinder` method's parameters, and Figure 9-8 shows an example of an elliptical cylinder in AutoCAD.

Table 9-8. *The* `AddEllipticalCylinder` *Method's Parameters*

Name	Data Type	Description
`CylinderCenter`	Variant	A three-element array of doubles specifying the center of the bounding box in WCS coordinates
`XLength`	Double	A positive value representing the length of the X-axis of the cylinder's elliptical base
`YLength`	Double	A positive value representing the length of the Y-axis of the cylinder's elliptical base
`Height`	Double	A positive value representing the cylinder's height



This example creates an elliptical cylinder based on user input:

```
Public Sub TestAddEllipticalCylinder()
      Dim varPick As Variant
      Dim dblXAxis As Double
      Dim dblYAxis As Double
      Dim dblHeight As Double
      Dim dblCenter(2) As Double
      Dim objEnt As Acad3DSolid

    '' set the default viewpoint
    SetViewpoint

    '' get input from user
    With ThisDrawing.Utility
        .InitializeUserInput 1
        varPick = .GetPoint(, vbCr & "Pick a base center point: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblXAxis = .GetDistance(varPick, vbCr & "Enter the X eccentricity: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblYAxis = .GetDistance(varPick, vbCr & "Enter the Y eccentricity: ")
        .InitializeUserInput 1 + 2 + 4, ""
        dblHeight = .GetDistance(varPick, vbCr & _
                                  "Enter the cylinder Z height: ")
    End With

    '' calculate center point from input
    dblCenter(0) = varPick(0)
    dblCenter(1) = varPick(1)
    dblCenter(2) = varPick(2) + (dblHeight / 2)

    '' draw the entity
    Set objEnt = ThisDrawing.ModelSpace.AddEllipticalCylinder(dblCenter, _
                                          dblXAxis, dblYAxis, dblHeight)
    objEnt.Update

    ThisDrawing.SendCommand "_shade" & vbCr
End Sub
```

Creating Extruded and Revolved Objects

You create extruded and revolved solids by taking planar `Region` objects and adding thickness to them, either by extruding them along the Z-axis or by revolving them around an arbitrary axis. The newly created entities are `3DSolid` objects.

The Extruded Solid

You create this solid by extruding a planar region along its Z-axis. You may extrude the region in a positive or negative direction, and you may taper or expand it in the extrusion direction.

```
Set 3DSolidObject = Object.AddExtrudedSolid(Region, Height, TaperAngle)
```

Table 9-9 presents the `AddExtrudedSolid` method's parameters.

Table 9-9. *The* `AddExtrudedSolid` *Method's Parameters*

Name	Data Type	Description
`Region`	`Region` object	A closed planar region to be extruded.
`Height`	Double	A nonzero value representing the extrusion's height. If this value is positive, the region is extruded in the OCS Z-direction. If this value is negative, the region is extruded in the –Z direction.
`TaperAngle`	Double	The angle of tapering or expansion of the extrusion given in radians. It must lie between –90 degrees and +90 degrees. Positive values taper in from the base, and negative values expand.

In Figure 9-9 you can see that the `Height` parameter is positive because the region is extruded in the +Z direction, and the `TaperAngle` parameter is negative because the object becomes wider as it's extruded.

AutoCAD won't create the extruded object if it intersects itself. In this event, AutoCAD will raise a run-time error. To avoid this possibility, you should be careful to choose the `Height` and `TaperAngle` parameters so that this doesn't occur.

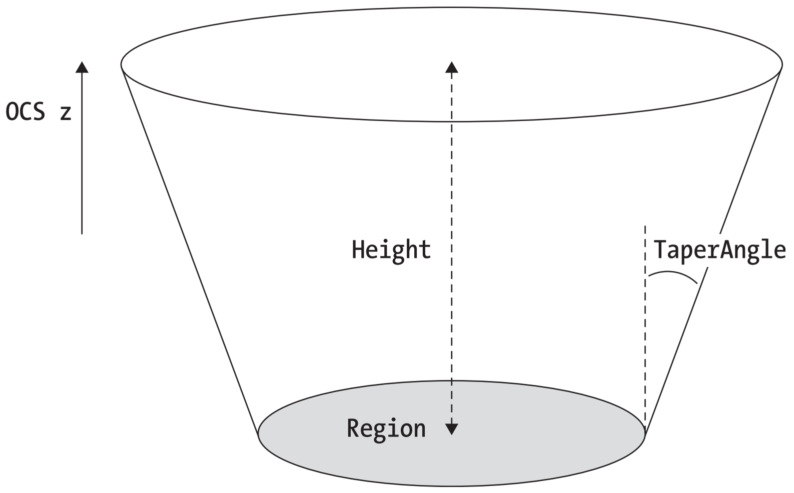


Figure 9-9. *An extruded solid*

The following example creates a circular region and extrudes it into a solid based on user input:

```
Public Sub TestAddExtrudedSolid()
      Dim varCenter As Variant
      Dim dblRadius As Double
      Dim dblHeight As Double
      Dim dblTaper As Double
      Dim strInput As String
      Dim varRegions As Variant
      Dim objEnts() As AcadEntity
      Dim objEnt As Acad3DSolid
      Dim varItem As Variant

On Error GoTo Done

    '' get input from user
    With ThisDrawing.Utility
        .InitializeUserInput 1
        varCenter = .GetPoint(, vbCr & "Pick the center point: ")
        .InitializeUserInput 1 + 2 + 4
        dblRadius = .GetDistance(varCenter, vbCr & "Indicate the radius: ")
        .InitializeUserInput 1 + 2 + 4
        dblHeight = .GetDistance(varCenter, vbCr & _
                                 "Enter the extrusion height: ")

        '' get the taper type
        .InitializeUserInput 1, "Expand Contract None"
        strInput = .GetKeyword(vbCr & _
                               "Extrusion taper [Expand/Contract/None]: ")

        '' if none, taper = 0
        If strInput = "None" Then
            dblTaper = 0

        '' otherwise, get the taper angle
        Else
            .InitializeUserInput 1 + 2 + 4
            dblTaper = .GetReal("Enter the taper angle ( in degrees): ")
            dblTaper = .AngleToReal(CStr(dblTaper), acDegrees)


            '' if expanding, negate the angle
            If strInput = "Expand" Then dblTaper = -dblTaper
        End If
    End With
    '' draw the entities
    With ThisDrawing.ModelSpace

        '' draw the outer region (circle)
        ReDim objEnts(0)
        Set objEnts(0) = .AddCircle(varCenter, dblRadius)

        '' create the region
        varRegions = .AddRegion(objEnts)
        '' extrude the solid
        Set objEnt = .AddExtrudedSolid(varRegions(0), dblHeight, dblTaper)

        '' update the extruded solid
        objEnt.Update
    End With

Done:
    If Err Then MsgBox Err.Description

    '' delete the temporary geometry
    For Each varItem In objEnts
        varItem.Delete
    Next
    For Each varItem In varRegions
        varItem.Delete
    Next

    ThisDrawing.SendCommand "_shade" & vbCr
End Sub
```

The Extruded Solid Along a Path

You use the `AddExtrudedSolidAlongPath` method to create a new `3DSolid` object that represents the extrusion of a closed planar region along a given path. This path may take the form of a polyline, spline, circle, ellipse, or arc. The new solid is extruded from the current location of the region using the translation of the path to the region's `Centroid`.

```
Set 3DSolidObject = Object.AddExtrudedSolidAlongPath(Region, Path)
```

Table 9-10 presents the `AddExtrudedSolidAlongPath` method's parameters, and Figure 9-10 shows an example of an extruded solid along a path.

Table 9-10. *The* `AddExtrudedSolidAlongPath` *Method's Parameters*

Name	Data Type	Description
`Region`	`Region` object	A closed planar region to be extruded.
`Path`	`Polyline`, `Spline`, `Circle`, `Ellipse`, or `Arc` object	The path along which the plane region is to be extruded. This path can't lie in the same plane as the region to be extruded.

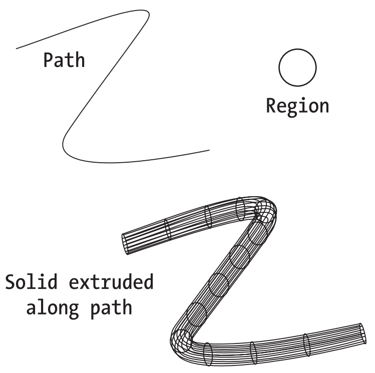


Figure 9-10. *An extruded solid along a path*

As with the extruded solid, the newly created object may not intersect itself. Therefore, you should avoid self-intersecting paths or those with sections of high curvature.

The following example extrudes a circular region into a solid along the path of a spline specified by the user:

```
Public Sub TestAddExtrudedSolidAlongPath()
      Dim objPath As AcadSpline
      Dim varPick As Variant
      Dim intI As Integer
      Dim dblCenter(2) As Double
      Dim dblRadius As Double
      Dim objCircle As AcadCircle
      Dim objEnts() As AcadEntity
      Dim objShape As Acad3DSolid
      Dim varRegions As Variant
      Dim varItem As Variant

    '' set default viewpoint
    SetViewpoint

    '' pick path and calculate shape points
    With ThisDrawing.Utility

        '' pick the path
On Error Resume Next
        .GetEntity objPath, varPick, "Pick a Spline for the path"
        If Err Then
            MsgBox "You did not pick a spline"
            Exit Sub
        End If
        objPath.Color = acGreen
        For intI = 0 To 2
            dblCenter(intI) = objPath.FitPoints(intI)
        Next
        .InitializeUserInput 1 + 2 + 4
        dblRadius = .GetDistance(dblCenter, vbCr & "Indicate the radius: ")

    End With


    '' draw the circular region, then extrude along path
    With ThisDrawing.ModelSpace

        '' draw the outer region (circle)
        ReDim objEnts(0)
        Set objCircle = .AddCircle(dblCenter, dblRadius)
        objCircle.Normal = objPath.StartTangent
        Set objEnts(0) = objCircle
        '' create the region
        varRegions = .AddRegion(objEnts)

        Set objShape = .AddExtrudedSolidAlongPath(varRegions(0), objPath)
        objShape.Color = acRed
    End With

    '' delete the temporary geometry
    For Each varItem In objEnts:  varItem.Delete:  Next
    For Each varItem In varRegions:  varItem.Delete:  Next

    ThisDrawing.SendCommand "_shade" & vbCr
End Sub
```

The Revolved Solid

You add a 3DSolid object representing a revolved solid, which you create by sweeping a planar region around an axis, to a drawing by using the AddRevolvedSolid method.

```
Set 3DSolidObject = Object.AddRevolvedSolid(Region, AxisPoint, _
                                            AxisDirection, Angle)
```

Table 9-11 presents the AddRevolvedSolid method's parameters, and Figure 9-11 shows an example of a revolved solid in AutoCAD.

Table 9-11. The `AddRevolvedSolid` *Method's Parameters*

Name	Data Type	Description
`Region`	`Region` object	A closed planar region to be revolved.
`AxisPoint`	Variant	A three-element array of doubles specifying the center of the axis of rotation in WCS coordinates.
`AxisDirection`	Variant	A three-element array of doubles specifying a directional vector for the axis of rotation.
`Angle`	Double	The angle through which the region is swept, measured in radians. You determine the positive direction of rotation by applying the arbitrary axis algorithm to `AxisDirection`.

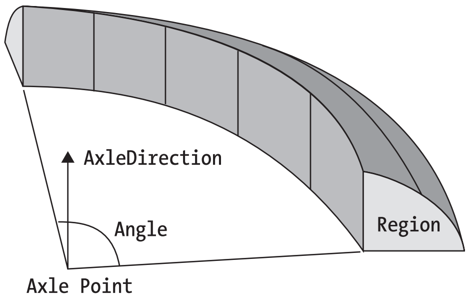


Figure 9-11. *A revolved solid*

The following example lets the user choose a closed region, which is then revolved into a solid based on user input:

```
Public Sub TestAddRevolvedSolid()
      Dim objShape As AcadLWPolyline
      Dim varPick As Variant
      Dim objEnt As AcadEntity
      Dim varPnt1 As Variant
      Dim dblOrigin(2) As Double
      Dim varVec As Variant
      Dim dblAngle As Double
      Dim objEnts() As AcadEntity
      Dim varRegions As Variant
      Dim varItem As Variant

    '' set default viewpoint
    SetViewpoint

    '' draw the shape and get rotation from user
    With ThisDrawing.Utility
```

```
        '' pick a shape
On Error Resume Next
        .GetEntity objShape, varPick, "pick a polyline shape"
        If Err Then
            MsgBox "You did not pick the correct type of shape"
            Exit Sub
        End If
On Error GoTo Done

        objShape.Closed = True

        '' add pline to region input array
        ReDim objEnts(0)
        Set objEnts(0) = objShape

        '' get the axis points
        .InitializeUserInput 1
        varPnt1 = .GetPoint(, vbLf & "Pick an origin of revolution: ")
        .InitializeUserInput 1
        varVec = .GetPoint(dblOrigin, vbLf & _
            "Indicate the axis of revolution: ")

        '' get the angle to revolve
        .InitializeUserInput 1
        dblAngle = .GetAngle(, vbLf & "Angle to revolve: ")
    End With

    '' make the region, then revolve it into a solid
    With ThisDrawing.ModelSpace

        '' make region from closed pline
        varRegions = .AddRegion(objEnts)

        '' revolve solid about axis
        Set objEnt = .AddRevolvedSolid(varRegions(0), varPnt1, varVec, _
                dblAngle)
        objEnt.Color = acRed
    End With

Done:
   If Err Then MsgBox Err.Description
    '' delete the temporary geometry
    For Each varItem In objEnts:  varItem.Delete:  Next
    If Not IsEmpty(varRegions) Then
        For Each varItem In varRegions:  varItem.Delete:  Next
    End If

    ThisDrawing.SendCommand "_shade" & vbCr
End Sub
```

Editing Solids

You may combine and edit individual 3DSolid objects to form new, more complex 3DSolid objects. You may perform operations to combine solids, subtract one solid from another, and find the common intersection between solids. You can use other methods to create sections and slices of individual solids. In each of this section's examples, you call the ORBIT command so that the user can view the resultant solids from whatever angle desired.

Boolean Operations

You perform the primary editing operations using the Boolean method (see Table 9-12 for parameter details). This method, whose name is derived from its algebraic nature, alters the solid according to the Operation parameter.

```
3DSolidObject.Boolean(Operation, SolidObject)
```

Table 9-12. *The* Boolean *Method's Parameters*

Name	Data Type	Description
Operation	Long	The type of operation that takes place. It must be one of the AcBooleanType constants: acUnion, acIntersection, or acSubtraction.
SolidObject	3DSolid object	The object the Boolean operation is performed against.

This method is destructive to the solid passed in the SolidObject parameter—that is, the 3DSolid object passed to this method is destroyed during the combination with the calling object.

■ **Note** If you specify the Intersection operation and the object passed in the SolidObject parameter doesn't spatially intersect the calling object, both 3DSolid objects are destroyed.

The following example combines selected solids using the Boolean operation specified. Try it on some of the solids you created in the preceding examples.

```
Public Sub TestBoolean()
      Dim objFirst As Acad3DSolid
      Dim objSecond As Acad3DSolid
      Dim varPick As Variant
      Dim strOp As String

On Error Resume Next
    With ThisDrawing.Utility
```

```
        '' get first solid from user
        .GetEntity objFirst, varPick, vbCr & "Pick a solid to edit: "
        If Err Then
            MsgBox "That is not an Acad3DSolid"
            Exit Sub
        End If

        '' highlight entity
        objFirst.Highlight True
        objFirst.Update

        '' get second solid from user
        .GetEntity objSecond, varPick, vbCr & "Pick a solid to combine: "
        If Err Then
            MsgBox "That is not an Acad3DSolid"
            Exit Sub
        End If

        '' exit if they're the same
        If objFirst Is objSecond Then
            MsgBox "You must pick 2 different solids"
            Exit Sub
        End If

        '' highlight entity
        objSecond.Highlight True
        objSecond.Update

        '' get boolean operation
        .InitializeUserInput 1, "Intersect Subtract Union"
        strOp = .GetKeyword(vbCr & _
                            "Boolean operation [Intersect/Subtract/Union]: ")

        '' combine the solids
        Select Case strOp
        Case "Intersect": objFirst.Boolean acIntersection, objSecond
        Case "Subtract": objFirst.Boolean acSubtraction, objSecond
        Case "Union": objFirst.Boolean acUnion, objSecond
        End Select

        '' highlight entity
        objFirst.Highlight False
        objFirst.Update
    End With

    '' shade the view, and start the interactive orbit command
    ThisDrawing.SendCommand "_shade" & vbCr & "_orbit" & vbCr
End Sub
```

Interference Operation

You can use another, nondestructive operation to deal with the common space shared by 3DSolid objects. The CheckInterference method (see Table 9-13 for parameter details) calculates the common space between the calling solid and a second object passed to it. It optionally creates and returns a new 3DSolid object.

```
[Set 3DSolidObject =] 3DSolidObject.CheckInterference(SolidObject, _
                                                      CreateInterferenceSolid)
```

Table 9-13. The CheckInterference *Method's Parameters*

Name	Data Type	Description
SolidObject	3DSolid object	The object that is checked against.
CreateInterferenceSolid	Boolean	Specifies whether a solid representing the interference of the two solids should be created. If it is set to True, the interference solid is created.

The following example combines selected solids using this method. Compare its results to the Boolean operators used in the previous example.

```
Public Sub TestInterference()
      Dim objFirst As Acad3DSolid
      Dim objSecond As Acad3DSolid
      Dim objNew As Acad3DSolid
      Dim varPick As Variant
      Dim varNewPnt As Variant

On Error Resume Next
    '' set default viewpoint
    SetViewpoint

    With ThisDrawing.Utility

        '' get first solid from user
        .GetEntity objFirst, varPick, vbCr & "Pick the first solid: "
        If Err Then
            MsgBox "That is not an Acad3DSolid"
            Exit Sub
        End If

        '' highlight entity
        objFirst.Highlight True
        objFirst.Update
        '' get second solid from user
        .GetEntity objSecond, varPick, vbCr & "Pick the second solid: "
        If Err Then
            MsgBox "That is not an Acad3DSolid"
            Exit Sub
        End If

        '' exit if they're the same
        If objFirst Is objSecond Then
            MsgBox "You must pick 2 different solids"
            Exit Sub
        End If

        '' highlight entity
        objSecond.Highlight True
        objSecond.Update

        '' combine the solids
        Set objNew = objFirst.CheckInterference(objSecond, True)
        If objNew Is Nothing Then
            MsgBox "Those solids don't intersect"
        Else
            '' highlight new solid
            objNew.Highlight True
            objNew.Color = acWhite
            objNew.Update

            '' move new solid
            .InitializeUserInput 1
            varNewPnt = .GetPoint(varPick, vbCr & "Pick a new location: ")
            objNew.Move varPick, varNewPnt
        End If

        '' dehighlight entities
        objFirst.Highlight False
        objFirst.Update
        objSecond.Highlight False
        objSecond.Update
    End With

    '' shade the view, and start the interactive orbit command
    ThisDrawing.SendCommand "_shade" & vbCr & "_orbit" & vbCr
End Sub
```

Slicing Solids

Use the `SliceSolid` method (see Table 9-14 for parameter details) to allow `3DSolid` objects to be sliced by an arbitrary plane. This plane, defined by three WCS points, destructively cuts the solid into a front and optional back half. If the back half is desired, then this method returns a new `3DSolid` object.

```
Set 3DSolidObject = 3DSolidObject.SliceSolid(PlanePoint1, _
                        PlanePoint2, PlanePoint3, Negative)
```

Table 9-14. *The* `SliceSolid` *Method's Parameters*

Name	Data Type	Description
`PlanePoint1`	Variant	A three-element array of doubles specifying a point in the slice plane.
`PlanePoint2`	Variant	A three-element array of doubles specifying another point in the slice plane.
`PlanePoint3`	Variant	A three-element array of doubles specifying a third point in the slice plane.
`Negative`	Boolean	A parameter controlling the return of the slice on the negative side of the plane. If it is `False`, no object is returned, and the back half of the slice is discarded. If it is `True`, the method returns the back half. In both cases, the calling object retains the front half.

The following example slices a selected solid and moves the new back half to another location. It might be quite difficult to choose three points that define a plane that slices through your selected object, but using overrides such as `Near` and `Perpendicular` should make this easier. Try it on some of the solids you created in the chapter's previous examples.

```
Public Sub TestSliceSolid()
      Dim objFirst As Acad3DSolid
      Dim objSecond As Acad3DSolid
      Dim objNew As Acad3DSolid
      Dim varPick As Variant
      Dim varPnt1 As Variant
      Dim varPnt2 As Variant
      Dim varPnt3 As Variant
      Dim strOp As String
      Dim blnOp As Boolean


On Error Resume Next
    With ThisDrawing.Utility

        '' get first solid from user
        .GetEntity objFirst, varPick, vbCr & "Pick a solid to slice: "
        If Err Then
            MsgBox "That is not a 3DSolid"
            Exit Sub
        End If
        '' highlight entity
        objFirst.Highlight True
        objFirst.Update

        .InitializeUserInput 1
        varPnt1 = .GetPoint(varPick, vbCr & "Pick first slice point: ")
        .InitializeUserInput 1
        varPnt2 = .GetPoint(varPnt1, vbCr & "Pick second slice point: ")
        .InitializeUserInput 1
        varPnt3 = .GetPoint(varPnt2, vbCr & "Pick last slice point: ")

        '' section the solid
        Set objNew = objFirst.SliceSolid(varPnt1, varPnt2, varPnt3, True)
        If objNew Is Nothing Then
            MsgBox "Couldn't slice using those points"
        Else
            '' highlight new solid
            objNew.Highlight False
            objNew.Color = objNew.Color + 1
            objNew.Update

            '' move section region to new location
            .InitializeUserInput 1
            varPnt2 = .GetPoint(varPnt1, vbCr & "Pick a new location: ")
            objNew.Move varPnt1, varPnt2
        End If
    End With

    '' shade the view
    ThisDrawing.SendCommand "_shade" & vbCr
End Sub
```

Sectioning Solids

The `SectionSolid` method (see Table 9-15 for parameter details) creates a new `Region` object defined by the intersection of a `3DSolid` object and an arbitrary plane. The plane is defined by three WCS points.

```
Set RegionObject = 3DSolidObject.SectionSolid (PlanePoint1, _
                                    PlanePoint2, PlanePoint3)
```

Table 9-15. The `SectionSolid` *Method's Parameters*

Name	Data Type	Description
`PlanePoint1`	Variant	A three-element array of doubles specifying a point in the intersection plane
`PlanePoint2`	Variant	A three-element array of doubles specifying another point in the intersection plane
`PlanePoint3`	Variant	A three-element array of doubles specifying a third point in the intersection plane

The following example sections a selected solid and moves the new region to another location. Remember to use object snap overrides such as `Near` and `Perpendicular` when you define the section plane. Try it on some of the solids you created earlier in this chapter.

```
Public Sub TestSectionSolid()
      Dim objFirst As Acad3DSolid
      Dim objSecond As Acad3DSolid
      Dim objNew As AcadRegion
      Dim varPick As Variant
      Dim varPnt1 As Variant
      Dim varPnt2 As Variant
      Dim varPnt3 As Variant

On Error Resume Next

    With ThisDrawing.Utility
        '' get first solid from user
        .GetEntity objFirst, varPick, vbCr & "Pick a solid to section: "
        If Err Then
            MsgBox "That is not an Acad3DSolid"
            Exit Sub
        End If

        '' highlight entity
        objFirst.Highlight True
        objFirst.Update

        .InitializeUserInput 1
        varPnt1 = .GetPoint(varPick, vbCr & "Pick first section point: ")
        .InitializeUserInput 1
        varPnt2 = .GetPoint(varPnt1, vbCr & "Pick second section point: ")
        .InitializeUserInput 1
        varPnt3 = .GetPoint(varPnt2, vbCr & "Pick last section point: ")
        '' section the solid
        Set objNew = objFirst.SectionSolid(varPnt1, varPnt2, varPnt3)
        If objNew Is Nothing Then
            MsgBox "Couldn't section using those points"
        Else
            '' highlight new solid
            objNew.Highlight False
            objNew.Color = acWhite
            objNew.Update

            '' move section region to new location
            .InitializeUserInput 1
            varPnt2 = .GetPoint(varPnt1, vbCr & "Pick a new location: ")
            objNew.Move varPnt1, varPnt2
        End If

        '' dehighlight entities
        objFirst.Highlight False
        objFirst.Update
    End With

    '' shade the view
    ThisDrawing.SendCommand "_shade" & vbCr
End Sub
```

Analyzing Solids: Mass Properties

Each 3DSolid object has a number of mass properties you can use for analysis. These properties include the center of gravity, the total volume of the solid, the radii of gyration, the product of inertia, and the moment of inertia.

The following example displays the mass properties for a selected solid. Try it on one of the solids you created earlier in the chapter.

```
Public Sub TestMassProperties()
      Dim objEnt As Acad3DSolid
      Dim varPick As Variant
      Dim strMassProperties As String
      Dim varProperty As Variant
      Dim intI As Integer

On Error Resume Next
    '' let user pick a solid
    With ThisDrawing.Utility
        .GetEntity objEnt, varPick, vbCr & "Pick a solid: "
        If Err Then
            MsgBox "That is not an Acad3DSolid"
            Exit Sub
        End If
    End With
```

```
'' format mass properties
With objEnt
    strMassProperties = "Volume: "
    strMassProperties = strMassProperties & vbCr & "    " & .Volume
    strMassProperties = strMassProperties & vbCr & vbCr & _
                        "Center Of Gravity: "
    For Each varProperty In .Centroid
        strMassProperties = strMassProperties & vbCr & "    "_
                             & varProperty
    Next
    strMassProperties = strMassProperties & vbCr & vbCr & _
                        "Moment Of Inertia: "
    For Each varProperty In .MomentOfInertia
        strMassProperties = strMassProperties & vbCr & "    " & _
                            varProperty
    Next
    strMassProperties = strMassProperties & vbCr & vbCr & _
                        "Product Of Inertia: "
    For Each varProperty In .ProductOfInertia
        strMassProperties = strMassProperties & vbCr & "    " & _
                            varProperty
    Next
    strMassProperties = strMassProperties & vbCr & vbCr & _
                        "Principal Moments: "
    For Each varProperty In .PrincipalMoments
        strMassProperties = strMassProperties & vbCr & "    " & _
                            varProperty
    Next
    strMassProperties = strMassProperties & vbCr & vbCr & _
                        "Radii Of Gyration: "
    For Each varProperty In .RadiiOfGyration
        strMassProperties = strMassProperties & vbCr & "    " & _
                            varProperty
    Next
    strMassProperties = strMassProperties & vbCr & vbCr & _
                        "Principal Directions: "
    For intI = 0 To UBound(.PrincipalDirections) / 3
        strMassProperties = strMassProperties & vbCr & "    (" & _
                       .PrincipalDirections((intI - 1) * 3) & ", " & _
                       .PrincipalDirections((intI - 1) * 3 + 1) & "," & _
                       .PrincipalDirections((intI - 1) * 3 + 2) & ")"
    Next

End With
```

```
    '' highlight entity
    objEnt.Highlight True
    objEnt.Update

    '' display properties
    MsgBox strMassProperties, , "Mass Properties"

    '' dehighlight entity
    objEnt.Highlight False
    objEnt.Update
End Sub
```

Summary

In this chapter you learned how to create each of the simple solid primitives. You considered how you can use planar figures to create more free-form solid entities by extruding and revolving regions. You also examined how you can combine existing solids to create sophisticated shapes and how to section and slice those shapes.

CHAPTER 10

Editing Objects

AutoCAD gives you plenty of methods and properties so you can edit drawing objects, which you'd normally have to do in the AutoCAD interface. By employing these methods and properties, you can adapt and combine editing tasks in macros and VBA applications as needed.

This chapter considers editing objects, first through their methods and then through their properties. Specifically, this chapter covers the following:

- Copying, deleting, exploding, highlighting, mirroring, moving, offsetting, rotating, and scaling objects
- Working with polar and rectangular arrays
- Changing the color, layer, linetype, and visibility of objects

In versions prior to 2006, whenever you modify an object in your code, the changes to the drawing don't appear until you call the object's `Update` method, the `Application` object's `Update` method, or the `Document` object's `Regen` method. In some cases, AutoCAD updates the display when your macro or program is complete, but it is safest for your code to force the update. This chapter's examples use the modified object's `Update` method to update the drawing display:

```
DrawingObject.Update
```

Editing with Methods

AutoCAD VBA editing methods edit differently from properties. Methods generally change an entity's shape, size, and position or create a new entity based on the original. Properties, on the other hand, tend to change the appearance of the object boundary lines, which represent the objects on the screen or on the plotter.

Unlike Visual LISP, VBA provides no equivalent functions for `(vlax`-method-applicable-p) for making sure an object exposes a specific method. For this reason, for each object against which you want to invoke a method, be sure that the object supports the method you are using. Your best tools for this are error trapping and error handling.

Copying Objects

Use the Copy method to copy an existing drawing object. The new object occupies the same position as the original object and is drawn on top of all other objects. This method has the following syntax:

```
Set DrawingObject = DrawingObject.Copy
```

The following code asks the user to pick a drawing object from the screen. It then copies the object and moves it to a point the user chooses.

```
Public Sub CopyObject()
      Dim objDrawingObject As AcadEntity
      Dim objCopiedObject As Object
      Dim varEntityPickedPoint As Variant
      Dim varCopyPoint As Variant

On Error Resume Next
    ThisDrawing.Utility.GetEntity objDrawingObject, varEntityPickedPoint, _
        "Pick an entity to copy: "
    If objDrawingObject Is Nothing Then
        MsgBox "You did not pick an object"
        Exit Sub
    End If

    'Copy the object
    Set objCopiedObject = objDrawingObject.Copy()
    varCopyPoint = ThisDrawing.Utility.GetPoint(, "Pick point to copy to: ")

    'put the object in its new position
    objCopiedObject.Move varEntityPickedPoint, varCopyPoint
    objCopiedObject.Update

End Sub
```

Deleting Objects

To remove an object from a drawing, use the Delete method, which has the following simple syntax:

```
Object.Delete
```

Many AutoCAD objects expose this method. For full details, see Appendix A. The following example shows how to implement this method:

```
Public Sub DeleteObject()
      Dim objDrawingObject As AcadEntity
      Dim varEntityPickedPoint As Variant
```

```
On Error Resume Next
    ThisDrawing.Utility.GetEntity objDrawingObject, varEntityPickedPoint, _
                                  "Pick an entity to delete: "
    If objDrawingObject Is Nothing Then
        MsgBox "You did not pick an object."
        Exit Sub
    End If

    'delete the object
    objDrawingObject.Delete

End Sub
```

The Erase method is similar to the Delete method, except it erases only SelectionSet groups. You can't use it to delete objects.

Exploding Objects

Use the Explode method to break a compound object into its subentities. It returns an array of objects that hold references to these subentities. This might mean breaking a block reference into its constituent drawing objects or a polyline into its individual straight-line polylines. Block Insertions leave the original block object behind, and the resulting exploded set of entities is actually a copy of each original nested entity in the Block Insertion. To produce behavior similar to the AutoCAD EXPLODE command, also delete the original Block Insertion entity. This method has the following syntax:

```
varObjectArray = Object.Explode
```

The 3DPoly, BlockRef, LightweightPolyline, MInsertBlock, PolygonMesh, Polyline, Xref, and Region objects expose this method.

This code example asks the user to choose a Region object. It then explodes the object and displays a message box that lists the types of the objects into which it was exploded.

```
Public Sub ExplodeRegion()
      Dim objDrawingObject As AcadEntity
      Dim varEntityPickedPoint As Variant

On Error Resume Next
    ThisDrawing.Utility.GetEntity objDrawingObject, varEntityPickedPoint, _
                                  "Please pick a region object."
    If objDrawingObject Is Nothing Or _
       objDrawingObject.ObjectName <> "AcDbRegion" Then
        MsgBox "You did not choose a region object."
        Exit Sub
    End If

Dim varObjectArray As Variant
Dim strObjectTypes As String
Dim intCount As Integer
```

```
    varObjectArray = objDrawingObject.Explode
    strObjectTypes = "The region you chose has been exploded " & _
             "into the following: " & UBound(varObjectArray) + 1 & " objects:"
    For intCount = 0 To UBound(varObjectArray)
        strObjectTypes = strObjectTypes & vbCrLf & _
                         varObjectArray(intCount).ObjectName
    Next
    MsgBox strObjectTypes

End Sub
```

■ **Note** In AutoCAD versions before AutoCAD 2004, the `Explode` method exploded nested MText entities in Block Insertions, resulting in individual Text entities. In AutoCAD 2004, the `Explode` method preserves nested MText entities as `MText`.

Highlighting Entities

Use the `Highlight` method to set whether an entity is highlighted in the drawing editor window. Changes to an entity's highlight become effective only when you update or regenerate the entity. You can apply this method to drawing entities and the `Group` and `SelectionSet` objects. This method has the following syntax:

```
Object.Highlight Highlighted
```

This method's `Highlighted` parameter, a `Boolean`, indicates whether to highlight the object (`True`) or not (`False`).

The following example illustrates the `Highlight` method in action. It asks the user to choose a selection of objects from the drawing area. It then highlights and unhighlights each of these objects.

```
Public Sub ToggleHighlight()
      Dim objSelectionSet As AcadSelectionSet
      Dim objDrawingObject As AcadEntity

    'choose a selection set name that you only use as temporary storage and
    'ensure that it does not currently exist
    On Error Resume Next
    ThisDrawing.SelectionSets("TempSSet").Delete
    Set objSelectionSet = ThisDrawing.SelectionSets.Add("TempSSet")

    'ask user to pick entities on the screen
    objSelectionSet.SelectOnScreen
    'change the highlight status of each entity selected
    For Each objDrawingObject In objSelectionSet
        objDrawingObject.Highlight True
        objDrawingObject.Update                               'not required for 2006
        MsgBox "Notice that the entity is highlighted"
        objDrawingObject.Highlight False                      'not required for 2006
        objDrawingObject.Update                               'not required for 2006
        MsgBox "Notice that the entity is not highlighted"
    Next

    objSelectionSet.Delete

End Sub
```

Mirroring Objects

Use the `Mirror` method to create an object that is the mirror image of an existing planar object. This method has the following syntax:

```
Set DrawingObject = DrawingObject.Mirror(Point1, Point2)
```

Table 10-1 explains this method's parameters.

Table 10-1. `Mirror` *Method Parameters*

Name	Data Type	Description
`Point1`	Variant	A three-element array of doubles specifying the 3-D WCS coordinates of a point on the mirror axis.
`Point2`	Variant	A three-element array of doubles specifying the 3-D WCS coordinates of a second point on the mirror axis.

The following code lets the user choose a selection of objects and a mirror axis, and then mirrors the objects around that axis.

```
Public Sub MirrorObjects()
      Dim objSelectionSet As AcadSelectionSet
      Dim objDrawingObject As AcadEntity
      Dim objMirroredObject As AcadEntity
      Dim varPoint1 As Variant
      Dim varPoint2 As Variant

    ThisDrawing.SetVariable "MIRRTEXT", 0
    'choose a selection set name that you only use as temporary storage and
    'ensure that it does not currently exist
    On Error Resume Next
    ThisDrawing.SelectionSets("TempSSet").Delete
    Set objSelectionSet = ThisDrawing.SelectionSets.Add("TempSSet")

    'ask user to pick entities on the screen
    ThisDrawing.Utility.Prompt "Pick objects to be mirrored." & vbCrLf
    objSelectionSet.SelectOnScreen

'change the highlight status of each entity selected
    varPoint1 = ThisDrawing.Utility.GetPoint(, _
        "Select a point on the mirror axis")
    varPoint2 = ThisDrawing.Utility.GetPoint(varPoint1, _
        "Select a point on the mirror axis")

    For Each objDrawingObject In objSelectionSet
        Set objMirroredObject = objDrawingObject.Mirror(varPoint1, varPoint2)
        objMirroredObject.Update
    Next

    objSelectionSet.Delete

End Sub
```

Note that the MIRRTEXT system variable has been set to 0. This setting mirrors text objects to the appropriate place on the drawing, but does not mirror the text itself, as shown in Figure 10-1. The default value of MIRRTEXT in AutoCAD 2004 is 0, but it defaulted to 1 in earlier versions of AutoCAD. If you set MIRRTEXT to 1, the text gets mirrored just as any other object.

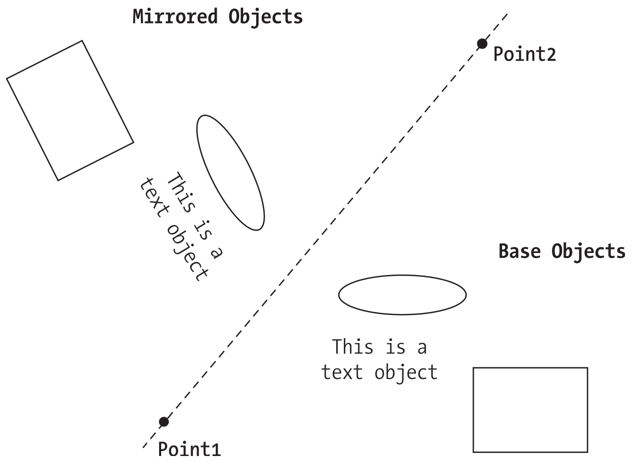


Figure 10-1. *Mirrored objects*

Use the `Mirror3D` method to reflect an object in a plane, where the new object lies outside the plane of the original object.

```
Set DrawingObject = DrawingObject.Mirror3D(PlanePoint1, PlanePoint2, PlanePoint3)
```

Table 10-2 explains this method's parameters.

Table 10-2. `Mirror 3D` *Method Parameters*

Name	Data Type	Description
`PlanePoint1`	Variant	A three-element array of doubles specifying the 3-D WCS coordinates of a point in the mirror plane.
`PlanePoint2`	Variant	A three-element array of doubles specifying the 3-D WCS coordinates of a second point in the mirror plane.
`PlanePoint3`	Variant	A three-element array of doubles specifying the 3-D WCS coordinates of a third point defining the mirror plane.

Figure 10-2 shows an object reflected on a plane.

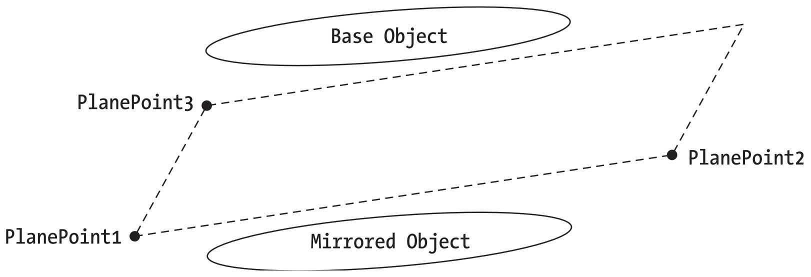


Figure 10-2. *A 3-D mirrored object*

The following code adds a new drawing object that reflects the base object around the WCS XY plane.

```
Public Sub MirrorObjectinXYplane()
      Dim objDrawingObject As AcadEntity
      Dim varEntityPickedPoint As Variant
      Dim objMirroredObject As AcadEntity
      Dim dblPlanePoint1(2) As Double
      Dim dblPlanePoint2(2) As Double
      Dim dblPlanePoint3(2) As Double

On Error Resume Next
    ThisDrawing.Utility.GetEntity objDrawingObject, varEntityPickedPoint, _
                                 "Please pick an entity to reflect: "
    If objDrawingObject Is Nothing Then
        MsgBox "You did not choose an object"
        Exit Sub
    'set plane of reflection to be the XY plane
    dblPlanePoint2(0) = 1#
    dblPlanePoint3(1) = 1#

    objDrawingObject.Mirror3D dblPlanePoint1, dblPlanePoint2, dblPlanePoint3

End Sub
```

Moving Objects

Use the `Move` method to perform three-dimensional translations on drawing objects. This method has the following syntax:

```
DrawingObject.Move Point1, Point2
```

Table 10-3 explains this method's parameters.

Table 10-3. `Move` *Method Parameters*

Name	Data Type	Description
`Point1`	Variant	A three-element array of doubles specifying the 3-D WCS coordinates of the translation vector's starting point.
`Point2`	Variant	A three-element array of doubles specifying the 3-D WCS coordinates of the translation vector's ending point.

Figure 10-3 shows an object moved from one place to another.

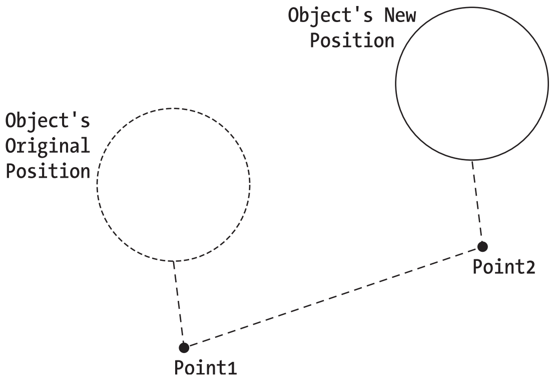


Figure 10-3. *A moved object*

The following example shows how to implement this method. The user first picks a selection of objects to be moved and specifies the translation vector on the screen.

```
Public Sub MoveObjects()
      Dim varPoint1 As Variant
      Dim varPoint2 As Variant
      Dim objSelectionSet As AcadSelectionSet
      Dim objDrawingObject As AcadEntity

    'choose a selection set name that you only use as temporary storage and
    'ensure that it does not currently exist
    On Error Resume Next
    ThisDrawing.SelectionSets("TempSSet").Delete
    Set objSelectionSet = ThisDrawing.SelectionSets.Add("TempSSet")

    'ask user to pick entities on the screen
    objSelectionSet.SelectOnScreen

    varPoint1 = ThisDrawing.Utility.GetPoint(, vbCrLf _
        & "Base point of displacement: ")
    varPoint2 = ThisDrawing.Utility.GetPoint(varPoint1, vbCrLf _
        & "Second point of displacement: ")

    'move the selection of entities
    For Each objDrawingObject In objSelectionSet
     objDrawingObject.Move varPoint1, varPoint2
     objDrawingObject.Update
    Next

    objSelectionSet.Delete

End Sub
```

Offsetting Objects

The `Offset` method creates a new object with boundaries offset a specified distance from an existing object's boundaries. You can apply this method to the `Arc`, `Circle`, `Ellipse`, `Line`, `LightweightPolyline`, `Polyline`, `Spline`, and `Xline` objects. This method returns an array of the newly created object(s). Even though this array frequently contains only one object of the same type as the object from which it is offset, this is not always this case, as seen in the example below.

```
varObjectArray = Object.Offset(OffsetDistance)
```

Table 10-4 explains this method's parameter.

Table 10-4. *The* `Object.Offset` *Method Parameter*

Name	Data Type	Description
`OffsetDistance`	Double	A nonzero number that indicates the offset's size and direction. Negative numbers mean that the offset makes the new object smaller than the original. If this has no meaning, as for a single straight line, a negative number positions the new object closer to the WCS origin.

In Figure 10-4, the center object is an `Ellipse` from which the two other objects have been offset. You can see that the new objects are not `Ellipses` as each point of each new object is the same distance from the original `Ellipse`. In these cases, the returned object is a `Spline`. In this example, the negative offset must be greater than 50.

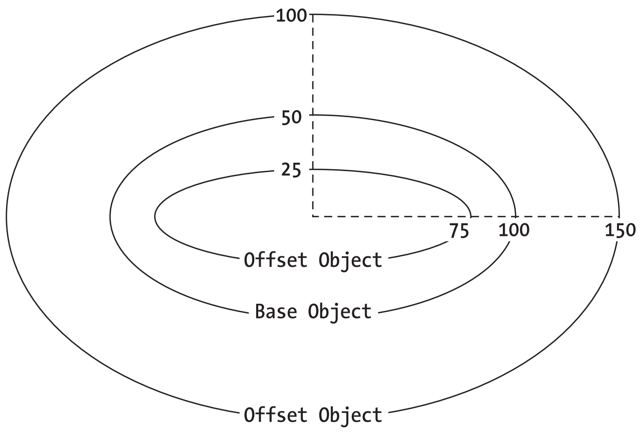


Figure 10-4. `Offset` *objects*

The following code sample creates the `Ellipse` and `Splines` in Figure 10-4 and shows the user a message box of the new objects' types.

```
Public Sub OffsetEllipse()
      Dim objEllipse As AcadEllipse
      Dim varObjectArray As Variant
      Dim dblCenter(2) As Double
      Dim dblMajor(2) As Double

    dblMajor(0) = 100#
    Set objEllipse = ThisDrawing.ModelSpace.AddEllipse(dblCenter, dblMajor, _
                                                         0.5)

    varObjectArray = objEllipse.Offset(50)
    MsgBox "The offset object is a " & varObjectArray(0).ObjectName

    varObjectArray = objEllipse.Offset(-25)
    MsgBox "The offset object is a " & varObjectArray(0).ObjectName

End Sub
```

■**Note** For the sake of accuracy, because offsetting an `Ellipse` creates nonelliptical `Spline` entities, instead consider creating new `Ellipse` entities with calculated geometry. This produces accurate `Ellipse` entities instead of approximated elliptical `Splines`. The same is true for offsetting `Spline` or curve-fitted `Polyline` or `LightweightPolyline` entities.

Rotating Objects

Use the `Rotate` method to rotate a drawing object around a given point in the User Coordinate System's (UCS) XY plane. This method has the following syntax:

```
DrawingObject.Rotate BasePoint, RotationAngle
```

Table 10-5 explains this method's parameters.

Table 10-5. `Rotate` *Method Parameters*

Name	Data Type	Description
`BasePoint`	Variant	A three-element array of doubles that specifies the 3-D WCS coordinates of the point through which the axis of rotation, parallel to the Z-axis of the UCS, passes.
`RotationAngle`	Double	The angle of rotation given in radians and measured counter-clockwise from the UCS's X-axis.

Figure 10-5 shows a rotated object.

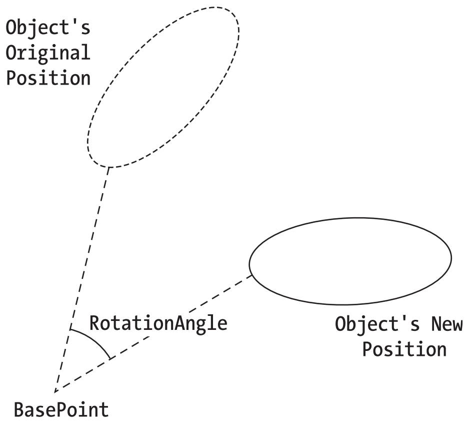


Figure 10-5. *A rotated object*

The following sample rotates an object based on user input.

```
Public Sub RotateObject()
      Dim objDrawingObject As AcadEntity
      Dim varEntityPickedPoint As Variant
      Dim varBasePoint As Variant
      Dim dblRotationAngle As Double

On Error Resume Next
    ThisDrawing.Utility.GetEntity objDrawingObject, varEntityPickedPoint, _
                              "Please pick an entity to rotate: "
    If objDrawingObject Is Nothing Then
        MsgBox "You did not choose an object."
        Exit Sub
    End If
    varBasePoint = ThisDrawing.Utility.GetPoint(, _
                   "Enter a base point for the rotation.")
    dblRotationAngle = ThisDrawing.Utility.GetReal( _
                       "Enter the rotation angle in degrees: ")

    'convert to radians
    dblRotationAngle = ThisDrawing.Utility. _
                       AngleToReal(CStr(dblRotationAngle), acDegrees)

    'Rotate the object
    objDrawingObject.Rotate varBasePoint, dblRotationAngle
    objDrawingObject.Update

End Sub
```

Use the `Rotate3D` method to perform a three-dimensional rotation of an object around an axis. This method has the following syntax:

```
DrawingObject.Rotate3D AxisPoint1, AxisPoint2, RotationAngle
```

Table 10-6 explains this method's parameters.

Table 10-6. `Rotate3D` *Method Parameters*

Name	Data Type	Description
`AxisPoint1`	Variant	A three-element array of doubles that specifies the 3-D WCS coordinates of a point through which the axis of rotation passes.
`AxisPoint2`	Variant	A three-element array of doubles that specifies the 3-D WCS coordinates of a second point through which the axis of rotation passes.
`RotationAngle`	Double	The angle of rotation given in radians and measured counter-clockwise about the specified axis.

Figure 10-6 shows an object rotated around an axis.

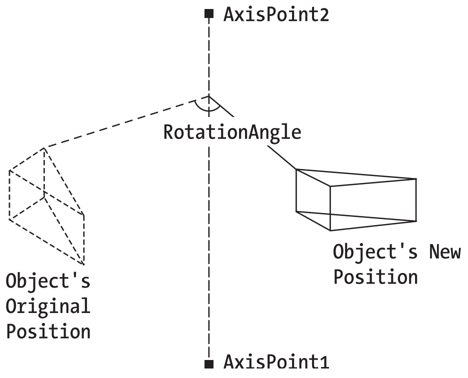


Figure 10-6. *An object rotated in 3-D*

The following code example lets the user choose an object to rotate, and the points that define the axis of rotation. It then rotates the object around that axis at an angle the user chooses.

```
Public Sub Rotate3DObject()
Dim objDrawingObject As AcadEntity
Dim varEntityPickedPoint As Variant
Dim objMirroredObject As AcadEntity
Dim varAxisPoint1 As Variant
Dim varAxisPoint2 As Variant
Dim dblRotationAxis As Double

On Error Resume Next
    ThisDrawing.Utility.GetEntity objDrawingObject, _
        varEntityPickedPoint, "Please select an entity to rotate: "
    If objDrawingObject Is Nothing Then
        MsgBox "You did not choose an object"
        Exit Sub
    End If

   'ask user for axis points and angle of rotation
   varAxisPoint1 = ThisDrawing.Utility.GetPoint(, _
                                "Enter first point of axis of rotation: ")
   varAxisPoint2 = ThisDrawing.Utility.GetPoint(, _
                                "Enter second point of axis of rotation: ")
   dblRotationAxis = ThisDrawing.Utility.GetReal( _
                                "Enter angle of rotation in degrees")
   'convert to radians
   dblRotationAxis = ThisDrawing.Utility.AngleToReal(CStr(dblRotationAxis), _
                                                    acDegrees)

   objDrawingObject.Rotate3D varAxisPoint1, varAxisPoint2, dblRotationAxis

End Sub
```

Scaling Objects

The `ScaleEntity` method scales a drawing object uniformly in all directions around a base point. This method has the following syntax:

```
DrawingObject.ScaleEntity BasePoint, ScaleFactor
```

Table 10-7 explains this method's parameters.

Table 10-7. `ScaleEntity` *Method Parameters*

Name	Data Type	Description
`BasePoint`	Variant	A three-element array of doubles that specifies the 3-D WCS coordinates of the scale's base point.
`ScaleFactor`	Double	A positive number that represents how much to scale the object relative to its current size. Each of the object's dimensions, measured from the base point, is multiplied by this parameter. A scale factor greater than 1 enlarges the object, while a scale factor less than 1 shrinks the object.

Figure 10-7 shows a scaled object.

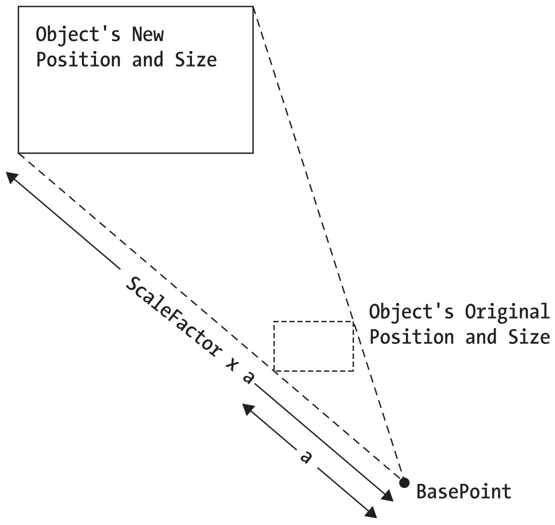


Figure 10-7. *A scaled object*

This example shows how to implement this method:

```
Public Sub ScaleObject()
      Dim objDrawingObject As AcadEntity
      Dim varEntityPickedPoint As Variant
      Dim varBasePoint As Variant
      Dim dblScaleFactor As Double
```

```
On Error Resume Next
    ThisDrawing.Utility.GetEntity objDrawingObject, varEntityPickedPoint, _
                                  "Please pick an entity to scale: "
    If objDrawingObject Is Nothing Then
        MsgBox "You did not choose an object"
        Exit Sub
    End If

    varBasePoint = ThisDrawing.Utility.GetPoint(, _
                   "Pick a base point for the scale:")
    dblScaleFactor = ThisDrawing.Utility.GetReal("Enter the scale factor: ")

    'Scale the object
    objDrawingObject.ScaleEntity varBasePoint, dblScaleFactor
    objDrawingObject.Update
End Sub
```

■ **Note** There is no direct equivalent to the `SCALE` command's Reference option. AutoCAD performs those calculations for you. To mimic that functionality, you have to write the code that performs the calculations.

Object Arrays

Use the `ArrayPolar` and `ArrayRectangular` methods to create an array of objects based on an existing object. Both methods copy the base object into a regular pattern at a specified distance from one another.

Creating a Polar Array of Objects

A polar array is a pattern of drawing objects lying on an arc's path. The `ArrayPolar` method creates a polar array, placing a specified number of objects over an angle. It returns an array of the new objects.

```
varObjectArray = DrawingObject.ArrayPolar (NumberofObjects, _
    AngleToFill, ArrayCenter)
```

Table 10-8 explains this method's parameters.

Table 10-8. `ArrayPolar` *Method Parameters*

Name	Data Type	Description
`NumberofObjects`	Long	The number of objects in the polar array.
`AngleToFill`	Double	A nonzero value representing the angle, measured counter-clockwise in radians, over which the array path will extend.
`ArrayCenter`	Variant	A three-element array of doubles specifying the 3-D WCS coordinates of the arc's center.

Always set the NumberofObjects parameter greater than 1. The original object is counted as an element of the polar array, even though this method doesn't return it in the object array. Consequently, the size of the object array is NumberofObjects-1. If you set NumberofObjects to 1, the method returns an empty array, but doesn't raise an error.

Figure 10-8 shows an ellipse arranged along an arc using the ArrayPolar method.

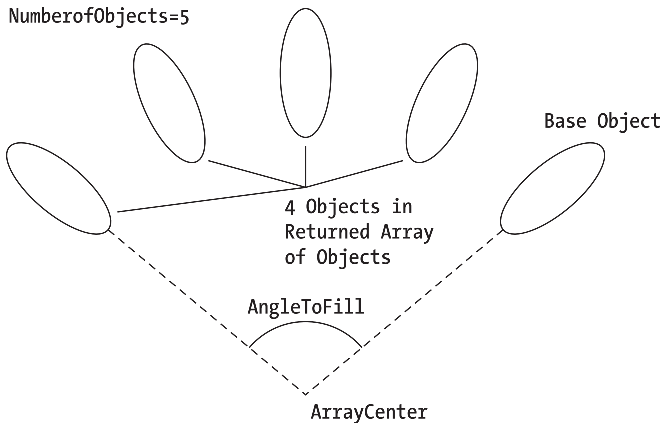


Figure 10-8. ArrayPolar *arrangement of ellipses*

The distance from the ArrayCenter to objects depends upon the object type. To specify the distance to the objects and the vector from which to measure the AngleToFill, use a circle or arc's center, a block or shape's insertion point, text's start point, or a line or trace's end point.

The following example shows how to implement this method. The objects returned in the array are designated as red.

```
Public Sub CreatePolarArray()
      Dim objDrawingObject As AcadEntity
      Dim varEntityPickedPoint As Variant
      Dim varArrayCenter As Variant
      Dim lngNumberofObjects As Long
      Dim dblAngletoFill As Double
      Dim varPolarArray As Variant
      Dim intCount As Integer

On Error Resume Next
    ThisDrawing.Utility.GetEntity objDrawingObject, varEntityPickedPoint, _
                      "Please select an entity to form the basis of a polar array"
    If objDrawingObject Is Nothing Then
        MsgBox "You did not choose an object"
        Exit Sub
    End If
    varArrayCenter = ThisDrawing.Utility.GetPoint(, _
                     "Pick the center of the array: ")
    lngNumberofObjects = ThisDrawing.Utility.GetInteger( _
                     "Enter total number of objects required in the array: ")
    dblAngletoFill = ThisDrawing.Utility.GetReal( _
          "Enter an angle (in degrees less than 360) over which the array should
extend: ")
    dblAngletoFill = ThisDrawing.Utility.AngleToReal _
                            (CStr(dblAngletoFill), acDegrees)
    If dblAngletoFill > 359 Then
      MsgBox "Angle must be less than 360 degrees", vbCritical
      Exit Sub
    End If
    varPolarArray = objDrawingObject.ArrayPolar(lngNumberofObjects, _
                                            dblAngletoFill, varArrayCenter)

    For intCount = 0 To UBound(varPolarArray)
        varPolarArray(intCount).Color = acRed
        varPolarArray(intCount).Update
    Next

End Sub
```

Note Unlike the AutoCAD `ARRAY` command, there is no ActiveX method available to control rotation of copied entities in polar arrays. You must handle that yourself through programming to rotate each copied entity appropriately along the array path's circumference.

Creating a Rectangular Array of Objects

A rectangular array is a grid pattern of drawing objects. This pattern may be two- or three-dimensional and you can construct it using the `ArrayRectangular` method. Just like the `ArrayPolar` method, this method returns an array of objects that hold a reference to the new drawing objects.

```
varObjectArray = DrawingObject.ArrayRectangular (NumberOfRows, _
    NumberOfColumns, NumberOfLevels, DistBetweenRows, _
    DistBetweenColumns, DistBetweenLevels)
```

Table 10-9 explains this method's parameters.

Table 10-9. `ArrayRectangular` *Method Parameters*

Name	Data Type	Description
`NumberofRows`	Long	A positive number representing the number of rows in the rectangular array. If this parameter is set to 1, then `NumberofColumns` must be greater than 1.
`NumberOfColumns`	Long	A positive number representing the number of columns in the rectangular array. If this parameter is set to 1, then `NumberofRows` must be greater than 1.
`NumberOfLevels`	Long	A positive number representing the number of levels in a 3-D array.
`DistBetweenRows`	Double	The distance between the rows. If this parameter is positive, rows extend upwards from the base object; if negative they extend downwards. If set to zero, objects are placed on top of the base object.
`DistBetweenColumns`	Double	The distance between the columns. If this parameter is positive, rows extend upwards from the base object; if negative they extend downwards. If set to zero, objects are placed on top of the base object.
`DistBetweenLevels`	Double	The distance between the levels. If this parameter is positive, rows extend upwards from the base object; if negative they extend downwards. If set to zero, objects are placed on top of the base object.

Figure 10-9 shows a two-dimensional rectangular array. A three-dimensional array is a simple extension of a two-dimensional array.

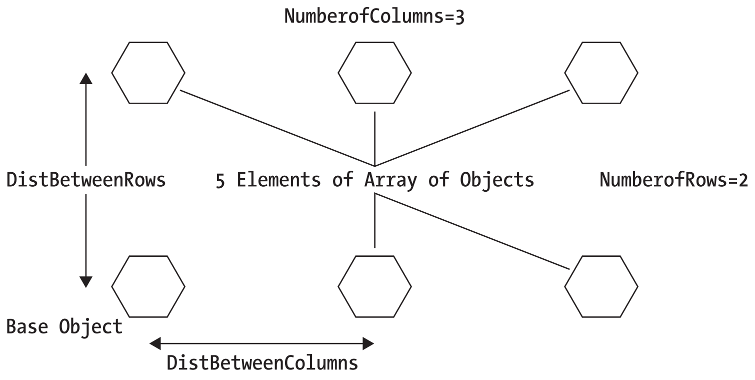


Figure 10-9. *Objects arranged with* `ArrayRectangular`

To create a two-dimensional array, set the `NumberOfLevels` parameters to 1, as shown in the following code example. Again, the new objects will be colored red in the array.

```
Public Sub Create2DRectangularArray()
      Dim objDrawingObject As AcadEntity
      Dim varEntityPickedPoint As Variant
      Dim lngNoRows As Long
      Dim lngNoColumns As Long
      Dim dblDistRows As Long
      Dim dblDistCols As Long
      Dim varRectangularArray As Variant
      Dim intCount As Integer

On Error Resume Next
    ThisDrawing.Utility.GetEntity objDrawingObject, varEntityPickedPoint, _
          "Please pick an entity to form the basis of a rectangular array: "
    If objDrawingObject Is Nothing Then
        MsgBox "You did not choose an object"
        Exit Sub
    End If

    lngNoRows = ThisDrawing.Utility.GetInteger( _
                         "Enter the required number of rows: ")
    lngNoColumns = ThisDrawing.Utility.GetInteger( _
                         "Enter the required number of columns: ")
    dblDistRows = ThisDrawing.Utility.GetReal( _
                         "Enter the required distance between rows: ")
    dblDistCols = ThisDrawing.Utility.GetReal( _
                         "Enter the required distance between columns: ")

    varRectangularArray = objDrawingObject.ArrayRectangular(lngNoRows, _
                         lngNoColumns, 1, dblDistRows, dblDistCols, 0)
    For intCount = 0 To UBound(varRectangularArray)
        varRectangularArray(intCount).Color = acRed
        varRectangularArray(intCount).Update
    Next

End Sub
```

Editing with Properties

Unlike the methods described previously, you can use a drawing object's properties to modify the on-screen appearance of the lines that represent AutoCAD objects. The rest of this chapter covers some of the most commonly used properties.

Changing an Object's Color

Use the `Color` property to read or set a drawing object's color. The `Group` and `Layer` objects also expose this property. Beginning with AutoCAD 2004, all objects have a new `TrueColor` property that supports the Color property. This new property uses RGB color values, Pantone color palettes, and color-book values. This property has the following syntax:

```
Object.Color = lngColor
```

lngColor may be one of 257 colors, 9 of which are represented by the predefined AutoCAD AcColor constants detailed in Table 10-10.

Table 10-10. AcColor *Constants*

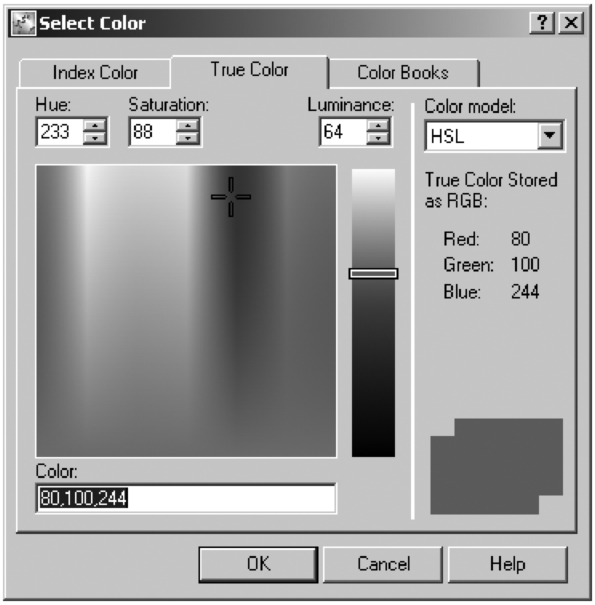
Constant	Color Index	Color
acByBlock	0	ByBlock
acRed	1	Red
acYellow	2	Yellow
acGreen	3	Green
acCyan	4	Cyan
acBlue	5	Blue
acMagenta	6	Magenta
acWhite	7	White or Black, depending on the screen background color
acByLayer	256	ByLayer

The following code changes the color of user-selected objects to green.

```
Public Sub ColorGreen()

      Dim objSelectionSet As AcadSelectionSet
      Dim objDrawingObject As AcadEntity

    'choose a selection set name that you only use as temporary storage and
    'ensure that it does not currently exist
    On Error Resume Next
    ThisDrawing.SelectionSets("TempSSet").Delete
    Set objSelectionSet = ThisDrawing.SelectionSets.Add("TempSSet")

    'ask user to pick entities on the screen
    objSelectionSet.SelectOnScreen

    For Each objDrawingObject In objSelectionSet
        objDrawingObject.Color = acGreen
        objDrawingObject.Update
    Next

    objSelectionSet.Delete

End Sub
```

Changing an Object's `TrueColor` Property

AutoCAD 2004's `TrueColor` property lets you specify RGB color values, Pantone color palette values, and color-book values. This is a vast improvement for users who need accurate color matching for precision rendering and presentation graphics. The following example shows how to create a new `Line` entity and assign it an RGB color value of 80, 100, 244. Figure 10-10 also shows how the new color palette options appear in AutoCAD 2004.


Select Color

Hue: 233
Saturation: 88
Luminance: 64
Color model: HSL

True Color Stored as RGB:

Red: 80
Green: 100
Blue: 244

Color: 80,100,244

Figure 10-10. *Color palette options*

```
Sub Example_TrueColor()
    ' This example draws a line and returns the RGB values
    Dim color As AcadAcCmColor
    Set color = AcadApplication.GetInterfaceObject("AutoCAD.AcCmColor.16")
    Call color.SetRGB(80, 100, 244)

    Dim line As AcadLine
    Dim startPoint(0 To 2) As Double
    Dim endPoint(0 To 2) As Double

    startPoint(0) = 1#: startPoint(1) = 1#: startPoint(2) = 0#
    endPoint(0) = 5#: endPoint(1) = 5#: endPoint(2) = 0#

    Set line = ThisDrawing.ModelSpace.AddLine(startPoint, endPoint)
    ZoomAll
    line.TrueColor = color
    Dim retcolor As AcadAcCmColor
    Set retcolor = line.TrueColor

    MsgBox "Red = " & retcolor.Red & vbCrLf & _
           "Green = " & retcolor.Green & vbCrLf & _
           "Blue = " & retcolor.Blue
End Sub
```

The AutoCAD 2006 developer's documentation gives more examples for using the `TrueColor` object.

Changing an Object's Color Properties

The `AcCmColor` object represents colors. Use the `AcCmColor` object to set colors and perform other color-related operations on objects. This object belongs to the AutoCAD class `AcadAcCmColor`. Invoke this class using one of these two methods:

- `GetInterfaceObject ("AutoCAD.AcCmColor.16")`
- `Dim col As New AcadAcCmColor`

The object includes properties for colors, color names, color-book names, and the color index. The following example shows how to invoke the `SetColorBook` method to use Pantone color books on a `Circle` entity.

```
    Dim fColor As AcadAcCmColor
    Set fColor = AcadApplication.GetInterfaceObject("AutoCAD.AcCmColor.16")
    Call color.SetRGB(80, 100, 244) _
    Call fColor.SetColorBookColor("PANTONE Yellow", _
        "Pantone solid colors-uncoated.acb")
    objCircle.TrueColor = fColor
```

The AutoCAD 2004 developer's documentation gives more examples for using the `AcCmColor` object.

Changing an Object's Layer

By changing a drawing or `Group` object's layer, any of the object's properties that are set to `acByLayer` change to reflect the new layer's attributes. Change an object's layer by setting its `Layer` property to the name of a different layer, as this syntax shows:

```
Object.Layer = strLayerName
```

The following code demonstrates the effects of changing a layer. It adds two circles to layer 0. The first circle is red, while the other circle's color is left at its default value of `ByLayer`. When the circles are moved to a new layer, the first circle stays red while the second adopts the new layer's color.

```
Public Sub ChangeLayer()
      Dim objNewLayer As AcadLayer
      Dim objCircle1 As AcadCircle
      Dim objCircle2 As AcadCircle
      Dim dblCenter1(2) As Double
      Dim dblCenter2(2) As Double
      dblCenter2(0) = 10#

    'reference a layer called "New Layer" is it exists or
    'add a new layer if it does not
    Set objNewLayer = ThisDrawing.Layers.Add("New Layer")

    objNewLayer.Color = acBlue

    ThisDrawing.ActiveLayer = ThisDrawing.Layers("0")
    Set objCircle1 = ThisDrawing.ModelSpace.AddCircle(dblCenter1, 10#)
    Set objCircle2 = ThisDrawing.ModelSpace.AddCircle(dblCenter2, 10#)
    objCircle1.Color = acRed
    objCircle1.Update

    objCircle1.Layer = "New Layer"
    objCircle2.Layer = "New Layer"

End Sub
```

Changing an Object's Linetype

To change the appearance of the lines that represent a drawing object on the screen, reset the object's `Linetype` property to the name of a `Linetype` object. This property also applies to the `Group` and `Layer` objects. This property has the following syntax:

```
Object.Linetype = strLinetypeName
```

This simple code sample sets an object's linetype to be continuous:

```
Public Sub ChangeLinetype()
      Dim objDrawingObject As AcadEntity
      Dim varEntityPickedPoint As Variant

On Error Resume Next
    ThisDrawing.Utility.GetEntity objDrawingObject, varEntityPickedPoint, _
                              "Please pick an object"
    If objDrawingObject Is Nothing Then
        MsgBox "You did not choose an object"
        Exit Sub
    End If
    ThisDrawing.Utility.GetEntity objDrawingObject, varEntityPickedPoint, _
                                  "Pick an entity to change linetype: "
    objDrawingObject.Linetype = "Continuous"
    objDrawingObject.Update

End Sub
```

■ **Tip** Always use strict error handling should someone assign a nonexistent linetype to an object. If you don't, your programs will crash. Chapter 6 covers `Linetype` objects in detail.

Changing an Object's Visibility

Use the `Visible` property to set whether a drawing object is visible or not. This method is also available to the `Application`, `Group`, and `Toolbar` objects, although for a `Group` it is write-only. Set this property `True` to make an object visible, and `False` to hide it. Making an object invisible can be useful for performing complex entity creation or modification tasks. This property has the following syntax:

```
Object.Visible = blnVisible
```

The following example shows how to implement this method. It hides, and then makes visible, an object.

```
Public Sub ToggleVisibility()
      Dim objDrawingObject As AcadEntity
      Dim varEntityPickedPoint As Variant

On Error Resume Next
    ThisDrawing.Utility.GetEntity objDrawingObject, varEntityPickedPoint, _
                                  "Choose an object to toggle visibility: "
    If objDrawingObject Is Nothing Then
        MsgBox "You did not choose an object"
        Exit Sub
    End If

  objDrawingObject.Visible = False
  objDrawingObject.Update
  MsgBox "The object was made invisible!"
  objDrawingObject.Visible = True
  objDrawingObject.Update
  MsgBox "Now it is visible again!"

End Sub
```

Note Other factors, such as whether the object's layer is frozen or turned off, can override this property.

The `Update` Method

This chapter and most of this book use the `Update` method extensively. It forces the AutoCAD graphics display engine to regenerate a specified object on the drawing screen. You need to do this often to accurately reflect changes made to objects.

Summary

This chapter examined several methods and properties you use to edit drawing entities. The next chapter covers adding dimensions and annotations to AutoCAD drawings.

CHAPTER 11

Dimensions and Annotations

Although AutoCAD provides an environment that allows users to make detailed and precise drawings, it's often necessary for users to incorporate further information. A typical user will add textual annotations and measurement annotations (*dimensions*) to a drawing for manufacturing, modeling, engineering, mapping, and surveying purposes and to help clarify design intent. AutoCAD VBA provides a number of methods to help users create annotations and dimensions that they normally create through the AutoCAD application interface.

In this chapter you'll learn how to create dimensions and annotations and add them to your drawings. You'll also see how to set up and use different text and dimension styles.

Working with Dimensions

You add dimension entities to drawings to show the dimension or size of different drawing elements. You can use them to measure angles, distances, and chords. In this section, you'll examine each of the seven dimension objects in turn. First, however, you'll look at the `DimStyle` object, which you use to determine the appearance of dimensions, leaders, and geometric tolerances.

Using the `DimStyle` Object

The `DimStyle` object represents some settings that determine the appearance of a dimension, tolerance, or leader. `DimStyle` objects are held in the `DimStyles` collection, and you may access them via the `DimStyles` collection's `Item` method.

To create a new `DimStyle` object through code, you need to use the `Add` method of the `DimStyles` collection.

```
Set DimStyleObject = DimStylesCollection.Add(DimStyleName)
```

Table 11-1 shows the `Add` method's parameter.

Table 11-1. *The* `Add` *Method's Parameter*

Name	Data Type	Description
`DimStyleName`	String	An identifying name for the new `DimStyle` object

The following code shows how to add a new DimStyle object:

```
Dim objDimStyle As AcadDimStyle

Set objDimStyle = ThisDrawing.DimStyles.Add("NewDimStyle")
```

Setting Dimension Styles

A DimStyle object that is newly created using the Add method inherits the dimension styles of the current active DimStyle object, regardless of any system variable settings that might be overriding the styles of the active DimStyle object. To set the styles for a DimStyle object, you need to set the system variables to reflect your required styles and then use the CopyFrom method explained in the next section.

Using the CopyFrom Method

You use the CopyFrom method to copy to a DimStyle object a set of dimension styles from a dimension, tolerance, leader, document, or another DimStyle object.

```
DimStyleObject.CopyFrom SourceObject
```

The SourceObject parameter is the object whose dimension styles are copied. The styles copied depend upon the source object used, as shown in Table 11-2.

Table 11-2. *The* CopyFrom *Method's Parameters*

Object	Styles Copied
Dimension, Tolerance, Leader	The style for that object plus any object overrides
Document	The active dimension style settings for the drawing plus any drawing overrides
DimStyle	The style data from that dimension style but no drawing overrides

The following code creates a new dimension style called NewDimStyle. This object inherits all the dimension style properties of the currently active dimension style, except that the color of dimension lines, extension lines, and dimension text are set to red, blue, and white, respectively.

```
Public Sub NewDimStyle
Dim objDimStyle As AcadDimStyle

    Set objDimStyle = ThisDrawing.DimStyles.Add("NewDimStyle")
    SetVariable "DIMCLRD", acRed
    SetVariable "DIMCLRE", acBlue
    SetVariable "DIMCLRT", acWhite
    SetVariable "DIMLWD", acLnWtByLwDefault
    objDimStyle.CopyFrom ThisDrawing
End Sub
```

Using Dimension Styles

You may associate a dimension style with a particular dimension object by setting its `StyleName` property, or you may set it to be the currently active style via the `ActiveDimStyle` property of the `Document` object. In the latter case, any objects created after the style is made active and not specifically associated with a different dimension style will adopt the style of the active `DimStyle` object.

Note No matter which property you use to set the dimension style for an object added to your drawing through code, the newly created object will not adopt any system variable overrides.

The `StyleName` Property

You use this property to set the dimension style associated with a dimension, tolerance, or leader. To set or change the `DimStyle` used, simply set or reset the value of this property.

```
Object.StyleName = DimStyleName
```

Table 11-3 shows the `StyleName` property's parameters.

Table 11-3. *The* `StyleName` *Property's Parameters*

Name	Data Type	Description
`Object`	`Dimension`, `Leader`, or `Tolerance` object	The object to which the dimension style is linked
`DimStyleName`	String	The identifying name of the `DimStyle` object

The following example asks the user to choose a dimension and changes the style of the dimension to one of the existing dimension styles. The `StyleName` property of the chosen object is then set appropriately.

```
Public Sub ChangeDimStyle()
Dim objDimension As AcadDimension
Dim varPickedPoint As Variant
Dim objDimStyle As AcadDimStyle
Dim strDimStyles As String
Dim strChosenDimStyle As String

On Error Resume Next
    ThisDrawing.Utility.GetEntity objDimension, varPickedPoint, _
                              "Pick a dimension whose style you wish to set"
    If objDimension Is Nothing Then
        MsgBox "You failed to pick a dimension object"
        Exit Sub
    End If
    For Each objDimStyle In ThisDrawing.DimStyles
        strDimStyles = strDimStyles & objDimStyle.Name & vbCrLf
    Next objDimStyle
    strChosenDimStyle = InputBox("Choose one of the following " & _
                        "Dimension styles to apply" & vbCrLf & strDimStyles)
    If strChosenDimStyle = "" Then Exit Sub

    objDimension.StyleName = strChosenDimStyle
End Sub
```

The ActiveDimStyle Property

You use the ActiveDimStyle property to make a dimension style the default for any newly created dimension, tolerance, or leader objects. Unlike the ActiveTextStyle property, the ActiveDimStyle property doesn't affect any preexisting objects.

```
Set DocumentObject.ActiveDimStyle = DimStyleObject
```

Table 11-4 shows the ActiveTextStyle property's parameter.

Table 11-4. *The* ActiveDimStyle *Property's Parameter*

Name	Data Type	Description
DimStyleObject	DimStyle object	The object holding the setting for the required dimension style

The following code displays the current dimension style to the user and provides an opportunity for the user to change it:

```
Public Sub SetActiveDimStyle()
Dim strDimStyles As String
Dim strChosenDimStyle As String
Dim objDimStyle As AcadDimStyle

    For Each objDimStyle In ThisDrawing.DimStyles
        strDimStyles = strDimStyles & objDimStyle.Name & vbCrLf
    Next

    strChosenDimStyle = InputBox("Choose one of the following Dimension " &  _
     "styles:" & vbCr & vbCr & strDimStyles, "Existing Dimension style is: " &  _
     ThisDrawing.ActiveDimStyle.Name, ThisDrawing.ActiveDimStyle.Name)
    If strChosenDimStyle = "" Then Exit Sub

On Error Resume Next
    ThisDrawing.ActiveDimStyle = ThisDrawing.DimStyles(strChosenDimStyle)
    If Err Then MsgBox "Dimension style was not recognized"
End Sub
```

Creating Dimensions

Dimension objects are entities that provide information about distances and angles within an AutoCAD drawing. It makes sense then that, just as for the drawing objects covered in Chapters 7 and 8, the `AddDimXXX` methods used to add dimensions to drawings are exposed by the `ModelSpace`, `PaperSpace`, and `Block` objects.

It's worth noting that not all properties of dimension entities are exposed through the ActiveX object model. For example, the dimension line control points of `DimAligned` objects are available only as DXF 13 and 14 codes. In addition, some dimension objects behave differently between AutoCAD 2002 and 2004 (and later versions), and I will emphasize that developers should research the changes in newer versions, especially with respect to dimensions and dimension styles.

The `Dim3PointAngular` Object

The `Dim3PointAngular` object displays the angular distance between three points. You can use the `AddDim3PointAngular` method to add this type of dimension to a drawing.

```
Set Dim3PointAngularObject = _
Object.AddDim3PointAngular(VertexPoint, Point1, Point2, _
 TextPosition)
```

Table 11-5 shows the `AddDim3PointAngular` method's parameters.

Table 11-5. *The* `AddDim3PointAngular` *Method's Parameters*

Name	Data Type	Description
`VertexPoint`	Variant	A three-element array of doubles specifying the WCS position of the vertex whose angle is to be measured
`Point1`	Variant	A three-element array of doubles specifying the WCS position of one of the end points
`Point2`	Variant	A three-element array of doubles specifying the WCS position of the other end point
`TextPosition`	Variant	A three-element array of doubles specifying the WCS position of the text displaying the angle size

Figure 11-1 depicts a three-point angular-style dimension.

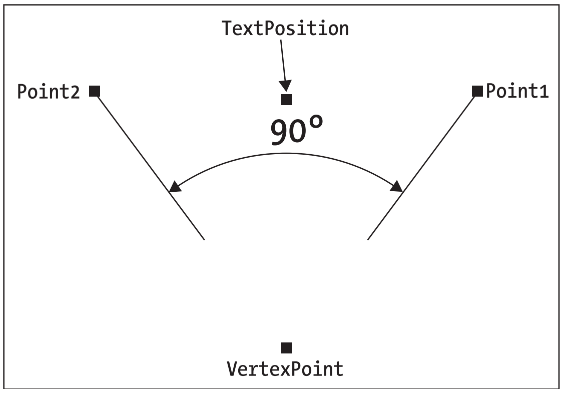


Figure 11-1. *A three-point angular dimension*

This code illustrates how to implement the `AddDim3PointAngular` method:

```
Public Sub Add3PointAngularDimension()
Dim varAngularVertex As Variant
Dim varFirstPoint As Variant
Dim varSecondPoint As Variant
Dim varTextLocation As Variant
Dim objDim3PointAngular As AcadDim3PointAngular

    'Define the dimension
    varAngularVertex = ThisDrawing.Utility.GetPoint(, _
                                            "Enter the center point: ")
    varFirstPoint = ThisDrawing.Utility.GetPoint(varAngularVertex, _
                                            "Select first point: ")
    varSecondPoint = ThisDrawing.Utility.GetPoint(varAngularVertex, _
                                            "Select second point: ")
    varTextLocation = ThisDrawing.Utility.GetPoint(varAngularVertex, _
                "Pick dimension text location: ")

    Set objDim3PointAngular = ThisDrawing.ModelSpace.AddDim3PointAngular( _
        varAngularVertex, varFirstPoint, varSecondPoint, varTextLocation)
    objDim3PointAngular.Update
End Sub
```

The `DimAligned` Object

You use the `DimAligned` object to display the length of a line. Extension lines emanate at right angles from the ends of the line to be measured to the level of the dimension text. You use the `AddDimAligned` method to add this object to your drawing.

```
Set DimAlignedObject = Object.AddDimAligned(Point1, Point2, TextPosition)
```

Table 11-6 shows the `AddDimAligned` method's parameters.

Table 11-6. *The* `AddDimAligned` *Method's Parameters*

Name	Data Type	Description
`Point1`	Variant	A three-element array of doubles specifying the WCS position of one end of the line to be measured
`Point2`	Variant	A three-element array of doubles specifying the WCS position of the other end of the line to be measured
`TextPosition`	Variant	A three-element array of doubles specifying the WCS position of the text to be displayed

Figure 11-2 depicts an aligned-style dimension.

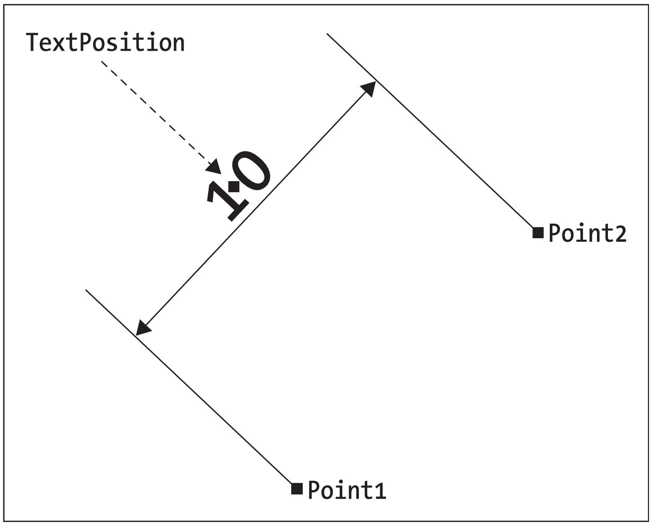


Figure 11-2. *An aligned dimension*

The following example asks users to select the points they would like to dimension. It then uses the `AddDimAligned` method to add the dimension. Finally, the units are changed to engineering units.

```
Public Sub AddAlignedDimension()
Dim varFirstPoint As Variant
Dim varSecondPoint As Variant
Dim varTextLocation As Variant
Dim objDimAligned As AcadDimAligned
    'Define the dimension
    varFirstPoint = ThisDrawing.Utility.GetPoint(, "Select first point: ")
    varSecondPoint = ThisDrawing.Utility.GetPoint(varFirstPoint, _
                                                  "Select second point: ")
    varTextLocation = ThisDrawing.Utility.GetPoint(, _
                                           "Pick dimension text location: ")
    'Create an aligned dimension
    Set objDimAligned = ThisDrawing.ModelSpace.AddDimAligned(varFirstPoint, _
                                              varSecondPoint, varTextLocation)
    objDimAligned.Update

    MsgBox "Now we will change to Engineering units format"
    objDimAligned.UnitsFormat = acDimLEngineering
    objDimAligned.Update
End Sub
```

The `DimAngular` Object

The `DimAngular` object is much like the `Dim3PointAngular` object. It's designed to hold dimension text displaying the angle between two lines or spanned by an arc or circle. You use the `AddDimAngular` method to create a new `DimAngular` object.

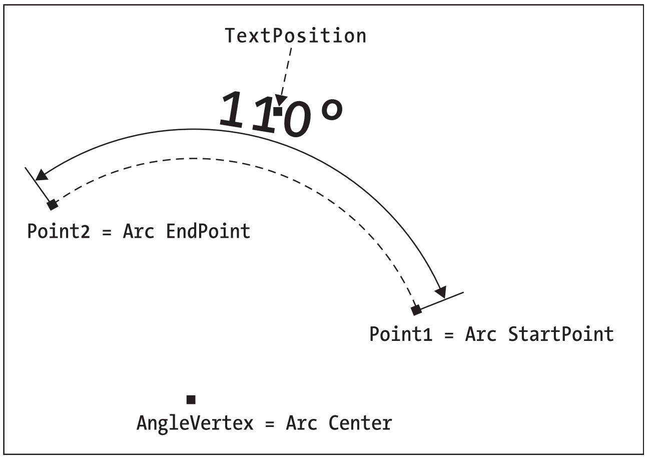
```
Set DimAngularObject = Object.AddDimAngular(Vertex, Point1, Point2, TextPosition)
```

Table 11-7 shows the `AddDimAngular` method's parameters.

Table 11-7. *The* `AddDimAngular` *Method's Parameters*

Name	Data Type	Description
`Vertex`	Variant	A three-element array of doubles specifying the WCS position of the vertex whose angle is to be measured
`Point1`	Variant	A three-element array of doubles specifying the WCS position of one of the end points
`Point2`	Variant	A three-element array of doubles specifying the WCS position of the other end point
`TextPosition`	Variant	A three-element array of doubles specifying the WCS position of the text displaying the angle size

Figure 11-3 depicts an angular dimension for an arc.



The following code sample shows how to use the `AddDimAngular` method. Once you've added the new dimension, its units are changed to degrees, minutes, and seconds.

```
Public Sub AddAngularDimension()
Dim varAngularVertex As Variant
Dim varFirstPoint As Variant
Dim varSecondPoint As Variant
Dim varTextLocation As Variant
Dim objDimAngular As AcadDimAngular

    'Define the dimension
    varAngularVertex = ThisDrawing.Utility.GetPoint(, _
                                             "Enter the center point: ")
    varFirstPoint = ThisDrawing.Utility.GetPoint(varAngularVertex, _
                                             "Select first point: ")
    varSecondPoint = ThisDrawing.Utility.GetPoint(varAngularVertex, _
                                              "Select second point: ")
    varTextLocation = ThisDrawing.Utility.GetPoint(varAngularVertex, _
            "Pick dimension text location: ")

    'Create an angular dimension
    Set objDimAngular = ThisDrawing.ModelSpace.AddDimAngular( _
           varAngularVertex, varFirstPoint, varSecondPoint, varTextLocation)
    objDimAngular.AngleFormat = acGrads
    objDimAngular.Update
    MsgBox "Angle measured in GRADS"

    objDimAngular.AngleFormat = acDegreeMinuteSeconds
    objDimAngular.TextPrecision = acDimPrecisionFour
    objDimAngular.Update
    MsgBox "Angle measured in Degrees Minutes Seconds"
End Sub
```

The `DimDiametric` Object

The `DimDiametric` object represents a dimension showing the length of a chord across a circle or arc. The `AddDimDiametric` method creates a new diametric dimension.

```
DimDiametricObject = Object.AddDimDiametric _
(ChordPoint1, ChordPoint2, LeaderLength)
```

Table 11-8 shows the `AddDimDiametric` method's parameters.

Table 11-8. *The* `AddDimDiametric` *Method's Parameters*

Name	Data Type	Description
`ChordPoint1`	Variant	A three-element array of doubles specifying the WCS position of one end of the chord to be measured
`ChordPoint2`	Variant	A three-element array of doubles specifying the WCS position of the other end of the chord to be measured
`LeaderLength`	Double	The distance from `ChordPoint1` to the dimension text

Figure 11-4 depicts a diametric-style dimension.

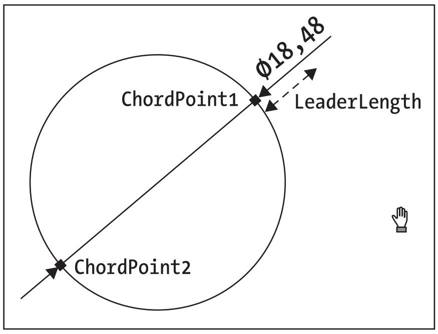


Figure 11-4. *A diametric dimension*

The following code sample adds a new diametric dimension. The object snap mode setting is changed so that the users can select only the diameter of their chosen circle. After the dimension has been added, this setting returns to its original value.

```
Public Sub AddDiametricDimension()
Dim varFirstPoint As Variant
Dim varSecondPoint As Variant
Dim dblLeaderLength As Double
Dim objDimDiametric As AcadDimDiametric
Dim intOsmode As Integer

    'get original object snap settings
    intOsmode = ThisDrawing.GetVariable("osmode")
    ThisDrawing.SetVariable "osmode", 512 ' Near

    With ThisDrawing.Utility
        varFirstPoint = .GetPoint(, "Select first point on circle: ")
        ThisDrawing.SetVariable "osmode", 128 ' Per
        varSecondPoint = .GetPoint(varFirstPoint, _
                                   "Select a point opposite the first: ")
        dblLeaderLength = .GetDistance(varFirstPoint, _
                                   "Enter leader length from first point: ")
    End With
    Set objDimDiametric = ThisDrawing.ModelSpace.AddDimDiametric( _
                             varFirstPoint, varSecondPoint, dblLeaderLength)
    objDimDiametric.UnitsFormat = acDimLEngineering
    objDimDiametric.PrimaryUnitsPrecision = acDimPrecisionFive
    objDimDiametric.FractionFormat = acNotStacked
    objDimDiametric.Update

    'reinstate original object snap settings
    ThisDrawing.SetVariable "osmode", intOsmode
End Sub
```

The `DimOrdinate` Object

The `DimOrdinate` object represents dimensioning text displaying the absolute value of the X-coordinate or Y-coordinate of a given point. You use the `AddDimOrdinate` method to create a new ordinate dimension.

```
Set DimOrdinateObject = Object.AddDimOrdinate(DefinitionPoint, _
LeaderEndPoint, UseXAxis)
```

Table 11-9 shows the `AddDimOrdinate` method's parameters.

Table 11-9. *The* `AddDimOrdinate` *Method's Parameters*

Name	Data Type	Description
`DefinitionPoint`	Variant	A three-element array of doubles specifying, with respect to the WCS, the point whose projection onto the X-axis or Y-axis is to be measured.
`LeaderEndPoint`	Variant	A three-element array of doubles specifying the WCS position of the dimension text placed at the end of an arrow.
`UseXAxis`	Boolean	Determines whether the `DefinitionPoint` is measured from the X-axis or Y-axis. If this is set to `True`, the X-axis is used as a reference.

Figure 11-5 depicts an ordinate-style dimension.

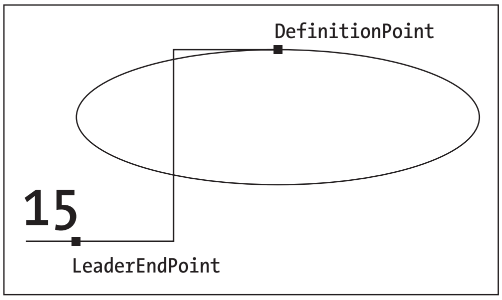


Figure 11-5. *An ordinate dimension*

The following example sets an ordinate dimension for either the X or Y direction:

```
Public Sub AddOrdinateDimension()
Dim varBasePoint As Variant
Dim varLeaderEndPoint As Variant
Dim blnUseXAxis As Boolean
Dim strKeywordList As String
Dim strAnswer As String
Dim objDimOrdinate As AcadDimOrdinate

    strKeywordList = "X Y"
    'Define the dimension
    varBasePoint = ThisDrawing.Utility.GetPoint(, _
                                 "Select ordinate dimension position: ")
    ThisDrawing.Utility.InitializeUserInput 1, strKeywordList
    strAnswer = ThisDrawing.Utility.GetKeyword("Along Which Axis? <X/Y>: ")

    If strAnswer = "X" Then
      varLeaderEndPoint = ThisDrawing.Utility.GetPoint(varBasePoint, _
                                  "Select X point for dimension text: ")
      blnUseXAxis = True
      Else
        varLeaderEndPoint = ThisDrawing.Utility.GetPoint(varBasePoint, _
                                  "Select Y point for dimension text: ")
        blnUseXAxis = False
    End If

    'Create an ordinate dimension
    Set objDimOrdinate = ThisDrawing.ModelSpace.AddDimOrdinate( _
                           varBasePoint, varLeaderEndPoint, blnUseXAxis)
    objDimOrdinate.TextSuffix = "units"
    objDimOrdinate.Update
End Sub
```

The `DimRadial` Object

A radial dimension displays the length of the radius of a circle or arc. You create it using the `AddDimRadial` method.

```
Set DimRadialObject = Object.AddDimRadial (CenterPoint, ChordPoint, LeaderLength)
```

Table 11-10 shows the `AddDimRadial` method's parameters.

Table 11-10. The `AddDimRadial` *Method's Parameters*

Name	Data Type	Description
`CenterPoint`	Variant	A three-element array of doubles specifying the WCS coordinates of the center of an arc or circle
`ChordPoint`	Variant	A three-element array of doubles holding the WCS coordinates of a point on the edge of the circle or arc at which an arrow and the text will be placed
`LeaderLength`	Double	The distance from the text to the `ChordPoint`

Figure 11-6 depicts a radial-style dimension.

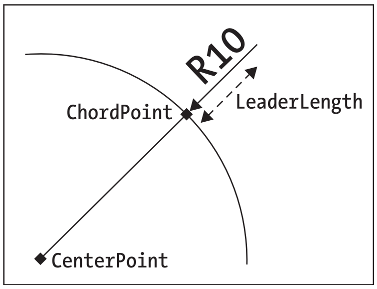


Figure 11-6. *A radial dimension*

The following example illustrates the `AddDimRadial` method by adding a radial dimension to a circle or arc picked by the user:

```
Public Sub AddRadialDimension()
Dim objUserPickedEntity As Object
Dim varEntityPickedPoint As Variant
Dim varEdgePoint As Variant
Dim dblLeaderLength As Double
Dim objDimRadial As AcadDimRadial
Dim intOsmode As Integer

    intOsmode = ThisDrawing.GetVariable("osmode")
    ThisDrawing.SetVariable "osmode", 512 ' Near

    'Define the dimension
    On Error Resume Next
    With ThisDrawing.Utility
        .GetEntity objUserPickedEntity, varEntityPickedPoint, _
                   "Pick Arc or Circle:"
        If objUserPickedEntity Is Nothing Then
            MsgBox "You did not pick an entity"
            Exit Sub
        End If
        varEdgePoint = .GetPoint(objUserPickedEntity.Center, _
                                 "Pick edge point")
        dblLeaderLength = .GetReal("Enter leader length from this point: ")
    End With

    'Create the radial dimension
    Set objDimRadial = ThisDrawing.ModelSpace.AddDimRadial( _
                  objUserPickedEntity.Center, varEdgePoint, dblLeaderLength)
    objDimRadial.ArrowheadType = acArrowArchTick
    objDimRadial.Update
    'reinstate original setting
    ThisDrawing.SetVariable "osmode", intOsmode
End Sub
```

The DimRotated Object

The `DimRotated` object measures the length of the projection of a line onto a direction specified by an angle from the X-axis.

```
Set DimRotatedObject = Object.AddDimRotated(Point1, Point2, _
DimLocationPoint, RotationAngle)
```

Table 11-11 shows the `AddDimRotated` method's parameters.

Table 11-11. *The* `AddDimRotated` *Method's Parameters*

Name	Data Type	Description
`Point1`	Variant	A three-element array of doubles specifying the WCS coordinates of one end of a line
`Point2`	Variant	A three-element array of doubles specifying the WCS coordinates of the other end of a line
`DimLocationPoint`	Variant	A three-element array of doubles specifying the WCS coordinates of a point lying on the line of the arrow and text
`RotationAngle`	Double	The angle of projection measured in radians from the X-axis

Figure 11-7 depicts a rotated-style dimension.

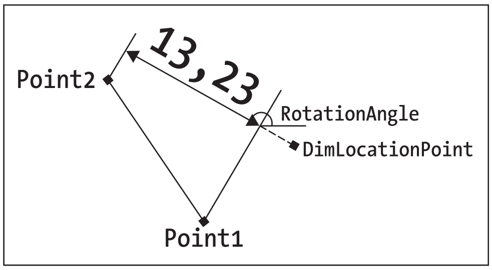


Figure 11-7. *A rotated dimension*

The following example illustrates how to add a rotated dimension at a position and angle chosen by the user:

```
Public Sub AddRotatedDimension()
Dim varFirstPoint As Variant
Dim varSecondPoint As Variant
Dim varTextLocation As Variant
Dim strRotationAngle As String
Dim objDimRotated As AcadDimRotated

    'Define the dimension
    With ThisDrawing.Utility
      varFirstPoint = .GetPoint(, "Select first point: ")
      varSecondPoint = .GetPoint(varFirstPoint, "Select second point: ")
      varTextLocation = .GetPoint(, "Pick dimension text location: ")
      strRotationAngle = .GetString(False, "Enter rotation angle in degrees")
    End With
    'Create a rotated dimension
    Set objDimRotated = ThisDrawing.ModelSpace.AddDimRotated(varFirstPoint, _
      varSecondPoint, varTextLocation, _
      ThisDrawing.Utility.AngleToReal(strRotationAngle, acDegrees))
    objDimRotated.DecimalSeparator = ","
    objDimRotated.Update
End Sub
```

Using the `Tolerance` Object

The `Tolerance` object represents a geometric tolerance in a feature control frame. Tolerances specify allowable deviations of orientation, location, and so on, from the exact geometry of a drawing.

```
Set ToleranceObject = Object.AddTolerance(Text, _
InsertionPoint, TextDirection)
```

Table 11-12 shows the `AddTolerance` method's parameters.

Table 11-12. *The* `AddTolerance` *Method's Parameters*

Name	Data Type	Description
`Text`	String	The text string to be displayed
`InsertionPoint`	Variant	A three-element array of doubles specifying the WCS point at which the tolerance is inserted
`TextDirection`	Variant	A three-element array of doubles specifying the three-dimensional vector for the direction of the tolerance text

Figure 11-8 depicts a tolerance feature flag.

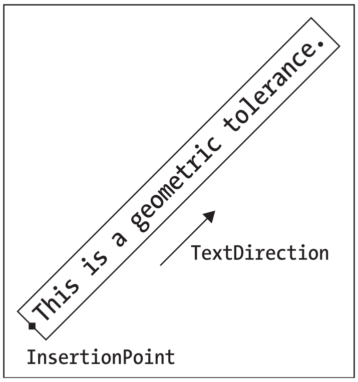


Figure 11-8. *A tolerance feature flag*

This code adds a `Tolerance` object to model space with the user-chosen text, position, and direction:

```
Public Sub CreateTolerance()
Dim strToleranceText As String
Dim varInsertionPoint As Variant
Dim varTextDirection As Variant
Dim intI As Integer
Dim objTolerance As AcadTolerance

  strToleranceText = InputBox("Please enter the text for the tolerance")
  varInsertionPoint = ThisDrawing.Utility.GetPoint(, _
                     "Please enter the insertion point for the tolerance")
  varTextDirection = ThisDrawing.Utility.GetPoint(varInsertionPoint, _
                             "Please enter a direction for the tolerance")

  For intI = 0 To 2
    varTextDirection(intI) = varTextDirection(intI) - varInsertionPoint(intI)
  Next

  Set objTolerance = ThisDrawing.ModelSpace.AddTolerance(strToleranceText, _
                                      varInsertionPoint, varTextDirection)

End Sub
```

Table 11-13 shows the formatting codes for `Tolerance` symbols.

Table 11-13. *Formatting Codes for the* `Tolerance` *Feature's Flag Symbols*

Code	Symbol
`{\Fgdt;a}`	Angularity symbol (similar to `<`)
`{\Fgdt;b}`	Perpendicularity symbol (two perpendicular lines)
`{\Fgdt;c}`	Flatness symbol (parallelogram)
`{\Fgdt;d}`	Profile of a surface symbol (closed half-circle)
`{\Fgdt;e}`	Circularity or roundness symbol (single circle)
`{\Fgdt;f}`	Parallelism symbol (`//`)
`{\Fgdt;g}`	Cylindricity symbol (`/O/`)
`{\Fgdt;h}`	Circular runout symbol (↗)
`{\Fgdt;i}`	Symmetry symbol (like a division sign)
`{\Fgdt;j}`	Positional symbol (circle with a cross in it)
`{\Fgdt;k}`	Profile of a line symbol (open half-circle)
`{\Fgdt;l}`	Least material condition symbol (circle with an *L* in it)
`{\Fgdt;m}`	Maximum material condition symbol (circle with an *M* in it)
`{\Fgdt;n}`	Diameter symbol
`{\Fgdt;o}`	Square symbol
`{\Fgdt;p}`	Projected tolerance zone symbol (circle with a P in it)
`{\Fgdt;q}`	Centerline symbol
`{\Fgdt;r}`	Concentricity symbol (two concentric circles)
`{\Fgdt;s}`	Regardless of feature size (circle with an *S* in it)
`{\Fgdt;t}`	Total runout symbol (↗↗)
`{\Fgdt;u}`	Straightness symbol (`-`)
`{\Fgdt;v}`	Counterbore symbol
`{\Fgdt;w}`	Countersink symbol
`{\Fgdt;x}`	Depth symbol
`{\Fgdt;y}`	Conical taper symbol
`{\Fgdt;z}`	Slope symbol
`%%v`	Used to create a vertical separation line between each symbol

Tolerances are related to the dimension objects and so are influenced by several dimension system variables: `DIMCLRD` controls the color of the feature control frame, `DIMCLRT` controls the color of the tolerance text, `DIMGAP` controls the gap between the feature control frame and the text, `DIMTXT` controls the size of the tolerance text, and `DIMTXTSTY` controls the style of the tolerance text.

Working with Annotations

An *annotation* is simply explanatory text added to a drawing to clarify some aspect of that drawing. The AutoCAD objects you may use for this purpose are the `Text` and `MText` objects, and for blocks you may use the `Attribute` and `AttributeReference` objects (see Chapter 13 for more information). In addition, you can use the `Tolerance` object to add text, but it's more related to the dimension objects, so I cover it later in the "Adding Annotations" section.. The format and style of the text is controlled by the `TextStyle` object as well as a number of system variables. In this section you'll concentrate on the `TextStyle` object and how to set and use text styles.

Using the `TextStyle` Object

A text style determines the appearance of any text you add to a drawing. `TextStyle` objects represent the settings for different styles and are held in the `TextStyles` collection. When you first create a drawing, this collection contains only one `TextStyle` object, named Standard. You can read the settings for this object through code using the `GetFont` method and the `Height`, `Width`, `ObliqueAngle`, and `TextGenerationFlag` properties outlined in the following sections. Alternatively, you can view the settings in the Text Style dialog box, which you access through the Format ➤ Text Style menu of the AutoCAD interface.

Figure 11-9 shows the Text Style dialog box.

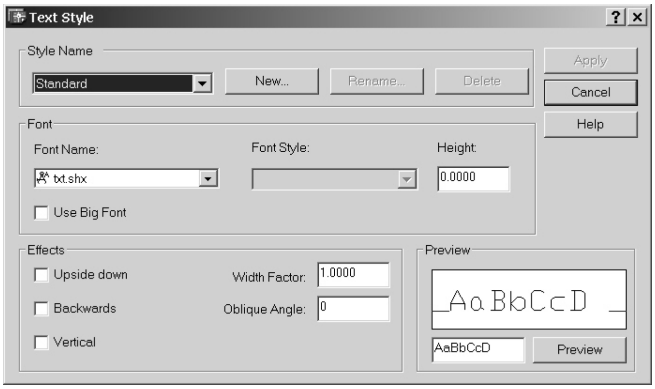

Text Style
Style Name: Standard
Font Name: txt.shx
Font Style:
Height: 0.0000
Use Big Font
Upside down
Backwards
Vertical
Width Factor: 1.0000
Oblique Angle: 0
Preview: AaBbCcD

Figure 11-9. *Viewing settings in the Text Style dialog box*

Creating `TextStyle` Objects

By creating new `TextStyle` objects for different types of text or information, you can have at your fingertips all the text styles that you require for a drawing. To create a new `TextStyle` object, use the `Add` method as follows:

```
Set TextStyleObject = TextStylesCollection.Add(TextStyleName)
```

Table 11-14 shows the `Add` method's parameter.

Table 11-14. *The* `Add` *Method's Parameter*

Name	Data Type	Description
`TextStyleName`	String	An identifying name for the new `TextStyle` object

If you try to add a `TextStyle` with the name of an existing `TextStyle` object, this method will return a reference to the existing object. The newly created object will have the same settings as the active `TextStyle` object.

Retrieving and Setting Text Styles

Once you've created a new `TextStyle` object, you'll need to set its font settings. You may also want to modify or check the settings of an existing `TextStyle` object. You can do this through the `SetFont` and `GetFont` methods. In addition, you can use the font settings held within an `.shx` file through the `FontFile` and `BigFontFile` properties.

The `SetFont` Method

You use the `SetFont` method to define the font for a `TextStyle` object. It sets the typeface, sets whether text will be bold and/or italicized, and sets the character set and pitch and family definitions.

```
TextStyleObject.SetFont TypeFace, Bold, Italic, CharacterSet, PitchAndFamily
```

■ **Note** Many TrueType fonts are licensed and aren't provided for public use without explicit permission. Be sure to comply with any licensing terms if you set a custom font and package the font with your drawing files to send to customers.

Table 11-15 shows the `SetFont` method's parameters.

Table 11-15. *The* `SetFont` *Method's Parameters*

Name	Data Type	Description
`TypeFace`	String	The name of a font, for example, Times New Roman. If the `TextStyle` is determined by an `.shx` file, this parameter holds an empty string.
`Bold`	Boolean	Determines if the text will be bold. If this value is set to `True`, the text will be bold. Otherwise, the text style isn't bold.
`Italic`	Boolean	Determines if the text will be italicized. If this value is set to `True`, the text will be italic. Otherwise, the text style isn't italic.

Continued

Table 11-15. *(Continued)*

Name	Data Type	Description

`CharacterSet`	Long	Specifies the character set for the font. Use the following constants in your application in the declaration section of your code:
		`Public Const ANSI_CHARSET = 0`
		`Public Const DEFAULT_CHARSET = 1`
		`Public Const SYMBOL_CHARSET = 2`
		`Public Const SHIFTJIS_CHARSET = 128`
		`Public Const OEM_CHARSET= 255`

`PitchAndFamily`	Long	Specifies the pitch and family values for the font. The value is a combination of three different settings for pitch, family, and optionally the `TrueType` flag (required only when you're specifying a TrueType font). Use the following constants in your application in the declaration section of your code:
		`' Pitch Values`
		`Public Const DEFAULT_PITCH = 0`
		`Public Const FIXED_PITCH = 1`
		`Public Const VARIABLE_PITCH = 2`
		`' Family Values`
		`Public Const FF_DONTCARE = 0`
		`Public Const FF_ROMAN = 16`
		`Public Const FF_SWISS = 32`
		`Public Const FF_MODERN = 48`
		`Public Const FF_SCRIPT = 64`
		`Public Const FF_DECORATIVE = 80`
		`' TrueType Flag`
		`Public Const TMPF_TRUETYPE = &H4`

This example creates a new `TextStyle` object and sets its font style to bold Greek symbols:

```
Public Sub AddTextStyle
Dim objTextStyle As AcadTextStyle

    Set objTextStyle = ThisDrawing.TextStyles.Add("Bold Greek Symbols")
    objTextStyle.SetFont "Symbol", True, False, 0, 0
End Sub
```

The `GetFont` Method

The `GetFont` method works in much the same way as the `SetFont` method, except it retrieves the font settings.

```
TextStyleObject.GetFont TypeFace, Bold, Italic, CharacterSet, PitchAndFamily
```

Table 11-16 shows the `GetFont` method's parameters.

Table 11-16. *The* `GetFont` *Method's Parameters*

Name	Data Type	Description
`TypeFace`	String	The name of a font, for example, Times New Roman. If the `TextStyle` is determined by a `.shx` file, this parameter holds an empty string.
`Bold`	Boolean	Determines if the text will be bold. If this value is set to `True`, the text will be bold. Otherwise, the text style isn't bold.
`Italic`	Boolean	Determines if the text will be italicized. If this value is set to `True`, the text will be italic. Otherwise the text style isn't italic.
`CharacterSet`	Long	See Table 11-15 for a description of this parameter.
`PitchAndFamily`	Long	See Table 11-15 for a description of this parameter.

The following example asks users for the name of a text style and then presents them with the font settings:

```
Public Sub GetTextSettings()
Dim objTextStyle As AcadTextStyle
Dim strTextStyleName As String
Dim strTextStyles As String
Dim strTypeFace As String
Dim blnBold As Boolean
Dim blnItalic As Boolean
Dim lngCharacterSet As Long
Dim lngPitchandFamily As Long
Dim strText As String

    ' Get the name of each text style in the drawing
    For Each objTextStyle In ThisDrawing.TextStyles
        strTextStyles = strTextStyles & vbCr & objTextStyle.Name
    Next
    ' Ask the user to select the Text Style to look at
    strTextStyleName = InputBox("Please enter the name of the TextStyle " & _
                "whose setting you would like to see" & vbCr & _
                strTextStyles,"TextStyles", ThisDrawing.ActiveTextStyle.Name)
    ' Exit the program if the user input was cancelled or empty
    If strTextStyleName = "" Then Exit Sub

On Error Resume Next
    Set objTextStyle = ThisDrawing.TextStyles(strTextStyleName)
    ' Check for existence the text style
    If objTextStyle Is Nothing Then
        MsgBox "This text style does not exist"
        Exit Sub
    End If
    ' Get the Font properties
    objTextStyle.GetFont strTypeFace, blnBold, blnItalic, lngCharacterSet, _
                lngPitchandFamily
    ' Check for Type face
    If strTypeFace = "" Then  ' No True type
        MsgBox "Text Style: " & objTextStyle.Name & vbCr & _
                "Using file font: " & objTextStyle.fontFile, _
                vbInformation, "Text Style: " & objTextStyle.Name

        Else
        ' True Type font info
       strText = "The text style: " & strTextStyleName & " has " & vbCrLf & _
                 "a " & strTypeFace & " type face"
       If blnBold Then strText = strText & vbCrLf & " and is bold"
       If blnItalic Then strText = strText & vbCrLf & " and is italicized"
       MsgBox strText & vbCr & "Using file font: " & objTextStyle.fontFile, _
              vbInformation, "Text Style: " & objTextStyle.Name
    End If
End Sub
```

The `FontFile` Property

You use the `FontFile` property to set the `TextStyle` object to adopt the font style held in an `.shx` file, or vice versa.

```
TextStyleObject.FontFile = FontFileName
```

Table 11-17 shows the `FontFile` property's parameter.

Table 11-17. *The* `FontFile` *Property's Parameter*

Name	Data Type	Description
`FontFileName`	String	The path and file name of the required font file

The following code sample sets the font file for a new `TextStyle` object named Roman to be the `romand.shx` file:

```
Public Sub SetFontFile
Dim objTextStyle As AcadTextStyle

    Set objTextStyle = ThisDrawing.TextStyles.Add("Roman")
    objTextStyle.fontFile = "romand.shx"
End Sub
```

To set or retrieve the font settings of an Asian-language, big-font file, use the `BigFontFile` property.

Other `TextStyle` Properties

You can use some other properties of a `TextStyle` object to set the way text appears: `Height`, `Width`, `ObliqueAngle`, and `TextGenerationFlag`.

Figure 11-10 shows the effects of setting the `Height` and `Width` properties. The `Height` property simply holds the height of uppercase text in the current drawing units. The `Width` property, on the other hand, sets the character spacing and may lie in the range `0 < Width ≤ 100`. A value of more than 1.0 expands the text; a value of less than 1.0 condenses it.

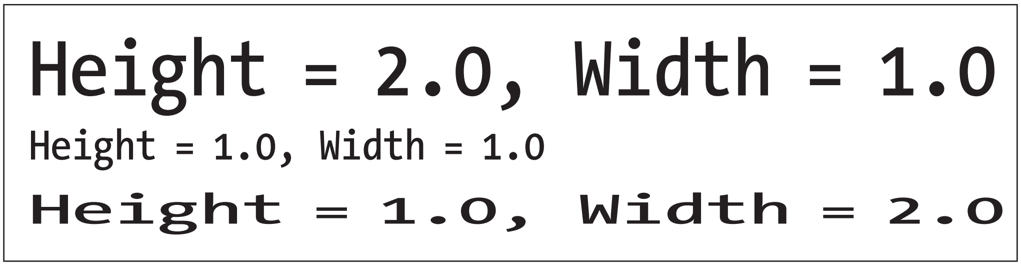

Height = 2.0, Width = 1.0
Height = 1.0, Width = 1.0
Height = 1.0, Width = 2.0

Figure 11-10. *Some* `Height` *variations*

You can use the `ObliqueAngle` property to set the angle of slant of the text away from its vertical axis. You may set it to any value between -858 and 858, with a negative value indicating a slope to the left. Figure 11-11 shows the result of setting this property.

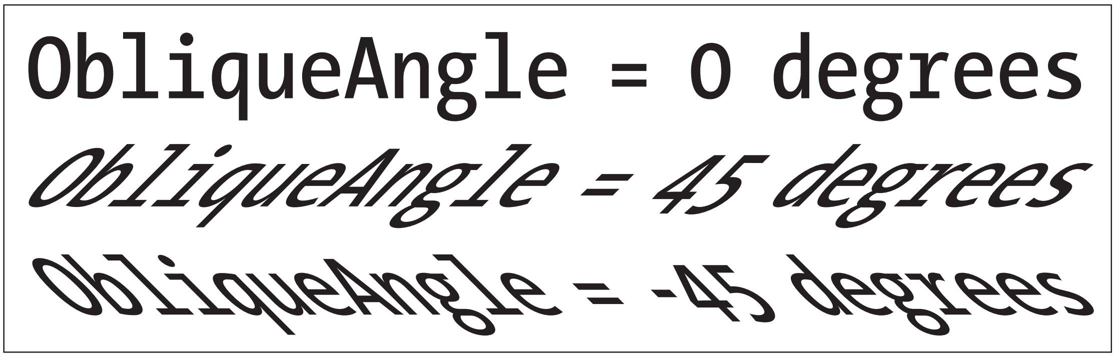


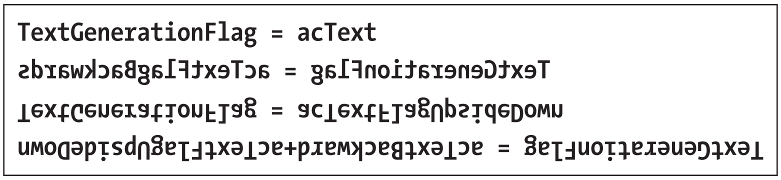
Figure 11-11. *Some* `ObliqueAngle` *variations*

Finally, the `TextGenerationFlag` property determines whether the text is written back to front and/or upside down. This property holds one or a combination of the `acTextGeneration` constants given in Table 11-18.

Table 11-18. `TextGeneration`*'s Constants*

Constant	Value	Description
`acTextFlagBackward`	2	The letters of the text appear in reverse order.
`acTextFlagUpsideDown`	4	The text appears upside down.

Figure 11-12 shows an example of each of the possible settings for the `TextGenerationFlag` property.



```
TextGenerationFlag = acText
TextGenerationFlag = acTextFlagBackwards
TextGenerationFlag = acTextFlagUpsideDown
TextGenerationFlag = acTextBackward+acTextFlagUpsideDown
```

Figure 11-12. *Some* `TextGenerationFlag` *variations*

Using a `TextStyle`

Once you've set up all the text fonts and formats for a `TextStyle` object, you'll probably want to use it. You have two ways to associate text to a particular style. The first way is to use the `StyleName` property, which sets the style for a particular object. The second way is to make a `TextStyle` object active, using the `ActiveStyle` property of the `Document` object, and set the text style for newly created objects and any existing text objects that don't have their `StyleName` property set.

The `StyleName` Property

You use this property to set the style used with an object. To set or change the `TextStyle` used with a `Text`, `MText`, `Attribute`, or `AttributeRef` object, simply set or reset the value of this property.

```
Object.StyleName = TextStyleName
```

Table 11-19 shows the `StyleName` property's parameters.

Table 11-19. *The* `StyleName` *Property's Parameters*

Name	Data Type	Description
`Object`	`Text`, `MText`, `Attribute`, or `AttributeRef` object	The object to which the text style is linked
`TextStyleName`	String	The identifying name of the `TextStyle` object

This code changes all text in model space to adopt a user's chosen text style:

```
Public Sub ChangeTextStyle()
Dim strTextStyles As String
Dim objTextStyle As AcadTextStyle
Dim objLayer As AcadLayer
Dim strLayerName As String
Dim strStyleName As String
Dim objAcadObject As AcadObject

On Error Resume Next
    For Each objTextStyle In ThisDrawing.TextStyles
        strTextStyles = strTextStyles & vbCr & objTextStyle.Name
    Next
    strStyleName = InputBox("Enter name of style to apply:" & vbCr & _
              strTextStyles, "TextStyles", ThisDrawing.ActiveTextStyle.Name)
    Set objTextStyle = ThisDrawing.TextStyles(strStyleName)
    If objTextStyle Is Nothing Then
        MsgBox "Style does not exist"
        Exit Sub
    End If

    For Each objAcadObject In ThisDrawing.ModelSpace
        If objAcadObject.ObjectName = "AcDbMText" Or _
            objAcadObject.ObjectName = "AcDbText" Then
            objAcadObject.StyleName = strStyleName
            objAcadObject.Update
        End If
    Next
End Sub
```

The `ActiveTextStyle` Property

You use the `ActiveTextStyle` property to make a text style the default for any newly created text and for any existing text that doesn't have a particular text style already associated with it.

```
Set DocumentObject.ActiveTextStyle = TextStyleObject
```

Table 11-20 shows the `ActiveTextStyle` property's parameter.

Table 11-20. *The* `ActiveTextStyle` *Property's Parameter*

Name	Data Type	Description
`TextStyleObject`	`TextStyle` object	The object holding the setting for the required text style

The following example sets the default text style based on user input:

```
Public Sub SetDefaultTextStyle()
Dim strTextStyles As String
Dim objTextStyle As AcadTextStyle
Dim strTextStyleName As String

    For Each objTextStyle In ThisDrawing.TextStyles
        strTextStyles = strTextStyles & vbCr & objTextStyle.Name
    Next
    strTextStyleName = InputBox("Enter name of style to apply:" & vbCr & _
              strTextStyles, "TextStyles", ThisDrawing.ActiveTextStyle.Name)
    If strTextStyleName = "" Then Exit Sub

On Error Resume Next
    Set objTextStyle = ThisDrawing.TextStyles(strTextStyleName)
    If objTextStyle Is Nothing Then
        MsgBox "This text style does not exist"
        Exit Sub
    End If

    ThisDrawing.ActiveTextStyle = objTextStyle
End Sub
```

Adding Annotations

To add annotations through VBA code, you need to add a `Text`, `MText`, or `AttributeReference` object. You may add each of these objects to model space, paper space, or a block definition. Chapters 8 and 13 provide details of how to add these objects. At this point, it's worth mentioning the `Leader` object.

Using the Leader Object

A `Leader` object consists of an arrowhead attached to a spline or to straight-line segments. You can use this object to associate text with a drawing object. To add a `Leader` object to your drawing, use the `AddLeader` method.

```
Set LeaderObject = Object.AddLeader(PointsArray, Annotationtype, ArrowType)
```

Table 11-21 shows the `AddLeader` method's parameters.

Table 11-21. *The* `AddLeader` *Method's Parameters*

Name	Data Type	Description
`PointsArray`	Variant	A three-element array of doubles specifying the 3-D WCS coordinates of the vertices of line segments or the fit points of a spline.
`AnnotationType`	Object	The annotation with which the leader is associated. This may by a `Tolerance`, `MText`, or `BlockRef` object or NULL.
`ArrowType`	Long	One of the `AcLeaderType` constants identifying the appearance of the leader.

Figure 11-13 shows each of the `AcLeaderType` constants and its result.

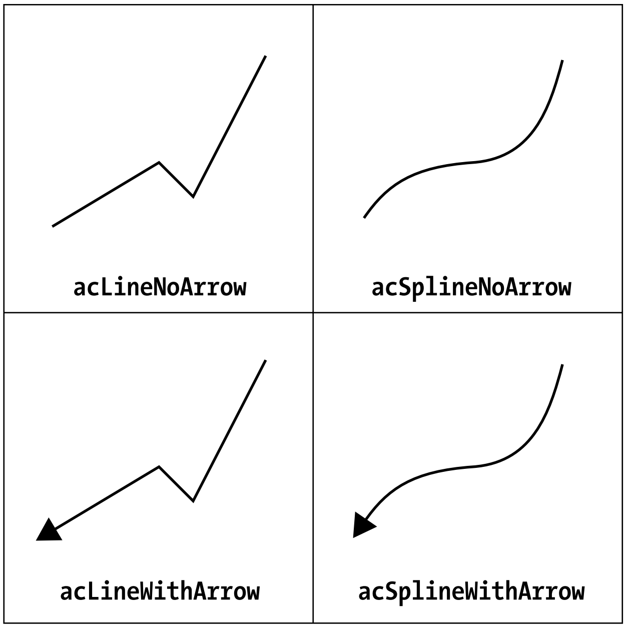


Figure 11-13. *Some* `LeaderType` *variations*

The following example adds a straight-line leader with associated text to model space. The alignment of the text is set depending on the direction of the leader.

```
Public Sub CreateStraightLeaderWithNote()
Dim dblPoints(5) As Double
Dim varStartPoint As Variant
Dim varEndPoint As Variant
Dim intLeaderType As Integer
Dim objAcadLeader As AcadLeader
Dim objAcadMtext As AcadMText
Dim strMtext As String
Dim intI As Integer

    intLeaderType = acLineWithArrow
    varStartPoint = ThisDrawing.Utility.GetPoint(, _
                                             "Select leader start point: ")
    varEndPoint = ThisDrawing.Utility.GetPoint(varStartPoint, _
                                              "Select leader end point: ")

    For intI = 0 To 2
      dblPoints(intI) = varStartPoint(intI)
      dblPoints(intI + 3) = varEndPoint(intI)
    Next

    strMtext = InputBox("Notes:", "Leader Notes")
    If strMtext = "" Then Exit Sub
    ' Create the text for the leader
    Set objAcadMtext = ThisDrawing.ModelSpace.AddMText(varEndPoint, _
    Len(strMtext) * ThisDrawing.GetVariable("dimscale"), strMtext)
    ' Flip the alignment direction of the text
    If varEndPoint(0) > varStartPoint(0) Then
        objAcadMtext.AttachmentPoint = acAttachmentPointMiddleLeft
    Else
        objAcadMtext.AttachmentPoint = acAttachmentPointMiddleRight
    End If
    objAcadMtext.InsertionPoint = varEndPoint

    'Create the leader object
    Set objAcadLeader = ThisDrawing.ModelSpace.AddLeader(dblPoints, _
                                              objAcadMtext, intLeaderType)
    objAcadLeader.Update
End Sub
```

Summary

In this chapter you examined the AutoCAD VBA objects for creating textual and dimension annotations. You use these objects to clarify a drawing's purpose and display the size and orientation of drawing elements. You also looked in detail at how to set up text and dimension styles using the TextStyle and DimStyle objects.

CHAPTER 12

Selection Sets and Groups

AutoCAD allows users to perform many operations on arbitrary sets of entities. The user builds these sets of entities either on an as-needed basis as a *selection set* or more permanently as a *group*. Selection sets (also referred to as PickSets) are transient and are destroyed when AutoCAD closes a drawing. Groups, on the other hand, are stored persistently in the drawing file.

This chapter covers the following topics:

- Creating and manipulating selection sets and groups
- Filtering sets of entities using selection set filters
- Interacting with users by building dynamic selection sets
- Working with the PickFirst selection set
- Manipulating groups

Selection Sets

A selection set is a temporary collection of drawing entities. With a selection set, users can perform an operation such as move, copy, or delete on a number of entities at once, rather than on one entity at a time.

Each AutoCAD drawing has a `SelectionSets` collection that contains `SelectionSet` objects. You can create any number of selection sets by adding new `SelectionSet` objects to the `SelectionSets` collection. Because selection sets aren't persistent, and so disappear when the drawing is closed, the `SelectionSets` collection will always be empty when a drawing is first opened or created.

Adding a `SelectionSet` Object

Selection sets are usually added to the collection on an as-needed basis. You can create as many as you need, but they're typically transient—you create them on demand and then discard them. You use the `Add` method (see Table 12-1 for parameter details) of the `SelectionSets` collection to create a `SelectionSet` object.

```
Set SelectionSetObject = SelectionSetsCollection.Add(SelectionSetName)
```

Table 12-1 explains this method's parameter.

Table 12-1. *The* Add *Method Parameter*

Name	Data Type	Description
SelectionSetName	String	Name of the newly created selection set

The Name parameter is simply used as a handle for the newly created selection set; it's never shown in the user interface. If a SelectionSet object already exists in the current drawing with that name, an exception is raised. Using the Item method, you can use the Name parameter to retrieve the selection set during the same session in which it was created.

The following example attempts to add a selection set using a name obtained from the user:

```
Public Sub TestAddSelectionSet()
      Dim objSS As AcadSelectionSet
      Dim strName As String

On Error Resume Next
    '' get a name from user
    strName = InputBox("Enter a new selection set name: ")
    If "" = strName Then Exit Sub

    '' create it
    Set objSS = ThisDrawing.SelectionSets.Add(strName)

    '' check if it was created
    If objSS Is Nothing Then
        MsgBox "Unable to Add '" & strName & "'"
    Else
        MsgBox "Added selection set '" & objSS.Name & "'"
    End If
End Sub
```

Try adding a few selection sets. You'll access them again in the next example.

Accessing and Iterating Selection Sets

As I mentioned previously, you access the SelectionSets collection via a Document object. In the following code, ThisDrawing is used as the active document:

```
Dim objSelections As AcadSelectionSets
Set objSelections = ThisDrawing.SelectionSets
```

To set a reference to an existing SelectionSet object, use the Item method of the SelectionSets collection:

```
Dim objSelection As AcadSelectionSet

Set objSelection = objSelections.Item(2)
Set objSelection = objSelections.Item("My SelectionSet")
```

The parameter of this method is either a number representing the position of the desired `SelectionSet` object within the `SelectionSets` collection, or a string representing the name of the desired `SelectionSet` object. If you use an index number, it must be between 0 and `SelectionSets.Count` minus 1. The `Count` property returns the total number of `SelectionSet` objects in the collection.

As with other AutoCAD collections, `Item` is the default method for `SelectionSets`. This means that you may omit the method name and the parameter is passed straight to the `SelectionSets` reference. This is often preferred because it's simpler to type and read, but you should avoid using default methods as Microsoft is moving away from implicitness to explicitness in programming technologies such as .NET. The following code does the same thing as the earlier example using the default method to specify the `SelectionSet` object:

```
Dim objSelection As AcadSelectionSet

Set objSelection = objSelections(2)
Set objSelection = objSelections("My SelectionSet")
```

While this example works, it is recommended that you explicitly indicate the method or property instead of relying on a default method or property.

Iterating the `SelectionSets` Collection

Situations will arise in which a programmer needs to step through each item in a collection one at a time—perhaps to check or alter some property of every element. Like all collections in VBA, `SelectionSets` has built-in support for iteration using a `For ... Each` loop.

The following example iterates the `SelectionSets` collection and displays the name of all the `SelectionSet` objects in a message box. Try it after running the previous `TestAddSelectionSet` example.

```
Public Sub ListSelectionSets()
      Dim objSS As AcadSelectionSet
      Dim strSSList As String

    For Each objSS In ThisDrawing.SelectionSets
        strSSList = strSSList & vbCr & objSS.Name
    Next
    MsgBox strSSList, , "List of Selection Sets"
End Sub
```

Selecting Entities

Newly created `SelectionSet` objects are empty. A `SelectionSet` is populated either by adding an array of objects, which I describe how to do later in this chapter, or by using one of the `SelectXXX` methods I describe in this section. There are `SelectXXX` methods for obtaining entities by specific point, by window, by arbitrary fence or polygon, by dynamic user selection, and by filtering the entities using specific criteria.

The Select Method

The Select method (see Table 12-2 for parameter details) is the basic way to add entities to SelectionSet objects. It allows you to select the previous selection set, the last visible entity, everything in the current space, or rectangular region.

```
SelectionSetObject.Select Mode [, Point1, Point2] _
[, FilterCodes, FilterValues]
```

The Select method has several modes of operation specified by the Mode parameter. AutoCAD provides the following AcSelect enum constants for use with this parameter, as shown in Table 12-3.

Table 12-2. Select *Method Parameters*

Name	Data Type	Description
Mode	Long	Determines the selection mode to be used. It must be one of the AcSelect constants described in Table 12-3.
Point1	Variant	An optional three-element array of doubles indicating the first corner of a rectangular region. It must be used in conjunction with Point2.
Point2	Variant	An optional three-element array of doubles indicating the other corner of a rectangular region. It must be used in conjunction with Point1.
FilterCodes	Variant	An optional array of integer entity selection filter codes. It must be used in conjunction with, and have the same length as, FilterValues.
FilterValues	Variant	An optional array of variant entity selection filter values. It must be used in conjunction with, and have the same length as, FilterCodes.

Table 12-3. Select *Method Modes*

Constant	Value	Description
acSelectionSetWindow	0	Entities completely contained within a window specified by Point1 and Point2 are selected.
acSelectionSetCrossing	1	Entities contained or crossing over a window specified by Point1 and Point2 are selected.
acSelectionSetPrevious	3	Entities from the most recent selection set in the current space are selected. Point1 and Point2 aren't used.
acSelectionSetLast	4	The most recently created entity within the current viewport is selected. Point1 and Point2 aren't used.
acSelectionSetAll	5	All entities in the current space are selected. Point1 and Point2 aren't used.

■**Note** The Select method is capable of selecting entities on all layers, even if a certain layer is frozen or locked.

The following example creates a new selection set and then populates it based on user input. Try each of the modes to see how they differ.

```
Public Sub TestSelect()
      Dim objSS As AcadSelectionSet
      Dim varPnt1 As Variant
      Dim varPnt2 As Variant
      Dim strOpt As String
      Dim lngMode As Long

On Error GoTo Done
    With ThisDrawing.Utility

        '' get input for mode
        .InitializeUserInput 1, "Window Crossing Previous Last All"
        strOpt = .GetKeyword(vbCr & _
        "Select [Window/Crossing/Previous/Last/All]: ")

        '' convert keyword into mode
        Select Case strOpt
        Case "Window":  lngMode = acSelectionSetWindow
        Case "Crossing":  lngMode = acSelectionSetCrossing
        Case "Previous":  lngMode = acSelectionSetPrevious
        Case "Last":  lngMode = acSelectionSetLast
        Case "All":  lngMode = acSelectionSetAll
        End Select

        '' create a new selectionset
        Set objSS = ThisDrawing.SelectionSets.Add("TestSelectSS")

        '' if it's window or crossing, get the points
        If "Window" = strOpt Or "Crossing" = strOpt Then

            '' get first point
            .InitializeUserInput 1
            varPnt1 = .GetPoint(, vbCr & "Pick the first corner: ")

            '' get corner, using dashed lines if crossing
            .InitializeUserInput 1 + IIf("Crossing" = strOpt, 32, 0)
            varPnt2 = .GetCorner(varPnt1, vbCr & "Pick other corner: ")

            '' select entities using points
            objSS.Select lngMode, varPnt1, varPnt2
        Else

            '' select entities using mode
            objSS.Select lngMode
        End If
        '' highlight the selected entities
        objSS.Highlight True

        '' pause for the user
        .GetString False, vbCr & "Enter to continue"

        '' unhighlight the entities
        objSS.Highlight False

    End With

Done:

    '' if the selectionset was created, delete it
    If Not objSS Is Nothing Then
        objSS.Delete
    End If
End Sub
```

Selection Set Filters

You use the optional FilterCodes and FilterValues parameters to filter the selection of entities based on some criteria. Common criteria include objects with a specific color or linetype, or objects on a particular layer.

The FilterCodes parameter is an array of integers that specifies the entity group codes to filter. Entity group codes are available for every entity property. See the AutoCAD DXF Reference for a complete listing of the available group codes. The FilterValues parameter is an array of variants that specifies the corresponding values for each entity group code present in the FilterCodes parameter. Because these two arrays have a one-to-one mapping, they must be of the same length.

The following example filters all the entities into a selection set based on a layer name entered by the user:

```
Public Sub TestSelectionSetFilter()
      Dim objSS As AcadSelectionSet
      Dim intCodes(0) As Integer
      Dim varCodeValues(0) As Variant
      Dim strName As String

On Error GoTo Done

    With ThisDrawing.Utility
        strName = .GetString(True, vbCr & "Layer name to filter: ")
        If "" = strName Then Exit Sub

        '' create a new selectionset
        Set objSS = ThisDrawing.SelectionSets.Add("TestSelectionSetFilter")
        '' set the code for layer
        intCodes(0) = 8

        '' set the value specified by user
        varCodeValues(0) = strName

        '' filter the objects
        objSS.Select acSelectionSetAll, , , intCodes, varCodeValues

        '' highlight the selected entities
        objSS.Highlight True

        '' pause for the user
        .Prompt vbCr & objSS.Count & " entities selected"
        .GetString False, vbLf & "Enter to continue "

        '' unhighlight the entities
        objSS.Highlight False
    End With

Done:

    '' if the selection was created, delete it
    If Not objSS Is Nothing Then
        objSS.Delete
    End If
End Sub
```

When multiple filter codes are specified, they're implicitly combined using a logical AND operator. You can explicitly control how codes contribute to the selection criteria by using *filter operators.* The filter operators are designated by a special group code of –4 and special string keywords for the values. You use filter operators in pairs surrounding a number of filter codes known as *operands.*

Table 12-4 lists the SelectionSet filter operators, their meanings, and the required number of operands.

Table 12-4. SelectionSet *Filter Operators*

Filter Operator	Start and End Value	Number of Operands	Description
"<AND"	… "AND>"	One or more	A logical AND of all the operands. If a criterion matches all of the operands, it will be included. This is the default for multiple criteria when no filter operators are specified.
"<OR"	… "OR>"	One or more	A logical OR of all the operands. If a criterion matches any of the operands, it will be included.

Continued

Table 12-4. *(Continued)*

Filter Operator	Start and End Value	Number of Operands	Description
"<XOR"	… "XOR>"	Exactly two	A logical `XOR` of two operands. If a criterion matches one but not the other, it will be included.
"<NOT"	… "NOT>"	Exactly one	A logical `NOT` of a single operand. If a criterion doesn't match the operand, it will be included.

The following example uses filter operators to limit the selection to lines, arcs, and circles that are *not* on a specified layer:

```
Public Sub TestSelectionSetOperator()
      Dim objSS As AcadSelectionSet
      Dim intCodes() As Integer
      Dim varCodeValues As Variant
      Dim strName As String

On Error GoTo Done
    With ThisDrawing.Utility
        strName = .GetString(True, vbCr & "Layer name to exclude: ")
        If "" = strName Then Exit Sub

        '' create a new selectionset
        Set objSS = ThisDrawing.SelectionSets.Add("TestSelectionSetOperator")

        '' using 9 filters
        ReDim intCodes(9):  ReDim varCodeValues(9)

        '' set codes and values - indented for clarity
        intCodes(0) = -4:  varCodeValues(0) = "<and"
        intCodes(1) = -4:    varCodeValues(1) = "<or"
        intCodes(2) = 0:       varCodeValues(2) = "line"
        intCodes(3) = 0:       varCodeValues(3) = "arc"
        intCodes(4) = 0:       varCodeValues(4) = "circle"
        intCodes(5) = -4:    varCodeValues(5) = "or>"
        intCodes(6) = -4:    varCodeValues(6) = "<not"
        intCodes(7) = 8:       varCodeValues(7) = strName
        intCodes(8) = -4:    varCodeValues(8) = "not>"
        intCodes(9) = -4:  varCodeValues(9) = "and>"

        '' filter the objects
        objSS.Select acSelectionSetAll, , , intCodes, varCodeValues

        '' highlight the selected entities
        objSS.Highlight True
        '' pause for the user
        .Prompt vbCr & objSS.Count & " entities selected"
        .GetString False, vbLf & "Enter to continue "

        '' unhighlight the entities
        objSS.Highlight False
    End With

Done:

    '' if the selection was created, delete it
    If Not objSS Is Nothing Then
        objSS.Delete
    End If
End Sub
```

The group code values can contain wild-card expressions if the code represents a string property. You can often use wild-card expressions to make filter criteria more succinct. For example, the preceding filter took nine group codes to define. By using wild cards, you can reduce it to just two:

```
ReDim intCodes(1):  ReDim varCodeValues(1)
intCodes(0) = 0:  varCodeValues(0) = "line,arc,circle"
intCodes(1) = 8:  varCodeValues(1) = "~" & strName
```

See the AutoCAD 2006 Customization Guide for more details on wild-card expressions.

The `SelectOnScreen` Method

You use this method to obtain a selection of entities interactively from the user (see Table 12-5 for parameter details). A Select Objects prompt is shown at the command line, where all the standard selection-set options such as `Window`, `Crossing`, and `Last` are available to the user for dynamic selection.

```
SelectionSetObject.SelectOnScreen [, FilterCodes, FilterValues]
```

Table 12-5. `SelectOnScreen` *Method Parameters*

Name	Data Type	Description
`FilterCodes`	Variant	An optional array of integer entity selection filter codes. This must be used in conjunction with, and have the same length as, `FilterValues`.
`FilterValues`	Variant	An optional array of variant entity selection filter values. This must be used in conjunction with, and have the same length as, `FilterCodes`.

Like the `Select` method, `SelectOnScreen` can filter the selected entities using criteria specified in the `FilterCodes` and `FilterValues` parameters. See the "Selection Set Filters" section for a full description of filtering.

The following example gets a selection of entities dynamically from the user:

```
Public Sub TestSelectOnScreen()
      Dim objSS As AcadSelectionSet

      On Error GoTo Done
    With ThisDrawing.Utility

        '' create a new selectionset
        Set objSS = ThisDrawing.SelectionSets.Add("TestSelectOnScreen")

        '' let user select entities interactively
        objSS.SelectOnScreen

        '' highlight the selected entities
        objSS.Highlight True

        '' pause for the user
        .Prompt vbCr & objSS.Count & " entities selected"
        .GetString False, vbLf & "Enter to continue "

        '' unhighlight the entities
        objSS.Highlight False

    End With

Done:

    '' if the selection was created, delete it
    If Not objSS Is Nothing Then
        objSS.Delete
    End If
End Sub
```

The SelectAtPoint Method

The SelectAtPoint method (see Table 12-6 for parameter details) selects a single entity that passes through a specified point. Unlike most of the other SelectXXX methods (except the acSelectionSetLast mode of Select), it selects only a single entity.

```
SelectionSet.SelectAtPoint Point [,FilterCodes, FilterValues]
```

Table 12-6. `SelectAtPoint` *Method Parameters*

Name	Data Type	Description
`Point`	Variant	A three-element array of doubles indicating the point through which the selected entity must pass.
`FilterCodes`	Variant	An optional array of integer entity selection filter codes. This must be used in conjunction with, and have the same length as, `FilterValues`.
`FilterValues`	Variant	An optional array of variant entity selection filter values. This must be used in conjunction with, and have the same length as, `FilterCodes`.

Like the other `Select` methods, `SelectAtPoint` can filter the selected entities using criteria specified in the `FilterCodes` and `FilterValues` parameters. (This was covered earlier in the "Selection Set Filters" section.)

The following example selects an entity passing through a point chosen by the user:

```
Public Sub TestSelectAtPoint()
      Dim varPick As Variant
      Dim objSS As AcadSelectionSet

      On Error GoTo Done

    With ThisDrawing.Utility

        '' create a new selectionset
        Set objSS = ThisDrawing.SelectionSets.Add("TestSelectAtPoint")

        '' get a point of selection from the user
        varPick = .GetPoint(, vbCr & "Select entities at a point: ")

        '' let user select entities interactively
        objSS.SelectAtPoint varPick

        '' highlight the selected entities
        objSS.Highlight True

        '' pause for the user
        .Prompt vbCr & objSS.Count & " entities selected"
        .GetString False, vbLf & "Enter to continue "

        '' unhighlight the entities
        objSS.Highlight False

    End With

Done:
    '' if the selection was created, delete it
    If Not objSS Is Nothing Then
        objSS.Delete
    End If
End Sub
```

The `SelectByPolygon` Method

This method (see Table 12-7 for parameter details) selects entities by using a boundary that may be either a closed polygon or an open figure called a *fence*. In the case of a closed polygon, you can further limit entities to only those lying completely within the polygon (`Window-Polygon`) or those that pass through the boundary, as well (`Crossing-Polygon`).

```
SelectionSetObject.SelectByPolygon Mode, Vertices _
[, FilterType, FilterData]
```

Table 12-7. `SelectByPolygon` *Method Parameters*

Name	Data Type	Description
`Mode`	Long	Determines the selection mode to be used. It must be one of the `AcSelect` constants described in Table 12-8.
`Vertices`	Variant	A required array of doubles specifying a list of 3-D vertex points in WCS coordinates. It's a simple array with a single dimension composed of alternating X, Y, and Z values (i.e., `p1x`, `p1y`, `p1z`, `p2x`, `p2y`, `p2z`, etc.).
`FilterType`	Variant	An optional array of integer entity selection filter codes. This must be used in conjunction with, and have the same length as, `FilterData`.
`FilterData`	Variant	An optional array of variant entity selection filter values. This must be used in conjunction with, and have the same length as, `FilterType`.

The `SelectByPolygon` method has several modes of operation specified by the `Mode` parameter. AutoCAD provides the `AcSelect` constants listed in Table 12-8 for use with this parameter.

Table 12-8. `SelectByPolygon` *Method Modes*

Constant	Value	Description
`acSelectionSetFence`	2	Entities crossing a multisegment fence specified by `Vertices`. At least two vertices must be specified.
`acSelectionSetWindowPolygon`	6	Entities completely contained within a polygon specified by the `Vertices` are selected. At least three vertices must be specified.
`acSelectionSetCrossingPolygon`	7	Entities contained within or crossing a polygon specified by the `Vertices` are selected. At least three vertices must be specified.

Like the other Select methods, SelectByPolygon can filter the selected entities using criteria specified in the FilterCodes and FilterValues parameters. See the “Selection Set Filters” section for a full description of filtering.

The following example selects entities based on a user-specified boundary and mode:

```
Public Sub TestSelectByPolygon()
      Dim objSS As AcadSelectionSet
      Dim strOpt As String
      Dim lngMode As Long
      Dim varPoints As Variant

      On Error GoTo Done
    With ThisDrawing.Utility

        '' create a new selectionset
        Set objSS = ThisDrawing.SelectionSets.Add("TestSelectByPolygon1")

        '' get the mode from the user
        .InitializeUserInput 1, "Fence Window Crossing"
        strOpt = .GetKeyword(vbCr & "Select by [Fence/Window/Crossing]: ")

        '' convert keyword into mode
        Select Case strOpt
             Case "Fence":  lngMode = acSelectionSetFence
             Case "Window":  lngMode = acSelectionSetWindowPolygon
             Case "Crossing":  lngMode = acSelectionSetCrossingPolygon
        End Select

        '' let user digitize points
        varPoints = InputPoints()

        '' select entities using mode and points specified
        objSS.SelectByPolygon lngMode, varPoints

        '' highlight the selected entities
        objSS.Highlight True

        '' pause for the user
        .Prompt vbCr & objSS.Count & " entities selected"
        .GetString False, vbLf & "Enter to continue "

        '' unhighlight the entities
        objSS.Highlight False

    End With

Done:
    '' if the selection was created, delete it
    If Not objSS Is Nothing Then
        objSS.Delete
    End If
End Sub
```

This example uses a helper function named `InputPoints` to obtain the boundary points. Here's the code for this function:

```
Function InputPoints() As Variant
      Dim varStartPoint As Variant
      Dim varNextPoint As Variant
      Dim varWCSPoint As Variant
      Dim lngLast As Long
      Dim dblPoints() As Double

      On Error Resume Next
    '' get first points from user
    With ThisDrawing.Utility
        .InitializeUserInput 1
        varStartPoint = .GetPoint(, vbLf & "Pick the start point: ")

        '' setup initial point
        ReDim dblPoints(2)
        dblPoints(0) = varStartPoint(0)
        dblPoints(1) = varStartPoint(1)
        dblPoints(2) = varStartPoint(2)
        varNextPoint = varStartPoint

        '' append vertexes in a loop
        Do
            '' translate picked point to UCS for basepoint below
            varWCSPoint = .TranslateCoordinates(varNextPoint, acWorld, _
                                                acUCS, True)

            '' get user point for new vertex, use last pick as basepoint
            varNextPoint = .GetPoint(varWCSPoint, vbCr & _
                                     "Pick another point <exit>: ")

            '' exit loop if no point picked
            If Err Then Exit Do

            '' get the upper bound
            lngLast = UBound(dblPoints)

            '' expand the array
            ReDim Preserve dblPoints(lngLast + 3)
            '' add the new point
            dblPoints(lngLast + 1) = varNextPoint(0)
            dblPoints(lngLast + 2) = varNextPoint(1)
            dblPoints(lngLast + 3) = varNextPoint(2)
        Loop
    End With

    '' return the points
    InputPoints = dblPoints
End Function
```

Figure 12-1 illustrates the results of the three selection options. The solid shapes represent AutoCAD drawing objects, and the dashed line represents the path of the selecting polygon.

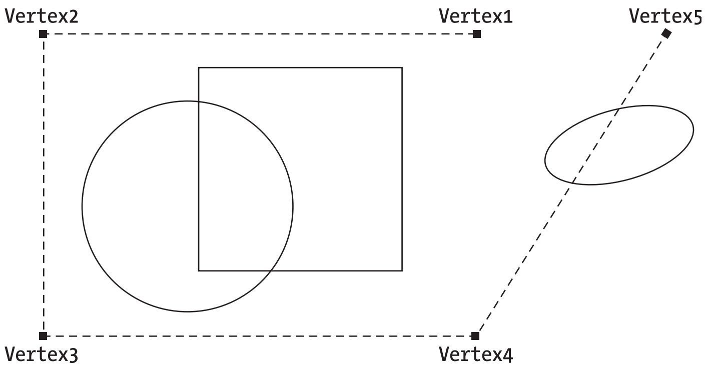


Figure 12-1. `TestSelectByPolygon` *example output*

The following parameters are used to define the area selected:

`acSelectionSetFence: only the ellipse is selected`

`acSelectionSetWindowPolygon: the circle and square are both selected`

`acSelectionSetCrossingPolygon: all objects are selected`

Adding and Removing Items

You can add individual entities to and remove individual entities from a selection set by using the `AddItem` and `RemoveItem` methods (see Table 12-9 for parameter details). These methods take an array of entities as a parameter. Both methods will raise exceptions if they're unable to add or remove all the entities in the array.

```
SelectionSetObject.AddItem(Entities)
```

```
SelectionSetObject.RemoveItem(Entities)
```

Table 12-9. *The* `AddItem` *and* `RemoveItem` *Method Parameter*

Name	Data Type	Description
`Entities`	Array of `AcadEntity` objects	The entities to add to or remove from the selection set

The following example adds and removes entities in a selection set until the user is satisfied with his or her choice:

```
Public Sub TestSelectAddRemoveClear()
      Dim objSS As AcadSelectionSet
      Dim objSStmp As AcadSelectionSet
      Dim strType As String
      Dim objEnts() As AcadEntity
      Dim intI As Integer

      On Error Resume Next

    With ThisDrawing.Utility
        '' create a new selectionset
        Set objSS = ThisDrawing.SelectionSets.Add("ssAddRemoveClear")
        If Err Then GoTo Done

        '' create a new temporary selection
        Set objSStmp = ThisDrawing.SelectionSets.Add("ssAddRemoveClearTmp")
        If Err Then GoTo Done

        '' loop until the user has finished
        Do
            '' clear any pending errors
            Err.Clear

            '' get input for type
            .InitializeUserInput 1, "Add Remove Clear Exit"
            strType = .GetKeyword(vbCr & "Select [Add/Remove/Clear/Exit]: ")

            '' branch based on input
            If "Exit" = strType Then

                '' exit if requested
                Exit Do

            ElseIf "Clear" = strType Then

                '' unhighlight the main selection
                objSS.Highlight False
                '' clear the main set
                objSS.Clear

            '' otherwise, we're adding/removing
            Else

                '' clear the temporary selection
                objSStmp.Clear

                objSStmp.SelectOnScreen
                '' highlight the temporary selection
                objSStmp.Highlight True

                '' convert temporary selection to array

                '' resize the entity array to the selection size
                ReDim objEnts(objSStmp.Count - 1)

                '' copy entities from the selection to entity array
                For intI = 0 To objSStmp.Count - 1
                    Set objEnts(intI) = objSStmp(intI)
                Next

                '' add/remove items from main selection using entity array
                If "Add" = strType Then
                    objSS.AddItems objEnts
                Else
                    objSS.RemoveItems objEnts
                End If

                '' unhighlight the temporary selection
                objSStmp.Highlight False

                '' highlight the main selection
                objSS.Highlight True
            End If
        Loop
    End With

Done:

    '' if the selections were created, delete them
    If Not objSS Is Nothing Then

        '' unhighlight the entities
        objSS.Highlight False
        '' delete the main selection
        objSS.Delete
    End If

    If Not objSStmp Is Nothing Then

        '' delete the temporary selection
        objSStmp.Delete
    End If
End Sub
```

The Clear, Delete, and Erase Methods

The remaining methods deal with clearing all entities from the selection, deleting the SelectionSet object itself, and erasing all the selected entities.

The Clear method simply clears the contents of a SelectionSet object. Entities that were in the selection are still present in the AutoCAD drawing—they're just no longer in the selection set.

```
SelectionSetObject.Clear
```

The Delete method deletes a SelectionSet object by removing it from the SelectionSets collection. Entities that were in the selection are still present in the AutoCAD drawing—the selection set is just deleted. Most of the examples in this chapter have used Delete after the program is complete.

```
SelectionSetObject.Delete
```

The Erase method erases the contents of a SelectionSet object from the AutoCAD drawing. Entities that were in the selection are immediately erased, but the SelectionSet object is still active and new entities may be added to it.

```
SelectionSetObject.Erase
```

The following example erases a selection of entities based on user input:

```
Public Sub TestSelectErase()
      Dim objSS As AcadSelectionSet

      On Error GoTo Done
    With ThisDrawing.Utility

        '' create a new selectionset
        Set objSS = ThisDrawing.SelectionSets.Add("TestSelectErase")

        '' let user select entities interactively
        objSS.SelectOnScreen

        '' highlight the selected entities
        objSS.Highlight True
        '' erase the selected entities
        objSS.Erase

        '' prove that the selection is empty (but still viable)
        .Prompt vbCr & objSS.Count & " entities selected"

    End With

Done:

    '' if the selection was created, delete it
    If Not objSS Is Nothing Then
        objSS.Delete
    End If
End Sub
```

The `PickFirstSelectionSet` Property

AutoCAD allows users to select entities in either a verb-noun or a noun-verb manner. A verb-noun selection is one in which a command is issued first, followed by a selection of entities to work with. A noun-verb selection is one in which entities are selected first and then a command that uses them is issued. One of the benefits of noun-verb selection is that by performing only one selection, you can run multiple independent commands in sequence on the same set of entities without having to explicitly use the Previous option.

The selection of entities built by the user when AutoCAD is idle at the command prompt is called the PickFirst selection set. This selection set is available via the read-only `PickfirstSelectionSet` property of each drawing object. A special `SelectionSet` named `PICKFIRST` is created by AutoCAD whenever the property is accessed. If the user does not preselect any entities, this selection set will be empty.

```
Set SelectionSetObject = DocumentObject.PickfirstSelectionSet
```

The `SelectionChanged` Event

By using the `SelectionChanged` event of each `Document` object, you can receive notification every time the user adds entities to or removes entities from the PickFirst selection set. This event is fired any time the PickFirst selection set is altered. Chapter 4 discusses events in more detail.

To demonstrate that the `SelectionChanged` event is triggered, the following example, which you should place in the `ThisDrawing` module, tells the user how many objects are in the PickFirst selection set and then changes the highlight of the PickFirst selection set after a half second.

```
Private Sub AcadDocument_SelectionChanged()
      Dim objSS As AcadSelectionSet
      Dim dblStart As Double

    '' get the pickfirst selection from drawing
    Set objSS = ThisDrawing.PickfirstSelectionSet
    '' highlight the selected entities
    objSS.Highlight True
    MsgBox "There are " & objSS.Count & " objects in selection set: " & objSS.Name
    '' delay for 1/2 second
    dblStart = Timer
    Do While Timer < dblStart + 0.5
    Loop

    '' unhighlight the selected entities
    objSS.Highlight False
End Sub
```

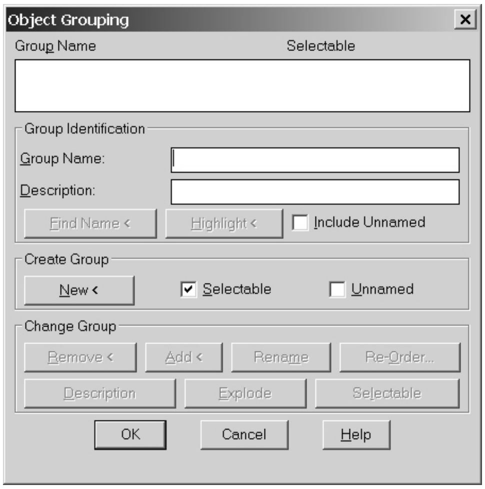
Groups

AutoCAD allows you to have a more permanent form of selection set called a *group*. Just like `SelectionSet` objects, `Group` objects associate an arbitrary set of entities into a collection that you can use all at once in many operations. You can associate entities with any number of groups, and you can have any number of entities in a single group.

By default, when an entity that belongs to a group is selected, all the other members of the group are automatically selected as well. The behavior is controlled by the `PICKSTYLE` system variable. See the AutoCAD User's Guide for details on other possible values for this variable.

Despite being collected into a group, entities that make up a group are still easily editable on an individual basis. This is in contrast to blocks in which, once entities are collected into a Block Definition and then inserted, the entities are quite difficult to edit individually. However, it's interesting to note that `Group` objects can have Xdata attached to them, and they have available nearly all the same default properties that other entities have, such as `Color`, `Linetype`, `Lineweight`, `PlotStyle`, `Visible`, and even `TrueColor`.

You normally create and edit groups through the Object Grouping dialog box (see Figure 12-2), which you can view by typing **GROUP** at the AutoCAD command prompt.



Adding a Group Object

Just like with selection sets, you use the Add method of the collection to create a new Group object as detailed in Table 12-10.

```
Set GroupObject = GroupsCollection.Add(Name)
```

Table 12-10. *The* Add *Method Parameter*

Name	Data Type	Description
Name	String	The name of the newly created group

If you try to add a Group object with a name that is already being used, this method will return a reference to the existing object.

The following example attempts to add a group using a name obtained from the user:

```
Public Sub TestAddGroup()
      Dim objGroup As AcadGroup
      Dim strName As String

      On Error Resume Next
    '' get a name from user
    strName = InputBox("Enter a new group name: ")
    If "" = strName Then Exit Sub

    Set objGroup = ThisDrawing.Groups.Item(strName)
    '' create it
    If Not objGroup Is Nothing Then
        MsgBox "Group already exists"
        Exit Sub
    End If

    Set objGroup = ThisDrawing.Groups.Add(strName)

    '' check if it was created
    If objGroup Is Nothing Then
        MsgBox "Unable to Add '" & strName & "'"
    Else
        MsgBox "Added group '" & objGroup.Name & "'"
    End If
End Sub
```

Accessing and Iterating Groups

The Groups collection is accessed via a Document object. In this code, ThisDrawing is used as the active document:

```
Dim objGroups As AcadGroups
Set objGroups = ThisDrawing.Groups
```

To set a reference to an existing `Group` object, use the `Item` method of the `Groups` collection:

```
Dim objGroup As AcadGroup

Set objGroup = objGroups.Item(2)
Set objGroup = objGroups.Item("My Group")
```

The parameter of this method is either a number representing the position of the desired `Group` object within the `Groups` collection or a string representing the name of the desired `Group` object. If you use an index number, it must be between `0` and `N-1`, in which `N` is the total number of groups in the collection.

Like other AutoCAD collections, `Item` is the default method for `Groups`. This means that the method name may be omitted, and the parameter passed straight to the `Groups` reference. Again, however, Microsoft guidelines recommend against using default methods and properties. You should always make a habit of invoking methods and accessing properties explicitly.

You may iterate through the `Groups` collection in the same way as you do the `SelectionSets` collection discussed at the beginning of this chapter.

```
Public Sub ListGroups()
      Dim objGroup As AcadGroup
      Dim strGroupList As String

    For Each objGroup In ThisDrawing.Groups
        strGroupList = strGroupList & vbCr & objGroup.Name
    Next
    MsgBox strGroupList, vbOKOnly, "List of Groups"
End Sub
```

Adding and Removing Items

You can add individual entities to and remove individual entities from a group by using the `AppendItem` and `RemoveItem` methods (see Table 12-11 for parameter details). These methods take an array of entities as a parameter. Both methods raise exceptions if they're unable to append or remove all the entities in the array. So if you try to append three items and one of them is already a member of the group, an error is raised and none of the items will be appended.

```
    GroupObject.AppendItem(Entities)

    GroupObject.RemoveItem(Entities)
```

Table 12-11. *The* `AppendItem` *and* `RemoveItem` *Method Parameter*

Name	Data Type	Description
`Entities`	Array of `AcadEntity` objects	The entities to append to or remove from the group

■Note The `RemoveItem` method doesn't remove the entities from your drawing. It simply dissociates them from the `Group` object.

The following example adds or removes entities in a selection set to a user-named group based on the user's choices:

```
Public Sub TestGroupAppendRemove()
      Dim objSS As AcadSelectionSet
      Dim objGroup As AcadGroup
      Dim objEnts() As AcadEntity
      Dim strName As String
      Dim strOpt As String
      Dim intI As Integer

      On Error Resume Next
    '' set pickstyle to NOT select groups
    ThisDrawing.SetVariable "Pickstyle", 2

    With ThisDrawing.Utility
        '' get group name from user
        strName = .GetString(True, vbCr & "Group name: ")
        If Err Or "" = strName Then GoTo Done

        '' get the existing group or add new one
        Set objGroup = ThisDrawing.Groups.Add(strName)

        '' pause for the user
        .Prompt vbCr & "Group contains: " & objGroup.Count & " entities" & _
                vbCrLf

        '' get input for mode
        .InitializeUserInput 1, "Append Remove"
        strOpt = .GetKeyword(vbCr & "Option [Append/Remove]: ")
        If Err Then GoTo Done

        '' create a new selectionset
        Set objSS = ThisDrawing.SelectionSets.Add("TestGroupAppendRemove")
        If Err Then GoTo Done

        '' get a selection set from user
        objSS.SelectOnScreen

        '' convert selection set to array
        '' resize the entity array to the selection size
        ReDim objEnts(objSS.Count - 1)

        '' copy entities from the selection to entity array
        For intI = 0 To objSS.Count - 1
            Set objEnts(intI) = objSS(intI)
        Next
        '' append or remove entities based on input
        If "Append" = strOpt Then
            objGroup.AppendItems objEnts
        Else
            objGroup.RemoveItems objEnts
        End If

        '' pause for the user
        .Prompt vbCr & "Group contains: " & objGroup.Count & " entities"

        '' unhighlight the entities
        objSS.Highlight False
    End With

Done:
    If Err Then MsgBox "Error occurred: " & Err.Description
    '' if the selection was created, delete it
    If Not objSS Is Nothing Then
        objSS.Delete
    End If
End Sub
```

The Delete Method

The Delete method deletes a Group object by removing it from the Groups collection. Entities that were in the group are still present in the AutoCAD drawing; the Group object is just deleted. This is somewhat like exploding a Block Insertion, except that when the Group object has been eliminated, so has its name in the collection stack.

```
GroupObject.Delete
```

The following example deletes a group based on user input:

```
Public Sub TestGroupDelete()
      Dim objGroup As AcadGroup
      Dim strName As String

      On Error Resume Next
    With ThisDrawing.Utility

        strName = .GetString(True, vbCr & "Group name: ")
        If Err Or "" = strName Then Exit Sub

        '' get the existing group
        Set objGroup = ThisDrawing.Groups.Item(strName)
        If Err Then
            .Prompt vbCr & "Group does not exist "
            Exit Sub
        End If
        '' delete the group
        objGroup.Delete
        If Err Then
            .Prompt vbCr & "Error deleting group "
            Exit Sub
        End If

        '' pause for the user
        .Prompt vbCr & "Group deleted"
    End With
End Sub
```

Summary

In this chapter you've seen how to build `SelectionSet` collections and `SelectionSet` objects, filter their content using specific criteria, allow the user to create dynamic selections, and handle PickFirst selections. You've also learned how to make `Group` objects for a more persistent type of entity association and manipulation capabilities.

CHAPTER 13

Blocks, Attributes, and External References

Blocks collect any number of AutoCAD entities into a single container object. Create a BlockReference object to insert the Block container object (also known as a block definition) into model space, paper space, or another block definition. Because the BlockReference object refers to the block definition's geometry, it is an efficient means for managing drawing data:

- You can add, move, copy, and so on, the entity collection as a whole.
- You can create a library of commonly used geometry ready for reuse.
- You can save memory by storing all block references in one block definition.
- You can make global changes by redefining the block definition at any time—all block references immediately reflect the changes.

Blocks differ from Group objects in several ways:

- Blocks are more distinctly named and constructed.
- You can also use ObjectDBX to import and export them between drawings because they contain entity construct data, whereas Group objects maintain only a list of entities that belong to them.
- AutoCAD lets you create Block objects and programmatically add them to your drawing, much as users do through the AutoCAD user interface.
- The number of entities in a block definition has no limit, and the number of block references has no limit.

When you define a block, you can use it as often as you need. You can create drawings faster because you don't have to create the individual objects in the block.

Attributes let you attach text to blocks. Like the blocks to which they are attached, attributes have both a definition and a reference. The Attribute object is the definition and is associated with a Block object. This *attribute definition* is the template for creating AttributeReference objects associated with new BlockReference objects.

While the geometry for each `BlockReference` is identical, the associated `Attribute` objects may differ. You can make the `Attribute` object visible, give it all the font characteristics of AutoCAD `Text` entities, or make it invisible if the text stores only data.

This chapter explains the following:

- How to create and manipulate `Block` objects
- How to insert a `Block` object into an AutoCAD drawing
- How to work with external reference files
- How to create `Attribute` objects
- How to insert `Block` objects with attributes

Blocks and Block References

The `Block` object represents a block definition, which contains a name and a set of entities. `Block` objects have two elements:

- A block definition, which is the abstract database structure that defines a `Block` object's entities
- A block reference (or block insertion), which is the actual insertion in a drawing

Changing a block definition also changes every block reference in the drawing. When you explode, modify, and re-create a block in a drawing, AutoCAD rebuilds the block definition in the drawing database. You can, however, directly modify the block definition database programmatically without having to do anything in the graphical interface.

Three kinds of blocks exist:

Simple block: The most common type of block definition, a simple block is a collection of objects that forms a new `Block` object. You can populate a simple block either by using new geometric entities, by copying existing drawing objects into it, or by inserting another AutoCAD drawing into it. You create references to simple blocks with the `InsertBlock` method, which returns a new `BlockReference` object. You can control the insertion point, scales, and rotation angle of this reference, or instance, in the drawing.

Externally referenced block: An *externally referenced* block is linked to another drawing file on disk. When someone makes changes to the drawing on disk, AutoCAD shows the changes the next time you generate the reference. Because the block definition is not in the current file, you can't modify its contents from the current drawing. You can get around this in the graphical environment using AutoCAD 2002 (and later) commands such as `REFEDIT`.

Layout block: Layout blocks contain the geometry associated with the `Layout` objects, which are represented as tabs at the bottom of the drawing window. You create layout blocks using the `Layouts` collection's `Add` method, which returns a new `Layout` object. You can use this object's `Block` property to access the layout block.

AutoCAD creates several layouts, which have the layout blocks listed in Table 13-1.

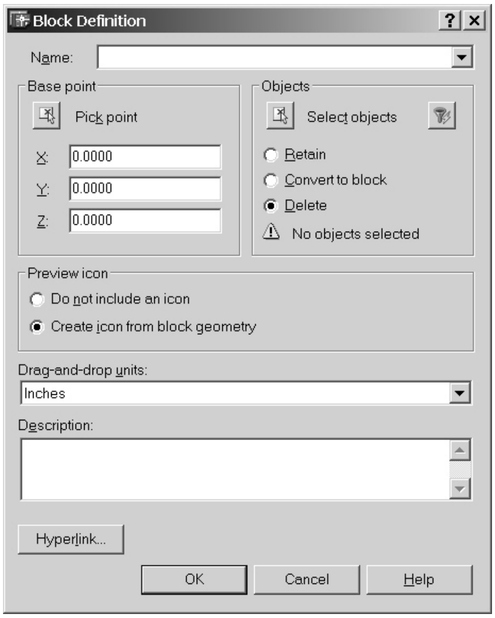
Table 13-1. *The Created Layouts*

Layout Name	Purpose
`*MODEL_SPACE`	The model space layout. This corresponds to the `ModelSpace` collection.
`*PAPER_SPACE`	The active paper space layout. This corresponds to the `PaperSpace` collection.
`*PAPER_SPACE0` through `*PAPER_SPACEn`	The first through last paper space layouts created.

■ **Note** When a paper space layout becomes active, it is renamed `*PAPER_SPACE`.

Use the `IsLayout` and `IsXRef` properties to identify the block definition type. If both of these properties are `False`, then the `Block` object is a simple block.

Users usually create simple block definitions using the Block Definition dialog box, as shown in Figure 13-1. Use the `BLOCK` command to open it.



Name:

Base point

X: 0.0000

Y: 0.0000

Z: 0.0000

Objects

Retain [✓ Delete] Convert to block

No objects selected

Preview icon

Do not include an icon [✓ Create icon from block geometry]

Drag-and-drop units: Inches

Description:

Figure 13-1. *Creating simple block definitions with the Block Definition dialog box*

All of the GUI's functionality is available in the `Block` object's properties and methods.

Accessing Block Objects

AutoCAD Document objects have a Blocks collection that contains all the drawing's Block definition objects. Access the Blocks collection with a Document object. The following code uses ThisDrawing as the active document:

```
Dim objBlocks As AcadBlocks
Set objBlocks = ThisDrawing.Blocks
MsgBox "There are " & objBlocks.Count & " Block objects"
```

To set a reference to an existing Block object, use the Blocks collection's Item method:

```
Dim objBlock As AcadBlock

Set objBlock = ThisDrawing.Blocks.Item(2)
Set objBlock = ThisDrawing.Blocks.Item("My Block")
```

As in other AutoCAD collections, Item is the default method for Blocks, so you can omit the method name and pass the parameter straight to the Blocks reference.

■ **Note** Even though the following method is simpler to type and read, it goes against Microsoft programming guidelines and is no longer supported in the .NET and VSA programming environments.

The following code performs the same task as the previous example but instead uses the default method to specify the Block object:

```
Dim objBlock As AcadBlock

Set objBlock = ThisDrawing.Blocks(2)
Set objBlock = ThisDrawing.Blocks("My Block")
```

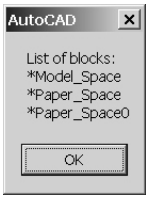
Iterating the Blocks Collection

As in all Visual Basic collections, the Blocks collection supports iteration using a For ... Each loop. The following example iterates through the Blocks collection, displaying each Block object's name in a message box:

```
Public Sub ListBlocks()
Dim objBlock As AcadBlock
Dim strBlockList As String

    strBlockList = "List of blocks: "

    For Each objBlock In ThisDrawing.Blocks
        strBlockList = strBlockList & vbCr & objBlock.Name
    Next

    MsgBox strBlockList
End Sub
```

Figure 13-2 shows this code's result for a newly created drawing.

Try adding a new layout to AutoCAD and then running this code again. It adds a new block. AutoCAD links these new Block objects to the different Layout objects and holds the geometry of the various drawing layouts.



```
List of blocks:
*Model_Space
*Paper_Space
*Paper_Space0
```

Figure 13-2. *Viewing the block code output*

Creating Blocks

You can create as many blocks as you need. You can create a simple block definition in two ways. The first uses the Blocks collection's Add method to create an empty Block object. You can then add entities to this new definition. The second uses the InsertBlock method of ModelSpace, PaperSpace, or a Block object to both create a block definition and insert a block reference from an external drawing file.

Using the Add Method

Use the Blocks collection's Add method to add a new Block object. The method takes a single string parameter—the new block's name. It returns a reference to the new Block object. If the parameter is not a valid block name, Add raises an exception. If you try to name a new block the same as an existing block, Add returns a reference to the existing Block object. This method has the following syntax:

```
Set BlockObject = BlocksCollection.Add(InsertionPoint, BlockName)
```

Table 13-2 explains this method's parameters.

When you create this object, it does not contain any entities. You can add them with the AddXXX methods discussed in the next section and in more detail in Chapters 8 and 9.

Table 13-2. *The Add Method's Parameters*

Name	Data Type	Description
InsertionPoint	Variant	A three-element array of doubles that specifies the block's insertion base point in WCS coordinates
BlockName	String	The new Block object's name

The following example adds a simple block definition to the Blocks collection and then adds a circle to it. The user can then insert the new block in the normal way.

```
Public Sub AddBlock()
Dim dblOrigin(2) As Double
Dim objBlock As AcadBlock
Dim strName As String

    '' get a name from user
    strName = InputBox("Enter a new block name: ")
    If "" = strName Then Exit Sub          ' exit if no old name
    '' set the origin point
    dblOrigin(0) = 0:  dblOrigin(1) = 0:   dblOrigin(2) = 0

    ''check if block already exists
On Error Resume Next
    Set objBlock = ThisDrawing.Blocks.Item(strName)
    If Not objBlock Is Nothing Then
        MsgBox "Block already exists"
        Exit Sub
    End If

    '' create the block
    Set objBlock = ThisDrawing.Blocks.Add(dblOrigin, strName)

    '' then add entities (circle)
    objBlock.AddCircle dblOrigin, 10

End Sub
```

AddXXX Methods

Use the AddXXX methods to add drawing entities to a Block object and therefore populate a new Block object. You've already seen a simple example of this in the AddBlock example. Chapters 8 and 9 cover the AddXXX methods fully.

CopyObject Method

Another way to populate a Block object with new entities is to use the Document object's CopyObject method to add duplicate entities. This method copies objects from one container to another. It also returns a variant array of the objects created during the copy. These new objects are exact duplicates, with the same relative positions, sizes, scales, and properties. This method has the following syntax:

```
varCopies = Owner.CopyObjects(Objects [, NewOwner] [, IdMap])
```

Table 13-3 explains this method's parameters.

Table 13-3. *The* CopyObject *Method's Parameters*

Name	Data Type	Description
Owner	Document, PaperSpace, ModelSpace, or Block objects	The current containing owner of the objects to copy.
Objects	Variant	An array of objects to copy. The objects must all belong to the Owner object.
NewOwner	Variant	Optional. Specifies the objects' new owner. If null, the objects are copied to the Owner object. This can also be another Document object.
IdMap	Variant	Optional. An array that holds IDPair objects.

The IdMap array is most useful for copying objects between databases or drawing files. The IDPair object describes how objects map from source to destination, including nonprimary but referenced objects. A full description of the IDPair object and its use is beyond the scope of this chapter. You will not need this optional functionality when you create blocks.

The following example creates and populates a block definition by copying the specified entities into the block:

```
Public Sub TestCopyObjects()
Dim objSS As AcadSelectionSet
Dim varBase As Variant
Dim objBlock As AcadBlock
Dim strName As String
Dim strErase As String
Dim varEnt As Variant
Dim objSourceEnts() As Object
Dim varDestEnts As Variant
Dim dblOrigin(2) As Double
Dim intI As Integer


    'choose a selection set name that you use only as temporary storage and
    'ensure that it does not currently exist
On Error Resume Next
    ThisDrawing.SelectionSets.Item("TempSSet").Delete
    Set objSS = ThisDrawing.SelectionSets.Add("TempSSet")
    objSS.SelectOnScreen

    '' get the other user input
    With ThisDrawing.Utility
        .InitializeUserInput 1
        strName = .GetString(True, vbCr & "Enter a block name: ")
        .InitializeUserInput 1
        varBase = .GetPoint(, vbCr & "Pick a base point: ")
        .InitializeUserInput 1, "Yes No"
        strErase = .GetKeyword(vbCr & "Erase originals [Yes/No]? ")
    End With

    '' set WCS origin
    dblOrigin(0) = 0: dblOrigin(1) = 0: dblOrigin(2) = 0

    '' create the block
    Set objBlock = ThisDrawing.Blocks.Add(dblOrigin, strName)

    '' put selected entities into an array for CopyObjects
    ReDim objSourceEnts(objSS.Count - 1)
    For intI = 0 To objSS.Count - 1
        Set objSourceEnts(intI) = objSS(intI)
    Next
    '' copy the entities into block
    varDestEnts = ThisDrawing.CopyObjects(objSourceEnts, objBlock)

    '' move copied entities so that base point becomes origin
    For Each varEnt In varDestEnts
        varEnt.Move varBase, dblOrigin
    Next

    '' if requested, erase the originals
    If strErase = "Yes" Then
        objSS.Erase
    End If

    '' we're done - prove that we did it
    ThisDrawing.SendCommand "._-insert" & vbCr & strName & vbCr

    '' clean up selection set
    objSS.Delete
End Sub
```

■**Note** If you try to copy the container object to itself, it reproduces itself an infinite number of times. You can't execute this method at the same time you iterate through a collection. An iteration opens the work space as read-only, while this method tries to perform a read-write operation. Complete any iteration before you call this method. The CopyObjects operation copies objects that the `Objects` parameter's primary objects own or reference.

Renaming a `Block` Object

To rename a block definition, assign a new string value to its `Name` property. Block references to the block definition are automatically adjusted. Take care to address other possible exceptions that may be thrown, such as an invalid name, existing name, and so on.

■**Note** Renaming an AutoCAD-defined layout block or any anonymous block, which is a block whose name begins with an asterisk (*), may crash AutoCAD.

The following example renames a `Block` definition based on user input:

```
Public Sub RenameBlock()
Dim strName As String
Dim objBlock As AcadBlock
```

```
On Error Resume Next                   ' handle exceptions inline
    strName = InputBox("Original Block name: ")
    If "" = strName Then Exit Sub          ' exit if no old name

    Set objBlock = ThisDrawing.Blocks.Item(strName)
    If objBlock Is Nothing Then            ' exit if not found
        MsgBox "Block '" & strName & "' not found"
        Exit Sub
    End If

    strName = InputBox("New Block name: ")
    If "" = strName Then Exit Sub          ' exit if no new name

    objBlock.Name = strName                ' try and change name
    If Err Then                            ' check if it worked
        MsgBox "Unable to rename block: " & vbCr & Err.Description
    Else
        MsgBox "Block renamed to '" & strName & "'"
    End If

End Sub
```

Deleting a `Block` Object

The `Delete` method removes a `Block` object from the `Blocks` collection. This method takes no parameters and returns nothing. It has the following syntax:

```
BlockObject.Delete
```

You can delete a block whenever you choose—unless the block can't be deleted. You can't delete a block when the following is true:

A `BlockReference` *object references it*: If a `BlockReference` object references a block definition in model space, in any paper space layout, or in any other block definition, you can't delete it. You have no definitive way to tell which entities reference a specific block, short of exhaustively searching each of these collections. If you delete all references or make them reference a different block, you can then delete the block definition. Keep in mind that deleting a `Block` object does not delete the block definition table entry. To completely remove a block definition, invoke a purge operation on the document.

It depends on an Xref: You create a block that depends on an Xref when you attach and activate an external reference file. You don't need to delete these blocks, however, because they're simply duplications of the blocks in the external drawing. AutoCAD doesn't save them with the current drawing.

■ **Note** AutoCAD creates certain blocks for its own purposes, such as the special layout blocks created for each layout. Deleting an AutoCAD-defined layout block or any other block whose name begins with an asterisk (*) can crash AutoCAD.

The following example deletes a block based on user input. Try it on some blocks created with the AddBlock code sample. In general, whenever you delete a collection member, such a block definition, execute a purge to clean up your drawing database. You'll see that you can't delete the block definition until you remove all the block references.

```
Public Sub DeleteBlock()
Dim strName As String
Dim objBlock As AcadBlock

On Error Resume Next                   ' handle exceptions inline
    strName = InputBox("Block name to delete: ")
    If "" = strName Then Exit Sub          ' exit if no old name

    Set objBlock = ThisDrawing.Blocks.Item(strName)
    If objBlock Is Nothing Then            ' exit if not found
        MsgBox "Block '" & strName & "' not found"
        Exit Sub
    End If

    objBlock.Delete                        ' try to delete it
    If Err Then                            ' check if it worked
        MsgBox "Unable to delete Block: " & vbCr & Err.Description
    Else
        MsgBox "Block '" & strName & "' deleted"
    End If
End Sub
```

Using the InsertBlock Method

The InsertBlock method of ModelSpace, PaperSpace, and Block objects serves two purposes. Its first purpose is to create a BlockReference object. Its second purpose is to create a block definition from a drawing file on disk. When you give it an external file name, InsertBlock imports the external drawing, creating a new Block object with the drawing's contents and inserting that file as a block reference.

■**Note** This operation imports any other blocks and drawing components (for example, layers and text styles) defined in the external drawing file.

Defining and Manipulating Blocks

After you define a block, you can use a number of methods to change the block's composition, select objects from the block, and break the block down into its parts.

Using the `Item` Method

Use the `Item` method to access the objects in the block definition object. This method has the following syntax:

```
Set objEntity = BlockObject.Item(Index)
```

This method has one parameter, `Index`; this is an Integer that gives the element's position in the `Block` collection object.

Because the `Item` method is the `Block` object's default method, you can omit it and use the following syntax. But it's best to avoid this in favor of using explicit references to all properties and methods (since Microsoft programming technologies are dropping this feature to gain performance, security, and consistency). For example, .NET and VSA no longer let you use default methods and properties.

```
Set objEntity = BlockObject(Index)
```

Using the `InsertBlock` Method

Use the `InsertBlock` method to add a `BlockReference` object to a drawing or to nest a block in another `Block` object. This can be either a block in the `Blocks` collection or an external drawing file. In either case, this method returns a `BlockReference` object. The `PaperSpace`, `ModelSpace`, and `Block` objects expose this method.

```
Set BlockReferenceObject = Object.InsertBlock(InsertionPoint, BlockName, _
    Xscale, Yscale, ZScale, RotationAngle)
```

Table 13-4 explains this method's parameters.

Table 13-4. *The* `InsertBlock` *Method's Parameters*

Name	Data Type	Description
`InsertionPoint`	Variant	A three-element array of doubles that specifies the 3-D WCS coordinates where the block will be inserted into `Object`.
`BlockName`	String	The name of a `Block` object in the `Blocks` collection or the path and file name of an AutoCAD drawing file to import.
`Xscale`	Double	A scaling factor for the block's X direction. It may not be 0. Negative numbers mirror the insertion on this axis.
`Yscale`	Double	A scaling factor for the block's Y direction. It may not be 0. Negative numbers mirror the insertion on this axis.
`Zscale`	Double	A scaling factor for the block's Z direction. It may not be 0. Negative numbers mirror the insertion on this axis.
`RotationAngle`	Double	The rotation angle relative to the WCS X-axis, expressed in radians.

The following example creates a block reference based on user input:

```
Public Sub TestInsertBlock()
Dim strName As String
Dim varInsertionPoint As Variant
Dim dblX As Double
Dim dblY As Double
Dim dblZ As Double
Dim dblRotation As Double

    '' get input from user
    With ThisDrawing.Utility
        .InitializeUserInput 1
        strName = .GetString(True, vbCr & "Block or file name: ")
        .InitializeUserInput 1
        varInsertionPoint = .GetPoint(, vbCr & "Pick the insert point: ")
        .InitializeUserInput 1 + 2
        dblX = .GetDistance(varInsertionPoint, vbCr & "X scale: ")
        .InitializeUserInput 1 + 2
        dblY = .GetDistance(varInsertionPoint, vbCr & "Y scale: ")
        .InitializeUserInput 1 + 2
        dblZ = .GetDistance(varInsertionPoint, vbCr & "Z scale: ")
        .InitializeUserInput 1
        dblRotation = .GetAngle(varInsertionPoint, vbCr & "Rotation angle: ")
    End With

    '' create the object
On Error Resume Next

    ThisDrawing.ModelSpace.InsertBlock varInsertionPoint, strName, dblX, _
                                       dblY, dblZ, dblRotation
    If Err Then MsgBox "Unable to insert this block."

End Sub
```

Try this example with the name of a block in the current drawing and with an external file name (including the .dwg file extension).

To give the user a friendlier way to input the path and file name, you can build a GUI with a UserForm and use a Windows CommonDialog control. This control is not available on the UserForm Toolbox, but you can add it by right-clicking the Toolbox, selecting Additional Controls, and checking the Microsoft Common Dialog Control, Version 6.0 option.

If you add a CommandButton control named cmdInsertBlock and a CommonDialog control named dlgOpenFile to a UserForm, as shown in Figure 13-3, you could use the following code to see the familiar Open dialog box, as shown in Figure 13-4, when you choose a drawing file.

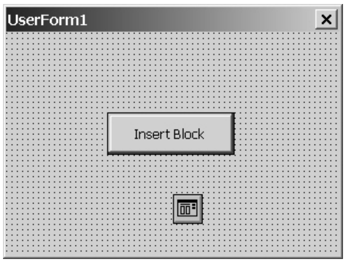


Figure 13-3. *Creating a* `UserForm` *with a* `CommandButton` *and a* `CommonDialog` *control*

```
Private Sub CommandButton1_Click()
    Dim objBlockRef As AcadBlockReference
    Dim varInsertionPoint As Variant
    Dim dblX As Double
    Dim dblY As Double
    Dim dblZ As Double
    Dim dblRotation As Double

    '' get input from user
    dlgOpenFile.Filter = "AutoCAD Blocks (*.DWG) | *.dwg"
    dlgOpenFile.InitDir = Application.Path
    dlgOpenFile.ShowOpen

    If dlgOpenFile.FileName = "" Then Exit Sub
    Me.Hide
    With ThisDrawing.Utility
        .InitializeUserInput 1
        varInsertionPoint = .GetPoint(, vbCr & "Pick the insert point: ")
        .InitializeUserInput 1 + 2
        dblX = .GetDistance(varInsertionPoint, vbCr & "X scale: ")
        .InitializeUserInput 1 + 2
        dblY = .GetDistance(varInsertionPoint, vbCr & "Y scale: ")
        .InitializeUserInput 1 + 2
        dblZ = .GetDistance(varInsertionPoint, vbCr & "Z scale: ")
        .InitializeUserInput 1
        dblRotation = .GetAngle(varInsertionPoint, vbCr & "Rotation angle: ")
    End With

    '' create the object
On Error Resume Next
    Set objBlockRef = ThisDrawing.ModelSpace.InsertBlock _
        (varInsertionPoint, dlgOpenFile.FileName, dblX, _
        dblY, dblZ, dblRotation)
    If Err Then
        MsgBox "Unable to insert this block"
        Exit Sub
    End If
    objBlockRef.Update
    Me.Show
End Sub
```

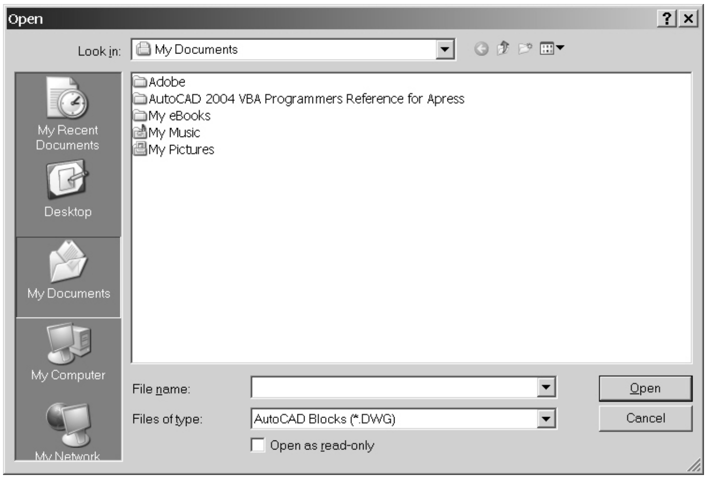


Figure 13-4. *Viewing the Open dialog box when choosing a drawing file*

To control the displayed file types, set the CommonDialog control's Filter property. You must hide the form before you can access the AutoCAD interface, because AutoCAD 2006 VBA UserForms are modal by default. The alternative is to set the ShowModal property of the UserForm to False.

Deleting a Block Reference

As with any other entity, use the Delete method to delete block references. This method removes only the BlockReference object, not the Block definition object. This method has the following syntax:

```
BlockReferenceObject.Delete
```

Using the Explode Method

The BlockReference object exposes the Explode method, which breaks down the block reference into its geometry. It returns an array of the block reference's objects. You can then select, move, copy, and so on, each entity individually. This method has the following syntax:

```
varArray = BlockReferenceObject.Explode
```

■Note This method actually creates a new copy of the block definition's entities and leaves the original unexploded block reference. If you don't want the reference, you must delete it yourself.

■Note In versions of AutoCAD before 2004, the `Explode` method explodes even nested `MText` objects into individual `Text` objects. The `Explode` method in AutoCAD 2004 and later leaves nested `MText` objects intact.

The following example explodes a block reference and moves the resulting entities to a new location:

```
Public Sub TestExplode()
    Dim objBRef As AcadBlockReference
    Dim varPick As Variant
    Dim varNew As Variant
    Dim varEnts As Variant
    Dim intI As Integer

On Error Resume Next
    '' get an entity and new point from user
    With ThisDrawing.Utility
        .GetEntity objBRef, varPick, vbCr & "Pick a block reference: "
        If Err Then Exit Sub
        varNew = .GetPoint(varPick, vbCr & "Pick a new location: ")
        If Err Then Exit Sub
    End With

    '' explode the blockref
    varEnts = objBRef.Explode
    If Err Then
        MsgBox "Error has occurred: " & Err.Description
        Exit Sub
    End If

    '' move resulting entities to new location
    For intI = 0 To UBound(varEnts)
        varEnts(intI).Move varPick, varNew
    Next
End Sub
```

Using the `WBlock` Method

This method's name means *write block*. This `Document` object method saves a `SelectionSet` object's contents to disk as a new drawing file. You can import this file into other drawings as a block definition using InsertBlock, as described earlier. The WBlock method returns nothing but has the following syntax:

```
DocumentObject.WBlock FileName, SelectionSet
```

Table 13-5 explains this method's parameters.

Table 13-5. *The* WBlock *Method's Parameters*

Name	Data Type	Description
FileName	String	The new drawing file's name. If you don't specify an extension, the method uses .dwg.
SelectionSet	SelectionSet object	A selection set containing the entities to write to the new file.

The method uses the selected entities' WCS origin as the base point for the drawing file. You may want to relocate the entities before you use WBlock so that the new drawing has a meaningful base point.

The following example asks the user for a selection set of entities, a file name, and a base point to use as the new file origin. It then moves the entities to the WCS origin using the selected base point. Next, it writes the entities to the file using the WBlock method. Finally, it moves the entities back to their original locations.

```
Public Sub TestWBlock()
Dim objSS As AcadSelectionSet
Dim varBase As Variant
Dim dblOrigin(2) As Double
Dim objEnt As AcadEntity
Dim strFilename As String

    'choose a selection set name that you use only as temporary storage and
    'ensure that it does not currently exist
On Error Resume Next
    ThisDrawing.SelectionSets("TempSSet").Delete
    Set objSS = ThisDrawing.SelectionSets.Add("TempSSet")
    objSS.SelectOnScreen

    With ThisDrawing.Utility
        .InitializeUserInput 1
        strFilename = .GetString(True, vbCr & "Enter a filename: ")
        .InitializeUserInput 1
        varBase = .GetPoint(, vbCr & "Pick a base point: ")
    End With

    '' WCS origin
    dblOrigin(0) = 0: dblOrigin(1) = 0: dblOrigin(2) = 0
    '' move selection to the origin
    For Each objEnt In objSS
        objEnt.Move varBase, dblOrigin
    Next

    '' wblock selection to file name
    ThisDrawing.Wblock strFilename, objSS

    '' move selection back
    For Each objEnt In objSS
        objEnt.Move dblOrigin, varBase
    Next

    '' clean up selection set
    objSS.Delete
End Sub
```

Using `MInsertBlock` Objects

An `MInsertBlock` object contains a planar array of block references in rows and columns. This section tells how to add and modify `MInsertBlock` objects.

The `AddMInsertBlock` method works much like `InsertBlock`, except that the resulting entity is an `MInsertBlock` object. Just like `InsertBlock`, this method lets you specify a new entity's insertion point, scale, and rotation angle. It has additional parameters for the arrayed rows and columns. The method has the following syntax:

```
Set MInsertBlockObject = Object.AddMInsertBlock(InsertionPoint, BlockName, _
    XScale, YScale, ZScale, RotationAngle, Rows, Columns, RowSpacing, _
    ColumnSpacing)
```

Table 13-6 explains this method's parameters.

Table 13-6. *The* `AddMInsertBlock` *Method's Parameters*

Name	Data Type	Description
`InsertionPoint`	Variant	A three-element array of doubles that specifies the 3-D WCS coordinates where the Xref will be inserted into the `Object`.
`BlockName`	String	The name of a `Block` object in the `Blocks` collection or the path and file name for an AutoCAD drawing file to import.
`Xscale`	Double	A scaling factor for the Xref's X direction. It may not be 0. Negative numbers mirror the insertion on this axis.
`Yscale`	Double	A scaling factor for the Xref's Y direction. It may not be 0. Negative numbers mirror the insertion on this axis.
`Zscale`	Double	A scaling factor for the Xref's Z direction. It may not be 0. Negative numbers mirror the insertion on this axis.
`RotationAngle`	Double	The rotation angle relative to the WCS X-axis, expressed in radians.

Continued

Table 13-6. *Continued*

Name	Data Type	Description
`Rows`	Long	A positive number that sets the number of rows.
`Columns`	Long	A positive number that sets the number of columns.
`RowSpacing`	Double	A nonzero number that specifies the spacing of rows in the array. A negative number creates rows in a negative X direction.
`ColumnSpacing`	Double	A nonzero number that specifies the spacing of columns in the array. A negative number creates columns in a negative Y direction.

The following example creates an `MInsertBlock` based on user input:

```
Public Sub TestAddMInsertBlock()
Dim strName As String
Dim varInsertionPoint As Variant
Dim dblX As Double
Dim dblY As Double
Dim dblZ As Double
Dim dR As Double
Dim lngNRows As Long
Dim lngNCols As Long
Dim dblSRows As Double
Dim dblSCols As Double

    '' get input from user
    With ThisDrawing.Utility
        .InitializeUserInput 1
        strName = .GetString(True, vbCr & "Block or file name: ")
        .InitializeUserInput 1
        varInsertionPoint = .GetPoint(, vbCr & "Pick the insert point: ")
        .InitializeUserInput 1 + 2
        dblX = .GetDistance(varInsertionPoint, vbCr & "X scale: ")
        .InitializeUserInput 1 + 2
        dblY = .GetDistance(varInsertionPoint, vbCr & "Y scale: ")
        .InitializeUserInput 1 + 2
        dblZ = .GetDistance(varInsertionPoint, vbCr & "Z scale: ")
        .InitializeUserInput 1
        dR = .GetAngle(varInsertionPoint, vbCr & "Rotation angle: ")
        .InitializeUserInput 1 + 2 + 4
        lngNRows = .GetInteger(vbCr & "Number of rows: ")
        .InitializeUserInput 1 + 2 + 4
        lngNCols = .GetInteger(vbCr & "Number of columns: ")
        .InitializeUserInput 1 + 2
        dblSRows = .GetDistance(varInsertionPoint, vbCr & "Row spacing: ")
        .InitializeUserInput 1 + 2
        dblSCols = .GetDistance(varInsertionPoint, vbCr & "Column spacing: ")
    End With
    '' create the object
    ThisDrawing.ModelSpace.AddMInsertBlock varInsertionPoint, strName, _
                 dblX, dblY, dblZ, dR, lngNRows, lngNCols, dblSRows, dblSCols
End Sub
```

While the `MInsertBlock` object is similar to a `BlockReference`, because it's an array, it has these extra properties: `Row`, `Column`, `RowSpacing`, and `ColumnSpacing`.

The following example adjusts an `MInsertBlock` object's array properties. Try it on the object you created in the previous example.

```
Public Sub TestEditobjMInsertBlock()
Dim objMInsert As AcadMInsertBlock
Dim varPick As Variant
Dim lngNRows As Long
Dim lngNCols As Long
Dim dblSRows As Double
Dim dblSCols As Double

On Error Resume Next
    '' get an entity and input from user
    With ThisDrawing.Utility
        .GetEntity objMInsert, varPick, vbCr & "Pick an MInsert: "
        If objMInsert Is Nothing Then
            MsgBox "You did not choose an MInsertBlock object"
            Exit Sub
        End If
        .InitializeUserInput 1 + 2 + 4
        lngNRows = .GetInteger(vbCr & "Number of rows: ")
        .InitializeUserInput 1 + 2 + 4
        lngNCols = .GetInteger(vbCr & "Number of columns: ")
        .InitializeUserInput 1 + 2
        dblSRows = .GetDistance(varPick, vbCr & "Row spacing: ")
        .InitializeUserInput 1 + 2
        dblSCols = .GetDistance(varPick, vbCr & "Column spacing: ")
    End With

    '' update the objMInsert
    With objMInsert
        .Rows = lngNRows
        .Columns = lngNCols
        .RowSpacing = dblSRows
        .ColumnSpacing = dblSCols
        .Update
    End With
End Sub
```

Introducing Dynamic Blocks

AutoCAD 2006 introduces *dynamic blocks*, which are more flexible and sophisticated than normal blocks. Since dynamic blocks may not be created programmatically, you will need to refer to the AutoCAD help files for instructions on how to create dynamic blocks.

■ **Note** You cannot create dynamic blocks through code using VBA. All of their properties, except for the `Value` property, are read-only.

You must follow a series of steps in order to access the properties of a dynamic block. The programming interface for dynamic block access is somewhat unconventional.

With the code example that follows, you can accomplish these steps:

1. Obtain a block reference object.
2. Cast the block reference to an `IAcadBlockReference2` object type.
3. Check the block reference to determine if it is dynamic.
4. If the block is dynamic, retrieve the dynamic properties of the block reference.
5. Iterate through each of the dynamic properties retrieving the `PropertyName` value and the associated `Value` property for display in the Immediate window.

Here's the code that accomplishes these steps:

```
Public Sub GetDynamicBlockProps()
Dim oBlockRef As IAcadBlockReference2
Dim Point As Variant
Dim oEntity As AcadEntity
Dim Props As Variant
Dim Index As Long

  On Error Resume Next

  ThisDrawing.Utility.GetEntity oEntity, Point, "Select block ..."

  If oEntity.ObjectName = "AcDbBlockReference" Then
    Set oBlockRef = oEntity

    Else
      'no block reference selected
      Exit Sub
  End If

  With oBlockRef
    If .IsDynamicBlock = True Then
      Props = .GetDynamicBlockProperties
      For Index = LBound(Props) To UBound(Props)
        Dim oProp As AcadDynamicBlockReferenceProperty

        Set oProp = Props(Index)

        'is the value an array for an insertion point
        If IsArray(oProp.Value) Then
          Dim SubIndex As Long

          For SubIndex = LBound(oProp.Value) To UBound(oProp.Value)
            Debug.Print oProp.PropertyName & ", " & oProp.Value(SubIndex)
          Next SubIndex

          Else
            Debug.Print oProp.PropertyName & ", " & oProp.Value
        End If
      Next Index
    End If
  End With
End Sub
```

■ **Note** When writing values to the `Value` property, no error will be raised if the value could not be set. Therefore, you will need to ensure the values you are attempting to write are the correct data types before attempting to write them to the `Value` property.

Using External References

External references, or Xrefs, are blocks that are not permanently loaded into the current drawing file. Instead, Xrefs refer to an external drawing file for their geometry (which is how they get their name).

External references share many properties and methods with simple blocks, and for many purposes you can treat them as simple blocks. But sometimes you might also need to use external references' special capabilities. This section explains the following Xref methods:

- Attaching and detaching
- Loading and unloading
- Binding

Attaching External References

The `AttachExternalReference` method works much like `InsertBlock`, except that the resulting entity is an external reference instead of a block reference. Just like `InsertBlock`, the `PaperSpace`, `ModelSpace`, and `Block` objects expose this method and let you specify the insertion point, scale, and rotation angle in the drawing.

```
Set ExternalReferenceObject = Object.AttachExternalReference(FileName, _
    BlockName, InsertionPoint, Xscale, Yscale, Zscale, _
    RotationAngle, Overlay)
```

Table 13-7 explains this method's parameters.

Table 13-7. *The* `AttachExternalReference` *Method's Parameters*

Name	Data Type	Description
`FileName`	String	The external AutoCAD drawing file's name. You must specify the `.dwg` extension. Optionally, you can specify a path to the file. If you don't, AutoCAD tries to find the file in the system search path.
`BlockName`	String	A name for the internal `Block` object that will point to the external drawing file.
`InsertionPoint`	Variant	A three-element array of doubles that specifies the 3-D WCS coordinates where the Xref will be inserted into the `Object`.
`Xscale`	Double	A nonzero number representing the scaling factor for the Xref's X direction. Negative numbers mirror the insertion on this axis.
`Yscale`	Double	A nonzero number representing the scaling factor for the Xref's Y direction. Negative numbers mirror the insertion on this axis.
`Zscale`	Double	A nonzero number representing the scaling factor for the Xref's Z direction. Negative numbers mirror the insertion on this axis.
`RotationAngle`	Double	The rotation angle relative to the WCS X-axis, expressed in radians.
`Overlay`	Boolean	Controls how the Xref is attached. If `True`, the Xref is brought in as an overlay. Overlay external references aren't visible if the current drawing is attached as an Xref to another drawing. In this way, overlay Xrefs can reduce the need to detach Xrefs before sharing drawings. If this parameter is `False`, the Xref is an attachment.

The following example creates an Xref based on user input:

```
Public Sub TestAttachExternalReference()
Dim strPath As String
Dim strName As String
Dim varInsertionPoint As Variant
Dim dblX As Double
Dim dblY As Double
Dim dblZ As Double
Dim dblRotation As Double
Dim strInput As String
Dim blnOver As Boolean
```

```
    '' get input from user
    With ThisDrawing.Utility
        .InitializeUserInput 1
        strPath = .GetString(True, vbCr & "External file name: ")
        .InitializeUserInput 1
        strName = .GetString(True, vbCr & "Block name to create: ")
        .InitializeUserInput 1
        varInsertionPoint = .GetPoint(, vbCr & "Pick the insert point: ")
        .InitializeUserInput 1 + 2
        dblX = .GetDistance(varInsertionPoint, vbCr & "X scale: ")
        .InitializeUserInput 1 + 2
        dblY = .GetDistance(varInsertionPoint, vbCr & "Y scale: ")
        .InitializeUserInput 1 + 2
        dblZ = .GetDistance(varInsertionPoint, vbCr & "Z scale: ")
        .InitializeUserInput 1
        dblRotation = .GetAngle(varInsertionPoint, vbCr & "Rotation angle: ")
        .InitializeUserInput 1, "Attach Overlay"
        strInput = .GetKeyword(vbCr & "Type [Attach/Overlay]: ")
        blnOver = IIf("Overlay" = strInput, True, False)
    End With

    '' create the object
    ThisDrawing.ModelSpace.AttachExternalReference strPath, strName, _
                   varInsertionPoint, dblX, dblY, dblZ, dblRotation, blnOver

End Sub
```

Detaching External References

You can detach an external reference from the current drawing using the `Block` object's `Detach` method. It has this syntax:

```
BlockObject.Detach
```

■Note When you detach an Xref's block definition, the method also removes all associated ExternalReference objects from the drawing. This includes linetypes, text styles, dim styles, nested block definitions, and layers.

Unloading External References

You can also unload external references without detaching them from the current drawing. Use the `Block` object's `Unload` method. It has this syntax:

```
BlockObject.Unload
```

Though not visible, unloaded Xrefs are still associated with the current drawing. To regenerate them, reload them.

Reloading External References

Use the `Block` object's `Reload` method to reload an external reference whenever you want, even if the Xref is already loaded. Reload an already-loaded Xref when you modify the underlying drawing and then want to update the in-memory copy in the current drawing. This method has the following syntax:

```
BlockObject.Reload
```

The following example demonstrates the `Detach`, `Reload`, and `Unload` methods using an external reference:

```
Public Sub TestExternalReference()
Dim strName As String
Dim strOpt As String
Dim objBlock As AcadBlock

On Error Resume Next    '' get input from user
    With ThisDrawing.Utility
        '' get the block name
        .InitializeUserInput 1
        strName = .GetString(True, vbCr & "External reference name: ")
        If Err Then Exit Sub

        '' get the block definition
        Set objBlock = ThisDrawing.Blocks.Item(strName)

        '' exit if not found
        If Err Then
            MsgBox "Unable to get block " & strName
            Exit Sub
        End If

        '' exit if not an xref
        If Not objBlock.IsXRef Then
            MsgBox "That is not an external reference"
            Exit Sub
        End If

        '' get the operation
        .InitializeUserInput 1, "Detach Reload Unload"
        strOpt = .GetKeyword(vbCr & "Option [Detach/Reload/Unload]: ")
        If Err Then Exit Sub
        '' perform operation requested
        If strOpt = "Detach" Then
            objBlock.Detach
        ElseIf strOpt = "Reload" Then
            objBlock.Reload
        Else
            objBlock.Unload
        End If
    End With
End Sub
```

Binding External References

Use the `Block` object's `Bind` method to convert external references to simple blocks. This operation builds an internal copy of the external drawing file in much the same way the `InsertBlock` method does using an external file name. Instead of referring to the external drawing database, `Bind` converts any former `ExternalReference` objects to simple block references. This method has the following syntax:

```
BlockObject.Bind(Merge)
```

This method has one parameter, `Merge`, which is a Boolean. When it's `True`, the method merges dependent symbol table entries in the external file with the current drawing's entries. When it's `False`, the method prefixes them to avoid collision with any other entry name in the current drawing. The prefix has the form `BlockName$X$EntryName`, where the following is true:

- `BlockName` is the block definition name for the current drawing's external reference.
- `X` is an automatically generated integer that makes the name unique in the current drawing.
- `EntryName` is the name of the symbol table entry in the externally referenced drawing file.

■**Note** If `Merge` is set to `True` and an entry is already present in the current drawing, the method maps the external entry to the current drawing entry. This is identical behavior to inserting block definitions that contain duplicate layers, linetypes, or text styles in the current drawing.

The following example binds the specified external reference using either style:

```
Public Sub TestBind()
Dim strName As String
Dim strOpt As String
Dim objBlock As AcadBlock

On Error Resume Next

    '' get input from user
    With ThisDrawing.Utility
```

```
        '' get the block name
        .InitializeUserInput 1
        strName = .GetString(True, vbCr & "External reference name: ")
        If Err Then Exit Sub

        '' get the block definition
        Set objBlock = ThisDrawing.Blocks.Item(strName)

        '' exit if not found
        If Err Then
            MsgBox "Unable to get block " & strName
            Exit Sub
        End If

        '' exit if not an xref
        If Not objBlock.IsXRef Then
            MsgBox "That is not an external reference"
            Exit Sub
        End If

        '' get the option
        .InitializeUserInput 1, "Prefix Merge"
        strOpt = .GetKeyword(vbCr & "Dependent entries [Prefix/Merge]: ")
        If Err Then Exit Sub

        '' perform the bind, using option entered
        objBlock.Bind ("Merge" = strOpt)
    End With
End Sub
```

Attributes

Attributes let you attach text to blocks. Each attribute has an identifier called the `Tag`, a prompt string called the `Prompt`, and a string field of user data called the `Value`. You can make this text either visible with all the font characteristics of standard AutoCAD text entities or invisible if the text is used to store data.

■ **Note** When a user inserts a block that contains attributes, AutoCAD asks the user to enter attribute values. See the AutoCAD User's Guide for an explanation of system variables that change this default behavior.

Creating Attributes

Like the blocks to which they are attached, attributes have both a definition and a reference. The `Attribute` object is the definition, and it's associated with a `Block` object. This attribute definition provides the template for creating `AttributeReference` objects associated with new `BlockReference` objects.

Using the `AddAttribute` Method

Use the `AddAttribute` method of a `PaperSpace`, `ModelSpace`, or `Block` object to create a new `Attribute` object. This attribute definition specifies the corresponding `AttributeReference`'s characteristics. The most important characteristics are passed as parameters to this method, including `Prompt` to display when inserting a block, `Tag` to identify the attribute, and `Value` (the default) to assign the attribute. This method has the following syntax:

```
Set AttributeObject = Object.AddAttribute(Height, Mode, Prompt, _
    InsertionPoint, Tag, Value)
```

Table 13-8 explains this method's attributes.

Table 13-8. *The* `AddAttribute` *Method's Parameters*

Name	Data Type	Description
`Height`	Double	The attribute definition's text size in the current drawing units.
`Mode`	Long	How the `Attribute` object behaves when a block containing this attribute definition is inserted. Table 13-9 lists this parameter's values.
`Prompt`	String	The method displays this prompt when a block containing this attribute definition is inserted. If null, the method displays the `Tag` parameter instead.
`InsertionPoint`	Variant	A three-element array of doubles that specifies the attribute's location in the block or drawing. It is a 3-D WCS coordinate.
`Tag`	String	An identifier used to access a specific attribute. It may contain any characters except spaces and exclamation points. AutoCAD converts it to uppercase.
`Value`	String	The attribute's default value.

AutoCAD provides several `AcAttributeMode` constants for the `Mode` parameter, which you can specify in any combination. Table 13-9 lists these values.

Table 13-9. The `Mode` *Parameter's Values*

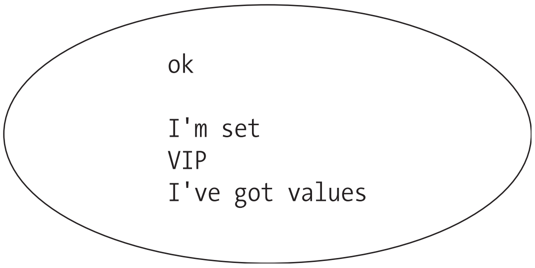
Constant	Value	Description
`acAttributeModeNormal`	0	The default mode, in which none of the other modes is applied.
`acAttributeModeInvisible`	1	When the block is inserted, the attribute's values aren't visible. The `ATTDISP` system variable overrides this mode setting.
`acAttributeModeConstant`	2	Each inserted block's attribute values are fixed. AutoCAD does not prompt for the attribute.
`acAttributeModeVerify`	4	When you insert the block, AutoCAD asks you to verify that the attribute value is correct.
`acAttributeModePreset`	8	AutoCAD inserts the block with its default attribute values. You can't edit these values.

The user can establish a preferred global mode for new attributes. The AFLAGS system variable stores this preferred global mode. You can access and set it using the GetVariable and SetVariable methods, which Appendix C discusses.

The following example creates a simple block containing an attribute of each type and then interactively inserts it:

```
Public Sub TestAddAttribute()
    Dim dblOrigin(2) As Double
    Dim dblEnt(2) As Double
    Dim dblHeight As Double
    Dim lngMode As Long
    Dim strTag As String
    Dim strPrompt As String
    Dim strValue As String
    Dim objBlock As AcadBlock
    Dim objEnt As AcadEntity

    '' create the block
    dblOrigin(0) = 0:  dblOrigin(1) = 0:  dblOrigin(2) = 0
    Set objBlock = ThisDrawing.Blocks.Add(dblOrigin, "Affirmations")

    '' delete existing entities (in case we've run before)
    For Each objEnt In objBlock
        objEnt.Delete
    Next

    '' create an ellipse in the block
    dblEnt(0) = 4:  dblEnt(1) = 0:  dblEnt(2) = 0
    objBlock.AddEllipse dblOrigin, dblEnt, 0.5

    '' set the height for all attributes
    dblHeight = 0.25
    dblEnt(0) = -1.5:  dblEnt(1) = 0:  dblEnt(2) = 0

    '' create a regular attribute
    lngMode = acAttributeModeNormal
    strTag = "Regular"
    strPrompt = "Enter a value"
    strValue = "I'm regular"
    dblEnt(1) = 1
    objBlock.AddAttribute dblHeight, lngMode, strPrompt, dblEnt, strTag, _
                          strValue
    '' create an invisible attribute
    lngMode = acAttributeModeInvisible
    strTag = "Invisible"
    strPrompt = "Enter a hidden value"
    strValue = "I'm invisible"
    dblEnt(1) = 0.5
    objBlock.AddAttribute dblHeight, lngMode, strPrompt, dblEnt, strTag, _
                          strValue

    '' create a constant attribute
    lngMode = acAttributeModeConstant
    strTag = "Constant"
    strPrompt = "Don't bother"
    strValue = "I'm set"
    dblEnt(1) = 0
    objBlock.AddAttribute dblHeight, lngMode, strPrompt, dblEnt, strTag, _
                          strValue

    '' create a verify attribute
    lngMode = acAttributeModeVerify
    strTag = "Verify"
    strPrompt = "Enter an important value"
    strValue = "I'm important"
    dblEnt(1) = -0.5
    objBlock.AddAttribute dblHeight, lngMode, strPrompt, dblEnt, strTag, _
                          strValue

    '' create a preset attribute
    lngMode = acAttributeModePreset
    strTag = "Preset"
    strPrompt = "No question"
    strValue = "I've got values"
    dblEnt(1) = -1
    objBlock.AddAttribute dblHeight, lngMode, strPrompt, dblEnt, strTag, _
                          strValue

    '' now insert block interactively using sendcommand
    ThisDrawing.SendCommand "._-insert" & vbCr & "Affirmations" & vbCr
End Sub
```

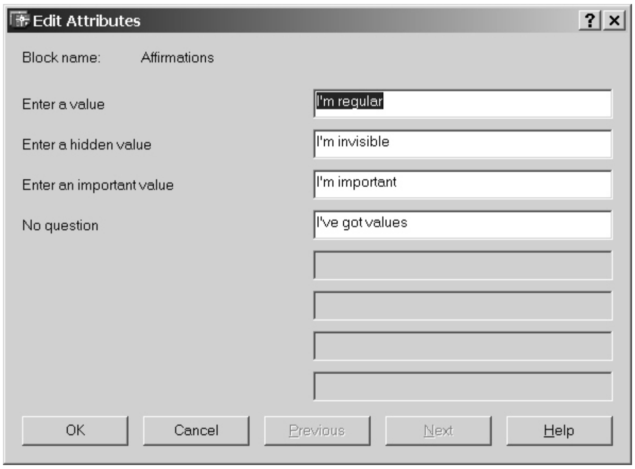
Neither constant nor preset attributes prompt for values. The `Verify` attribute prompt repeats. Only four attributes are visible in the drawing, as shown in Figure 13-5. The `Invisible` attribute is present but hidden.



```
ok

I'm set
VIP
I've got values
```

Figure 13-5. *Viewing the code output*

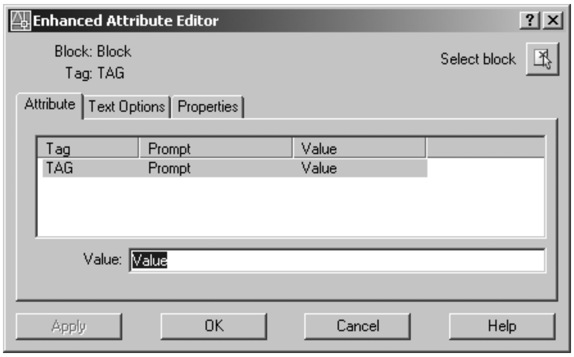
If, after you perform an insertion, you run the `DDATTE` command to review these values in the Edit Attributes dialog box, four of them are editable, as shown in Figure 13-6. The `Constant` attribute is not available, but you can change both the invisible values and the preset values.



Block name:	Affirmations
Enter a value	I'm regular
Enter a hidden value	I'm invisible
Enter an important value	I'm important
No question	I've got values

Figure 13-6. *Editing the four attributes*

In AutoCAD 2005 and 2006, after you perform the insertion, you can double-click the block to invoke the Enhanced Attribute Editor, as shown in Figure 13-7. You can then review the attribute values and make changes appropriately.



Tag	Prompt	Value
TAG	Prompt	Value

Figure 13-7. *Editing attributes in AutoCAD 2005/2006*

Manipulating Attribute References

Although the `BlockReference` object has identical geometry, each block reference's `AttributeReference` objects may have different values.

You can access these attribute references using the `GetAttributes` method.

Using the `GetAttributes` Method

Use the `GetAttributes` method of a `BlockReference` or `MInsertBlock` object to retrieve any associated `AttributeReference` objects. It returns an array, `varAttributeRefs`, that contains all the nonconstant attributes. This method has the following syntax:

```
varAttributeRefs = Object.GetAttributes()
```

The following example uses the Debug window to display information about each `AttributeRef` object associated with a block reference. Try it on an attributed block, such as the one created in the previous example.

```
Public Sub TestGetAttributes()
Dim varPick As Variant
Dim objEnt As AcadEntity
Dim objBRef As AcadBlockReference
Dim varAttribs As Variant
Dim strAttribs As String
Dim intI As Integer

On Error Resume Next
    With ThisDrawing.Utility

        '' get an entity from user
        .GetEntity objEnt, varPick, vbCr & "Pick a block with attributes: "
        If Err Then Exit Sub
        '' cast it to a blockref
        Set objBRef = objEnt

        '' exit if not a block
        If objBRef Is Nothing Then
            .Prompt vbCr & "That wasn't a block."
            Exit Sub
        End If

        '' exit if it has no attributes
        If Not objBRef.HasAttributes Then
            .Prompt vbCr & "That block doesn't have attributes."
            Exit Sub
        End If

        '' get the attributerefs
        varAttribs = objBRef.GetAttributes

        '' show some information about each
        strAttribs = "Block Name: " & objBRef.Name & vbCrLf
        For intI = LBound(varAttribs) To UBound(varAttribs)
           strAttribs = strAttribs & " Tag(" & intI & "): " & _
           varAttribs(intI).TagString & vbTab & "  Value(" & intI & "): " & _
           varAttribs(intI).TextString & vbCrLf
        Next

    End With
    MsgBox strAttribs

End Sub
```

Using the GetConstantAttributes Method

Use the GetConstantAttributes method of a BlockReference or MInsertBlock object to retrieve associated constant Attribute objects. This method has the following syntax:

```
varAttributes = Object.GetConstantAttributes()
```

It returns an array, varAttributes, that contains the constant attributes.

The following example uses the Debug window to display information about each constant Attribute object associated with a block reference. Try it on a block that has constant attributes, such as the one created earlier:

```
Public Sub TestGetConstantAttributes()
Dim varPick As Variant
Dim objEnt As AcadEntity
Dim objBRef As AcadBlockReference
Dim varAttribs As Variant
Dim strAttribs As String
Dim intI As Integer
```

```
    On Error Resume Next
    With ThisDrawing.Utility

        '' get an entity from user
        .GetEntity objEnt, varPick, vbCr & _
                   "Pick a block with constant attributes: "
        If Err Then Exit Sub

        '' cast it to a blockref
        Set objBRef = objEnt

        '' exit if not a block
        If objBRef Is Nothing Then
            .Prompt vbCr & "That wasn't a block."
            Exit Sub
        End If

        '' exit if it has no attributes
        If Not objBRef.HasAttributes Then
            .Prompt vbCr & "That block doesn't have attributes."
            Exit Sub
        End If

        '' get the constant attributes
        varAttribs = objBRef.GetConstantAttributes

        '' show some information about each
        strAttribs = "Block Name: " & objBRef.Name & vbCrLf
        For intI = LBound(varAttribs) To UBound(varAttribs)
            strAttribs = strAttribs & " Tag(" & intI & "): " & _
            varAttribs(intI).TagString & vbTab & "Value(" & intI & "): " & _
            varAttribs(intI).TextString
        Next

    End With
    MsgBox strAttribs
End Sub
```

Iterating Attribute Definitions

The `BlockReference` object exposes methods to retrieve its associated `AttributeReference` and `Attribute` objects. Unfortunately, similar functionality is not available for a `Block` object's attribute definitions. You can see this when you modify a block insertion in a drawing that contains multiple insertions of the same block. If it contains `Attributes` and you change the `ATDEF` location or properties and redefine the block, the other insertions do not update their display properties in unison with the changed insertion.

Use the following function to obtain a collection of attributes from a block. The collection uses the `TagString` as its key so it can quickly find specific attributes.

```
Function GetAttributes(objBlock As AcadBlock) As Collection
Dim objEnt As AcadEntity
Dim objAttribute As AcadAttribute
Dim coll As New Collection

    '' iterate the block
    For Each objEnt In objBlock

        '' if it's an attribute
        If objEnt.ObjectName = "AcDbAttributeDefinition" Then

            '' cast to an attribute
            Set objAttribute = objEnt

            '' add attribute to the collection
            coll.Add objAttribute, objAttribute.TagString
        End If
    Next

    '' return collection
    Set GetAttributes = coll
End Function
```

This example uses the `GetAttributes` function to display information about attributes created in the earlier example code:

```
Public Sub DemoGetAttributes()
Dim objAttribs As Collection
Dim objAttrib As AcadAttribute
Dim objBlock As AcadBlock
Dim strAttribs As String

    '' get the block
    Set objBlock = ThisDrawing.Blocks.Item("Affirmations")

    '' get the attributes
    Set objAttribs = GetAttributes(objBlock)

    '' show some information about each

    For Each objAttrib In objAttribs
        strAttribs = objAttrib.TagString & vbCrLf
        strAttribs = strAttribs & "Tag: " & objAttrib.TagString & vbCrLf & _
           "Prompt: " & objAttrib.PromptString & vbCrLf & " Value: " & _
            objAttrib.TextString & vbCrLf & "  Mode: " & _
            objAttrib.Mode
        MsgBox strAttribs
    Next
    '' find specific attribute by TagString
    Set objAttrib = objAttribs.Item("PRESET")

    '' prove that we have the right one
    strAttribs = "Tag: " & objAttrib.TagString & vbCrLf & "Prompt: " & _
       objAttrib.PromptString & vbCrLf & "Value: " & objAttrib.TextString & _
       vbCrLf & "Mode: " & objAttrib.Mode
    MsgBox strAttribs
End Sub
```

Inserting Blocks with Attributes

Depending on the setting of the ATTREQ system variable, AutoCAD can automatically handle retrieving and setting attribute values as user inputs. But if you use the InsertBlock method to insert a block reference, you are responsible for setting attribute values.

The following example inserts a block and sets several attribute values by locating the appropriate Tag identifiers. It uses the Affirmations block defined earlier.

```
Public Sub TestInsertAndSetAttributes()
Dim objBRef As AcadBlockReference
Dim varAttribRef As Variant
Dim varInsertionPoint As Variant
Dim dblX As Double
Dim dblY As Double
Dim dblZ As Double
Dim dblRotation As Double

    '' get block input from user
    With ThisDrawing.Utility
        .InitializeUserInput 1
        varInsertionPoint = .GetPoint(, vbCr & "Pick the insert point: ")
        .InitializeUserInput 1 + 2
        dblX = .GetDistance(varInsertionPoint, vbCr & "X scale: ")
        .InitializeUserInput 1 + 2
        dblY = .GetDistance(varInsertionPoint, vbCr & "Y scale: ")
        .InitializeUserInput 1 + 2
        dblZ = .GetDistance(varInsertionPoint, vbCr & "Z scale: ")
        .InitializeUserInput 1
        dblRotation = .GetAngle(varInsertionPoint, vbCr & "Rotation angle: ")
    End With

    '' insert the block
    Set objBRef = ThisDrawing.ModelSpace.InsertBlock(varInsertionPoint, _
                              "Affirmations", dblX, dblY, dblZ, dblRotation)

    '' iterate the attributerefs
    For Each varAttribRef In objBRef.GetAttributes
```

```
        '' change specific values based on Tag
        Select Case varAttribRef.TagString
        Case "Regular":
            varAttribRef.TextString = "I have new values"
        Case "Invisible":
            varAttribRef.TextString = "I'm still invisible"
        Case "Verify":
            varAttribRef.TextString = "No verification needed"
        Case "Preset":
            varAttribRef.TextString = "I can be changed"
        End Select
    Next

End Sub
```

Notice that AutoCAD lets attribute references be changed freely, even if the attribute definition specifies that a value is preset or should be verified. You can decide to handle attribute modes in any way you want.

Summary

This chapter explained how to create and manipulate blocks and external references. These objects give you many ways to efficiently store, manage, share, and use common graphic data. This chapter also explained how to associate text with graphics using attributes. Because every block's attribute reference values are unique, they provide a simple form of nongraphic data storage inside the AutoCAD drawing.

CHAPTER 14

Views and Viewports

A view is a named viewpoint configuration in model space. A View object contains spatial references to the point of view, the target of view, and other parameters. A user can name and save a view in his drawing and reuse it later. AutoCAD VBA provides methods and properties that let you create, set, and delete views.

A *viewport*, on the other hand, is a bounded area that displays some part of model or paper space. There are two viewport objects: the Viewport object for model space and the PViewport object for paper space, used for plotting.

This chapter covers these topics:

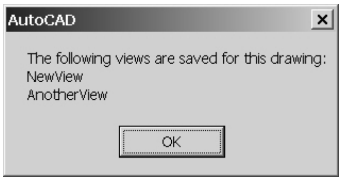
- Creating, setting, and deleting views
- Creating model-space viewports
- Creating paper-space viewports

Views

AutoCAD VBA uses a View object to represent a view. When you create a new drawing, no views are set up. All the views that you save in a drawing become members of the Views collection. The following code iterates through the Views collection and displays the names of saved views.

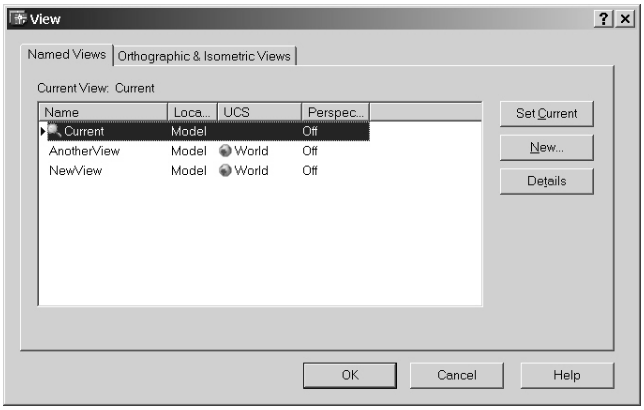
```
Public Sub DisplayViews()
    Dim objView As AcadView
    Dim strViewNames As String

    If ThisDrawing.Views.Count > 0 Then
        For Each objView In ThisDrawing.Views
            strViewNames = strViewNames & objView.Name & vbCrLf
        Next
        MsgBox "The following views are saved for this drawing:" & vbCrLf _
                & strViewNames
    Else
        MsgBox "There are no saved View objects in the Views collection."
    End If

End Sub
```

Figure 14-1 shows this code's output. This is equivalent to the views displayed in the View dialog box's Named Views tab (see Figure 14-2).



```
The following views are saved for this drawing:
NewView
AnotherView
```

Figure 14-1. *A list of views in a drawing*



Current View: Current

Name	Loca...	UCS	Perspec...
Current	Model		Off
AnotherView	Model	World	Off
NewView	Model	World	Off

Figure 14-2. *The View dialog box's Named Views tab*

The view named Current is not saved and is not part of the `Views` collection. Also, the `Views` collection is not imported into other drawings when you insert one drawing into another.

Creating a View

To create a new view in the `Views` collection, use the `Add` method. It has the following syntax:

```
Set ViewObject = ViewsCollection.Add(ViewName)
```

This method has one parameter, `ViewName`, a String. It contains the new `View` object's name.

After you add a `View` object to the `Views` collection, set the view's properties. The `Center`, `Target`, and `Direction` properties determine the drawing's viewing angle. Table 14-1 describes these properties.

Table 14-1. *View Properties*

Property	Description
`Center`	The view's center. A 2-D coordinate.
`Target`	The vector from the view's `Center` to its `Target` is the view's line of sight. A 3-D WCS coordinate.
`Direction`	The vector from which the drawing is viewed. A 3-D coordinate.

The `Height` and `Width` properties have no effect on the viewport. They only determine the view's visible portion in the viewport.

Use the following code to add a view to a drawing. It takes its settings from the active viewport, so users must set the view they want on-screen and then use this macro to create a named view for that setup.

```
Public Sub AddView()
    Dim objView As AcadView
    Dim objActViewPort As AcadViewport
    Dim strNewViewName As String
    Dim varCenterPoint As Variant
    Dim dblPoint(1) As Double

    strNewViewName = InputBox("Enter name for new view: ")
    If strNewViewName = "" Then Exit Sub

    On Error Resume Next
    Set objView = ThisDrawing.Views.Item(strNewViewName)
    If objView Is Nothing Then
        Set objView = ThisDrawing.Views.Add(strNewViewName)
        ThisDrawing.ActiveSpace = acModelSpace

        varCenterPoint = ThisDrawing.GetVariable("VIEWCTR")
        dblPoint(0) = varCenterPoint(0): dblPoint(1) = varCenterPoint(1)

        Set objActViewPort = ThisDrawing.ActiveViewport

        'Current view info. is stored in the new AcadView object
     With objView
        .Center = dblPoint
        .Direction = ThisDrawing.GetVariable("VIEWDIR")
        .Height = objActViewPort.Height
        .Target = objActViewPort.Target
        .Width = objActViewPort.Width
     End With
```

```
        MsgBox "A new view called " & objView.Name & _
               " has been added to the Views collection."
    Else
        MsgBox "This view already exists."
    End If

End Sub
```

The `VIEWCTR` and `VIEWDIR` system variables hold center and direction information. Appendix C lists all system variables.

Setting a View as Current

After you create a view, you'll probably want to use it. Use a `Viewport` object's `SetView` method to set one of the `View` objects as the current view. This method has the following syntax:

```
ViewportObject.SetView ViewObject
```

The parameter `ViewObject` is a reference to a `View` object, not the `View` object's name. The following code asks the user to enter a new view to use. If this view exists in the `Views` collection, the code sets it as the current view.

```
Public Sub SetView()
Dim objView As AcadView
Dim objActViewPort As AcadViewport
Dim strViewName As String

    ThisDrawing.ActiveSpace = acModelSpace
    Set objActViewPort = ThisDrawing.ActiveViewport
    'Redefine the current ViewPort with the View info
    strViewName = InputBox("Enter the view you require.")

    If strViewName = "" Then Exit Sub

    On Error Resume Next
    Set objView = ThisDrawing.Views.Item(strViewName)
    If Not objView Is Nothing Then
        objActViewPort.SetView objView
        ThisDrawing.ActiveViewport = objActViewPort
    Else
        MsgBox "View was not recognized."
    End If

End Sub
```

Deleting a View

When you finish using a view, you can delete it. Use the `Delete` method as follows:

```
ViewObject.Delete
```

The following example illustrates how to implement this method.

```
Public Sub DeleteView()
Dim objView As AcadView
Dim strViewName As String
Dim strExistingViewNames As String

    For Each objView In ThisDrawing.Views
        strExistingViewNames = strExistingViewNames & objView.Name & vbCrLf
    Next

    strViewName = InputBox("Existing Views: " & vbCrLf & _
                     strExistingViewNames & vbCrLf & _
                     "Enter the view you wish to delete from the list.")
    If strViewName = "" Then Exit Sub

    On Error Resume Next
    Set objView = ThisDrawing.Views.Item(strViewName)
    If Not objView Is Nothing Then
        objView.Delete
    Else
        MsgBox "View was not recognized."
    End If

End Sub
```

Viewports

Viewports are windowed views of your drawing in model and paper space. The `ActiveSpace` property determines which type of viewport is in use.

You construct paper-space and model-space viewports differently. The rest of this chapter explains these two approaches.

The Model-Space Viewport

Model-space viewports give you a number of windows in which to view and edit your model. The `MAXACTVP` system variable controls the maximum number of active viewports, and it's initially set to 64. Remember, though, that each new viewport adversely affects performance, so keep the number of active viewports to a minimum.

The following code shows how to create a model-space viewport. The paragraphs after the code explain this fairly involved procedure.

```
Public Sub CreateViewport()
Dim objViewPort As AcadViewport
Dim objCurrentViewport As AcadViewport
Dim varLowerLeft As Variant
Dim dblViewDirection(2) As Double
Dim strViewPortName As String
```

```
strViewPortName = InputBox("Enter a name for the new viewport.")
'user cancelled
If strViewPortName = "" Then Exit Sub
'check if viewport already exists
On Error Resume Next
Set objViewPort = ThisDrawing.Viewports.Item(strViewPortName)
If Not objViewPort Is Nothing Then
    MsgBox "Viewport already exists"
    Exit Sub
End If

'Create a new viewport
Set objViewPort = ThisDrawing.Viewports.Add(strViewPortName)

'Split the screen viewport into 4 windows
objViewPort.Split acViewport4

For Each objCurrentViewport In ThisDrawing.Viewports
    If objCurrentViewport.LowerLeftCorner(0) = 0 Then
        If objCurrentViewport.LowerLeftCorner(1) = 0 Then
            'this takes care of Top view
            dblViewDirection(0) = 0
            dblViewDirection(1) = 0
            dblViewDirection(2) = 1
            objCurrentViewport.Direction = dblViewDirection
        Else
            'this takes care of Front view
            dblViewDirection(0) = 0
            dblViewDirection(1) = -1
            dblViewDirection(2) = 0
            objCurrentViewport.Direction = dblViewDirection
        End If
    End If
    If objCurrentViewport.LowerLeftCorner(0) = 0.5 Then
        If objCurrentViewport.LowerLeftCorner(1) = 0 Then
            'this takes care of the Right view
            dblViewDirection(0) = 1
            dblViewDirection(1) = 0
            dblViewDirection(2) = 0
            objCurrentViewport.Direction = dblViewDirection
         Else
           'this takes care of the Isometric view
            dblViewDirection(0) = 1
            dblViewDirection(1) = -1
            dblViewDirection(2) = 1
            objCurrentViewport.Direction = dblViewDirection
        End If
    End If
    Next
    'make viewport active to see effects of changes
    ThisDrawing.ActiveViewport = objViewPort

End Sub
```

First, use the `Viewports` collection's `Add` method to create a new viewport. This method has the following syntax:

```
Set ViewportObject = ViewportsCollection.Add(ViewportName)
```

This method has one parameter, `ViewportName`, a String that contains the new `Viewport` object's name.

Here's how the example code implements this method:

```
'Create a new viewport
Set objViewPort = ThisDrawing.Viewports.Add(strViewPortName)
```

Next, the code uses the `Split` method to create the windows in the viewport.

```
ViewportObject.Split NumberOfWindows
```

The `NumberOfWindows` parameter must be one of the AutoCAD `AcViewportSplitType` constants in Table 14-2.

Table 14-2. `AcViewportSplitType` *Constants*

Constant	Value	Description
`acViewport2Horizontal`	0	Splits the viewport horizontally into two equal sections.
`acViewport2Vertical`	1	Splits the viewport vertically into two equal sections.
`acViewport3Left`	2	Splits the viewport into two vertical halves. The left half is split horizontally into two equal sections.
`acViewport3Right`	3	Splits the viewport into two vertical halves. The right half is split horizontally into two equal sections.
`acViewport3Horizontal`	4	Splits the viewport horizontally into three equal sections.
`acViewport3Vertical`	5	Splits the viewport vertically into three equal sections.
`acViewport3Above`	6	Splits the viewport into two horizontal halves. The top half is a single viewport. The bottom half is split horizontally into two equal sections.
`acViewport3Below`	7	Splits the viewport into two horizontal halves. The bottom half is a single viewport. The top half is split horizontally into two equal sections.
`acViewport4`	8	Splits the viewport horizontally and vertically into four equal sections.

The following example shows how the code sample implements this method. For the effects of this method to become apparent, reset the viewport as the active viewport. You can reset the active viewport at the very end of the code, when it's time to view the changes.

```
'Split the screen viewport into 4 windows
objViewPort.Split acViewport4
```

■Note Although the code calls the `Split` method, the changes don't appear until the `ActiveViewport` property makes the viewport active. You have to do this even if the viewport was active before the call to `Split`. A viewport does not have to be active for this method to work, but you must reset it to active, or you won't see the method's results.

You can identify each viewport using either the `LowerLeftCorner` or the `UpperRightCorner` property, each of which holds a two-element array of doubles. Figure 14-3 applies to the code sample using `acViewport4`.

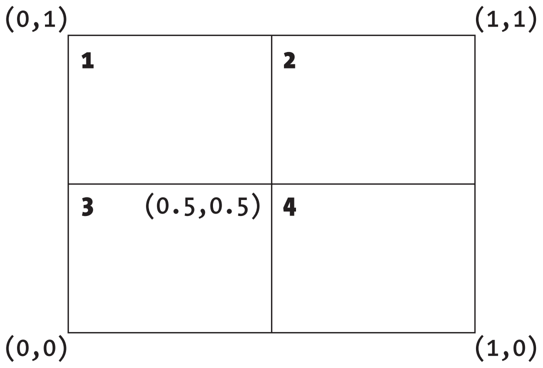


Figure 14-3. *Viewports*

Table 14-3 indicates the coordinates used to create the viewports as shown in Figure 14-3.

Table 14-3. *Viewport Coordinates*

Viewport	`LowerLeftCorner` Property	`UpperRightCorner` Property
1	(0, 0.5)	(0.5, 1)
2	(0.5, 0.5)	(1, 1)
3	(0, 0)	(0.5, 0.5)
4	(0.5, 0)	(1, 0.5)

The following code shows how this section's example code uses the `LowerLeftCorner` property to identify the viewports.

```
For Each objCurrentViewport In ThisDrawing.Viewports
  If objCurrentViewport.LowerLeftCorner(0) = 0 Then
    If objCurrentViewport.LowerLeftCorner(1) = 0 Then
      'this takes care of Top view
       dblViewDirection(0) = 0
       dblViewDirection(1) = 0
       dblViewDirection(2) = 1
    Else
       'this takes care of Front view
       dblViewDirection(0) = 0
       dblViewDirection(1) = -1
       dblViewDirection(2) = 0
       objCurrentViewport.Direction = dblViewDirection
    End If
  End If
  If objCurrentViewport.LowerLeftCorner(0) = 0.5 Then
    If objCurrentViewport.LowerLeftCorner(1) = 0 Then
       'this takes care of the Right view
       dblViewDirection(0) = 1
       dblViewDirection(1) = 0
       dblViewDirection(2) = 0
       objCurrentViewport.Direction = dblViewDirection
    Else
       'this takes care of the Isometric view
        dblViewDirection(0) = 1
        dblViewDirection(1) = -1
        dblViewDirection(2) = 1
       objCurrentViewport.Direction = dblViewDirection
    End If
  End If
Next
```

Set the `Direction` property for each viewport. This property has the following syntax:

```
ViewportObject.Direction = ViewDirection
```

`ViewDirection` is a three-element array of doubles that specifies the direction from which the drawing is viewed for that viewport.

Users typically have a Top, Front, Right, and maybe an isometric view of their drawing. Each of these views has a specific set of points relative to an imaginary target point at (0, 0, 0), which defines the direction of view. Table 14-4 lists the standard views and their `Direction` properties.

Table 14-4. *Views and Their Direction Properties*

View	Direction
Top	(0, 0, 1)
Bottom	(0, 0, –1)
Front	(0, –1, 0)
Back	(0, 1, 0)
Left	(–1, 0, 0)
Right	(1, 0, 0)
Northeast	(1, 1, 1)
Northwest	(–1, 1, 1)
Southeast	(1, –1, 1)
Southwest	(–1, –1, 1)

In this section's example, the following code sets the Top view's `Direction` property:

```
'this takes care of the Top view
dblViewDirection(0) = 0: dblViewDirection(1) = 0: dblViewDirection(2) = 1
objCurrentViewport.Direction = dblViewDirection
```

When you finish setting up the viewports, reset the viewport to be the active viewport to see the effects of your changes. Here's how to set the active viewport in code using the `ActiveViewport` property:

```
ThisDrawing.ActiveViewport = ViewportObject
```

At the end of the macro, use the following code to see the code's effects:

```
ThisDrawing.ActiveViewport = objViewPort
```

Figure 14-4 shows the result of running this macro on a drawing.

That may have seemed like a lot of work, but it's really not as difficult as it looks. Experimentation is your greatest ally when dealing with viewports. Remember to use the `LowerLeftCorner` and `UpperRightCorner` properties to determine which window you are dealing with.

You must activate a viewport before you can change it. To tell which viewport is active, look for the viewport with the bold border (as in Figure 14-6). Also, only the active viewport contains the crosshair cursor. When you finish making changes, reset the viewport to see them.

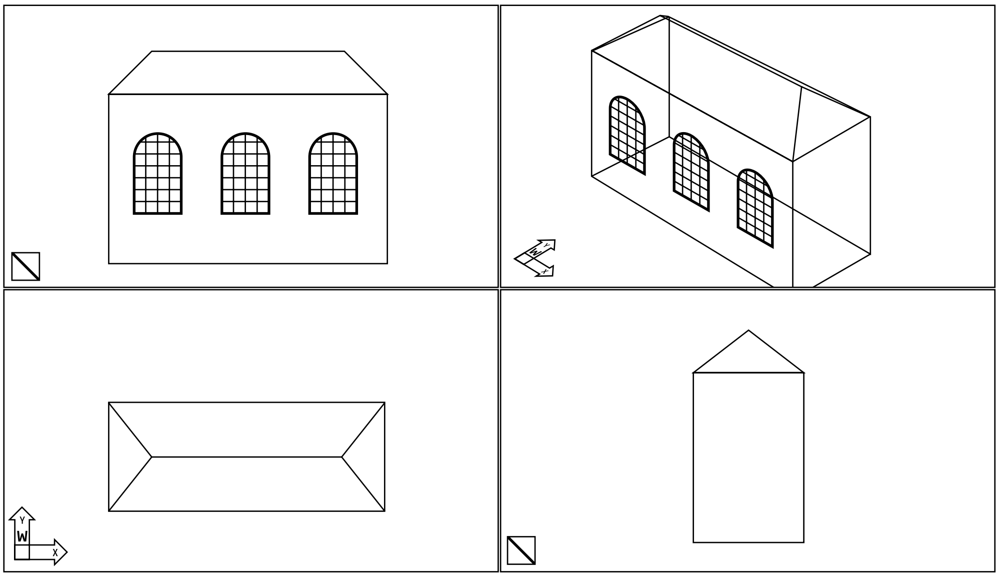


Figure 14-4. *Four-viewport view*

The Paper-Space Viewport

Paper-space viewports, represented by `PViewport` objects, are created for plotting purposes using the `AddPViewport` method as follows:

```
Set PViewportObject =  PaperSpaceCollection.AddPViewport(CenterPoint, _
                                             Width, Height)
```

Table 14-5 explains this method's parameters.

Table 14-5. `AddPViewport` *Method Parameters*

Name	Data Type	Description
`CenterPoint`	Variant	A three-element array of doubles that specifies the viewport's center coordinates
`Width`	Double	A positive number that specifies the viewport's width
`Height`	Double	A positive number that specifies the viewport's height

There is currently no programmatic means to quickly create, select, or manipulate polygonal `PViewports`, since they are made up of `LwPolyline` entities with additional information to relate the `LwPolyline` to the `Pviewport`.

The following example shows how to set up four different viewports for paper space. It's quite a long section of code, so the paragraphs that follow split it up into more manageable chunks and explain each step.

```
Public Sub CreatePViewports()
Dim objTopVPort As AcadPViewport
Dim objFrontVPort As AcadPViewport
Dim objRightVPort As AcadPViewport
Dim objIsoMetricVPort As AcadPViewport
Dim objLayout As AcadLayout
Dim objAcadObject As AcadObject
Dim dblPoint(2) As Double
Dim dblViewDirection(2) As Double
Dim dblOrigin(1) As Double
Dim dblHeight As Double
Dim dblWidth As Double
Dim varMarginLL As Variant
Dim varMarginUR As Variant

    ThisDrawing.ActiveSpace = acPaperSpace

    Set objLayout = ThisDrawing.ActiveLayout
    dblOrigin(0) = 0: dblOrigin(1) = 0
    objLayout.PlotOrigin = dblOrigin
    If objLayout.PlotRotation = ac0degrees Or objLayout.PlotRotation = _
                                          ac180degrees Then
        objLayout.GetPaperSize dblWidth, dblHeight
    Else
        objLayout.GetPaperSize dblHeight, dblWidth
    End If

    objLayout.GetPaperMargins varMarginLL, varMarginUR

    dblWidth = dblWidth - (varMarginUR(0) + varMarginLL(0))
    dblHeight = dblHeight - (varMarginUR(1) + varMarginLL(1))
    dblWidth = dblWidth / 2#
    dblHeight = dblHeight / 2#

    'Clear the layout of old PViewports
    For Each objAcadObject In ThisDrawing.PaperSpace
        If TypeName(objAcadObject) = "IAcadPViewport" Then
        objAcadObject.Delete
        End If
    Next

    'create Top Viewport
    dblPoint(0) = dblWidth - dblWidth * 0.5 '25
    dblPoint(1) = dblHeight - dblHeight * 0.5 '75
    dblPoint(2) = 0#

    Set objTopVPort = ThisDrawing.PaperSpace.AddPViewport(dblPoint, _
                                                    dblWidth, dblHeight)

    'need to set view direction
    dblViewDirection(0) = 0
    dblViewDirection(1) = 0
    dblViewDirection(2) = 1
    objTopVPort.Direction = dblViewDirection

    objTopVPort.Display acOn
    ThisDrawing.MSpace = True

    ThisDrawing.ActivePViewport = objTopVPort
    ThisDrawing.Application.ZoomExtents
    ThisDrawing.Application.ZoomScaled 0.5, acZoomScaledRelativePSpace

    'create Front Viewport
    dblPoint(0) = dblWidth - dblWidth * 0.5
    dblPoint(1) = dblHeight + dblHeight * 0.5
    dblPoint(2) = 0
```

```
Set objFrontVPort = ThisDrawing.PaperSpace.AddPViewport(dblPoint, _
                                                    dblWidth, dblHeight)

'need to set view direction
dblViewDirection(0) = 0
dblViewDirection(1) = -1
dblViewDirection(2) = 0
objFrontVPort.Direction = dblViewDirection

objFrontVPort.Display acOn
ThisDrawing.MSpace = True

ThisDrawing.ActivePViewport = objFrontVPort
ThisDrawing.Application.ZoomExtents
ThisDrawing.Application.ZoomScaled 0.5, acZoomScaledRelativePSpace

'create Right Viewport
dblPoint(0) = dblWidth + dblWidth * 0.5
dblPoint(1) = dblHeight - dblHeight * 0.5
dblPoint(2) = 0

Set objRightVPort = ThisDrawing.PaperSpace.AddPViewport(dblPoint, _
                                                    dblWidth, dblHeight)

'need to set view direction
dblViewDirection(0) = 1
dblViewDirection(1) = 0
dblViewDirection(2) = 0
objRightVPort.Direction = dblViewDirection

objRightVPort.Display acOn
ThisDrawing.MSpace = True

ThisDrawing.ActivePViewport = objRightVPort
ThisDrawing.Application.ZoomExtents
ThisDrawing.Application.ZoomScaled 0.5, acZoomScaledRelativePSpace

'create Isometric Viewport
dblPoint(0) = dblWidth + dblWidth * 0.5
dblPoint(1) = dblHeight + dblHeight * 0.5
dblPoint(2) = 0

Set objIsoMetricVPort = ThisDrawing.PaperSpace.AddPViewport(dblPoint, _
                                                    dblWidth, dblHeight)
```

```
    'need to set view direction
    dblViewDirection(0) = 1
    dblViewDirection(1) = -1
    dblViewDirection(2) = 1
    objIsoMetricVPort.Direction = dblViewDirection

    objIsoMetricVPort.Display acOn
    ThisDrawing.MSpace = True

    ThisDrawing.ActivePViewport = objIsoMetricVPort
    ThisDrawing.Application.ZoomExtents
    ThisDrawing.Application.ZoomScaled 0.5, acZoomScaledRelativePSpace

    'make paper space active again and we're almost done
    ThisDrawing.ActiveSpace = acPaperSpace
    ThisDrawing.Application.ZoomExtents

    'regen in all viewports
    ThisDrawing.Regen acAllViewports
End Sub
```

First, this code sets the active space to be paper space and uses the active `Layout` object's `GetPaperSize` method to find the paper's size. Because the viewports need to stay inside the plottable area, the code finds out the margins' size. It uses the `GetPaperMargins` method, which returns two two-element arrays that hold the offset of the lower-left and upper-right plot area from the lower-left and upper-right paper corners.

```
ThisDrawing.ActiveSpace = acPaperSpace

Set objLayout = ThisDrawing.ActiveLayout
dblOrigin(0) = 0: dblOrigin(1) = 0

objLayout.PlotOrigin = dblOrigin
If objLayout.PlotRotation = ac0degrees Or objLayout.PlotRotation = _
                                          ac180degrees Then
   objLayout.GetPaperSize dblWidth, dblHeight
Else
   objLayout.GetPaperSize dblHeight, dblWidth
End If

objLayout.GetPaperMargins varMarginLL, varMarginUR

dblWidth = dblWidth - (varMarginUR(0) + varMarginLL(0))
dblHeight = dblHeight - (varMarginUR(1) + varMarginLL(1))
dblWidth = dblWidth / 2#
dblHeight = dblHeight / 2#
```

Figure 14-5 shows a template of a viewport labeled with the different parameters to create.

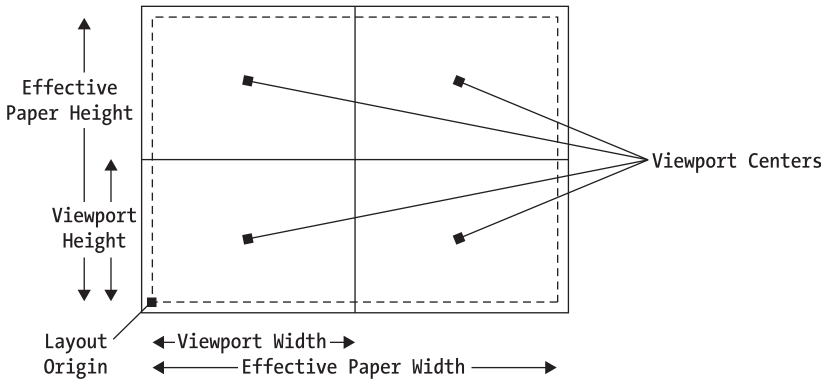


Figure 14-5. *Viewport parameters*

The code then halves the width and height dimensions to use as each viewpoint's size. It also uses this information to calculate each viewport's center.

The code then ensures that no old viewports exist, since they'd overlap or interfere with our new viewport.

```
'Clear the layout of old PViewports
For Each objAcadObject In ThisDrawing.PaperSpace
    If TypeName(objAcadObject) = "IAcadPViewport" Then
        objAcadObject.Delete
    End If
Next
```

Now the code creates the viewports. The code is similar for each viewport. Here's how it creates the Front viewport:

```
'create Front Viewport
dblPoint(0) = dblWidth - dblWidth * 0.5
dblPoint(1) = dblHeight + dblHeight * 0.5
dblPoint(2) = 0

Set objFrontVPort = ThisDrawing.PaperSpace.AddPViewport(dblPoint, _
                                                        dblWidth, dblHeight)
```

Next, the code sets the viewport's viewpoint with the `Direction` property, as follows:

```
'need to set view direction
dblViewDirection(0) = 0
dblViewDirection(1) = -1
dblViewDirection(2) = 0

objFrontVPort.Direction = dblViewDirection
```

Next, the code sets what the viewport displays using the `Display` method to enable the viewport display, and then setting the `MSpace` method to allow editing of model space from a floating paper-space viewport. Finally, the code sets the drawing to fill the viewport by scaling it to half its size relative to paper-space units.

```
objFrontVPort.Display acOn

ThisDrawing.MSpace = True

ThisDrawing.ActivePViewport = objFrontVPort
ThisDrawing.Application.ZoomExtents
ThisDrawing.Application.ZoomScaled 0.5, acZoomScaledRelativePSpace
```

After the code sets up all the viewports, the only thing left for it to do is regenerate the drawing to see all of the changes.

```
'regen in all viewports
ThisDrawing.Regen acAllViewports
```

Figure 14-6 shows this code's result.

AutoCAD gives you a number of properties to control the viewport's view. You can also control the layer on which the viewport is created and the linetype used to create the viewport. For full details, see Appendix A.

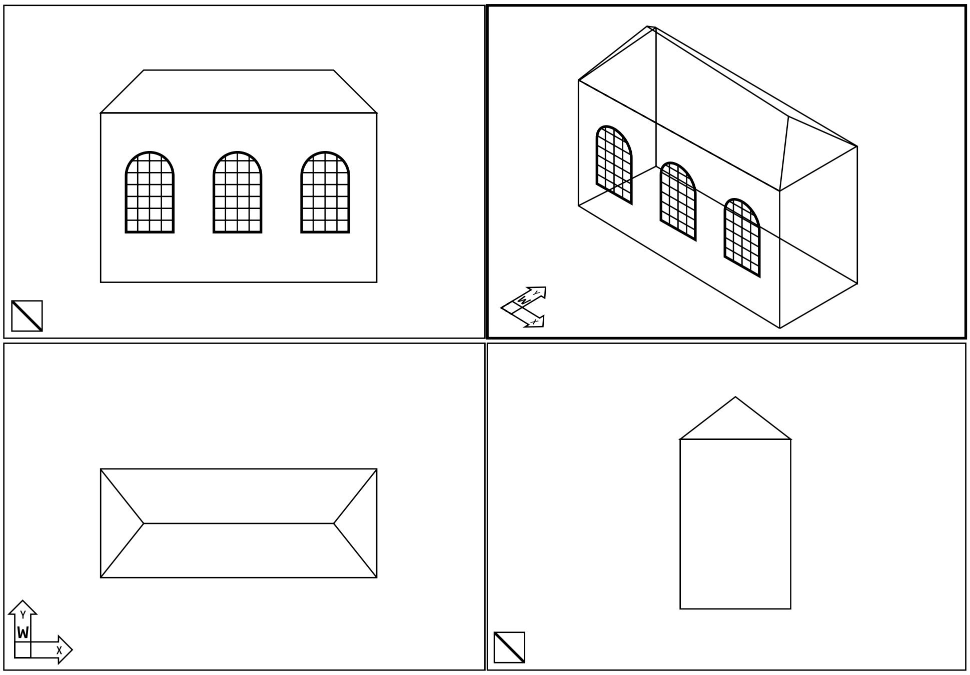


Figure 14-6. *Four-viewport view*

Summary

This chapter showed you how to name, define, and save a view for use later. AutoCAD provides methods and properties that let you create, set, and delete views. Viewports are bounded areas that display a specific view of your drawing in model space and paper space. This chapter examined the processes of creating both model-space and paper-space viewports.

CHAPTER 15

Layout and Plot Configurations

Being able to draw on the screen is great, but eventually you'll want a hard-copy printout of your design. For most businesses, this is a requirement as part of the customer-deliverable product. AutoCAD provides a number of methods to create through code plots of your drawing that the user normally generates using the AutoCAD Plot dialog box interface.

This chapter covers the following topics:

- Examining the `Plot` and `PlotConfiguration` objects
- Previewing and plotting a drawing
- Understanding the `Layouts` collection and the `Layout` object
- Controlling plot parameters

The `Plot` Object

The `Plot` object consists of a number of methods and properties that are used to plot layouts. The `Layout` object (discussed later in the chapter) holds the plot settings and visual properties of a model-space or paper-space block. You access the `Plot` object via the `Document` object using the following syntax:

```
Set PlotObject = DocumentObject.Plot
```

To declare and set a reference to the `Plot` object for the currently active drawing, you might use this code:

```
Dim objPlot As AcadPlot

Set objPlot = ThisDrawing.Plot
```

Through the methods and properties of the `Plot` object, you can display a plot preview, plot to a plotting device or to a file, and employ batch-mode plotting. A detailed discussion of batch-mode plotting falls outside the scope of this book, but Appendix A covers the relevant properties and methods.

Plotting Your Drawing

This section details the various means used to plot your drawings.

The DisplayPlotPreview Method

It's a good idea to preview your drawing before you print it out. AutoCAD provides a means of previewing the active layout of your drawing using the `DisplayPlotPreview` method. This method displays the Plot Preview box, which must be dismissed by the user rather than through code. The drawing for which the `DisplayPlotPreview` method is invoked will become the active drawing and will remain so even after you've dismissed the Plot Preview box.

```
PlotObject.DisplayPlotPreview Preview
```

Table 15-1 shows the `DisplayPlotPreview` property's parameter.

Table 15-1. *The* `DisplayPlotPreview` *Property Parameter*

Name	Data Type	Description
`Preview`	Long	This determines the kind of preview used. It must be one of the `AcPreviewMode` constants described in Table 15-2.

In AutoCAD 2005 and later, there is only one type of Preview mode, full preview, which takes longer to perform than a partial preview because it requires a regeneration of the drawing, although no optimization or vector sorting takes place. For backward compatibility, two `AcPreviewMode` constants are available, though only the full preview is used.

Table 15-2 shows the `Preview` type's constants.

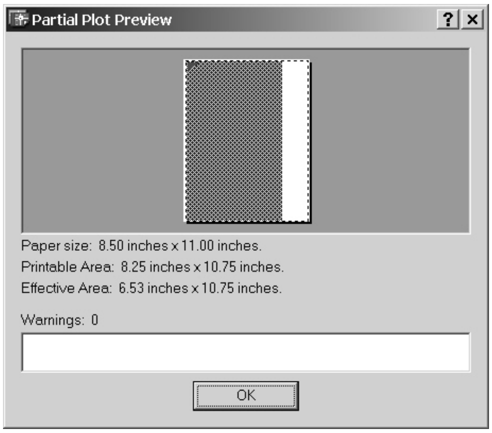
Table 15-2. `Preview` *Type Constants*

Constant	Value	Description
`acPartialPreview`	0	This type of preview shows the effective plot area relative to the paper size.
`acFullPreview`	1	This type of preview displays the drawing on screen, as it will appear when plotted.

The following code allows the user to decide the type of plot preview:

```
Public Sub PlotPreview()
    If MsgBox("A preview of your drawing will be displayed." & _
                   "Would you like to see a full preview?", vbYesNo) =
vbYes Then
        ThisDrawing.Plot.DisplayPlotPreview acFullPreview
    Else
        ThisDrawing.Plot.DisplayPlotPreview acPartialPreview
    End If
End Sub
```

Figure 15-1 depicts the two different types of plot previews.



Paper size: 8.50 inches x 11.00 inches.
Printable Area: 8.25 inches x 10.75 inches.
Effective Area: 6.53 inches x 10.75 inches.

Warnings: 0

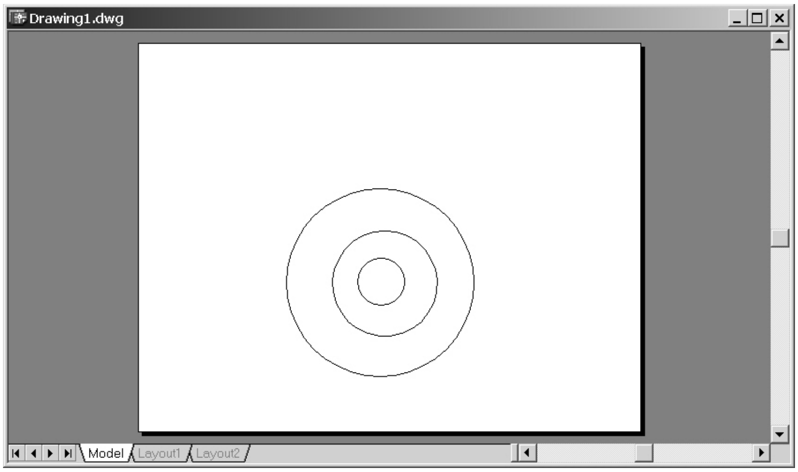


Figure 15-1. *Partial versus full plot preview*

Plotting with Layouts

Layout configurations that you create to control plot settings and visual properties of model- or paper-space geometry are held in the `Layouts` collection. ActiveX layouts differ from their AutoCAD user interface counterparts by splitting the standard AutoCAD layout into two objects: a `Layout` object containing the visual properties and plot settings as they would appear in the standard AutoCAD interface, and a `Block` object containing the geometry.

You access the `Layouts` collection via the `Document` object using the following syntax:

```
Set LayoutsCollection = DocumentObject.Layouts
```

To declare and set a reference to the Layouts object for the currently active drawing, you might use this code:

```
Dim objLayouts As AcadLayouts

Set objLayouts = ThisDrawing.Layouts
```

A Layout object has only one Block object associated with it and you access it via the Block property. Similarly, the one layout is associated with a Block object and you can access it through the Layout property.

You can access a Layout object using the Item method of the Layouts collection or you can reference the active layout through the ActiveLayout property of the Document object:

```
Set LayoutObject = DocumentObject.ActiveLayout
```

You can use many of the properties and methods of the Layout object to control plotting features such as plot rotation and scale. These properties are also exposed by the PlotConfiguration object and are covered later in this chapter.

The SetLayoutsToPlot Method

You use the SetLayoutsToPlot method to specify the layout or layouts that you want to plot. You'll normally use this method prior to a call to the PlotToDevice or PlotToFile method to override the default plotting of the active layout. After you've called either of these plotting methods, the default returns to the active layout.

```
PlotObject.SetLayoutsToPlot(Layouts)
```

Table 15-3 shows the SetLayoutsToPlot method's parameter.

Table 15-3. *The* SetLayoutsToPlot *Method Parameter*

Name	Data Type	Description
Layouts	Variant	An array of string names identifying the layouts to plot. If this list is Null, this method is effectively useless and the default layout to plot will remain as the active layout.

The following code plots the layouts specified by the user:

```
Public Sub PlotLayouts()
Dim objLayout As AcadLayout
Dim strLayoutList() As String
Dim intCount As Integer
Dim objPlot As AcadPlot
```

```
    intCount = -1
    For Each objLayout In ThisDrawing.Layouts
      If MsgBox("Do you wish to plot the layout: " _
                & objLayout.Name, vbYesNo) = vbYes Then
        intCount = intCount + 1
        ReDim Preserve strLayoutList(intCount)
          strLayoutList(intCount) = objLayout.Name
      End If
    Next objLayout

    Set objPlot = ThisDrawing.Plot
    objPlot.SetLayoutsToPlot strLayoutList
    objPlot.PlotToDevice
End Sub
```

The `PlotToDevice` Method

You may print or plot a layout of the active drawing to a file by any device connected to your system using the `PlotToDevice` method. This method plots using the current configuration or using configuration settings held in a PC3 file. You can specify the device through the `ConfigName` property for the `Layout` or `PlotConfiguration` objects. This method returns a value of `True` if the plot was successful; otherwise, it returns `False`.

By default, this method will print the currently active layout. If you want to print a different selection of layouts, you need to call the `SetLayoutsToPlot` method, described previously, to specify the required layouts.

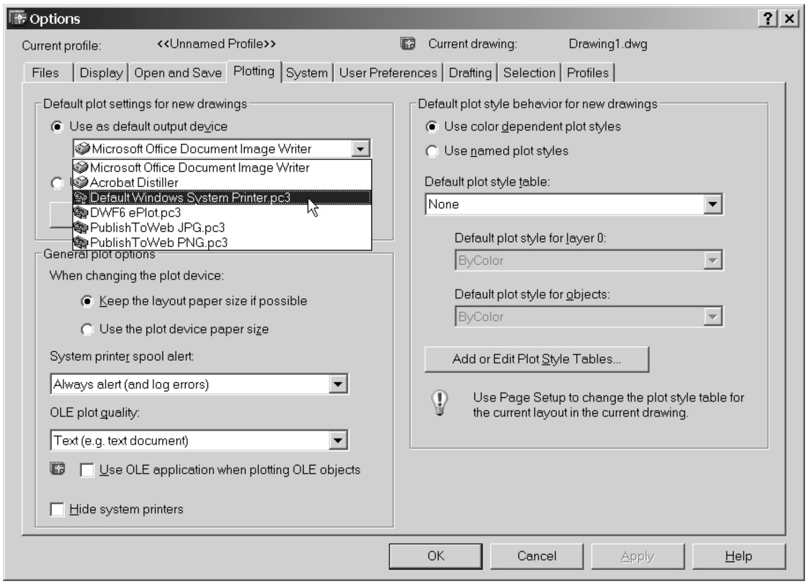
```
blnPlotSuccessful = PlotObject.PlotToDevice [PlotConfiguration]
```

Table 15-4 shows the `PlotToDevice` method's parameter.

Table 15-4. *The* `PlotToDevice` *Method Parameter*

Name	Data Type	Description
`PlotConfiguration`	String	This optional parameter specifies the full path and file name of a PC3 file to use instead of the current configuration. If the file isn't found, AutoCAD will search the printer configuration path for the file. If this parameter isn't provided or is invalid, the current configuration will be used.

You can look at the available printers using the Options dialog box inside AutoCAD, as shown in Figure 15-2. The Plotting tab displays all the available printer device names. Unfortunately, AutoCAD doesn't provide a means to see this list programmatically.



Current profile: <<Unnamed Profile>>
Current drawing: Drawing1.dwg

[✓ Use as default output device]: Microsoft Office Document Image Writer

When changing the plot device:
[✓ Keep the layout paper size if possible]
Use the plot device paper size

System printer spool alert: Always alert (and log errors)

OLE plot quality: Text (e.g. text document)

☐ Use OLE application when plotting OLE objects

☐ Hide system printers

[✓ Use color dependent plot styles]
Use named plot styles

Default plot style table: None

Default plot style for layer 0: ByColor

Default plot style for objects: ByColor

Use Page Setup to change the plot style table for the current layout in the current drawing.

Figure 15-2. *The Plotting tab of the Options dialog box*

The `PlotToFile` Method

This method works in a similar way to the `PlotToDevice` method except that it plots layouts of the active drawing to a specified file. The default layout to be plotted is the active layout, but this may be overridden by the `SetLayoutsToPlot` method given earlier in this chapter. This method returns a Boolean indicating whether the plot to file was successful. A value of `True` indicates success.

```
blnPlotSuccessful = PlotObject.PlotToFile _
   (PlotFile [, PlotConfiguration])
```

Table 15-5 shows the `PlotToFile` method's parameters.

Table 15-5. `PlotToFile` *Method Parameters*

Name	Data Type	Description
`PlotFile`	String	The name of the file that the layout or layouts is plotted to. When you're plotting multiple layouts, it's possible for the file name for each plot to be generated from the drawing and layout names. This automatic generation of plot file names will occur if a path but no file name is supplied. Otherwise, the last layout specified will be plotted to the file name provided.
`PlotConfiguration`	String	This optional parameter specifies the full path and file name of a PC3 file to use instead of the current configuration. If the file isn't found, AutoCAD will search the printer configuration path for the file. If this parameter isn't provided or is invalid, the current configuration will be used.

It isn't necessary to provide an extension for the plot file. However, if an extension isn't provided, then the generated extension depends upon the default extension for the driver or device specified in the `PlotConfiguration` parameter. If a file extension is provided, it will be overwritten for certain raster output drivers that replace user-provided extensions with `.gif`.

Some plot configurations will invoke a plot to file indirectly, such as Adobe Acrobat PDF or the newer Microsoft Document Imaging (MDI) output. For example, a PC3 configuration like a PDF or MDI will still work with `PlotToDevice`, but it will then hand the request to the Adobe rendering service to produce the PDF output or the MDI rendering service for producing MDI or TIFF documents.

Plot Configurations

When you plot a drawing, its appearance depends upon the plot configuration you use. AutoCAD provides the `PlotConfiguration` object to hold the settings for a particular configuration. A plot configuration is similar to a layout, as it contains the same plot information, and its corresponding AutoCAD objects expose almost identical properties and methods. However, a `PlotConfiguration` object isn't associated with any Block Definition, and so consists of a collection of plot settings that you may use with any geometry.

`PlotConfiguration` Objects

You can have as many `PlotConfiguration` objects as you need, identified by a name and held in the `PlotConfigurations` collection. You access this collection through the `Document` object as follows:

```
Set PlotConfigurationsCollection = DocumentObject.PlotConfigurations
```

A `PlotConfiguration` object belongs either to model space or paper space (layouts) only, not both. Additionally, when you import a `PlotConfiguration` object by way of a `PageSetup` import (by command or programmatically), it will apply and attach only to the appropriate working space that it's related to in the source drawing. To create a new `PlotConfiguration` object, you use the `Add` method, which for the `PlotConfigurations` collection takes the syntax shown here:

```
Set PlotConfigurationObject = PlotConfigurationsCollection.Add(Name[, ModelType])
```

Table 15-6 shows the `Add` method's parameters.

Table 15-6. `Add` *Method Parameters*

Name	Data Type	Description
`Name`	String	The identifying name of the new `PlotConfiguration` object.
`ModelType`	Boolean	An optional parameter that determines which layouts the plot configuration may apply to. If this is set to `True`, the configuration applies only to model space; otherwise, it applies to all layouts. The default is `False`.

To create a new PlotConfiguration object named PlotConfig1 that applies only to model space, you could use the following code:

```
Public Sub AddPConfig()
Dim objPlotConfigs As AcadPlotConfigurations
Dim objPlotConfig As AcadPlotConfiguration

  Set objPlotConfigs = ThisDrawing.PlotConfigurations
  Set objPlotConfig = objPlotConfigs.Add("PlotConfig1", True)

End Sub
```

Each of the properties and methods discussed in the sections that follow determine the appearance of a plot. Both the PlotConfiguration and the Layout objects expose them, although for a Layout object they apply only to the individual layout. Remember, a Layout object defines a specific view of your drawing geometry for plotting, whereas a PlotConfiguration object is generic and not associated with any particular view or drawing-view layout. Effectively, they both achieve the same result but with the capability to use completely different input data.

When I present the syntax of each method or property in the sections that follow, I use the word "object" to refer to either the PlotConfiguration or Layout object.

The PlotType Property

You use this property to read or set how much of a drawing or layout is plotted. Modifications to this property come into effect only after the drawing is regenerated.

```
Object.PlotType = lngPlotType
```

The PlotType property holds one of the AcPlotType constants listed in Table 15-7.

Table 15-7. AcPlotType *Constants*

Constant	Value	Description
acDisplay	0	Everything within the current display is printed.
acExtents	1	Everything within the extents of the currently selected space is printed.
acLimits	2	Everything within the limits of the current space is printed.
acView	3	The view named by the ViewToPlot property is printed. You must call the ViewToPlot property before you can set PlotType to acView.
acWindow	4	Everything in the window specified by the SetWindowToPlot method is printed. You must call the SetWindowToPlot method before you can set the PlotType to acWindow.
acLayout	5	Everything within the margins of the specified paper size is printed. The origin is calculated from the (0, 0) coordinate location of the layout. This option doesn't apply to model space.

The `ViewToPlot` Property

AutoCAD allows you to plot a previously named view by setting the `ViewToPlot` property to the name of the view you want to use. For this to have an effect, it's then necessary to set the `PlotType` property of the `Layout` or `PlotConfiguration` object to `acView`. You supply the view name as a string.

```
Object.ViewToPlot = strViewName
```

I discuss how to manipulate views through code in Chapter 14. To view the currently available views through the AutoCAD user interface, navigate to the View dialog box via the View ➤ Named Views menu. Figure 15-3 shows the View dialog box.

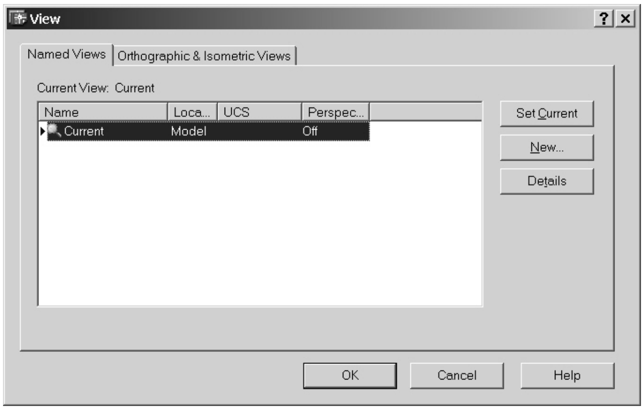


Figure 15-3. *The View dialog box*

The following code example sets the view to be plotted based on user input and then plots that view to file:

```
Public Sub PlotView()
Dim intCount As Integer
Dim strViewToPlot As String
Dim objView As AcadView
Dim strFilename As String

    strViewToPlot = InputBox("Enter name of view to plot: ", "Plot View")

On Error Resume Next
    Set objView = ThisDrawing.Views(strViewToPlot)
    If objView Is Nothing Then
        MsgBox "This view does not exist"
        Exit Sub
    End If
    strFilename = InputBox("Enter a filename to plot to")
    If strFilename = "" Then Exit Sub

    'set view to plot
    ThisDrawing.ModelSpace.Layout.ViewToPlot = strViewToPlot
    ThisDrawing.ModelSpace.Layout.PlotType = acView

    'Initiate the plot
    ThisDrawing.Plot.PlotToFile strFilename
End Sub
```

The `SetWindowToPlot` Method

AutoCAD allows you to plot part of a layout within a rectangular window by using the `SetWindowToPlot` method to set the coordinates defining the lower-left and upper-right coordinates of the window. For this method to have an effect, it's necessary to set the `PlotType` property of the `Layout` or `PlotConfiguration` object to `acWindow`.

```
Object.SetWindowToPlot(LowerLeftCorner, UpperRightCorner)
```

Table 15-8 shows the `SetWindowToPlot` method's parameters.

Table 15-8. `SetWindowToPlot` *Method Parameters*

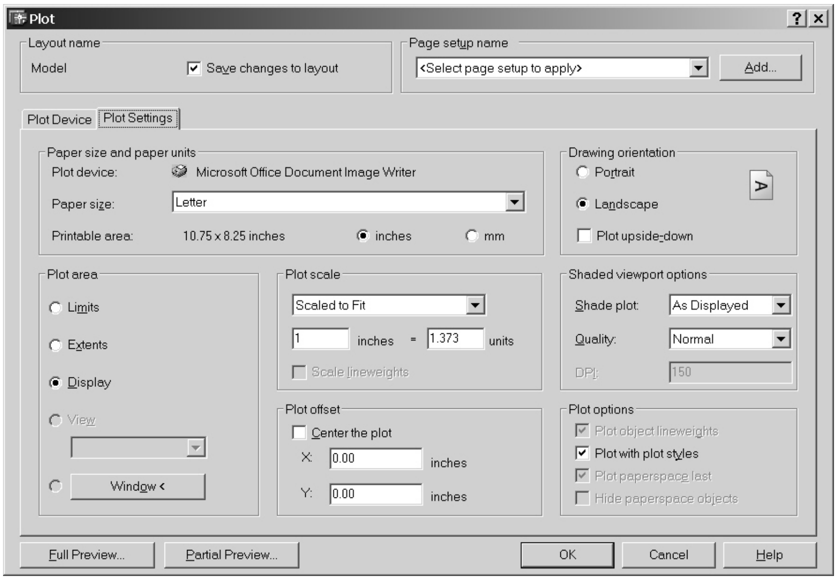
Name	Data Type	Description
`LowerLeftCorner`	Variant	A two-element array of doubles specifying in paper units the X and Y values, measured from the origin, for the lower-left corner of the window to be plotted
`UpperRightCorner`	Variant	A two-element array of doubles specifying in paper units the X and Y values, measured from the origin, for the upper-right corner of the window to be plotted

The following example illustrates how to implement this method by retrieving a window to plot from the user and then plotting that part of the drawing to file:

```
Public Sub PlotWindow()
Dim varLowerLeftCorner As Variant
Dim varUpperRightCorner As Variant
Dim dblLowerLeftCorner(1) As Double
Dim dblUpperRightCorner(1) As Double
Dim intCount As Integer
Dim strFilename As String

    'set the plot type
    varLowerLeftCorner = ThisDrawing.Utility.GetPoint(, _
                                 "Select lower-left corner of window: ")
    varUpperRightCorner = ThisDrawing.Utility.GetCorner(varLowerLeftCorner, _
                                "Select upper-right corner of window: ")
    For intCount = 0 To 1
        dblLowerLeftCorner(intCount) = CDbl(varLowerLeftCorner(intCount))
        dblUpperRightCorner(intCount) = CDbl(varUpperRightCorner(intCount))
    Next intCount
    ThisDrawing.ActiveLayout.SetWindowToPlot dblLowerLeftCorner, _
                                            dblUpperRightCorner

    ThisDrawing.ActiveLayout.PlotType = acWindow
    'initiate the plot
    strFilename = InputBox("Enter a filename to plot to")
    If strFilename = "" Then Exit Sub

    ThisDrawing.Plot.PlotToFile strFilename

End Sub
```

Controlling Plot Parameters

A number of parameters are normally set using the Plot dialog box shown in Figure 15-4. AutoCAD provides a means to control these settings programmatically. In this section I cover the options that are most commonly modified.


Plot

Layout name: Model — [✓] Save changes to layout

Page setup name: <Select page setup to apply> — Add...

Plot Device | Plot Settings

Paper size and paper units
- Plot device: Microsoft Office Document Image Writer
- Paper size: Letter
- Printable area: 10.75 x 8.25 inches — [● inches] ○ mm

Drawing orientation
- ○ Portrait
- ● Landscape
- ☐ Plot upside-down

Plot area
- ○ Limits
- ○ Extents
- ● Display
- ○ View
- ○ Window <

Plot scale
- Scaled to Fit
- 1 inches = 1.373 units
- ☐ Scale lineweights

Shaded viewport options
- Shade plot: As Displayed
- Quality: Normal
- DPI: 150

Plot offset
- ☐ Center the plot
- X: 0.00 inches
- Y: 0.00 inches

Plot options
- [✓] Plot object lineweights
- [✓] Plot with plot styles
- [✓] Plot paperspace last
- ☐ Hide paperspace objects

Full Preview... | Partial Preview... | OK | Cancel | Help

Figure 15-4. *The Plot dialog box*

All the methods and properties for controlling the plot parameters are available to both the `PlotConfiguration` and `Layout` objects. Consequently, when I present the syntax in the following sections, I use the word "object" to denote either object.

The CanonicalMediaName Property

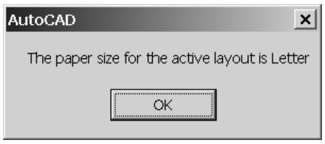
You can read or set the paper size to be used when plotting using the `CanonicalMediaName` property of the `Layout` or `PlotConfiguration` object. You specify the paper size by a name given as a string, and changes to this property won't take effect until the drawing has been regenerated.

```
Object.CanonicalMediaName = strPaperSize
```

This code snippet shows how to read the current setting of the paper size for the active layout of a drawing:

```
Public Sub PaperSizeNames()
    MsgBox "The paper size for the active layout is " & _
           ThisDrawing.ActiveLayout.CanonicalMediaName
End Sub
```

The result of this code may be similar to that shown in Figure 15-5.


AutoCAD

The paper size for the active layout is Letter

Figure 15-5. `PaperSizeNames` *output*

The GetCanonicalMediaNames Method

You can use the `GetCanonicalMediaNames` method to retrieve the names of the available paper sizes for a specified plot device. The return value for this method is an array of strings holding the names of the available paper sizes.

```
varPaperSizeNames = Object.GetCanonicalMediaNames()
```

It's advisable to call the `RefreshPlotDeviceInfo` method before you use this method for the first time, and each time you're changing the default plot device. This ensures that the plot, paper size names, and plot style table information accurately reflect the current system state.

The GetPaperSize Method

Although you can read the `CanonicalMediaName` property to find out the name of the paper size to be used, you may want to know the actual width and height dimensions. The `GetPaperSize` method retrieves the width and height of the configured paper, given in the current paper units.

```
Object.GetPaperSize Width, Height
```

Table 15-9 shows the `GetPaperSize` method's parameters.

Table 15-9. `GetPaperSize` *Method Parameters*

Name	Data Type	Description
`Width`	Double	The width of the paper in units specified by the `PaperUnits` property of the layout or plot configuration
`Height`	Double	The height of the paper in units specified by the `PaperUnits` property of the layout or plot configuration

The following code example incorporates the property and methods associated with the paper size. First, the available paper sizes for the current layout are retrieved and displayed to the user. If the user then enters one of these in an input box, the dimensions of the chosen paper size display.

```
Public Sub PaperSize()
Dim varPaperSizeNames As Variant
Dim strPaperSizeNames As String
Dim intCount As Integer
Dim strChoosenPaperSize As String

    varPaperSizeNames = ThisDrawing.ActiveLayout.GetCanonicalMediaNames
    strPaperSizeNames = "These are the paper sizes available:" & vbCrLf
    For intCount = 0 To UBound(varPaperSizeNames)
        strPaperSizeNames = strPaperSizeNames & _
                varPaperSizeNames(intCount) & ", "
    Next intCount
    strPaperSizeNames = strPaperSizeNames & vbCrLf & " Please choose one."

    strChoosenPaperSize = InputBox(strPaperSizeNames, "Pick a paper size")

    For intCount = 0 To UBound(varPaperSizeNames)
        If StrComp(strChoosenPaperSize, varPaperSizeNames(intCount), 1) = 0 _
            Then GoTo DisplaySize
    Next intCount
    MsgBox "You did not enter a valid paper size name."
    Exit Sub

DisplaySize:
Dim dblPaperWidth As Double
Dim dblPaperHeight As Double
Dim lngPaperUnits As Long
Dim strPaperUnits As String
```

```
    ThisDrawing.ActiveLayout.GetPaperSize dblPaperWidth, dblPaperHeight
    lngPaperUnits = ThisDrawing.ActiveLayout.PaperUnits
    Select Case lngPaperUnits
        Case 0
           strPaperUnits = "inches"
           dblPaperWidth = dblPaperWidth / 25.4
           dblPaperHeight = dblPaperHeight / 25.4
        Case 1
        strPaperUnits = "millimeters"
    End Select
    MsgBox dblPaperWidth & " by " & dblPaperHeight & " " & strPaperUnits
End Sub
```

The Plot Scale

You normally draw an AutoCAD drawing in units that reflect the true size of the object being represented. Therefore, when you print your drawing you'll probably need to scale the plot so that it fits comfortably onto the paper. There are two types of plot scales, and both give the ratio of the plot size to drawing size. The first is the standard scale and it's set to one of AutoCAD's predefined scales. The second, custom scale, can be set to any value.

The `StandardScale` Property

You can use the `StandardScale` property to set the plot scale to one of the predefined AutoCAD scales.

```
Object.StandardScale = lngAcPlotScale
```

This must be one of the `AcPlotScale` constants detailed in Table 15-10.

Table 15-10. `AcPlotScale` *Constants*

Constant	Value	Description
acScaleToFit	0	Scale to Fit
ac1_128in_1ft	1	1/128″ : 1′
ac1_64in_1ft	2	1/64″ : 1′
ac1_32in_1ft	3	1/32″ : 1′
ac1_16in_1ft	4	1/16″ : 1′
ac3_32in_1ft	5	3/32″ : 1′
ac1_8in_1ft	6	1/8″ : 1′
ac3_16in_1ft	7	3/16″ : 1′
ac1_4in_1ft	8	1/4″ : 1′
ac3_8in_1ft	9	3/8″ : 1′
ac1_2in_1ft	10	1/2″ : 1′
ac3_4in_1ft	11	3/4″ : 1′
ac1in_1ft	12	1″ : 1′
ac3in_1ft	13	3″ : 1′

Constant	Value	Description
`ac6in_1ft`	14	6″ : 1′
`ac1ft_1ft`	15	1′ : 1′
`ac1_1`	16	1:1
`ac1_2`	17	1:2
`ac1_4`	18	1:4
`ac1_8`	19	1:8
`ac1_10`	20	1:10
`ac1_16`	21	1:16
`ac1_20`	22	1:20
`ac1_30`	23	1:30
`ac1_40`	24	1:40
`ac1_50`	25	1:50
`ac1_100`	26	1:100
`ac2_1`	27	2:1
`ac4_1`	28	4:1
`ac8_1`	29	8:1
`ac10_1`	30	10:1
`ac100_1`	31	100:1

The `GetCustomScale` Method

You use this method to examine the scale for a layout or plot configuration.

```
Object.GetCustomScale(Numerator, Denominator)
```

Table 15-11 shows the `GetCustomScale` method's parameters.

Table 15-11. `GetCustomScale` *Method Parameters*

Name	Data Type	Description
`Numerator`	Double	The numerator in the scale ratio. This value represents the number of inches or millimeters for the plot. The unit of measurement is held in the `PaperUnits` parameter.
`Denominator`	Double	The denominator in the scale ratio. This value represents the number of drawing units used to scale to the measurement given in the numerator.

You can use the following code to retrieve the scales used in each of the layouts of your drawing:

```
Public Sub GetScales()
Dim objLayout As AcadLayout
Dim dblNumerator As Double
Dim dblDenominator As Double
```

```
    For Each objLayout In ThisDrawing.Layouts
        'Get custom scale information
        objLayout.GetCustomScale dblNumerator, dblDenominator
        If objLayout.PaperUnits = acInches Then
            MsgBox "The scale of " & objLayout.Name & _
                   " is " & dblNumerator & " inches = " & _
                   dblDenominator & " Drawing Units"
        ElseIf objLayout.PaperUnits = acMillimeters Then
            MsgBox "The scale of " & objLayout.Name & _
                   " is " & dblNumerator & " millimeters = " & _
                   dblDenominator & " Drawing Units"
        Else
        MsgBox "The scale of " & objLayout.Name & " is " & _
                dblNumerator & " pixels = " & dblDenominator & _
               " Drawing Units"
    End If
  Next
End Sub
```

The `SetCustomScale` Method

This method works in a similar way to `GetCustomScale`, described previously, except it sets the plot scale. Any changes you make through this method become effective only after the drawing has been regenerated.

```
Object.SetCustomScale(Numerator, Denominator)
```

Table 15-12 shows the `SetCustomScale` method's parameters.

Table 15-12. `SetCustomScale` *Method Parameters*

Name	Data Type	Description
`Numerator`	Double	A positive value representing the number of inches or millimeters of the plot. The unit of measurement is held in the `PaperUnits` parameter.
`Denominator`	Double	A positive value representing the number of drawing units for the drawing that will be scaled to the measurement given in the numerator.

The `UseStandardScale` Property

You use this property to read or set whether a plot should use a standard or custom scale. It holds a Boolean value that is set to `True` if a standard scale is in use or `False` if a custom plot scale is in use.

```
Object.UseStandardScale = blnStandardScale
```

The following code sample employs the `UseStandardScale` property to retrieve the type of scale used for each layout and displays that information to the user:

```
Public Sub UseStandardScale()
Dim objLayout As AcadLayout

    For Each objLayout In ThisDrawing.Layouts
        If objLayout.UseStandardScale Then
            MsgBox "The scale of " & objLayout.Name & " is a Standard scale"
        Else
            MsgBox "The scale of " & objLayout.Name & " is a Custom scale"
        End If
    Next
End Sub
```

The `PlotRotation` Property

If you want to plot a layout at an angle other than the default of 08, then you'll need to specify the angle of rotation using the `PlotRotation` property. This property allows you to select predefined angles of 08, 908, 1808, and 2708, measured counterclockwise in the XY plane from the X-axis of the WCS. Changes to the `PlotRotation` property won't take effect until the drawing has been regenerated.

```
Object.PlotRotation = lngAcPlotRotation
```

Any `AcPlotRotation` constant shown in Table 15-13 is an acceptable value for this property.

Table 15-13. `AcPlotRotation` *Constants*

Constant	Value	Description
`ac0degrees`	0	The layout and plot have the same orientation.
`ac90degrees`	1	The plot is rotated by an angle of 90° from the layout.
`ac180degrees`	2	The plot is rotated by an angle of 180° from the layout.
`ac270degrees`	3	The plot is rotated by an angle of 270° from the layout.

In the following code sample, the user is asked at which angle he or she would like to preview the active layout. The preview then displays, and finally the `PlotRotation` property returns to its original value.

```
Public Sub PlotAngle()
Dim strPlotAngle As String
Dim lngStoreAngle As Long

    lngStoreAngle = ThisDrawing.ActiveLayout.PlotRotation
    strPlotAngle = InputBox("For a plot preview please enter the angle" & _
                   "(in degrees: 0, 90, 180 or 270) that you would like" & _
                   " your layout plotted at", , "0")
    Select Case strPlotAngle
      Case "0"
          ThisDrawing.ActiveLayout.PlotRotation = ac0degrees
      Case "90"
         ThisDrawing.ActiveLayout.PlotRotation = ac90degrees
     Case "180"
         ThisDrawing.ActiveLayout.PlotRotation = ac180degrees
      Case "270"
         ThisDrawing.ActiveLayout.PlotRotation = ac270degrees
      Case Else
        MsgBox "You entered an invalid value"
        Exit Sub
    End Select
    ThisDrawing.Regen acActiveViewport
    ThisDrawing.Plot.DisplayPlotPreview acFullPreview
    ThisDrawing.ActiveLayout.PlotRotation = lngStoreAngle
    ThisDrawing.Regen acActiveViewport
End Sub
```

Summary

In this chapter you explored several topics specific to plotting. This is one area in which AutoCAD 2000 and later versions are vastly different from previous versions. In AutoCAD 2000 and higher, Autodesk added a lot of plotting power, and when you're programming your plotting applications, the use of plot configuration files can reduce the amount of hard coding required by taking advantage of the features built into the AutoCAD product. Some differences exist between AutoCAD 2000, 2000i, 2002, 2004, 2005, and 2006 as far as programmatic capabilities with respect to plotting and plot configuration management, be sure to refer to the appropriate documentation for your platform.

CHAPTER 16

Controlling Menus and Toolbars

AutoCAD 2000 (and higher) gives you the ability to control the menus and toolbars programmatically. You can manipulate existing menus or create new entries using the objects exposed by the AutoCAD object model. Although you can't create a completely new menu structure programmatically, you can make changes to the existing menus.

This chapter covers the following topics in detail:

- Loading, saving, and unloading menu groups
- Assigning accelerator keys
- Manipulating the menu bar
- Creating and editing menus
- Creating and editing toolbars
- Floating and docking toolbars

In AutoCAD 2006, the menu system has been radically changed. The system looks the same externally, but the internal functioning of the system and the files the system uses have been overhauled. Fortunately, this is invisible to most VBA programs, as most of the objects and methods have not changed. However, if you want your programs to work with both 2006 and earlier releases, you should know about some of the differences.

- In AutoCAD 2005 and earlier, the menus are stored in `.mnu`, `.mns`, `.mnc`, and `.mnr` files. AutoCAD 2006 uses just `.cui` and `.mnr` files. In AutoCAD 2006, when a partial menu is loaded using the `MENULOAD` command or the `CUILOAD` command, a pointer to the partial menu is stored in the current main `.cui` file.
- AutoCAD 2006 can load existing `.mnu` or `.mns` files, at which point they are converted to `.cui` files. Subsequent modification to the menus in AutoCAD 2006 will update only the `.cui` files.
- `.cui` files are stored in XML format. Although XML documentation is readily available, the `.cui` file structure itself is not documented. Unlike the situation with `.mnu` and `.mns` from earlier releases, Autodesk strongly discourages creating or editing `.cui` files manually. Instead, users are expected to use AutoCAD 2006's new `CUI` command to perform all `.cui` file modifications using a dialog box interface.

- AutoCAD 2006 introduces the concept of *work spaces*. These are collections of menu settings, specifying the order of pull-downs, location, and visibility of toolbars, and so on. The work spaces are stored in the current main `.cui` file. Unfortunately, there is no ActiveX access to work spaces.
- In AutoCAD 2005 and earlier, there is only one main menu and many partial menus. In AutoCAD 2006, there are Main and Enterprise (read-only) menus, each of which can have partial menus attached. To attach menus to the Enterprise menu, it is necessary to temporarily make it the Main menu while using the `CUILOAD` command. Unfortunately, there is no ActiveX access to the Enterprise menu setting. The user has to change this in the Options dialog box.

In summary, the AutoCAD 2006 menu overhaul, while introducing some useful concepts, has placed various obstacles in the way of the VBA programmer's ability to safely make changes to the menu system. In light of these problems, you may want to consider using AutoCAD's `CUI` interface to create partial menus for your users, rather than doing it under program control as described in this chapter. If you still want VBA control over your AutoCAD 2006 menus, read on.

■**Note** This chapter's content is specific to AutoCAD 2000 and later. The previous object models didn't expose any objects to deal with menus and toolbars.

I will start this chapter with the `MenuGroups` collection because it's the parent object of the `MenuGroup` object, which in turn is the parent object of the `ToolBars` and `PopupMenus` collections. The `MenuBar` collection holds all the `PopupMenu` objects that are currently displayed in the AutoCAD menu bar.

The MenuGroups Collection

The menus loaded into the current session of AutoCAD are contained in the `MenuGroups` collection. These menus, grouped into `MenuGroup` objects, may or may not be visible on the menu bar, but they're all still contained within this collection. Each `MenuGroup` object provides access to the toolbars and pop-up menus available within an AutoCAD session.

Loading Menu Groups

You can use the `Load` method of the `MenuGroups` collection to load a new menu group contained in a menu file (`.mnc`, `.mns`, `.mnu`, or `.cui`) into an AutoCAD session.

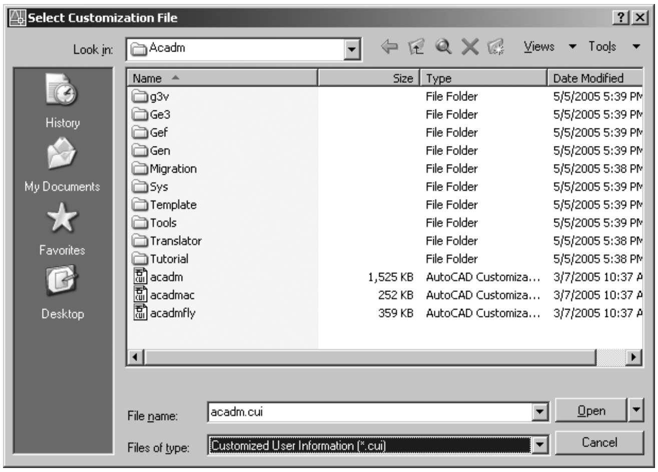
```
Set MenuGroupObject = MenuGroupsCollection.Load (MenuFileName [,BaseMenu])
```

Table 16-1 shows the `Load` method's parameters.

Table 16-1. *The* `Load` *Method's Parameters*

Name	Data Type	Description
`MenuFileName`	String	The path and file name of the menu file to be loaded.
`BaseMenu`	Boolean	This optional parameter determines whether the menu group is loaded as a base or partial menu. If it's set to `True`, then the menu group is loaded as a base menu. Otherwise, the menu group is loaded as a partial menu. The default is `False`.

Using the `Load` method with the `BaseMenu` parameter set to `True` equates to executing the `MENU` command inside the AutoCAD application and selecting a file through the Select Customization File dialog box (see Figure 16-1). Alternatively, you can execute the `MENULOAD` or `CUILOAD` command and check the Replace All option in the Menu Customization dialog box (see Figure 16-2). The newly loaded menu file becomes the only loaded menu group and completely replaces the previous menu bar.



Name	Size	Type	Date Modified
g3v		File Folder	5/5/2005 5:39 P
Ge3		File Folder	5/5/2005 5:39 P
Gef		File Folder	5/5/2005 5:39 P
Gen		File Folder	5/5/2005 5:39 P
Migration		File Folder	5/5/2005 5:38 P
Sys		File Folder	5/5/2005 5:39 P
Template		File Folder	5/5/2005 5:39 P
Tools		File Folder	5/5/2005 5:39 P
Translator		File Folder	5/5/2005 5:38 P
Tutorial		File Folder	5/5/2005 5:38 P
acadm	1,525 KB	AutoCAD Customiza...	3/7/2005 10:37 A
acadmac	252 KB	AutoCAD Customiza...	3/7/2005 10:37 A
acadmfly	359 KB	AutoCAD Customiza...	3/7/2005 10:37 A

File name: acadm.cui

Files of type: Customized User Information (*.cui)

Figure 16-1. *Using the Select Customization File dialog box to load a menu file*

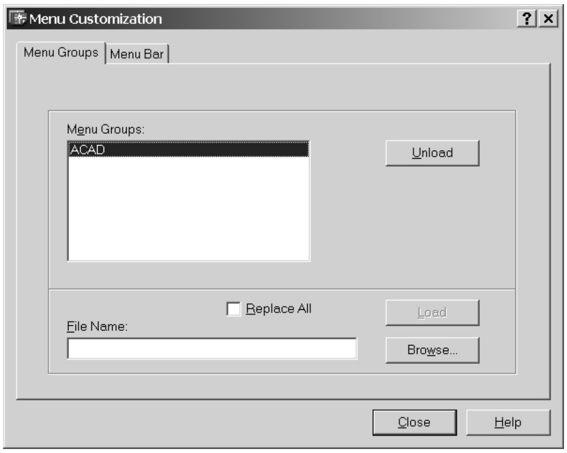


Figure 16-2. *Using the Menu Customization dialog box to load a menu group*

Alternatively, using the `Load` method with the `BaseMenu` parameter set to `False` equates to executing the `MENULOAD` or `CUILOAD` command inside the AutoCAD application and selecting a file through the Menu Customization dialog box. The menu group is loaded in addition to the already existing menu groups.

To find out what type of menu groups are loaded into an AutoCAD session, you can examine the `Type` property for each `MenuGroup` object.

```
lngMenuGroupType = MenuGroupObject.Type
```

This read-only property holds one of the `AcMenuGroupType` constants listed in Table 16-2.

Table 16-2. *The* `AcMenuGroupType` *Constants*

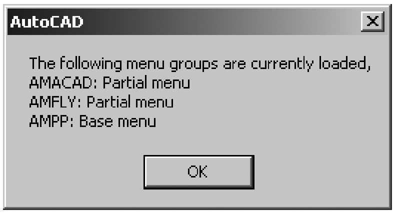
Constant	Value	Description
`AcBaseMenuGroup`	0	The menu group is a base menu group.
`AcPartialMenuGroup`	1	The menu group is a partial menu group.

The following code lists all the currently loaded menu groups and their types:

```
Public Sub ListMenuGroups()
Dim objMenuGroup As AcadMenuGroup
Dim strMenuGroupNames As String

    strMenuGroupNames = "The following menu groups are currently loaded, "

    For Each objMenuGroup In Application.MenuGroups
        If objMenuGroup.Type = acBaseMenuGroup Then
            strMenuGroupNames = strMenuGroupNames & vbCrLf & _
                                objMenuGroup.Name & ": Base menu"
        Else
            strMenuGroupNames = strMenuGroupNames & vbCrLf & _
                                objMenuGroup.Name & ": Partial menu"
        End If
    Next
    MsgBox strMenuGroupNames

End Sub
```

For my setup, this code yields the result shown in Figure 16-3.

Note that the MenuGroups collection doesn't expose an Add method, so you can't create new menu groups through code. However, you can copy an .mns file, give it a new file name, and then load and edit it as you require. You can also write a new .mnu or .mns file and load it entirely from VBA. It's usually best to work with partial menus and avoid modifying or replacing the AutoCAD menu system entirely, for two main reasons:



```
The following menu groups are currently loaded,
AMACAD: Partial menu
AMFLY: Partial menu
AMPP: Base menu
```

Figure 16-3. *Viewing the* `ListMenuGroups()` *code output*

- You don't want to upset your users (customers) by changing the familiar menu system entirely.
- Keeping your menus separate avoids the possibility of an AutoCAD service pack wiping out your hard work.

Until you feel confident with the material covered in this chapter, you should make a copy of an existing menu group file, unload the original, and load and use this copy to try the code in this chapter. You can then unload the copy and reload the original to restore your original AutoCAD menu structure.

■ **Note** You can't edit image menu items, screen menus, or tablet menus programmatically. However, you can load and unload these menu types using AutoCAD VBA.

The MenuGroup Object

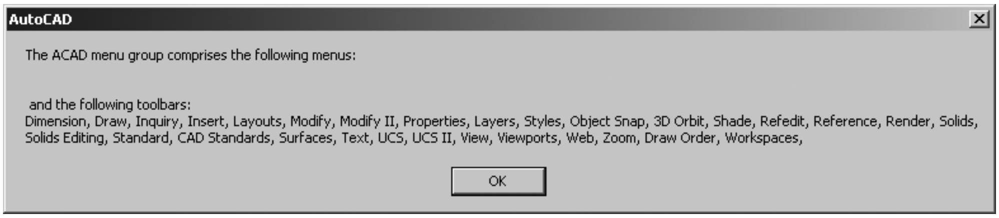
Once you've loaded a menu group, its corresponding MenuGroup object is added to the MenuGroups collection. A MenuGroup object contains two collections, PopupMenus and Toolbars. All the menus within a menu group are members of the PopupMenus collection, and all the toolbars are members of the Toolbars collection. You access these two collections via the Menus and Toolbars properties of the MenuGroup object with the following syntax:

```
Set PopupMenusCollection = MenuGroupObject.Menus

Set ToolbarsCollection = MenuGroupObject.Toolbars
```

The following code sample sets a reference to the `MenuGroup` object representing the ACAD menu group if it's loaded into the current session. It then displays a list of each of the menus and toolbars that make up this menu group. Here I used the `NameNoMnemonic` property of the menus to display their names rather than the `Name` property because I didn't want to show all the accelerator keys and hot keys.

```
Public Sub ListMenusAndToolbars()
Dim objMenuGroup As AcadMenuGroup
Dim objPopupMenu As AcadPopupMenu
Dim objToolBar As AcadToolbar
Dim strMenusAndToolbars As String

    On Error Resume Next
    Set objMenuGroup = ThisDrawing.Application.MenuGroups.Item("ACAD")
    If objMenuGroup Is Nothing Then
        MsgBox "ACAD menu group is not loaded"
        Exit Sub
    End If

    strMenusAndToolbars = _
              "The ACAD menu group comprises the following menus: " & vbCrLf
    For Each objPopupMenu In objMenuGroup.Menus
        strMenusAndToolbars = strMenusAndToolbars & _
                              objPopupMenu.NameNoMnemonic & ", "
    Next

    strMenusAndToolbars = strMenusAndToolbars & vbCrLf & vbCrLf & _
                          " and the following toolbars: " & vbCrLf

    For Each objToolBar In objMenuGroup.Toolbars
        strMenusAndToolbars = strMenusAndToolbars & objToolBar.Name & ", "
    Next
    MsgBox strMenusAndToolbars

End Sub
```

Figure 16-4 shows the result of this code. As you can see, the File, Edit, and View menus (as well as the Draw, Modify, and Dimension toolbars) belong to this group. Note that many of these menus and toolbars aren't normally visible when you use the AutoCAD interface.



```
The ACAD menu group comprises the following menus:


 and the following toolbars:
Dimension, Draw, Inquiry, Insert, Layouts, Modify, Modify II, Properties, Layers, Styles, Object Snap, 3D Orbit, Shade, Refedit, Reference, Render, Solids,
Solids Editing, Standard, CAD Standards, Surfaces, Text, UCS, UCS II, View, Viewports, Web, Zoom, Draw Order, Workspaces,
```

Figure 16-4. *Viewing the loaded menu group*

Saving Menu Groups

In versions up to AutoCAD 2005, you have two methods for saving a menu group: `Save` and `SaveAs`.

■ **Note** AutoCAD 2006 does not support saving menu groups.

As you would expect, `Save` saves the menu group to the file it's already associated with, and `SaveAs` allows you to save the menu group under a new name.

```
MenuGroupObject.Save MenuFileType

MenuGroupObject.SaveAs FileName, MenuFileType
```

Table 16-3 shows the `SaveAs` method's parameters.

Table 16-3. *The* `SaveAs` *Method's Parameters*

Name	Data Type	Description
`MenuFileType`	Long	Determines whether the menu group is saved as a source or a compiled file. It must take one of the `AcMenuFileType` constants in Table 16-4.
`FileName`	String	The full path and file name for the file to save to. The menu group then adopts this new name.

Table 16-4 presents the `AcMenuFileType` constants.

Table 16-4. *The* `AcMenuFileType` *Constants*

Constant	Value	Description
`acMenuFileCompiled`	0	A compiled menu file (`.mnc` extension)
`acMenuFileSource`	1	A source menu file (`.mns` extension)

You can use the following code to take a compiled menu and save it as an `.mns` file, ensuring that the user doesn't overwrite the ACAD menu group file:

```
Public Sub SaveMenuGroupAsSource()
Dim strMenuGroup As String
Dim strSaveto As String
Dim objMenuGroup As AcadMenuGroup

    strMenuGroup = InputBox("Enter the name of the menu group you" & _
                            " wish to save")
    If strMenuGroup = "" Then Exit Sub
```

```
On Error Resume Next
    Set objMenuGroup = ThisDrawing.Application.MenuGroups.Item(strMenuGroup)

    If objMenuGroup Is Nothing Then
        MsgBox "The menu group was not recognized"
        Exit Sub
    End If

    strSaveto = InputBox("Enter the filename to save" & strMenuGroup & " to")
    If UCase(strSaveto) = "ACAD" Then
        MsgBox "You should not overwrite the ACAD file"
        Exit Sub
    End If

    objMenuGroup.SaveAs strSaveto, acMenuFileSource

End Sub
```

Unloading Menu Groups

If you're sure that you no longer require a menu group, you can unload it from the AutoCAD session using the `Unload` method, which is the programmatic counterpart to using the `MENULOAD` and `MENUUNLOAD` commands.

```
MenuGroupObject.Unload
```

When you unload a menu group, any references you've set up through code to the menus and toolbars contained in that group become invalid. It's wise, therefore, to release all such references before you use this method.

Accelerator Keys

You can include accelerator keys in the name of a menu or menu item. You do this by placing an ampersand (&) before the character you want to be the accelerator key. For example, setting a menu name to &File causes it to appear as File and allows it to be quickly accessed by pressing Alt+F.

Changing the Menu Bar

As I pointed out earlier, AutoCAD 2000 and later allow you to completely replace the current menu bar with a new menu group by loading it as the base menu. The individual menus that the menu bar displays are contained in the `MenuBar` collection. You can add, rearrange, and remove menus within a menu group from the menu bar, and it's this modification of the menu bar that you'll consider next.

AutoCAD 2006 introduces the concept of *work spaces*, which are stored in the current `.cui` file. Work spaces contain the status of the various interface elements, including the layout of the toolbars and pull-down menus. AutoCAD 2006's default state after installation has no current work space, and it is important that one should be set before doing any menu customization. Also, any menu changes need to be saved in a work space, or they will be lost. AutoCAD 2006 provides no API for work spaces, so it is up to the user to do this using the `WORKSPACE` or `WSSAVE` command.

■**Note** Changes will affect the `.mns`, `.mnc`, and `.mnr` files, leaving the `.mnu` file unchanged as a crude kind of backup file. In AutoCAD 2006, the changes will affect the `.cui` file directly. Bugs in AutoCAD 2006 have been known to corrupt the `.cui` file, making it important to back up any `.cui` file before modifying it. Innocent changes your program makes to the menu might trigger the complete collapse of AutoCAD's menu system, rendering AutoCAD practically unusable. It is important to provide the user with an escape route from such a situation.

Adding Menus to the Menu Bar

You can add a new menu to the menu bar in two ways. The first uses the `InsertInMenuBar` method of the `PopupMenu` object to be inserted. When a menu not displayed on the AutoCAD menu bar is added to the menu bar, its `OnMenuBar` property changes from `False` to `True`.

```
PopupMenuObject.InsertInMenuBar(Index)
```

■**Note** AutoCAD 2006's default state has no current work space, so you should set one before customizing any menus. See the AutoCAD User's Guide for details about creating a work space. Work spaces cannot be created using VBA.

Table 16-5 shows the `InsertInMenuBar` method's parameters.

Table 16-5. *The* `InsertInMenuBar` *Method's Parameters*

Name	Data Type	Description
`Index`	Variant	The position within the menu bar where the pop-up menu will be added. It can be either an integer between 0 and N, where N is the number of objects in the menu bar, or a string giving the name of an existing menu (including the & accelerator key character) that the new item will be placed directly before. If the menu specified by the index doesn't exist, then the new menu is added at the end of the menu bar.

This example places a user-named menu at the beginning of the menu bar:

```
Public Sub InsertMenu1()
Dim objMenuGroup As AcadMenuGroup
Dim strMenuGroupNames As String
Dim strChosenMenuGroup As String
Dim objMenu As AcadPopupMenu
Dim strMenuNames As String
Dim strChosenMenu As String
```

```
    strMenuGroupNames = "Choose one of the loaded menu groups: " & vbCrLf
    For Each objMenuGroup In ThisDrawing.Application.MenuGroups
        strMenuGroupNames = strMenuGroupNames & objMenuGroup.Name & vbCrLf
    Next objMenuGroup

    strChosenMenuGroup = InputBox(strMenuGroupNames, "Choose a menu group")
    If strChosenMenuGroup = "" Then Exit Sub

    On Error Resume Next
    Set objMenuGroup = ThisDrawing.Application.MenuGroups.Item(strChosenMenuGroup)
    If objMenuGroup Is Nothing Then
        MsgBox "The menu group you chose does not exist"
        Exit Sub
    End If

    strMenuNames = "Choose a menu to add to the menu bar:" & vbCrLf
    For Each objMenu In objMenuGroup.Menus
        strMenuNames = strMenuNames & objMenu.Name & vbCrLf
    Next objMenu

    strChosenMenu = InputBox(strMenuNames, "Choose a menu")
    If strChosenMenu = "" Then Exit Sub

    ' check if menu is already on menu bar
    Set objMenu = ThisDrawing.Application.MenuBar.Item(strChosenMenu)
    If Not objMenu Is Nothing Then
        MsgBox "This menu is already present on the menu bar"
        Exit Sub
    End If
    Set objMenu = objMenuGroup.Menus.Item(strChosenMenu)
    If objMenu Is Nothing Then
        MsgBox "The menu you chose does not exist"
        Exit Sub
    End If


    objMenu.InsertInMenuBar 0

End Sub
```

This example places a user-named menu just before an existing menu, also chosen by the user:

```
Public Sub InsertMenu2()
Dim objMenuGroup As AcadMenuGroup
Dim strMenuGroupNames As String
Dim strChosenMenuGroup As String
Dim objMenu As AcadPopupMenu
Dim strMenuNames As String
Dim strChosenMenu As String
Dim strMenuPosition As String
```

```
    strMenuGroupNames = "Choose one of the loaded menu groups: " & vbCrLf
    For Each objMenuGroup In ThisDrawing.Application.MenuGroups
        strMenuGroupNames = strMenuGroupNames & objMenuGroup.Name & vbCrLf
    Next

    strChosenMenuGroup = InputBox(strMenuGroupNames, "Choose a menu group")
    If strChosenMenuGroup = "" Then Exit Sub

    On Error Resume Next
    Set objMenuGroup = ThisDrawing.Application.MenuGroups.Item(strChosenMenuGroup)
    If objMenuGroup Is Nothing Then
        MsgBox "The menu group you chose does not exist"
        Exit Sub
    End If

    strMenuNames = "Choose a menu to add to the menu bar:" & vbCrLf
    For Each objMenu In objMenuGroup.Menus
        strMenuNames = strMenuNames & objMenu.Name & vbCrLf
    Next

    strChosenMenu = InputBox(strMenuNames, "Choose a menu")
    If strChosenMenu = "" Then Exit Sub

    Set objMenu = objMenuGroup.Menus.Item(strChosenMenu)
    If objMenu Is Nothing Then
        MsgBox "The menu you chose does not exist"
        Exit Sub
    End If

    strMenuPosition = InputBox("Enter name of menu you wish to place it" & _
                               " before (including accelerator keys)")

    objMenu.InsertInMenuBar strMenuPosition

End Sub
```

Figure 16-5 shows the result of adding the Context Menu for Viewport Object menu of the Viewport menu group before the Draw menu of the ACAD menu group. Note that if the existing menu on the menu bar wasn't recognized, then the new menu will be placed at the end of the menu bar.

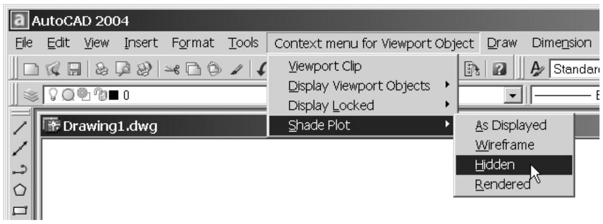


Figure 16-5. *Setting menu indices*

AutoCAD 2006 does this automatically, defaulting to placing all menus on the end. If this is not the desired location, you need to remove them and place them in the right place. AutoCAD 2005 and earlier do not automatically insert the menus.

The second way to add a menu to the menu bar employs the `InsertMenuInMenuBar` method of a `PopupMenus` collection.

```
PopupMenusCollection.InsertMenuInMenuBar MenuName, Index
```

Table 16-6 shows the `InsertMenuInMenuBar` method's parameters.

Table 16-6. *The* `InsertMenuInMenuBar` *Method's Parameters*

Name	**Data Type**	**Description**
`MenuName`	String	The identifying name of the pop-up menu to be added to the menu bar.
`Index`	Variant	The position within the menu bar where the pop-up menu will be added. It can be either an integer between 0 and N, where N is the number of objects in the menu bar, or a string giving the name of an existing menu (including the & accelerator key character) that the new item will be placed directly before. If the menu specified by the index doesn't exist, then the new menu is added at the end of the menu bar.

The following example again adds a user-chosen menu to the menu bar. If the menu that the user has chosen isn't recognized, no error is raised; it's simply not added to the menu bar.

```
Public Sub InsertMenu3()
Dim objMenuGroup As AcadMenuGroup
Dim strMenuGroupNames As String
Dim strChosenMenuGroup As String
Dim objMenu As AcadPopupMenu
Dim strMenuNames As String
Dim strChosenMenu As String
Dim strMenuPosition As String

    strMenuGroupNames = "Choose one of the loaded menu groups: " & vbCrLf
    For Each objMenuGroup In ThisDrawing.Application.MenuGroups
        strMenuGroupNames = strMenuGroupNames & objMenuGroup.Name & vbCrLf
    Next

    strChosenMenuGroup = InputBox(strMenuGroupNames, "Choose a menu group")
    If strChosenMenuGroup = "" Then Exit Sub

    On Error Resume Next
    Set objMenuGroup = ThisDrawing.Application.MenuGroups.Item(strChosenMenuGroup)
    If objMenuGroup Is Nothing Then
        MsgBox "The menu group you chose does not exist"
        Exit Sub
    End If
    strMenuNames = "Choose a menu to add to the menu bar:" & vbCrLf
    For Each objMenu In objMenuGroup.Menus
        strMenuNames = strMenuNames & objMenu.Name & vbCrLf
    Next

    strChosenMenu = InputBox(strMenuNames, "Choose a menu")
    If strChosenMenu = "" Then Exit Sub

    strMenuPosition = InputBox("Enter name of menu you wish to place it" & _
                               " before (including accelerator keys)")

    objMenuGroup.Menus.InsertMenuInMenuBar strChosenMenu, strMenuPosition
End Sub
```

You can use the specified position and order in which menus are added to the menu bar to arrange/rearrange the menus that make up your menu bar.

Removing Menus

Just as with adding menus to the menu bar, you have two ways to remove them: the `RemoveFromMenuBar` method of a `PopupMenu` object and the `RemoveMenuFromMenuBar` method of a `PopupMenus` collection. The first method takes the following simple syntax:

```
PopupMenuObject.RemoveFromMenuBar
```

This code iterates through the menus on the menu bar and removes those that the user specifies:

```
Sub RemoveMenus()
Dim objMenu As AcadPopupMenu

    For Each objMenu In ThisDrawing.Application.MenuBar
        If MsgBox("Remove " & objMenu.Name & "?", vbYesNo) = vbYes Then
            objMenu.RemoveFromMenuBar
        end if
    Next

End Sub
```

When you use the second method, you need to specify which menu you want to remove.

```
PopupMenusCollection.RemoveMenuFromMenuBar Index
```

Table 16-7 shows the `RemoveMenuFromMenuBar` method's parameters.

Table 16-7. *The* `RemoveMenuFromMenuBar` *Method's Parameters*

Name	Data Type	Description
`Index`	Variant	The position within the menu bar of the menu to be removed. It can be either an integer between 0 and N-1, where N is the number of objects in the menu bar, or a string giving the name of the menu (including the & accelerator key character). If the menu specified by the index doesn't exist, then this method does nothing.

To allow the user to choose a menu to remove, you use this code:

```
Public Sub RemoveMenu()
Dim objMenuGroup As AcadMenuGroup
Dim strMenuGroupNames As String
Dim strChosenMenuGroup As String
Dim strOnMenuBar As String
Dim objMenu As AcadPopupMenu
Dim strChosenMenu As String

    strMenuGroupNames = "Choose one of the loaded menu groups: " & vbCrLf
    For Each objMenuGroup In ThisDrawing.Application.MenuGroups
        strMenuGroupNames = strMenuGroupNames & objMenuGroup.Name & vbCrLf
    Next

    strChosenMenuGroup = InputBox(strMenuGroupNames, "Choose a menu group")
    If strChosenMenuGroup = "" Then Exit Sub

    On Error Resume Next
    Set objMenuGroup = ThisDrawing.Application.MenuGroups.Item(strChosenMenuGroup)
    If objMenuGroup Is Nothing Then
        MsgBox "The menu group you chose does not exist"
        Exit Sub
    End If

    strOnMenuBar = "the following menus with the " & objMenuGroup.Name & _
                   " menu group are displayed in the menu bar." & vbCrLf
    For Each objMenu In objMenuGroup.Menus
        If objMenu.OnMenuBar = True Then
            strOnMenuBar = strOnMenuBar & objMenu.Name & vbCrLf
        end if
    Next
    strOnMenuBar = strOnMenuBar & "Please choose a menu to remove"

    strChosenMenu = InputBox(strOnMenuBar)

    objMenuGroup.Menus.RemoveMenuFromMenuBar strChosenMenu

End Sub
```

Note Removed menus are still available in their designated menu group even though they're no longer visible to the user.

Editing Menus

Programmatically, you can create and customize two types of menus: pull-down and shortcut menus. You access pull-down menus via the AutoCAD menu bar, and you access shortcut menus by pressing Shift and right-clicking. Both types of menus are displayed as cascading-style menus.

Pull-down menu structures are limited to 999 items, and shortcut menu structures are limited to 499 items. These limits include everything in the menu structure. AutoCAD will ignore any menu items beyond these limits. Figure 16-6 reveals how the numbering of menu items works. You can see that the separators count as menu items.

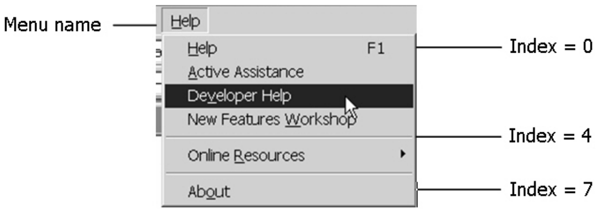


Figure 16-6. *Figuring out the numbering of a menu bar*

If the space available on the graphics screen is smaller than that required to display the menu structure, the menu will be truncated to fit within the available space. In addition, arrows will be placed at the top and bottom to allow navigation of the menu.

Creating New Menus

You can add a new menu, represented by a `PopupMenu` object, to a `PopupMenus` collection using the `Add` method.

```
Set PopupMenuObject = PopupMenusCollection.Add(MenuName)
```

Table 16-8 shows the `Add` method's parameter.

Table 16-8. *The* `Add` *Method's Parameter*

Name	Data Type	Description
`MenuName`	String	The identifying name of the newly created `PopupMenu` object

The `MenuName` parameter may be a simple string such as `VBATestMenu` or one containing special characters such as an ampersand (&) placed before a character specifying it as the accelerator key. By setting the menu name to `&VBATestMenu`, it will appear as VBATestMenu in the menu bar; you can view it by pressing Alt+V. For a complete list of special codes, please see the AutoCAD Customization Guide.

The following example illustrates how to create a new Apress menu and how to define a shortcut for the *r* character within it (see Figure 16-7):

```
Public Sub CreateApressMenu1()
Dim objMenus As AcadPopupMenus
Dim objMyMenu As AcadPopupMenu
Dim strNewMenuName As String
Dim objMyMenuItem As AcadPopupMenuItem

    Set objMenus = ThisDrawing.Application.MenuGroups.Item(0).Menus

    On Error Resume Next
    Set objMyMenu = objMenus.Item("Ap&ress")
    If Not objMyMenu Is Nothing Then
        MsgBox "Menu already exists"
        Exit Sub
    End If

    Set objMyMenu = objMenus.Add("Ap&ress")

    objMyMenu.InsertInMenuBar ThisDrawing.Application.MenuBar.Count
    Set objMyMenuItem = objMyMenu.AddMenuItem(0, "Add &Circle", _
        "-vbarun AddCircle ")

End Sub
```

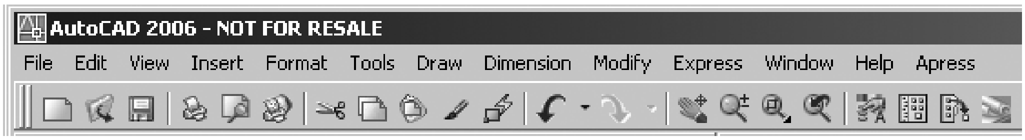

AutoCAD 2006 - NOT FOR RESALE
File Edit View Insert Format Tools Draw Dimension Modify Express Window Help Apress

Figure 16-7. *Creating a custom menu*

Note that the Apress menu has an accelerator key of *r*. The accelerator keys in PopupMenu names are no longer underlined by default. To see the underline, users must hold the Alt key. Users can select the Apress menu by pressing Alt+R. In this example, I also added an item to the menu to avoid problems caused by adding empty menus.

Next you'll look at some specifics of how to add items to a menu.

Adding New Menu Items

You can add a new item, represented by a PopupMenuItem object, to a menu through the AddMenuItem method of the PopupMenu object.

```
Set PopupMenuItemObject = PopupMenuObject.AddMenuItem(Index, Label, Macro)
```

Table 16-9 shows the `AddMenuItem` method's parameters.

Table 16-9. *The* `AddMenuItem` *Method's Parameters*

Name	Data Type	Description
`Index`	Variant	The position within the menu where the pop-up menu item will be added. It can be either an integer between 0 and N, where N is the number of objects in the menu, or a string giving the name of an existing menu item (including the & accelerator key character) that the new item will be placed directly before. If the menu item specified by the index doesn't exist, then the new menu item is added at the end of the menu.
`Label`	String	A label for the menu item. The label may contain `DIESEL` string expressions, which conditionally change the label each time it's displayed. Labels also identify the accelerator keys.
`Macro`	String	A string that will be written to the AutoCAD command prompt when the menu item is clicked.

You can't change the `Index` value of the menu item once it has been created through the `Index` property, as it's read-only. To rearrange menu items, you must first delete a menu item and add it again at the position you want it within the menu structure.

The following example code adds a menu item to the Apress menu. I've included a simple macro to demonstrate how the `Macro` parameter works. In this case, you can call a macro that you've written yourself by using the `-VBARUN` command described in Chapter 1. Note that you need to leave a space at the end of the macro name so that AutoCAD knows the end of the command has been reached.

```
Public Sub CreateApressMenu2()
Dim objMenus As AcadPopupMenus
Dim objMyMenu As AcadPopupMenu
Dim strNewMenuName As String
Dim objMyMenuItem As AcadPopupMenuItem

    Set objMenus = ThisDrawing.Application.MenuGroups.Item(0).Menus

    On Error Resume Next
    Set objMyMenu = objMenus.Item("Ap&ress")
    If objMyMenu Is Nothing Then
        Set objMyMenu = objMenus.Add("Ap&ress")
    End If

    'Check if the Menu is displayed on the MenuBar
    If Not objMyMenu.OnMenuBar Then
    'Display the menu on the menu bar
        objMyMenu.InsertInMenuBar ThisDrawing.Application.MenuBar.Count
    End If
    Set objMyMenuItem = objMyMenu.AddMenuItem(0, "Add &Circle", _
        "-vbarun AddCircle ")
    objMyMenuItem.HelpString = "This adds a circle at the origin"
```

```
End Sub
Public Sub AddCircle()
Dim dblCenter(2) As Double
  If ThisDrawing.ActiveSpace = acModelSpace Then
     ThisDrawing.ModelSpace.AddCircle dblCenter, 2
  Else
     ThisDrawing.PaperSpace.AddCircle dblCenter, 2
  End If
End Sub
```

You can see from the preceding code that it's possible to set the `HelpString` property of a `PopupMenuIem` object so that text, which provides information about what the menu item does, appears in the AutoCAD status line. If you hover your mouse over the newly created menu item, you'll see the phrase "This adds a circle at the origin" in the AutoCAD status line. This property is also exposed by the `Toolbar` and `ToolbarItem` objects.

Adding Separators

You can use the `AddSeparator` method to add a separator to your menu structure or to a toolbar. This method takes a single parameter and returns either a `PopupMenuItem` object or a `ToolbarItem` object. Here's the syntax for adding a separator to a menu:

```
Set PopupMenuItemObject = PopupMenuObject.AddSeparator(Index)
```

Table 16-10 shows the `AddSeparator` method's parameter.

■ **Note** You can't have adjacent separators or a separator at the beginning of a menu or toolbar.

Table 16-10. *The* `AddSeparator` *Method's Parameter*

Name	Data Type	Description
`Index`	Variant	The position within the menu where the separator will be added. It can be either an integer between 0 and N, where N is the number of objects in the menu, or a string giving the name of an existing menu item (including the & accelerator key character) that the separator will be placed directly before. If the menu item specified by the index doesn't exist, then the separator is placed at the end of the menu.

Creating Cascading Submenus

You can create cascading submenus by adding a submenu to an existing menu via the `AddSubMenu` method. This new cascading menu is blank, and you can modify it in precisely the same way as any other `PopupMenu` object.

```
Set PopupMenuObject = PopupMenuObject.AddSubMenu(Index, Label)
```

Table 16-11 shows the `AddSubMenu` method's parameters.

Table 16-11. *The* AddSubMenu *Method's Parameters*

Name	Data Type	Description
Index	Variant	The position within the menu where the submenu will be added. It can be either an integer between 0 and N, where N is the number of objects in the menu, or a string giving the name of an existing menu item (including the & accelerator key character) that the submenu will be placed directly before. If the menu item specified by the index doesn't exist, then the submenu is placed at the end of the menu.
Label	String	A label for the menu item. The label may contain DIESEL string expressions, which conditionally change the label each time it's displayed. Labels also identify the accelerator keys.

The following example illustrates how to create a complete cascading menu structure:

```
Public Sub CreateSubMenus()
Dim objMenus As AcadPopupMenus
'main menu definition
Dim objMenuCreates As AcadPopupMenu
'submenu definitions
Dim objMenuLine As AcadPopupMenuItem
Dim objMenuRectangle As AcadPopupMenuItem
Dim objSubMenuCircle As AcadPopupMenu
Dim objSubMenuCenterRadius As AcadPopupMenuItem
Dim objSubMenuCenterDiameter As AcadPopupMenuItem
Dim objSubMenuText As AcadPopupMenu
Dim objSubMenuMulti As AcadPopupMenuItem
Dim objSubMenuSingle As AcadPopupMenuItem
Dim strEscEsc As String

    Set objMenus = ThisDrawing.Application.MenuGroups.Item(0).Menus

    'Create the new main menu
    On Error Resume Next
    Set objMenuCreates = objMenus.Item("Creates")
    If objMenuCreates Is Nothing Then
      Set objMenuCreates = objMenus.Add("Creates")
    end if
    strEscEsc = Chr(27) & Chr(27) & Chr(95)
    'Add a menu items to the new main menu
    If objMenuCreates("Rectangle") Is Nothing Then
       Set objMenuRectangle = _
        objMenuCreates.AddMenuItem(objMenuCreates.Count, "Rectangle", _
                strEscEsc & "rectang ")
    end if
    If objMenuCreates("Line") Is Nothing Then
       Set objMenuLine = objMenuCreates.AddMenuItem(objMenuCreates.Count, _
                "Line", strEscEsc & "line ")
    End if
    'create the circle submenu
    If objMenuCreates("Circle") Is Nothing Then
     Set objSubMenuCircle = objMenuCreates.AddSubMenu(objMenuCreates.Count, _
               "Circle")
       Set objSubMenuCenterRadius = objSubMenuCircle.AddMenuItem( _
        objSubMenuCircle.Count, "Center, Radius", strEscEsc & "circle ")
       Set objSubMenuCenterDiameter = objSubMenuCircle.AddMenuItem( _
            objSubMenuCircle.Count, "Center, Diameter", _
                             strEscEsc & "circle \d ")
    End If

    'create the text submenu
    If objMenuCreates("Text") Is Nothing Then
       Set objSubMenuText = objMenuCreates.AddSubMenu(objMenuCreates.Count, _
               "Text")
       Set objSubMenuMulti = objSubMenuText.AddMenuItem( _
        objSubMenuText.Count, "MultiLine Text", strEscEsc & "mtext ")
       Set objSubMenuSingle = objSubMenuText.AddMenuItem( _
        objSubMenuText.Count, "SingleLine Text", strEscEsc & "dtext ")
    End If

    'Display the menu on the menu bar
    If objMenuCreates.OnMenuBar = False Then
      objMenuCreates.InsertInMenuBar (ThisDrawing.Application.MenuBar.Count)
    End if
End Sub
```

Figure 16-8 shows the result of this code.

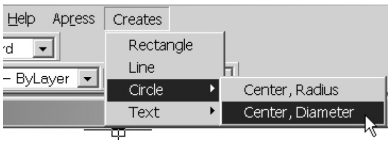


Figure 16-8. *Creating a cascading menu*

Deleting Menu Items

To delete a menu item from a `PopupMenu` object, you simply use the `Delete` method:

```
PopupMenuItemObject.Delete
```

You can use the following macro to delete menus on the menu bar. The user specifies which menu to delete. The code then iterates through the menu items in that menu and asks the user if he or she wants to delete each item.

```
Public Sub DeleteMenu()
Dim objMenu As AcadPopupMenu
Dim objMenuItem As AcadPopupMenuItem
Dim strDeleteMenuName As String
Dim strMenuName As String

    strDeleteMenuName = InputBox("Enter name of Menu to remove: ")
    If strDeleteMenuName = "" Then Exit Sub

On Error Resume Next
    Set objMenu = ThisDrawing.Application.MenuBar.Item(strDeleteMenuName)
    If objMenu Is Nothing Then
        MsgBox "Menu is not on the menu bar"
        Exit Sub
    End If

    For Each objMenuItem In objMenu
        strMenuName = objMenuItem.Caption
        If strMenuName = "" Then strMenuName = "Separator"

        If MsgBox("Delete " & strMenuName, vbYesNo, _
                  "Confirm deletion of a MenuItem") = _
           vbYes Then
            objMenuItem.Delete
        End If
    Next

End Sub
```

Adding Menu Items to the Shortcut Menu

The shortcut menu accessed by pressing Shift and right-clicking is a menu within the AutoCAD base menu group. The Boolean `ShortcutMenu` property of a `PopupMenu` object is set to `True` if the menu is a shortcut menu. You can add menu items to the shortcut menu in precisely the same way as you do to a drop-down menu, as I discussed earlier in this chapter. The following code sample adds a Zoom Extents menu item to the shortcut menu:

```
Public Sub AddMenuItemToShortcutMenu()
Dim objMenuGroup As AcadMenuGroup
Dim objShortcutMenu As AcadPopupMenu
Dim objMenuItem As AcadPopupMenu

    'only need the ACAD menugroup
    Set objMenuGroup = ThisDrawing.Application.MenuGroups.Item("ACAD")
    'find the shortcut menu
    For Each objMenuItem In objMenuGroup.Menus
        If objMenuItem.ShortcutMenu = True Then
            Set objShortcutMenu = objMenuItem
        End If
    Next


    'Add menu item to the shortcut menu
    objShortcutMenu.AddMenuItem "", "&Zoom &Extents", "_zoom e "
End Sub
```

Editing Toolbars

All the toolbars that belong to a `MenuGroup` object loaded in the current session of AutoCAD are contained within its `Toolbars` collection. These toolbars may or may not be visible in the AutoCAD interface. The code in "The MenuGroup Object" section shows how to display a list of all toolbars that are loaded. In the following sections, you'll learn how to create, edit, and delete AutoCAD toolbars.

Creating New Toolbars

You can create and add a new `Toolbar` object to the `Toolbars` collection through the `Add` method, which returns a reference to the new `Toolbar` object.

```
Set ToolbarObject = ToolbarsCollection.Add(ToolbarName)
```

Table 16-12 shows the `Add` method's parameter.

Table 16-12. *The* `Add` *Method's Parameter*

Name	Data Type	Description
`ToolbarName`	String	The identifying name of the newly created `Toolbar` object

The following example illustrates how to create a new toolbar in the AutoCAD collection. A check is in place to ensure that a toolbar of the same name doesn't already exist, as this would cause an error to be raised.

```
Public Sub AddNewToolbar()
Dim strTBName As String
Dim objToolbar As AcadToolbar

    strTBName = InputBox("Please enter name for new toolbar")
    If strTBName = "" Then Exit Sub
```

```
On Error Resume Next
    Set objToolbar = ThisDrawing.Application.MenuGroups.Item("ACAD"). _
                     Toolbars.Item(strTBName)
    If Not objToolbar Is Nothing Then
        MsgBox "This toolbar already exists"
    Else
        ThisDrawing.Application.MenuGroups.Item("ACAD").Toolbars.Add strTBName
    End If

End Sub
```

As you can see from Figure 16-9, the newly created toolbar, which becomes visible after creation, will be empty, so the next thing you'll want to do is add some toolbar buttons.

Figure 16-9. *Creating a new toolbar*

Adding Toolbar Buttons

A toolbar button is represented by a `ToolbarItem` object. You can use the `AddToolbarButton` method to add a new toolbar item at a specified position within a toolbar.

```
Set ToolbarItemObject = ToolbarObject.AddToolbarButton(Index, ButtonName, _
HelpString, Macro[, FlyoutButton])
```

Table 16-13 shows the `AddToolbarButton` method's parameters.

Table 16-13. *The* `AddToolbarButton` *Method's Parameters*

Name	Data Type	Description
`Index`	Variant	The position within the toolbar where the toolbar button will be added. It can be either an integer between 0 and N, where N is the number of objects in the menu, or a string giving the name of an existing toolbar button that the button will be placed directly before. If the button specified by the index doesn't exist, then the new button is placed at the end of the toolbar.
`ButtonName`	String	An identifying name for the new toolbar button. It's displayed when the user hovers over the button. It must consist of only alpha-numeric characters, hyphens (`-`), and underscores (`_`).
`HelpString`	String	A string that appears in the AutoCAD status line when a user high-lights a toolbar item for selection.
`Macro`	String	A string that is written to the AutoCAD command prompt when the menu item is clicked.
`FlyoutButton`	Boolean	An optional parameter that determines whether the new button will be a flyout button. You use a flyout button to nest a set of but-tons under a single toolbar button. Set this to `True` to create a flyout button. The default is `False`.

Once you've created the toolbar button, you can't change the `Index` value of the toolbar button using the `Index` property. To do this, you must first delete the toolbar button and then add it again in the position within the toolbar structure that you require.

The following example illustrates how to use the AddToolbarButton method by creating a toolbar with the same functionality as the cascading menu you created earlier in the chapter:

```
Public Sub CreateNewToolbars()
Dim objMenuGroup As AcadMenuGroup
Dim objToolbars As AcadToolbars
Dim objToolbar0 As AcadToolbar
Dim objToolbar1 As AcadToolbar
Dim objToolbar2 As AcadToolbar
Dim objToolBarCircle As AcadToolbarItem
Dim objToolBarText As AcadToolbarItem
Dim strEscEsc As String

  Set objMenuGroup = ThisDrawing.Application.MenuGroups.Item(0)
  Set objToolbars = objMenuGroup.Toolbars

  'Create three new toolbars
On Error Resume Next
  Set objToolbar0 = objToolbars.Item("Custom Toolbar Create")
  If Not objToolbar0 Is Nothing Then
      MsgBox "The 'Custom Toolbar Create’ already exists"
      Exit Sub
  End If

  Set objToolbar0 = objToolbars.Add("Custom Toolbar Create")
  Set objToolbar1 = objToolbars.Add("Circle")
  Set objToolbar2 = objToolbars.Add("Text")

  'Add a toolbar item to the new toolbar
  objToolbar0.AddToolbarButton objToolbar0.Count, "Line", _
       "Draw a line", strEscEsc & "line "
  objToolbar0.AddToolbarButton objToolbar0.Count, "Rectangle", _
      "Draw a rectangle", strEscEsc & "rectang "

  'create the circle flyout toolbar button
  Set objToolBarCircle = objToolbar0.AddToolbarButton(objToolbar0.Count, _
      "Circle", "Draw a circle", "Circle", True)
  'create the circle flyout toolbar
  objToolbar1.AddToolbarButton objToolbar1.Count, "Center, Radius", _
      "Draw circle using Center point and radius", strEscEsc & "circle "
  objToolbar1.AddToolbarButton objToolbar1.Count, "Center, Diameter", _
      "Draw circle using Center point and diameter", strEscEsc & "circle \d "
  'create the text submenu
  Set objToolBarText = objToolbar0.AddToolbarButton(objToolbar0.Count, _
      "Text", "Create text strings", "Text", True)
  'create the text flyout toolbar
  objToolbar2.AddToolbarButton objToolbar2.Count, "MultiLine Text", _
      "Create multi-text", strEscEsc & "mtext "
  objToolbar2.AddToolbarButton objToolbar2.Count, "SingleLine Text", _
      "Create single-line text", strEscEsc & "dtext "

  objToolBarCircle.AttachToolbarToFlyout objMenuGroup.Name, objToolbar1.Name
  objToolBarText.AttachToolbarToFlyout objMenuGroup.Name, objToolbar2.Name

  'SetIcons "C:\Apress\smLineIcon.bmp", "C:\Apress\lgLineIcon.bmp"

  objToolbar0.Visible = True
  objToolbar1.Visible = False
  objToolbar2.Visible = False
End Sub
```

In Figure 16-10 you can see the newly created toolbar. Note the ButtonName and HelpString text displayed as the tooltip and in the status line of AutoCAD. In addition, you can see that flyout buttons have a small black triangle in the lower-right corner.

The default icon when you add a toolbar button through code is the ballooned question mark. You'll now look at how to set the icons to images of your choosing.

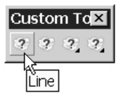


Figure 16-10. *Creating a toolbar with buttons*

Defining the Toolbar Button Image

To set and retrieve the bitmaps of the icons associated with a toolbar button, you can use the SetBitmaps and GetBitmaps methods. Both methods have similar syntax and require two parameters (see Table 16-14), one for each of the two icons (small and large).

```
ToolbarItemObject.SetBitmaps SmallIconName, LargeIconName

ToolbarItemobject.GetBitmaps SmallIconName, LargeIconName
```

Table 16-14. *The* SetBitmaps *and* GetBitmaps *Methods' Parameters*

Name	Data Type	Description
SmallIconName	String	The path and file name for the small bitmap (16×15 pixels)
LargeIconName	String	The path and file name for the small bitmap (24×22 pixels)

The following code example sets the small and large icons for the Line button of the toolbar you just created with the previous code. Note that here I'm using bitmaps I created and stored in the C:\Apress\ folder, so, if you want to try this code, you'll need to uncomment the SetIcons line in the previous code and change the path and file information to suitable files on your machine.

```
Public Sub SetIcons(strIconSmall As String, strIconLarge As String)
Dim objToolBarItem As AcadToolbarItem
    Set objToolBarItem = ThisDrawing.Application.MenuGroups(0).Toolbars( _
                                            "Custom Toolbar Create")("Line")

    objToolBarItem.SetBitmaps strIconSmall, _
                              strIconLarge

End Sub
```

■ **Note** Autodesk has sought Microsoft Windows XP logo certification for the entire AutoCAD product line. One of the requirements of this logo certification involves toolbar icon graphics. If you intend to develop software for marketing, you should seriously investigate these requirements if you want to produce products that coincide with the base platform (AutoCAD) on which they're used. Refer to the MSDN Web site at `http://msdn.microsoft.com` for more information.

Adding Separators

You can use the `AddSeparator` method to add a separator to your menu structure or to a toolbar. This method takes a single parameter and returns either a `PopupMenuItem` object or a `ToolbarItem` object. Here's the syntax for adding a separator to a toolbar:

```
Set ToolbarItemObject = ToolbarObject.AddSeparator(Index)
```

Table 16-15 shows the `AddSeparator` method's parameter.

Table 16-15. *The* `AddSeparator` *Method's Parameter*

Name	Data Type	Description
`Index`	Variant	The position within the toolbar where the separator will be added. It can be either an integer between 0 and N, where N is the number of objects in the toolbar collection, or a string giving the name of an existing toolbar item (including the & accelerator key character) that the separator will be placed directly before. If the toolbar item specified by the index doesn't exist, then the separator is placed at the end of the menu.

■ **Note** You can't have adjacent separators or a separator at the beginning of a menu or toolbar.

Floating and Docking Toolbars

Toolbars may either float on-screen or be docked to an edge of the AutoCAD drawing window. You use the `Float` method to float a toolbar at a specified point on the screen. This point may lie outside the AutoCAD window.

```
ToolbarObject.Float Top, Left, NumberOfRows
```

Table 16-16 shows the `Float` method's parameters.

Table 16-16. *The* `Float` *Method's Parameters*

Name	Data Type	Description
`Top`	Long	The position of the top edge of the toolbar from the top of the screen given in pixels.
`Left`	Long	The position of the left side of the toolbar from the left edge of the screen given in pixels.
`NumberOfRows`	Long	The number of rows that the toolbar buttons should be distributed over. If this is set to more than the number of buttons on the toolbar, this parameter will be ignored.

This code sample shows how to use the `Float` method by floating the AutoCAD Draw toolbar to the top-left corner of the screen:

```
Public Sub FloatDrawToolbar()
Dim objToolBarDraw As AcadToolbar

     Set objToolBarDraw = _
     ThisDrawing.Application.MenuGroups.Item("ACAD").Toolbars.Item("Draw")

     objToolBarDraw.Float 0, 0, 3

End Sub
```

Figure 16-11 shows the floating toolbar.

Note that the toolbar lies outside the AutoCAD window and is split over three rows.

To dock a toolbar, use the `Dock` method.

```
ToolbarObject.Dock DockStatus
```

Table 16-17 shows the `Dock` method's parameter.

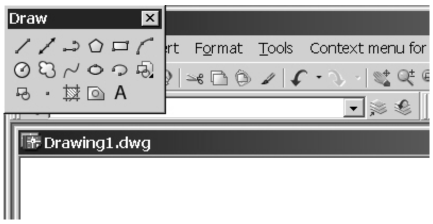


Figure 16-11. *Floating a toolbar*

Table 16-17. *The* `Dock` *Method's Parameter*

Name	Data Type	Description
`DockStatus`	Long	This specifies which edge of the window to dock to. It must be one of the `AcToolbarDockStatus` constants presented in Table 16-18.

Table 16-18 contains the AcToolbarDockStatus constants.

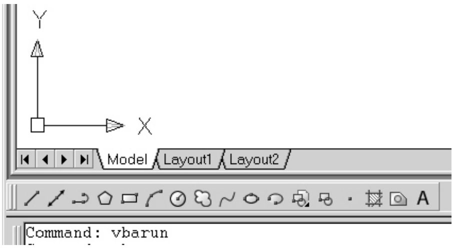
Table 16-18. *The* AcToolbarDockStatus *Constants*

Constant	Value	Description
acToolbarDockTop	0	The toolbar is docked to the top of the drawing window.
acToolbarDockBottom	1	The toolbar is docked to the bottom of the drawing window.
acToolbarDockLeft	2	The toolbar is docked to the left side of the drawing window.
acToolbarDockRight	3	The toolbar is docked to the right side of the drawing window.

This time the AutoCAD Draw toolbar is docked to the bottom of the drawing window:

```
Public Sub DockDrawToolbar()
Dim objToolBarDraw As AcadToolbar

    Set objToolBarDraw =  _
    ThisDrawing.Application.MenuGroups.Item("ACAD").Toolbars.Item("Draw")

    objToolBarDraw.Dock acToolbarDockBottom

End Sub
```

Figure 16-12 shows the docked toolbar.



```
Command: vbarun
```

Figure 16-12. *Docking a toolbar*

Deleting Toolbars and Toolbar Buttons

You can delete both toolbars and toolbar buttons with the Delete method.

```
ToolbarObject.Delete

ToolbarItemObject.Delete
```

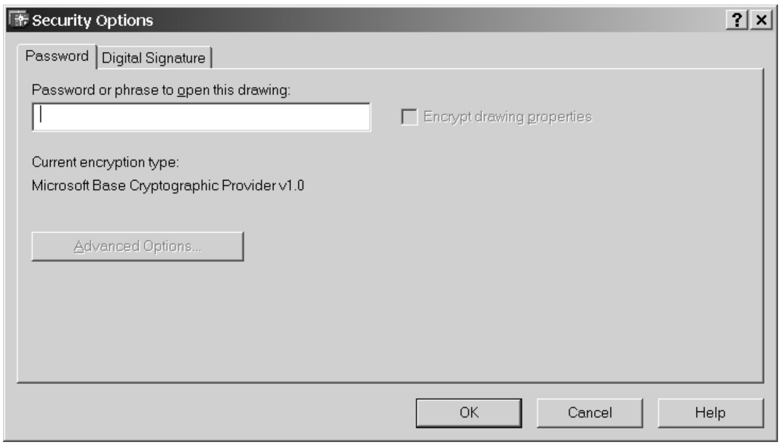
Summary

I covered a lot of material in this chapter. The part of the AutoCAD object model that represents the menu and toolbar structure is quite complicated. The information in this chapter should provide you with a sound basis for modifying menus and toolbars. For details of every property and method of the menu structure objects, please consult Appendix A.

CHAPTER 17

Drawing Security

AutoCAD 2006 has two features aimed at providing security to drawing files: Digital Signatures and Password Protection. You can access and manipulate both features from a command prompt, a GUI interface, or programmatically. The command interface is `SECURITYOPTIONS`, which displays the Security Options dialog box for configuring the Digital Signature and Password Protection features, as shown in Figure 17-1.



Current encryption type:
Microsoft Base Cryptographic Provider v1.0

Figure 17-1. *Digital Signature and Password Protection options*

Digital Signatures

To support the Digital Signatures feature from a programming standpoint, Autodesk added in 2004 a new object called `SecurityParams`. This object allows you to set various security options, including drawing and properties encryption, digital signatures, and timestamps. Several options are available that provide varying levels of security, as presented in this chapter. Although it isn't possible to provide extensive details about cryptography, further details are available on the MSDN Web site (`http://msdn.microsoft.com`).

To attach a digital signature to a document of any kind, you first need a digital ID from a certificate authority such as VeriSign. You can generate your own "self-signed" certificates, but these are intended for internal use, not for use by your customers. Your digital signature is valid as long as the drawing doesn't change. Anyone receiving your drawing can validate it to see that it really came from you. If your drawing has an invalid signature, the recipient can easily tell that the drawing was changed after you attached the digital signature.

The following example illustrates how to use the properties of the `SercurityParams` object to save and encrypt a drawing file:

```
Public Sub SaveWithEncryption()
Dim oSecurity As New AcadSecurityParams
  With oSecurity
    .Action = AcadSecurityParamsType.ACADSECURITYPARAMS_ENCRYPT_DATA
    .Algorithm = AcadSecurityParamsConstants.ACADSECURITYPARAMS_ALGID_RC4
    .Comment = "Add comment to drawing"
    .Issuer = "Your Company Name"
    .KeyLength = 40
    'AutoCAD converts all passwords to uppercase before applying them
    .Password = UCase("br549")
    .ProviderName = "Microsoft Base Cryptographic Provider v1.0"
    .ProviderType = 1
    .SerialNumber = "BR549"
    .Subject = "The Flying DeLorean"
    .TimeServer = "NIST(time.nist.gov)"
  End With

  ThisDrawing.SaveAs "C:\MyDrawing.dwg", , oSecurity
End Sub
```

The `Action` Property

The `Action` property is the heart of the `SecurityParams` object (and, of course, the `Password` property, which is pretty straightforward).

When specifying values for the `Action` property, you must specify one or more of the constants in Table 17-1 for drawing encryption, drawing properties encryption, a digital signature, or a timestamp.

Table 17-1. *The* `Action` *Property's Constants*

Constant	Value
ACADSECURITYPARAMS_ENCRYPT_DATA	0x00000001
ACADSECURITYPARAMS_ENCRYPT_PROPS	0x00000002
ACADSECURITYPARAMS_SIGN_DATA	0x00000010
ACADSECURITYPARAMS_ADD_TIMESTAMP	0x00000020

Each constant represents a security-related operation. When you use the `Action` property, at some point in the set of operations you must set the `Action` property to the `ACADSECURITYPARAMS_ENCRYPT_DATA` constant, to the `ACADSECURITYPARAMS_SIGN_DATA` constant, or to both.

To specify multiple security-related operations, add the constants representing the operations. The following example shows how to sign a drawing and use a timestamp:

```
ACADSECURITYPARAMS_SIGN_DATA + ACADSECURITYPARAMS_ADD_TIMESTAMP
```

Note You can specify only `AcadSecurityParamsConstants.ACADSECURITYPARAMS_ALGID_RC4` for the `Algorithm` property. Any other value will generate an error.

The `Algorithm` Property

The `Algorithm` property specifies the identifier of the encryption algorithm to be used in signing the drawing file.

```
object.Algorithm
```

Table 17-2 shows the `Algorithm` property's parameters.

Table 17-2. *The* `Algorithm` *Property's Parameters*

Name	Returns	Description
`object`	`SecurityParams`	The object or objects to which this property applies
`Algorithm`	Long; `AcadSecurityParams`➥ `Constants` enum; read-write	Specifies only the following constant: `ACADSECURITYPARAMS_ALGID_RC4_0x00006801`

Here's an example of how to set the `Algorithm` property:

```
Dim sp As New AcadSecurityParams
sp.Algorithm = AcadSecurityParamsConstants.ACADSECURITYPARAMS_ALGID_RC4
```

The `Issuer` Property

The `Issuer` property specifies the name of the digital certificate issuer. The `Put` option requires one parameter, `Issuer`, which is a string value.

```
object.Issuer
```

Table 17-3 shows the `Issuer` property's parameters.

Table 17-3. *The* `Issuer` *Property's Parameters*

Name	Returns	Description
`object`	`SecurityParams`	The object or objects to which this property applies
`Issuer`	String; read-write	The issuer name of the digital certificate

Here's an example of how to set the `Issuer` property:

```
Dim sp As New AcadSecurityParams
sp.Issuer = "Personal Freemail RSA 2000.8.30"
```

The `ProviderName` Property

This property specifies the name of the provider of the digital certificate.

```
object.ProviderName
```

Table 17-4 shows the `ProviderName` property's parameters.

Table 17-4. *The* `ProviderName` *Property's Parameters*

Name	Returns	Description
`object`	`SecurityParams`	The object or objects to which this property applies
`ProviderName`	String; read-write	The encryption provider name

More information about cryptography providers is available on MSDN.

The `SerialNumber` Property

The `SerialNumber` property specifies the serial number identifier for the digital certificate. It uses the following options:

```
object.SerialNumber
```

Table 17-5 shows the `SerialNumber` property's parameters.

Table 17-5. *The* `SerialNumber` *Property's Parameters*

Name	Returns	Description
`object`	`SecurityParams`	The object or objects to which this property applies
`SerialNumber`	String; read-write	The serial number of the digital certificate

Here's an example of how to set the `SerialNumber` property:

```
Dim sp As New AcadSecurityParams
sp.SerialNumber = "073848"
```

The `Subject` Property

This property specifies a subject string name value of the digital certificate.

```
object.Subject
```

Table 17-6 shows the `Subject` property's parameters.

Table 17-6. The `Subject` *Property's Parameters*

Name	Returns	Description
`object`	`SecurityParams`	The object or objects to which this property applies
`Subject`	String; read-write	The subject name of the digital certificate

The `TimeServer` Property

The `TimeServer` property specifies the name of the time server to be used for a digital signature. If you don't set this property, the time from the local machine is used as the timestamp for the digital signature.

```
object.TimeServer
```

Table 17-7 shows the `TimeServer` property's parameters.

Table 17-7. *The* `TimeServer` *Property's Parameters*

Name	Returns	Description
`object`	`SecurityParams`	The object or objects to which this property applies
`TimeServer`	String; read-write	The name of the time server to be used for a digital signature

Here's an example of how to set the `TimeServer` property:

```
Dim sp As New AcadSecurityParams
sp.TimeServer = "NIST(time.nist.gov)"
```

Password Protection

Probably one of the most controversial security features in AutoCAD 2006 is Password Protection (originally introduced in AutoCAD 2004). Basically, this feature allows a user to lock a drawing file with a password to prevent unwanted people from opening and viewing or modifying it. It's an all-or-nothing feature in that the drawing is either completely locked or completely unlocked; no intermediate locking capability exists (such as locking layers or permitting viewing but not modifying or printing, as is possible with products such as Adobe Acrobat).

This quotes Autodesk:

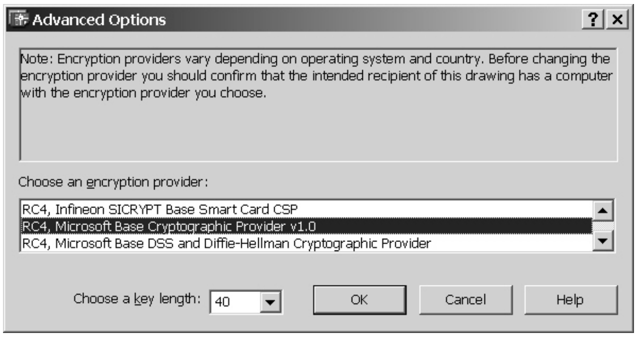
> *When you password-protect a drawing, you encrypt the drawing, altering it with a secret code. A password-protected or encrypted drawing can only be opened by someone who knows the correct password.*

Additionally, Autodesk says this:

Warning! If you lose the password, it is not recoverable. Before you add a password, you should create a backup that is not protected with a password.

You can disable Password Protection during installation, which is how most sites with large numbers of users handle this.

If you decide to employ this feature, you should know that it is indeed secure. In addition to setting a unique password, you can configure other options for the encryption method. From the GUI interface, you must enter a default password in the password entry box in order to access the Advanced Options button, which opens the Advanced Options dialog box shown in Figure 17-2. Quite a few options are available.


Note: Encryption providers vary depending on operating system and country. Before changing the encryption provider you should confirm that the intended recipient of this drawing has a computer with the encryption provider you choose.

Choose an encryption provider:
RC4, Infineon SICRYPT Base Smart Card CSP
RC4, Microsoft Base Cryptographic Provider v1.0
RC4, Microsoft Base DSS and Diffie-Hellman Cryptographic Provider

Choose a key length: 40

Figure 17-2. *The Advanced Options dialog box*

From a programmatic aspect, the same object model applies to password locking (aka encryption) as to digital certificates. In fact, it uses the same `SecurityParams` object to provide the interface for saving an encrypted drawing.

■ **Note** It's worth noting that AutoCAD converts the password string value to uppercase before storing it in the drawing. The result is a case-insensitive password entry match when you attempt to open the drawing.

The following example shows how to save an encrypted drawing file using this object:

```
Sub SaveLockedDrawing()
  ' This example encrypts and saves a file.
  Dim acadApp As New AcadApplication
  Dim sp As New AcadSecurityParams
  acadApp.Visible = True
  sp.Action = AcadSecurityParamsType.ACADSECURITYPARAMS_ENCRYPT_DATA
  sp.Algorithm =   AcadSecurityParamsConstants.ACADSECURITYPARAMS_ALGID_RC4
  sp.KeyLength = 40
  sp.Password = UCase("unique-password)
  sp.ProviderName = "Microsoft Base Cryptographic Provider v1.0"
  sp.ProviderType = 1
  acadApp.ActiveDocument.SaveAs "C:\LockedDrawing1.dwg", , sp
End Sub
```

Summary

This chapter covered the two most important security features provided in AutoCAD 2006: Digital Signatures and Password Protection. Though these features impose serious consideration of potential risks, they offer obvious security benefits to users who need such protection for their work.

CHAPTER 18

Using the Windows API

This chapter tells you how to take advantage of the Windows API. This isn't a definitive guide to the Windows API, but it gives you enough information to get you started, including common examples such as the `Open` and `SaveAs` common dialogs.

Declarations

You must declare every Windows API function that you use, *period.* Unlike in VBA, the Windows API includes no type library against which to resolve your function calls. So for them to be resolved—and you must resolve all function calls—you have to declare each one before you use it. The following code shows how to declare a function call:

```
Public Declare Function SetForegroundWindow Lib "user32.dll" _
  (ByVal hwnd As Long) As Long
```

The syntax breakdown is as follows:

```
[Scope] Declare Function <FunctionName> Lib <DLL Filename string> _
  (List of Parameters) As <DataType>
```

The easiest way to get the declaration is to use the API Viewer (which is available at msdn.microsoft.com) shown in Figure 18-1.

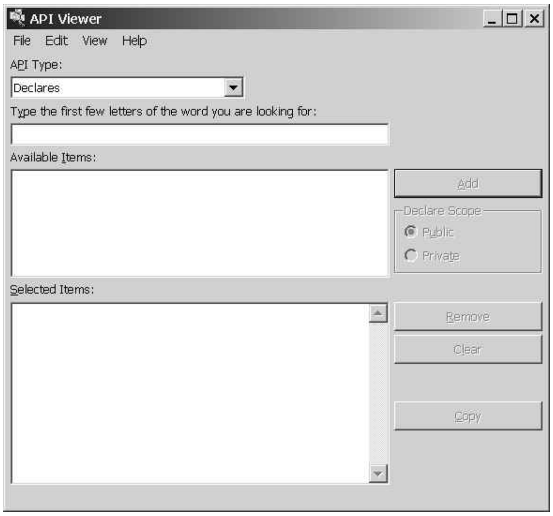


Figure 18-1. *The API Viewer dialog box*

Windows Data Structures

A data structure is a collection of related information that is accessible through a single variable name. Data structures are widely used in the C and, to some extent, the C++ languages, but *classes* have replaced them in modern programming practices. Classes let you define functions that work on the class's data, letting you hide variables and functionality from the outside world. On the other hand, data structures are nothing more than data structures. But you're in luck because the Windows API has a ton of data structures, as the following code demonstrates:

```
Public Type POINTS
  x As Integer
  y As Integer
End Type
```

Visual Basic-to-DLL Calling Conventions

To call DLL function procedures from Visual Basic, convert the C syntax that documents them into valid `Declare` statements. Visual Basic can call these procedures using the appropriate parameter data type declarations.

You must convert the C data types into Visual Basic data types. Also, specify whether the calling convention is `ByVal` (by value) or `ByRef` (by reference). Table 18-1 illustrates the conversions for 32-bit Windows C-language data types to Visual Basic.

Table 18-1. *C vs. Visual Basic Data Type Declarations*

C Language Data Type	Declare in Visual Basic As	Call With
`ATOM`	`ByVal` *variable* As Integer	An expression that evaluates to an Integer
`BOOL`	`ByVal` *variable* As Long	An expression that evaluates to a Long
`BYTE`	`ByVal` *variable* As Byte	An expression that evaluates to a Byte
`CHAR`	`ByVal` *variable* As Byte	An expression that evaluates to a Byte
`COLORREF`	`ByVal` *variable* As Long	An expression that evaluates to a Long
`DWORD`	`ByVal` *variable* As Long	An expression that evaluates to a Long
`HWND`, `HDC`, `HMENU`, etc. (Windows handles)	`ByVal` *variable* As Long	An expression that evaluates to a Long
`INT`, `UINT`	`ByVal` *variable* As Long	An expression that evaluates to a Long
`LONG`	`ByVal` *variable* As Long	An expression that evaluates to a Long
`LPARAM`	`ByVal` *variable* As Long	An expression that evaluates to a Long
`LPDWORD`	*variable* As Long	An expression that evaluates to a Long
`LPINT`, `LPUINT`	*variable* As Long	An expression that evaluates to a Long
`LPRECT`	*variable* As type	Any variable of that user-defined type
`LPSTR`, `LPCSTR`	`ByVal` *variable* As String	An expression that evaluates to a String

C Language Data Type	Declare in Visual Basic As	Call With
`LPVOID`	*variable* As Any	Any variable (use `ByVal` when passing a string)
`LPWORD`	*variable* As Integer	An expression that evaluates to an Integer
`LRESULT`	`ByVal` *variable* As Long	An expression that evaluates to a Long
`NULL`	As Any or `ByVal` *variable* As Long	`ByVal` Nothing or `ByVal` `0&` or `vbNullString`
`SHORT`	`ByVal` *variable* As Integer	An expression that evaluates to an Integer
`VOID`	`Sub` procedure	Not applicable
`WORD`	`ByVal` *variable* As Integer	An expression that evaluates to an Integer
`WPARAM`	`ByVal` *variable* As Long	An expression that evaluates to a Long

Specifying the Library

Visual Basic uses the `Declare` statement's `Lib` clause to find the `.dll` file that contains the procedures. When you declare a function that uses one of the core Windows API libraries, specify the file name extension `.dll`.

Tip For consistency, get into the habit of specifying this extension all the time, because you'll need to specify it for noncore API libraries that you use.

Here's how to use the `Lib` clause:

```
Public Declare Function SetForegroundWindow Lib "user32.dll" _
    (ByVal hwnd As Long) As Long
```

For noncore API libraries, you can specify a path in the `Lib` clause. If you don't specify a path, then Visual Basic searches for the file in the following order:

1. Directory containing the `.exe` file
2. Current directory
3. Windows system directory (usually C:\Windows\System)
4. Windows directory (usually C:\Windows)
5. Path environment variable

The Major Windows DLLs

Table 18-2 lists the most commonly used libraries of Windows API functions.

Table 18-2. *Common Windows API Libraries*

File Name	Description
`Advapi32.dll`	Advanced API services library supporting numerous APIs, including many security and registry calls
`Comdlg32.dll`	Common dialog API library
`Gdi32.dll`	Graphics Device Interface API library
`Kernel32.dll`	Core Windows 32-bit base API support
`Lz32.dll`	32-bit compression routines
`Mpr.dll`	Multiple Provider Router library
`Netapi32.dll`	32-bit Network API library
`Shell32.dll`	32-bit Shell API library
`User32.dll`	Library for user interface routines
`Version.dll`	Version library
`Winmm.dll`	Windows multimedia library
`Winspool.drv`	Print spooler interface that contains print-spooler API calls

Working with Windows API Procedures That Use Strings

The `Declare` statement's `Alias` clause is required when you call Windows API procedures that use strings to specify the correct character set. There are actually two formats for procedures that contain strings: ANSI and Unicode.

For example, the `SetWindowText` function does not really exist. Instead, you use two separate functions depending on whether you use ANSI or Unicode. The following code illustrates the ANSI version:

```
Private Declare Function SetWindowText Lib "user32" Alias _
   "SetWindowTextA" (ByVal hwnd As Long, ByVal lpString As String) _
   As Long
```

The string that follows the Alias clause must be the procedure's true, case-sensitive name.

Specify the ANSI version of functions in Visual Basic, because only Windows NT supports the Unicode versions. Use Unicode only when you're certain your application will run on Windows NT.

Passing Arguments by Value or by Reference

By default, Visual Basic passes arguments by reference. Instead of passing the actual value of the argument, it passes a 32-bit address specifying the value's location. The `ByRef` keyword is not required, but it's a good idea to specify the exact method of passing the argument to make your code more readable.

■**Note** This is no longer true with Visual Basic .NET or any of the .NET languages, which default to `ByValue` arguments.

Many DLL procedures expect you to pass an argument by value. The function expects to receive the actual value instead of its memory location. If you pass the argument to the function using `ByRef`, the function receives information that it has no idea how to handle.

To pass an argument by value, place the `ByVal` keyword in front of the `Declare` statement's argument declaration. The `InvertRect` procedure accepts its first argument by value and its second by reference, as the following example shows:

```
Declare Function InvertRect Lib "user32" Alias "InvertRectA" _
   (ByVal hdc As Long, lpRect As RECT) As Long
```

■**Note** When you look at DLL procedure documentation that uses C-language syntax, remember that C passes all arguments except arrays by value.

Learning by Example

The best way to learn the Windows API is to follow other programmers' examples and try different situations on your own. So this section gives you examples of using the Windows API. These examples illustrate some of the most common Windows API functions that AutoCAD developers use, and should give you enough information to pursue using Windows API functions in your own application development.

`OpenFile` Common Control Dialog Replacement for VBA

Using the `OpenFile` common control dialog adds a look of consistency to your application. The `OpenFile` dialog is part of the `comdlg32.dll` library of Windows API routines and is easily accessed. The following example illustrates using these routines to request a drawing file to open.

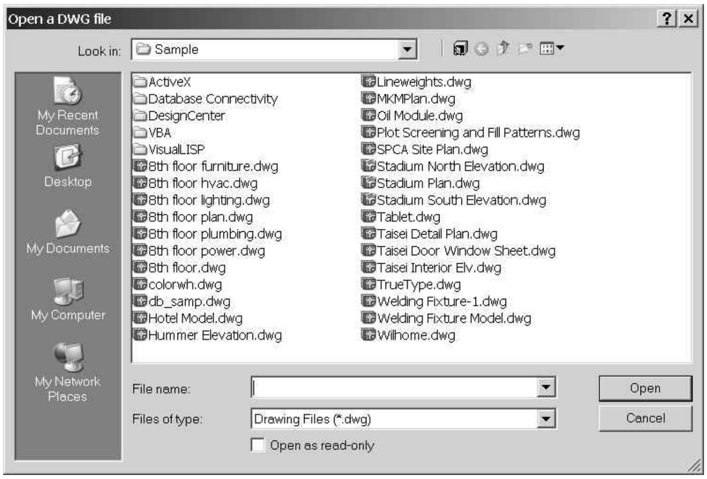
```
Private Declare Function GetOpenFileName Lib "comdlg32.dll" Alias _
   "GetOpenFileNameA" (pOpenfilename As OPENFILENAME) As Long
Private Type OPENFILENAME
  lStructSize As Long
  hwndOwner As Long
  hInstance As Long
  lpstrFilter As String
  lpstrCustomFilter As String
  nMaxCustFilter As Long
  nFilterIndex As Long
  lpstrFile As String
  nMaxFile As Long
  lpstrFileTitle As String
  nMaxFileTitle As Long
  lpstrInitialDir As String
  lpstrTitle As String
  flags As Long
  nFileOffset As Integer
  nFileExtension As Integer
  lpstrDefExt As String
  lCustData As Long
  lpfnHook As Long
  lpTemplateName As String
End Type

Public Function ShowOpen(Filter As String, _
                         InitialDir As String, _
                         DialogTitle As String) As String

Dim OFName As OPENFILENAME

  'Set the structure size
  OFName.lStructSize = Len(OFName)
  'Set the owner window
  OFName.hwndOwner = 0
  'Set the filter
  OFName.lpstrFilter = Filter
  'Set the maximum number of chars
  OFName.nMaxFile = 255
  'Create a buffer
  OFName.lpstrFile = Space(254)
  'Create a buffer
  OFName.lpstrFileTitle = Space$(254)
  'Set the maximum number of chars
  OFName.nMaxFileTitle = 255
  'Set the initial directory
  OFName.lpstrInitialDir = InitialDir
  'Set the dialog title
  OFName.lpstrTitle = DialogTitle
  'no extra flags
  OFName.flags = 0
  'Show the 'Open File' dialog
  If GetOpenFileName(OFName) Then
    ShowOpen = Trim(OFName.lpstrFile)

    Else
      ShowOpen = ""
  End If
End Function
```

The following sample code illustrates using the `ShowOpen` routine, which returns the selected file name as a string.

```
Dim Filter As String
Dim InitialDir As String
Dim DialogTitle As String
Dim OutputStr As String

  Filter = "Drawing Files (*.dwg)" + Chr$(0) + "*.dwg" + Chr$(0) + _
    "All Files (*.*)" + Chr$(0) + "*.*" + Chr$(0)
  InitialDir = "C:\Program Files\AutoCAD 2006\Sample"
  DialogTitle = "Open a DWG file"

  OutputStr = ShowOpen(Filter, InitialDir, DialogTitle)
  MsgBox OutputStr
```

The `Filter` parameter is a string that details which file types, by extension, to display when the `OpenFile` dialog box appears. The `InitialDir` parameter specifies which directory to display by default. To give a name to your `OpenFile` dialog box, use the `DialogTitle` parameter.

With each of these parameters defined, executing this code results in the `OpenFile` dialog box shown in Figure 18-2.



ActiveX
Database Connectivity
DesignCenter
VBA
VisualLISP
8th floor furniture.dwg
8th floor hvac.dwg
8th floor lighting.dwg
8th floor plan.dwg
8th floor plumbing.dwg
8th floor power.dwg
8th floor.dwg
colorwh.dwg
db_samp.dwg
Hotel Model.dwg
Hummer Elevation.dwg
Lineweights.dwg
MKMPlan.dwg
Oil Module.dwg
Plot Screening and Fill Patterns.dwg
SPCA Site Plan.dwg
Stadium North Elevation.dwg
Stadium Plan.dwg
Stadium South Elevation.dwg
Tablet.dwg
Taisei Detail Plan.dwg
Taisei Door Window Sheet.dwg
Taisei Interior Elv.dwg
TrueType.dwg
Welding Fixture-1.dwg
Welding Fixture Model.dwg
Wilhome.dwg

Figure 18-2. *An* `OpenFile` *dialog box example*

`SaveAsFile` Common Control Dialog Replacement for VBA

Using the `SaveAsFile` common control dialog adds a look of consistency to your application. The `SaveAsFile` dialog is part of the `comdlg32.dll` library of Windows API routines and is easily accessed. The following example uses the `comdlg32.dll` routines to save a drawing file.

```
Private Declare Function GetSaveFileName Lib "comdlg32.dll" Alias _
   "GetSaveFileNameA" (pOpenfilename As OPENFILENAME) As Long

Private Type OPENFILENAME
  lStructSize As Long
  hwndOwner As Long
  hInstance As Long
  lpstrFilter As String
  lpstrCustomFilter As String
  nMaxCustFilter As Long
  nFilterIndex As Long
  lpstrFile As String
  nMaxFile As Long
  lpstrFileTitle As String
  nMaxFileTitle As Long
  lpstrInitialDir As String
  lpstrTitle As String
  flags As Long
  nFileOffset As Integer
  nFileExtension As Integer
  lpstrDefExt As String
  lCustData As Long
  lpfnHook As Long
  lpTemplateName As String
End Type
```

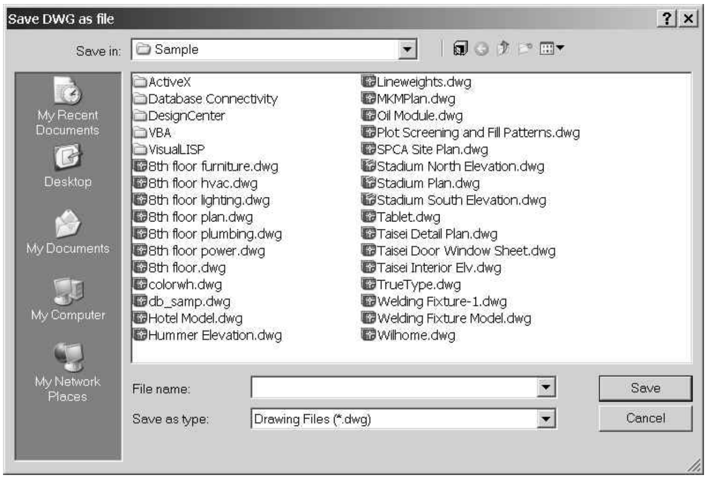
```
Public Function ShowSave(Filter As String, _
                         InitialDir As String, _
                         DialogTitle As String) As String

Dim OFName As OPENFILENAME

  'Set the structure size
  OFName.lStructSize = Len(OFName)
  'Set the owner window
  OFName.hwndOwner = 0
  'Set the filter
  OFName.lpstrFilter = Filter
  'Set the maximum number of chars
  OFName.nMaxFile = 255
  'Create a buffer
  OFName.lpstrFile = Space(254)
  'Create a buffer
  OFName.lpstrFileTitle = Space$(254)
  'Set the maximum number of chars
  OFName.nMaxFileTitle = 255
  'Set the initial directory
  OFName.lpstrInitialDir = InitialDir
  'Set the dialog title
  OFName.lpstrTitle = DialogTitle
  'no extra flags
  OFName.flags = 0
  'Show the 'SaveAs File' dialog
  If GetSaveFileName(OFName) Then
    ShowSave = Trim(OFName.lpstrFile)

    Else
      ShowSave = ""
  End If
End Function
```

The following sample code shows the ShowSave routine:

```
Dim Filter As String
Dim InitialDir As String
Dim DialogTitle As String
Dim OutputStr As String

  Filter = "Drawing Files (*.dwg)" + Chr$(0) + "*.dwg" + Chr$(0) + _
    "All Files (*.*)" + Chr$(0) + "*.*" + Chr$(0)
  InitialDir = "C:\Program Files\AutoCAD 2006\Sample"
  DialogTitle = "Save DWG as file"

  OutputStr = ShowSave(Filter, InitialDir, DialogTitle)
```

The `Filter` parameter is a string that details which file types, by extension, to display when the `SaveAsFile` dialog box appears. The `InitialDir` parameter specifies which directory to display by default. To give a name to your `SaveAsFile` dialog box, use the `DialogTitle` parameter.

With each of these parameters defined, executing this code opens the `SaveAsFile` dialog box shown in Figure 18-3.


Save DWG as file

Save in: Sample

File name:

Save as type: Drawing Files (*.dwg)

Figure 18-3. *A* `SaveAsFile` *dialog box example*

Retrieving the Status of the Caps Lock, Num Lock, and Scroll Lock Keys

Sometimes you need to know the status of the Caps Lock, Num Lock, or Scroll Lock key. Visual Basic doesn't let you do this directly, but the Windows API does. The following example shows how:

```
Private Const VK_CAPITAL = &H14
Private Const VK_NUMLOCK = &H90
Private Const VK_SCROLL = &H91

Private Type KeyboardBytes
  kbByte(0 To 255) As Byte
End Type

Private Declare Function GetKeyState Lib "user32" (ByVal nVirtKey As Long) _
  As Long
Private Declare Function GetKeyboardState Lib "user32" (kbArray As KeyboardBytes) _
  As Long
Private Declare Function SetKeyboardState Lib "user32" (kbArray As KeyboardBytes) _
  As Long

Dim kbArray As KeyboardBytes
```

This example implements these routines. This first example uses the key passed to it in the vkKey parameter to turn that key on.

```
Public Sub TurnOn(vkKey As Long)
  'Get the keyboard state
  GetKeyboardState kbArray
  'Change a key
  kbArray.kbByte(vkKey) = 1
  'Set the keyboard state
  SetKeyboardState kbArray
End Sub
```

Note The effect of running these examples may not be apparent because in certain circumstances your keyboard lights may not change from On to Off and vice versa. However, the code does change the way characters will be displayed when typed.

The following example does the opposite of the TurnOn routine from the previous example. It uses the key passed to it in the vkKey parameter to turn off that key.

```
Public Sub TurnOff(vkKey As Long)
  'Get the keyboard state
  GetKeyboardState kbArray
  'change a key
  kbArray.kbByte(vkKey) = 0
  'set the keyboard state
  SetKeyboardState kbArray
End Sub
```

This final example gets the status of the key passed in the vkKey parameter and returns True if the key is on or False if it's off.

```
Public Function GetKeyStatus(vkKey As Long) As Boolean
  'Get the keyboard state
  GetKeyboardState kbArray
  'change a key
  GetKeyStatus = kbArray.kbByte(vkKey)
End Function
```

Using these routines helps you enforce the case users input. If you need everything entered to be all capital letters, these routines ensure that you get only uppercase input.

Don't be frightened by the verbosity of these code examples. The power implied here is with respect to reuse. If you put this code to work, you'll quickly realize the benefit of encapsulating this code in class modules, which you can reuse many times with minimal coding. You can easily create custom dialog forms or other adapted features and make them basic class items from which you can draw.

Summary

This short chapter presented an enormous amount of information. The best way for you to really be proficient at accessing the Windows API is through trial and error. This chapter presented some useful routines that should help get you started writing robust applications using the Windows API.

CHAPTER 19

Connecting to External Applications

In this chapter you'll learn the basics of connecting to applications external to AutoCAD, such as Microsoft Excel and Word. Once connected, you'll see how to create basic documents and populate those external documents with information you control from within AutoCAD.

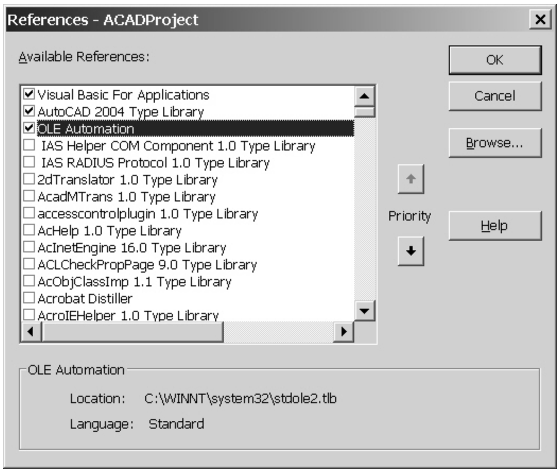
This chapter is an introduction to connecting and performing basic functions with external applications. It doesn't present in-depth coverage of all the tasks that you could accomplish, as there are numerous books dedicated to the subject of customization for these products.

At the end of this chapter, you'll examine how to connect to a database created in Microsoft Access, which is the typical way AutoCAD developers use and manipulate database files.

Making the Connection

Most applications you connect to will have similar connection methods. First, you'll need to include the necessary references in the programming environment, as illustrated in Figure 19-1.

Next, you will need a routine that checks for a running instance of the application, and that has additional code to start an instance of the application if it is not already running


References - ACADProject

Available References:

- ☑ Visual Basic For Applications
- ☑ AutoCAD 2004 Type Library
- ☑ OLE Automation
- ☐ IAS Helper COM Component 1.0 Type Library
- ☐ IAS RADIUS Protocol 1.0 Type Library
- ☐ 2dTranslator 1.0 Type Library
- ☐ AcadMTrans 1.0 Type Library
- ☐ accesscontrolplugin 1.0 Type Library
- ☐ AcHelp 1.0 Type Library
- ☐ AcInetEngine 16.0 Type Library
- ☐ ACLCheckPropPage 9.0 Type Library
- ☐ AcObjClassImp 1.1 Type Library
- ☐ Acrobat Distiller
- ☐ AcroIEHelper 1.0 Type Library

OLE Automation

Location: C:\WINNT\system32\stdole2.tlb

Language: Standard

Figure 19-1. *The References dialog box*

The following example illustrates starting Microsoft Excel programmatically:

```
Public Sub StartExcel(App As Excel.Application, Visible As Boolean)
  'handle errors inline
  On Error Resume Next
    Set App = GetObject(, "Excel.Application")  'depends on application
  'check to see if application is running
  If Err Then
    'no, application will need to be started
    Err.Clear
    Set App = CreateObject("Excel.Application")  'depends on application
    'check to see if application was started
    If Err Then
      'no, application could not be started - exit
      Exit Sub
    End If
  End If
  'set the application visibility
  App.Visible = Visible
End Sub
```

■**Note** Depending on the version of Excel you're using (especially if you have multiple versions installed), you should make a habit of requesting type libraries by explicit version name (e.g., Excel.Application.11). In fact, AutoCAD 2006 requires you to reference the application object using AutoCAD.Application.16, (which isn't necessary within AutoCAD 2006 VBA because the `Application` object is always present).

Here's how you can use this routine to ensure an instance of Excel is running before doing anything else:

```
Public Sub Start()
Dim oExcel As Excel.Application
  'attempt to start Excel
  StartExcel oExcel, True

  If Not oExcel Is Nothing Then
    MsgBox "Success"

    Else
      MsgBox "Could not start Excel, exiting ...", vbCritical
      Exit Sub
  End If
End Sub
```

■**Note** If Excel was not previously running, the Excel application will close when the `Sub` ends.

In these examples, you can start two applications with a slight modification to the code listed, as shown in Table 19-1. The acceptable variations are listed in Table 19-2.

Table 19-1. *Application Class Identification*

Application	Class Identification
Excel	Excel.Application.*x* (where *x* is the product version index)
Word	Word.Application.*x* (where *x* is the product version index)

Table 19-2. *Acceptable Microsoft Product Variations*

Product	Version	Example
Office 95	7	`Word.Application.7`
Office 97	8	`Excel.Application.8`
Office 2000	9	`Word.Application.9`
Office XP	10	`Powerpoint.Application.10`
Office 2003	11	`Outlook.Application.11`

Connecting to Microsoft Excel

Now that you have a connection to Microsoft Excel, you're going to learn how to create a new workbook and worksheet. Then you'll write and read values from the cells of a worksheet. Finally, you'll save the worksheet and close Excel.

Creating a New Workbook

To create a workbook, declare a variable of the `Excel.Workbook` object data type. Then, using the `Add` method of the Excel application object, add a new workbook as in the following example:

```
Dim Workbook As Excel.Workbook
Set Workbook = oExcel.Workbooks.Add
```

Creating a New Worksheet

To create a worksheet, declare a variable of the `Excel.WorkSheet` object data type. Then use the `Add` method of the `WorkSheets` collection object to create a new worksheet, as illustrated in the following example:

```
Dim WorkSheet As Excel.WorkSheet
Set WorkSheet = oExcel.Worksheets.Add
```

Accessing a Worksheet

To access a worksheet, declare a variable of the `Excel.WorkSheet` object data type. Then assign the `Worksheets` collection object of the `Workbook` object to the newly declared variable as in the following example:

```
Dim WorkSheet As Excel.WorkSheet
Set WorkSheet = oExcel.Worksheets(1)
```

Writing and Reading Cells

Using the `WorkSheet` object, you can assign values to `Cells`, as shown in the following example:

```
  With WorkSheet
    .Cells(1, 1).Value = 1: .Cells(1, 2).Value = 2
    .Cells(2, 1).Value = 1.5: .Cells(2, 2).Value = 3
    .Cells(3, 1).Value = "Text 1": .Cells(3, 2).Value = "Text 2"
  End With
```

■ **Note** Items such as the row and column indexes start at 1, not 0.

Reading the values contained in `Cells` is the reverse of writing them, as the following example illustrates:

```
  With WorkSheet
    MsgBox .Cells(1, 1).Value & ", " & .Cells(1, 2).Value
    MsgBox .Cells(2, 1).Value & ", " & .Cells(2, 2).Value
    MsgBox .Cells(3, 1).Value & ", " & .Cells(3, 2).Value
  End With
```

Saving and Exiting Excel

Using the `Close` method of the `Workbook` object, you can close the workbook. The first parameter of the `Close` method determines whether the workbook should be saved. You supply the second parameter, a file name, and indicate whether the first parameter is `True` as in the following example:

```
Workbook.Close True, "C:\Test.xls"
```

Once you've completed your work with the Excel application, use the `Quit` method of the Excel application object to properly close the application:

```
App.Quit
```

Connecting to Microsoft Word

This section presents the code necessary to connect to Microsoft Word programmatically.

Creating a New Document

To create a document, declare a variable of the `Word.Document` object data type. Then use the `Add` method of the `Documents` collection object to add a new document, as shown here:

```
Dim oDocument As Word.Document
Set oDocument = oWord.Documents.Add
```

Adding Text to the Document

To add text to your document, simply set the `Text` property of the `Content Range` object of the document, as in the following example:

```
oDocument.Content.Text = "This is some sample text"
```

Setting Page Orientation

You make the page orientation either portrait or landscape by setting the `Orientation` property of the `PageSetup` object for the document using this:

```
oDocument.PageSetup.Orientation = wdOrientPortrait
```

or this:

```
oDocument.PageSetup.Orientation = wdOrientLandscape
```

Setting Margins

Set the margins for your document by setting the `LeftMargin`, `RightMargin`, `TopMargin`, or `BottomMargin` property of the `PageSetup` object for the document, as shown in the following example:

```
oDocument.PageSetup.LeftMargin = InchesToPoints(0.5)
oDocument.PageSetup.RightMargin = InchesToPoints(0.5)
oDocument.PageSetup.TopMargin = InchesToPoints(0.5)
oDocument.PageSetup.BottomMargin = InchesToPoints(0.5)
```

■**Note** The value supplied to the `XXXMargin` properties must be in points. If you want to supply the values in inches (as in the example) you'll need to convert the values to points using the built-in VBA function `InchesToPoints`.

Setting the Document Header and Footer

You can set the document's header and/or footer by setting the Text property of the Headers and/or Footers collection object like this:

```
oDocument.Sections(1).Headers(wdHeaderFooterPrimary) _
  .Range.Text = "This is Header text"
```

or this:

```
oDocument.Sections(1).Footers(wdHeaderFooterPrimary) _
  .Range.Text
```

Saving and Exiting Word

Use the SaveAs method of the Document object to save your document:

```
  oDocument.SaveAs "Sample"
```

To close your document, use the Close method of the Documents collection object:

```
  oWord.Documents("Sample.doc").Close True
```

Finally, use the Quit method of the Word.Application object to actually close the Word application, as shown here:

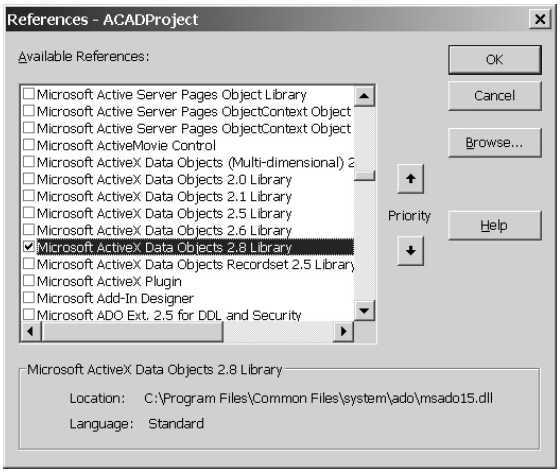
```
oWord.Quit
```

Connecting to a Microsoft Access Database

Most AutoCAD developers use Microsoft Access to create the database structure file, and then they use the Microsoft ActiveX Data Objects (ADO) library through VBA to access the .mdb file created by Access. You can use the same approach (and quite often the same code) to work with other data sources, such as Microsoft SQL Server, Oracle, DB2, and even Microsoft Exchange. This section details the steps necessary to connect to and access the data contained in a database file in a Microsoft Access application. Please refer to the same database file to follow along with these examples.

Connecting to a Database File

First, you need to add a reference to the Microsoft ActiveX Data Objects library, as shown in Figure 19-2.



- ☐ Microsoft Active Server Pages Object Library
- ☐ Microsoft Active Server Pages ObjectContext Object
- ☐ Microsoft Active Server Pages ObjectContext Object
- ☐ Microsoft ActiveMovie Control
- ☐ Microsoft ActiveX Data Objects (Multi-dimensional) 2
- ☐ Microsoft ActiveX Data Objects 2.0 Library
- ☐ Microsoft ActiveX Data Objects 2.1 Library
- ☐ Microsoft ActiveX Data Objects 2.5 Library
- ☐ Microsoft ActiveX Data Objects 2.6 Library
- ☑ Microsoft ActiveX Data Objects 2.8 Library
- ☐ Microsoft ActiveX Data Objects Recordset 2.5 Library
- ☐ Microsoft ActiveX Plugin
- ☐ Microsoft Add-In Designer
- ☐ Microsoft ADO Ext. 2.5 for DDL and Security

Microsoft ActiveX Data Objects 2.8 Library
Location: C:\Program Files\Common Files\system\ado\msado15.dll
Language: Standard

Figure 19-2. *Microsoft ActiveX Data Objects library reference*

Next, you need to establish a `Connection` object similar to the following example:

```
Dim oAccess As New ADODB.Connection

oAccess.Open "Provider=Microsoft.Jet.OLEDB.4.0;" & _
             "Data Source=" & "C:\AutoCAD-VBA.mdb" & ";"
```

■ **Note** The example database for this exercise was created in Microsoft Access 2000. If you want to use a different file-format database, such as Access 97 or 2002, you'll need to change the `Provider` string appropriately or use an ODBC data source name (DSN) to provide the interface connection directives.

Retrieving a Set of Records

The ability to retrieve the records stored in a database file is essential. The following example illustrates how to retrieve all the records from a table called Layers, and displays each record in a typical `MsgBox`:

```
  'retrieve records from the Layers table
  Dim oRecordset As New ADODB.Recordset

  oRecordset.Open "Select * From Layers", oAccess, adOpenKeyset

  With oRecordset
    Do While Not .EOF
      MsgBox !LayerName & ", Color is " & !color
      .MoveNext
    Loop
    .Close
  End With
```

Writing Values to the Database File

Another essential operation is the ability to perform writing data to a table within the database file. The following example iterates through the `Layers` collection object and writes each layer's name and color value to the Layers table:

```
Dim oLayer As AcadLayer

oRecordset.Open "Select * From Layers", oAccess, adOpenKeyset, adLockOptimistic

For Each oLayer In ThisDrawing.Layers
  With oRecordset
    .AddNew
    !LayerName = oLayer.Name
    !Color = oLayer.Color
    .Update
  End With
Next oLayer

oRecordset.Close
```

Closing the Connection

Once you've finished using the connection to a database file, you should always close the database connection, as shown in the following example:

```
oAccess.Close
```

You can go even further and create new databases, and then build the tables and queries within them using ADO and ADOX features. You can find a full reference of the ADO and ADOX objects, properties, methods, and events on the Microsoft MSDN Web site (`http://msdn.microsoft.com`) or Microsoft's Microsoft Data Access and Storage Developer Center Web site (`http://www.microsoft.com/data`).

■ **Tip** One of the most often overlooked time-savers for working with external applications is the Macro Recorder feature. You can save an enormous amount of time and frustration in trying to figure out the specifics of performing a task in Excel by just doing the task while recording it, and then opening the results. Quite often, you can copy and paste the results directly into the AutoCAD VBA editor with minor modifications, and be on your way. This is true with any VBA-enabled product that has the Macro Recorder feature, including Microsoft Visio, PowerPoint, and Outlook.

Working with Other Databases

Although Microsoft Access is a very popular database, it is by no means the only one you will encounter. Many companies, especially larger firms, rely on larger-scale database systems such as Oracle, SQL Server, and DB2. These products provide much more power and reliability than desktop databases such as Access, dBASE, and FoxPro. You should be aware of the differences you'll commonly encounter when you move data from one place to another, and between different database environments.

For example, the data type aspects can vary widely and you should *never* assume data types are consistent between such products as Microsoft Access, Microsoft SQL Server, DB2, and Oracle. For instance, when you convert databases in Access 2000 to SQL Server 2000, memo fields are converted into ntext or nvarchar fields, which are very different. Other common differences relate to date and time values, currency, and special fields such as Pictures, Hyperlinks, and BLOBs (binary large objects).

Connectivity Automation Objects

The dbConnect feature added in AutoCAD 2000 provides a powerful means for linking drawing entities to external data sources such as Excel spreadsheets, Access databases, and other ODBC data sources. It also provides a means for automatically labeling entities using Leader callouts with special control over the text labeling that is driven directly by linked values in the external data source.

This functionality is provided through a special API called Connectivity Automation Objects, or CAO. CAO is a very simple, compact object model, with a small set of objects, properties, methods, and so on (compared to AutoCAD, anyway). It exposes all of the power and capability of dbConnect to developers, both from within and from outside of AutoCAD. Like most ActiveX features in AutoCAD, you can employ CAO from VBA within AutoCAD, or from VBA in an external application.

Advanced Database Issues

Don't be lulled into thinking that because you feel comfortable with the AutoCAD drawing "database" environment, you can translate it quickly into the world of database development. The differences are staggering. Although many similarities exist, you'll encounter numerous issues in database development that aren't present in AutoCAD. For example, referential integrity, normalization, indexing, views, and job scheduling are all common in mainstream database development. And although these topics may seem unimportant or remote to you as an AutoCAD developer, they're crucial to database developers and the work they do every day.

If you aren't a seasoned database developer, it often helps to partner with one while you're learning the ropes of database programming. Although you can certainly learn database development skills, it's truly a world unto its own. Finally, before you begin building a large, complex database application with AutoCAD, consult an experienced database developer to make sure you don't reinvent the wheel or waste precious time building a solution that would have been easier or more robust if it were instead developed on the database end.

Working with Services and Other APIs

Like applications, services are processes that run within the operating-system-protected namespaces. Unlike applications, though, services normally don't provide a graphical interface and are instead intended for under-the-hood purposes such as monitoring, logging, reacting, or protecting other system resources. Many services are very useful for making sure the overall environment is known.

One service you might be interested in using is the Active Directory Service Interfaces (ADSI) object for accessing Active Directory Services on a Windows 2000 or 2003 network. An example of a commonly used API is the FileSystemObject (FSO), which is used by many applications to create, edit, read, and delete files, as well as perform file system operations on drives, folders, and so forth.

To find out the drive type and file system in use on the C: drive, for example, you could access the FSO `Drive` object for C: and then query its `DriveType` and `FileSystem` properties:

```
Function ShowDriveType(drvpath)
   Dim fso, d, dt
   Set fso = CreateObject("Scripting.FileSystemObject")
   Set d = fso.GetDrive(drvpath)
   Select Case d.DriveType
      Case 0: dt = "Unknown"
      Case 1: dt = "Removable"
      Case 2: dt = "Fixed"
      Case 3: dt = "Network"
      Case 4: dt = "CD-ROM"
      Case 5: dt = "RAM Disk"
   End Select
   ShowDriveType = dt
End Function

Function ShowFileSystemType(drvspec)
   Dim fso,d
   Set fso = CreateObject("Scripting.FileSystemObject")
   Set d = fso.GetDrive(drvspec)
   ShowFileSystemType = d.FileSystem
End Function

MsgBox "C: Drive Type is " & ShowDriveType("C")
MsgBox "C: File System is " & ShowFileSystemType("C")
```

The ADSI object is built into Windows 2000, Windows XP Professional, and corresponding server versions of these products. It provides an API for interfacing with directory services such as Lightweight Directory Access Protocol (LDAP) and Active Directory. A common task might be to retrieve a list of all the users, groups, or computers on your domain. To do this, you could invoke the ADSI interface to the `WinNT` object, as shown here:

```
Sub ShowAllComputers()
  Dim adsDomain
  Set adsDomain = GetObject("WinNT://MyDomainName")
  For each objMember in adsDomain
    If Ucase(objMember.Class) = "COMPUTER" Then
      Debug.Print objMember.Name  & vbCrLf
    End If
  Next
  Set adsDomain = Nothing
End Sub
```

You might have noticed in the preceding code that `GetObject` is used instead of `CreateObject`. This is because a true service is always available through the `GetObject` interface and doesn't require instantiation to obtain an instance.

The FSO, ADSI, and LanManServer API references are all published on the MSDN Web site, as are many other useful APIs and services such as WMI/WBEM, CDO/CDONTS, LDAP, and Windows Scripting Host (WSH).

Summary

This chapter briefly touched upon the capabilities that VBA provides for working with external applications; it's by no means exhaustive. You can interface with an almost limitless number of external applications, services, and processes from within AutoCAD VBA. The more you look and experiment, the more you'll be amazed at what you can do.

CHAPTER 20

Creating Tables

AutoCAD 2005 introduced a new object aimed at creating drawing tables for schedules or bills of materials. The new object, `Table`, allows you to create a rectangular array of any number of rows and columns and populate it with virtually any data you like. Although demonstrating every conceivable way of using a table is not possible, in this chapter I will show you enough of the basics of creating and manipulating tables that you can experiment on your own to find the best fit for your situation.

The `AddTable` Method

You can create a `Table` object by specifying the insertion point, number of rows, number of columns, row height, and column width using the `AddTable` method.

```
Set TableObject = Object.AddTable(InsertionPoint, NumberOfRows, _
 NumberOfColumns, RowHeight, ColumnWidth)
```

Table 20-1 explains this method's parameters.

Table 20-1. *The* `AddTable` *Method's Parameters*

Name	Data Type	Description
`InsertionPoint`	Double	Insertion point of table
`NumberOfRows`	Long	Number of rows in table
`NumberOfColumns`	Long	Number of columns in table
`RowHeight`	Double	Height of each row in table
`ColumnWidth`	Double	Width of each column in table

The following example shows how to create a table based upon user input:

```
Public Sub TestAddTable_Click()
Dim vInsertionPoint As Variant
Dim lNumberOfRows As Long
Dim lNumberOfColumns As Long
Dim dRowHeight As Double
Dim dColumnWidth As Double
Dim oTable As AcadTable
```

```
  On Error Resume Next

  With ThisDrawing.Utility
    vInsertionPoint = .GetPoint(, vbCr & "Pick the insertion point: ")
    lNumberOfRows = .GetInteger(vbCr & "Enter number of rows: ")
    lNumberOfColumns = .GetInteger(vbCr & "Enter number of columns: ")
    dRowHeight = .GetReal(vbCr & "Enter row height: ")
    dColumnWidth = .GetReal(vbCr & "Enter column width: ")
  End With

  If Err Then Exit Sub

  Set oTable = ThisDrawing.ModelSpace.AddTable(vInsertionPoint, _
lNumberOfRows, lNumberOfColumns, dRowHeight, dColumnWidth)
End Sub
```

The `RegenerateTableSuppressed` Property

The `RegenerateTableSuppressed` property specifies whether the table will be repainted each time a change to the table is made.

```
object.RegenerateTableSuppressed
```

Table 20-2 shows the `RegenerateTableSuppressed` property's parameters.

Table 20-2. *The* `RegenerateTableSuppressed` *Property's Parameters*

Value	Description
`True`	The table regeneration is suppressed or off.
`False`	The table regeneration is not suppressed or on.

The `GetText` Method

Text within a Table can be read using the `GetText` method to retrieve the text string value of a specified row and column of an existing table.

The `GetText` method returns a string and has the following syntax:

```
object.GetText Row, Column
```

Table 20-3 explains this method's parameters.

Table 20-3. *The* `GetText` *Method's Parameters*

Name	Data Type	Description
`Row`	Long	A zero-based row index for the cell containing text
`Column`	Long	A zero-based column index for the cell containing text

The following example illustrates how to read the text value from row 2 and column 3 of a user-selected table:

```
Public Sub TestGetText()
Dim vPoint As Variant
Dim oTable As AcadTable
Dim Text As String

  On Error Resume Next

  ThisDrawing.Utility.GetEntity oTable, vPoint, "Select table: "

  If TypeOf oTable Is AcadTable Then
    Text = oTable.GetText(1, 2)
  End If

  MsgBox Text
End Sub
```

The `SetText` Method

Use the `SetText` method to set the text string value of a specified row and column of an existing table.

The `SetText` method returns nothing but has the following syntax:

```
object.SetText Row, Column, Text
```

Table 20-4 explains this method's parameters.

Table 20-4. *The* `SetText` *Method's Parameters*

Name	Data Type	Description
`Row`	Long	A zero-based row index for the cell receiving text
`Column`	Long	A zero-based column index for the cell receiving text
`Text`	String	The string to place in the cell specified by row and column of the table

The following example illustrates how to set the text value for row 2 and column 3 of a user-selected table:

```
Public Sub TestSetText()
Dim vPoint As Variant
Dim oTable As AcadTable

  On Error Resume Next

  ThisDrawing.Utility.GetEntity oTable, vPoint, "Select table: "
  If TypeOf oTable Is AcadTable Then
    With oTable
      .SetText 1, 2, "New string"
      Text = .GetText(1, 2)
    End With
  End If

  MsgBox Text
End Sub
```

The GetTextHeight Method

Use the `GetTextHeight` method to retrieve the row height of a specific type of row.

The `GetTextHeight` method returns the height of the row type specified and has the following syntax:

```
object.GetTextHeight RowType
```

Table 20-5 explains this method's parameter.

Table 20-5. *The* `GetTextHeight` *Method's Parameter*

Name	Data Type	Description

`RowType`	AcRowType	One of these row types:
		`AcRowType` enumerations
		`AcDataRow`
		`AcHeaderRow`
		`AcTitleRow`
		`AcUnknownRow`

The following example illustrates how to retrieve the row height for each of the types of rows in a table:

```
Public Sub TestGetTextHeight()
Dim vPoint As Variant
Dim oTable As AcadTable
Dim Index As Long

  On Error Resume Next

 ThisDrawing.Utility.GetEntity oTable, vPoint, "Select table: "

  If TypeOf oTable Is AcadTable Then
    With oTable
      For Index = 0 To 3
        Select Case Index
          Case 0
            MsgBox "DataRow height: " & .GetTextHeight(acDataRow)
          Case 1
            MsgBox "HeaderRow height: " & .GetTextHeight(acHeaderRow)
          Case 2
            MsgBox "TitleRow height: " & .GetTextHeight(acTitleRow)
          Case 3
            MsgBox "UnknownRow height: " & .GetTextHeight(acUnknownRow)
        End Select
      Next Index
    End With
  End If
End Sub
```

The SetTextHeight Method

Use the SetTextHeight method to set the row height of a specific type of row. The method returns nothing but has the following syntax:

```
object.SetTextHeight RowType, TextHeight
```

Table 20-6 explains this method's parameters.

Table 20-6. *The* SetTextHeight *Method's Parameters*

Name	Data Type	Description
RowType	AcRowType	One of these row types: AcRowType enumerations AcDataRow AcHeaderRow AcTitleRow AcUnknownRow
TextHeight	Double	Height of the text contained in specified row

The following example illustrates how to set the row height for each of the types of rows in a table:

```
Public Sub TestSetTextHeight()
Dim vPoint As Variant
Dim oTable As AcadTable
Dim Index As Long

  On Error Resume Next

 ThisDrawing.Utility.GetEntity oTable, vPoint, "Select table: "

  If TypeOf oTable Is AcadTable Then
    With oTable
      For Index = 0 To 3
        Select Case Index
```

```
          Case 0
            .SetTextHeight acDataRow, 0.25
            MsgBox "DataRow height: " & .GetTextHeight(acDataRow)
          Case 1
            .SetTextHeight acHeaderRow, 0.25
            MsgBox "HeaderRow height: " & .GetTextHeight(acHeaderRow)
          Case 2
            .SetTextHeight acTitleRow, 0.25
            MsgBox "TitleRow height: " & .GetTextHeight(acTitleRow)
          Case 3
            .SetTextHeight acUnknownRow, 0.25
            MsgBox "UnknownRow height: " & .GetTextHeight(acUnknownRow)
        End Select
      Next Index
    End With
  End If
End Sub
```

The `InsertRows` Method

Use the `InsertRows` method to insert additional rows in an existing table.

■ **Note** Row(s) are inserted *before* the row index provided as a parameter to this method.

The method returns nothing but has the following syntax:

```
object.InsertRows Index, RowHeight, Rows
```

Table 20-7 explains this method's parameters.

Table 20-7. *The* `InsertRows` *Method's Parameters*

Name	Data Type	Description
`Index`	Long	Zero-based row index
`RowHeight`	Double	The height of row(s) to be added
`Rows`	Long	The number of rows to be added

The following example illustrates how to insert two rows that are 0.25 inches high before the second row of an existing table:

```
Public Sub TestInsertRows()
Dim vPoint As Variant
Dim oTable As AcadTable

  On Error Resume Next
  ThisDrawing.Utility.GetEntity oTable, vPoint, "Select table: "

  If TypeOf oTable Is AcadTable Then
    oTable.InsertRows 1, 0.25, 2
    'second row, insert two 0.25 high rows
  End If
End Sub
```

Figure 20-1 shows the table before `InsertRows`, and Figure 20-2 shows the table after `InsertRows`.

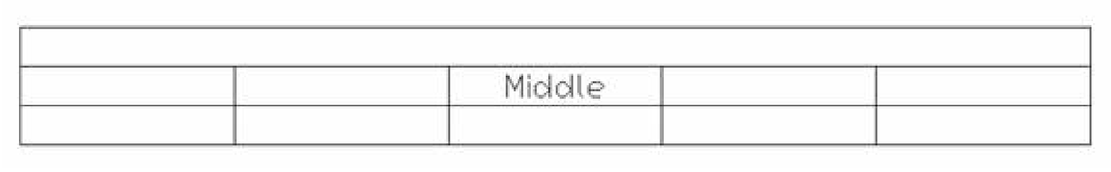


Figure 20-1. *Viewing the table before* `InsertRows`

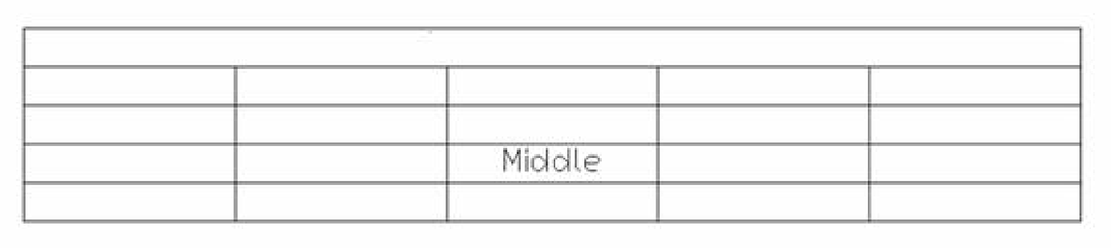


Figure 20-2. *Viewing the table after* `InsertRows`

The `InsertColumns` Method

Use the `InsertColumns` method to insert additional columns in an existing table.

■**Note** Column(s) are inserted *before* the column index provided as a parameter to this method.

The method returns nothing but has the following syntax:

```
object.InsertColumns Index, ColumnWidth, Columns
```

Table 20-8 explains this method's parameters.

Table 20-8. *The* `InsertColumns` *Method's Parameters*

Name	Data Type	Description
`Index`	Long	Zero-based column index
`ColumnWidth`	Double	The width of column(s) to be added
`Columns`	Long	The number of columns to be added

The following example illustrates how to insert two columns that are 1.25 inches wide before the second column of an existing table:

```
Public Sub TestInsertColumns()
Dim vPoint As Variant
Dim oTable As AcadTable

  On Error Resume Next

 ThisDrawing.Utility.GetEntity oTable, vPoint, "Select table: "

  If TypeOf oTable Is AcadTable Then
    oTable.InsertColumns 1, 1.25, 2
    'second column, insert two 1.25 wide columns
  End If
End Sub
```

Figure 20-3 shows the table before `InsertColumns`, and Figure 20-4 shows the table after `InsertColumns`.

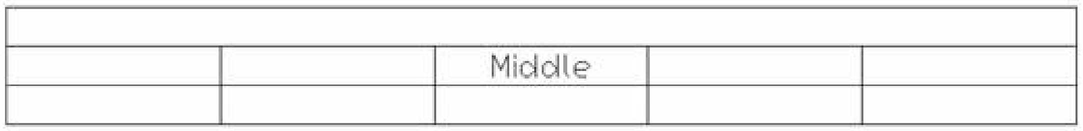


Figure 20-3. *Viewing the table before* `InsertColumns`

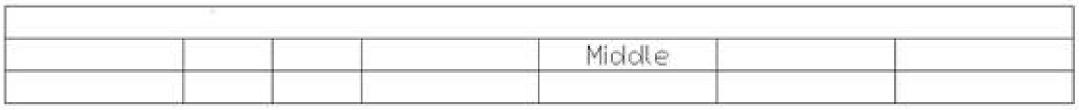


Figure 20-4. *Viewing the table after* `InsertColumns`

Putting It All Together

You now have an idea of some of the features and characteristics possible for creating and manipulating a Table object within AutoCAD. However, this may not seem all that useful just yet. In this final part of the chapter, I will show how to build a simple bill of materials, or *schedule* as some disciplines called it, using a table populated with the drawing's block insertions.

First, you need to determine what constitutes a component and create a class object representation. For this example, a component consists of the PartNumber and Description. Additionally, you can keep a rolling count of the number of components as follows:

```
Public PartNumber As String
Public Count As Long
Public Description As String
```

Type the previous code in a class module named Component.

Second, to generate a list of components or block insertions contained within the drawing, you will need a collection of Component objects, which is created using the following:

```
Public Components As New Collection         'global variable for creating BOM
```

Type this code example into a standard .bas module.

Next, you need to create a selection set containing just block insertions. For a discussion of selection set creation, refer to Chapter 12. Type the following line of code into the same standard .bas module that you placed the Components collection object declaration in:

```
Public Const Entity = 0                     'an entity
```

You will use this constant declaration when you create the actual selection set by typing in the following function in the ThisDrawing module:

```
Public Function CreateSelectionSet(SelectionSet As AcadSelectionSet, _
                                   FilterCode As Integer, _
                                   FilterValue As String) As Boolean
Dim intFilterCode(0) As Integer
Dim varFilterValue(0) As Variant

'assume success
CreateSelectionSet = True

  intFilterCode(0) = FilterCode
  varFilterValue(0) = FilterValue

  SelectionSet.Select acSelectionSetAll, , , intFilterCode, varFilterValue

  If SelectionSet.Count < 1 Then
    CreateSelectionSet = False
  End If
End Function
```

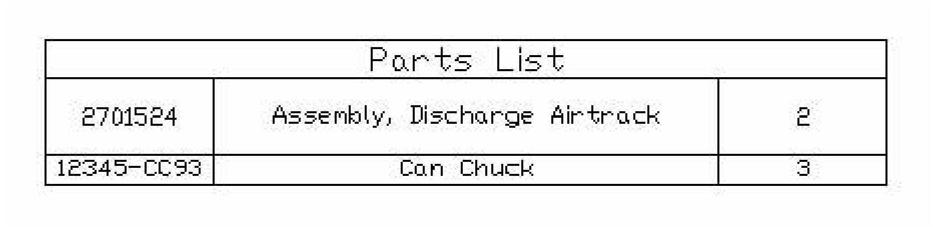
Next, you are ready to generate the selection set and populate the Components collection with each Component within the selection set. Basically, the routine will generate a selection set and then iterate through the selection set looking for block insertions that contain attributes. These attributes were predefined and match the criteria for a `Component` object. If you needed something different, you would change these definitions to meet your own requirements. Each block insertion with attributes is then added to the `Components` collection. If the `Component` exists in the collection, then just the `Count` property is incremented. The following code wraps all this functionality into a single routine called `CreateBillOfMaterials`:

```
Public Sub CreateBillOfMaterials()
Dim oSS As AcadSelectionSet
Dim oAttribute As AcadAttribute
Dim BlockRef As AcadBlockReference
Dim Index As Long
Dim Component As New Component
Dim Attribs As Variant

  On Error Resume Next
  ThisDrawing.SelectionSets("Inserts").Delete
  On Error GoTo 0

  Set oSS = ThisDrawing.SelectionSets.Add("Inserts")

  'zero the Components class object
  Set Components = Nothing

  'create a selection set of BlockRefs
  If CreateSelectionSet(oSS, Entity, "INSERT") Then
    For Index = 0 To oSS.Count - 1
      Set BlockRef = oSS(Index)

      If BlockRef.HasAttributes Then
        Attribs = BlockRef.GetAttributes

        On Error Resume Next
        Set Component = Components(Attribs(0).TextString)

        If Err Then
          Err.Clear
          On Error GoTo 0

          With Component
            .Count = 1
            .PartNumber = Attribs(0).TextString
            .Description = Attribs(1).TextString
          End With

          Components.Add Component, Component.PartNumber
          Else
            Components(Attribs(0).TextString).Count = _
         Components(Attribs(0).TextString).Count + 1
        End If

        Set Component = Nothing
      End If
    Next Index
  End If
  'create the table on the drawing
  CreateTableBOM
End Sub
```

Notice the last statement in the `CreateBillOfMaterials` routine. This is the routine you will use to actually create the table on the drawing and populate its contents with the data you generated using the following `CreateBillOfMaterials` routine:

```
Private Sub CreateTableBOM()
Dim vInsertionPoint As Variant
Dim lNumberOfRows As Long
Dim lNumberOfColumns As Long
Dim dRowHeight As Double
Dim dColumnWidth As Double
Dim oTable As AcadTable
Dim Row As Long
Dim Column As Long

  On Error Resume Next

  With ThisDrawing.Utility
    vInsertionPoint = .GetPoint(, vbCr & "Pick the insertion point: ")
  End With

  If Err Then Exit Sub

  lNumberOfRows = Components.Count + 1
  lNumberOfColumns = 3
  dRowHeight = 0.25
  dColumnWidth = 2

  Set oTable = ThisDrawing.ModelSpace.AddTable(vInsertionPoint, _
lNumberOfRows, lNumberOfColumns, dRowHeight, dColumnWidth)

  'put in the header row
  oTable.SetText Row, Column, "Parts List"
  For Row = 1 To Components.Count
    With Components(Row)
      oTable.SetText Row, Column, .PartNumber
      Column = Column + 1
      oTable.SetText Row, Column, .Description
      Column = Column + 1
      oTable.SetText Row, Column, .Count
    End With
    'reset column back to zero
    Column = 0
  Next Row

  'adjust the Description column width
  oTable.SetColumnWidth 1, 6
End Sub
```

Executing the code presented, you will be asked to select a location for the table. By default, the table will be created in an upper-left to lower-right direction. So, the point you select constitutes the upper-left corner of your bill of materials table, as shown in Figure 20-5.



Parts List		
2701524	Assembly, Discharge Airtrack	2
12345-CC93	Can Chuck	3

Figure 20-5. *Creating a bill of materials*

Summary

In this chapter I discussed how to create a table. Additionally, I gave examples of adding and manipulating text and introduced how to insert rows and columns and manipulate their appearance. In addition, I explained how and when to regenerate the table contents. Finally, you looked at how to put all you learned in this chapter to good use by creating a bill of materials on a drawing.

CHAPTER 21

The SummaryInfo Object

AutoCAD 2005 introduces the means for creating and storing properties of a drawing. Author, title, subject, and other information may be added, queried, and manipulated. Users can manipulate the standard built-in drawing properties or create custom properties specific to their needs. You may access and manipulate this feature from a command prompt, a GUI interface, or programmatically. The command interface is `DWGPROPS`, which displays a dialog form for adding and changing drawing properties, as shown in Figure 21-1.

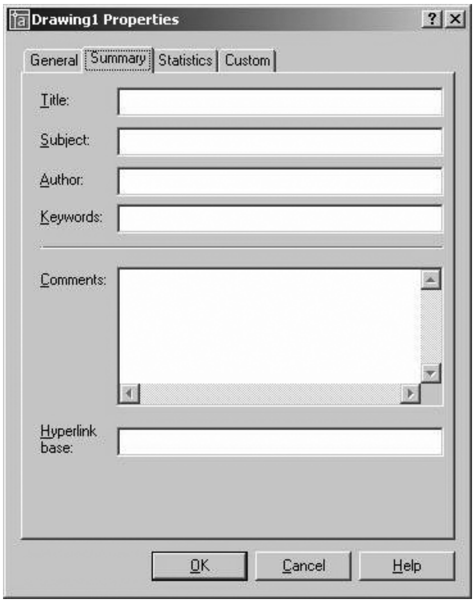


Figure 21-1. *Drawing properties*

This dialog is accessible in AutoCAD through the File ➤ Drawing Properties menu item.

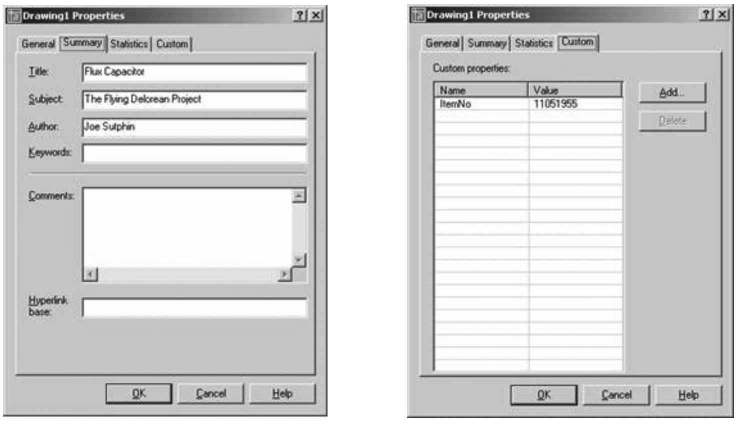
To support these features programmatically Autodesk added a new object called `SummaryInfo`. This object allows you to add, change, and manipulate various built-in drawing properties as well as create your own custom properties.

Overview

The following example illustrates using the `AddCustomInfo` method and setting properties of the `SummaryInfo` object to manipulate standard built-in drawing properties and create custom properties:

```
Public Sub TestSummaryInfoObject()
  With ThisDrawing.SummaryInfo
    .Author = "Joe Sutphin"
    .RevisionNumber = "3"
    .Subject = "The Flying Delorean Project"
    .Title = "Flux Capacitor"
    .AddCustomInfo "ItemNo", "11051955"
  End With
End Sub
```

Figure 21-2 illustrates the resulting `DWGPROPS` dialog box when you execute the code in the preceding example above. Note the added custom property `ItemNo`.



Title: Flux Capacitor
Subject: The Flying Delorean Project
Author: Joe Sutphin

Name	Value
ItemNo	11051955

Figure 21-2. *Summary and custom property views*

Properties

The rest of this chapter will outline how to manipulate standard built-in properties and how to create your own custom properties as you did with the preceding code.

The `Author` Property

The `Author` property specifies the drawing author's name:

```
object.Author
```

Table 21-1 shows the `Author` property's parameter.

Table 21-1. *The* `Author` *Property Parameter*

Name	Returns	Description
`Author`	String; read-write	The drawing author's name

Here's an example of setting the `Author` property:

```
ThisDrawing.SummaryInfo.Author = "Joe Sutphin"
```

The `RevisionNumber` Property

The `RevisionNumber` property specifies a revision level for the drawing:

```
object.RevisionNumber
```

Table 21-2 shows the `RevisionNumber` property's parameters.

Table 21-2. *The* `RevisionNumber` *Property Parameter*

Name	Returns	Description
`RevisionNumber`	String; read-write	The drawing's revision level

Here's an example of setting the `RevisionNumber` property:

```
ThisDrawing.SummaryInfo.RevisionNumber = "1"
```

The following example illustrates incrementing the RevisionNumber property. The code is written so that you do not need to concern yourself with whether the previous revision was numeric or alphabetic.

```
ThisDrawing.SummaryInfo.RevisionNumber = _
   Chr(Asc (ThisDrawing.SummaryInfo.RevisionNumber) + 1)
```

The result of executing the code will be to set the `RevisionNumber` property to one greater than its previous value. If it was set to 1 initially, for example, it will now be 2. If it were set to B it would now be C, and so on.

The `Subject` Property

The `Subject` property specifies a subject for the drawing:

```
object.Subject
```

Table 21-3 shows the `Subject` property's parameter.

Table 21-3. *The* `Subject` *Property Parameter*

Name	Returns	Description
`Subject`	String; read-write	The drawing's subject

Here's an example of setting the `Subject` property:

```
ThisDrawing.SummaryInfo.Subject = "The Flying DeLorean Project"
```

The `Title` Property

The `Title` property specifies a title for the drawing:

```
object.Title
```

Table 21-4 shows the `Title` property's parameter.

Table 21-4. *The* `Title` *Property Parameter*

Name	Returns	Description
`Title`	String; read-write	The drawing's title

Here's an example of setting the `Title` property:

```
ThisDrawing.SummaryInfo.Title = "Flux Capacitor"
```

Adding Custom SummaryInfo

Although the standard `SummaryInfo` property offerings may be adequate for some users, others will inevitably need additional properties added to the drawing. In this section I will present the methods to add custom properties in the `SummaryInfo` object.

The `AddCustomInfo` Method

Use the `AddCustomInfo` method to create a custom property within your drawing. The `AddCustomInfo` method returns nothing, and has the following syntax:

```
object.AddCustomInfo Key, Value
```

Table 21-5 details this method's parameters.

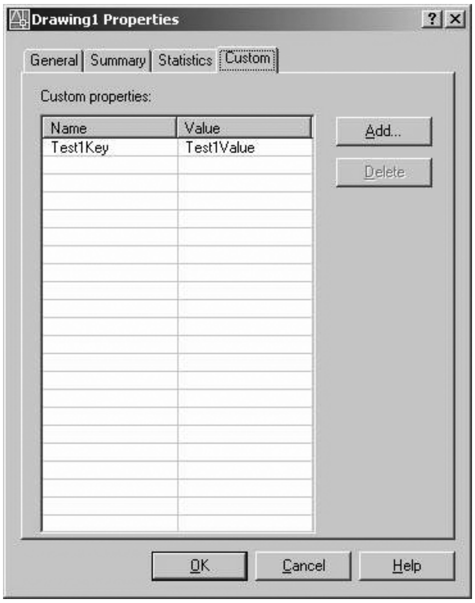
Table 21-5. `AddCustomInfo` *Method Parameters*

Name	Returns	Description
`Key`	String	The name for the custom property
`Value`	String	The value for the custom property

The following example illustrates adding a custom property called `Key0`, which can represent any value you desire.

```
Public Sub TestAddCustomInfo()
Dim Key0 As String
Dim Value0 As String

  Key0 = "Test1Key": Value0 = "Test1Value"
  ThisDrawing.SummaryInfo. AddCustomInfo Key0, Value0
End Sub
```

You can view custom properties by clicking File ➤ Drawing Properties, then selecting the Custom tab to display the dialog shown in Figure 21-3.



Name	Value
Test1Key	Test1Value

Figure 21-3. *Custom property entry*

The `GetCustomByKey` Method

Use the `GetCustomByKey` method to retrieve the custom properties set in the drawing. The `GetCustomByKey` method returns nothing, and has the following syntax:

```
object.GetCustomByKey Key, Value
```

Table 21-6 details this method's parameters.

Table 21-6. `GetCustomByKey` *Method Parameters*

Name	Data Type	Description
`Key`	String	The name of the custom property to get
`Value`	String	The value of the custom property

The following example illustrates retrieving a custom property called `Key0`:

```
Public Sub TestGetCustomInfo()
Dim Key0 As String
Dim Value0 As String

  ThisDrawing.SummaryInfo. GetCustomByKey Key0, Value0
End Sub
```

The `NumCustomInfo` Method

To iterate through all the custom property fields set in a drawing you will need to know how many fields there are in the drawing. The `NumCustomInfo` method returns that value. The method has no parameters, but has the following syntax:

```
RetVal = object.NumCustomInfo
```

Table 21-7. `NumCustomInfo` *Method*

Name	Data Type	Description
`RetVal`	Long	The number of custom property fields defined in this drawing

The following example illustrates using the `NumCustomInfo` method and the `GetCustomByIndex` method (discussed next) to iterate through each custom property field defined in the drawing:

```
Public Sub TestGetCustomByIndex()
Dim Index As Long
Dim CustomKey As String
Dim CustomValue As String

  'iterate all custom properties of drawing
  For Index = 0 To ThisDrawing.SummaryInfo.NumCustomInfo - 1
    ThisDrawing.SummaryInfo.GetCustomByIndex Index, CustomKey, CustomValue
    MsgBox "Key = " & CustomKey & vbCrLf & _
           "Value = " & CustomValue
  Next Index
End Sub
```

The `GetCustomByIndex` Method

This method retrieves the custom property field and value based upon an index value. The method returns nothing, and has the following syntax:

```
object.GetCustomByIndex Index, Key, Value
```

Table 21-8 details this method's parameters.

Table 21-8. `GetCustomByIndex` *Method Parameters*

Name	Data Type	Description
`Index`	Long	The index value of the custom property field; this index can be from zero to the number of custom properties minus one.
`Key`	String	The name of the custom property to get.
`Value`	String	The variable to hold the value of the custom property.

The `RemoveCustomByKey` Method

Use the `RemoveCustomByKey` to remove a custom property set in the drawing using a key or string value. The `RemoveCustomByKey` method returns nothing, and has the following syntax:

```
object.RemoveCustomByKey Key
```

Table 21-9 details this method's single parameter.

Table 21-9. *The* `RemoveCustomByKey` *Method Parameter*

Name	Data Type	Description
`Key`	String	The name of the custom property to remove

The following example illustrates removing a custom property called `Key0`:

```
Public Sub TestRemoveCustomInfoByKey()
  ThisDrawing.SummaryInfo. RemoveCustomByKey "Key0"
End Sub
```

The `RemoveCustomByIndex` Method

Use `RemoveCustomByIndex` to remove a custom property set in the drawing using a numeric index value. The `RemoveCustomByIndex` method returns nothing, and has the following syntax:

```
object.RemoveCustomByKey Index
```

Table 21-10 details this method's single parameter.

Table 21-10. *The* `RemoveCustomByIndex` *Method Parameter*

Name	Data Type	Description
`Index`	Long	The index value of the property to remove; the acceptable range of values is from zero to the number of custom properties minus one.

The following example illustrates removing the third custom property set in the drawing:

```
Public Sub TestRemoveCustomInfoByIndex()
  ThisDrawing.SummaryInfo. RemoveCustomByIndex 2
End Sub
```

The `SetCustomByKey` Method

Use the `SetCustomByKey` method to change the custom properties set in the drawing. The `SetCustomByKey` method returns nothing, and has the following syntax:

```
object.SetCustomByKey Key, Value
```

Table 21-11 details this method's parameters.

Table 21-11. `SetCustomByKey` *Method Parameters*

Name	Data Type	Description
`Key`	String	The name of the custom property to set
`Value`	String	The value of the custom property

The following example illustrates changing a custom property called `Key0`:

```
Public Sub TestSetCustomInfo()
Dim Value0 As String

      Value0 = "NewValue"
  ThisDrawing.SummaryInfo. SetCustomByKey Key0, Value0
End Sub
```

The `SetCustomByIndex` Method

This method sets the custom property field and value based upon an index value. The method returns nothing, and has the following syntax:

```
object.SetCustomByIndex Index, Key, Value
```

Table 21-12 details this method's parameters.

Table 21-12. `SetCustomByIndex` *Method Parameters*

Name	Data Type	Description
`Index`	Long	The index value of the custom property field; this index can be from zero to the number of custom properties minus one.
`Key`	String	The name of the custom property to get.
`Value`	String	The variable to hold the value of the custom property.

The following example illustrates iterating through all the custom property fields and changing each custom property field using an index:

```
Public Sub TestSetCustomByIndex()
Dim Index As Long
Dim CustomKey As String
Dim CustomValue As String

  'iterate all custom properties of drawing
  For Index = 0 To ThisDrawing.SummaryInfo.NumCustomInfo - 1
    ThisDrawing.SummaryInfo.GetCustomByIndex Index, CustomKey, CustomValue
    MsgBox "Key = " & CustomKey & vbCrLf & _
           "Value = " & CustomValue
  Next Index
End Sub
```

Summary

This chapter provided an overview of the various features and capabilities of the `SummaryInfo` object. We covered changing the standard built-in properties and creating and manipulating your own custom properties.

CHAPTER 22

An Illustrative VBA Application

In this chapter we will develop a real-life application involving the use of AutoCAD and Microsoft Word. Executed from AutoCAD, this application will accomplish the following list of tasks:

- Open Microsoft Word.
- Create a new document.
- Create a table with the appropriate number of columns.
- Make adjustments to various table parameters.
- Fill the table with AutoCAD layer data.

Additionally, many small details not in the list will be explained throughout this chapter. Our goal is to finish with an application that has a user interface like the one shown in Figure 22-1, and an output like that shown in Figure 22-2.

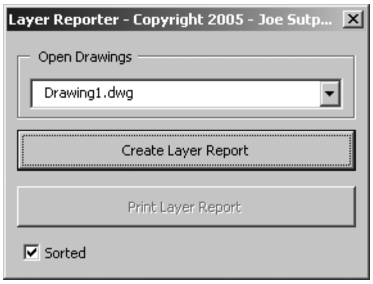
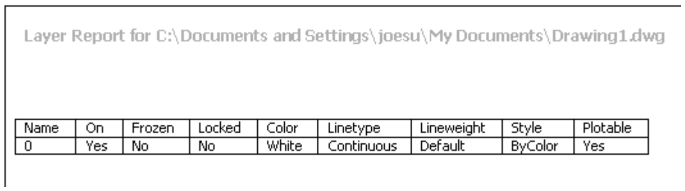


Figure 22-1. *A sample user interface*



Layer Report for C:\Documents and Settings\joesu\My Documents\Drawing1.dwg

Name	On	Frozen	Locked	Color	Linetype	Lineweight	Style	Plotable
0	Yes	No	No	White	Continuous	Default	ByColor	Yes

Figure 22-2. *Sample output*

Reporting on the layers and their settings is a fairly common requirement, especially if you work with clients. Many times the customer wants something with a specific format and typeface. This chapter will give you the skills to further develop the application into exactly what your clients desire.

Start Building the Application

Since this is just a simple automation-style project, I am not going to discuss all the theoretical implications of software design, or patterns, or any of that other boring stuff. We are going to do what most AutoCAD developers do—write code.

First, as you noticed in Figure 22-1 this application will be form-based, so you need to design a form. Make sure your form includes two `CommandButtons`, a `ComboBox`, and a `Checkbox`. The `Frame` isn't required but improves the form's aesthetics. I like to use names for my controls that make sense, like `cmdCreateLayerReport` instead of `CommandButton1`. So, rename your controls to those listed in Table 22-1.

Table 22-1. *Default Control Names and Their New Names*

Default Name	Rename To
`ComboBox1`	`cboDrawingName`
`CommandButton1`	`cmdCreateLayerReport`
`CommandButton2`	`cmdPrintLayerReport`
`CheckBox1`	`chkSorted`

Next you need to add a reference to the Microsoft Word Object Library. Locate it in the Available References dialog on the Tools pull-down, check its box, and press OK. Be sure to save your work frequently. The visual portion of this project is now complete. As you can see, the user interface is rather simple. However, the code behind the scenes is a bit more complex.

Writing the Code

This section will lay out the details to accomplish our task of creating a Microsoft Word–based report of the layers defined within a particular AutoCAD drawing.

Initial Declarations

Create a standard `.bas` module. In this module you will declare global variables for use throughout the application. The following code should get you started with the application:

```
Public mobjWord As Word.Application
Public mobjDoc As Word.Document
Public mobjTable As Word.Table
```

Next, you want to populate the Open Drawings combo box list with the names of all the drawings open in the current session and make the current selection that of the active document by default. Place the following code in the `Userform_Initialize` event routine:

```
Dim oDocument As AcadDocument
  For Each oDocument In Application.Documents
    cboDrawingName.AddItem oDocument.Name
  Next oDocument
  'set the starting list value to active document
  Dim Index As Long
    For Index = 0 To cboDrawingName.ListCount - 1
      With cboDrawingName
        .ListIndex = Index
        If .List(.ListIndex) = Application.ActiveDocument.Name Then
          Exit For
        End If
      End With
    Next Index
```

If you were to execute your application with just what you have done up to this point and with several drawings open, you would get a dialog box similar to Figure 22-3 when you click the down arrow on the Open Drawings combo box:

At this point the user would be able to create a layer report for the currently active document or pick from any of the other open drawings. Next, you want to actually create the layer report using the following list as a guide:

- Create a session of Microsoft Word.
- Create a new document in Word.
- Create a table within the Word document.
- Populate the table with AutoCAD layer data.

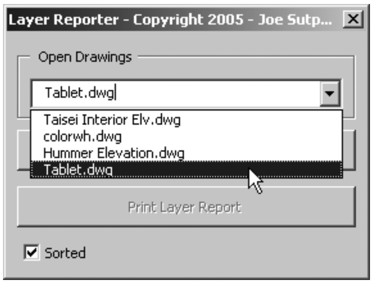


Figure 22-3. *Layer Reporter main dialog box*

While this seems like a simple list of tasks to accomplish, you'll need to attend to numerous details if you want your layer report to look professional. When the user clicks on the Create Layer Report button, the series of events outlined in the following sections will occur.

Note The following lines of code go in the `cmdCreateLayerReport_Click` event routine.

Create a Session of Microsoft Word

This application uses automation with Microsoft Word for formatted printing purposes. To use Word you must first connect or instantiate a running instance of Microsoft Word, as illustrated in the following:

```
Set mobjWord = CreateObject("Word.Application")
```

Simply instantiating Microsoft Word does not make it visible, however. You must make it visible manually by setting the object's `Visible` property to `True`:

```
mobjWord.Visible = True
```

Create a New Document in Word

With an instance of Microsoft Word now running, create a new document:

```
Set mobjDoc = mobjWord.Documents.Add
```

Create a Table with the Word Document

Inside your newly created document you want to create a table. First, determine how many columns are needed. Table 22-2 outlines the various columns you want to represent in your table:

Table 22-2. *Layer States Being Reported*

Heading	Description
On	Determines if the layer is on or off
Frozen	Determines if the layer is frozen or thawed
Locked	Determines if the layer is locked or unlocked
Color	Layer color
Linetype	Layer linetype
Lineweight	Layer lineweight
Style	Layer plot style
Plot	Determines if the layer will be plotted

Nine columns of data and the number of layers within the drawing will control the size of the table, as illustrated in the following:

```
Set mobjTable = mobjWord.ActiveDocument.Tables.Add _
 (mobjWord.ActiveDocument.Range, _
    Application.Documents(cboDrawingName.Text).Layers.Count + 1, 9)
```

Notice the row count (the number of layers within the drawing) is the layer count plus one. This extra row will be used to place a column header in our report.

Create Column Headings

With a column created for each piece of data we want to represent in the table, you want to identify each column with a header using the following code example:

```
Dim lngRow As Long
Dim lngColumn As Long

  'row counter must begin at 1
  lngRow = 1
  'column counter must begin at 1
  lngColumn = 1

  With mobjTable
    .Cell(lngRow, lngColumn).Range.Text = "Name"
    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = "On"
    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = "Frozen"
    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = "Locked"
    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = "Color"
    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = "Linetype"
    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = "Lineweight"
    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = "Style"
    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = "Plottable"
  End With
```

Populate the Table with AutoCAD Layer Data

Placing the layer data contained in the drawing is the core functionality of the entire application. The following code accomplishes that task by iterating through the AutoCAD `Layers` collection and placing each layer and its associated data in a single row of the table, as shown in Figure 22-2.

```
  'increment the row counter
  lngRow = lngRow + 1

  'column must begin at 1
  lngColumn = 1

Dim objLayer As AcadLayer
```

```
'put layer data in table
For Each objLayer In Application.Documents(cboDrawingName.Text).Layers
  With mobjTable
    .Cell(lngRow, lngColumn).Range.Text = objLayer.Name
    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = ConvertToWord(objLayer.LayerOn)
    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = ConvertToWord(objLayer.Freeze)
    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = ConvertToWord(objLayer.Lock)
    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = ConvertColorToString(objLayer.Color)
    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = objLayer.Linetype
    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = ConvertLineweight(objLayer.Lineweight)
    lngColumn = lngColumn + 1

    If ThisDrawing.Application.Preferences.Output.PlotPolicy Then
      .Cell(lngRow, lngColumn).Range.Text = "ByColor"

      Else
        .Cell(lngRow, lngColumn).Range.Text = objLayer.PlotStyleName
    End If

    lngColumn = lngColumn + 1
    .Cell(lngRow, lngColumn).Range.Text = ConvertToWord(objLayer.Plottable)

    'increment the row counter
    lngRow = lngRow + 1

    'column counter must begin at 1
    lngColumn = 1
  End With
Next objLayer
```

Helper Functions

If you were impatient and attempted to run the preceding code you undoubtedly saw all the errors generated by missing functions. Instead of writing a 0 or a 1 for a No or Yes value, you want to use the actual words No and Yes. Also, you want to convert the color numbers returned by AutoCAD to the actual color name for the first nine colors. Lastly, you want to convert the lineweight integer values returned by AutoCAD to something more meaningful, like `ByBlock` and `ByLayer`.

ConvertColorToString Function

```
Public Function ConvertColorToString(Color As Integer) As String
  Select Case Color
    Case 0
      ConvertColorToString = "ByBlock"
    Case 1
      ConvertColorToString = "Red"
    Case 2
      ConvertColorToString = "Yellow"
    Case 3
      ConvertColorToString = "Green"
    Case 4
      ConvertColorToString = "Cyan"
    Case 5
      ConvertColorToString = "Blue"
    Case 6
      ConvertColorToString = "Magenta"
    Case 7
      ConvertColorToString = "White"
    Case 256
      ConvertColorToString = "ByLayer"
    Case Else
      ConvertColorToString = CStr(Color)
  End Select
End Function
```

ConvertLineweight Function

```
Public Function ConvertLineweight(Lineweight As Integer) As String
  Select Case Lineweight
    Case acLnWtByBlock
      ConvertLineweight = "ByBlock"
    Case acLnWtByLayer
      ConvertLineweight = "ByLayer"
    Case acLnWtByLwDefault
      ConvertLineweight = "Default"
    Case Else
      ConvertLineweight = Format(Lineweight / 100, "#.00") & "mm"
  End Select
End Function
```

ConvertToWord Function

```
Public Function ConvertToWord(Value As Boolean) As String
  Select Case Value
    Case acTrue
      ConvertToWord = "Yes"
    Case acFalse
      ConvertToWord = "No"
  End Select
End Function
```

Each of the conversions used in this application are performed by one of the helper functions. You need to include these functions in your standard `.bas` module created earlier in this exercise.

Adjust the Fonts

You can adjust the fonts used in the report to any font available within Microsoft Word. The following example demonstrates changing the Normal, Header, Footer, and Page Number sections' font to Tahoma and various font sizes:

■ **Note** You'll add the following code snippets to the `cmdCreateLayerReport_Click` event routine.

```
With mobjWord.ActiveDocument
  'set text font and size for each section
  .Styles("Normal").Font.Name = "Tahoma"
  .Styles("Normal").Font.Size = "8"
  .Styles("Header").Font.Name = "Tahoma"
  .Styles("Header").Font.Size = "9"
  .Styles("Header").Font.Bold = True
  .Styles("Footer").Font.Name = "Tahoma"
  .Styles("Footer").Font.Size = "9"
  .Styles("Footer").Font.Italic = True
  .Styles("Page Number").Font.Name = "Tahoma"
  .Styles("Page Number").Font.Size = "11"
  .Styles("Page Number").Font.Bold = True
End With
```

Sort the Table

Your client may request the list of layers to be sorted. The `Microsoft Word Table` object provides a method to accomplish this task, as shown in the following code, based upon whether the user selected the Sorted option in the dialog box:

```
  If chkSorted.Value Then
    'sort the table of layers
    With mobjTable
      .Select
      .Sort ExcludeHeader:=True, FieldNumber:="Column 1", SortFieldType _
        :=wdSortFieldAlphanumeric, SortOrder:=wdSortOrderAscending
    End With
  End If
```

As you might imagine, you can sort any column in either ascending or descending order, adding flexibility to your layer reporting tool. In the preceding example, the Name column is sorted in ascending order.

Autofit Column Text

With all the data in each column and row, you want to adjust the table for the longer strings in each column. Microsoft Word provides an `AutoFit` method on the `Table` object to allow for autofitting the width of the column to the longest data string in each column.

```
  With mobjTable
    .AutoFitBehavior wdAutoFitContent
    .Columns(1).AutoFit      'Layer name
    .Columns(2).AutoFit      'Off/On status
    .Columns(3).AutoFit      'Frozen/Thawed status
    .Columns(4).AutoFit      'Locked/Unlocked status
    .Columns(5).AutoFit      'Layer color
    .Columns(6).AutoFit      'Layer linetype
    .Columns(7).AutoFit      'Layer lineweight
    .Columns(8).AutoFit      'Layer Plot style
    .Columns(9).AutoFit      'Plot this layer?
  End With
```

Add Page Header and Footer

A page header and footer allow you to add some other useful information, such as the path of the drawing or the report creation date. The example illustrates adding a page header and footer to our report.

```
  With mobjDoc
    'set the document header
    .Sections(1).Headers(wdHeaderFooterPrimary).Range.Text = _
        "Layer Report for " & Application.Documents(cboDrawingName.Value).Path _
        & "\" & Application.Documents(cboDrawingName.Value).Name

    'set the document footer
    .Sections(1).Footers(wdHeaderFooterPrimary).Range.Text = _
     "Date Created: " & Date & vbTab & "Total # of Layers: " &_
    Application.Documents(cboDrawingName.Value).Layers.Count
    .Sections(1).Footers(wdHeaderFooterPrimary).PageNumbers.Add _
    wdAlignPageNumberRight
  End With
```

Print the Report

After all the complexity you have endured up to this point, you deserve a break. The act of printing out the report you just created requires simply a single line of code in the `cmdPrintLayerReport_Click` event routine:

```
mobjWord.ActiveDocument.PrintOut
```

Summary

In this chapter we have successfully built a layer reporting application using AutoCAD and Microsoft Word. The chapter demonstrated a lot of functionality and should serve as an excellent starting point for your own automation project. I hope this chapter will serve as an inspiration to delve into what is possible using the automation tools at your disposal.

APPENDIX A

AutoCAD Object Summary

This appendix contains a reference to the full AutoCAD object model. The first part of the appendix provides information about properties, methods, and events that are common to a number of the AutoCAD objects. The second and major part of the appendix explains the methods, properties, and events of each object and collection in alphabetical order.

AutoCAD Collections

The AutoCAD object model has the following collections:

```
AcadBlock                    AcadModelSpace
AcadBlocks                   AcadPaperSpace
AcadDictionaries             AcadPlotConfigurations
AcadDictionary               AcadPopupMenu
AcadDimStyles                AcadPopupMenus
AcadDocuments                AcadRegisteredApplications
AcadFileDependencies         AcadSelectionSet
AcadGroup                    AcadSelectionSets
AcadGroups                   AcadTextStyles
AcadHyperlinks               AcadToolbar
AcadLayers                   AcadToolbars
AcadLayouts                  AcadUCSs
AcadLineTypes                AcadViewports
AcadMenuBar                  AcadViews
AcadMenuGroups
```

Common Collection Methods

All the collections in the AutoCAD object model support the `Item` method, which is used to select an item from the collection.

Name	Returns	Description
`Item`	Object	Returns the object at the given index in the collection. Parameter: `Index As Variant` (Integer or String). If the `Index` value is a String, it must match an existing object's name.

Common Collection Property

All collections in the AutoCAD object model support the following property.

Name	Returns	Description
`Count`	Long	Gets the number of items in the collection.

The `Application` Property

Just one property is common to all the objects in the AutoCAD object model: the `Application` property.

Name	Returns	Description
`Application`	`AcadApplication`	Gets the `AcadApplication` object, which represents the application's frame controls and path settings and provides the means to navigate down the object hierarchy. This property's value is read-only.

The `AcadObject` Object

This object represents methods and properties common to the following objects:

`Acad3DFace`	`AcadDictionaries`
`Acad3DPolyline`	`AcadDictionary`
`Acad3DSolid`	`AcadDim3PointAngular`
`AcadArc`	`AcadDimAligned`
`AcadAttribute`	`AcadDimAngular`
`AcadAttributeReference`	`AcadDimArcLength`
`AcadBlock`	`AcadDimDiametric`
`AcadBlockReference`	`AcadDimension`
`AcadBlocks`	`AcadDimOrdinate`
`AcadCircle`	`AcadDimRadial`

AcadDimRadialLarge

AcadDimRotated

AcadDimStyle

AcadDimStyles

AcadDynamicBlockReferenceProperty

AcadEllipse

AcadEntity

AcadExternalReference

AcadGroup

AcadGroups

AcadHatch

AcadLayer

AcadLayers

AcadLayout

AcadLayouts

AcadLeader

AcadLine

AcadLineType

AcadLineTypes

AcadLWPolyline

AcadMInsertBlock

AcadMLine

AcadModelSpace

AcadMText

AcadOle

AcadPaperSpace

AcadPlotConfiguration

AcadPlotConfigurations

AcadPoint

AcadPolyfaceMesh

AcadPolygonMesh

AcadPolyline

AcadPViewport

AcadRasterImage

AcadRay

AcadRegion

AcadRegisteredApplication

AcadRegisteredApplications

AcadShape

AcadSolid

AcadSortentsTable

AcadSpline

AcadTable

AcadTableStyle

AcadText

AcadTextStyle

AcadTextStyles

AcadTolerance

AcadTrace

AcadUCS

AcadUCSs

AcadView

AcadViewport

AcadViewports

AcadViews

AcadXLine

AcadXRecord

Common AcadObject Methods

The following table presents common AcadObject methods.

Name	Description
Delete	Deletes a specified object. If an object in a collection is deleted, all the remaining items are reassigned a new index based on the current count. Note that you can't delete collections, so you can't use this method for collections without creating an error.
GetExtensionDictionary	Returns AcadDictionary and gets the extension dictionary associated with an object. If an object doesn't have an extension dictionary, this method will create one for that object and return it.
GetXData	Gets the extended data (XData), that is, instance-specific data, associated with an object. A NULL string value for the AppName parameter will return all the data attached to the object, regardless of the application that created it. Supplying an application name will return only the data that was created by the specified application. Parameters: AppName As String, XDataType As Variant (array of Integers), and XDataValue As Variant (array of Variants).
SetXData	Sets the extended data (XData), that is, instance-specific data associated with an object. Parameters: XDataType As Variant (array of Integers) and XDataValue As Variant (array of Variants).

Common AcadObject Properties

The following table presents common AcadObject properties.

Name	Returns	Description
Application	Object	Gets the Application object. This property's value is read-only. The Application object represents the application's frame controls and path settings and provides the means to navigate down the object hierarchy.
Document	Object	Gets the document or drawing in which the object belongs. This property's value is read-only.
Handle	String	Gets the handle of an object. This property's value is read-only. An object ID and a handle are the two ways of referencing an object. A handle stays the same in a drawing for the lifetime of the object.
HasExtensionDictionary	Boolean	Determines whether the object has an extension dictionary associated with it. This property's value is read-only.
ObjectID	Integer	Gets the object ID of the object. This property's value is read-only. You can use an object ID or a handle to reference an object. It's better to use a handle unless you plan to work with certain ObjectARX functions that require an object ID.
ObjectName	String	Gets the AutoCAD class name of the object. This property's value is read-only.
OwnerID	Integer	Gets the object ID of the owner (i.e., the parent object). This property's value is read-only.

Common AcadObject Event

Apart from the AcadApplication and AcadDocument objects, which expose a variety of events, AutoCAD provides only one other event, the Modified event. This event is common to all the objects listed previously. When coding in VBA, you must provide an event handler for all objects enabled for the Modified event; otherwise, your application may crash. No events are fired while a modal dialog box is displayed.

Event	Description
Modified	Triggered whenever an object is modified. Note that setting the value of a property even if that value remains unchanged is considered to be a modification. Parameter: Entity As AcadEntity, the object in the drawing that is modified. The AcadEntity object is covered in the next section.

The AcadEntity Object

The AcadEntity object represents a generic object that exposes all the methods and properties common to a set of AutoCAD objects known as the *Drawing Objects*. These objects, which define all the drawing capabilities (including drawing planar and solid objects, inserting text and hyperlinks, and linking external documents) are as follows:

Acad3DFace	AcadDimRotated
Acad3DPolyline	AcadEllipse
Acad3DSolid	AcadEntity
AcadArc	AcadExternalReference
AcadAttribute	AcadHatch
AcadAttributeReference	AcadLeader
AcadBlockReference	AcadLine
AcadCircle	AcadLWPolyline
AcadDim3PointAngular	AcadMInsertBlock
AcadDimAligned	AcadMLine
AcadDimAngular	AcadMText
AcadDimArcLength	AcadOle
AcadDimDiametric	AcadPoint
AcadDimension	AcadPolyfaceMesh
AcadDimOrdinate	AcadPolygonMesh
AcadDimRadial	AcadPolyline
AcadDimRadialLarge	AcadPViewport

AcadRasterImage	AcadTable
AcadRay	AcadText
AcadRegion	AcadTolerance
AcadShape	AcadTrace
AcadSolid	AcadXline
AcadSpline	

Common AcadEntity Methods

In addition to the methods inherited from the AcadObject object, the AcadEntity object supports the following methods.

Name	Returns	Description
ArrayPolar	Variant	Creates an array of selected objects in a polar pattern. The distance is determined from the array's center point to a reference point on the last object selected, which in turn depends on the type of object previously selected. AutoCAD uses the center point of a circle or arc, the insertion point of a block or shape, the start point of text, and one end point of a line or trace. Note that you can't use this method for the AcadAttributeReference object. Note also that you can't use this method while iterating through a collection. Parameters: NumberOfObjects As Integer, AngleToFill As Double, and CenterPoint As Variant. The NumberOfObjects parameter specifies the number of objects to be created in the polar array. This must be a positive integer greater than 1. The AngleToFill parameter specifies the angle to fill in radians. A positive value indicates counterclockwise rotation. This parameter can't be equal to 0. The CenterPoint parameter specifies the 3-D WCS coordinates indicating the center point for the polar array.
ArrayRectangular	Variant	Creates an array of selected objects in a rectangular pattern. AutoCAD builds the rectangular array along a baseline defined by the current snap rotation angle, which is 0 by default. Note that you can't use this method for the AcadAttributeReference object. Note also that you can't use this method while iterating through a collection. Parameters: NumberOfRows As Integer, NumberOfColumns As Integer, NumberOfLevels As Integer, DistBetweenRows As Double, DistBetweenCols As Double, and DistBetweenLevels As Double. The NumberOfRows parameter specifies the number of rows in the rectangular array. This must be a positive number. If NumberOfColumns is 1, then NumberOfRows must be greater than 1. The NumberOfColumns parameter specifies the number of columns in the rectangular array. This must be a positive number. If NumberOfRows is 1, then NumberOfColumns must be greater than 1. The NumberOfLevels parameter specifies the number of levels in a 3-D array. The DistBetweenRows parameter specifies the distance between the rows. If positive, rows are added upward from the base entity; otherwise, they're added downward. The DistBetweenCols parameter specifies the distance between the columns. If positive, columns are added to the right of the base entity; otherwise, they're added to the left. The DistBetweenLevels parameter specifies the distance between the

Name	Returns	Description
CopyObject		Duplicates the given object to the same location. Note that you can't use this method for the AcadAttributeReference object. Note also that you can't use this method while iterating through a collection.
GetBoundingBox		Gets two points of a box enclosing the specified object. Parameters: MinPoint As Variant and MaxPoint As Variant (both of these are three-element arrays of Doubles).
Highlight		Sets the highlight status for the given object or for all objects in a given selection set. Note that you'll see the changes only once the drawing is updated. Parameter: HighlightFlag As Boolean.
IntersectWith	Variant	Returns the points at which one object intersects another object in the drawing as an array of Doubles. The object supplied as the parameter can be any drawing object. However, this method isn't supported by the AcadPolygonMesh and AcadPViewport objects. Parameters: IntersectObject As Object and option As AcExtendOption. For a list of the values of the AcExtendOption enumerated type, see Appendix B.
MirrorObject		Creates a mirror image copy of a planar object around an axis, defined by two points. This method places the reflected image into the drawing and retains the original object. To remove the original object, use the Delete method. To manage the reflection properties of text objects, use the MIRRTEXT system variable. Note that you can't use this method for the AcadAttributeReference object. Note also that you can't use this method while iterating through a collection. Parameters: Point1 As Variant and Point2 As Variant.
Mirror3D	Object	Creates a mirror image copy of a planar object reflected in a plane, defined by three points. AutoCAD checks to see whether the object to be copied owns any other object. If it does, it performs a copy on those objects as well. The process continues until all owned objects have been copied. Note that you can't use this method for the AcadAttributeReference object. Note also that you can't use this method while iterating through a collection. Parameters: Point1 As Variant, Point2 As Variant, and Point3 As Variant.
Move		Moves an object along a vector, defined by two points. Parameters: FromPoint As Variant and ToPoint As Variant.
Rotate		Rotates an object around a base point. Parameters: BasePoint As Variant and RotationAngle As Double. The RotationAngle parameter should be specified in radians.
Rotate3D		Rotates an object around an axis in 3-D space, defined by two points, which become the axis of rotation. Parameters: Point1 As Variant, Point2 As Variant, and RotationAngle As Double. The RotationAngle parameter should be in radians.
ScaleEntity		Scales an object equally in the X, Y, and Z directions. Parameters: BasePoint As Variant and ScaleFactor As Double. The BasePoint parameter is a set of 3-D WCS coordinates and is the only common point in the original and scaled system. The ScaleFactor parameter determines the level of scaling. A value greater than 1 enlarges the object. A value between 0 and 1 shrinks the object. The scale factor must be greater than 0.
TransformBy		Moves, scales, and rotates an object given a 4×4 transformation matrix. See the AutoCAD documentation for more information. Parameter: TransformationMatrix As Variant (a 4×4 array of Doubles).
Update		Updates the object to the drawing screen.

Common AcadEntity Properties

In addition to the properties inherited from the AcadObject, the AcadEntity object supports the following properties.

Name	Returns	Description
Color	AcColor	Gets or sets the color of an entity. For a list of possible values for the AcColor enumerated type, see Appendix B.
EntityName	String	Gets the class name of the object. Retained for backward compatibility only. Use the VBA TypeName function instead. This property's value is read-only.
EntityType	AcEntityName	Gets the entity type of the object. Retained for backward compatibility only. Use the VBA TypeOf keyword instead. This property's value is read-only. For a list of the values for the AcEntityName enumerated type, see Appendix B.
Hyperlinks	AcadHyperlinks	Gets the AcadHyperlinks collection for an entity. This property's value is read-only.
Layer	String	Gets or sets the layer for an entity.
Linetype	String	Gets or sets the linetype of an entity.
LinetypeScale	Double	Gets or sets the linetype scale of an entity. The default value is 1.0.
Lineweight	AcLineWeight	Gets or sets the lineweight of an individual entity or the default lineweight for the drawing. For a list of possible values for the AcLineWeight enumerated type, see Appendix B.
PlotStyleName	String	Gets or sets the object's plot-style name.
TrueColor	AcCmColor	Gets or sets the RGB color index, color name, or Pantone color book settings.
Visible	Boolean	Specifies whether the object (or application) is visible.

■ **Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadDimension Object

The AcadDimension object represents a generic object with a set of properties that are common to the nine dimension objects, namely, the following:

AcadDim3PointAngular	AcadDimOrdinate
AcadDimAligned	AcadDimRadial
AcadDimAngular	AcadDimRadialLarge
AcadDimArcLength	AcadDimRotated
AcadDimDiametric	

In addition, it inherits all the methods and properties of the AcadEntity and AcadObject objects.

Common AcadDimension Properties

The AcadDimension object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
DecimalSeparator	String	Gets or sets the character used as the decimal separator for dimension values. The initial value for this property is the period (.), though any character will be accepted as a valid value for this property. Use this property only when the UnitsFormat property is set to acDimDecimal. This property overrides the value of the DIMSEP system variable.
Normal	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the dimension object.
Rotation	Double	Gets or sets the rotation angle (in radians) for the dimension line, relative to the X-axis with positive values going counterclockwise when viewed along the Z-axis toward the origin.
ScaleFactor	Double	Gets or sets the object's relative X scale factor. The initial value for this property is 1.0000.
StyleName	String	Gets or sets the name of the object's style. Use the AcadDimStyle object to change the attributes of a given dimension style. The name given must already be defined in the drawing.
SuppressTrailingZeros	Boolean	Gets or sets whether leading/trailing zeros are suppressed in the object's values. It overrides the value of the DIMZIN system variable. The initial value for both these properties is False.
TextColor	AcColor	Gets or sets the color of the text. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in the default color) or acByLayer (where AutoCAD draws objects in the color specified for the layer). For a list of possible values for the AcColor enumerated type, see Appendix B.
TextGap	Double	Gets or sets the distance between the text and the dimension line when the dimension line is broken to make room for the text. A negative value creates basic dimensioning, that is, dimension text with a box around it. This property overrides the value of the DIMGAP system variable.
TextHeight	Double	Gets or sets the height of the text. The initial value for this property is 0.1800.
TextMovement	AcDimTextMovement	Gets or sets how text is drawn when it's moved. The initial value for this property is acMoveTextNoLeader. For a list of possible values for the AcDimTextMovement enumerated type, see Appendix B.

Continued

Name	Returns	Description
TextOverride	String	Gets or sets the text string, which has a maximum length of 256 characters. The user string replaces the calculated dimension value. You can revert to the calculated dimension value by setting the text to a NULL string (""). You can append or prefix text to the primary dimension value by using a closed set of brackets (<>) to represent the value. The primary dimension value will replace the brackets when the string is displayed.
TextPosition	Variant	Gets or sets the position of the text as a set of WCS coordinates.
TextPrefix	String	Gets or sets the prefix of the dimension value. The initial value for this property is NULL. To turn off an established prefix, set this property equal to a single period.
TextRotation	Double	Gets or sets the angle of the text in radians. The valid range is 0 to 6.28.
TextStyle	String	Gets or sets the style of the text. The initial value for this property is STANDARD.
TextSuffix	String	Gets or sets the suffix of the dimension value. The initial value for this property is NULL. To turn off an established suffix, set this property equal to a single period.
ToleranceDisplay	AcDimToleranceMethod	Gets or sets whether the tolerances are displayed with the text. It overrides the value of the DIMTOL system variable. The initial value for this property is acTolNone. For a list of possible values for the AcDimToleranceMethod enumerated type, see Appendix B.
ToleranceHeightScale	Double	Gets or sets the scale factor of the text's height for the tolerance values relative to the object's general text height. The initial value for this property is 1.0000. This property is available only when the ToleranceDisplay property is set to any value other than acTolNone.
ToleranceJustification	AcDimToleranceJustify	Gets or sets the vertical justification of tolerance values relative to that of the object's general text. The initial value for this property is acTolMiddle. This property is available only when the ToleranceDisplay property is set to any value other than acTolNone. For a list of possible values for the AcDimToleranceJustify enumerated type, see Appendix B.
ToleranceLowerLimit	Double	Gets or sets the minimum tolerance for text. The initial value for this property is 0.0000. This property is available only when the ToleranceDisplay property is set to acTolDeviation or acTolLimits.
TolerancePrecision	AcDimPrecision	Gets or sets the precision of tolerance values in the primary dimensions. This property isn't available when ToleranceDisplay is set to acTolNone. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.

Name	Returns	Description
`ToleranceSuppressLeadingZeros`	Boolean	Gets or sets whether leading zeros in tolerance values are suppressed. It overrides the value of the `DIMTZIN` system variable. The initial value for this property is `False`.
`ToleranceSuppressTrailingZeros`	Boolean	Gets or sets whether trailing zeros in tolerance values are suppressed. It overrides the value of the `DIMTZIN` system variable. The initial value for this property is `False`.
`ToleranceUpperLimit`	Double	Gets or sets the maximum tolerance for text. The initial value for this property is 0.0000. This property is available only when the `ToleranceDisplay` property is set to `acTolSymmetrical`, `acTolDeviation`, or `acTolLimits`.
`VerticalTextPosition`		Gets or sets the vertical position of the text relative to the dimension line. It overrides the value of the `DIMTAD` system variable. The initial value for this property is `acVertCentered`. For a list of possible values for the `AcDimVerticalJustification` enumerated type, see Appendix B.

Note Because this object inherits from `AcadObject`, it supports the `Modified` event.

AutoCAD Object Reference

The following sections detail the remaining AutoCAD object model objects derived from the base class objects detailed earlier in this appendix.

Acad3DFace Object

The `Acad3DFace` object represents a 3-D triangle or a quadrilateral plane section. You create it using the `Add3DFace` method of the `AcadBlock`, `AcadModelSpace`, or `AcadPaperSpace` object. You can specify different Z coordinates for each corner point of a `Acad3DFace` object, and you can build solid objects by joining 3-D faces together (e.g., a cube consists of six 3-D faces joined together). You control which edges of the face are visible through the `SetInvisibleEdge` method, which allows accurate modeling of objects with holes.

Acad3DFace Object Methods

The `Acad3DFace` object inherits all the methods of the `AcadEntity` and `AcadObject` objects. It also supports the following methods.

Name	Returns	Description
GetInvisibleEdge	Boolean	Gets the visibility setting for an edge of a Acad3DFace object at a given index. It's True if the edge is invisible and False otherwise. Parameter: Index As Integer.
SetInvisibleEdge		Sets the visibility state of an edge at a given index. To see any changes in visibility, the drawing must be regenerated. Parameters: Index As Integer and State As Boolean. The Index parameter must be in the range 0 to 3. Set the State parameter to True for an invisible edge and False otherwise.

Acad3DFace Object Properties

The Acad3DFace object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Coordinate	Variant	Gets or sets the coordinate of a single vertex in the object. This will replace the specified vertex of the object. Use standard array-handling techniques to process the values contained in this property. It returns a three-element array of Doubles containing 3-D coordinates in WCS. Note that the Z coordinate will default to 0 on the active UCS. Parameter: Index As Integer (the index in the zero-based array of vertices for the vertex you want to set or query).
Coordinates	Variant	Gets or sets the coordinates for each vertex in the object. This will replace any existing coordinates for the specified object. Use standard array-handling techniques to process the coordinates contained in this property. Note that you can't change the number of coordinates in the object by using this property; you can change only the location of existing coordinates. It returns a three-element array of Doubles containing 3-D coordinates in WCS. Note also that he Z coordinate will always default to 0 on the active UCS.
VisibilityEdge1	Boolean	Gets or sets whether edge 1 is visible. For this property and the three that follow, True means the edge is visible and False means the edge isn't visible.
VisibilityEdge2	Boolean	Gets or sets whether edge 2 is visible.
VisibilityEdge3	Boolean	Gets or sets whether edge 3 is visible.
VisibilityEdge4	Boolean	Gets or sets whether edge 4 is visible. In the case of a triangle, this property will still return a value, because a triangle is defined as a quadrilateral in which point 3 is equal to point 4.

■Note Because this object inherits from AcadObject, it supports the Modified event.

Acad3DPolyline Object

The Acad3DPolyline object represents a 3-D polyline of straight-line segments. You create it using the Add3DPoly method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

Acad3DPolyline Object Methods

The Acad3DPolyline object inherits all the methods of the AcadEntity and AcadObject objects. It also supports the following methods.

Name	Returns	Description
AppendVertex		Appends a vertex to the polyline, which is an array of 3-D coordinates. Parameter: vertex As Variant (a three-element array of Doubles).
Explode	Variant	Explodes the polyline into its constituent lines and returns them as an array of Objects.

Acad3DPolyline Object Properties

The Acad3DPolyline object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Closed	Boolean	Gets or sets whether the polyline is closed (True) or open (False).
Coordinate	Variant	Gets or sets the coordinate of a single vertex in the object. This will replace the specified existing vertex of the object. Use standard array-handling techniques to process the values contained in this property. It returns a Variant (three-element array of Doubles). These coordinates can be converted to and from other coordinate systems using the TranslateCoordinates method. Parameter: Index As Integer (the index in the zero-based array for the vertex you want to set or query).
Coordinates	Variant	Gets or sets the coordinates for each vertex in the object. This will replace any existing coordinates for the specified object. Use standard array-handling techniques to process the coordinates contained in this property. When you set the coordinates for a polyline, if you supply fewer coordinates than the object currently possesses, the polyline will be truncated. Any fit points applying to the truncated vertices will also be truncated. If you supply more coordinates than the object currently possesses, the extra vertices will be appended to the polyline. It returns the vertex points as a Variant (a three-element array of Doubles) containing the X, Y, and Z coordinates for the vertices in WCS. These coordinates can be converted to and from other coordinate systems using the TranslateCoordinates method.
Type	Ac3DPolylineType	Gets or sets the type of the 3-D polyline. You can use this property to convert a 3-D polyline into a spline. For a list of the values of the Ac3DPolylineType enumerated type, see Appendix B.

■ **Note** Because this object inherits from AcadObject, it supports the Modified event.

Acad3DSolid Object

The Acad3DSolid object represents a solid object with free-form surface support. You don't create general Acad3DSolid objects, as there's no Add3DSolid method supported by the AcadBlock, AcadModelSpace, and AcadPaperSpace objects. Instead, you use this object to represent a whole series of 3-D shapes for which no specific AutoCAD object exists: boxes, cones, cylinders, spheres, wedges, tori (donuts), and others. You can then combine these shapes to create more complex solids by joining or subtracting them or finding their intersecting (overlapping) volume. You can also create solids by sweeping a 2-D object along a path or revolving it around an axis.

To create an Acad3DSolid object, use one of the following methods, all of which are common to the AcadBlock, AcadModelSpace, and AcadPaperSpace objects:

AddBox	AddExtrudedSolidAlongPath
AddCone	AddRevolvedSolid
AddCylinder	AddSphere
AddEllipticalCone	AddTorus
AddEllipticalCylinder	AddWedge
AddExtrudedSolid	

See the section about the AcadBlock object for more details on each of these.

Acad3DSolid Object Methods

The Acad3DSolid object inherits all the methods of the AcadEntity and AcadObject objects. It also supports the following methods.

Name	Returns	Description
Boolean		Performs a destructive operation returning the union, intersection, or subtraction between the object and another Acad3DSolid or AcadRegion object. Parameters: Operation As AcBooleanType and SolidObject As Acad3DSolid. For a list of possible values for the AcBooleanType enumerated type, see Appendix B.
CheckInterference	Acad3DSolid	Checks for interference between the object and another Acad3DSolid object and creates an object representing the interference if required. Parameters: Object As Acad3DSolid and CreateInterferenceSolid As Boolean. The second parameter is True if an interference solid is required and False otherwise.
SectionSolid	AcadRegion	Creates an AcadRegion object representing the intersection of the object with the plane containing the specified points. Parameters: Point1 As Variant, Point2 As Variant, and Point3 As Variant.

Name	Returns	Description
SliceSolid	Acad3DSolid	Creates an Acad3DSolid object representing the portion of the object on one side of the plane containing the specified points. Parameters: Point1 As Variant, Point2 As Variant, Point3 As Variant, and Negative As Boolean. The fourth parameter determines whether the resulting solid should be returned on the negative side of the plane, which is True if this is the case and False otherwise. The original Acad3DSolid object retains the positive side of the slice.

Acad3DSolid Object Properties

The Acad3DSolid object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Centroid	Variant	Gets the center of area or mass for a solid as a 2-D coordinate. It returns a two-element array of Doubles. This property is read-only.
MomentOfInertia	Variant	Gets the moment of inertia for the solid as a 3-D coordinate. It returns a three-element array of Doubles. This property is read-only.
PrincipalDirections	Variant	Gets the principal directions of the solid as X, Y, and Z coordinates calculated on the current coordinate system. This property is read-only.
PrincipalMoments	Variant	Gets the principal moments property of the solid as X, Y, and Z coordinates calculated on the current coordinate system. This property's value is read-only.
ProductOfInertia	Variant	Gets the product of inertia of the solid as X, Y, and Z coordinates calculated on the current coordinate system. This property's value is read-only.
RadiiOfGyration	Variant	Gets the radius of gyration of the solid as X, Y, and Z coordinates calculated on the current coordinate system. This property's value is read-only.
Volume	Double	Gets the object's volume. This property's value is read-only.

Note Because this object inherits from AcadObject, it supports the Modified event.

AcadApplication Object

The AcadApplication object represents an instance of the AutoCAD application and is accessed in VBA by using ThisDrawing.Application. The AcadApplication object is also the global object for the ActiveX interface, and all methods and properties are available in the global namespace. The object supports 19 methods, which control the loading or listing of the currently loaded external applications and interface objects, and 20 properties, which reflect the properties of the main application window. In addition, more than 20 events are associated with the AcadApplication object.

AcadApplication Object Methods

The AcadApplication object supports the following methods.

Name	Returns	Description
Eval		Executes a line of VBA code without the need to create a macro. Parameter: Expression As String.
GetAcadState	AcadState	Returns an AcadState object that will monitor the state of the AutoCAD application from other applications.
GetInterfaceObject	Object	Loads a program ID into AutoCAD as an in-process server. Although the object will be loaded into AutoCAD, it won't show up in its type library. The object will have its own type library. This method lets you connect to an ActiveX Automation server. Parameter: ProgID As String.
ListARX	Variant	Returns the currently loaded AutoCAD ARX applications as an array. It returns empty if none are loaded.
LoadARX		Loads an AutoCAD ARX application. Parameter: Name As String.
LoadDVB		Loads an AutoCAD VBA project file. Parameter: Name As String.
Quit		Closes the drawing file and the AutoCAD application. If AutoCAD is in the middle of a command, an error message will be generated. If the modified document hasn't been saved, a prompt message will be generated.
RunMacro		Runs a VBA macro. Parameter: MacroPath As String. If the full path name isn't specified, AutoCAD will use its search path to find the macro or search all currently loaded projects. Syntax: RunMacro "MyProject.Module1".
UnloadARX		Unloads an unlocked AutoCAD ARX application. Parameter: Name As String. Don't attempt to unload the file acadvba.arx.
UnloadDVB		Unloads an AutoCAD VBA project file. Parameter: Name As String.
Update		Updates the screen.
ZoomAll		Zooms the current active model space or paper space viewport to display the entire drawing. For a 2-D view, AutoCAD displays the whole drawing or the current extent, whichever is greater. For a 3-D view, this method is equivalent to ZoomExtents.
ZoomCenter		Zooms the current active model space or paper space viewport as specified by the parameters. Parameters: Center As Variant and Magnify As Double.
ZoomExtents		Zooms the current active model space or paper space viewport to the extent of the drawing.

Name	Returns	Description
ZoomPickWindow		Zooms the current active model space or paper space viewport to a window defined by points picked on the screen.
ZoomPrevious		Zooms the current active model space or paper space viewport to its previous state.
ZoomScaled		Zooms the current active model space or paper space viewport to the specified scale factor. Parameters: `Scale As Double` and `ScaleType As AcZoomScaleType`. For the values of the second parameter, see Appendix B.
ZoomWindow		Zooms the current active model space or paper space viewport as specified by the parameters. Parameters: `LowerLeft As Variant` and `UpperRight As Variant`. Parameters are three-element arrays of Doubles specifying WCS coordinates.

`AcadApplication` Object Properties

The `AcadApplication` object supports the following properties.

Name	Returns	Description
ActiveDocument	AcadDocument	Gets or sets a reference to the active document.
Application	AcadApplication	Gets the `AcadApplication` object, which represents the application's frame controls and path settings and provides the means to navigate down the object hierarchy.
Caption	String	Gets the text seen by the user in the window caption bar. This property is read-only. Note that if the caption is modified using the Win32 API function `SetWindowText`, the caption property will return the modified caption text.
Documents	AcadDocuments	Gets the `Documents` collection. This property is read-only.
FullName	String	Gets the full name, including the path, of the application. This property's value is read-only.
Height	Integer	Gets or sets the height of the application window in pixels.
LocaleID	Long	Gets the regional settings for the session as defined by the Windows 95 or Windows NT operating systems. This property's value is read-only.
MenuBar	AcadMenuBar	Gets the `AcadMenuBar` object for the session. This property's value is read-only.
MenuGroups	AcadMenuGroups	Gets the `MenuGroups` collection for the session.
Name	String	Gets the name of the application. This property's value is read-only.
Path	String	Gets the path of the application without the file name. This property's value is read-only.
Preferences	AcadPreferences	Gets the application's `AcadPreferences` object. This property's value is read-only.

Continued

Name	Returns	Description
StatusID	Boolean	Gets the current active status of the viewport. It's True if the specified viewport is active and False otherwise. This property's value is read-only. Parameter: VportObj As AcadViewport.
VBE	VBAIDE	Gets the VBAIDE extensibility object. This isn't available if the acadvba.arx application hasn't been loaded. This property's value is read-only.
Version	String	Gets the version of the AutoCAD application. This property's value is read-only. For AutoCAD 2000, the value is 15.0.
Visible	Boolean	Gets or sets whether the application is visible.
Width	Integer	Gets or sets the width of the application window in pixels.
WindowLeft	Integer	Gets or sets the distance between the left edge of the application window and the left edge of the screen in pixels. This distance will establish the X coordinate of the upper-left corner (0,0) of the application.
WindowState	AcWindowState	Gets or sets the state of the application window. Possible values are acMin (minimized), acMax (maximized), or acNorm (normal).
WindowTop	Integer	Gets or sets the distance between the top of the application window and the top of the screen in pixels. This distance will establish the Y coordinate of the upper-left corner (0,0) of the application.

AcadApplication Object Events

The following events are triggered during various stages in the lifetime of the AcadApplication object. Note that no events will be fired while a modal dialog box is displayed.

Event	Description
AppActivate	Occurs before the main application window is activated.
AppDeactivate	Occurs before the main application window is deactivated.
ARXLoaded	Occurs after an ObjectARX application is loaded. Parameter: FullPathName As String.
ARXUnloaded	Occurs after an ObjectARX application is unloaded. Parameter: FullPathName As String.
BeginCommand	Occurs after a command is issued but before it is completed. Parameter: CommandName As String.
BeginFileDrop	Occurs when a file is dropped on the main application window. Parameters: FileName As String and Cancel As Boolean. If Cancel is set to True, this aborts the loading of the drawing, LISP file, or ARX file; otherwise, loading is allowed to continue.
BeginLISP	Occurs after a request to evaluate a LISP expression is received. Parameter: FirstLine As String. FirstLine won't have any case conversion of the alpha characters.
BeginModal	Occurs before a modal dialog box is displayed. You should never issue a message box from within an event handler for the BeginModal event.

Event	Description
`BeginOpen`	Occurs after a request to open an existing drawing is received. Parameter: `Filename As String`.
`BeginPlot`	Occurs after a request to print a drawing is received. Parameter: `DrawingName As String`.
`BeginQuit`	Occurs just before the session ends. Parameter: `Cancel As Boolean`.
`BeginSave`	Occurs after a request to save a drawing is received. Parameter: `Filename As String`.
`EndCommand`	Occurs after a command is completed. Parameter: `CommandName As String`.
`EndLISP`	Occurs after a LISP expression is executed.
`EndModal`	Occurs after a modal dialog box is dismissed.
`EndOpen`	Occurs after an existing drawing is opened. Parameter: `Filename As String`.
`EndPlot`	Occurs after a document is sent to the printer. Parameter: `DrawingName As String`.
`EndSave`	Occurs after a drawing is saved. Parameter: `Filename As String`.
`LISPCancelled`	Occurs when the evaluation of a LISP expression is canceled.
`NewDrawing`	Occurs before a drawing is created.
`SysVarChanged`	Occurs when the value of a system variable is changed. Parameters: `SysVarName As String` and `NewVal As Variant`.
`WindowChanged`	Occurs when the application window is changed. Parameter: `WindowState As Integer` (`acMin`, `acMax`, or `acNorm`).
`WindowMovedOrResized`	Occurs after one document window is moved or resized. Parameters: `HWNDFrame As Long` and `Moved As Boolean`. If `Moved` is set to `True`, the window was moved; otherwise, it was resized.

`AcadArc` Object

The `AcadArc` object represents a circular arc, which is always drawn counterclockwise from the start point to the end point. The `StartPoint` and `EndPoint` properties of an arc are calculated through the `StartAngle`, `EndAngle`, and `Radius` properties. The `AcadArc` object is created using the `AddArc` method of the `AcadBlock`, `AcadModelSpace`, or `AcadPaperSpace` object.

`AcadArc` Object Methods

The `AcadArc` object inherits all the methods of the `AcadEntity` and `AcadObject` objects. It also supports the following method.

Name	Returns	Description
`Offset`	Variant	Creates a new arc by offsetting the current arc by a specified distance, which must be a nonzero number. If the offset is positive, a concentric arc with a larger radius is created. If the offset is negative, a smaller arc is created. The return value isn't an arc but a Variant, which is an array of Objects. Parameter: `Distance As Double`.

AcadArc Object Properties

The AcadArc object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
ArcLength	Double	Gets the length of the arc. This property's value is read-only.
Area	Double	Gets the enclosed area of the arc. This property's value is read-only.
Center	Variant	Gets or sets the center of the arc as an array of the Doubles.
EndAngle	Double	Gets or sets the end angle of the arc in radians.
EndPoint	Variant	Gets the end point of the arc as an array of three Doubles. This property's value is read-only.
Normal	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the arc.
Radius	Double	Gets or sets the radius of the arc.
StartAngle	Double	Gets or sets the start angle of the arc in radians.
StartPoint	Variant	Gets the start point of the arc as an array of three Doubles. This property's value is read-only.
Thickness	Double	Gets or sets the distance the AcadArc object is extruded above or below its elevation, in a Z-axis direction. The default is 0.0.
TotalAngle	Double	Gets the total angle of the arc in radians. This property's value is read-only.

■ **Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadAttribute Object

The AcadAttribute object represents an object appearing as a text string that describes the characteristics of an attribute reference, or, in other words, an attribute definition. You create it using the AddAttribute method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadAttribute Object Methods

The AcadAttribute object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadAttribute Object Properties

The AcadAttribute object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
`Alignment`	`AcAlignment`	Gets or sets the horizontal and vertical alignments of the attribute. For a list of the values for the `AcAlignment` enumeration, see Appendix B.
`Backward`	Boolean	Gets or sets whether the text is backward.
`Constant`	Boolean	Gets or sets whether the attribute has the same value in every occurrence. AutoCAD doesn't prompt for a value of constant attributes.
`FieldLength`	Integer	Gets or sets the field length of the attribute.
`Height`	Double	Gets or sets the height of the attribute.
`HorizontalAlignment`	`AcHorizontalAlignment`	Gets or sets the horizontal alignment of the attribute.
`InsertionPoint`	Variant	Gets the insertion point of the attribute.
`Invisible`	Boolean	Gets or sets whether the attribute is invisible. An invisible attribute will not be displayed or plotted. This property's value is read-only.
`Mode`	Integer	Gets or sets the mode of the attribute definition. Values: `acAttributeModeNormal`, `acAttributeModeInvisible`, `acAttributeModeConstant`, `acAttributeModeVerify`, or `acAttributeModePreset`.
`Normal`	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the attribute.
`ObliqueAngle`	Double	Gets or sets the oblique angle of the attribute and is measured from the vertical axis. The units are radians within the range of –85 to +85 degrees. A positive value denotes a lean toward the right.
`Preset`	Boolean	Gets or sets whether the attribute is preset. A preset attribute sets the attribute to its default value when the user inserts the block.
`PromptString`	String	Gets or sets the prompt string for the attribute that appears when a block containing this attribute is inserted. The default for this string is the `TagString` property.
`Rotation`	Double	Gets or sets the rotation angle (in radians) for the attribute, relative to the X-axis, with positive values going counterclockwise when viewed along the Z-axis toward the origin.
`ScaleFactor`	Double	Gets or sets the scale factor for the attribute. It must be positive. A scale factor greater than 1 enlarges the object. A scale factor from 0 to 1 shrinks the object. The initial value for this property is 1.0.
`StyleName`	String	Gets or sets the name of the style used with the object, the default being the current style.
`TagString`	String	Gets or sets the tag string, which identifies each occurrence of the attribute. The tag string identifies each occurrence of the attribute and may contain any characters other than spaces and exclamation marks. Lowercase letters are changed to uppercase.

Continued

Name	Returns	Description
`TextAlignmentPoint`	Variant	Gets or sets the alignment point for text, returning a three-element array of Doubles. This property's value is read-only when the `Alignment` property is set to `acAlignmentLeft`.
`TextGenerationFlag`	`AcTextGenerationFlag`	Gets or sets the attribute text generation flag. Values: `acTextFlagBackward` and `acTextFlagUpsideDown`. The values can be combined.
`TextString`	String	Gets or sets the text string for the attribute.
`Thickness`	Double	Gets or sets the distance the `Attribute` object is extruded above or below its elevation, in a Z-axis direction. The default is 0.0.
`UpsideDown`	Boolean	Specifies whether the text is upside down.
`Verify`	Boolean	Specifies whether the user is prompted for verification when inserting the block.
`VerticalAlignment`	`AcVerticalAlignment`	Gets or sets the vertical alignment of the attribute. For a list of possible values for the `AcVerticalAlignment` enumeration, see Appendix B.

■ **Note** Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadAttributeReference` Object

The `AcadAttributeReference` object represents an object containing text that links to a block or to an instance of the `AcadAtribute` object. These objects can't be directly created but are added to a drawing when a block containing an attribute definition is inserted. You achieve this by calling the `InsertBlock` method, which is common to the `AcadBlock`, `AcadModelSpace`, and `AcadPaperSpace` objects. The `GetAttributes` method of the `AcadBlockReference` object returns an array of all attribute references attached to the inserted block.

`AcadAttributeReference` Object Methods

The `AcadAttributeReference` object inherits all the methods of the `AcadEntity` and `AcadObject` objects. It supports no other methods.

`AcadAttributeReference` Object Properties

The `AcadAttributeReference` object inherits all the properties of the `AcadEntity` and `AcadObject` objects, as well as the common `Application` property. It also supports all the other properties supported by the `AcadAttribute` object except for `Mode`, `Preset`, `PromptString`, and `Verify`. It supports no other properties.

■ **Note** Because this object inherits from `AcadObject`, it supports the `Modified` event.

AcadBlock Object

The AcadBlock object represents a block definition containing a name and a set of objects. There's no limit to the number of objects a block can contain. Three kinds of block exist: a simple block, an Xref block, and a layout block. Chapter 12 discusses blocks.

AcadBlock Object Methods

In addition to the methods inherited from the AcadObject object, the AcadBlock object supports the following methods.

Name	Returns	Description
Add3DFace	Acad3DFace	Creates an Acad3DFace object using the four vertices. If the last parameter is omitted, a three-sided face is created. Points must be entered in a clockwise or counterclockwise order to create the object. Parameters: Point1 As Variant, Point2 As Variant, Point3 As Variant, and Point4 As Variant.
Add3DMesh	AcadPolygonMesh	Creates a free-form AcadPolygonMesh object representing an M×N free-form mesh from the given array of coordinates. Parameters: M As Integer, N As Integer, and PointsMatrix As Variant. The size of the mesh in both the M and N directions is limited to between 2 and 256. The PointsMatrix parameter is a 3×M×N array of 3-D WCS coordinates. All the coordinates for row M must be supplied before any vertices for row M+1.
Add3DPoly	Acad3DPolyline	Creates an Acad3DPolyline object from the given array of 3-D WCS coordinates. The polyline will be created according to the order of the coordinates in the array. Parameter: PointsArray As Variant.
AddArc	AcadArc	Creates an AcadArc object as specified by the parameters. Parameters: Center As Variant (an array of three Doubles representing 3-D WCS coordinates), Radius As Double, StartAngle As Double, and EndAngle As Double. The start and end angles must be in radians. If StartAngle is greater than EndAngle, a clockwise arc is created.
AddAttribute	AcadAttribute	Creates an AcadAttribute object as specified by the parameters. Parameters: Height As Double, Mode As AcAttributeMode, Prompt As String, InsertionPoint As Variant (an array of three Doubles representing 3-D WCS coordinates), Tag As String, and Value As String. For the values of the Mode parameter, see Appendix B.
AddBox	Acad3DSolid	Creates an Acad3DSolid object representing a solid box with center at the given origin and with edges parallel to the axes of the WCS. Parameters: Origin As Variant (an array of three Doubles as 3-D WCS coordinates), Length As Double, Width As Double, and Height As Double.

Continued

Name	Returns	Description
AddCircle	AcadCircle	Creates an AcadCircle object as specified by the parameters. Parameters: Center As Variant (an array of three Doubles representing 3-D WCS coordinates) and Radius As Double.
AddCone	Acad3DSolid	Creates an Acad3DSolid object representing a solid cone with a base on the XY plane of the WCS. Parameters: Center As Variant (an array of three Doubles representing 3-D WCS coordinates), BaseRadius As Double, and Height As Double.
AddCustomObject	Object	Creates a custom object. Parameter: ClassName As String. ClassName must be an ARX file prior to the call.
AddCylinder	Acad3DSolid	Creates an Acad3DSolid object representing a cylinder whose base is on the XY plane of the WCS. Parameters: Center As Variant (an array of three Doubles representing 3-D WCS coordinates), Radius As Double, and Height As Double.
AddDim3PointAngular	AcadDim3PointAngular	Creates an AcadDim3PointAngular object as specified by the parameters. Parameters: AngleVertex As Variant, FirstEndPoint As Variant, SecondEndPoint As Variant, and TextPoint As Variant. All four parameters are arrays of three Doubles representing 3-D WCS coordinates.
AddDimAligned	AcadDimAligned	Creates an AcadDimAligned object as specified by the parameters. Parameters: ExtLine1Point As Variant, ExtLine2Point As Variant, and TextPosition As Variant. All three parameters are arrays of three Doubles representing 3-D WCS coordinates.
AddDimAngular	AcadDimAngular	Creates an AcadDimAngular object representing the angular dimension for an arc, two lines, or a circle. Parameters: AngleVertex As Variant, FirstEndPoint As Variant, SecondEndPoint As Variant, and TextPoint As Variant. All four parameters are arrays of three Doubles representing 3-D WCS coordinates.
AddDimDiametric	AcadDimDiametric	Creates a diametric dimension for a circle or arc given the two points on the diameter and the length of the leader line. Parameters: ChordPoint As Variant, FarChordPoint As Variant, and LeaderLength As Double. The first two parameters are arrays of three Doubles representing 3-D WCS coordinates.
AddDimOrdinate	AcadDimOrdinate	Creates an ordinate dimension given the definition point and leader end point. Parameters: DefinitionPoint As Variant, LeaderEndPoint As Variant, and UseXAxis As Integer. The first two parameters are arrays of three Doubles representing 3-D WCS coordinates.
AddDimRadial	AcadDimRadial	Creates a radial dimension for the selected object at the given location. Parameters: Center As Variant, ChordPoint As Variant, and LeaderLength As Double. The first two parameters are arrays of three Doubles representing 3-D WCS coordinates.

Name	Returns	Description
AddDimRotated	AcadDimRotated	Creates an AcadDimRotated object as specified by the parameters. Parameters: ExtLine1Point As Variant, ExtLine2Point As Variant, DimLineLocation As Variant, and RotationAngle As Double. The first three parameters are arrays of three Doubles representing 3-D WCS coordinates.
AddEllipse	AcadEllipse	Creates an AcadEllipse object representing an ellipse in the XY plane of the WCS as specified by the parameters. Parameters: Center As Variant, MajorAxis As Variant, and RadiusRatio As Double. The first two parameters are arrays of three Doubles representing 3-D WCS coordinates.
AddEllipticalCone	Acad3DSolid	Creates an Acad3DSolid object representing an elliptical cone in the XY plane of the WCS given the center, major radius, minor radius, and height. Parameters: Center As Variant (an array of three Doubles representing 3-D WCS coordinates), MajorRadius As Double, MinorRadius As Double, and Height As Double.
AddEllipticalCylinder	Acad3DSolid	Creates an Acad3DSolid object representing an elliptical cylinder as specified by the parameters with its base in the XY plane. Parameters: Center As Variant (an array of three Doubles representing 3-D WCS coordinates), MajorRadius As Double, MinorRadius As Double, and Height As Double.
AddExtrudedSolid	Acad3DSolid	Creates an Acad3DSolid object representing an extruded solid as specified by the parameters. Parameters: Profile As AcadRegion, Height As Double, and TaperAngle As Double. The Height parameter specifies the height of the extrusion along the Z-axis of the object's coordinate system. The TaperAngle parameter must be provided in radians. The range of the taper angle is from –90 to +90 degrees. Positive angles taper in from the base; negative angles taper out. The default angle, 0, extrudes a 2-D object perpendicular to its plane. Only 2-D planar regions can be extruded.
AddExtrudedSolid AlongPath	Acad3DSolid	Creates an Acad3DSolid object representing an extruded solid, given the profile and an extrusion path. Only 2-D planar regions can be extruded, and the path shouldn't lie on the same plane as the profile, and it shouldn't have areas of high curvature. Parameters: Profile As AcadRegion and Path As Object.
AddHatch	AcadHatch	Creates an AcadHatch object as specified by the parameters. Parameters: PatternType As AcPatternType, PatternName As String, and Associativity As Boolean. For a list of possible values for the AcPatternType enumerated type, see Appendix B. The Associativity parameter is True if the hatch is to be associative and False otherwise. Don't forget to create the outer boundary for the hatch and to append it using Obj.AppendOuterLoop(outerLoop), in which Obj is the newly created hatch and outerLoop is an array of drawing objects forming the hatch boundary. (See the section on the AcadHatch object for details.)

Continued

Name	Returns	Description
AddLeader	AcadLeader	Creates an AcadLeader object as specified by the parameters. Parameters: PointsArray As Variant, Annotation As AcadEntity, and Type As AcLeaderType. The PointsArray parameter must be an array of 3-D WCS coordinates (Doubles) and has a minimum of two points (six elements). The Annotation parameter must be an AcadTolerance, AcadMText, or AcadBlockReference object or NULL. See Appendix B for the values of the AcLeaderType enumeration.
AddLightWeightPolyline	AcadLWPolyline	Creates an AcadLWPolyline object as specified by the parameter. Parameter: VerticesList As Variant. VerticesList is an array of 2-D OCS coordinates specifying the vertices of the polyline. A minimum of two vertices or four elements is required. You can convert between OCS and WCS using the TranslateCoordinates method (see the section on the AcadUtility object for details). Arcs are added to vertices using the SetBulge method (see the section on the AcadLWPolyline object for details).
AddLine	AcadLine	Creates an AcadLine object representing a line passing through the given points. Parameters: StartPoint As Variant and EndPoint As Variant. Both parameters are arrays of three Doubles representing 3-D WCS coordinates.
AddMInsertBlock	AcadMInsertBlock	Inserts an array of AcadBlockReferences from an original block definition of a given Name, as specified by the parameters. Parameters: InsertionPoint As Variant, Name As String, Xscale As Double, Yscale As Double, Zscale As Double, Rotation As Double, NumRows As Integer, NumColumns As Integer, RowSpacing As Integer, and ColumnSpacing As Integer. The InsertionPoint parameter is a set of 3-D WCS coordinates. The rotation angle is in radians.
AddMLine	AcadMLine	Creates an AcadMLine representing a set of lines passing through the points specified in the parameter. Parameter: VertexList As Variant. VertexList is an array of the 3-D WCS coordinates specifying the vertices for the multiline.
AddMText	AcadMText	Creates an AcadMText object as specified by the parameters. Parameters: InsertionPoint As Variant, Width As Double, and Text As String. The InsertionPoint parameter is a set of 3-D WCS coordinates.
AddPoint	AcadPoint	Creates an AcadPoint object as specified by the 3-D WCS coordinates specified. Parameter: Point As Variant.
AddPolyfaceMesh	AcadPolyfaceMesh	Creates an AcadPolyfaceMesh object from a list of vertices. Parameters: VertexList As Variant and FaceList As Variant. VertexList is an array of 3-D WCS coordinates used to create the polyface mesh vertices. At least four points (12 elements) are required for constructing a polyface mesh object. FaceList is an array of integers representing the vertex numbers for each face. Faces are defined in groups of alues.

Name	Returns	Description
AddPolyline	AcadPolyline	Creates an AcadPolyline object from a list of vertices. The polyline vertices are an array of OCS coordinates. At least two points (six values) are required to form a polyline. Parameter: VerticesList As Variant. You can convert between OCS and WCS using the TranslateCoordinates method (see the section on the AcadUtility object for details). Arcs are added to vertices using the SetBulge method (see the section on the AcadPolyline object for details).
AddRaster	AcadRasterImage	Creates an AcadRasterImage object based on the given image file. Parameters: ImageFileName As String, InsertionPoint As Variant, ScaleFactor As Double, and RotationAngle As Double. The InsertionPoint parameter is a set of 3-D WCS coordinates. The default value for the ScaleFactor parameter is 1. The RotationAngle is in radians.
AddRay	AcadRay	Creates an AcadRay object representing a ray passing through two points. Parameters: Point1 As Variant and Point2 As Variant. Both parameters are arrays of three Doubles representing 3-D WCS coordinates.
AddRegion	Variant	Creates a region from an array of AcadEntity objects. The array must consist of AcadLine, AcadArc, AcadCircle, AcadEllipse, AcadLWPolyline, and AcadSpline objects, which must form a closed coplanar region. Parameter: ObjectList As Variant.
AddRevolvedSolid	Acad3DSolid	Creates an Acad3DSolid object representing a revolved solid as specified by the parameters. Parameters: Profile As AcadRegion, AxisPoint As Variant, AxisDir As Variant, and Angle As Double. The AxisPoint parameter specifies the start point of the axis of revolution and is defined with an array of 3-D WCS coordinates. The AxisDir parameter specifies the direction of the axis of revolution. The Angle parameter is the angle of revolution in radians. Enter 6.28 for a full circle revolution.
AddShape	AcadShape	Creates an AcadShape object based on a template identified by name, at the given insertion point, scale factor, and rotation. Parameters: Name As String, InsertionPoint As Variant, ScaleFactor As Double, and RotationAngle As Double. The InsertionPoint parameter is a set of 3-D WCS coordinates.
AddSolid	AcadSolid	Creates an AcadSolid object representing a 2-D solid polygon. The first two points define one edge of the polygon. For a four-sided solid, the third point is defined to be diagonally opposite from the second point. If the third and fourth points are equal, then a triangle is created. The AcadSolid is filled when the FILLMODE system variable is set to 1. Parameters: Point1 As Variant, Point2 As Variant, Point3 As Variant, and Point4 As Variant.

Continued

Name	Returns	Description
AddSphere	Acad3DSolid	Creates an Acad3DSolid object representing a sphere as specified by the parameters. Parameters: Center As Variant (an array of three Doubles representing 3-D WCS coordinates) and Radius As Double.
AddSpline	AcadSpline	Creates an AcadSpline object as specified by the parameters, the first one being an array of 3-D WCS coordinates defining the spline curve. The StartTangent and EndTangent parameters specify the tangents (a 3-D vector) at the two ends of the spline. At least two points (six elements) are required for constructing an AcadSpline object. Parameters: PointsArray As Variant, StartTangent As Variant, and EndTangent As Variant.
AddText	AcadText	Creates an AcadText object representing a single line of text. Parameters: TextString As String, InsertionPoint As Variant, and Height As Double. The InsertionPoint parameter is a set of 3-D WCS coordinates.
AddTolerance	AcadTolerance	Creates an AcadTolerance object from the supplied parameters, the second one being a set of 3-D WCS coordinates and the last a 3-D vector. Parameters: Text As String, InsertionPoint As Variant, and Direction As Variant.
AddTorus	Acad3DSolid	Creates an Acad3DSolid object representing a torus from the parameters. Parameters: Center As Variant, TorusRadius As Double, and TubeRadius As Double. The first parameter is a set of 3-D WCS coordinates, and the values for both radii must be positive.
AddTrace	AcadTrace	Creates an AcadTrace object from an array of points. Parameter: PointsArray As Variant (an array of three Doubles representing 3-D WCS coordinates).
AddWedge	Acad3DSolid	Creates an Acad3DSolid object representing a wedge with edges parallel to the axes given the length, width, and height. Parameters: Center As Variant (an array of three Doubles representing 3-D WCS coordinates), Length As Double, Width As Double, and Height As Double.
AddXline	AcadXline	Creates an AcadXline object representing an infinite line passing through the given points. Parameters: Point1 As Variant and Point2 As Variant. Both parameters are arrays of three Doubles representing 3-D WCS coordinates. The width and fill mode are set using the TRACEWID and FILLMODE system variables.

Name	Returns	Description
AttachExternal Reference	AcadExternalReference	Attaches an AcadExternalReference (Xref) object to the AcadBlock, given the full path name of the external drawing, the name of the Xref, and other parameters. Note that attached AcadExternalReference objects can be nested. Parameters: PathName As String, Name As String, InsertionPoint As Variant, Xscale As Double, Yscale As Double, Zscale As Double, Rotation As Double, and Overlay As Boolean. The InsertionPoint parameter is a set of 3-D WCS coordinates. The Rotation parameter specifies the angle of rotation of the Xref, in radians. The Overlay parameter specifies whether the Xref is to be an overlay (True) or an attachment (False). An overlaid Xref will be ignored when the container drawing is referenced at its turn.
Bind		Binds an AcadExternalReference (Xref) object as a nested block, complete with all its named dependent symbols (dimension styles, linetypes, etc.). This operation makes the Xref a permanent part of the drawing and not an externally referenced file. Named objects from the Xref can be used in the current drawing. Parameter: PrefixName As Boolean. If this parameter is True, then the symbols named in the Xref are prefixed with <blockname>x, where x is an integer that is automatically incremented to avoid overriding existing block functions. If this parameter is False, no prefix is used. If there is a clash of names, AutoCAD uses the symbols already defined in the drawing.
Detach		Detaches an AcadExternalReference (Xref) object from the block, including all copies of the Xref and the definition itself.
InsertBlock	AcadBlockReference	Inserts an AcadBlockReference object representing a file or a block that has been defined in the current drawing. Parameters: InsertionPoint As Variant, Name As String, Xscale As Double, Yscale As Double, Zscale As Double, and Rotation As Double. The InsertionPoint parameter is a set of 3-D WCS coordinates. If the Name is a file name, include any path information necessary for AutoCAD to find the file and the .dwg extension.
Item	AcadEntity	Gets the member object at a given index in the block. Parameter: Index As Variant (String or Integer).
Reload		Reloads the most recently saved version of the external reference (Xref) or an unloaded Xref.
Unload		Unloads an external reference (Xref). This means that the drawing opens faster and uses less memory. The Xref and its table of symbols don't appear in the drawing.

AcadBlock Object Properties

In addition to the properties inherited from the AcadEntity object and the common Application property, the AcadBlock object supports the following properties.

Name	Returns	Description
Count	Integer	Gets the number of items in the block. This property is read-only.
IsLayout	Boolean	Determines whether the block is a layout block. This property's value is read-only. If the values for IsLayout and IsXref are both False, then the block is a simple block.
IsXref	Boolean	Determines whether the block is an Xref block. This property's value is read-only.
Layout	AcadLayout	Gets the plot settings (AcadLayout object) for the block. This property's value is read-only.
Name	String	Gets the name of the block.
Origin	Variant	Gets or sets the origin of the block in WCS coordinates.
XrefDatabase	AcadDatabase	Gets the AcadDatabase object that defines the contents of the block. Available only if the IsXref property for the block is True. This property's value is read-only.

■ **Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadBlockReference Object

The AcadBlockReference object represents an instance of a block definition inserted into a drawing. The creation of a new block definition is done automatically when an external block file is inserted with the InsertBlock method. To add or delete items from the block reference, you may first use the Explode method to break it into its component objects (subentities). However, the block definition still remains in the drawing's block symbol table with all its constituents.

AcadBlockReference Object Methods

The AcadBlockReference object inherits all the methods of the AcadEntity and AcadObject objects. It also supports the following methods.

Name	Returns	Description
Explode	Variant	Explodes the block and returns the subentities as an array of Objects
GetAttributes	Variant	Returns an array of editable attribute references attached to the block reference, only if the HasAttributes property is set to True
GetConstantAttributes	Variant	Returns an array of constant attributes for the block

AcadBlockReference Object Properties

The AcadBlockReference object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
HasAttributes	Boolean	Specifies if the block has any attributes in it. This property's value is read-only.
InsertionPoint	Variant	Gets or sets an insertion point in the block as a set of 3-D WCS coordinates. It returns a three-element array of Doubles.
Name	String	Gets or sets the name of the block.
Normal	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the block.
Rotation	Double	Gets or sets the rotation angle (in radians) for the block, relative to the X-axis with positive values going counterclockwise when viewed along the Z-axis toward the origin.
XScaleFactor	Double	Gets or sets the X scale factor for the block.
YScaleFactor	Double	Gets or sets the Y scale factor for the block
ZScaleFactor	Double	Gets or sets the Z scale factor for the block.

■ **Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadBlocks Collection

The AcadBlocks collection contains all the blocks in the drawing. The collection contains two special collections: the AcadModelSpace collection for model space entities, and the AcadPaperSpace collection for paper space entities. Although the AcadBlocks collection inherits a Delete method, you can't actually delete the collection. There's no limit to the number of blocks you can create in your drawing. However, there can be only one instance of the AcadBlocks collection predefined for each drawing. You can make multiple references to it by using the Blocks property of the AcadDocument object.

AcadBlocks Collection Methods

In addition to the methods inherited from the AcadObject object, the AcadBlocks collection supports the following methods.

Name	Returns	Description
Add	AcadBlock	Creates an AcadBlock object and adds it to the collection. Parameters: InsertionPoint As Variant and Name As String. The InsertionPoint parameter is a set of 3-D WCS coordinates that specify where the AcadBlock object will be added. Once the AcadBlock has been created, you can add subentities to it.
Item	AcadBlock	Returns the object at the given index in the collection. Parameter: Index As Variant (Integer or String). If the Index value is a String, it must match an existing block definition.

AcadBlocks Collection Properties

The AcadBlocks collection supports the Count property, the common Application property, and those properties inherited from the AcadObject object. It supports no other properties.

Note Because this collection inherits from AcadObject, it supports the Modified event.

AcadCircle Object

The AcadCircle object represents a full circle. It is created using the AddCircle method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadCircle Object Methods

The AcadCircle object inherits all the methods of the AcadEntity and AcadObject objects. It also supports the following method.

Name	Returns	Description
Offset	Variant	Creates a new circle by offsetting the current circle by a specified distance, which must be nonzero. If the offset is positive, a larger circle is created with the same origin. If the offset is negative, a smaller circle is created. Parameter: Distance As Double.

AcadCircle Object Properties

The AcadCircle object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Area	Double	Gets the area of the circle in square drawing units. This property's value is read-only.
Center	Variant	Gets or sets the center of the circle as a set of 3-D coordinates.
Circumference	Double	Gets or sets the circumference of the circle.
Diameter	Double	Gets or sets the diameter of the circle.
Normal	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the object.
Radius	Double	Gets or sets the radius of the circle.
Thickness	Double	Gets or sets the distance the AcadCircle object is extruded above or below its elevation, in a Z-axis direction. The default is 0.0.

Note Because this object inherits from AcadObject, it supports the Modified event.

AcadDatabase Object

The AcadDatabase object represents the contents of an external reference (Xref) block. This object provides access to the contents of an external reference block. It's available only on blocks whose IsXref property is equal to True.

AcadDatabase Object Methods

The AcadDatabase object supports the following methods. These are identical to three of the methods supported by the AcadDocument object, except they act upon the contents of an Xref block, not the active document.

Name	Returns	Description
CopyObjects	Variant	Duplicates an object along with other objects owned and referenced by it. This is called *deep cloning*. Parameters: Objects As Variant, [Owner As Variant], and [IDPairs As Variant]. The Objects parameter is array of primary objects to be copied. They must all have the same owner. The Owner parameter is the new owner for the copied objects. If unspecified, the original owner is assumed. The IDPairs parameter gives information on what happened during the copy and translation process. Input is an empty Variant. Output is an array of AcadIDPair objects. You can't execute this method while simultaneously iterating through a collection.
HandleToObject	Object	Returns the object that corresponds to the given handle. The object must be in the current Xref drawing. Parameter: Handle As String. A handle is persistent. It can be stored for reuse with this method for the lifetime of the object.
ObjectIDToObject	Object	Returns the object that corresponds to the given object ID. The object must be in the current Xref drawing. Parameter: ObjectID As Integer. An object ID isn't persistent. Its lifetime is limited to a drawing session.

AcadDatabase Object Properties

As well as the common Application property, the AcadDatabase object supports the following properties. These are identical to the corresponding properties of the AcadDocument object, except they refer to the contents of an Xref block, not the active document.

Name	Returns	Description
Blocks	AcadBlocks	Gets the AcadBlocks collection for the Xref drawing. This property's value is read-only.
Dictionaries	AcadDictionaries	Gets the AcadDictionaries collection for the Xref drawing. This property's value is read-only.
DimStyles	AcadDimStyles	Gets the AcadDimStyles collection for the Xref drawing. This property's value is read-only.
ElevationModelSpace	Double	Gets or sets the elevation setting in the model space. The current elevation is the Z value that is used whenever a 3-D point is expected but only the X and Y values are supplied.

Continued

Name	Returns	Description
ElevationPaperSpace	Double	Gets or sets the elevation setting in the paper space. The current elevation is the Z value that is used whenever a 3-D point is expected but only the X and Y values are supplied.
Groups	AcadGroups	Gets the AcadGroups collection for the Xref drawing. This property's value is read-only.
Layers	AcadLayers	Gets the AcadLayers collection for the Xref drawing. This property's value is read-only.
Layouts	AcadLayouts	Gets the AcadLayouts collection for the Xref drawing. This property's value is read-only.
Limits	Variant	Gets or sets the Xref drawing limits in the XY plane. Two sets of WCS coordinates specify the lower-left (LIMMIN) and upper-right corners (LIMMAX) of the plane.
Linetypes	AcadLineTypes	Gets the AcadLinetypes collection for the Xref drawing. This property's value is read-only.
ModelSpace	AcadModelSpace	Gets the AcadModelSpace collection for the Xref drawing. This property's value is read-only.
PaperSpace	AcadPaperSpace	Gets the AcadPaperSpace collection for the Xref drawing. This property's value is read-only.
PlotConfigurations	AcadPlotConfigurations	Gets the AcadPlotConfigurations collection for the Xref drawing. This property's value is read-only.
Preferences	AcadDatabasePreferences	Gets the AcadPreferences object for the Xref drawing. The AcadPreferences object holds the options from the Options dialog box that resides in the Registry. Options that reside in the drawing can be found through the AcadDatabasePreferences object. This property's value is read-only.
RegisteredApplications	AcadRegisteredApplications	Gets the AcadRegisteredApplications collection for the Xref drawing. This property's value is read-only.
TextStyles	AcadTextStyles	Gets the AcadTextStyles collection for the Xref drawing. This property's value is read-only.
UserCoordinateSystems	AcadUCSs	Gets the AcadUCSs collection for the Xref drawing. This property's value is read-only.
Viewports	AcadViewports	Gets the AcadViewports collection for the Xref drawing. This property's value is read-only.
Views	AcadViews	Gets the AcadViews collection for the Xref drawing. This property's value is read-only.

AcadDatabasePreferences Object

The AcadDatabasePreferences object specifies the current AutoCAD drawing-specific settings, that is, all the options from the Options dialog box that reside in a drawing. (Options that reside in the Registry can be found through the main AcadPreferences object.) This object has no methods.

AcadDatabasePreferences Object Properties

As well as the common Application property, the AcadDatabasePreferences object supports the following properties.

Name	Returns	Description
AllowLongSymbolNames	Boolean	Determines whether symbol names are extended (True) or restricted (False). ("Symbol names" refers to the names of nongraphical objects such as linetypes and layers.) Extended symbol names can be up to 255 characters and can use any symbol not reserved by Microsoft or AutoCAD, in addition to the letters, numerals, and the space. This is the default. Restricted symbol names can only be up to 31 letters long and are restricted to the letters and numerals, the underscore (_), dollar ($), and hyphen (-). The value for this property is stored in the EXTNAMES system variable.
ContourLinesPerSurface	Integer	Gets or sets the number of contour lines per surface on an object. The valid range is 0 to 2047. The number is stored in the ISOLINES system variable.
DisplaySilhouette	Boolean	Determines whether silhouette curves of solid objects are displayed in wireframe mode: True for silhouette curves and False for isolines (default). The value is stored in the DISPSILH system variable.
Lineweight	AcLineWeight	Gets or sets the default lineweight for the drawing. For a list of possible values for the AcLineWeight enumerated type, see Appendix B.
LineweightDisplay	Boolean	Determines whether lineweights are displayed in the model space. True is the default. This property is read/write.
MaxActiveViewports	Integer	Gets or sets the maximum number of active viewports. The range is 2 to 48. The value of is stored in the MAXACTVP system variable.
ObjectSortByPlotting	Boolean	Specifies whether objects are sorted by their plotting order. True is the default. The values of this property and the five that follow are stored in the SORTENTS system variable. Note that setting these additional ObjectSortBy... properties to True slows your application.
ObjectSortByPSOutput	Boolean	Specifies whether objects are sorted by their PostScript output order. True is the default.
ObjectSortByRedraws	Boolean	Specifies whether objects are sorted by their redraw order. False is the default.
ObjectSortByRegens	Boolean	Specifies whether objects are sorted by their regeneration order. False is the default.
ObjectSortBySelection	Boolean	Specifies whether drawing objects are sorted by object selection. False is the default.
ObjectSortBySnap	Boolean	Gets or sets whether drawing objects are sorted by object snap. False is the default.
OLELaunch	Boolean	Specifies whether the parent application is launched when OLE objects are plotted. The value of this property is stored in the OLESTARTUP system variable. The initial value of this property is False. This gives a high-quality plot but at the cost of speed.

Continued

Name	Returns	Description
RenderSmoothness	Double	Gets or sets the smoothness of shaded, rendered, and hidden line–removed objects. The value of this property is stored in the FACETRES system variable. The initial value for this property is 0.5. Values less than or equal to 1 increase performance.
SegmentPerPolyline	Integer	Gets or sets the number of line segments for each polyline curve. The value of this property is stored in the SPLINESEGS system variable. The initial value is 8.
SolidFill	Boolean	Specifies whether multilines, traces, solids, hatches, and wide polylines are filled in. The value of this property is stored in the FILLMODE system variable. The default value is True.
TextFrameDisplay	Boolean	Specifies whether the frames for text objects are displayed rather than the text itself. The value of this property is stored in the QTEXTMODE system variable. The default is False.
XrefEdit	Boolean	Determines whether the current drawing can be edited in place while being referenced by another user. The value of this property is stored in the XEDIT system variable. The default is True.
XrefLayerVisibility	Boolean	Specifies whether Xref-dependent layers are visible and whether nested Xref path changes are saved. If True, Xref-dependent layer changes made in the current drawing take precedence (default). If False, the layer table as stored in the reference drawing (Xref) takes precedence. The value of this property is stored in the VISRETAIN system variable.

AcadDictionaries Collection

The AcadDictionaries collection contains all the dictionaries in the drawing. Although this collection inherits a Delete method, you can't actually delete it. If you need to delete a specific dictionary, use the Delete method found in the AcadDictionary object. There's no limit to the number of dictionaries you can create in your drawing. However, there can be only one instance of the AcadDictionaries collection, which is predefined for each drawing. You can make multiple references to it by using the Dictionaries property.

AcadDictionaries Collection Methods

In addition to the methods inherited from the AcadObject object, the AcadDictionaries collection supports the following methods.

Name	Returns	Description
Add	AcadDictionary	Creates a member object and adds it to the collection. Parameter: Name As String.
Item	AcadObject	Gets the member object at a given index in a collection. Parameter: Index As Variant (an Integer or a String). If the value for Index is a String, it must match an existing object name in the collection.

AcadDictionaries Collection Properties

The AcadDictionaries collection supports the Count property, the common Application property, and the properties inherited from the AcadObject object. It supports no other properties.

Note Because this collection inherits from AcadObject, it supports the Modified event.

AcadDictionary Object

The AcadDictionary object represents a container object for storing and retrieving objects, with associated string keywords by which they are referenced. A dictionary can contain any type of object, including other dictionaries. You can create new dictionaries, add entries to an existing dictionary, and get the keyword for a given object or the object for a given keyword. You can change the object associated with a given keyword or rename the keyword for a given object. However, it doesn't perform type checking of entries.

Note that this object doesn't represent the spell-checking dictionary, which is specified in the AcadPreferences object.

AcadDictionary Object Methods

In addition to the methods inherited from the AcadObject object, the AcadDictionary object supports the following properties.

Name	Returns	Description
AddObject	AcadObject	Adds an object to a named dictionary. Note that an entry replaces one already existing with the same name. Parameters: Keyword As String and ObjectName As String. The first parameter is the keyword to be listed in the dictionary for this object. The second parameter is the name of the object to be stored. The ARX application defining the object must first be loaded with the LoadArx method.
AddXRecord	AcadXRecord	Creates an XRecord object in any dictionary. Parameter: Keyword As String.
GetName	String	Gets the keyword of an object in a dictionary. The parameter is the object's name. Parameter: Object As AcadObject.
GetObject	AcadObject	Gets the object in a dictionary, given the keyword for the object. Parameter: Name As String.
Item	AcadObject	Gets the member object at a given index in the dictionary. Parameter: Index As Variant (Integer or String). If the value for Index is a String, it must match an existing object name in the collection.
Remove	AcadObject	Removes a named object from the dictionary, given its keyword. Parameter: Name As String.
Rename		Renames the keyword of an item in the dictionary. Parameters: OldName As String and NewName As String.
Replace		Replaces an item in the dictionary by a given item. This changes the object but retains the keyword. Parameters: OldName As String and Obj As AcadObject.

AcadDictionary Object Properties

In addition to the properties inherited from the AcadObject object and the common Application property, the AcadDictionary object supports the following properties.

Name	Returns	Description
Count	Integer	Gets the number of items in the dictionary. This property's value is read-only.
Name	String	Gets or sets the name of the dictionary.

■**Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadDim3PointAngular Object

The AcadDim3PointAngular object represents a dimension of the angular distance defined by three points. When creating a three-point angular dimension, AutoCAD draws the dimension line arc between the extension lines, which are drawn from the angle end points to the intersection of the dimension line arc. AutoCAD uses the location of the dimension line arc to choose between the minor and major angles specified by the angle vertex and extension lines. To create a three-point angular dimension, use the AddDim3PointAngular method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadDim3PointAngular Object Methods

The AcadDim3PointAngular object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadDim3PointAngular Object Properties

The AcadDim3PointAngular object inherits all the properties of the AcadEntity object, the AcadObject object, and the AcadDimension object, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
AngleFormat	AcAngleUnits	Gets or sets the unit format for angles. The initial value is acDegrees. For the values of the AcAngleUnit enumerated type, see Appendix B.
AngleVertex	Variant	Gets or sets a 3-D set of coordinates representing the vertex.
Arrowhead1Block Arrowhead2Block	String	Gets or sets the block used as the custom arrowhead at the first/second end of the dimension line. It overrides the value of the DIMBLK1 or DIMBLK2 system variable.
Arrowhead1Type Arrowhead2Type		Gets or sets the type of arrowhead at the first/second end of the dimension line. For the values of the AcDimArrowheadType enumerated type, see Appendix B.

Name	Returns	Description
ArrowheadSize	Double	Gets or sets the size of the dimension line arrowheads, leader line arrowheads, and hook lines. The initial value for this property is 0.1800.
DimensionLineColor	AcColor	Gets or sets the color of the dimension lines. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in the default color) or acByLayer (where AutoCAD draws objects in the color specified for the layer). For a list of possible values for the AcColor enumerated type, see Appendix B.
DimensionLineWeight	AcLineWeight	Gets or sets the lineweight of the dimension lines. It overrides the value of the DIMLWD system variable. For a list of possible values for the AcLineWeight enumerated type, see Appendix B.
DimLine1Suppress DimLine2Suppress	Boolean	Gets or sets whether the first/second dimension line is suppressed. It overrides the value of the DIMSD1 or DIMSD2 system variable. The initial value for this property is False. When this property is set to True, the display of the dimension line and arrowhead between the first extension line and the text is suppressed.
DimLineInside	Boolean	Gets or sets whether the dimension lines are only displayed inside the extension lines. It overrides the value of the DIMSOXD system variable. The initial value for this property is False.
ExtensionLineColor	AcColor	Gets or sets the color of the extension lines. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in the default color), acByLayer (where AutoCAD draws objects in the color specified for the layer), or the AcColor enumerated type (see Appendix B). It overrides the value of the DIMCLRE system variable.
ExtensionLineExtend	Double	Gets or sets the distance that the extension line extends beyond the dimension line. It overrides the value of the DIMEXE system variable. The initial value for this property is 0.1800.
ExtensionLineOffset	Double	Gets or sets the distance that the extension lines are offset from the origin points. It overrides the value of the DIMEXO system variable. The initial value for this property is 0.0625.
ExtensionLineWeight	AcLineWeight	Gets or sets the lineweight of the extension lines. It overrides the value of the DIMLWE system variable. For a list of possible values for the AcLineWeight enumerated type, see Appendix B.
ExtLine1Point ExtLine2Point	Variant	Gets or sets the end point of the first/second extension line.
ExtLine1Suppress ExtLine2Suppress	Boolean	Gets or sets whether the first/second extension line is suppressed. It overrides the value of the DIMSE1 or DIMSE2 system variable. The initial value for this property is False.

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Name	Returns	Description
`Fit`	`AcDimFit`	Gets or sets whether text and arrowheads are placed inside or outside extension lines given the availability of space between the extension lines. This property sets priorities for moving text and arrowheads when space isn't available to fit both within the extension lines. It overrides the value of the `DIMAFIT` system variable. The initial value for this property is `acBestFit`. For a list of the possible values, see Appendix B.
`ForceLineInside`	Boolean	Gets or sets whether a dimension line is drawn between the extension lines when the text is outside the extension lines. It overrides the value of the `DIMTOFL` system variable. The initial value for this property is `False`.
`FractionFormat`		Gets or sets the horizontal position of the text. For a list of possible values for the `AcDimHorizontalJustification` enumerated type, see Appendix B.
`Measurement`	Double	Gets the measurement for the dimension. For angular dimensions, this property overrides the value for the `DIMAUNIT` system variable. This property's value is read-only.
`TextInside`	Boolean	Gets or sets whether the text appears inside the extension lines. It overrides the value of `DIMTIX` system variable. The initial value for this property is `True`. If `False`, AutoCAD places it inside if there is room.
`TextInsideAlign`	Boolean	Gets or sets the position of any text inside/outside the extension lines. The values for these properties override the values of the `DIMTIH` or `DIMTOH` system variable, respectively. The initial value for this property is `True`. The `TextInsideAlign` property is available only when the `TextInside` property is set to `True`.
`TextPrecision`	`AcDimPrecision`	Gets or sets the precision of the text. For the possible values for the `AcDimPrecision` enumerated type, see Appendix B.

■Note Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadDimAligned` Object

The `AcadDimAligned` object represents a linear dimension, measuring the distance between two points, that's displayed parallel to the points being measured. In aligned dimensions, the dimension line is parallel to the extension line origins. The extension line origins are specified using the `ExtLine1Point` and `ExtLine2Point` properties. The `AcadDimAligned` object is created using the `AddDimAligned` method of the `AcadBlock`, `AcadModelSpace`, or `AcadPaperSpace` object.

AcadDimAligned Object Methods

The AcadDimAligned object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadDimAligned Object Properties

The AcadDimAligned object inherits all the properties of the AcadEntity object, the AcadObject object, and the AcadDimension object, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
AltRoundDistance	Double	Gets or sets the rounding of alternative units, if the AltUnits property is turned on. This property, and the two that follow, all override the value of the DIMALTZ system variable.
AltSuppressLeadingZeros	Boolean	Gets or sets whether leading zeros in alternative dimension values are suppressed. The initial value for this property is False.
AltSuppressTrailingZeros	Boolean	Gets or sets whether trailing zeros in alternative dimension values are suppressed. The initial value for this property is False.
AltSuppressZeroFeet AltSuppressZeroInches	Boolean	Gets or sets whether a zero foot/inches measurement is suppressed in alternative dimension values. Both properties change the value of the DIMALTZ system variable. The initial value for this property is False.
AltTextPrefix AltTextSuffix	String	Gets or sets a prefix/suffix for the alternative dimension measurement. Both properties override the value of the DIMAPOST system variable. The initial value for this property is a NULL string. To turn off an established prefix, set this property equal to a single period (.).
AltTolerancePrecision	AcDimPrecision	Gets or sets the precision of tolerance values in alternative dimensions. The initial value for this property is acDimPrecisionTwo. This property is available only when the ToleranceDisplay property is set to any value other than acTolNone, and it overrides the value of the DIMTDEC system variable. For a list of the values for the AcDimPrecision enumerated type, see Appendix B.
AltToleranceSuppressLeadingZeros	Boolean	Gets or sets whether leading zeros are suppressed in alternative dimension values. This property, and the three that follow, all override the DIMALTTZ system variable. The initial value for this property is False.
AltToleranceSuppressTrailingZeros	Boolean	Gets or sets whether trailing zeros are suppressed in alternative dimension values. The initial value for this property is False.

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Name	Returns	Description
AltToleranceSuppressLeadingZeros	Boolean	Gets or sets whether a zero foot measurement is suppressed in alternative tolerance values. The initial value for this property is False.
AltSuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed in alternative tolerance values. The initial value for this property is False.
AltUnits	Boolean	Gets or sets whether alternative units dimensioning is enabled. It overrides the value of the DIMALT system variable.
AltUnitsFormat	AcDimUnits	Gets or sets the units format for alternative units. The initial value for this property is acDimDecimal. It overrides the value of the DIMALTU system variable. For a list of possible values for the AcDimUnits enumerated type, see Appendix B.
AltUnitsPrecision	AcDimPrecision	Gets or sets the number of decimal places in alternative units. It overrides the value of the DIMALTD system variable. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.
AltUnitsScale	Double	Gets or sets the scale factor for alternative units. If the AltUnits property is turned on, this property multiplies linear dimensions by a factor to produce a value in an alternate system of measurement. The initial value represents the number of millimeters in an inch. It overrides the value of the DIMALTF system variable.
Arrowhead1Block Arrowhead2Block	String	Gets or sets the block used as the custom arrowhead at the first/second end of the dimension line. It overrides the value of the DIMBLK1 or DIMBLK2 system variable.
Arrowhead1Type Arrowhead2Type	AcDimArrowheadType	Gets or sets the type of arrowhead at the first/second end of the dimension line. For a list of possible values for the AcDimArrowheadType enumerated type, see Appendix B.
ArrowheadSize	Double	Gets or sets the size of the dimension line arrowheads, leader line arrowheads, and hook lines. It overrides the value of the DIMASZ system variable. The initial value for this property is 0.1800.
DimensionLineColor	AcColor	Gets or sets the color of the dimension lines. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in the default color) or acByLayer (where AutoCAD draws objects in the color specified for the layer). For a list of possible values for the AcColor enumerated type, see Appendix B.

Name	Returns	Description
`DimensionLineExtend`	Double	Gets or sets the distance the dimension line extends beyond the extension line when oblique strokes rather than arrowheads are used. It overrides the value of the `DIMDLE` system variable. The initial value for this property is 0.0000.
`DimensionLineWeight`	`AcLineWeight`	Gets or sets the lineweight of the dimension lines. It overrides the value of the `DIMLWD` system variable. For a list of possible values for the `AcLineWeight` enumerated type, see Appendix B.
`DimLine1Suppress` `DimLine2Suppress`	Boolean	Gets or sets whether the first/second dimension line is suppressed. It overrides the value of the `DIMSD1` or `DIMSD2` system variable. The initial value for this property is `False`. When this property is set to `True`, the display of the dimension line and arrowhead between the first extension line and the text is suppressed.
`DimLineInside`	Boolean	Gets or sets whether the dimension lines are displayed only inside the extension lines. It overrides the value of the `DIMSOXD` system variable. The initial value for this property is `False`.
`ExtensionLineColor`	`AcColor`	Gets or sets the color of the extension lines. Use a color index number from 0 to 256 or one of the constants listed here: `acByBlock` (where AutoCAD draws objects in the default color), `acByLayer` (where AutoCAD draws objects in the color specified for the layer), or the `AcColor` enumerated type (see Appendix B). It overrides the value of the `DIMCLRE` system variable.
`ExtensionLineExtend`	Double	Gets or sets the distance that the extension line extends beyond the dimension line. It overrides the value of the `DIMEXE` system variable. The initial value for this property is 0.1800.
`ExtensionLineOffset`	Double	Gets or sets the distance that the extension lines are offset from the origin points. It overrides the value of the `DIMEXO` system variable. The initial value for this property is 0.0625.
`ExtensionLineWeight`	`AcLineWeight`	Gets or sets the lineweight of the extension lines. It overrides the value of the `DIMLWE` system variable. For a list of possible values for the `AcLineWeight` enumerated type, see Appendix B.
`ExtLine1Point` `ExtLine2Point`	Variant	Gets or sets the end point of the first/second extension line.
`ExtLine1Suppress` `ExtLine2Suppress`	Boolean	Gets or sets whether the first/second extension line is suppressed. It overrides the value of the `DIMSE1` or `DIMSE2` system variable. The initial value for this property is `False`.
`Fit`	`AcDimFit`	Gets or sets whether text and arrowheads are placed inside or outside extension lines given the availability of space between the extension lines. This property sets priorities for moving text and arrowheads when space isn't available to fit both within the extension lines. It overrides the value of the `DIMAFIT` system variable. The initial value for this property is `acBestFit`. For a list of the possible values, see Appendix B.
`ForceLineInside`	Boolean	Gets or sets whether a dimension line is drawn between the extension lines when the text is outside the extension lines. It overrides the value of the `DIMTOFL` system variable. The initial value for this property is `False`.

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Name	Returns	Description
FractionFormat	AcDimFractionType	Gets or sets the formats of fractional values. It overrides the value of the DIMFRAC system variable. This property is used when the UnitsFormat property is set to acDimLArchitectural or acDimLFractional. Possible values: acHorizontal, acDiagonal, and acNotStacked.
HorizontalTextPosition	AcDimHorizontalJustification	Gets or sets the horizontal position of the text. It overrides the value of the DIMJUST system variable. For a list of possible values for the AcDimHorizontalJustification enumerated type, see Appendix B.
LinearScaleFactor	Double	Gets or sets the global scale factor for linear dimensioning measurements. It overrides the value of the DIMLFAC system variable. The initial value for this property is 1.0000. When LinearScaleFactor is assigned a negative value, the factor is applied only in paper space.
Measurement	Double	Gets the measurement for the dimension. This property overrides the value for the DIMLUNIT system variable. This property's value is read-only.
PrimaryUnitsPrecision	AcDimPrecision	Gets or sets the number of decimal places displayed for the primary units. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.
RoundDistance	Double	Gets or sets the rounding of units. It overrides the value of the DIMRND system variable.
SuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
SuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
TextInside	Boolean	Gets or sets whether the text appears inside the extension lines. It overrides the value of DIMTIX system variable. The initial value for this property is True. If False, AutoCAD places it inside if there is room.

Name	Returns	Description
TextInsideAlign	Boolean	Gets or sets the position of any text inside/outside the extension lines. The values for these properties override the values of the DIMTIH or DIMTOH system variable, respectively. The initial value for this property is True. The TextInsideAlign property is available only when the TextInside property is set to True.
ToleranceSuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed in tolerance values. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
ToleranceSuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed in tolerance values. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
UnitsFormat	AcDimLUnits	Gets or sets the unit format. The initial value for this property is acDimLDecimal, in which the format specified by the DecimalSeparator and PrimaryUnitsPrecision properties will be used to format the decimal value. For a list of possible values for the AcDimLUnits enumerated type, see Appendix B.

Note Because this object inherits from AcadObject, it supports the Modified event.

AcadDimAngular Object

The AcadDimAngular object represents a dimension of the angular distance between two lines or the angle of a circular arc. If you need extension lines, they'll be added automatically. The end points provided will be used as origin points for the extension lines. The AcadDimAngular object is created using the AddDimAngular method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadDimAngular Object Methods

The AcadDimAngular object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadDimAngular Object Properties

The AcadDimAngular object inherits all the properties of the AcadEntity object, the AcadObject object, and the AcadDimension object, as well as the common Application property. It also supports all the other properties supported by the AcadDim3PointAngular object except for the AngleVertex property. This object, however, has the following two extra properties.

Name	Returns	Description
ExtLine1StartPoint ExtLine2StartPoint	Variant	Gets or sets the start point of the first/second extension line.

Note Because this object inherits from AcadObject, it supports the Modified event.

AcadDimArcLength Object

This is a circular-style dimension measuring the length of an arc.

AcadDimArcLength Object Methods

The AcadDimArcLength object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadDimArcLength Object Properties

The AcadDimArcLength object inherits all the properties of the AcadEntity object, the AcadObject object, and the AcadDimension object, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
AltRoundDistance	Double	Gets or sets the rounding of alternative units, if the AltUnits property is turned on. This property, and the two that follow, all override the value of the DIMALTZ system variable.
AltSuppressLeadingZeros	Boolean	Gets or sets whether leading zeros in alternative dimension values are suppressed. The initial value for this property is False.
AltSuppressTrailingZeros	Boolean	Gets or sets whether trailing zeros in alternative dimension values are suppressed. The initial value for this property is False.
AltSuppressZeroFeet AltSuppressZeroInches	Boolean	Gets or sets whether a zero foot/inches measurement is suppressed in alternative dimension values. Both properties change the value of the DIMALTZ system variable. The initial value for this property is False.
AltTextPrefix AltTextSuffix	String	Gets or sets a prefix/suffix for the alternative dimension measurement. Both properties override the value of the DIMAPOST system variable. The initial value for this property is a NULL string. To turn off an established prefix, set this property equal to a single period (.).

Name	Returns	Description
AltTolerancePrecision	AcDimPrecision	Gets or sets the precision of tolerance values in alternative dimensions. The initial value for this property is acDimPrecisionTwo. This property is available only when the ToleranceDisplay property is set to any value other than acTolNone, and it overrides the value of the DIMTDEC system variable. For a list of the values for the AcDimPrecision enumerated type, see Appendix B.
AltToleranceSuppressLeadingZeros	Boolean	Gets or sets whether leading zeros are suppressed in alternative dimension values. This property, and the three that follow, all override the DIMALTTZ system variable. The initial value for this property is False.
AltToleranceSuppressTrailingZeros	Boolean	Gets or sets whether trailing zeros are suppressed in alternative dimension values. The initial value for this property is False.
AltToleranceSuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed in alternative tolerance values. The initial value for this property is False.
AltToleranceSuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed in alternative tolerance values. The initial value for this property is False.
AltUnits	Boolean	Gets or sets whether alternative units dimensioning is enabled. This overrides the value of the DIMALT system variable.
AltUnitsFormat	AcDimUnits	Gets or sets the units format for alternative units. The initial value for this property is acDimDecimal. This overrides the value of the DIMALTU system variable. For a list of possible values for the AcDimUnits enumerated type, see Appendix B.
AltUnitsPrecision	AcDimPrecision	Gets or sets the number of decimal places in alternative units. This overrides the value of the DIMALTD system variable. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.
AltUnitsScale	Double	Gets or sets the scale factor for alternative units. If the AltUnits property is turned on, this property multiplies linear dimensions by a factor to produce a value in an alternate system of measurement. The initial value represents the number of millimeters in an inch. This overrides the value of the DIMALTF system variable.
ArcEndParam	Double	Gets or sets the end parameter of the arc as a 3-D coordinate.
ArcPoint	Variant	Gets or sets a point on the arc as a 3-D coordinate.
ArcStartParam	Double	Gets or sets the start parameter of the arc as a 3-D coordinate.

Continued

Name	Returns	Description
Arrowhead1Block Arrowhead2Block	String	Gets or sets the block used as the custom arrowhead at the first/second end of the dimension line. This overrides the value of the DIMBLK1 or DIMBLK2 system variable.
Arrowhead1Type Arrowhead2Type	AcDimArrowheadType	Gets or sets the type of arrowhead at the first/second end of the dimension line. For a list of possible values for the AcDimArrowheadType enumerated type, see Appendix B.
ArrowheadSize	Double	Gets or sets the size of the dimension line arrowheads, leader line arrowheads and hook lines. This overrides the value of the DIMASZ system variable. The initial value for this property is 0.1800.
CenterPoint	Variant	Gets or sets the center as a set of 3-D coordinates.
DimensionLineColor	AcColor	Gets or sets the color of the dimension lines. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in default color) or acByLayer (where AutoCAD draws objects in the color specified for the layer). For a list of possible values for the AcColor enumerated type, see Appendix B.
DimensionLineExtend	Double	Gets or sets the distance the dimension line extends beyond the extension line when oblique strokes rather than arrowheads are used. This overrides the value of the DIMDLE system variable. The initial value for this property is 0.0000.
DimensionLineWeight	AcLineWeight	Gets or sets the line weight of the dimension lines. This overrides the value of the DIMLWD system variable. For a list of possible values for the AcLineWeight enumerated type, see Appendix B.
DimLine1Suppress DimLine2Suppress	Boolean	Gets or sets whether the first/second dimension line is suppressed. This overrides the value of the DIMSD1 or DIMSD2 system variable. The initial value for this property is False. When this property is set to True, the display of the dimension line and arrowhead between the first extension line and the text is suppressed.
DimLineInside	Boolean	Gets or sets whether the dimension lines are displayed only inside the extension lines. This overrides the value of the DIMSOXD system variable. The initial value for this property is False.
ExtensionLineColor	AcCmColor	Gets or sets the color of the extension lines. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in default color), acByLayer (where AutoCAD draws objects in the color specified for the layer), and the AcColor enumerated type (see Appendix B). This overrides the value of the DIMCLRE system variable.
ExtensionLineExtend	Double	Gets or sets the distance that the extension line extends beyond the dimension line. This overrides the value of the DIMEXE system variable. The initial value for this property is 0.1800.

Name	Returns	Description
ExtensionLineOffset	Double	Gets or sets the distance that the extension lines are offset from the origin points. This overrides the value of the DIMEXO system variable. The initial value for this property is 0.0625.
ExtensionLineWeight	AcLineWeight	Gets or sets the line weight of the extension lines. This overrides the value of the DIMLWE system variable. For a list of possible values for the AcLineWeight enumerated type, see Appendix B.
ExtLine1Point ExtLine2Point	Variant	Gets or sets the endpoint of the first/second extension line.
ExtLine1Suppress ExtLine2Suppress	Boolean	Gets or sets whether the first/second extension line is suppressed. This overrides the value of the DIMSE1 or DIMSE2 system variable. The initial value for this property is False.
Fit	AcDimFit	Gets or sets whether text and arrowheads are placed inside or outside extension lines given the availability of space between the extension lines. This property sets priorities for moving text and arrowheads when space is not available to fit both within the extension lines. This overrides the value of the DIMAFIT system variable. The initial value for this property is acBestFit. For a list of the possible values, see Appendix B.
ForceLineInside	Boolean	Gets or sets whether a dimension line is drawn between the extension lines when the text is outside the extension lines. This overrides the value of the DIMTOFL system variable. The initial value for this property is False.
FractionFormat	AcDimFractionType	Gets or sets the formats of fractional values. This overrides the value of the DIMFRAC system variable. This property is used when the UnitsFormat property is set to acDimLArchitectural or acDimLFractional. Possible values: acHorizontal, acDiagonal and acNotStacked.
HasLeader	Boolean	Specifies whether the dimension has a leader. The initial value for this property is False.
HorizontalTextPosition	AcDimHorizontalJustification	Gets or sets the horizontal position of the text. This overrides the value of the DIMJUST system variable. For a list of possible values for the AcDimHorizontalJustification enumerated type, see Appendix B.
IsPartial	Boolean	Gets or sets whether the dimension is for a partial arc. The initial value for this property is False.
Leader1Point	Variant	Gets or sets the origin for Leader 1 as a set of 3-D coordinates.

Continued

Name	Returns	Description
Leader2Point	Variant	Gets or sets the origin for Leader 2 as a set of 3-D coordinates.
LinearScaleFactor	Double	Gets or sets the global scale factor for linear dimensioning measurements. This overrides the value of the DIMLFAC system variable. The initial value for this property is 1.0000. When LinearScaleFactor is assigned a negative value, the factor is applied only in paper space.
Measurement	Double	Gets the measurement for the dimension. This property overrides the value for the DIMLUNIT system variable. This is read-only.
PrimaryUnitsPrecision	AcDimPrecision	Gets or sets the number of decimal places displayed for the primary units. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.
RoundDistance	Double	Gets or sets the rounding of units. This overrides the value of the DIMRND system variable.
SuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed. This overrides the value of the DIMZIN system variable. The initial value for this property is False.
SuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed. This overrides the value of the DIMZIN system variable. The initial value for this property is False.
SymbolPosition	AcDimArcLengthSymbol	Gets and sets placement of the arc length dimension symbol. For a list of possible values for the AcDimArcLengthSymbol enumerated type, see Appendix B.
TextInside	Boolean	Gets or sets whether the text appears inside the extension lines. This overrides the value of DIMTIX system variable. The initial value for this property is True. If False, AutoCAD places it inside if there is room.
TextInsideAlign	Boolean	Gets or sets the position of any text inside/outside the extension lines. The values for these properties override the values of the DIMTIH or DIMTOH system variable, respectively. The initial value for this property is True. The TextInsideAlign property is available only when the TextInside property is set to True.
TextOutsideAlign	Boolean	Gets or sets the position of any text inside/outside the extension lines. The values for these properties override the values of the DIMTIH or DIMTOH system variables, respectively. The initial value for this property is True. The TextInsideAlign property is available only when the TextInside property is set to True.
ToleranceSuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed in tolerance values. This overrides the value of the DIMZIN system variable. The initial value for this property is False.

Name	Returns	Description
ToleranceSuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed in tolerance values. This overrides the value of the DIMZIN system variable. The initial value for this property is False.
TrueColor	AcCmColor	Gets or sets an object's color. Colors are identified by an AcCmColor object. This object can hold an RGB value, an ACI number (an integer from 1 to 255), or a named color. Using an RGB value, you can choose from millions of colors.
UnitsFormat	AcDimLUnits	Gets or sets the unit format. The initial value for this property is acDimLDecimal, where the format specified by the DecimalSeparator and PrimaryUnitsPrecision properties will be used to format the decimal value. For a list of possible values for the AcDimLUnits enumerated type, see Appendix B.

■ **Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadDimDiametric Object

The AcadDimDiametric object represents a dimension of the diameter of a circle or arc. The position of the text, set in the TextPosition property, determines the location of the dimension line. The AcadDimDiametric object is created using the AddDimDiametric method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadDimDiametric Object Methods

The AcadDimDiametric object inherits all the properties of the AcadEntity and AcadObject objects. It supports no other methods.

AcadDimDiametric Object Properties

The AcadDimDiametric object supports all the properties inherited from the AcadEntity object, the AcadObject object, and the AcadDimension object, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
AltRoundDistance	Double	Gets or sets the rounding of alternative units if the AltUnits property is turned on. This property and the two that follow override the value of the DIMALTZ system variable.
AltSuppressLeadingZeros	Boolean	Gets or sets whether leading zeros in alternative dimension values are suppressed. The initial value for this property is False.
AltSuppressTrailingZeros	Boolean	Gets or sets whether trailing zeros in alternative dimension values are suppressed. The initial value for this property is False.

Continued

Name	Returns	Description
AltSuppressZeroFeet AltSuppressZeroInches	Boolean	Gets or sets whether a zero foot/inches meas urement is suppressed in alternative dimension values. Both properties change the value of the DIMALTZ system variable. The initial value for this property is False.
AltTextPrefix AltTextSuffix	String	Gets or sets a prefix/suffix for the alternative dimension measurement. Both properties override the value of the DIMAPOST system variable. The initial value for this property is a NULL string. To turn off an established prefix, set this property equal to a single period (.).
AltTolerancePrecision	AcDimPrecision	Gets or sets the precision of tolerance values in alternative dimensions. The initial value for this property is acDimPrecisionTwo. This property is available only when the ToleranceDisplay property is set to any value other than acTolNone, and it overrides the value of the DIMTDEC system variable. For a list of the values for the AcDimPrecision enumerated type, see Appendix B.
AltToleranceSuppressLeadingZeros	Boolean	Gets or sets whether leading zeros are suppressed in alternative dimension values. This property and the three that follow override the DIMALTTZ system variable. The initial value for this property is False.
AltToleranceSuppressTrailingZeros	Boolean	Gets or sets whether trailing zeros are suppressed in alternative dimension values. The initial value for this property is False.
AltToleranceSuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed in alternative tolerance values. The initial value for this property is False.
AltToleranceSuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed in alternative tolerance values. The initial value for this property is False.
AltUnits	Boolean	Gets or sets whether alternative units dimensioning is enabled. It overrides the value of the DIMALT system variable.
AltUnitsFormat	AcDimUnits	Gets or sets the units format for alternative units. The initial value for this property is acDimDecimal. It overrides the value of the DIMALTU system variable. For a list of possible values for the AcDimUnits enumerated type, see Appendix B.
AltUnitsPrecision	AcDimPrecision	Gets or sets the number of decimal places in alternative units. It overrides the value of the DIMALTD system variable. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.

Name	Returns	Description
AltUnitsScale	Double	Gets or sets the scale factor for alternative units. If the AltUnits property is turned on, this property multiplies linear dimensions by a factor to produce a value in an alternate system of measurement. The initial value represents the number of millimeters in an inch. It overrides the value of the DIMALTF system variable.
Arrowhead1Block/Arrowhead2Block	String	Gets or sets the block used as the custom arrowhead at the first/second end of the dimension line. It overrides the value of the DIMBLK1 or DIMBLK2 system variable.
AcDimArrowheadType		Gets or sets the type of arrowhead at the first/second end of the dimension line. For a list of possible values for the AcDimArrowheadType enumerated type, see Appendix B.
ArrowheadSize	Double	Gets or sets the size of the dimension line arrowheads, leader line arrowheads, and hook lines. It overrides the value of the DIMASZ system variable. The initial value for this property is 0.1800.
CenterMarkSize	Double	Gets or sets the size of the center mark. It overrides the value of the DIMCEN system variable. The initial value for this property is 0.0900. This property isn't available if the CenterType property is set to acCenterNone.
CenterType	AcDimCenterType	Gets or sets the type of the center mark. The center mark is visible only if you place the dimension line outside the circle or arc. It overrides the value of the DIMCEN system variable. For a list of possible values for the AcDimCenterType enumerated type, see Appendix B.
DimensionLineColor	AcColor	Gets or sets the color of the dimension lines. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in the default color) or acByLayer (where AutoCAD draws objects in the color specified for the layer). For a list of possible values for the AcColor enumerated type, see Appendix B.
DimensionLineWeight	AcLineWeight	Gets or sets the lineweight of the dimension lines. It overrides the value of the DIMLWD system variable. For a list of possible values for the AcLineWeight enumerated type, see Appendix B.
DimLine1Suppress DimLine2Suppress	Boolean	Gets or sets whether the first/second dimension line is suppressed. It overrides the value of the DIMSD1 or DIMSD2 system variable. The initial value for this property is False. When this property is set to True, the display of the dimension line and arrowhead between the first extension line and the text is suppressed.

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Name	Returns	Description
`Fit`	`AcDimFit`	Gets or sets whether text and arrowheads are placed inside or outside extension lines given the availability of space between the extension lines. This property sets priorities for moving text and arrowheads when space isn't available to fit both within the extension lines. It overrides the value of the `DIMAFIT` system variable. The initial value for this property is `acBestFit`. For a list of the possible values, see Appendix B.
`ForceLineInside`	Boolean	Gets or sets whether a dimension line is drawn between the extension lines when the text is outside the extension lines. It overrides the value of the `DIMTOFL` system variable. The initial value for this property is `False`.
`FractionFormat`	`AcDimFractionType`	Gets or sets the formats of fractional values. It overrides the value of the `DIMFRAC` system variable. This property is used when the `UnitsFormat` property is set to `acDimLArchitectural` or `acDimLFractional`. Possible values: `acHorizontal`, `acDiagonal`, and `acNotStacked`.
`HorizontalTextPosition`	`AcDimHorizontalJustification`	Gets or sets the horizontal position of the text. For a list of possible values for the `AcDimHorizontalJustification` enumerated type, see Appendix B.
`LeaderLength`	Double	Sets the length of the leader. This property is write-only; it's used when the object is created and isn't stored.
`LinearScaleFactor`	Double	Gets or sets the global scale factor for linear dimensioning measurements. It overrides the value of the `DIMLFAC` system variable. The initial value for this property is 1.0000. When `LinearScaleFactor` is assigned a negative value, the factor is applied only in paper space.
`Measurement`	Double	Gets the measurement for the dimension. This property overrides the value for the `DIMLUNIT` system variable. This property's value is read-only.
`PrimaryUnitsPrecision`	`AcDimPrecision`	Gets or sets the number of decimal places displayed for the primary units. For a list of possible values for the `AcDimPrecision` enumerated type, see Appendix B.
`RoundDistance`	Double	Gets or sets the rounding of units. It overrides the value of the `DIMRND` system variable.
`SuppressZeroFeet`	Boolean	Gets or sets whether a zero foot measurement is suppressed. It overrides the value of the `DIMZIN` system variable. The initial value for this property is `False`.

Name	Returns	Description
SuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
TextInside	Boolean	Gets or sets whether the text appears inside the extension lines. It overrides the value of DIMTIX system variable. The initial value for this property is True. If False, AutoCAD places it inside if there's room.
TextInsideAlign	Boolean	Gets or sets the position of any text inside/outside the extension lines. The values for these properties override the values of the DIMTIH and DIMTOH system variables, respectively. The initial value for this property is True. The TextInsideAlign property is available only when the TextInside property is set to True.
ToleranceSuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed in tolerance values. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
ToleranceSuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed in tolerance values. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
UnitsFormat	AcDimLUnits	Gets or sets the unit format. The initial value for this property is acDimLDecimal, where the format specified by the DecimalSeparator and PrimaryUnitsPrecision properties will be used to format the decimal value. For a list of possible values for the AcDimLUnits enumerated type, see Appendix B.

■**Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadDimOrdinate Object

The AcadDimOrdinate object represents a dimension of the absolute X or Y position of a point from the origin. Ordinate dimensions display the X or Y UCS coordinate of an object along with a simple leader line. The absolute value of the coordinate is used according to the prevailing standards for ordinate dimensions. To change the extension lines of an ordinate dimension, change the Rotation property and/or the TextPosition property. The extension lines will be recalculated to fit the new requirements. The AcadDimOrdinate object is created using the AddDimOrdinate method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadDimOrdinate Object Methods

The AcadDimOrdinate object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadDimOrdinate Object Properties

The AcadDimOrdinate object inherits all the properties of the AcadEntity object, the AcadObject object, and the AcadDimension object, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
AltRoundDistance	Double	Gets or sets the rounding of alternative units if the AltUnits property is turned on. This property and the two that follow override the value of the DIMALTZ system variable.
AltSuppressLeadingZeros	Boolean	Gets or sets whether leading zeros in alternative dimension values are suppressed. The initial value for this property is False.
AltSuppressTrailingZeros	Boolean	Gets or sets whether trailing zeros in alternative dimension values are suppressed. The initial value for this property is False.
AltSuppressZeroFeet AltSuppressZeroInches	Boolean	Gets or sets whether a zero foot/inches measurement is suppressed in alternative dimension values. Both properties change the value of the DIMALTZ system variable. The initial value for this property is False.
AltTextPrefix AltTextSuffix	String	Gets or sets a prefix/suffix for the alternative dimension measurement. Both properties override the value of the DIMAPOST system variable. The initial value for this property is a NULL string. To turn off an established prefix, set this property equal to a single period (.).
AltTolerancePrecision	AcDimPrecision	Gets or sets the precision of tolerance values in alternative dimensions. The initial value for this property is acDimPrecisionTwo. This property is available only when the ToleranceDisplay property is set to any value other than acTolNone, and it overrides the value of the DIMTDEC system variable. For a list of the values for the AcDimPrecision enumerated type, see Appendix B.
AltToleranceSuppressLeadingZeros	Boolean	Gets or sets whether leading zeros are suppressed in alternative dimension values. This property and the three that follow override the DIMALTTZ system variable. The initial value for this property is False.
AltToleranceSuppressTrailingZeros	Boolean	Gets or sets whether trailing zeros are suppressed in alternative dimension values. The initial value for this property is False.
AltToleranceSuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed in alternative tolerance values. The initial value for this property is False.
AltToleranceSuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed in alternative tolerance values. The initial value for this property is False.

Name	Returns	Description
AltUnits	Boolean	Gets or sets whether alternative units dimensioning is enabled. It overrides the value of the DIMALT system variable.
AltUnitsFormat	AcDimUnits	Gets or sets the units format for alternative units. The initial value for this property is acDimDecimal. It overrides the value of the DIMALTU system variable. For a list of possible values for the AcDimUnits enumerated type, see Appendix B.
AltUnitsPrecision	AcDimPrecision	Gets or sets the number of decimal places in alternative units. It overrides the value of the DIMALTD system variable. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.
AltUnitsScale	Double	Gets or sets the scale factor for alternative units. If the AltUnits property is turned on, this property multiplies linear dimensions by a factor to produce a value in an alternate system of measurement. The initial value represents the number of millimeters in an inch. It overrides the value of the DIMALTF system variable.
ArrowheadSize	Double	Gets or sets the size of the dimension line arrowheads, leader line arrowheads, and hook lines. It overrides the value of the DIMASZ system variable. The initial value for this property is 0.1800.
ExtensionLineColor	AcColor	Gets or sets the color of the extension lines. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in the default color), acByLayer (where AutoCAD draws objects in the color specified for the layer), or the AcColor enumerated type (see the AcadEntity object in Appendix B). It overrides the value of the DIMCLRE system variable.
ExtensionLineOffset	Double	Gets or sets the distance that the extension lines are offset from the origin points. It overrides the value of the DIMEXO system variable. The initial value for this property is 0.0625.
ExtensionLineWeight	AcLineWeight	Gets or sets the lineweight of the extension lines. It overrides the value of the DIMLWE system variable. For a list of possible values for the AcLineWeight enumerated type, see Appendix B.
FractionFormat	AcDimFractionType	Gets or sets the formats of fractional values. It overrides the value of the DIMFRAC system variable. This property is used when the UnitsFormat property is set to acDimLArchitectural or acDimLFractional. Possible values: acHorizontal, acDiagonal, and acNotStacked.
LinearScaleFactor	Double	Gets or sets the global scale factor for linear dimensioning measurements. It overrides the value of the DIMLFAC system variable. The initial value for this property is 1.0000. When LinearScaleFactor is assigned a negative value, the factor is applied only in paper space.
Measurement	Double	Gets the measurement for the dimension. This property overrides the value for the DIMLUNIT system variable. This property's value is read-only.
PrimaryUnitsPrecision	AcDimPrecision	Gets or sets the number of decimal places displayed for the primary units. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.

Continued

Name	Returns	Description
RoundDistance	Double	Gets or sets the rounding of units. It overrides the value of the DIMRND system variable.
SuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
SuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
TextInside	Boolean	Gets or sets whether the text appears inside the extension lines. It overrides the value of DIMTIX system variable. The initial value for this property is True. If False, AutoCAD places it inside if there's room.
TextInsideAlign	Boolean	Gets or sets the position of any text inside/ outside the extension lines. The values for these properties override the values of the DIMTIH and DIMTOH system variables, respectively. The initial value for this property is True. The TextInsideAlign property is available only when the TextInside property is set to True.
ToleranceSuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed in tolerance values. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
ToleranceSuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed in tolerance values. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
UnitsFormat	AcDimLUnits	Gets or sets the unit format. The initial value for this property is acDimLDecimal, where the format specified by the DecimalSeparator and PrimaryUnitsPrecision properties will be used to format the decimal value. For a list of possible values for the AcDimLUnits enumerated type, see Appendix B.

Note Because this object inherits from AcadObject, it supports the Modified event.

AcadDimRadial Object

The AcadDimRadial object represents a dimension of the radius of a circle or arc. The position of the text, set in the TextPosition property, determines the location of the dimension line. The AcadDimRadial object is created using the AddDimRadial method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadDimRadial Object Methods

The AcadDimRadial object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadDimRadial Object Properties

The AcadDimRadial object inherits all the properties of the AcadEntity object, the AcadObject object, and the AcadDimension object, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
AltRoundDistance	Double	Gets or sets the rounding of alternative units if the AltUnits property is turned on. This property and the two that follow override the value of the DIMALTZ system variable.
AltSuppressLeadingZeros	Boolean	Gets or sets whether leading zeros in alternative dimension values are suppressed. The initial value for this property is False.
AltSuppressTrailingZeros	Boolean	Gets or sets whether trailing zeros in alternative dimension values are suppressed. The initial value for this property is False.
AltSuppressZeroFeet AltSuppressZeroInches	Boolean	Gets or sets whether a zero foot/inches measurement is suppressed in alternative dimension values. Both properties change the value of the DIMALTZ system variable. The initial value for this property is False.
AltTextPrefix AltTextSuffix	String	Gets or sets a prefix/suffix for the alternative dimension measurement. Both properties override the value of the DIMAPOST system variable. The initial value for this property is a NULL string. To turn off an established prefix, set this property equal to a single period (.).
AltTolerancePrecision	AcDimPrecision	Gets or sets the precision of tolerance values in alternative dimensions. The initial value for this property is acDimPrecisionTwo. This property is available only when the ToleranceDisplay property is set to any value other than acTolNone, and it overrides the value of the DIMTDEC system variable. For a list of the values for the AcDimPrecision enumerated type, see Appendix B.
AltToleranceSuppressLeadingZeros	Boolean	Gets or sets whether leading zeros are suppressed in alternative dimension values. This property and the three that follow override the DIMALTTZ system variable. The initial value for this property is False.
AltToleranceSuppressTrailingZeros	Boolean	Gets or sets whether trailing zeros are suppressed in alternative dimension values. The initial value for this property is False.
AltToleranceSuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed in alternative tolerance values. The initial value for this property is False.
AltToleranceSuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed in alternative tolerance values. The initial value for this property is False.

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Name	Returns	Description
AltUnits	Boolean	Gets or sets whether alternative units dimensioning is enabled. It overrides the value of the DIMALT system variable.
AltUnitsFormat	AcDimUnits	Gets or sets the units format for alternative units. The initial value for this property is acDimDecimal. It overrides the value of the DIMALTU system variable. For a list of possible values for the AcDimUnits enumerated type, see Appendix B.
AltUnitsPrecision	AcDimPrecision	Gets or sets the number of decimal places in alternative units. It overrides the value of the DIMALTD system variable. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.
AltUnitsScale	Double	Gets or sets the scale factor for alternative units. If the AltUnits property is turned on, this property multiplies linear dimensions by a factor to produce a value in an alternate system of measurement. The initial value represents the number of millimeters in an inch. It overrides the value of the DIMALTF system variable.
ArrowheadBlock	String	Gets or sets the block used as the custom arrowhead.
ArrowheadSize	Double	Gets or sets the size of the dimension line arrowheads, leader line arrowheads, and hook lines. It overrides the value of the DIMASZ system variable. The initial value for this property is 0.1800.
ArrowheadType	AcDimArrowheadType	Gets or sets the type of the arrowhead. It overrides the value of the DIMLDRBLK system variable. For a list of possible values for the AcDimArrowheadType enumerated type, see Appendix B.
CenterMarkSize	Double	Gets or sets the size of the center mark. The initial value for this property is 0.0900. This property isn't available if the CenterType property is set to acCenterNone.
CenterType	AcDimCenterType	Gets or sets the type of the center mark. The center mark is visible only if you place the dimension line outside the circle or arc. It overrides the value of the DIMCEN system variable. For a list of possible values for the AcDimCenterType enumerated type, see Appendix B.
DimensionLineColor	AcColor	Gets or sets the color of the dimension lines. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in the default color) or acByLayer (where AutoCAD draws objects in the color specified for the layer). For a list of possible values for the AcColor enumerated type, see Appendix B.
DimensionLineWeight	AcLineWeight	Gets or sets the lineweight of the dimension lines. It overrides the value of the DIMLWD system variable. For a list of possible values for the AcLineWeight enumerated type, see Appendix B.
DimLineSuppress	Boolean	Specifies whether the dimension line is suppressed. It overrides the value of the DIMSD2 system variable. The initial value for this property is False. When this property is set to True, the display of the dimension line and arrowhead between the extension line and the text is suppressed.

Name	Returns	Description
Fit	AcDimFit	Gets or sets whether text and arrowheads are placed inside or outside extension lines given the availability of space between the extension lines. This property sets priorities for moving text and arrowheads when space isn't available to fit both within the extension lines. It overrides the value of the DIMAFIT system variable. The initial value for this property is acBestFit. For a list of the possible values, see Appendix B.
ForceLineInside	Boolean	Gets or sets whether a dimension line is drawn between the extension lines when the text is outside the extension lines. It overrides the value of the DIMTOFL system variable. The initial value for this property is False.
FractionFormat	AcDimFractionType	Gets or sets the formats of fractional values. It overrides the value of the DIMFRAC system variable. This property is used when the UnitsFormat property is set to acDimLArchitectural or acDimLFractional. Possible values: acHorizontal, acDiagonal, and acNotStacked.
HorizontalTextPosition	AcDimHorizontalJustification	Gets or sets the horizontal position of the text. For a list of possible values for the AcDimHorizontalJustification enumerated type, see Appendix B.
LeaderLength	Double	Sets the length of the leader. This property is write-only; it's used when the object is created and isn't stored.
LinearScaleFactor	Double	Gets or sets the global scale factor for linear dimensioning measurements. It overrides the value of the DIMLFAC system variable. The initial value for this property is 1.0000. When LinearScaleFactor is assigned a negative value, the factor is applied only in paper space.
Measurement	Double	Gets the measurement for the dimension. This property overrides the value for the DIMLUNIT system variable. This property's value is read-only.
PrimaryUnitsPrecision	AcDimPrecision	Gets or sets the number of decimal places displayed for the primary units. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.
RoundDistance	Double	Gets or sets the rounding of units. It overrides the value of the DIMRND system variable.
SuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed. It overrides the value of the DIMZIN system variable. The initial value for this property is False.

Continued

Name	Returns	Description
SuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
TextInside	Boolean	Gets or sets whether the text appears inside the extension lines. It overrides the value of the DIMTIX system variable. The initial value for this property is True. If False, AutoCAD places it inside if there's room.
TextInsideAlign	Boolean	Gets or sets the position of any text inside/outside the extension lines. The values for these properties override the values of the DIMTIH and DIMTOH system variables, respectively. The initial value for this property is True. The TextInsideAlign property is available only when the TextInside property is set to True.
ToleranceSuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed in tolerance values. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
ToleranceSuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed in tolerance values. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
UnitsFormat	AcDimLUnits	Gets or sets the unit format. The initial value for this property is acDimLDecimal, where the format specified by the DecimalSeparator and PrimaryUnitsPrecision properties will be used to format the decimal value. For a list of possible values for the AcDimLUnits enumerated type, see Appendix B.

■**Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadDimRadialLarge Object

This is a dimension measuring a large radius, also known as a *jogged radius dimension*.

AcadDimRadialLarge Object Methods

The AcadDimRadialLarge object inherits all the methods of the AcadEntity and AcadObject objects. It doesn't support any other methods.

AcadDimRadialLarge Object Properties

The AcadDimRadialLarge object inherits all the properties of the AcadEntity object, the AcadObject object, and the AcadDimension object, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
AltRoundDistance	Double	Gets or sets the rounding of alternative units, if the AltUnits property is turned on. This property, and the two that follow, all override the value of the DIMALTZ system variable.
AltSuppressLeadingZeros	Boolean	Gets or sets whether leading zeros in alternative dimension values are suppressed. The initial value for this property is False.
AltSuppressTrailingZeros	Boolean	Gets or sets whether trailing zeros in alternative dimension values are suppressed. The initial value for this property is False.
AltSuppressZeroFeet AltSuppressZeroInches	Boolean	Gets or sets whether a zero foot/inches measurement is suppressed in alternative dimension values. Both properties change the value of the DIMALTZ system variable. The initial value for this property is False.
AltTextPrefix AltTextSuffix	String	Gets or sets a prefix/suffix for the alternative dimension measurement. Both properties override the value of the DIMAPOST system variable. The initial value for this property is a NULL string. To turn off an established prefix, set this property equal to a single period (.).
AltTolerancePrecision	AcDimPrecision	Gets or sets the precision of tolerance values in alternative dimensions. The initial value for this property is acDimPrecisionTwo. This property is available only when the ToleranceDisplay property is set to any value other than acTolNone, and it overrides the value of the DIMTDEC system variable. For a list of the values for the AcDimPrecision enumerated type, see Appendix B.
AltToleranceSuppressLeadingZeros	Boolean	Gets or sets whether leading zeros are suppressed in alternative dimension values. This property, and the three that follow, all override the DIMALTTZ system variable. The initial value for this property is False.
AltToleranceSuppressTrailingZeros	Boolean	Gets or sets whether trailing zeros are suppressed in alternative dimension values. The initial value for this property is False.
AltToleranceSuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed in alternative tolerance values. The initial value for this property is False.
AltToleranceSuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed in alternative tolerance values. The initial value for this property is False.
AltUnits	Boolean	Gets or sets whether alternative units dimensioning is enabled. This overrides the value of the DIMALT system variable.
AltUnitsFormat	AcDimUnits	Gets or sets the units format for alternative units. The initial value for this property is acDimDecimal. This overrides the value of the DIMALTU system variable. For a list of possible values for the AcDimUnits enumerated type, see Appendix B.

Continued

Name	Returns	Description
AltUnitsPrecision	AcDimPrecision	Gets or sets the number of decimal places in alternative units. This overrides the value of the DIMALTD system variable. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.
AltUnitsScale	Double	Gets or sets the scale factor for alternative units. If the AltUnits property is turned on, this property multiplies linear dimensions by a factor to produce a value in an alternate system of measurement. The initial value represents the number of millimeters in an inch. This overrides the value of the DIMALTF system variable.
ArrowheadBlock	String	Gets or sets the block used as the custom arrowhead at the end of the dimension line. This overrides the value of the DIMBLK1 or DIMBLK2 system variable.
ArrowheadSize	Double	Gets or sets the size of the dimension line arrowheads, leader line arrowheads, and hook lines. This overrides the value of the DIMASZ system variable. The initial value for this property is 0.1800.
ArrowheadType	AcDimArrowheadType	Gets or sets the type of arrowhead at the end of the dimension line. For a list of possible values for the AcDimArrowheadType enumerated type, see Appendix B.
Center	Variant	Gets or sets the center of the circle as a set of 3-D coordinates.
CenterMarkSize	Double	Gets or sets the size of the center mark. This overrides the value of the DIMCEN system variable. The initial value for this property is 0.0900. This property is not available if the CenterType property is set to acCenterNone.
CenterType	AcDimCenterType	Gets or sets the type of the center mark. The center mark is visible only if you place the dimension line outside the circle or arc. This overrides the value of the DIMCEN system variable. For a list of possible values for the AcDimCenterType enumeration, see Appendix B.
ChordPoint	Variant	Gets or sets the chord point for the arc as a set of 3-D coordinates.
DimensionLineColor	AcColor	Gets or sets the color of the dimension lines. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in default color) or acByLayer (where AutoCAD draws objects in the color specified for the layer). For a list of possible values for the AcColor enumerated type, see Appendix B.
DimensionLineWeight	AcLineWeight	Gets or sets the line weight of the dimension lines. This overrides the value of the DIMLWD system variable. For a list of possible values for the AcLineWeight enumerated type, see Appendix B.
DimLineSuppress	Boolean	Gets or sets whether the dimension line is suppressed. This overrides the value of the DIMSD1 or DIMSD2 system variable. The initial value for this property is False. When this property is set to True, the display of the dimension line and arrowhead between the first extension line and the text is suppressed.

Name	Returns	Description
Fit	AcDimFit	Gets or sets whether text and arrowheads are placed inside or outside extension lines given the availability of space between the extension lines. This property sets priorities for moving text and arrowheads when space is not available to fit both within the extension lines. This overrides the value of the DIMAFIT system variable. The initial value for this property is acBestFit. For a list of the possible values, see Appendix B.
ForceLineInside	Boolean	Gets or sets whether a dimension line is drawn between the extension lines when the text is outside the extension lines. This overrides the value of the DIMTOFL system variable. The initial value for this property is False.
FractionFormat	AcDimFractionType	Gets or sets the formats of fractional values. This overrides the value of the DIMFRAC system variable. This property is used when the UnitsFormat property is set to acDimLArchitectural or acDimLFractional. Possible values: acHorizontal, acDiagonal and acNotStacked.
HorizontalTextPosition	AcDimHorizontalJustification	Gets or sets the horizontal position of the text. This overrides the value of the DIMJUST system variable. For a list of possible values for the AcDimHorizontalJustification enumeration, see Appendix B.
JogAngle	ACAD_ANGLE	Gets or sets the jog angle. For a list of possible values for the ACAD_ANGLE enumeration, see Appendix B
JogLocation	Variant	Gets or sets the jog location or user pick point as a set of 3-D coordinates.
LinearScaleFactor	Double	Gets or sets the global scale factor for linear dimensioning measurements. This overrides the value of the DIMLFAC system variable. The initial value for this property is 1.0000. When LinearScaleFactor is assigned a negative value, the factor is applied only in paper space.
Measurement	Double	Gets the measurement for the dimension. This property overrides the value for the DIMLUNIT system variable. It is read-only.
OverrideCenter	Variant	Gets or sets the override center or pick point as a set of 3-D coordinates.
PrimaryUnitsPrecision	AcDimPrecision	Gets or sets the number of decimal places displayed for the primary units. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.

Continued

Name	Returns	Description
RoundDistance	Double	Gets or sets the rounding of units. This overrides the value of the DIMRND system variable.
SuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed. This overrides the value of the DIMZIN system variable. The initial value for this property is False.
SuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed. This overrides the value of the DIMZIN system variable. The initial value for this property is False.
TextInside	Boolean	Gets or sets whether the text appears inside the extension lines. This overrides the value of DIMTIX system variable. The initial value for this property is True. If False, AutoCAD places it inside if there is room.
TextInsideAlign	Boolean	Gets or sets the position of any text inside/outside the extension lines. The values for these properties override the values of the DIMTIH or DIMTOH system variable, respectively. The initial value for this property is True. The TextInsideAlign property is available only when the TextInside property is set to True.
TextOutsideAlign	Boolean	Gets or sets the position of any text inside/outside the extension lines. The values for these properties override the values of the DIMTIH or DIMTOH system variable, respectively. The initial value for this property is True. The TextInsideAlign property is available only when the TextInside property is set to True.
ToleranceSuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed in tolerance values. This overrides the value of the DIMZIN system variable. The initial value for this property is False.
ToleranceSuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed in tolerance values. This overrides the value of the DIMZIN system variable. The initial value for this property is False.
TrueColor	AcCmColor	Gets or sets an object's color. Colors are identified by an AcCmColor object. This object can hold an RGB value, an ACI number (an integer from 1 to 255), or a named color. Using an RGB value, you can choose from millions of colors.
UnitsFormat	AcDimLUnits	Gets or sets the unit format. The initial value for this property is acDimLDecimal, where the format specified by the DecimalSeparator and PrimaryUnitsPrecision properties will be used to format the decimal value. For a list of possible values for the AcDimLUnits enumeration, see Appendix B.

■**Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadDimRotated Object

The AcadDimRotated object represents a dimension that measures the distance between two points and is displayed at a given rotation. To change the extension lines of a rotated dimension, change the Rotation property and/or the TextPosition property. The extension lines will be recalculated to fit the new requirements. The AcadDimRotated object is created using the AddCircle method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadDimRotated Object Methods

The AcadDimRotated object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadDimRotated Object Properties

The AcadDimRadial object inherits all the properties of the AcadEntity object, the AcadObject object, and the AcadDimension object, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
AltRoundDistance	Double	Gets or sets the rounding of alternative units if the AltUnits property is turned on. This property and the two that follow override the value of the DIMALTZ system variable.
AltSuppressLeadingZeros	Boolean	Gets or sets whether leading zeros in alternative dimension values are suppressed. The initial value for this property is False.
AltSuppressTrailingZeros	Boolean	Gets or sets whether trailing zeros in alternative dimension values are suppressed. The initial value for this property is False.
AltSuppressZeroFeet AltSuppressZeroInches	Boolean	Gets or sets whether a zero foot/inches measurement is suppressed in alternative dimension values. Both properties change the value of the DIMALTZ system variable. The initial value for this property is False.
AltTextPrefix AltTextSuffix	String	Gets or sets a prefix/suffix for the alternative dimension measurement. Both properties override the value of the DIMAPOST system variable. The initial value for this property is a NULL string. To turn off an established prefix, set this property equal to a single period (.).
AltTolerancePrecision	AcDimPrecision	Gets or sets the precision of tolerance values in alternative dimensions. The initial value for this property is acDimPrecisionTwo. This property is available only when the ToleranceDisplay property is set to any value other than acTolNone, and it overrides the value of the DIMTDEC system variable. For a list of the values for the AcDimPrecision enumerated type, see Appendix B.

Continued

Name	Returns	Description
AltToleranceSuppressLeadingZeros	Boolean	Gets or sets whether leading zeros are suppressed in alternative dimension values. This property and the three that follow override the DIMALTTZ system variable. The initial value for this property is False.
AltToleranceSuppressTrailingZeros	Boolean	Gets or sets whether trailing zeros are suppressed in alternative dimension values. The initial value for this property is False.
AltToleranceSuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed in alternative tolerance values. The initial value for this property is False.
AltToleranceSuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed in alternative tolerance values. The initial value for this property is False.
AltUnits	Boolean	Gets or sets whether alternative units dimensioning is enabled. It overrides the value of the DIMALT system variable.
AltUnitsFormat	AcDimUnits	Gets or sets the units format for alternative units. The initial value for this property is acDimDecimal. It overrides the value of the DIMALTU system variable. For a list of possible values for the AcDimUnits enumerated type, see Appendix B.
AltUnitsPrecision	AcDimPrecision	Gets or sets the number of decimal places in alternative units. It overrides the value of the DIMALTD system variable. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.
AltUnitsScale	Double	Gets or sets the scale factor for alternative units. If the AltUnits property is turned on, this property multiplies linear dimensions by a factor to produce a value in an alternate system of measurement. The initial value represents the number of millimeters in an inch. It overrides the value of the DIMALTF system variable.
ArrowheadBlock	String	Gets or sets the block used as the custom arrowhead at the first/second end of the dimension line. It overrides the value of the DIMBLK1 or DIMBLK2 system variable.
ArrowheadSize	Double	Gets or sets the size of the dimension line arrowheads, leader line arrowheads, and hook lines. It overrides the value of the DIMASZ system variable. The initial value for this property is 0.1800.
ArrowheadType	AcDimArrowheadType	Gets or sets the type of arrowhead at the first/second end of the dimension line. For a list of possible values for the AcDimArrowheadType enumerated type, see Appendix B.

Name	Returns	Description
DimensionLineColor	AcColor	Gets or sets the color of the dimension lines. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in the default color) or acByLayer (where AutoCAD draws objects in the color specified for the layer). For a list of possible values for the AcColor enumerated type, see Appendix B.
DimensionLineExtend	Double	Gets or sets the distance the dimension line extends beyond the extension line when oblique strokes rather than arrowheads are used. It overrides the value of the DIMDLE system variable. The initial value for this property is 0.0000.
DimLineInside	Boolean	Gets or sets whether the dimension lines are displayed only inside the extension lines. It overrides the value of the DIMSOXD system variable. The initial value for this property is False.
DimensionLineWeight	AcLineWeight	Gets or sets the lineweight of the dimension lines. It overrides the value of the DIMLWD system variable. For a list of possible values for the AcLineWeight enumerated type, see Appendix B.
DimLineSuppress	Boolean	Gets or sets whether the first/second dimension line is suppressed. It overrides the value of the DIMSD1 or DIMSD2 system variable. The initial value for this property is False. When this property is set to True, the display of the dimension line and arrowhead between the first extension line and the text is suppressed.
ExtensionLineColor	AcColor	Gets or sets the color of the extension lines. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in the default color), acByLayer (where AutoCAD draws objects in the color specified for the layer), or the AcColor enumerated type (see Appendix B). It overrides the value of the DIMCLRE system variable.
ExtensionLineExtend	Double	Gets or sets the distance that the extension line extends beyond the dimension line. It overrides the value of the DIMEXE system variable. The initial value for this property is 0.1800.
ExtensionLineOffset	Double	Gets or sets the distance that the extension lines are offset from the origin points. It overrides the value of the DIMEXO system variable. The initial value for this property is 0.0625.
ExtensionLineWeight	AcLineWeight	Gets or sets the lineweight of the extension lines. It overrides the value of the DIMLWE system variable. For a list of possible values for the AcLineWeight enumerated type, see Appendix B.
ExtLine1Suppress ExtLine2Suppress	Boolean	Gets or sets whether the first/second extension line is sup pressed. It overrides the value of the DIMSE1 or DIMSE2 system variable. The initial value for this property is False.
Fit	AcDimFit	Gets or sets whether text and arrowheads are placed inside or outside extension lines given the availability of space between the extension lines. This property sets priorities for moving text and arrowheads when space isn't available to fit both within the extension lines. It overrides the value of the DIMAFIT system variable. The initial value for this property is acBestFit. For a list of the possible values, see Appendix B.
ForceLineInside	Boolean	Gets or sets whether a dimension line is drawn between the extension lines when the text is outside the extension lines. It overrides the value of the DIMTOFL system variable. The initial value for this property is False.

Continued

Name	Returns	Description
FractionFormat	AcDimFractionType	Gets or sets the formats of fractional values. It overrides the value of the DIMFRAC system variable. This property is used when the UnitsFormat property is set to acDimLArchitectural or acDimLFractional. Possible values: acHorizontal, acDiagonal, and acNotStacked.
HorizontalTextPosition	AcDimHorizontalJustification	Gets or sets the horizontal position of the text. It overrides the value of the DIMJUST system variable. For a list of possible values for the AcDimHorizontalJustification enumerated type, see Appendix B.
LinearScaleFactor	Double	Gets or sets the global scale factor for linear dimensioning measurements. It overrides the value of the DIMLFAC system variable. The initial value for this property is 1.0000. When LinearScaleFactor is assigned a negative value, the factor is applied only in paper space.
Measurement	Double	Gets the measurement for the dimension. This property overrides the value for the DIMLUNIT system variable. This property's value is read-only.
PrimaryUnitsPrecision	AcDimPrecision	Gets or sets the number of decimal places displayed for the primary units. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.
RoundDistance	Double	Gets or sets the rounding of units. It overrides the value of the DIMRND system variable.
SuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
SuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
TextInside	Boolean	Gets or sets whether the text appears inside the extension lines. It overrides the value of DIMTIX system variable. The initial value for this property is True. If False, AutoCAD places it inside if there's room.
TextInsideAlign	Boolean	Gets or sets the position of any text inside/outside the extension lines. The values for these properties override the values of the DIMTIH and DIMTOH system variables, respectively. The initial value for this property is True. The TextInsideAlign property is available only when the TextInside property is set to True.

Name	Returns	Description
ToleranceSuppressZeroFeet	Boolean	Gets or sets whether a zero foot measurement is suppressed in tolerance values. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
ToleranceSuppressZeroInches	Boolean	Gets or sets whether a zero inch measurement is suppressed in tolerance values. It overrides the value of the DIMZIN system variable. The initial value for this property is False.
UnitsFormat	AcDimLUnits	Gets or sets the unit format. The initial value for this property is acDimLDecimal, where the format specified by the DecimalSeparator and PrimaryUnitsPrecision properties will be used to format the decimal value. For a list of possible values for the AcDimLUnits enumerated type, see Appendix B.

Note Because this object inherits from AcadObject, it supports the Modified event.

AcadDimStyle Object

The AcadDimStyle object represents a group of dimension settings that determine the appearance of new dimensions created in the drawing. It is created by the Add method of the AcadDimStyles collection and is activated through the ActiveDimStyle property of the Document object. To change the style of a dimension, use the StyleName property found on the dimension. To control the settings of the current document overrides, use the dimensioning system variables.

When you change a dimensioning system variable, you aren't changing the active dimension style itself. Rather, you're setting a document override for the dimension style, which means that all newly created dimensions will still be created with the active dimension style. They won't reflect the overrides from the system variables until the active dimension style is updated. To change the settings of any dimension style, use the CopyFrom method. Dimensions created via ActiveX use the active dimension style only. To apply the system overrides for dimensions created by ActiveX, use the CopyFrom method to copy the dimension style from the document to the active dimension style.

AcadDimStyle Object Methods

In addition to the methods inherited from the AcadObject object, the AcadDimStyle object supports the following method.

Name	Description
CopyFrom	Copies the dimension style data from a source object, which can be a document, dimension, or another dimension style. Parameter: StyleSource As Object. If StyleSource is an AcadDocument object, this method copies the active dimension style settings for the drawing plus any drawing overrides. If it's a dimension, an AcadTolerance object, or an AcadLeader object, this method copies the style for that object plus any object overrides. If StyleSource is an AcadDimStyle object, then the style data from that dimension style is copied.

AcadDimStyle Object Properties

In addition to the properties inherited from the AcadObject object and the common Application property, the AcadDimStyle object supports the following property.

Name	Returns	Description
Name	String	Gets or sets the object's name.

■**Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadDimStyles Collection

The AcadDimStyles collection contains all the dimension styles in the drawing. Although this collection inherits a Delete method, you can't actually delete it. If you need to delete a specific dimension style, use the Delete method found in the AcadDimStyle object. There's no limit to the number of dimension styles you can create in your drawing. However, there can be only one instance of the AcadDimStyles collection, which is predefined for each drawing. You can make multiple references to it by using the DimStyles property.

AcadDimStyles Collection Methods

In addition to the methods inherited from the AcadObject object, the AcadDimStyles collection supports the following methods.

Name	Returns	Description
Add	AcadDimStyle	Creates an AcadDimStyle object and adds it to the collection. Parameter: Name As String.
Item	AcadDimStyle	Gets the member object at a given index in the collection. Parameter: Index As Variant (Integer or String). If the Index value is a String, it must match an existing style definition.

AcadDimStyles Collection Properties

The AcadDimStyles collection supports the Count property, the common Application property, and the properties inherited from the AcadObject object. It supports no other properties.

■**Note** Because this collection inherits from AcadObject, it supports the Modified event.

AcadDocument Object

The AcadDocument object represents an AutoCAD drawing. The active document can be accessed using ThisDrawing in VBA and the ActiveDocument property of the AcadApplication object in Visual Basic.

Certain objects in the drawing establish the format, location, or style that new objects will adopt. These include the AcadDimStyle, AcadTextStyle, AcadLinetype, AcadViewport, and AcadLayer objects. A drawing may contain many objects of these types, but only one can be active at a time. The AcadDocument object also contains a link property to all the collections, all of which are created when the document is created. You can thus iterate through every object in the drawing.

You control the model space and paper space settings from the AcadDocument object by using its ActiveSpace property, its MSpace property, and the Display method of the AcadPViewport object. By default, a drawing is opened in the model space with the tiled viewport setting.

Changes to most active objects will appear immediately. However, several objects must be reset for changes to appear: the active text style, the active UCS, and the active viewport. These are reset using the ActiveTextStyle, ActiveUCS, and ActiveViewport properties, respectively. An example syntax for resetting the active viewport is as follows:

```
ThisDrawing.ActiveViewport = modifiedViewportObj
```

AcadDocument Object Methods

The AcadDocument object supports the following methods.

Name	Returns	Description
Activate		Makes the specified drawing active.
AuditInfo		Evaluates the drawing's integrity. For every error detected, AutoCAD provides a description of the error and recommends corrective action. Parameter: FixError As Boolean. If the parameter is set to True, AutoCAD should attempt to fix any errors.
Close		Closes the specified drawing. If no drawing is specified, the drawing is closed only if it already has a valid file name or wasn't modified. Parameters: SaveChanges As Boolean and FileName As String. If the first parameter is set to True, changes are saved. The second parameter specifies the name to assign to the drawing. The document will be closed and no error will occur if the drawing hasn't been modified or if the first parameter is set to False. The document will be saved and closed if the first parameter is set to True (the default value) and the drawing was previously saved or was opened from an existing drawing file. In any other case, a valid name has to be specified.
CopyObjects	Variant	Copies objects, their child objects, and so on recursively (deep cloning). Parameters: Objects As Variant, [Owner As Variant], and [IDPairs As Variant]. For more information on this method, see the AcadDatabaseObject section.
EndUndoMark		Marks the end of a block of operations. See also the StartUndoMark method.

Continued

Name	Returns	Description
Export		Exports the drawing to a .wmf, .sat, .eps, .dxf, or .bmp format. Parameters: FileName As String, Extension As String, and SelectionSet As AcadSelectionSet. When exporting to wmf, .sat, or .bmp format, a nonempty selection set must be provided. For the .sat format, only AcadRegion and Acad3DSolid objects can be exported. When exporting to .eps and .dxf formats, the selection set is ignored and the entire drawing is exported.
GetVariable	Variant	Returns the current setting of an AutoCAD system variable. For a list of all the AutoCAD system variables and their types, please refer to Appendix C. Parameter: Name As String.
HandleToObject	Object	Returns the object that corresponds to a handle. This method can return only objects in the current document. Parameter: Handle As String. A handle is persistent. It can be stored for reuse with this method for the lifetime of the object.
Import	Object	Imports a drawing in .wmf, .sat, .eps, or .dxf format. In the case of importing a .wmf file, an AcadBlockReference object is returned. In all other cases, the return value is NULL. Parameters: FileName As String, InsertionPoint As Variant, and ScaleFactor As Double.
LoadShapeFile		Loads an .shx (shape) file. This method makes all the shapes in the shape file available to the current drawing. Parameter: FullName As String.
New	AcadDocument	Creates a new document in SDI mode. It's good practice to immediately save the new document with a valid file name. This will simplify later using the save and close methods. Parameter: TemplateFileName As String.
ObjectIDToObject	Object	Returns the object that corresponds to an object ID. This method can return objects only in the current document. Parameter: ID As Long. An object ID isn't persistent. Its lifetime is limited to a drawing session.
Open		Opens an existing .dwg (drawing) file when working in SDI mode. The file becomes the active document. Parameters: Name As String and [ReadOnly As Boolean]. The Name parameter is an existing file name, including the path or the valid URL address of the drawing to open. The second parameter is True if the drawing is to be read-only and False otherwise (the default).
PurgeAll		Removes all unused named references from the document. Deleted objects remain in the document until they're purged using this method.
Regen		Regenerates the entire drawing and updates the screen coordinates and view resolution for all objects. Parameter: WhichViewports As AcRegenType (acActiveViewport or acAllViewports).
Save		Saves the document. When the user is saving to a secure URL, a dialog box will prompt the user for the necessary password information. Message boxes may also appear if the user hasn't suppressed this activity in the browser.

Name	Returns	Description
`SaveAs`		Saves the document as specified by the parameters. Parameters: `FileName As String` and `FileType As AcMenuFileType`. For a list of possible values for the `AcMenuFileType` enumeration, see Appendix B.
`SendCommand`		Sends a command string from a VB or VBA application to the document to be processed. If the drawing specified isn't active, it will be made active. This method processes any AutoCAD command-line function, including LISP expressions. This method is generally synchronous. Only when responding to user interaction or when called from an event handler will commands be processed asynchronously. You should never use this method to issue a command for which there is an ActiveX method available. Instead, use the `LoadDVB` method. Parameter: `Command As String`.
`SetVariable`		Sets the value of an AutoCAD system variable. For a list of all AutoCAD system variables and their types, please refer to Appendix C. Parameters: `Name As String` and `Value As Variant`.
`StartUndoMark`		Marks the beginning of a block of operations that will be treated as a unique operation in an Undo command. Unfortunately, the Undo command hasn't been wrapped for VBA but is available synchronously using the `SendCommand` method.
`WBlock`		Creates a new drawing file from the selection set. Parameter: `FileName As String` and `SelectionSet As AcadSelectionSet`.

■ **Note** The insertion point isn't specified. If it isn't already at the origin, each object within the selection set will have to be copied and moved before a call to `WBlock`.

`AcadDocument` Object Properties

As well as the common `Application` property, the `AcadDocument` object supports the following properties.

Name	Returns	Description
`Active`	Boolean	Specifies whether the document is active. This property's value is read-only.
`ActiveDimStyle`	`AcadDimStyle`	Gets or sets the active dimension style. This style will be applied to all newly created dimensions.
`ActiveLayer`	`AcadLayer`	Gets or sets the active layer. New objects are placed on the active layer as they're created.
`ActiveLayout`	`AcadLayout`	Gets or sets the active layout.
`ActiveLinetype`	`AcadLinetype`	Gets or sets the active linetype. The specified linetype must already exist in the drawing.
`ActivePViewport`	`AcadPViewport`	Gets or sets the active paper space viewport. A viewport must be set active using this property before you can see any changes to it.

Continued

Name	Returns	Description
ActiveSelectionSet	AcadSelectionSet	Gets or sets the active selection set.
ActiveSpace	AcActiveSpace	Gets or sets whether the paper space or the model space is active. The value is stored in the TILEMODE system variable. For model space and floating viewports, this value is set to acPaperSpace. Even though you have the ability to edit the model, when the MSpace property is set to True, you're still technically in paper space. Values for AcActiveSpace: acModelSpace or acPaperSpace.
ActiveTextStyle	AcadTextStyle	Gets or sets the active text style. New text added to the drawing will adopt this text style. Existing text that has no distinct text style specified will adopt the new style. Note that a call to the Regen method is necessary to see any changes.
ActiveUCS	AcadUCS	Gets or sets the active UCS. Changes made to the current active UCS will become visible only after that UCS is reset as the active UCS.
ActiveViewport	AcadViewport	Gets or sets the active viewport. Changes made to the current active viewport will become visible only after you reset the viewport as the active viewport.
Blocks	AcadBlocks	Gets the drawing's blocks collection. This property's value is read-only.
Database	AcadDatabase	Gets the database to which the document belongs. This property's value is read-only.
Dictionaries	AcadDictionaries	Gets the document's dictionaries collection. This property's value is read-only.
DimStyles	AcadDimStyles	Gets the document's dimension styles collection. This property's value is read-only.
ElevationModelSpace	Double	Gets or sets the elevation setting for the model space. The current elevation is the Z value that is used whenever a 3-D point is expected but only the X and Y values are supplied.
ElevationPaperSpace	Double	Gets or sets the elevation setting for the paper space. The current elevation is the Z value that is used whenever a 3-D point is expected but only the X and Y values are supplied.
FullName	String	Gets the document's name, including the path. It returns a null string for a new drawing. This property's value is read-only.
Groups	AcadGroups	Gets the document's groups collection. This property's value is read-only.
HWND	Long	Gets the window handle of the document's window frame. This property's value is read-only.
Layers	AcadLayers	Gets the document's layers collection.
Layouts	AcadLayouts	Gets the document's layouts collection.
Limits	Variant	Gets or sets the document's drawing limits as an array of four Double numbers: the X and Y coordinates of the lower-left limit, and the second pair of values define the X and Y coordinates of the upper-right limit. You can't specify limits for the Z-axis.
Linetypes	AcadLinetypes	Gets the document's linetypes collection. This property's value is read-only.

Name	Returns	Description
`ModelSpace`	`AcadModelSpace`	Gets the document's model space object. This property's value is read-only.
`MSpace`	Boolean	Specifies whether it is possible to edit the model from the floating paper space viewports.
`Name`	String	Gets the document's name but not the path. It returns a null string for a new drawing. This property's value is read-only.
`ObjectSnapMode`	Boolean	Specifies whether the document's snap mode is on or off. `True` indicates that it is on.
`PaperSpace`	`AcadPaperSpace`	Gets the document's paper space object. This property's value is read-only.
`Path`	String	Gets the document's path but not the file name. This property's value is read-only.
`PickfirstSelectionSet`	`AcadSelectionSet`	Gets the `PickFirst` selection set. This property's value is read-only.
`Plot`	`AcadPlot`	Gets the document's `AcadPlot` object. This property's value is read-only.
`PlotConfigurations`	`AcPlotConfigurations`	Gets the document's plot configurations collection. This property's value is read-only.
`Preferences`	`AcadPreferences`	Gets the document's `AcadPreferences` object. This property's value is read-only.
`ReadOnly`	Boolean	Specifies whether the document is read-only. This property's value is read-only.
`RegisteredApplications`	`AcadRegisterApplications`	Gets the document's registered applications collection. This property's value is read-only.
`Saved`	Boolean	Specifies whether the document has any unsaved changes. It's `True` if there are no unsaved changes and `False` if there are. This property's value is read-only.
`SelectionSets`	`AcadSelectionSets`	Gets the document's selection set collection. This property's value is read-only.
`TextStyles`	`AcadTextStyles`	Gets the document's text styles collection. This property's value is read-only.
`UserCoordinateSystems`	`AcadUCSs`	Gets the document's UCSs collection. This property's value is read-only.
`Utility`	`AcadUtility`	Gets the document's `AcadUtility` object. This property's value is read-only.
`Viewports`	`AcadViewports`	Gets the document's viewports collection. This property's value is read-only.
`Views`	`AcadViews`	Gets the document's views collection. This property's value is read-only.
`WindowState`	`AcWindowState`	Gets or sets the state of the document window. Values: `acMin`, `acMax`, and `acNorm`.
`WindowTitle`	String	Gets the title of the document window, which will be the same as for the `Name` property for a saved document. This property's value is read-only.

AcadDocument Object Events

The following events are triggered during various stages in the lifetime of the AcadDocument object. Note that no events will be fired while a modal dialog box is displayed.

Event	Description
Activate	Occurs when the document window is activated.
BeginClose	Occurs after a request to close a drawing is received.
BeginCommand	Occurs after a command is issued but before it's initiated. Parameter: Command As String.
BeginDoubleClick	Occurs after the user double-clicks anywhere in the drawing. Parameter: PickPoint As Variant (array of WCS coordinates).
BeginLISP	Occurs after a request to evaluate a LISP expression is received. Parameter: FirstLine As String. FirstLine won't have any case conversion of the alpha characters.
BeginPlot	Occurs after a request to print a drawing is received. Parameter: DrawingName As String.
BeginRightClick	Occurs after the user right-clicks anywhere in the drawing window. Parameter: PickPoint As Variant.
BeginSave	Occurs after a request to save a drawing is received. Parameter: FileName As String.
BeginShortcutMenuCommand	Occurs after the user right-clicks in the drawing window but before the menu is in command mode. Parameters: ShortcutMenu As AcadPopupMenu and Command As String.
BeginShortcutMenuDefault	Occurs after the user right-clicks in the drawing window but before the menu is in default mode. Parameter: ShortcutMenu As AcadPopupMenu.
BeginShortcutMenuEdit	Occurs after the user right-clicks in the drawing window but before the menu is in edit mode. Parameters: ShortcutMenu As AcadPopupMenu and SelectionSet As AcadSelectionSet.
BeginShortcutMenuGrip	Occurs after the user right-clicks in the drawing window but before the menu is in grip mode. Parameter: ShortcutMenu As AcadPopupMenu.
BeginShortcutMenuOSnap	Occurs after the user right-clicks in the drawing window but before the menu is in osnap mode. Parameter: ShortcutMenu As AcadPopupMenu.
Deactivate	Occurs when the drawing window is deactivated.
EndCommand	Occurs after a command is executed. Parameter: CommandName As String.
EndLISP	Occurs after a LISP expression is evaluated.
EndPlot	Occurs after a document is sent to the printer. Parameter: DrawingName As String.
EndSave	Occurs after a drawing has been saved. Parameter: FileName As String.
EndShortcutMenu	Occurs after a shortcut menu appears. Use this event to perform any cleanup work on the shortcut menu. Parameter: ShortcutMenu As AcadPopupMenu.
LayoutSwitched	Occurs after the user changes the layout. Parameter: LayoutName As String.
LISPCancelled	Occurs when the evaluation of a LISP expression is canceled.

Event	Description
ObjectAdded	Occurs after an object is added to the drawing. Parameter: Entity As AcadEntity.
ObjectErased	Occurs after an object is erased from the drawing. Parameter: ObjectID As Long.
ObjectModified	Occurs after an object in the drawing is modified. Parameter: Entity As AcadEntity.
SelectionChanged	Occurs when the current PickFirst selection set changes.
WindowChanged	Occurs when the drawing or document window is changed. Parameter: WindowState As acMin or acMax or acNorm. This event (and the one that follows) is helpful when implementing floating palettes, toolbars, or modeless dialog boxes that track with the application or document window. The VB or ObjectARX application can use the HWNDFrame parameter to get the coordinates of the window, convert those coordinates to either screen or parent coordinates, and use this information to position other windows.
WindowMovedOrResized	Occurs after the drawing or document window is moved or resized. Parameters: HWNDFrame As Long and Moved As Boolean. If Moved is set to True, the window was moved; otherwise, it was resized.

AcadDocuments Collection

The AcadDocuments collection contains all AutoCAD drawings open in the current session. To add a new member to the collection, use the Add or Open method. To select a specific document, use the Item method. To close all the documents, use the Close method similar to the one found in the AcadDocument object. To close all documents, use the Close method found in this collection. There's no limit to the number of documents you can create in your drawing. However, there can be only one instance of the AcadDocuments collection, which is predefined for each application. You can make multiple references to it by using the Documents property or the AcadDocuments object.

AcadDocuments Collection Methods

The AcadDocuments collection supports these four methods.

Name	Returns	Description
Add	AcadDocument	Creates an AcadDocument object and adds it to the collection. Optional parameter: Name As String. This method should not be called when AutoCAD runs in SDI mode.
Close		Closes all open drawings in MDI mode. Note that you can't call this method while in SDI mode. Use the Close method of the AcadDocument object.
Item	AcadDocument	Returns the object at the given index in the collection. Parameter: Index As Variant (Integer or String). If the Index value is a String, it must match an existing document in the collection.
Open		Opens an existing .dwg (drawing) file in MDI mode. The document then becomes the active document. When the user is accessing a secure URL, a dialog box will be posted that prompts the user for the necessary password information. Message boxes may also appear if the user hasn't suppressed this activity in the browser. Parameters: Name As String and [ReadOnly As Boolean]. The second parameter specifies whether the drawing is read-only.

AcadDocuments Collection Properties

The AcadDocuments collection supports the common Application property and the Count property but no other properties.

AcadDynamicBlockReferenceProperty Object

This is an object containing the properties of a dynamic block. Dynamic blocks cannot be created through VBA; however, properties of dynamic blocks may be queried.

AcadDynamicBlockReferenceProperty Methods

The AcadDynamicBlockReferenceProperty object does not support any methods.

AcadDynamicBlockReferenceProperty Properties

The AcadDynamicBlockReferenceProperty object inherits no properties. However, it does support the following properties.

Name	Returns	Description
AllowedValuesVariant		Gets the allowed values for the property. Not all dynamic block properties have a restricted set of property values. Some properties are unrestricted, some specify a minimum or maximum value (or both), and some have a list of allowed values. This property supports only allowed property values that are defined by a list; minimum, maximum, and unrestricted property types are not supported. Read-only.
Description	String	Gets the description of an object. The description for a dynamic block reference property is set by the block author and cannot be edited.
PropertyName	String	Gets the name for the property. Property names are guaranteed to be unique among all properties on a given block reference. Read-only.
ReadOnly	Boolean	Gets if the document or property is read-only or read-write. The property is read-only when the ReadOnly property is set to True.
Show	Boolean	Gets whether the property is shown in the user interface. If this property is True, the dynamic block reference property is displayed in the Properties palette when the block reference containing it is selected. It is read-only.
UnitsType	AcDynamicBlockReference PropertyUnitsType	Gets and sets the current display units type for the property. For a list of possible values for the AcDynamicBlockReferencePropertyUnitsType enumeration, see Appendix B.
Value	Variant	Gets and sets the current value for the property.

AcadEllipse Object

The AcadEllipse object represents an elliptical arc or full ellipse, drawing a true ellipse, not a polyline approximation. It is created using the AddEllipse method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadEllipse Object Methods

The AcadEllipse object inherits all the methods of the AcadEntity and AcadObject objects. It also supports the following method.

Name	Returns	Description
Offset	Variant	Creates a new ellipse by offsetting the current ellipse by a specified distance, which must be nonzero. If the offset is positive, a larger ellipse is created with the same origin. If the offset is negative, a smaller ellipse is created. Parameter: Distance As Double.

AcadEllipse Object Properties

The AcadEllipse object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Area	Double	Gets the area of the ellipse. This property's value is read-only.
Center	Variant	Gets or sets the center of the ellipse as an array of the Doubles.
EndAngle	Double	Gets or sets the end angle of the ellipse in radians. Use 6.28 radians to specify a closed ellipse.
EndParameter	Double	Gets or sets the end parameter for an ellipse. The valid range is 0 to 2*PI.
EndPoint	Variant	Gets the 3-D coordinates of the end point of the ellipse. This property's value is read-only.
MajorAxis	Variant	Gets or sets the direction of the major (i.e., longer) axis of the ellipse as a 3-D vector. The vector originates at the ellipse center.
MajorRadius	Double	Gets or sets the length of the major (i.e., longer) axis of the ellipse.
MinorAxis	Variant	Gets the direction of the minor (i.e., shorter) axis of the ellipse as a 3-D vector. This property's value is read-only.
MinorRadius	Double	Gets or sets the length of the minor (i.e., shorter) axis of the ellipse.
Normal	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the ellipse.
RadiusRatio	Double	Gets or sets the major to minor axis ratio of an ellipse. A value of 1.0 denotes a circle.
StartAngle	Double	Gets or sets the start angle of the ellipse in radians. Use 6.28 radians to specify a closed ellipse.
StartParameter	Double	Gets or sets the start parameter for an ellipse. The valid range is 0 to 2*PI.
StartPoint	Variant	Gets the X, Y, Z coordinate of the start point of the ellipse. This property's value is read-only.

Note Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadExternalReference` Object

The `AcadExternalReference` object represents an instance of an external reference (Xref) inserted into a drawing (i.e., where a second drawing is linked into the current drawing). It differs from the `AcadBlockReference` object in the following respect: if a block is inserted into a drawing, its definition and all associated geometry are stored in the current drawing database and aren't updated if the original drawing changes. However, when you insert a drawing as an Xref, changes are updated in the externally referenced file. Like a block reference, an Xref is displayed in the current drawing as a single object. However, it doesn't significantly increase the file size of the current drawing and can't be exploded. The `AcadExternalReference` object is created using the `AttachExternalReference` method of the `AcadBlock`, `AcadModelSpace`, or `AcadPaperSpace` object. Note that many methods and properties of the `AcadBlock` and `AcadDatabase` objects are specific to Xrefs.

`AcadExternalReference` Object Methods

The `AcadExternalReference` object inherits all the methods of the `AcadEntity` and `AcadObject` objects. It also supports the following method.

Name	Returns	Description
`GetConstantAttributes`	Variant	Returns an array of constant attributes for the external reference.

`AcadExternalReference` Object Properties

The `AcadExternalReference` object inherits all the properties of the `AcadEntity` and `AcadObject` objects, as well as the common `Application` property. It also supports the following properties.

Name	Returns	Description
`HasAttributes`	Boolean	Specifies whether the external reference has any attributes. This property's value is read-only.
`InsertionPoint`	Variant	Gets or sets the insertion point for the external reference as a set of 3-D WCS coordinates. It returns a three-element array of Doubles.
`Name`	String	Gets or sets the name of the external reference.
`Normal`	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the external reference.
`Path`	String	Gets or sets the path of the external reference. This doesn't include the file name.
`Rotation`	Double	Specifies the rotation angle (in radians) for the block, relative to the X-axis with positive values going counterclockwise when viewed along the Z-axis toward the origin.
`XScaleFactor`	Double	Gets or sets the X scale factor for the block.
`YScaleFactor`	Double	Gets or sets the Y scale factor for the block.
`ZScaleFactor`	Double	Gets or sets the Z scale factor for the block.

Note Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadFileDependency` Object

The `AcadFileDependency` object describes an entry in the File Dependency List. The files on which a drawing file depends are in its File Dependency List, which is the `AcadFileDependencies` collection object. This object has no methods, but it supports the following properties.

Name	Returns	Description
`AffectsGraphics`	Boolean	If `True`, affects; if `False`, doesn't affect.
`Feature`	String	Feature name (e.g., "Acad:Xref").
`FileName`	String	File name of current file.
`FileSize`	Long	File size in bytes. This property's value is read-only.
`FingerprintGUID`	String	The GUID fingerprint of the file.
`FoundPath`	String	Alternate path found in (if any).
`FullFileName`	String	Saved full file name.
`IsModified`	Boolean	If `True`, modified; if `False`, not modified.
`ReferenceCount`	Long	Current item count of reference.
`TimeStamp`	Long	Current time/date in seconds since 1/1/1980.
`VersionGUID`	String	GUID version stamp.

`AcadFileDependencies` Object

The `AcadFileDependencies` object creates, updates, retrieves, and removes items in the File Dependency List. The File Dependency List is a list of the files on which the drawing file depends. You can perform operations on the File Dependency List using the `FileDependencies` collection.

`AcadFileDependencies` Object Methods

In addition to the methods inherited from the `AcadDocument` object, the following methods are available.

Name	Returns	Description
`CreateEntry`	Long	An index into the File Dependency List
`IndexOf`	Long	The index of an entry in the File Dependency List
`RemoveEntry`		Removes an entry from the File Dependency List
`UpdateEntry`		Updates an entry in the File Dependency List

AcadGroup Object

The AcadGroup object represents a named AcadSelectionSet object. It is created using the Add method of the AcadGroups collection and can be accessed by the Item method of the AcadGroups collection. See the AcadSelectionSet object for more information on selection sets.

AcadGroup Object Methods

In addition to the methods inherited from the AcadObject object, the AcadGroup object supports the following methods.

Name	Returns	Description
AppendItems		Appends one or more entities to the specified group. Parameter: Objects As Variant.
Highlight		Determines whether the objects in a given group are to be highlighted. Parameter: HighlightFlag As Boolean.
Item	AcadEntity	Gets the member object at a given index in a group. Parameter: Index As Variant (Integer or String). If the Index value is a String, it must match an existing object name in the collection.
RemoveItems		Removes specified items from the group. Note that items removed from a group remain in the drawing; they're no longer associated with the group. Parameter: Objects As Variant.
Update		Updates the object to the drawing screen.

AcadGroup Object Properties

In addition to the properties inherited from the AcadObject object, the AcadGroup object supports the following properties.

Name	Returns	Description
Color	AcColor	Gets or sets the color of the group object. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in the default color) or acByLayer (where AutoCAD draws objects in the color specified for the layer). For a list of possible values for the AcColor enumerated type, see Appendix B.
Count	Integer	Gets the number of items in the group. This property's value is read-only.
Layer	String	Gets or sets the layer for the group.
Linetype	String	Gets or sets the linetype of the group.
LinetypeScale	Double	Gets or sets the linetype scale of the group. The default value is 1.0.
Lineweight	AcLineWeight	Gets or sets the lineweight of the group. For a list of possible values for the AcLineWeight enumerated type, see Appendix B.
Name	String	Gets or sets the name of the group.
PlotStyleName	String	Gets or sets the group's plot style name.
Visible	Boolean	Specifies whether the group is visible.

Note Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadGroups` Collection

The `AcadGroups` collection contains all the groups in the drawing. Although this collection inherits a `Delete` method, you can't actually delete it. If you need to delete a specific group, use the `Delete` method found in the `AcadGroup` object. There is no limit to the number of groups you can create in your drawing. However, there can be only one instance of the `AcadGroups` collection, which is predefined for each drawing. You can make multiple references to it by using the `Groups` property.

`AcadGroups` Collection Methods

In addition to the methods inherited from the `AcadObject` object, the `AcadGroups` collection supports the following methods.

Name	Returns	Description
`Add`	`AcadGroup`	Creates a member object and adds it to the collection. Parameter: `Name As String`.
`Item`	`AcadGroup`	Gets the member object at a given index in a group. Parameter: `Index As Variant` (an Integer or a String). If the `Index` value is a String, it must match an existing group in the collection.

`AcadGroups` Collection Properties

The `AcadGroups` collection supports the `Count` property, the common `Application` property, and the properties inherited from the `AcadObject` object. It supports no other properties.

Note Because this collection inherits from `AcadObject`, it supports the `Modified` event.

`AcadHatch` Object

The `AcadHatch` object represents an area fill consisting of a pattern of lines. It is created using the `AddHatch` method of the `AcadBlock`, `AcadModelSpace`, or `AcadPaperSpace` object. After creating the hatch, what you must do next is add the outer loop using the `AppendOuterLoop` method. You have to do this for the hatch to be a valid AutoCAD object; otherwise, AutoCAD will enter an unpredictable state. Once you've successfully created the outer loop, you can add any inner loops using the `AppendInnerLoop` method or any other operation carried out on the hatch.

`AcadHatch` Object Methods

In addition to the methods inherited from the `AcadObject` object, the `AcadHatch` object supports the following methods.

Name	Description
`AppendInnerLoop`	Appends an inner loop to the hatch. You must add the outer loop before you can add any inner loop. Parameter: `ObjectArray As Variant`. The parameter is an array of objects forming a closed boundary, which can consist of one or more objects. If more than one object is used, their end points must coincide for the loop to be created properly. The loop may contain the following types of objects: `AcadLine`, `AcadPolyline`, `AcadCircle`, `AcadEllipse`, `AcadSpline`, and `AcadRegion`. It's important that the object array used in the call to `AppendInnerLoop` forms a closed boundary.
`AppendOuterLoop`	Appends an outer loop to the hatch. The outer loop must be closed and must be created before any inner loops can be added. Parameter: `ObjectArray As Variant`. For a description of the parameter, see the `AppendInnerLoop` method. It's important that the object array used in the call to `AppendOuterLoop` forms a closed boundary.
`Evaluate`	Evaluates the hatch lines or solid fill using the specified hatch pattern. For regular hatch patterns, this method performs intersection calculations between pattern definition lines and hatch boundary curves to form hatch lines. For solid fill hatch patterns, this method performs triangulation of the hatch area and fills in the triangular meshes with the given color. AutoCAD may not succeed in evaluating an `AcadHatch` object when either the inner or outer loop is too small for the hatch patterns and will return an ambiguous output error.
`GetLoopAt`	Gets the objects used to define a loop at the given index of the hatch. Parameters: `Index As Integer` and `ObjectArray As Variant`. The `Index` parameter can take any value from 0 (the index for the outer loop) to the index of the last loop created with `AppendInnerLoop`. The second parameter is an object or an array of objects that makes up the loop.
`InsertLoopAt`	Inserts a loop at a given index of a hatch. Parameters: `Index As Integer`, `LoopType As AcLoopType`, and `ObjectArray As Variant`. Values for the `AcLoopType` enumeration are given in Appendix B. For a description of the third parameter, see the `AppendInnerLoop` method.
`SetPattern`	Sets the pattern type and name of the hatch. Note that the integer value for the pattern type is supplied. Parameters: `PatternType As AcPatternType` and `PatternName As String`. See the `AcPatternType` enumeration table in Appendix B.

`AcadHatch` Object Properties

In addition to the properties inherited from the `AcadEntity` object, the `AcadObject` object, and the common `Application` property, the `AcadHatch` object supports the following properties.

Name	Returns	Description
`AssociativeHatch`	Boolean	Specifies whether the hatch is associative. An associative hatch is updated when its boundaries are modified.
`Elevation`	Double	Gets or sets the current elevation of the hatch object.
`Gradient1`	`AcCmColor`	Gets or sets the start color of the gradient. It must be one of the constants `acColorMethodByACI` or `acColorMethodByBlock`.
`Gradient2`	`AcCmColor`	Gets or sets the end color of the gradient. It must be one of the constants `acColorMethodByACI` or `acColorMethodByBlock`.

Name	Returns	Description
GradientCentered	Boolean	Gets or sets whether the gradient is centered.
GradientName	String	Gets or sets the pattern name. The pattern name for the gradient may be LINEAR, CYLINDER, INVCYLINDER, SPHERICAL, HEMISPHERICAL, CURVED, INVSPHERICAL, INVHEMISPHERICAL, or INVCURVED.
HatchObjectType	AcHatchObject	Gets or sets the hatch object type to be hatch, classic, or gradient. The default value of this property is 0, acHatchObject, which creates a classic hatch. If the value of this property is 1, acGradientObject, a gradient is created by the AddHatch method for hatch creation. If a gradient is created, then PatternType should be AcGradientPatternType and PatternName should contain a gradient pattern name of LINEAR, CYLINDER, INVCYLINDER, SPHERICAL, HEMISPHERICAL, CURVED, INVSPHERICAL, INVHEMISPHERICAL, or INVCURVED.
HatchStyle	AcHatchStyle	Gets or sets the hatch style. For a list of possible values for the AcHatchStyle enumerated type, see Appendix B.
ISOPenWidth	AcISOPenWidth	Gets or sets the ISO pen width of an ISO hatch pattern. For a list of possible values for the AcISOPenWidth enumerated type, see Appendix B. When you query a hatch and receive a value of acPenWidthUnk, use the PatternScale property to obtain the nonstandard value. Setting the pen width of any hatch to acPenWidthUnk has no effect.
Normal	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the hatch object.
NumberOfLoops	Integer	Gets the number of loops in the hatch boundary. This property's value is read-only.
PatternAngle	Double	Gets or sets the angle of the hatch pattern in radians. The valid range is 0 to 6.28. The value is stored in the HPANG system variable.
PatternDouble	Boolean	Specifies whether the user-defined hatch is double-hatched. The value of this property is stored in the HPDOUBLE system variable. If the PatternType property is set to acHatchPatternTypePreDefined or acHatchPatternTypeCustomDefined, this property isn't used.
PatternName	String	Gets or sets the pattern name of the hatch, which can be up to 34 characters long but can't have spaces (by default). To remove the default setting, enter a period (.), and a NULL string is returned. If the PatternType property is set to acHatchPatternTypePreDefined, then this property refers to the name of a hatch pattern in acad.pat. If the PatternType property is set to acHatchPatternTypeCustomDefined, then this property refers to the name of a hatch pattern in a custom .pat file. If the PatternType property is set to acHatchPatternTypeUserDefined, then this property isn't used. The value of this property is stored in the HPNAME system variable. This property's value is read-only.

Continued

Name	Returns	Description
PatternScale	Double	Gets or sets the hatch pattern scale. The value of this property must not be zero, and it is stored in the HPSCALE system variable. If the PatternType property is set to acHatchPatternTypeUserDefined, then this property isn't used.
PatternSpace	Double	Gets or sets the user-defined hatch pattern spacing. This value is also controlled by the HPSPACE system variable. If the PatternType property is set to acHatchPatternTypePreDefined or acHatchPatternTypeCustomDefined, then this property isn't used.
PatternType	AcPatternType	Gets the pattern type of the hatch. For a list of possible values for the AcPatternType enumerated type, see Appendix B. This property's value is read-only.

■ **Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadHyperlink Object

The AcadHyperlink object represents a URL and URL description, which is stored within the XData of the object's corresponding entity and not in the drawing itself. This means a hyperlink object is created every time a URL is requested, and the name and description is read from the XData. In this manner, it's similar to the AcadUtility object. You should take care to not create multiple hyperlink objects referencing the same URL. If one such object is updated, the others won't be.

AcadHyperlink Object Method

The AcadHyperlink object doesn't inherit from AcadEntity or AcadObject. It supports only a single method.

Name	Description
Delete	Deletes a specified hyperlink object

AcadHyperlink Object Properties

In addition to the common Application property, the AcadHyperlink object supports the following properties.

Name	Returns	Description
URL	String	Gets or sets the URL for the hyperlink object. The file pointed to by the URL can be stored locally on a network drive or via an Internet connection.
URLDescription	String	Gets or sets the URL description for the hyperlink object. This is useful if the URL name itself isn't very helpful in identifying the contents of the file.
URLNamedLocation	String	Gets or sets the named location for the hyperlink object. This can be another AutoCAD drawing or a completely different application, such as a word processing program. If you specify a named view to jump to in an AutoCAD drawing, AutoCAD restores that view when the hyperlink is opened.

AcadHyperlinks Collection

The AcadHyperlinks collection contains all the hyperlinks for a given entity. To add a new member to the collection, use the Add method. To select a specific hyperlink, use the Item method. It is accessed via the Hyperlinks property on all AcadEntity objects.

AcadHyperlinks Collection Methods

The AcadHyperlinks collection doesn't inherit from AcadObject. It supports the following methods.

Name	Returns	Description
Add	AcadHyperlink	Creates a named AcadHyperlink object and adds it to the collection. Parameters: Name As String, [Description As Variant], and [NamedLocation As Variant].
Item	AcadHyperlink	Gets the member object at a given index in a collection, group, or selection set. Parameter: Index As Integer (An Integer of a String). If the value for Index is a String, it must match an existing object name in the collection.

AcadHyperlinks Collection Properties

The AcadHyperlinks collection supports the common Application and Count properties. It supports no other properties.

AcadIDPair Object

The AcadIDPair object is a transient object used as an optional parameter in the CopyObjects method of the AcadDatabase and AcadDocument objects. It contains the object IDs of both the source and destination objects. See the AcadDatabase object for more details. The CopyObjects method populates the AcadIDPair object with the object IDs of the source objects and the newly copied objects, thus creating a map between all the objects copied and created. This object also contains information about the copy and translation process.

Note that this object is used for advanced cloning functions. General users should avoid using this optional functionality.

The AcadIDPair object has no methods.

AcadIDPair Object Properties

The AcadIDPair object supports the common Application property and the following properties.

Name	Returns	Description
IsCloned	Boolean	Determines whether the source object in a CopyObjects operation has been cloned. This property's value is read-only.
IsOwnerXlated	Boolean	Determines whether the owning object in a CopyObjects operation has been translated. This property's value is read-only.
IsPrimary	Boolean	Determines whether the source object in a CopyObjects operation was part of the primary set of objects being copied or if it was simply owned by a member in the primary set. This property's value is read-only.
Key	Integer	The object ID of the source object in the CopyObjects operation. This property's value is read-only.
Value	Integer	The object ID of the newly created cloned object in the CopyObjects operation. This property's value is read-only.

AcadLayer Object

The AcadLayer object represents a logical grouping of data, similar to transparent acetate overlays on a drawing. It is created using the Add method of the AcadLayers collection and can be accessed by the Item method of the AcadLayers collection. All new objects are added to the active layer. To make a layer active, use the ActiveLayer property of the Document object.

AcadLayer Object Methods

The AcadLayer object supports the methods inherited from the AcadObject object and no other methods. Note that layer 0 can't be deleted and layer DEFPOINTS shouldn't. This restriction applies also to layers referenced in a block definition.

AcadLayer Object Properties

In addition to the properties inherited from the AcadObject object, and the common Application property, the AcadLayer object supports the following properties.

Name	Returns	Description
Color	AcColor	Gets or sets the color of the layer object, which will determine the acByLayer color of the entities on the specific layer. Use a color index number from 0 to 256 or the AcColor enumeration (see Appendix B).
Freeze	Boolean	Specifies the freeze status of a layer: True if frozen, False if thawed. Freezing layers makes them invisible and excludes them from regeneration and plotting. Thawing a layer enables these capabilities. You can't freeze the active layer or make a frozen layer active.
LayerOn	Boolean	Specifies whether a layer is on or off. Layers that are "off" aren't displayed or plotted, though they're regenerated when the drawing is regenerated.
Linetype	String	Gets or sets the linetype for the layer. The default linetype for any new entity will become the linetype of the layer (ByLayer).

Name	Returns	Description
`Lineweight`	`AcLineWeight`	Gets or sets the lineweight for the layer. For a list of possible values for the `AcLineWeight` enumerated type, see Appendix B.
`Lock`	Boolean	Specifies whether a layer is locked. You can't edit objects on a locked layer, though they're still visible if the layer is on and thawed. You can activate locked layers and add objects to them, apply snap modes, freeze and turn them off, and change their associated colors.
`Name`	String	Gets or sets the name of the layer. Layer `0` can't be renamed. Layer `DEFPOINTS` is used for dimensioning shouldn't be renamed either.
`PlotStyleName`	String	Gets or sets the layer's plot style name.
`Plottable`	Boolean	Specifies whether the layer is plottable.
`ViewportDefault`	Boolean	Specifies whether the layer is to be frozen in new viewports.

Note Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadLayers` Collection

The `AcadLayers` collection contains all the layers in the drawing. Although this collection inherits a `Delete` method, you can't actually delete this collection. If you need to delete a specific layer, use the `Delete` method found in the `AcadLayer` object. There is no limit to the number of layers you can create in your drawing. However, there can be only one instance of the `AcadLayers` collection, which is predefined for each drawing. You can make multiple references to it by using the `Layers` property.

`AcadLayers` Collection Methods

In addition to the methods inherited from the `AcadObject` object, the `AcadLayers` collection supports the following methods.

Name	Returns	Description
`Add`	`AcadLayer`	Creates a member object and adds it to the collection. Parameter: `Name As String`.
`Item`	`AcadLayer`	Gets the member object at a given index in a collection, group, or selection set. Parameter: `Index As Variant` (an Integer or a String). If the `Index` value is a String, it must match an existing object name in the collection.

`AcadLayers` Collection Properties

The `AcadLayers` collection supports the `Count` property, the common `Application` property, and the properties inherited from the `AcadObject` object. It supports no other properties.

Note Because this object inherits from `AcadObject`, it supports the `Modified` event.

AcadLayout and AcadPlotConfiguration Objects

The AcadLayout object contains the plot settings and visual properties of a model space or paper space block, as displayed in the AutoCAD user interface. Each AcadLayout object is associated with a single AcadBlock object, which can be accessed through the Block property. Alternatively, the AcadLayout object can be accessed through the Item property of the AcadLayouts collection, the Layout property of the AcadBlock (when IsLayout is set to True), the AcadModelSpace and AcadPaperSpace objects, or the ActiveLayout property of the Document object.

The AcadPlotConfiguration object is similar to the AcadLayout object in that it contains the same plot settings and visual properties. The difference between the two is as follows: whereas a layout is defined only for a particular block, a plot configuration isn't specific to any one block; it's a named collection of plot settings that can be applied to any geometry.

As the two objects are all but identical, I will present their methods and properties together.

AcadLayout and AcadPlotConfiguration Objects: Methods

The AcadLayout and AcadPlotConfiguration objects support all the methods inherited from the AcadObject object. They also support the following methods.

Name	Returns	Description
CopyFrom		Copies the settings from the given plot configuration. Parameter: PlotConfig As AcadPlotConfiguration. This method is specific to the AcadPlotConfiguration object.
GetCanonicalMediaNames	Variant	Returns all available canonical media names for the specified plot device. You should call RefreshPlotDeviceInfo before calling this method. This method returns the names as an array of strings.
GetCustomScale		Gets the custom scale for a layout or plot configuration, in millimeters or inches, depending on the value of the PaperUnits property. Parameters: Numerator As Double and Denominator As Double.
GetLocaleMediaName	String	Given a canonical media name, this method returns the localized version of the canonical media name. Parameter: Name As String.
GetPaperMargins		Gets the margins offsets for the layout or plot configuration, the units of which depend on the values of the PaperUnits property. The print origin is offset from the edge of the paper by the values returned by this method. Parameters: LowerLeft As Variant and UpperRight As Variant. (Both parameters are two-element arrays of Doubles.)
GetPaperSize		Gets the width and height of the configured paper using the values specified by the PaperUnits property. Parameters: Width As Double and Height As Double.
GetPlotDeviceNames	Variant	Returns all available plot device names. You should call RefreshPlotDeviceInfo before calling this method. This method returns the names as an array of strings.
GetPlotStyleTableNames	Variant	Returns all available plot style table names. You should call RefreshPlotDeviceInfo before calling this method. This method returns the names as an array of strings.

Name	Returns	Description
GetWindowToPlot		Gets the coordinates that define the portion of the layout to plot, the units of which depend on the values of the PaperUnits property. The coordinates of the window are taken from the origin. For these coordinates to be used for the plot, the PlotType property must be set to acWindows. Parameters: LowerLeft As Variant and UpperRight As Variant. (Both parameters are two-element arrays of Doubles.)
RefreshPlotDeviceInfo		Updates the plot, canonical media, and plot-style table information to reflect the current system state. You should call this method once before calling the GetCanonicalMediaNames, GetPlotDeviceNames, or GetPlotStyleTableNames method.
SetCustomScale		Sets the custom scale for a layout or plot configuration, in millimeters or inches depending on the value of the PaperUnits property. Parameters: Numerator As Double and Denominator As Double. Note that the parameters must be greater than zero.
SetWindowToPlot		Sets the coordinates that define the portion of the layout to plot, the units of which depend on the values of the PaperUnits property. The coordinates of the window are taken from the origin. For these coordinates that can be used for the plot, the PlotType property must be set to acWindows. Parameters: LowerLeft As Variant and UpperRight As Variant.

AcadLayout and AcadPlotConfiguration Objects: Properties

The AcadLayout and AcadPlotConfiguration objects inherit all the properties inherited from the AcadObject object, as well as the common Application property. They also support the following properties.

Name	Returns	Description
Block	AcadBlock	This property is specific to the AcadLayout object. It gets the block associated with the layout. This property's value is read-only.
CanonicalMediaName	String	Gets or sets the name of the media that specifies the paper size. Note that changes to this property won't be visible until the drawing is regenerated.
CenterPlot	Boolean	Specifies whether there is centering of the plot on the media. Note that changes to this property won't be visible until the drawing is regenerated. Note also that this property can't be set to True if the PlotType property is set to acLayout.
ConfigName	String	Gets or sets the name of the PC3 file or print device to be used by the layout or plot configuration. Note that this property takes only the file name for the configuration file. Use the PrinterConfigPath property to specify the path for printer configuration files.

Continued

Name	Returns	Description
ModelType	Boolean	Specifies whether the plot configuration applies only to model space or to all layouts. If True, it applies only to the model space layout; if False, it applies to all layouts. This property's value is read-only.
Name	String	Gets or sets the name of the object.
PaperUnits	AcPlotPaperUnits	Gets or sets the units for the display of the layout or plot configuration in the user interface. Note that changes to this property won't be visible until the drawing is regenerated. Possible values for the AcPlotPaperUnits enumeration are acInches, acMillimeters, and acPixels.
PlotHidden	Boolean	Specifies whether objects in the paper space are to be hidden during a plot. This is True if objects are to be hidden and False otherwise. Note that this property does *not* affect objects inside floating model space viewports.
PlotOrigin	Variant	Gets or sets the origin of the UCS, block, layout, or raster image in WCS coordinates. It returns a two-element array of Doubles representing the XY coordinates of the origin. The origin is offset from the media edge by the paper margin and is given in millimeters. Note that changes to this property won't be visible until the drawing is regenerated.
PlotRotation	AcPlotRotation	Gets or sets the rotation angle for the layout or plot configuration relative to the X-axis, with positive values going counterclockwise when viewed along the Z-axis toward the origin. Note that changes to this property won't be visible until the drawing is regenerated. Possible values for the AcPlotRotation enumeration are ac0degrees, ac90degrees, ac180degrees, and ac270degrees.
PlotType	AcPlotType	Gets or sets the type of layout or plot configuration. Note that changes to this property won't be visible until the drawing is regenerated. Note also that the ViewToPlot property or SetWindowToPlot method needs to be called before you can set the PlotType to acView or acWindow, respectively. For a list of possible values for the AcPlotType enumerated type, see Appendix B.
PlotViewportBorders	Boolean	Specifies whether the viewport borders are to be plotted.
PlotViewportsFirst	Boolean	Specifies whether all geometry in paper space viewports is plotted first rather than the geometry in the paper space. This is True if geometry in the paper space viewports is plotted first and False otherwise.
PlotWithLineweights	Boolean	Specifies whether objects are plotted with the lineweights assigned from a plot style or from a drawing file. This is True if the plot uses the lineweights assigned from a plot style and False otherwise.
PlotWithPlotStyles	Boolean	Specifies whether objects are plotted with the plot configuration assigned from a plot style or from a drawing file. This is True if the plot uses the plot configuration assigned from a plot style and False otherwise.

Name	Returns	Description
`ScaleLineweights`	Boolean	Specifies whether the lineweight is scaled with the rest of the geometry when a layout is printed. The value of this property is stored in the `LWSCALE` system variable. Note that this property is disabled for the model space layout.
`ShowPlotStyles`	Boolean	Specifies whether plot styles are to be used in the plot (`True`) or use the plot styles assigned to objects in the drawing (`False`).
`StandardScale`	`AcPlotScale`	Gets or sets the standard scale for the layout, viewport, or plot configuration. For a list of possible values for the `AcPlotScale` enumerated type, see Appendix B.
`StyleSheet`	String	Gets or sets the name of the style sheet for the layout or plot configuration.
`TabOrder`	Integer	Gets or sets the tab order of a layout. Controls the order in which the layouts are displayed in the tab control. The tab order should be unique and sequential among all layouts in the database. The model space tab must have a tab order of 0. Paper space tabs must have a tab order of 1 or greater. This applies to the `AcadLayout` object only.
`UseStandardScale`	Boolean	Specifies whether the plot is to use a standard or custom scale. This is `True` if the standard scale is to be used and `False` otherwise. See the `SetCustomScale` method.
`ViewToPlot`	String	Gets or sets the name of the view to be plotted. Note that the `PlotType` property for the layout or plot configuration has to be set to `acView`.

■ **Note** Because these objects inherit from `AcadObject`, they both support the `Modified` event.

`AcadLayouts` and `AcadPlotConfigurations` Collections

The `AcadLayouts` collection contains all the `AcadLayout` objects in the drawing. It is accessed by the `Layouts` property of the `Document` object. It is created when the document is created and can't be deleted (even though the collection inherits a `Delete` method from `AcadObject`). To remove a layout definition, use the `Delete` method of the `AcadLayout` object.

The `AcadPlotConfigurations` collection is similar to the `AcadLayouts` collection, except it contains all the `AcadPlotConfiguration` objects for a project. Whereas a layout is specific to a particular block, a plot configuration contains a named collection of plot settings that can be applied to any geometry. Once again, this collection inherits a `Delete` method that you can't use. To remove a plot configuration, use the `Delete` method of the `AcadPlotConfiguration` object.

As the two collections are all but identical, I will present their methods and properties together.

AcadLayouts and AcadPlotConfigurations Collections: Methods

The AcadLayouts and AcadPlotConfigurations collections support all the methods inherited from the AcadObject object. They also support the following methods.

Name	Returns	Description
Add		Creates a named member object and adds it to the appropriate collection. Parameter: Name As String. Additional optional parameter (for the AcadPlotConfigurations collection only): ModelType As Variant (this is True if the plot configuration applies to a model space tab and False otherwise).
Item		Gets the member object at a given index in a collection. Parameter: Index As Variant (an Integer or a String). If the Index value is a String, it must match an existing object name in the collection.

AcadLayouts and AcadPlotConfigurations Collections: Properties

The AcadLayouts and AcadPlotConfigurations collections support all the properties inherited from the AcadObject object, the Count property, and the common Application property. They support no other properties.

■ **Note** Because these collections inherit from AcadObject, they both support the Modified event.

AcadLeader Object

A leader line is an object that can be composed of an arrowhead attached to either splines or straight-line segments. In addition, a short horizontal line (hook line) connects descriptive text and/or feature control frames to the leader line. The AcadLeader object specifies such a complex leader line, and the Leader property of the AcadDimDiametric and AcadDimRadial objects specifies a simple leader line.

The AcadLeader object is created using the AddLeader method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object. Any annotation attached to the leader line is defined in the Annotation property of the AcadLeader object.

AcadLeader Object Methods

The AcadLeader object inherits all the methods of the AcadEntity and AcadObject objects. It also supports the following method.

Name	Description
Evaluate	Evaluates the relative position of the leader line to its associated annotation and updates the leader line's geometry if necessary.

AcadLeader Object Properties

The AcadLeader object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Annotation	AcadEntity	Gets or sets the annotation object for a leader, which can be a tolerance object, a multiline text object, or a block reference. Note that setting the Annotation property to a new annotation object modifies the drawing but returns an invalid input error.
ArrowheadBlock	String	Gets or sets the name of the block to use as the custom arrowhead for a leader line.
ArrowheadSize	Double	Gets or sets the size of the leader line arrowheads and hook lines. The initial value for this property is 0.1800.
ArrowheadType	AcDimArrowheadType	Gets or sets the type of arrowhead used for the leader line. The initial value for this property is acArrowDefault. Note that if you use the ArrowheadBlock property to specify a block to use as a custom arrowhead, this property will be set to acArrowUserDefined. For a list of possible values for the AcDimArrowheadType enumerated type, see Appendix B.
Coordinate	Variant	Gets or sets the coordinate of a single vertex in the object. This will replace the existing vertex for the specified object. Use standard array-handling techniques to process the values contained in this property. It returns a three-element array of Doubles containing 3-D coordinates in WCS. Note that the Z coordinate will default to 0 on the active UCS. Parameter: Index As Integer (the index in the zero-based array for the vertex you want to set or query).
Coordinates	Variant	Gets or sets the coordinates for each vertex in the object. This will replace any existing coordinates for the specified object. Use standard array-handling techniques to process the coordinates contained in this property. Note that you can't change the number of coordinates in the object by using this property; you can change only the location of existing coordinates. It returns a three-element array of Doubles containing 3-D coordinates in WCS. Note also that the Z coordinate will always default to 0 on the active UCS.
DimensionLineColor	AcColor	Gets or sets the color of the leader object. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in the default color), acByLayer (where AutoCAD draws objects in the color specified for the layer), or the AcColor enumeration (see Appendix B). This property overrides the value of the DIMCLRD system variable for the given leader object.
DimensionLineWeight	AcLineWeight	Gets or sets the lineweight for the leader lines. See Appendix B for a list of the values of the AcLineWeight enumeration.
Normal	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the leader line.

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Name	Returns	Description
ScaleFactor	Double	Gets or sets the scale factor for the object. It has to be a value greater than 0.0. A scale factor greater than 1 enlarges the hook line. A scale factor from 0 to 1 shrinks the hook line. The initial value for this property is 1.0000. This property overrides the DIMSCALE system variable.
StyleName	String	Gets or sets the name of the style used with the object, the default being the current style. Use the TextStyle object to change the attributes of a given text style. Note that the name given must already be defined in the drawing.
TextGap	Double	Gets or sets the gap between the annotation and a hook line on a leader object.
Type	AcLeaderType	Gets or sets type of a leader line. For a list of possible values for the AcLeaderType enumerated type, see Appendix B.
VerticalTextPosition	AcDimVerticalJustification	Gets or sets the vertical position of text in relation to the leader line. This property overrides the value of the DIMTAD system variable for the given dimension, leader, or tolerance object. The initial value for this property is acVertCentered. (For a list of the values of the AcDimVerticalJustification enumerated type, see Appendix B.)

■ **Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadLine Object

The AcadLine object represents a single line segment. In AutoCAD, lines can be one segment or a series of connected segments, but each is a separate AcadLine object. You use this object if you want to edit individual segments. If, however, you need to draw a series of line segments as a single object, use the AcadLWPolyline object. The AcadLine object is created using the AddLine method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadLine Object Methods

The AcadLine object inherits all the methods of the AcadEntity and AcadObject objects. It also supports the following method.

Name	Returns	Description
Offset	Variant	Creates a new line by offsetting the current line by a specified distance, which can be positive or negative but can't be zero. If the offset is negative, this means that the line is drawn closer to the WCS origin. Parameter: Distance As Double.

AcadLine Object Properties

The AcadLine object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Angle	Double	Gets the angle of the line in degrees, starting at the X-axis and proceeding counterclockwise. This property's value is read-only.
Delta	Variant	Gets the delta of the line. It returns a three-element array of Doubles representing the deltaX, deltaY, and deltaZ values for the line. This property's value is read-only.
EndPoint	Variant	Gets or sets a set of 3-D coordinates representing the end point of the line.
Length	Double	Gets the length of the line. This property's value is read-only.
Normal	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the line object.
StartPoint	Variant	Gets or sets the set of 3-D WCS coordinates representing the start point for the line.
Thickness	Double	Gets or sets the distance the AcadLine object is extruded above or below its elevation, in a Z-axis direction. The default is 0.0.

Note Because this object inherits from AcadObject, it supports the Modified event.

AcadLineType Object

The AcadLineType object contains line definitions or components consisting of combinations of dashes, dots, spaces, text, and shapes. You can create default linetypes programmatically only using the Add method of the AcadLinetypes collection. However, you can load an existing linetype definition to your drawing by using the Load method of the AcadLinetypes collection. Although there is no limit to the number of linetypes defined in any one drawing, only one can be active at a time. New objects are created using the linetype specified for the active layer. If no such linetype exists for the active layer, new objects are created with the active linetype. Linetypes are activated using the ActiveLinetype property.

AcadLineType Object Methods

The AcadLineType object inherits all the methods of the AcadObject object. It supports no other methods.

AcadLineType Object Properties

The AcadLineType object inherits all the properties of the AcadObject object, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
`Description`	String	Gets or sets a text description of the linetype object. This can be up to 47 characters long and can contain text or a series of characters to represent the linetype pattern itself (using underscores, dots, dashes, and spaces).
`Name`	String	Gets or sets the name of the linetype definition.

■**Note** Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadLineTypes` Collection

The `AcadLinetypes` collection contains all the linetypes defined for a drawing. You use the `Add` method to add a new linetype definition to the collection, and you can select a particular linetype using the `Item` method. You can load an existing linetype definition using the `Load` method. The collection also inherits a `Delete` method from `AcadObject`, but you can't use it, as the collection can't be deleted. To remove a linetype, use the `Delete` method of the `AcadLinetype` object. There is no limit to the number of linetypes you can create in your drawing. However, there can be only one instance of the `AcadLinetypes` collection, which is predefined for each drawing. You can make multiple references to it by using the `Linetypes` property.

`AcadLineTypes` Collection Methods

The `AcadLineTypes` collection inherits all the methods of the `AcadObject` object. It also supports the following methods.

Name	Returns	Description
`Add`	`AcadLineType`	Creates a named `AcadLinetype` object and adds it to the collection. Parameter: `Name As String`.
`Item`	`AcadLineType`	Gets the `AcadLinetype` object at a given index in the collection. Parameter: `Index As Variant` (an Integer or a String). If the `Index` value is a String, it must match an existing linetype definition.
`Load`		Loads a definition of a named linetype from a library (`.lin`) file. Parameters: `Name As String` and `FileName As String`.

`AcadLineTypes` Collection Properties

The `AcadLineTypes` collection inherits all the properties of the `AcadObject` object and supports the `Count` property and the common `Application` property. It supports no other properties.

■**Note** Because this collection inherits from `AcadObject`, it supports the `Modified` event.

AcadLWPolyline Object

The AcadLWPolyline object represents a 2-D line of adjustable width that's composed of line and arc segments. It's created using the AddLWPolyline method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadLWPolyline Object Methods

The AcadLWPolyline object inherits all the methods of the AcadEntity and AcadObject objects. It also supports the following methods.

Name	Returns	Description
AddVertex		Adds a vertex to the lightweight polyline, a vertex being the end point for a new line segment. See the SetBulge method for creating an arc. Each vertex is added to a three-element array of Doubles. Parameters: Index As Integer and Vertex As Variant. (Index refers to the position in the array where the new vertex is added. Vertex refers to the 3-D OCS coordinates where the vertex is to be created.)
Explode	Variant	Explodes the lightweight polyline and returns the subentities as an array of Objects (AcadLine and AcadArc).
GetBulge	Double	Returns the bulge value at a given index of the lightweight polyline. A bulge is the tangent of 1/4 of the included angle for the arc between the selected vertex and the next vertex in the polyline's vertex list. A negative bulge value indicates that the arc goes clockwise from the selected vertex to the next vertex. A bulge of 0 indicates a straight segment, and a bulge of 1 is a semicircle. Parameter: Index As Integer.
GetWidth		Returns segment width at a given index of the lightweight polyline. Parameters: Index As Integer, StartWidth As Double, and EndWidth As Double.
Offset	Variant	Creates a new lightweight polyline by offsetting the existing polyline by a given distance, which must be nonzero. If the offset is negative, this means that the line is drawn closer to the WCS origin. Parameter: Distance As Double.
SetBulge		Sets the vertex bulge of the lightweight polyline at a given index. See GetBulge for the definition of a bulge. Parameters: Index As Integer and bulge As Double.
SetWidth		Sets the segment width of the lightweight polyline for a given index. Parameters: Index As Integer, StartWidth As Double, and EndWidth As Double.

AcadLWPolyline Object Properties

The AcadLWPolyline object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
`Area`	Double	Gets the area of a lightweight polyline in square drawing units. The area is computed as though a straight line connects the start and end points. This property's value is read-only.
`Closed`	Boolean	Specifies whether the lightweight polyline is closed (`True`) or open (`False`). Open is the default.
`ConstantWidth`	Double	Gets or sets a global width for all segments in a lightweight polyline.
`Coordinate`	Variant	Gets or sets the coordinate of a single vertex in the object. This will replace any existing vertices for the specified object. Use standard array-handling techniques to process the values contained in this property. This returns a two-element array of Doubles containing 2-D coordinates in OCS. These coordinates can be converted to and from other coordinate systems using the `TranslateCoordinates` method. Parameter: `Index As Integer` (the index in the zero-based array of vertices for the vertex you want to set or query).
`Coordinates`	Variant	Gets or sets the coordinates for each vertex in the object. This will replace any existing coordinates for the specified object. Use standard array-handling techniques to process the coordinates contained in this property. Note that you can't change the number of coordinates in the object by using this property; you can change only the location of existing coordinates. This returns the array point as a two-element array of Doubles containing 2-D points in OCS. These coordinates can be converted to and from other coordinate systems using the `TranslateCoordinates` method.
`Elevation`	Double	Gets or sets the current elevation of the lightweight polyline.
`LinetypeGeneration`	Boolean	Specifies the linetype generation of a lightweight polyline. If this is set to `True`, the linetype is generated in a continuous pattern through the polyline vertices; if it's set to `False`, the linetype is generated starting and ending with a dash at each polyline vertex.
`Normal`	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the lightweight polyline.
`Thickness`	Double	Gets or sets the distance the `AcadLWPolyline` object is extruded above or below its elevation, in a Z-axis direction. The default is 0.0.

■ **Note** Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadMenuBar` Collection

The `AcadMenuBar` collection contains all the `AcadPopupMenu` objects displayed on the current AutoCAD menu bar. Any menu from a currently loaded menu group can be displayed on the menu bar. To display a menu on the menu bar, use the `InsertInMenuBar` method found on the AcadPopupMenu object. To remove a menu from the current menu bar, use the RemoveFromMenuBar method also found on the AcadPopupMenu object.

AcadMenuBar Collection Methods

Just one method is supported.

Name	Returns	Description
Item	Object	Returns the object at the specified index in the collection. Parameter: Index As Variant (Integer or String). If the Index value is a String, it must match an existing object name in the collection.

AcadMenuBar Collection Properties

As well as the common Application and Count properties, the AcadMenuBar collection supports the following property.

Name	Returns	Description
Parent	AcadApplication	Returns the parent of the AcadMenuBar collection. This property's value is read-only.

AcadMenuGroup Object

The AcadMenuGroup object represents an AutoCAD menu group, which contains menus and toolbars, some or all of which may be currently displayed in AutoCAD. Each menu group contains an AcadPopupMenus collection and an AcadToolbars collection. The former collection contains all the menus within the menu group and can be accessed through the Menus property. The latter contains all the toolbars within the menu group and can be accessed through the Toolbars property. Menu groups are loaded into the AutoCAD session using the Load method, which adds the menu group to the AcadMenuGroups collection.

You can't create new menu groups. However, you can load an existing menu file containing a menu group and save it again with a new menu group name and to new menu file. The menus and toolbars thus become available for editing.

AcadMenuGroup Object Methods

Three methods are supported by the AcadMenuGroup object.

Name	Description
Save	Saves the menu group. Note that certain changes to the appearance of the AutoCAD user interface, such as changing system colors, will cause the menus and toolbars to be reloaded from the menu file. Be certain to save changes to your menu group often, using the Save method, to avoid losing your customizations. Parameter: MenuFileType As AcMenuFileType. For a list of possible values for the AcMenuFileType enumerated type, see Appendix B.
SaveAs	Saves the menu group and gives it the specified name using the full path and file name, or valid URL address, for the file. The active menu group takes on the new name. Parameters: FileName As String and MenuFileType As AcMenuFileType.
Unload	Unloads the menu group. When a menu group is unloaded from the drawing, any references to the menus and toolbars within that group become invalid. Always delete or set to NULL any references to toolbars and menus before unloading.

AcadMenuGroup Object Properties

As well as the common Application property, the AcadMenuGroup object supports the following properties.

Name	Returns	Description
MenuFileName	String	Gets the name of the file in which the menu group is located. This property's value is read-only.
Menus	AcadPopupMenus	Gets the AcadPopupMenus collection for the menu group.
Name	String	Gets the name of the menu group. The name is limited to 32 characters and can't contain spaces or punctuation marks. This property's value is read-only.
Parent	AcadMenuGroups	Gets the parent of the AcadMenuGroup object. This property's value is read-only.
Toolbars	AcadToolbars	Gets the AcadToolbars collection for the menu group.
Type	AcMenuGroupType	Gets the type of the menu group. This property's value is read-only. The possible values for the AcMenuGroupType enumeration are acBaseMenuGroup and acPartialMenuGroup.

AcadMenuGroups Collection

The AcadMenuGroups collection contains all the AcadMenuGroup objects, from which you can get a reference to all pop-up menus and toolbars loaded in the current AutoCAD session. To load a new menu group from a file into this collection, use the Load method. Note that you can't create new menu groups. However, you can load an existing menu group and save it out again with a new name and to new menu file. The menus and toolbars thus become available for editing. To unload a menu group from this collection, use the Unload method found on the AcadMenuGroup object.

AcadMenuGroups Collection Methods

Three methods are supported by the AcadMenuGroups object.

Name	Returns	Description
Item	AcadMenuGroup	Returns the object at the given index in the collection. Parameter: Index As Variant (Integer or String). If the Index value is a String, it must match an existing object name in the collection.
Load	AcadMenuGroup	Loads a menu group from a menu file. Parameters: MenuFileName As String and [BaseMenu As Variant]. The MenuFileName should have an .mnc, .mns, or .mnu extension. If the BaseMenu parameter is set to True, then the menu group will be loaded as a base menu, similar to the AutoCAD MENU command. If this parameter is omitted, the menu group will be loaded as a partial menu, similar to the AutoCAD MENULOAD command.

AcadMenuGroups Collection Properties

As well as the common Application property and the Count property, the AcadMenuGroups collection supports the following property.

Name	Returns	Description
Parent	AcadApplication	Gets the parent of the collection. This property's value is read-only.

AcadMInsertBlock Object

The AcadMInsertBlock object represents a rectangular array of block references. It is created using the AddMInsertBlock method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadMInsertBlock Object Methods

The AcadMInsertBlock object inherits all the methods of the AcadEntity and AcadObject objects. It also supports the following methods.

Name	Returns	Description
Explode	Variant	Explodes the block and returns the subentities as an array of AcadBlockReference objects.
GetAttributes	Variant	Returns an array of editable attribute references attached to the multiple block reference. After you edit one AcadAttribute object and update the AcadMInsertBlock object, changes will appear in each nested block because there's only one set of attribute references for the multiple inserted blocks.
GetConstantAttributes	Variant	Returns an array of constant attributes for the multiple blocks.

AcadMInsertBlock Object Properties

The AcadMInsertBlock object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Columns	Integer	Gets or sets the number of columns in a block array.
ColumnSpacing	Double	Gets or sets the spacing of the columns in a block array.
HasAttributes	Boolean	Indicates whether the block has any attributes in it. This property's value is read-only.
InsertionPoint	Variant	Represents an insertion point in the block as a set of 3-D WCS coordinates. It returns a three-element array of Doubles.
Name	String	Gets or sets the name of the multiple block, which is identical to the name of the block used in the AddMInsertBlock method.
Normal	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the block.

Continued

Name	Returns	Description
`Rotation`	Double	Gets or sets the rotation angle (in radians) for the block relative to the X-axis, with positive values going counterclockwise when viewed along the Z-axis toward the origin.
`Rows`	Integer	Gets or sets the number of rows in a block array.
`RowSpacing`	Double	Gets or sets the spacing of the rows in a block array.
`XScaleFactor`	Double	Gets or sets the X scale factor for the block.
`YScaleFactor`	Double	Gets or sets the Y scale factor for the block.
`ZScaleFactor`	Double	Gets or sets the Z scale factor for the block.

■ **Note** Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadMLine` Object

The `AcadMLine` object represents multiple parallel lines. One `AcadMLine` object can comprise up to 16 lines. The object is created using the `AddMLine` method of the `AcadBlock`, `AcadModelSpace`, or `AcadPaperSpace` object.

`AcadMLine` Object Methods

The `AcadMLine` object inherits all the methods of the `AcadEntity` and `AcadObject` objects. It supports no other methods.

`AcadMLine` Object Properties

The `AcadMLine` object inherits all the properties of the `AcadEntity` and `AcadObject` objects, as well as the common `Application` property. It also supports the following properties.

Name	Returns	Description
`Coordinates`	Variant	Gets or sets the coordinates for each vertex in the object. This will replace any existing coordinates for the specified object. Use standard array-handling techniques to process the coordinates contained in this property. Note that you can't change the number of coordinates in the object by using this property; you can change only the location of existing coordinates. Returns a three-element array of Doubles containing 3-D coordinates in WCS. Note also that the Z coordinate will always default to 0 on the active UCS.
`StyleName`	String	Gets the `MLine` style name. This property's value is read-only.

■ **Note** Because this object inherits from `AcadObject`, it supports the `Modified` event.

AcadModelSpace Collection

The AcadModelSpace collection contains all the objects in model space. It is similar to the AcadBlock object, and it supports the same methods and properties. Moreover, objects contained in this collection can also be found in the blocks collection in a special block called *MODEL_SPACE, which is to say that both the block named *MODEL_SPACE and the AcadModelSpace collection actually point to the same data. The AcadModelSpace collection was created to provide a faster and easier means of accessing the data in model space. There is a single AcadModelSpace collection provided in the drawing. The collection can't be created or deleted. To add objects to the collection, you use the same Add* methods presented for the AcadBlock object.

Use model space for drafting and design work and to create 2-D drawings or 3-D models.

Methods and Properties

The AcadModelSpace collection supports all the methods and properties supported by the AcadBlock object, as well as the common Application property. See the AcadBlock object for more details.

■ **Note** Because this collection inherits from AcadObject, it supports the Modified event.

AcadMText Object

The AcadMText object represents a paragraph of alphanumeric characters that fits within a nonprinting text boundary. Each AcadMText object is a single object regardless of the number of lines it contains, and AutoCAD uses word wrap to break long lines into paragraphs. Text can be imported into an AcadMText object, but all embedded formatting will be lost. The text boundary definition specifies the width and alignment of the paragraph, and although it isn't plotted or printed, it remains part of the object's framework. AcadMText objects are created using the AddMText method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadMText Object Methods

The AcadMText object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadMText Object Properties

The AcadMText object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
AttachmentPoint	AcAttachmentPoint	Gets or sets the attachment point for an AcadMText object. The option selected determines both the text justification and text spill in relation to the text boundary. When the property is changed, the position of the existing bounding box doesn't change. The text is simply rejustified within the bounding box. For a list of possible values for the AcAttachmentPoint enumerated type, see Appendix B.
DrawingDirection	AcDrawingDirection	Gets or sets the direction in which the MText paragraph is to be read. Possible values for the AcDrawingDirection enumeration are acLeftToRight and acTopToBottom. (Three other values are reserved for future use: acRightToLeft, acBottomToTop, and acByStyle.)
Height	Double	Sets the height of uppercase text, measured in the current units. This property is used as a scale factor for both the height and width of the text. \H also indicates a height change.
InsertionPoint	Variant	Sets the insertion point for the text and specifies the location for a corner of the text boundary. The AttachmentPoint property specifies which corner of the text boundary is to be positioned at this insertion point.
LineSpacingFactor	Double	Gets or sets the relative line spacing factor for the AcadMText object, defined as the vertical distance between the baseline of one text line and the baseline of the next text line. It's set as a multiple of single line spacing and should have a value between 2.5 and 4.0.
LineSpacingStyle	AcLineSpacingStyle	Gets or sets the line spacing style for the AcadMText object. For a list of possible values for the AcLineSpacingStyle enumerated type, see Appendix B.
Normal	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the AcadMText object.
Rotation	Double	Gets or sets the rotation angle (in radians) for MText relative to the X-axis, with positive values going counterclockwise when viewed along the Z-axis toward the origin.
StyleName	String	Gets or sets the name of the style used with the object, the default being the current style. Use the TextStyle object to change the attribute of a given text style. Note that the name given must already be defined in the drawing.
TextString	String	Gets or sets the text string for the entity. The text string may contain format codes; for example, you can underline text, add a line over text, and create stacked text.
Width	Double	Gets or sets the width of the text boundary in the current units. AutoCAD wraps the text within the text boundary; therefore, the width must be a positive number large enough to accommodate the text. If the width isn't large enough, the text may be difficult to read or may not be visible at all. \W also indicates a width change.

Note Because this object inherits from AcadObject, it supports the Modified event.

AcadPaperSpace Collection

The AcadPaperSpace collection contains all the objects in paper space layout, and if a new paper space is made active, the data in this collection will be changed to reflect the data in the new active layout. It is similar to the AcadBlock object, and it supports the same methods and properties, with just one additional method. Moreover, objects contained in this collection can also be found in the blocks collection in a special block called *PAPER_SPACE, which is to say that both the block named *PAPER_SPACE and the AcadPaperSpace collection actually point to the same data. The AcadPaperSpace collection was created to provide a faster and easier means of accessing the data in the active paper space layout.

Paper space layouts are used for creating a finished layout for printing or plotting the drawing and not doing drafting or design work, which is carried out in the model space. Although both 2-D and 3-D objects can exist in paper space, commands that render a 3-D viewpoint are disabled. The AcadPaperSpace collection can't be created or deleted.

Use paper space to create a finished layout of a drawing for printing or plotting.

AcadPaperSpace Collection Methods

The AcadPaperSpace collection inherits all the methods of the AcadBlock and AcadModelSpace collections. It also supports the following method.

Name	Returns	Description
`AddPViewport`	`AcadPViewport`	Adds a paper space viewport, given the coordinates for the center of the viewport, its height, and its width. Parameters: `Center As Variant`, `Width As Double`, and `Height As Double`.

AcadPaperSpace Collection Properties

The AcadPaperSpace collection inherits all the properties of the AcadBlock and AcadModelSpace collections, as well as the common Application property. It supports no other properties.

Note Because this collection inherits from AcadObject, it supports the Modified event.

AcadPlot Object

The AcadPlot object represents the set of methods and properties used for plotting specified layouts. You can display a plot preview, plot to a file or plot to a device, or batch-plot several plot commands. You can set the number of copies of the drawing to be plotted and specify quiet error mode so that the plot session will be uninterrupted.

However, it should be noted that you can't create a configured plotter (PC3) file using ActiveX. To create a PC3 file, use the Add-a-Plotter wizard in AutoCAD. For more information on PC3 files, see "Plotter-Specific Configuration Information" in the AutoCAD documentation.

AcadPlot Object Methods

The AcadPlot object supports the following methods.

Name	Returns	Description
DisplayPlotPreview		Displays the plot preview dialog box for the active layout. Parameter: Preview As AcPreviewMode. For a description of possible values for the AcPreviewMode enumerated type, see Appendix B. Note that there's no hourglass indicating the regeneration process and that the user needs to dismiss the preview for this method to return. Any VBA UserForm should be hidden during a plot preview.
PlotToDevice	Boolean	Plots a layout to a device and returns whether the plot was successfully sent. Parameter: plotConfig As String. Specifies the full path and file name of the PC3 file to use instead of the current configuration. If this parameter isn't provided, the current configuration will be used. A dialog window will indicate the printing progress. Any VBA UserForm should be hidden during a PlotToDevice operation.
PlotToFile	Boolean	Plots a layout to a file and returns whether the plot was successfully sent. Parameters: PlotFile As String and [PlotConfig As String]. PlotFile specifies the name of the file to which to plot the active layout. When you plot multiple layouts, the file name for each plot will be generated from the drawing and layout names. A dialog window will indicate the printing progress. Any VBA UserForm should be hidden during a PlotToFile operation. PlotConfig specifies the full path and file name of the PC3 file to use instead of the current configuration. If this parameter isn't provided, the current configuration will be used.
SetLayoutsToPlot		Sets the layout or layouts to be plotted. If you want a layout other than the active layout to be plotted, you must call this method. Parameter: LayoutList As Variant (an array of layout objects). If the LayoutList parameter is NULL or this method isn't called at all, the active layout is sent to the plot. Note that a call to this method has to precede each layout plotting in an iteration. Before you enter a batch operation, you have to specify the number of layouts to plot using the StartBatchMode method.
StartBatchMode		Invokes batch mode printing. This method allows the plot progress dialog box to show the progress of batch plots. This remains active until the batch is completed. Parameter: EntryCount As Long. The EntryCount parameter specifies the number of calls to either the PlotToFile method or the PlotToDevice method that must follow the StartBatchMode method.

AcadPlot Object Properties

As well as the common Application property, the AcadPlot object supports the following properties.

Name	Returns	Description
`QuietErrorMode`	Boolean	Specifies whether quiet error mode is enabled. Quiet error mode logs all plot-related error messages into a log file. This is essential for certain plotting applications, such as batch plotting, that require uninterrupted application execution. When the quiet error mode is disabled, errors during printing are displayed in alert boxes.
`NumberOfCopies`	Long	Gets or sets the number of copies to be plotted.
`BatchPlotProgress`	Boolean	Specifies whether the batch plot is in progress. Setting this property to `False` terminates the batch plot. You can achieve this programmatically or by clicking the Cancel button in the progress dialog box.

`AcadPlotConfiguration` Object

See the `AcadLayout` object.

`AcadPlotConfigurations` Collection

See the `AcadLayouts` collection.

`AcadPoint` Object

The `AcadPoint` object represents a point marker on a drawing appearing as a dot, square, circle, X, tick, plus sign, or as any combination of these. You can specify a full 3-D location for a point. However, the current elevation is assumed if you supply two coordinates. Points can also act as nodes onto which you can snap objects. The `Coordinates` property provides an array that can be used in any method requiring a point as an input parameter. `AcadPoint` objects are created using the `AddPoint` method of the `AcadBlock`, `AcadModelSpace`, or `AcadPaperSpace` object.

The `PDMODE` and `PDSIZE` system variables control the appearance of point objects. The values for `PDMODE` are as follows:

0	Draws point as a dot (.)
1	No symbol drawn
2	Draws point as a plus (+)
3	Draws point as a cross (x)
4	Draws point as a vertical dash (|)

You can add 32, 64, or 96 to any of the preceding values to specify a shape to draw around the point in addition to the point symbol itself. The values represent a circle, a square, and a circle within a square, respectively.

The `PDSIZE` variable controls the size of the point symbols for `PDMODE` values 2 and 4. A setting of 0 generates the point at 5 percent of the graphics area height. A positive value specifies an absolute size for the point figures; a negative value is interpreted as a percentage of the viewport size.

Any changes to the points (`PDMODE` or `PDSIZE`) are displayed when the drawing is regenerated.

AcadPoint Object Methods

The AcadPoint object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadPoint Object Properties

The AcadPoint object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Coordinates	Variant	Gets or sets the coordinates. It returns a three-element array of Doubles containing 3-D coordinates in WCS.
Normal	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the mesh.
Thickness	Double	Gets or sets the distance an AcadPoint object is extruded above or below its elevation, in a Z-axis direction. The default is 0.0.

■**Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadPolyfaceMesh Object

The AcadPolyfaceMesh object represents a 3-D polyface mesh. It is created using the AddPolyfaceMesh method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadPolyfaceMesh Object Methods

The AcadPolyfaceMesh object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadPolyfaceMesh Object Properties

The AcadPolyfaceMesh object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Coordinate	Variant	Gets or sets the coordinate of a single vertex in the object. This will replace an existing vertex for the specified object. Use standard array-handling techniques to process the values contained in this property. It returns a three-element array of Doubles containing 3-D coordinates in WCS. Note that the Z coordinate will default to 0 on the active UCS. Parameter: Index As Integer (the index in the zero-based array of vertices for the vertex you want to set or query).

Name	Returns	Description
`Coordinates`	Variant	Gets or sets the coordinates for each vertex in the object. This will replace any existing coordinates for the specified object. Use standard array-handling techniques to process the coordinates contained in this property. Note that you can't change the number of coordinates in the object by using this property; you can change only the location of existing coordinates. It returns a three-element array of Doubles containing 3-D coordinates in WCS. Note that the Z coordinate will always default to 0 on the active UCS.
`NumberOfFaces`	Integer	Gets the number of faces in the mesh. This property's value is read-only.
`NumberOfVertices`	Integer	Gets the number of vertices in the mesh. This property's value is read-only.

Note Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadPolygonMesh` Object

The `AcadPolygonMesh` object simulates the geometric representation of a mesh laid out on a 3-D surface. It is specified as an M×N mesh, where M represents the number of vertices in a row of the mesh and N represents the number of vertices in a column. A mesh can be open or closed in either or both the M and N directions. A mesh that is closed in a particular direction is considered to be continuous from the last row or column on to the first row or column. If it is open, it is treated as discontinuous between the last row and the first row. The `AcadPolygonMesh` object is created using the `AddPolygonMesh` method of the `AcadBlock`, `AcadModelSpace`, or `AcadPaperSpace` object.

`AcadPolygonMesh` Object Methods

The `AcadPolygonMesh` object inherits all the methods of the `AcadEntity` and `AcadObject` objects. It also supports the following methods.

Name	Returns	Description
`AppendVertex`		Appends a vertex to the polygon mesh, which is an array of 3-D WCS coordinates specifying the appended row of vertices. Note that appending a vertex is the equivalent of adding an extra row to the polygon mesh matrix. Parameter: `Vertex As Variant` (a 3×M array of Doubles). Note that an array of the wrong size will result in an error, because a polygon mesh is a regular M×N matrix.
`Explode`	Variant	Explodes the polygon mesh and returns the subentities as an array of Objects (`Acad3DFace` objects).

AcadPolygonMesh Object Properties

The AcadPolygonMesh object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Coordinate	Variant	Gets or sets the coordinate of a single vertex in the object. This will replace an existing vertex for the specified object. Use standard array-handling techniques to process the values contained in this property. For simple polylines, this property specifies simple vertices. For splines or curve-fit polylines, this property specifies control point vertices. It returns a three-element array of Doubles containing 3-D coordinates in WCS. Note that the Z coordinate will default to 0 on the active UCS. Parameter: Index As Integer (the index in the zero-based array of vertices for the vertex you want to set or query).
Coordinates	Variant	Gets or sets the coordinates for each vertex in the object. This will replace any existing coordinates for the specified object. Use standard array-handling techniques to process the coordinates contained in this property. For simple polylines, this property specifies simple vertices. For splines or curve-fit polylines, this property specifies control point vertices. Note that you can't change the number of coordinates in the object by using this property; you can change only the location of existing coordinates. It returns a 3×M×N-element array of Doubles containing 3-D coordinates in WCS.
MClose	Boolean	Specifies whether the polygon mesh is closed in the M direction. The document has to be regenerated for the changes to appear.
MDensity	Integer	Gets or sets the smooth surface density of a polygon mesh in the M direction, the value of which is an integer between 2 and 255. The initial value for this property is derived from the value in the SURFU system variable plus 1. The M surface density is the number of vertices in the M direction for AcadPolygonMesh objects of the following types: acQuadSurfaceMesh, acCubicSurfaceMesh, and acBezierSurfaceMesh.
MVertexCount	Integer	Gets the vertex count in the M direction of the polygon mesh. This property's value is read-only. This is the number of vertices that will be used to make up an M row in the polygon mesh if the Type property is acSimpleMesh. For any other polygon mesh type, the MDensity value will be used as the row size. Note that you can't change the number of vertices for a simple mesh. This property is necessary when appending new vertices.
NClose	Boolean	Specifies if the polygon mesh is closed in the N direction. The document has to be regenerated for the changes to appear.

Name	Returns	Description
NDensity	Integer	Gets or sets the smooth surface density of a polygon mesh in the N direction, the value of which is an integer between 2 and 255. The initial value for this property is derived from the value in the SURFV system variable plus 1. The N surface density is the number of vertices in the N direction for AcadPolygonMesh objects of the following types: acQuadSurfaceMesh, acCubicSurfaceMesh, and acBezierSurfaceMesh.
NVertexCount	Integer	Gets the vertex count in the N direction of the polygon mesh. This property's value is read-only. This is the number of vertices that will be used to make up an N row in the polygon mesh if the Type property is acSimpleMesh. For any other polygon mesh type, the MDensity value will be used as the row size. Note that you can't change the number of vertices for a simple mesh.
Type	AcPolymeshType	Gets or sets type of a polygon mesh object. For a list of possible values for the AcPolymeshType enumerated type, see Appendix B.

Note Because this object inherits from AcadObject, it supports the Modified event.

AcadPolyline Object

The AcadPolyline object represents a 2-D line of adjustable width or a 3-D line or mesh of non-adjustable width composed of line and arc segments. Use the AcadLWPolyline object to create polylines with an optimized format that saves memory and disk space. The AcadPolyline object is created using the AddPolyline method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadPolyline Object Methods

The AcadPolyline object inherits all the methods of the AcadEntity and AcadObject objects. It also supports the following methods.

Name	Returns	Description
AppendVertex		Appends a vertex to the polyline, which is an array of 3-D coordinates. The X and Y coordinates are given in OCS; the Z coordinate is ignored. Parameter: vertex As Variant (a three-element array of doubles).
Explode	Variant	Explodes the polyline and returns the subentities as an array of Objects (AcadLine and AcadArc).

Continued

Name	Returns	Description
`GetBulge`	Double	Returns the vertex bulge value at a given index of the polyline. A bulge is the tangent of ¼ of the included angle for the arc between the selected vertex and the next vertex in the polyline's vertex list. A negative bulge value indicates that the arc goes clockwise from the selected vertex to the next vertex. A bulge of 0 indicates a straight segment, and a bulge of 1 is a semicircle. Note that this method will fail if the polyline `Type` property isn't `acSimplePoly`. Parameter: `Index As Integer`.
`GetWidth`		Returns segment width at a given index of the polyline. Parameters: `Index As Integer`, `StartWidth As Double`, and `EndWidth As Double`.
`Offset`	Variant	Creates a new polyline by offsetting the polyline by a given distance, which must be nonzero. If the offset is negative, this means that the line is drawn closer to the WCS origin. Parameter: `Distance As Double`.
`SetBulge`		Sets the vertex bulge of the polyline at a given index. See `GetBulge` for the definition of a bulge. Note that this method will fail if the polyline `Type` property isn't `acSimplePoly`. Parameters: `Index As Integer` and `bulge As Double`.
`SetWidth`		Sets the segment width of the polyline for a given index. Parameters: `Index As Integer`, `StartWidth As Double`, and `EndWidth As Double`.

`AcadPolyline` Object Properties

The `AcadPolyline` object inherits all the properties of the `AcadEntity` and `AcadObject` objects, as well as the common `Application` property. It also supports the following properties.

Name	Returns	Description
`Area`	Double	Gets the area of the polyline in square drawing units. The area is computed as though a straight line connects the start and end points. This property's value is read-only.
`Closed`	Boolean	Specifies whether the polyline is closed (`True`) or open (`False`). Open is the default.
`ConstantWidth`	Double	Gets or sets a global width for all segments in a polyline.
`Coordinate`	Variant	Gets or sets the coordinate of a single vertex in the object. This will replace an existing vertex for the specified object. Use standard array-handling techniques to process the values contained in this property. For simple polylines, this property specifies simple vertices. For splines or curve-fit polylines, this property specifies control point vertices. It returns a three-element array of Doubles containing 3-D coordinates in OCS (the Z coordinate is ignored). These coordinates can be converted to and from other coordinate systems using the `TranslateCoordinates` method. Parameter: `Index As Integer` (the index in the zero-based array of vertices for the vertex you want to set or query).

Name	Returns	Description
Coordinates	Variant	Gets or sets the coordinates for each vertex in the object. This will replace any existing coordinates for the specified object. Use standard array-handling techniques to process the coordinates contained in this property. For simple polylines, this property specifies simple vertices. For splines or curve-fit polylines, this property specifies control point vertices. Note that you can't change the number of coordinates in the object by using this property; you can change only the location of existing coordinates. It returns the array point as a three-element array of Doubles containing 3-D points in OCS (the Z coordinate is ignored). These coordinates can be converted to and from other coordinate systems using the TranslateCoordinates method.
Elevation	Double	Gets or sets the current elevation of the polyline.
LinetypeGeneration	Boolean	Specifies the linetype generation of a polyline. If it's set to True, the linetype is generated in a continuous pattern through the polyline vertices; if it's set to False, the linetype is generated starting and ending with a dash at each polyline vertex.
Normal	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the polyline.
Thickness	Double	Gets or sets the distance an AcadPolyline object is extruded above or below its elevation, in a Z-axis direction. The default is 0.0.
Type	AcPolylineType	Gets or sets type of a polyline object. For a list of possible values for the AcPolylineType enumerated type, see Appendix B.

Note Because this object inherits from AcadObject, it supports the Modified event.

AcadPopupMenu Object

The AcadPopupMenu object represents an AutoCAD cascading menu of which there are two types: pull-down menus, which are accessed from the AutoCAD menu bar, and shortcut menus, which are accessed by pressing Shift and right-clicking. A pop-up menu is a collection of menu items that includes AcadPopMenuItems, separators, and submenus, which can be added to the collection using the AddMenuItem, AddSeparator, and AddSubmenu methods.

To create a new pull-down menu, use the Add method to add a new AcadPopupMenu object to the AcadPopupMenus collection, and then add the menu to the menu bar using the InsertInMenuBar method. To create a new shortcut menu, you must first delete any existing shortcut menu, as there can be only one shortcut menu per menu group. If there is no other shortcut menu in a menu group, you can then add a menu with the label POP0, which will be recognized as the new shortcut menu. This label will tell AutoCAD that you want to create a cursor menu. Consult the AcadPreferenceUser object for the list of properties to customize the shortcut menu.

AcadPopupMenu Object Methods

The AcadPopupMenu object supports the following methods.

Name	Returns	Description
AddMenuItem	AcadPopupMenuItem	Adds an item to the pop-up menu. Parameters: Index As Variant (Integer or String), Label As String, and Macro As String. The Index parameter for this property and the four that follow must be between 0 and N minus 1 (if it's an Integer), where N is the number of objects in the pop-up menu. The new item will be added immediately before the specified index location. To add the new menu item to the end of a menu, set the index to be greater than N. If a String is specified and the indexed item doesn't exist, then the new menu item is added at the end of the menu. The Label parameter may contain DIESEL string expressions. Labels also identify the accelerator keys (keyboard key sequences) that correspond to the menu item by placing an ampersand (&) in front of the accelerator character.
AddSeparator	AcadPopupMenuItem	Adds a separator to the pop-up menu. The first item in a menu can't be a separator. You can't add a separator immediately next to another separator. Parameter: Index As Variant (Integer or String).
AddSubmenu	AcadPopupMenu	Adds a submenu to the pop-up menu. Parameters: Index As Variant (Integer or String) and Label As String. The Label parameter may contain DIESEL string expressions. Labels also identify the accelerator keys (keyboard key sequences) that correspond to the menu item by placing an ampersand (&) in front of the accelerator character.
InsertInMenuBar		Inserts the pop-up menu into the menu bar at the location specified by the index. Parameter: Index As Variant.
Item	AcadPopupMenuItem	Returns the member of the collection specified by the index. Parameter: Index As Variant (Integer or String). If the Index value is a String, it must match an existing object name in the collection. To determine if the returned item is a menu item, a separator, or a submenu, use the Type property of the returned AcadPopupMenuItem object.
RemoveFromMenuBar		Removes the pop-up menu from the menu bar.

AcadPopupMenu Object Properties

As well as the common Application property and the Count property, the AcadPopupMenu object supports the following properties.

Name	Returns	Description
Name	String	Gets or sets the name of the AcadPopupMenu object.
NameNoMnemonic	String	Gets the name of the menu without the underscore mnemonic.
OnMenuBar	Boolean	Specifies whether the menu is on the menu bar. This property's value is read-only.
Parent	Object	Gets the parent of the AcadPopupMenu object. This property's value is read-only. The valid object types are AcadMenuBar and AcadPopupMenus.
ShortcutMenu	Boolean	Specifies whether the menu is a shortcut menu.
TagString	String	Gets the menu's tag string, which can consist of alphanumeric and underscore (_) characters. This string uniquely identifies the item within a given menu file. This string is automatically assigned when the object is created and is used internally by AutoCAD for toolbar and menu identification. This property's value is read-only.

AcadPopupMenuItem Object

The AcadPopupMenuItem object represents a single menu item on an AutoCAD pull-down menu. There are three types: a simple menu item, a separator, and a submenu item. They are created using the AddMenuItem, AddSeparator, or AddSubmenu method of the AcadPopupMenu object.

Simple menu items have an index, a label, a tag, and a macro. The index is the position on the menu where the individual menu item is located. The label is a string that defines the content and formatting of the menu item as it appears to the user. The tag is a string consisting of alphanumeric and underscore (_) characters, which uniquely identifies the menu item within a given menu. The macro is a series of commands that execute specific actions when a menu item is selected.

Separators have only an index, which specifies where on the menu the separator is to appear. Submenu items have an index, a label, and a tag, but no macro. Instead, they have an entire AcadPopupMenu object associated with them, which appears when the user selects the submenu item. To find an AcadPopupMenu object associated with a submenu item, use the SubMenu property.

AcadPopupMenuItem Object Methods

The AcadPopupMenuItem object supports a single method.

Name	Description
Delete	Deletes the item from the menu

AcadPopupMenuItem Object Properties

As well as the common Application property, the AcadPopupMenuItem object supports the following properties.

Name	Returns	Description
Caption	String	Gets the text seen by the user for the menu item. This property is derived from the Label property by removing any DIESEL string expressions. This property's value is read-only.
Check	Boolean	Specifies the check status for the menu item (the tick to the left of the menu item caption).
Enable	Boolean	Specifies whether the item is enabled (grayed out).
EndSubMenuLevel	Integer	Gets or sets the last item in a submenu. The default value for this property is 0, specifying that this menu item isn't the last item in the submenu. A value of 1 indicates that this item is the last item in a submenu. A value of 2 indicates that this item is the last item of a submenu and also of its parent menu. This value should be incremented for each parent menu for which this item terminates.
HelpString	String	Gets or sets the help string for the menu item, which appears in the AutoCAD status line when a user highlights a menu item.
Index	Integer	Gets the index for the menu item. The first position in the index is 0. This property's value is read-only.
Label	String	Gets or sets the content and formatting of the menu item as they appear to the user.
Macro	String	Gets or sets the name of the macro associated with the menu item.
Parent	AcadPopupMenu	Gets the parent of the object. This property's value is read-only.
SubMenu	AcadPopupMenu	Gets the submenu, if the item is associated with one. This information can be obtained from the object Type property. This property's value is read-only.
TagString	String	Gets or sets the item's tag string, which can consist of alphanumeric and underscore (_) characters. This string uniquely identifies the item within a given menu file. This string is automatically assigned when the object is created and is used internally by AutoCAD for toolbar and menu identification.
Type	AcMenuItemType	Gets the item's type. This property's value is read-only. Possible values for the AcMenuItemType enumeration are acMenuItem, acMenuSeparator, and acMenuSubMenu.

AcadPopupMenus Collection

The AcadPopupMenus collection contains all the AcadPopupMenu objects loaded in the menu group, some or all of which may be currently displayed in AutoCAD. To load an existing menu into the current session, use the Load method of the AcadMenuGroups collection. To access this collection, use the Menus property on the specific AcadMenuGroup. To create a new menu, use the Add method. To display a menu on the AutoCAD menu bar, use the InsertMenuInMenuBar method. To remove a menu from the menu bar, use the RemoveMenuFromMenuBar method.

AcadPopupMenus Collection Methods

The AcadPopupMenus collection supports four methods.

Name	Returns	Description
Add	AcadPopupMenu	Creates an AcadPopupMenu object and adds it to the collection. Parameter: Name As String.
InsertMenuInMenuBar		Inserts a menu into the menu bar immediately before the position specified by the index. Parameters: MenuName As String and Index As Variant (Integer or String). If the Index is an Integer, it must be between 0 and N minus 1, where N is the number of pop-up menus in the menu bar. If the Index is a String, the name of the pop-up menu must contain any underscore (_) that appears in the menu. This method is equivalent to the InsertInMenuBar method of the AcadPopupMenu object.
Item	AcadPopupMenu	Returns the item in the collection specified by the index. Parameter: Index As Variant (Integer or String).
RemoveMenuFromMenuBar		Removes the menu specified by the index from the menu bar. Parameter: Index As Variant (Integer or String). If the Index is an Integer, it must be between 0 and N minus 1, where N is the number of pop-up menus in the menu bar. If it is a String, the name of the pop-up menu must contain any underscore (_) that appears in the menu, which corresponds to the TagString property of the AcadPopupMenu object.

AcadPopupMenus Collection Properties

As well as the common Application property, the AcadPopupMenus collection supports the following property.

Name	Description
ParentAcadMenuGroup	Gets the collection's parent object. This property's value is read-only.

AcadPreferences Object

The AcadPreferences object specifies the current AutoCAD setting, holding all the options from the Options dialog box that reside in the Registry. Consult this dialog box from the Tools menu for a list of all the current settings, most of them being accessed using the AcadPreferences object. Options that reside in the drawing can be found through the AcadDatabasePreferences object. The AcadPreferences object can be referenced from the Preferences property on the AcadApplication object. The AcadPreferences object has no methods.

AcadPreferences Object Properties

In addition to the common Application property, the AcadPreferences object supports the following properties.

Name	Returns	Description
Drafting		Gets the AcadPreferencesDrafting object for the application. This property's value is read-only.
Display		Gets the AcadPreferencesDisplay object for the application. This property's value is read-only.
Files		Gets the AcadPreferencesFiles object for the application. This property's value is read-only.
OpenSave		Gets the AcadPreferencesOpenSave object for the application. This property's value is read-only.
Output		Gets the AcadPreferencesOutput object for the application. This property's value is read-only.
Profiles		Gets the AcadPreferencesProfiles object for the application. This property's value is read-only.
Selection		Gets the AcadPreferencesSelection object for the application. This property's value is read-only.
System		Gets the AcadPreferencesSystem object for the application. This property's value is read-only.
User		Gets the AcadPreferencesUser object for the application. This property's value is read-only.

AcadPreferencesDisplay Object

The AcadPreferencesDisplay object contains the options from the Display tab on the Options dialog box. The object has no methods.

AcadPreferencesDisplay Object Properties

In addition to the common Application property, the AcadPreferencesDisplay object supports the following properties.

Name	Returns	Description
AutoTrackingVecColor	VBA.ColorConstants	Gets or sets the color of the autotracking vector. The default color for this property is vbWhite. Possible values: any member of VBA.ColorConstants.
CursorSize	Integer	Gets or sets the size of the crosshairs as a percentage of the size of the screen. The default value for this property is 5 percent. The allowable range is 1 to 100 percent of the total screen. At 100 percent, the ends of the crosshairs aren't visible. The value for this property is stored in the CURSORSIZE system variable.
DisplayLayoutTabs	Boolean	Specifies whether the Model and Layout tabs appear in the drawing editor. True is the initial value.

Name	Returns	Description
`DisplayScreenMenu`	Boolean	Specifies whether the screen menu appears on the right side of the drawing window. `False` is the initial value.
`DisplayScrollBars`	Boolean	Specifies whether scrollbars appear at the bottom and right sides of the drawing window. `False` is the initial value.
`DockedVisibleLines`	Integer	Gets or sets the number of lines of text visible in the command window. The initial value for this property is 3. Note that while it's still available, this property has no effect in AutoCAD 2004 products.
`GraphicsWinLayout-BackgrndColor`	`VBA.ColorConstants`	Gets or sets the background color of the paper space layouts. The initial value for this property is a color halfway between the Windows standard background color and the Windows standard active button color. Possible values: any member of `VBA.ColorConstants`.
`GraphicsWinModel-BackgrndColor`	`VBA.ColorConstants`	Gets or sets the background color of the model space window. The default color for this property is `vbBlack`. Possible values: any member of `VBA.ColorConstants`.
`HistoryLines`	Integer	Gets or sets the number of lines of text from the text window kept in the memory. The initial value is 400.
`ImageFrameHighlight`	Boolean	Specifies how the raster image is displayed when the image is selected. If `True`, only the raster image is displayed. If `False`, the raster image content is displayed. The value for this property is stored in the `IMAGEHLT` system variable.
`LayoutCreateViewport`	Boolean	Specifies whether a viewport is automatically created for each new layout. `True` is the initial value.
`LayoutCrosshairColor`	`VBA.ColorConstants`	Gets or sets the color of the crosshairs and text in paper space layouts. The initial value for this property is the Windows standard text color. Possible values: any member of `VBA.ColorConstants`.
`LayoutDisplayMargins`	Boolean	Specifies whether margins are displayed in layouts. `True` is the initial value.
`LayoutDisplayPaper`	Boolean	Specifies whether the paper background is displayed in layouts. `True` is the initial value.
`LayoutDisplayPaperShadow`	Boolean	Specifies whether the paper background shadow is displayed in layouts. `True` is the initial value.
`LayoutShowPlotSetup`	Boolean	Specifies whether the plot setup dialog box is displayed when a new layout is created. `True` is the initial value.

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Name	Returns	Description
MaxAutoCADWindow	Boolean	Specifies whether AutoCAD occupies the entire screen at the start of the session.
ModelCrosshairColor	VBA.ColorConstants	Gets or sets the color of the crosshairs and of the text for the model space. The initial value of this property is vbWhite. Possible values: any member of VBA.ColorConstants.
ShowRasterImage	Boolean	Specifies how raster images are displayed during real-time pan and zooms. If True, the image content is displayed. If False, only the image outline is displayed (the default). If the dragging display is turned on and you enable ShowRasterImage, a copy of the image moves with the cursor as you reposition the original image. The value for this property is stored in the RTDISPLAY system variable.
TextFontSize	Integer	Gets or sets the font size for new text. The initial value for this property is 10.
TextFont	String	Gets or sets the font for new text. The initial value for this property is Courier.
TextFontStyle	AcTextFontStyle	Gets or sets the font style for new text. For a list of possible values for the AcTextFontStyle enumerated type, see Appendix B.
TextWinBackgrndColor	VBA.ColorConstants	Gets or sets the text window's background color. The default color for this property is the Windows standard window color. Possible values: any member of VBA.ColorConstants.
TextWinTextColor	VBA.ColorConstants	Gets or sets the text window's text color. The default color for this property is the Windows standard window text color. Possible values: any member of VBA.ColorConstants.
TrueColorImages	Boolean	Gets or sets how the colors in raster images and render images are displayed. If True, the highest possible color resolution is used. If False, palette colors are used (the default).
XrefFadeIntensity	Long	Gets or sets the percentage of fade for external references. The value must be between 1 and 90. The default value is 50.

AcadPreferencesDrafting Object

The AcadPreferencesDrafting object contains the options from the Drafting tab on the Options dialog box. The object has no methods.

AcadPreferencesDrafting Object Properties

In addition to the common Application property, the AcadPreferencesDrafting object supports the following properties.

Name	Returns	Description
AlignmentPointAcquisition	AcAlignmentPointAcquisition	Gets or sets how autoalignment points are acquired. For a list of possible values for the AcAlignmentPointAcquisition enumerated type, see Appendix B.
AutoSnapAperture	Boolean	Specifies whether the autosnap aperture is displayed in the center of the crosshairs when you snap to an object. This option is available only when the autosnap marker, magnet, or tooltip is enabled. The value for this property is stored in the APBOX system variable. The initial value for this property is True.
AutoSnapApertureSize	Long	Gets or sets whether the size of the autosnap aperture in pixels. This option is available only when the autosnap marker, magnet, or tooltip is enabled. The value for this property is stored in the APERTURE system variable. The initial value is 10.
AutoSnapMagnet	Boolean	Specifies whether the autosnap magnet is enabled. The value for this property is stored in the AUTOSNAP system variable. True is the initial value.
AutoSnapMarker	Boolean	Specifies whether the autosnap marker is enabled. The value for this property is stored in the AUTOSNAP system variable. True is the initial value.
AutoSnapMarkerColor	AcColor	Gets or sets the autosnap marker's color. For a list of possible values for the AcColor enumerated type, see Appendix B.
AutoSnapMarkerSize	Long	Gets or sets the autosnap marker's size in pixels. The value must be between 1 and 20.
AutoSnapToolTip	Boolean	Specifies whether the autosnap tooltips are enabled. The value for this property is stored in the AUTOSNAP system variable. True is the initial value.
AutoTrackToolTip	Boolean	Specifies whether the autotrack tooltips are displayed. When it's enabled, the user sees a tooltip whenever the cursor is over an alignment path. When it's disabled, the tooltip doesn't appear. The value for this property is stored in the AUTOSNAP system variable. True is the initial value.
FullScreenTrackingVector	Boolean	Specifies whether full-screen tracking vectors are displayed. The value for this property is stored in the TRACKPATH system variable. True is the initial value.
PolarTrackingVector	Boolean	Specifies whether polar tracking vectors are displayed. The value for this property is stored in the TRACKPATH system variable. True is the initial value.

AcadPreferencesFiles Object

The AcadPreferencesFiles object contains the options from the Files tab on the Options dialog box.

AcadPreferencesFiles Object Methods

The AcadPreferencesFiles object supports just two methods.

Name	Returns	Description
GetProjectFilePath	String	Returns the directory where external reference files are sought. This name is also controlled by the PROJECTNAME system variable. Parameter: ProjectName As String.
SetProjectFilePath		Sets the directory where external reference files are sought. This name is also controlled by the PROJECTNAME system variable. Parameters: ProjectName As String and ProjectFilePath As String.

AcadPreferencesFiles Object Properties

In addition to the common Application property, the AcadPreferencesFiles object supports the following properties.

Name	Returns	Description
AltFontFile	String	Gets or sets the location of the font file used if the original font can't be located and an alternative font isn't specified in the font-mapping file.
AltTabletMenuFile	String	Gets or sets the path for an alternative menu to swap with the AutoCAD standard tablet menu. This property is applicable only if you use a digitizer.
AutoSavePath	String	Gets or sets the path for the file that is created if automatic saving is enabled with the AutoSaveInterval property of the AcadPreferencesOpenSave object. The value for this property is stored in the SAVEFILEPATH system variable.
ConfigFile	String	Gets the location of the configuration file where information about the hardware device driver is stored. This value is read-only and can be changed only using the /c command-line switch, which is added to the acad.exe command line at startup.
CustomDictionary	String	Gets or sets the custom dictionary for the AutoCAD session. This property contains its drive, path, and file name.
DefaultInternetURL	String	Gets or sets the default Internet address. The value for this property is stored in the INETLOCATION system variable.
DriversPath	String	Gets or sets the directory where to look for ADI device drivers for the video display, pointing devices, printers, and plotters. Specify a drive letter and path for each path. Separate multiple directory listing with a semicolon (;).
FontFileMap	String	Gets or sets the location of the file that defines how fonts are converted if they can't be located.
HelpFilePath	String	Gets or sets the location of the AutoCAD help file. This property specifies its drive, path, and file name.

Name	Returns	Description
LicenseServer	String	Gets a list of client license servers available to the network license manager program. The value for this property is stored in the ACADSERVER environment variable. This property's value is read-only.
LogFilePath	String	Gets or sets the location of the log file. The value for this property is stored in the LOGFILEPATH system variable. Use the LogFileOn property of the AcadPreferencesOpenSave object to enable or disable log file capabilities.
MainDictionary	String	Gets or sets the dictionary used for spell checking.
MenuFile	String	Gets or sets the location of the menu file for the AutoCAD session. This property contains the drive, path, and file name for an .mnu, .mns, or .mnc file.
ObjectARXPath	String	Gets or sets the location for ObjectARX applications.
PostScriptPrologFile	String	Gets or sets a name for a customized prolog section in the acad.psf file. The value for this property is stored in the PSPROLOG system variable.
PrinterConfigPath	String	Gets or sets the location of the printer configuration files. A semicolon is used to separate multiple directories but shouldn't be used at the end of the string.
PrinterDescPath	String	Gets or sets the location of the printer description files. A semicolon is used to separate multiple directories but shouldn't be used at the end of the string.
PrinterStyleSheetPath	String	Gets or sets the location of the printer style sheet files. A semicolon is used to separate multiple directories but shouldn't be used at the end of the string.
PrintFile	String	Gets or sets an alternative name for the temporary plot file. The default is the name of the drawing followed by the extension .plt.
PrintSpoolerPath	String	Gets or sets the directory of the print spool files. The plot is written to this location.
PrintSpoolExecutable	String	Gets or sets the application used for print spooling including the drive, the path, and the application name of the print spooling executable used with autospool.
SupportPath	String	Gets or sets the directories to look for support files (support, fonts, help, and bonus files). A semicolon is used to separate multiple directories but shouldn't be used at the end of the string.
TempFilePath	String	Gets or sets the directory used to store temporary files.
TemplateDWGPath	String	Gets or sets the path for the template files used by the startup wizards.
TempXrefPath	String	Gets or sets the location of the external reference files. This location is used for the copy of the external reference if you choose acEnableWithCopy demand loading on the XrefDemandLoad property of the AcadPreferencesOpenSave object.
TextEditor	String	Gets or sets the name of the text editor for the MTEXT command.
TextureMapPath	String	Gets or sets the directory where to look for rendering texture maps. Multiple paths can be entered by using a semicolon (;) to paths.
WorkspacePath	String	Gets or sets the path for the database workspace file.

AcadPreferencesOpenSave Object

The AcadPreferencesOpenSave object contains the options from the Open and Save tab on the Options dialog box. The object has no methods.

AcadPreferencesOpenSave Object Properties

In addition to the common Application property, the AcadPreferencesOpenSave object supports the following properties.

Name	Returns	Description
AutoEdit	Boolean	Specifies whether an audit should be performed after a DXFIN or DXBIN interchange command is rendered.
AutoSaveInterval	Integer	Gets or sets an interval for automatic saving. The value must be between 0 and 600. A value of 0 indicates that no automatic saving is required. The timer starts as soon as you make a change to a drawing. It is reset and restarted whenever the drawing is saved. The value for this property is stored in the SAVETIME system variable. The initial setting for this property is 120.
CreateBackup	Boolean	Specifies whether a backup file is used. The value of this property is stored in the ISAVEBAK system variable. True is the initial value.
DemandLoadARXApp	AcARXDemandLoad	Gets and sets when a third-party application is loaded if a drawing contains custom objects created in that application. The value for this property is stored in the DEMANDLOAD system variable. For a list of possible values for the AcARXDemandLoad enumerated type, see Appendix B.
FullCRCValidation	Boolean	Specifies whether a cyclic redundancy check (CRC) should be performed whenever an object is read into the drawing. False is the initial value. If your drawings are being corrupted and you suspect a hardware problem or an AutoCAD error, set this property to True.
IncrementalSavePercent	Integer	Gets or sets the wasted space allowed in a file as a percentage. When the specified percentage is reached, a full save occurs instead of an incremental save. Full saves eliminate wasted space. If this property is 0, every save is a full save. Incremental saves increase the size of the drawing, but full saves take longer. The value of this property is stored in the ISAVEPERCENT system variable. The initial value of this property is 50. Autodesk recommends a setting of zero (0) when problems occur during file saves in network environments.
LogFileOn	Boolean	Specifies whether the contents of the text window are written to a log file. The value for this property is stored in the LOGFILEMODE system variable. True is the initial value.
MRUNumber	Long	Gets the number of most recently used files appearing in the File menu. The default value for this property is 4. This property's value is read-only.
ProxyImage	AcProxyImage	Gets or sets how objects are displayed in a drawing that has been created in a third-party application. Possible values for the AcProxyImage enumeration are acProxyNotShow, acProxyShow (default), and acProxyBoundingBox.

Name	Returns	Description
SaveAsType	AcSaveAsType	Gets or sets the drawing type the drawing is saved as. The initial value for this property is ac2004_DWG. For a list of possible values for the AcSaveAsType enumerated type, see Appendix B.
SavePreviewThumbnail	Boolean	Specifies whether .bmp preview images are saved with the drawing. The value for this property is saved in the RASTERPREVIEW system variable. True is the initial value.
ShowProxyDialogBox	Boolean	Specifies whether a warning message is displayed when a drawing that contains custom objects is opened. The value of this property is stored in the PROXYNOTICE system variable. True is the initial value.
TempFileExtension	String	Gets or sets the extension for temporary files. The string can contain up to three characters. The default value is "ac$".
XrefDemandLoad	AcXrefDemandLoad	Gets or sets demand loading of external references. The value for this property is stored in the XLOADCTL system variable. Demand loading improves performance by loading only the parts of the referenced drawing needed to regenerate the current drawing. The initial value for this property is acDemandLoadEnabled. For a list of possible values for the AcXrefDemandLoad enumerated type, see Appendix B.

AcadPreferencesOutput Object

The AcadPreferencesOutput object contains the options from the Output tab on the Options dialog box. The object has no methods.

AcadPreferencesOutput Object Properties

As well as the common Application property, the AcadPreferencesOutput object supports the following properties.

Name	Returns	Description
DefaultOutputDevice	String	Gets or sets the default output device for new layouts and model space. If there are no devices on the system, None will be returned.
DefaultPlotStyleForLayer	String	Gets or sets the default plot style for new drawings or for drawings created with earlier releases that haven't been saved in AutoCAD 2000 format. This property is available only when PlotPolicy is set to acPolicyNamed. The initial value of this property is Normal. The value for this property is stored in the DEFLPLSTYLE system variable.
DefaultPlotStyleForObjects	String	Gets or sets the default plot style name for newly created objects. This property is available only if PlotPolicy is set to acPolicyNamed. The initial value is ByLayer. The value for this property is stored in the DEFPLSTYLE system variable.

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Name	Returns	Description
OLEQuality	AcOleQuality	Gets or sets the plot quality of OLE objects. The initial value is acOQText. The value for this property is stored in the OLEQUALITY system variable. Possible values for the AcOleQuality enumeration are acOQLineArt, acOQText (default), acOQGraphics, acOQPhoto, and acOQHighPhoto.
PlotLegacy	Boolean	Gets or sets whether legacy plot scripts can run. The value for this property is stored in the PLOTLEGACY system variable. The initial value is False.
PlotPolicy	AcPlotPolicy	Gets or sets whether an object's color property is associated with its plot style name when a new drawing is created. The value for this property is stored in the PSTYLEPOLICY system variable. The initial value for this property is acPolicyNamed. For a list of possible values for the AcPlotPolicy enumerated type, see Appendix B.
PrinterPaperSizeAlert	Boolean	Specifies whether the user is alerted when a layout is configured with a paper size that is different from the default setting for the PC3 file. The value for this property is stored in the PAPERALERT system variable. The initial value is True.
PrinterSpoolAlert	AcPrinterSpoolAlert	Gets or sets whether the user is alerted when output needs to be spooled through a system printer because of a conflict with an I/O port. The value for this property is stored in the PSPOOLALERT system variable. The initial value is acPrinterAlwaysAlert. For a list of possible values for the AcPrinterSpoolAlert enumerated type, see Appendix B.
UseLastPlotSettings	Boolean	Specifies whether the settings of the last successful plot are applied.

AcadPreferencesProfiles Object

The AcadPreferencesProfiles object contains the options from the Profiles tab on the Options dialog box.

AcadPreferencesProfiles Object Methods

This object supports the following methods.

Name	Description
CopyProfile	Copies a profile in the Registry. Parameters: oldProfileName As String and newProfileName As String.
DeleteProfile	Deletes a profile from the Registry. Parameter: ProfileName As String.
ExportProfile	Exports a profile to a file so that other users can use it. Parameters: Profile As String and RegFile As String. The extension of the RegFile should be .arg.
GetAllProfileNames	Gets all profiles available to the system. Parameter: PNames As Variant.

Name	Description
ImportProfile	Imports a profile from a file created by another user. Parameters: Profile As String, RegFile As String, and IncludePathInfo As Boolean. The imported Registry file specified by the RegFile parameter should have an .arg extension. The IncludePathInfo parameter is True if the path information in the Registry file will be preserved and False otherwise.
RenameProfile	Renames a profile. Parameters: OrigProfileName As String and NewProfileName As String.
ResetProfile	Resets the value in a profile to its default values. The specified profile must be the currently active profile. Parameter: Profile As String.

AcadPreferencesProfiles Object Properties

As well as the common Application property, the AcadPreferencesProfiles object supports the following property.

Name	Returns	Description
ActiveProfile	String	Gets or sets the active profile for the session. The available profile names are returned by the GetAllProfileNames method.

AcadPreferencesSelection Object

The AcadPreferencesSelection object contains the options from the Selection tab on the Options dialog box. The object has no methods.

AcadPreferencesSelection Object Properties

As well as the common Application property, the AcadPreferencesSelection object supports the following properties.

Name	Returns	Description
DisplayGrips	Boolean	Specifies whether selection set grips are displayed for the stretch, move, rotate, scale, and mirror grip modes. The value for this property is stored in the GRIPS system variable. The initial value is True.
DisplayGripsWithinBlocks	Boolean	Specifies whether grips are assigned within blocks. The value for this property is stored in the GRIPBLOCK system variable. The initial value is False.
GripColorSelected	AcColor	Gets or sets the color of selected grips. The value for this property is stored in the GRIPHOT system variable. The initial value is acRed. For a list of possible values for the AcColor enumerated type, see Appendix B.
GripColorUnselected	AcColor	Gets or sets the color of grips that aren't selected. The value for this property is stored in the GRIPCOLOR system variable. The initial value is acBlue. For a list of possible values for the AcColor enumerated type, see Appendix B.

Continued

Name	Returns	Description
GripSize	Long	Gets or sets the size of grips in pixels. The value for this property is stored in the GRIPSIZE system variable. The value may be between 1 and 255, but a size greater than 20 isn't recommended. The initial value is 3.
PickAdd	Boolean	Specifies whether objects are added to the selection set by using the Shift key. The value for this property is stored in the PICKADD system variable. The initial value is False.
PickAuto	Boolean	Specifies whether windowing is automatic at the select objects prompt. The value for this property is stored in the PICKAUTO system variable. The initial value is True.
PickBoxSize	Long	Gets or sets the size of the object selection target. The value for this property is stored in the PICKBOX system variable. The initial value is 3.
PickDrag	Boolean	Specifies how a selection window is drawn. The value for this property is stored in the PICKDRAG system variable. If True, the selection window is drawn by dragging. If False, the selection window is drawn by clicking the corners. The initial value is False.
PickFirst	Boolean	Specifies whether objects are selected before a command is issued. The value for this property is stored in the PICKFIRST system variable. The initial value is True.
PickGroup	Boolean	Specifies whether picking an object in a group selects the entire group. The initial value is False.

Note The GRIPHOVER system variable added in AutoCAD 2004 isn't exposed through ActiveX in any API.

AcadPreferencesSystem Object

The AcadPreferencesSystem object contains the options from the System tab on the Options dialog box. The object has no methods.

AcadPreferencesSystem Object Properties

As well as the common Application property, the AcadPreferencesSystem object supports the following properties.

Name	Returns	Description
BeepOnError	Boolean	Specifies whether an invalid entry causes an alarm to beep. The initial value is False.
DisplayOLEScale	Boolean	Specifies whether the OLE scaling dialog box is displayed when an OLE object is inserted into a drawing. The initial value is True.
EnableStartupDialog	Boolean	Specifies whether the startup dialog box is displayed at the beginning of the session. The initial value is True.

Name	Returns	Description
LoadAcadLspInAllDocuments	Boolean	Specifies when acad.lsp is loaded. The value for this property is stored in the ACADLSPASDOC system variable. If True, it's loaded with each drawing. If False, it's loaded at startup.
ShowWarningMessages	Boolean	Specifies whether all dialog boxes with a Don't Display This Warning Again check box are displayed again. The initial value is False.
SingleDocumentMode	Boolean	Specifies whether AutoCAD runs in single-document mode rather than in multiple-document mode. The value for this property is stored in the SDI system variable. The initial is False.
StoreSQLIndex	Boolean	Specifies whether the SQL index is stored in the drawing. The initial value is True.
TablesReadOnly	Boolean	Specifies whether database tables are opened in read-only mode rather than read-write mode. The initial value is False.

AcadPreferencesUser Object

The AcadPreferencesUser object contains the options from the User tab on the Options dialog box. The object has no methods.

AcadPreferencesUser Object Properties

As well as the common Application property, the AcadPreferencesUser object supports the following properties.

Name	Returns	Description
ADCInsertUnits-DefaultSource	AcInsertUnits	Gets or sets the units that are automatically used for objects in the AutoCAD Design Center for a source drawing without assigned insert units. The value of this property is stored in the INSUNITSDEFSOURCE system variable. The initial value is acInsertUnitsUnitless. For a list of possible values for the AcInsertUnits enumerated type, see Appendix B.
ADCInsertUnits-DefaultTarget	AcInsertUnits	Gets or sets the units that are automatically used for objects in the AutoCAD Design Center for a target drawing without assigned insert units. The value of this property is stored in the INSUNITSDEFTARGET system variable. The initial value is acInsertUnitsUnitless. For a list of possible values for the AcInsertUnits enumerated type, see Appendix B.
HyperlinkDisplayCursor	Boolean	Specifies whether the hyperlink cursor and shortcut menu are displayed. When it's enabled, the user sees a small bitmap (just below and to the right of the crosshairs) whenever the cursor is over an object that contains a hyperlink. The initial value is True.

Continued

Name	Returns	Description
HyperlinkDisplayTooltip	Boolean	Specifies whether the hyperlink tooltips are displayed. When it's enabled, the user sees a tooltip whenever the cursor is over an object that contains a hyperlink. This property is automatically disabled when the HyperlinkDisplayCursor property is disabled. The initial value is True.
KeyboardAccelerator	AcKeyboardAccelerator	Gets or sets the keyboard used. The initial value is acPreferenceCustom. For a list of possible values for the AcKeyboardAccelerator enumerated type, see Appendix B.
KeyboardPriority	AcKeyboardPriority	Gets or sets the response to the input of coordinate data. The value of this property is stored in the OSNAPCOORD system variable. The initial value is acKeyboardEntryExceptScripts. For a list of possible values for the AcKeyboardPriority enumerated type, see Appendix B.
SCMCommandMode	AcDrawingAreaSCMCommand	Gets or sets the right-click functionality in the drawing area while in command mode (that is, while a command is currently in progress). The value of this property (and the three that follow) is stored in the SHORTCUTMENU system variable. This property can be set only when the ShortcutMenuDisplay property is set to True. The initial value is acEnableSCMOptions. For a list of possible values for the AcDrawingAreaSCMCommand enumerated type, see Appendix B.
SCMDefaultMode	AcDrawingAreaSCMDefault	Gets or sets the right-click functionality in the drawing area while in default mode (that is, while no objects are selected and no commands are in progress). This property can be set only when the ShortcutMenuDisplay property is set to True. The initial value is acSCM. The initial value is acEnableSCMOptions. For a list of possible values for the AcDrawingAreaSCMDefault enumerated type, see Appendix B.
SCMEditMode	AcDrawingAreaSCMEdit	Gets or sets the right-click functionality in the drawing area while in edit mode (that is, while one or more objects are selected and no commands are in progress). This property can be set only when the ShortcutMenuDisplay property is set to True. The initial value is acEdSCM. For a list of possible values for the AcDrawingAreaSCMEdit enumerated type, see Appendix B.
ShortCutMenuDisplay	Boolean	Specifies whether right-clicking in the drawing area displays a shortcut. If False, a right-click will be interpreted as an Enter. The initial value is True.

AcadPViewport Object

The AcadPViewport object represents rectangular objects created in paper space that display views. The functionality of this object is slightly different from what is used in the AutoCAD user interface. In

ActiveX Automation, the ActiveSpace property (of the Document object) is used to control the TILEMODE system variable. Also, the MSpace property (also of the Document object) is the equivalent of both AutoCAD's MSPACE and PSPACE commands. In addition, the ActiveX Automation user is required to use the Display method before setting the MSpace property to acOn. It's good practice to activate the display using the Display method for at least one AcadPViewport object before setting the MSpace property, or else the latter will throw an exception.

The AcadPViewport object is created using the AddPViewport method of the AcadPaperSpace object.

AcadPViewport Object Methods

The AcadPViewport object inherits all the methods of the AcadEntity and AcadObject objects. It also supports the following methods.

Name	Description
Display	Determines whether viewport is on or off. The display control must be on before the MSpace property can be used to activate the model space editing capabilities. Use the ViewportOn property to determine whether a paper space viewport display has already been turned on with this method. Parameter: Status As Boolean.
GetGridSpacing	Gets the grid spacing for the viewport. Parameters: XSpacing As Double and YSpacing As Double.
GetSnapSpacing	Gets the snap spacing for the viewport. Parameters: XSpacing As Double and YSpacing As Double.
SetGridSpacing	Sets the grid spacing for the viewport. Parameters: XSpacing As Double and YSpacing As Double.
SetSnapSpacing	Sets the snap spacing for the viewport. Parameters: XSpacing As Double and YSpacing As Double.

AcadPViewport Object Properties

The AcadPViewport object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
ArcSmoothness	Integer	Gets or sets the smoothness of circles, arcs, and ellipses. The initial value for this property is 100. The higher the number, the smoother the object, but AutoCAD needs longer to regenerate it. Performance can be improved by setting this property to a low value for drawing and increasing the value for rendering. The valid range is 1 to 20,000.
Center	Variant	Gets or sets the center of the viewport as a set of 2-D coordinates (as Doubles). The default center is (0,0).
Clipped	Boolean	Determines whether the viewport has been clipped. This property's value is read-only.
CustomScale	Double	Gets or sets the custom scale factor for the viewport. To set the viewport to a custom scale, first set the StandardScale property to acVpCustomScale, and then use this property to define the custom scale value.

Continued

Name	Returns	Description
Direction	Variant	Gets or sets the viewing direction for a 3-D visualization of the drawing. This property puts the viewer in a position to look at the drawing as if looking back at the origin (0, 0, 0) from a specified point in space. It's similar to AutoCAD's VPOINT command. It returns a three-element array of Doubles.
DisplayLocked	Boolean	Determines whether the viewport is locked, that is, whether it can or can't be modified (scaled, rotated, or moved). It's True if it is locked and False otherwise.
GridOn	Boolean	Specifies the status of the viewport grid. It's True if it's on and False otherwise. The value of this property is stored in the GRIDMODE system variable. Note that the grid is used only for visual reference. It isn't plotted and it isn't considered part of the drawing. The grid origin is set by the SnapBasePoint method. When you turn the grid on or off on a viewport, the visibility of the grid won't change until the viewport is made active or is reset (using the ActiveViewport property).
Height	Double	Use to set the height of the viewport, which is the Y-axis measurement of the viewport frame. The height also determines the magnification parameter in a ZoomCenter.
LensLength	Double	Gets or sets the lens length (in millimeters) used in perspective viewing, the value of which is stored in the LENSLENGTH system variable.
RemoveHiddenLines	Boolean	Specifies whether hidden lines are to be plotted on a paper space viewport.
SnapBasePoint	Variant	Gets or sets the snap base point for the viewport, the value of which is stored in the SNAPBASE system variable. The snap base point can't be changed for the active paper space viewport. Any changes to this property aren't reflected in the display until the drawing is regenerated.
SnapOn	Boolean	Specifies the status of snap mode. The value of this property is stored in the SNAPMODE system variable, and its style is changed using the SNAPSTYLE system variable. This property activates snap mode using the current snap grid resolution, rotation, and style. Note that the snap grid is invisible. Use the GridOn property to activate a separate visible grid.
SnapRotationAngle	Double	Gets or sets the snap rotation angle (in radians) of the viewport relative to the current UCS, the value of which is stored in the SNAPANG system variable. Changes to this property aren't reflected in the drawing until it's updated. The valid range is 0 to 6.28.
StandardScale	AcViewportScale	Gets or sets the standard scale for the viewport. Changes to this property won't be visible until the drawing is regenerated. For a list of possible values for the AcViewportScale enumerated type, see Appendix B.
StyleSheet	String	Gets or sets the named style sheet for the viewport.

Name	Returns	Description
`Target`	Variant	Gets or sets the target point for the viewport as a set of 3-D WCS coordinates representing the target point. The line of sight is drawn from the center to the target point. It returns a three-element array of Doubles.
`TwistAngle`	Double	Gets or sets the twist angle (in radians) for the viewport. This method twists or tilts the view around the line of sight, measured counterclockwise.
`UCSIconAtOrigin`	Boolean	Specifies whether the UCS icon is displayed at the origin.
`UCSIconOn`	Boolean	Specifies whether the UCS icon is on.
`UCSPerViewport`	Boolean	Specifies whether the UCS is saved with the viewport.
`ViewportOn`	Boolean	Specifies the display status of the viewport. This property is set by the `Display` method.
`Width`	Double	Sets the width of the viewport, which is the X-axis measurement of the viewport frame.

Note Because this object inherits from `AcadObject`, it supports the `Modified` event. There is no direct programmatic means to create or manipulate polygonal `PViewport` objects.

`AcadRasterImage` Object

The `AcadRasterImage` object represents an image consisting of a rectangular grid, or *raster*, of small squares or dots known as pixels. It is created using the `AddRaster` method of the `AcadBlock`, `AcadModelSpace`, or `AcadPaperSpace` object.

`AcadRasterImage` Constant Methods

The `AcadRasterImage` object inherits all the methods of the `AcadEntity` and `AcadObject` objects. It also supports the following method.

Name	Description
`ClipBoundary`	Sets the clipping boundary for a raster image as an array of 2-D WCS coordinates. This must be a closed and visible polygon or rectangle completely contained within the boundaries of the image. You can use this method to define a region of an image for display or plotting. Multiple instances of the same image can have different boundaries. Parameter: `Boundary As Variant` (an array of two-element array of Doubles).

AcadRasterImage Constant Properties

The AcadRasterImage object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Brightness	Integer	Gets or sets the current brightness value of an image, with values between 0 and 100. The default is 50. You adjust this value to darken or lighten an image. Note that you can't use this property with bitonal images.
ClippingEnabled	Boolean	Enables or disables the clipping boundary, defined with the ClipBoundary method.
Contrast	Integer	Gets or sets the current contrast value of an image, with values between 0 and 100. The default is 50. You adjust this value to make a dull image clearer. Note that you can't use this property with bitonal images.
Fade	Integer	Gets or sets the current fade value of an image, with values of 0 to 100. You adjust this value to make vectors easier to see over images and to create a watermark effect in your plotted output. Note that you can't use this property with bitonal images.
Height	Double	Gets the height of the raster image in pixels. This property's value is read-only.
ImageFile	String	Gets or sets the full path and file name of the raster image file. This property can be used to load a new raster image into an existing raster entity.
ImageHeight	Double	Gets or sets the height of the raster image in current units. For uniform stretching/shrinking of the image, use the ScaleFactor property.
ImageVisibility	Boolean	Specifies whether image visibility is on or off. Image redrawing speed is increased by turning off image visibility. Hidden images aren't displayed or plotted only the image boundary. Images can be hidden regardless of the current viewport coordinate system or when it isn't aligned with the current viewport coordinate system.
ImageWidth	Double	Gets or sets the width of the raster image in current units. For uniform stretching/shrinking of the image, use the ScaleFactor property.
Name	String	Gets or sets the name of the raster object, but not its path (see the ImageFile property).
Origin	Variant	Gets or sets the origin of the raster image in WCS coordinates, relative to the lower-left corner. Returned an array of Doubles.
Rotation	Double	Gets or sets the rotation angle (in radians) for the object, relative to the X-axis of the object's WCS with positive angles going counterclockwise when viewed down from the Z-axis toward the origin.
ScaleFactor	Double	Gets or sets the scale factor for the object. It has to be a value greater than 0. A scale factor greater than 1 enlarges the object. A scale factor between 0 and 1 shrinks the object. The initial value for this property is 1.0000.
ShowRotation	Boolean	Determines whether a raster image is displayed at the value of its Rotation property.

Name	Returns	Description
`Transparency`	Boolean	Specifies whether the transparency for a particular bitonal image is on or off. Attached bitonal images inherit current layer settings for background and foreground color. In addition to the modifications you can make to any attached image, you can modify bitonal images by turning the transparency of the background on and off. Note that bitonal images and bitonal image boundaries are always the same color.
`Width`	Double	Gets the width of the raster image in pixels. This property's value is read-only.

■ **Note** Because this object inherits from `AcadObject`, it supports the `Modified` event.

AcadRay Object

The `AcadRay` object represents a semi-infinite line and is commonly used as a construction line. A ray has a finite starting point and extends to infinity. The `AcadRay` object is created using the `AddRay` method of the `AcadBlock`, `AcadModelSpace`, or `AcadPaperSpace` object.

AcadRay Object Methods

The `AcadRay` object inherits all the methods of the `AcadEntity` and `AcadObject` objects. It supports no other methods.

AcadRay Object Properties

The `AcadRay` object inherits all the properties of the `AcadEntity` and `AcadObject` objects, as well as the common `Application` property. It also supports the following properties.

Name	Returns	Description
`BasePoint`	Variant	Gets or sets the point through which the ray passes. It returns a set of 3-D coordinates as a three-element array of Doubles.
`DirectionVector`	Variant	Gets or sets the direction for the ray through a vector. It returns a three-element array of Doubles.
`SecondPoint`	Variant	Gets or sets the second point of the ray. It returns a set of 3-D WCS coordinates as a three-element array of Doubles. Note that you can't set the second point of a ray equal to its base point.

■ **Note** Because this object inherits from `AcadObject`, it supports the `Modified` event.

AcadRegion Object

The AcadRegion object represents a bounded planar face consisting of lines, circles, arcs, elliptical arcs, and spline curves. All objects in the region retain their layer, linetype, and color. AutoCAD deletes the original objects after converting them to regions and doesn't hatch the regions by default. The AcadRegion object is created using the AddRegion method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object. Regions are mostly used as loops for AcadHatch objects to calculate the area of a composite figure or to find its Centroid and other physical characteristics.

AcadRegion Object Methods

The AcadRegion object inherits all the methods of the AcadEntity and AcadObject objects. It also supports the following methods.

Name	Returns	Description
Boolean		Performs a Boolean operation against another region, allowing you to create composite regions from the intersection, union, or subtraction of one region from another. Parameters: Operation As AcBooleanType and Object As AcadRegion. For a list of AcBooleanType constants, see Appendix B.
Explode	Variant	Explodes the region and returns the subentities as an array of Objects. The individual loops of the regions are converted to lines, circular or elliptical arcs, or spline curves.

AcadRegion Object Properties

The AcadRegion object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Area	Double	Gets the enclosed area of a region, which equals the combined area for all the objects in the region. The area of the object is specified in square drawing units. This property's value is read-only.
Centroid	Variant	Gets the center of area or mass for a region or solid as a 2-D coordinate. It returns as a two-element array of Doubles. This property's value is read-only.
MomentOfInertia	Variant	Gets the moment of inertia for the region as a 3-D coordinate. It returns as a three-element array of Doubles. This property's value is read-only.
Normal	Variant	Gets the 3-D (Z-axis) normal unit vector for the region. This property's value is read-only.
Perimeter	Double	Gets the total length of the inner and outer region loops in drawing units. This property's value is read-only.
PrincipalDirections	Variant	Gets the principal directions of the region as X, Y, and Z coordinates calculated on the current coordinate system. This property's value is read-only.
PrincipalMoments	Variant	Gets the principal moments property of the region as X, Y, and Z coordinates calculated on the current coordinate system. This property's value is read-only.

Name	Returns	Description
ProductOfInertia	Double	Gets the product of inertia of the region, as X, Y, and Z coordinates calculated on the current coordinate system. This property's value is read-only.
RadiiOfGyration	Variant	Gets the radius of gyration of the region, as X, Y, and Z coordinates calculated on the current coordinate system. This property's value is read-only.

■**Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadRegisteredApplication Object

The AcadRegisteredApplication object represents an external application that has been added to the drawing. To be recognized by AutoCAD, an application must register the name or names that it uses. This happens only once per drawing. These application names are then saved along with the extended data (XData) of each entity that uses them. Every instance of extended data referenced in a drawing must have its application registered in the drawing. Each AcadRegisteredApplication object is created using the Add method of the AcadRegisteredApplications collection and can be accessed using the Item method of the AcadRegisteredApplications collection.

AcadRegisteredApplication Object Methods

The AcadRegisteredApplication object inherits all the methods of the AcadObject object. It supports no other methods.

AcadRegisteredApplication Object Properties

The AcadRegisteredApplication object inherits all the properties of the AcadObject object, as well as the common Application property. It also supports the following property.

Name	Returns	Description
Name	String	Gets or sets the name of the registered application

■**Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadRegisteredApplications Collection

The AcadRegisteredApplications collection contains a collection of all registered applications in the drawing. To add a new member to the collection, use the Add method. To select a specific registered application, use the Item method. Although this collection inherits a Delete method, you can't actually delete this collection. If you need to delete a specific registered application, use the `Delete` method found in the `AcadRegisteredApplication` object. There is no limit to the number of registered applications you can create in your drawing. However, there can be only one instance of the `AcadRegisteredApplications` collection, which is predefined for each drawing. You can make multiple references to it by using the `RegisteredApplications` property.

`AcadRegisteredApplications` Collection Methods

The `AcadRegisteredApplications` collection inherits all the methods of the `AcadObject` object. It also supports the following methods.

Name	Returns	Description
`Add`		Creates a member object and adds it to the collection. Parameter: `Name As String`.
`Item`		Gets the member object at a given index in a collection. Parameter: `Index As Variant` (an Integer or a String). If the `Index` value is a String, it must match an existing object name in the collection.

`AcadRegisteredApplications` Collection Properties

The `AcadRegisteredApplications` collection inherits all the properties of the `AcadObject` object, the `Count` property, and the common `Application` property. It supports no other properties.

■ **Note** Because this collection inherits from `AcadObject`, it supports the `Modified` event.

`AcadSelectionSet` Object

The `AcadSelectionSet` object represents a group of one or more AutoCAD objects specified for processing as a single unit. It is created by the `Add` method of the `AcadSelectionSets` collections and can be accessed by the `Item` method of the `AcadSelectionSets` collection. Four methods can be used to remove selection sets or their contents: `Clear`, `RemoveItems`, `Erase`, and `Delete`.

`AcadSelectionSet` Object Methods

The `AcadSelectionSet` object supports the following methods. Note that it does not inherit from `AcadObject`.

Name	Returns	Description
`AddItems`		Adds an object or objects to the selection set. Parameter: `Items As Variant`. The parameter must be an array containing a unique or multiple objects.
`Clear`		The `Clear` method removes all the items from the selection set. The objects still exist, but they no longer reside in the selection set.

Name	Returns	Description
Delete		The Delete method deletes a selection set object, but not the objects that were contained within it. The key (Name) is removed from the collection and is available for further use.
Erase		The Erase method deletes all items in a selection set. The selection set still exists but the objects removed do not.
Highlight		Sets the highlight status for all objects in the selection set. Once the highlight flag for an object has been set, a call to the Update or Regen method is required to view the change. Parameter: HighlightFlag As Boolean. The parameter is True if the object is highlighted and False otherwise.
Item	Object	Gets the member of the selection set specified by the index. Parameter: Index As Variant (Integer or String). If the Index value is a String, it must match an existing object name in the collection.
RemoveItems		The RemoveItems method removes one or more items from a selection set. These objects still exist but they no longer reside in the selection set. Parameter: Objects As Variant. The parameter must be an array containing a unique or multiple objects.
Select		Adds objects to the selection set. Parameters: Mode As AcSelect, [Point1 As Variant, Point2 As Variant], and [FilterType As Integer, FilterData As Variant]. For a list of possible values for the AcSelect enumeration (the Mode parameter), see Appendix B. The second and third parameters are three-element arrays of Doubles representing 3-D WCS coordinates. For this method and the three that follow, FilterType is a DXF group code specifying the type of filter to use. FilterData is the value to filter on. All four Select methods support the filtering mechanism.
SelectAtPoint		Adds an object containing a point to the selection set. Parameters: Point As Variant and [FilterType As Integer, FilterData As Variant]. Point is a three-element array of Doubles representing 3-D WCS coordinates.
SelectByPolygon		Adds any objects inside a polygon to the selection set. Parameters: Mode As AcSelect, PointsList As Variant, and [FilterType As Integer, FilterData As Variant]. For a list of possible values for the AcSelect enumeration (the Mode parameter), see Appendix B. PointsList is a three-element array of Doubles representing 3-D WCS coordinates.
SelectOnScreen		Prompts the user to select an object from the screen. Parameters: [FilterType As Integer, FilterData As Variant].
Update		Updates the selection set.

AcadSelectionSet Object Properties

As well as the common Application property, the AcadSelectionSet object supports the following properties.

Name	Returns	Description
Count	Integer	Gets the number of items in the selection set.
Name	String	Gets name of the selection set. This property's value is read-only.

AcadSelectionSets Collection

The AcadSelectionSets collection contains all the selection sets in the drawing. To add a new member to the collection, use the Add method. To select a specific selection set, use the Item method. To delete a specific selection set, use the Delete method of the AcadSelectionSet object. AutoCAD allows you up to 128 selection sets objects. However, there can be only one instance of the AcadSelectionSets collection, which is predefined for each drawing. You can make multiple references to it by using the SelectionSets property of the AcadDocument object.

AcadSelectionSets Collection Methods

The AcadSelectionSets collection supports just two methods. Note that it does not inherit from AcadObject.

Name	Returns	Description
Add	AcadSelectionSet	Creates an AcadSelectionSet object and adds it to the collection. Parameter: Name As String.
Item	AcadSelectionSet	Returns the member of the collection specified by the index. Parameter: Index As Variant (Integer or String). If the Index value is a String, it must match an existing object name in the collection.

AcadSelectionSets Collection Properties

The AcadSelectionSets collection supports the Application property and the Count property but no others.

AcadShape Object

The AcadShape object represents an object comprising lines, arcs, and circles defined in an .shx file. Note that before inserting a shape, you must load the file containing the desired shape file by using the LoadShapeFile method. The AcadShape object is created using the AddShape method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadShape Object Methods

The AcadShape object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadShape Object Properties

The AcadShape object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Height	Double	Sets the height of the shape object in drawing units.
InsertionPoint	Variant	Gets or sets the insertion point for a shape as a set of 3-D WCS coordinates. It returns a three-element array of Doubles.
Name	String	Gets or sets the name of the object.

Name	Returns	Description
`Normal`	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the shape object.
`ObliqueAngle`	Double	Gets or sets the oblique angle of the attribute and is measured from the vertical axis. The units are radians within the range of –85 to +85 degrees. A positive value denotes a lean toward the right.
`Rotation`	Double	Gets or sets the rotation angle (in radians) for the shape, relative to the X-axis with positive values going counterclockwise when viewed along the Z-axis toward the origin.
`ScaleFactor`	Double	Gets or sets the scale factor for the object. It has to be a value greater than 0.0. A scale factor greater than 1 enlarges the object. A scale factor between 0 and 1 shrinks the object. The initial value for this property is 1.0000.
`Thickness`	Double	Gets or sets the distance the `AcadShape` object is extruded above or below its elevation, in a Z-axis direction. The default is 0.0.

Note Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadSolid` Object

The `AcadSolid` object represents a solid-filled polygon. Solid objects are filled only when the `FILLMODE` system variable is on, which can be set or queried by the `SetVariable` and `GetVariable` methods. The `AcadSolid` object is created using the `AddSolid` method of the `AcadBlock`, `AcadModelSpace`, or `AcadPaperSpace` object.

`AcadSolid` Object Methods

The `AcadSolid` object inherits all the methods of the `AcadEntity` and `AcadObject` objects. It supports no other methods.

`AcadSolid` Object Properties

The `AcadSolid` object inherits all the properties of the `AcadEntity` and `AcadObject` objects, as well as the common `Application` property. It also supports the following properties.

Name	Returns	Description
`Coordinate`	Variant	Gets or sets the coordinate of a single vertex in the object. This will replace an existing vertex for the specified object. Use standard array-handling techniques to process the values contained in this property. It returns a three-element array of Doubles containing 3-D coordinates in WCS. Note that the Z coordinate will default to 0 on the active UCS. Parameter: `Index As Integer` (the index in the zero-based array of vertices for the vertex you want to set or query).

Continued

Name	Returns	Description
`Coordinates`	Variant	Gets or sets the coordinates for each vertex in the object. This will replace any existing coordinates for the specified object. Use standard array-handling techniques to process the coordinates contained in this property. Note that you can't change the number of coordinates in the object by using this property, you can only change the location of existing coordinates. It returns a three-element array of Doubles containing 3-D coordinates in WCS.
`Normal`	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the solid object.
`Thickness`	Double	Gets or sets the distance the `AcadSolidobject` is extruded above or below its elevation, in a Z-axis direction. The default is 0.0.

■**Note** Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadSortEntsTable` Object

This object contains and manipulates draw order information.

`AcadSortEntsTable` Methods

The `AcadSortEntsTable` object does not support any methods.

`AcadSortEntsTable` Properties

The `AcadSortEntsTable` object inherits no properties. However, it does support the following properties.

Name	Returns	Description
`Block`	`AcadBlock`	Returns the block of a `SortentsTable` object.
`GetFullDrawOrder`		Returns all of the objects in a block, sorted by draw order with the bottom object first. Parameters: `Objects As Variant`, `HonorSortentsSysVar As Boolean`. If `True`, uses the setting in the `DRAWORDERCTL` system variable.
`GetRelativeDrawOrder`		Returns the objects specified, sorted by the draw order, with the bottom object first. Parameters: `Objects As Variant`, `HonorSortentsSysVar As Boolean`. If `True`, uses the setting in the `DRAWORDERCTL` system variable.
`MoveAbove`		Moves objects above the target in the draw order. Parameters: `Objects As Variant`, `Target As AcadEntity`.
`MoveBelow`		Moves objects below the target in the draw order. Parameters: `Objects As Variant`, `Target As AcadEntity`.
`MoveToBottom`		Moves objects to the bottom of the draw order. Parameters: `Objects As Variant`.
`MoveToTop`		Moves objects to the top of the draw order. Parameters: `Objects As Variant`.

Name	Returns	Description
`SetRelativeDrawOrder`		Sets the relative draw order specified, with the bottom objects first. Parameters: `Objects As Variant`.
`SwapOrder`		Swaps the draw order positions of two objects. Parameters: `Object1 As AcadEntity`, `Object2 As AcadEntity`.

`AcadSpline` Object

A spline is a smooth curve passing through a given set of points. AutoCAD uses a particular type of spline known as a nonuniform rational B-spline (NURBS) curve, which produces a smooth curve between control points. Splines are useful for creating irregularly shaped curves, for example, drawing contour lines for geographic information system applications or automobile design. Splines are created by specifying coordinate points and can form a closed loop if the start and end points and tangents are coincident. You can also alter the spline-fitting tolerance. Fit tolerance refers to how closely the spline fits the set of given coordinates. The lower the tolerance, the more closely the spline fits the points. Zero tolerance means that the spline passes through all the points.

The `AcadSpline` object represents a NURBS curve and is created using the `AddSpline` method of the `AcadBlock`, `AcadModelSpace`, or `AcadPaperSpace` object.

`AcadSpline` Object Methods

The `AcadSpline` object inherits all the methods of the `AcadEntity` and `AcadObject` objects. It also supports the following methods.

Name	Returns	Description
`AddFitPoint`		Adds the fit point (a set of 3-D WCS coordinates) to the spline at a given index. If the index is negative, then the point is added to the beginning of the spline. If the index exceeds the maximum value of the index, then the point is added to the end of the spline. The fit point is added to the spline and is refitted through the new set of points. The changes can be viewed only if the drawing is regenerated. Parameters: `Index As Integer` and `FitPoint As Variant`.
`DeleteFitPoint`		Deletes the fit point of the spline at a given index, which must be a positive integer between 0 and N minus 1, where N is the total number of fit points for the spline. The fit point is removed, and the spline refitted through the remaining points. There must be at least three fit points in the spline for this method to succeed. The changes can be viewed only if the drawing is regenerated. Parameter: `Index As Integer`.
`ElevateOrder`		Elevates the order of the spline. The range of values is one above the current order (which is the value of the `Degree` property plus 1) to a maximum of 26. When a spline is elevated, it is converted from an interpolated (fit) spline to a control point spline. Note that, once elevated, the spline order can't be reduced. The `StartTangent` and `EndTangent` properties are no longer used or accessible, the spline being editable by the mean of its control points. Parameter: `Order As Integer`.

Continued

Name	Returns	Description
GetControlPoint	Variant	Returns the 3-D WCS coordinates of the control point of the spline at a given index as an array of Doubles. Control points fine-tune a spline definition by adding weight to a portion of the spline curve. Parameter: Index As Integer.
GetFitPoint	Variant	Returns the 3-D WCS coordinates of the fit point of the spline at a given index as an array of Doubles. Parameter: Index As Integer.
GetWeight	Double	Returns the weight of the spline at a given control point index. Parameter: Index As Integer.
Offset	Variant	Creates a new spline by offsetting the current spline by a specified distance, which must be nonzero. If the offset is negative, this means that the spline is drawn closer to the WCS origin. Parameter: Distance As Double.
PurgeFitData		Purges the fit data of the spline.
Reverse		Reverses the direction of the spline.
SetControlPoint		Sets the indexed control point of the spline at a specified point. Parameters: Index As Integer and ControlPoint As Variant.
SetFitPoint		Sets the indexed fit point of the spline at a specified point. Parameters: Index As Integer and FitPoint As Variant.
SetWeight		Sets the weight of the spline for a given control point index. Parameters: Index As Integer and Weight As Double.

AcadSpline Object Properties

The AcadSpline object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Area	Double	Gets the enclosed area of the spline in square drawing units. The area is computed as though a straight line connects the start and end points. This property's value is read-only.
Closed	Boolean	Determines whether the spline is closed. When you close a spline, you make it tangent-continuous at both ends. If the spline's start and end points are already the same, closing the spline makes it tangent-continuous at both points. When you open a closed spline, if the start and end points were the same before the close, this property returns the spline to its original state. The start and end points remain the same but lose their tangent continuity. If the spline's start and end points were not the same before the close, opening the spline returns it to its original state and removes tangent continuity. This property's value is read-only.
ControlPoints	Variant	Gets or sets the control points of a spline storing the 3-D WCS coordinates as an array of Doubles. Control points fine-tune a spline definition by adding weight or density to a portion of the spline curve.

Name	Returns	Description
`Degree`	Integer	Gets the degree of the spline's polynomial representation. The return value must be in the range 1 to 25. This property's value is read-only. To set a higher value for the degree, use the `ElevateOrder` method. The higher the degree is, the higher the control points are.
`EndTangent`	Variant	Gets or sets the end tangent of the spline as a directional vector. This property is inaccessible to control points.
`FitPoints`	Variant	Gets or sets the fit points of a spline, which define the path of the spline.
`FitTolerance`	Double	Refits the spline to the existing points with new tolerance values. If this value is 0, the spline passes through all fit points. A value greater than zero allows the curve to pass through the fit points within the specified tolerance.
`IsPeriodic`	Boolean	Determines whether the given spline is periodic. This property's value is read-only.
`IsPlanar`	Boolean	Determines whether the spline is planar. This property's value is read-only.
`IsRational`	Boolean	Determines whether the given spline is rational. This property's value is read-only.
`Knots`	Variant	Gets the knot vector for a spline.
`NumberOfControlPoints`	Integer	Gets the number of control points of the spline. This property's value is read-only.
`NumberOfFitPoints`	Integer	Gets the number of fit points of the spline. This property's value is read-only.
`StartTangent`	Variant	Gets or sets the start tangent for the spline as a directional vector. This property is inaccessible to control points for an elevated spline.
`Weights`	Variant	Gets the weight vector for spline.

Note Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadState` Object

The `AcadState` object is a special object for use in monitoring the state of AutoCAD from out-of-process applications. It is a transient object that is returned from the `GetAcadState` method of the `AcadApplication` object and is used to check for AutoCAD quiescence from out-of-process applications. This object has no methods.

`AcadState` Object Properties

Other than the common `Application` property, the `AcadState` object supports just one property.

Name	Returns	Description
`IsQuiescent`	Boolean	Returns `True` if AutoCAD is idle and `False` if AutoCAD is busy. This property's value is read-only.

AcadSummaryInfo Object

This object accesses drawing properties such as the `Title`, `Subject`, `Author`, and `Keywords` properties.

AcadSummaryInfo Methods

The `AcadSummaryInfo` object does not inherit any methods. However, it does support the following methods.

Name	Returns	Description
`AddCustomInfo`		Adds a custom property field at the end of the existing list of fields. Parameters: `Key As String`, `Value As String`.
`GetCustomByIndex`	String	Returns the custom property name (key) and value that correspond to an index value. Parameters: `Index As Long`, `Key As String`, `Value As String`.
`GetCustomByKey`	String	Returns the custom property value that corresponds to a name (key) value. Parameters: `Key As String`, `Value As String`.
`NumCustomInfo`	Long	Returns the number of custom property fields that have been set.
`RemoveCustomByIndex`		Deletes the custom property name (key) and value that correspond to an index value. Parameter: `Index As Long`.
`RemoveCustomByKey`		Deletes the custom property value that corresponds to a name (key) value. Parameter: `Key As String`.
`SetCustomByIndex`		Sets the custom property name (key) and value that correspond to an index value. Parameters: `Index As Long`, `Key As String`, `Value As String`.
`SetCustomByKey`		Sets the custom property value that corresponds to a name (key) value. Parameters: `Key As String`, `Value As String`.

AcadSummaryInfo Properties

The `AcadSummaryInfo` object inherits no properties. However, it does support the following properties.

Name	Returns	Description
`Author`	String	Gets or sets the author of the drawing.
`Comments`	String	Gets or sets the comments you want to use to locate the drawing.
`HyperlinkBase`	String	Gets or sets the base address used for all relative links inserted within the drawing. An Internet address or path to a folder on a network drive may be specified as well.
`Keywords`	String	Gets or sets the keywords you want to use to locate the drawing.
`LastSavedBy`	String	Gets or sets the name of the last person who modified the file.
`RevisionNumber`	String	Gets or sets the drawing revision number.
`Subject`	String	Gets or sets the subject of the drawing.
`Title`	String	Gets or sets the title used to locate the drawing.

`AcadTable` Object

A table contains data in rows and columns. You can use the `Table` object to add and modify tables in a drawing.

To add a table to a drawing, use the `AddTable` method. After you add a table, you can modify the number of columns, the number of rows, and other settings.

All methods that change the `Table` object do the following:

1. Open the `Table` object in write mode.
2. Modify the `Table` object based on the input parameters.
3. Close the `Table` object, which recomputes the `Table`.

Recomputing large tables consumes a lot of time and memory because the `Table` object is reconstructed from scratch. If you are making multiple calls to update a table, you can improve performance by setting `RegenerateTableSuppressed` before and after making the modifications.

`AcadTable` Methods

The `AcadTable` object inherits all the methods of the `AcadEntity` and `AcadObject` objects. It also supports the following methods.

Name	Returns	Description
`ClearSubSelection`		Removes a subselection set of cells from a table.
`ClearTableStyleOverrides`		Clears table style overrides. Parameter: `Flag As Long`. 0 deletes all table and cell overrides. 1 deletes all table overrides but retains cell overrides. 2 deletes all cell overrides but retains table overrides.
`DeleteCellContent`		Deletes the cell content in the specified row and column. Parameters: `Row As Long`, `Column As Long`.
`DeleteColumns`		Deletes columns from a table. Parameters: `Row As Long`, `Column As Long`.
`DeleteRows`		Deletes rows from a table. Parameters: `Row As Long`, `Rows As Long`.
`GenerateLayout`		Generates the layout of a table. This method updates the `Table` object according to the current table style. This method computes the table geometry, including gridlines, text content, block content, and background fills.
`GetAlignment`	`AcRowType`	Returns the cell alignment for a row type. For a list of the values for the `AcRowType` enumeration, see Appendix B.
`GetAttachmentPoint`		Sets the attachment point for the specified row and column. Parameters: `Row As Long`, `Column As Long`.
`GetAutoScale`	Boolean	Returns the value specifying whether auto scale is used in the specified row and column. Parameters: `Row As Long`, `Column As Long`.
`GetBackgroundColor`	`AcCmColor`	Returns the background color value for the specified row type. Parameter: `RowType As AcRowType`. For a list of possible values for the `AcRowType` enumeration, see Appendix B.

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Name	Returns	Description
GetBackgroundColorNone	Boolean	Returns the value specifying that there is no background color for the specified row type. For a list of the values for the AcRowType enumeration, see Appendix B.
GetBlockAttributeValue	String	Returns the attribute value of the specified cell for the attribute definition object contained in the block. Parameters: Row As Long, Column As Long, AttDefId As Long.
GetBlockRotation	Double	Returns the block rotation angle for the specified row and column. Parameters: Row As Long, Column As Long.
GetBlockScale	Double	Returns the block scale value for the specified row and column. Parameters: Row As Long, Column As Long.
GetBlockTableRecordId	Long	Returns the block table record ID of the cell. Parameters: Row As Long, Column As Long.
GetCellAlignment	AcCellAlignment	Returns the alignment for the cell in the specified row and column. Parameters: Row As Long, Column As Long. For a list of possible values for the AcCellAlignment enumeration, see Appendix B.
GetCellBackgroundColor	AcCmColor	Returns the background true color value of a cell in the specified row and column. Parameters: Row As Long, Column As Long. For a list of possible values for the AcCmColor enumeration, see Appendix B.
GetCellBackgroundColorNone	Boolean	Returns the value specifying that there is no background color for the specified row and column. Parameters: Row As Long, Column As Long.
GetCellContentColor	AcCmColor	Returns the true color value for the content of the specified row and column. Parameters: Row As Long, Column As Long. For a list of possible values for the AcCmColor enumeration, see Appendix B.
GetCellExtents		Returns the cell extents for the specified row and column. Parameters: Row As Long, Column As Long, OuterCell As Boolean. The specified cell is an outer cell when the OuterCell parameter is True, an inner cell when False.
GetCellGridColor	AcCmColor	Returns the grid color value for an edge of the specified row and column. Parameters: Row As Long, Column As Long, Edge As AcCellEdgeMask. For a list of possible values for the AcCmColor and AcCellEdgeMask enumerations, see Appendix B.
GetCellGridLineWeight	Acad_LWeight	Returns the grid lineweight value for an edge of the specified row and column. Parameters: Row As Long, Column As Long, Edge As AcCellEdgeMask. For a list of possible values for the Acad_LWeight and AcCellEdgeMask enumerations, see Appendix B.
GetCellGridVisibility	Boolean	Returns the grid visibility value for an edge of the specified row and column. Parameters: Row As Long, Column As Long, Edge As AcCellEdgeMask. For a list of possible values for the AcCellEdgeMask enumeration, see Appendix B.
GetCellStyleOverrides		Returns the cell style overrides. Parameters: Row As Long, Column As Long.

Name	Returns	Description
GetCellTextHeight	Double	Returns the text height for the specified row and column. Parameters: Row As Long, Column As Long.
GetCellTextStyle	String	Returns the text style name for the specified row and column. Parameters: Row As Long, Column As Long.
GetCellType	AcCellType	Returns the cell type for the specified row and column. Parameters: Row As Long, Column As Long. For a list of possible values for the AcCellType eumeration, see Appendix B.
GetColumnWidth	Double	Returns the column width for the column at the specified column index in the table. Parameters: Column As Long.
GetContentColor	AcCmColor	Returns the true color value for the specified row type. Parameters: RowType As AcRowType. For a list of possible values for the AcCmColor and AcRowType enumerations, see Appendix B.
GetFieldId	Long	Returns the field object ID of the specified cell. Parameters: Row As Long, Column As Long.
GetGridColor	AcCmColor	Returns the grid color value for a grid linetype and row type. Parameters: GridLineType As AcGridLineType, RowType As AcRowType. For a list of possible values for the AcCmColor, AcGridLineType, and AcRowType enumerations, see Appendix B.
GetGridLineWeight	Acad_LWeight	Returns a grid lineweight value for a grid linetype and row type. Parameters: GridLineType As AcGridLineType, RowType As AcRowType. For a list of possible values for the Acad_LWeight, AcGridLineType, and AcRowType enumerations, see Appendix B.
GetGridVisibility	Boolean	Returns a grid visibility value for the specifed grid linetype and row type. Parameters: GridLineType As AcGridLineType, RowType As AcRowType. For a list of possible values for the AcGridLineType, and AcRowType enumerations, see Appendix B.
GetMinimumColumnWidth	Double	Gets the minimum column width for the column at the specified column index in the table. Parameter: Column As Long.
GetMinimumRowHeight	Double	Returns the minimum row height for the specified row. Parameter: Row As Long.
GetRowHeight	Double	Returns the row height for the row at the specified row index in the table. Parameter: Row As Long.
GetRowType	AcRowType	Returns the row type for the specified row. Parameter: Row As Long. For a list of possible values for the AcRowType enumeration, see Appendix B.
GetSubSelection		Returns the row and column indexes of the cells in a subselection set. This method gets the row and column indices of the cells in the subselection set. The total number of selected cells is equal to (rowMax – rowMin + 1) * (colMax - colMin + 1). Parameters: RowMin As Long, RowMax As Long, ColumnMin As Long, ColumnMax As Long.
GetText	String	Returns the text value for the specified row and column. Parameters: Row As Long, Column As Long.
GetTextHeight	Double	Returns the text height for the specified row type. Parameter: RowType As AcRowType. For a list of possible values for the AcRowType enumeration, see Appendix B.

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Name	Returns	Description
GetTextRotation	AcRotationAngle	Returns the text rotation for the specified row and column. Parameters: Row As Long, Column As Long. For a list of possible values for the AcRotationAngle enumeration, see Appendix B.
GetTextStyle	String	Returns the text style name for the specified row type. Parameter: RowType As AcRowType. For a list of possible values for the AcRowType enumeration, see Appendix B.
HitTest	Boolean	Returns the cell at the specified location. This function performs a hit test by specifying a point and viewing direction. The cell hit by the ray is returned. Parameters: InputPickPoint As Variant, ViewVector As Variant, ResultRowIndex As Long, ResultColumnIndex As Long.
InsertColumns		Inserts columns in a table. Parameters: StartColumn As Long, Width As Double, NumberOfColumns As Long.
InsertRows		Inserts rows in a table. Parameters: StartRow As Long, Height As Double, NumberOfRows As Long.
IsMergedCells	Boolean	Returns the merge status of a cell. Parameters: Row As Long, Column As Long, MinRow As Long, MaxRow As Long, MinColumn As Long, MaxColumn As Long.
MergedCells		Merges cells in a table. This method merges a rectangular region of cells. The total number of cells to be merged is equal to (maxRow - minRow + 1) * (maxCol - minCol + 1). Parameters: MinRow As Long, MaxRow As Long, MinColumn As Long, MaxColumn As Long.
RecomputeTableBlock		Updates a table block. This function updates the table block record that the Table object references to match changes made to the Table object since the most recent update of the table block record. Parameter: ForceUpdate As Boolean. An update will occur when the parameter value is True.
ReselectSubRegion		Returns subentities of a selection set in a table.
Select		Selects a cell in a table. This function selects a cell in the table by specifying a point, viewing direction, and orientation. Parameters: InputPickPoint As Variant, ViewVector As Variant, ViewOrientationVector As Variant, ApertureBoxWidth As Double, ApertureBoxHeight As Double, AllowOutside As Boolean, ResultRowIndex As Long, ResultColumnIndex As Long.
SelectSubRegion		Selects a set of cells in a table. Parameters: WindowPoint1 As Variant, WindowPoint2 As Variant, ViewVector As Variant, ViewOrientationVector As Variant, Type As AcSelectType, IncludeCurrentSelection As Boolean, MinRow As Long, MaxRow As Long, MinColumn As Long, MaxColumn As Long.
SetAlignment		Sets a cell alignment for the specified row types. Parameters: RowTypes As Long, CellAlignment As AcCellAlignment. For a list of possible values for the AcCellAlignment enumeration, see Appendix B.
SetAutoScale		Sets the value specifying whether auto scale is used in the specified row and column. Parameters: Row As Long, Column As Long, AutoScale As Boolean. Autoscale is used when True.

Name	Returns	Description
SetBackgroundColor		Sets a background color value for the specifed row types. Parameters: RowTypes As Long, Color As AcCmColor. For a list of possible values for the AcCmColor enumeration, see Appendix B.
SetBackgroundColorNone		Sets the value specifying whether there is no background color for the specified row types. Parameters: RowTypes As Long, Value As Boolean. No background color used when Value is True.
SetBlockAttributeValue		Sets the attribute value of the specified cell for the attribute definition object that is contained in the block. Parameters: Row As Long, Column As Long, AttDefId As Long, Attribute-Value As String.
SetBlockRotation		Sets the block rotation angle for the specified row and column. Parameters: Row As Long, Column As Long, Rotation As Double.
SetBlockScale		Sets the block scale value for the specified row and column. Parameters: Parameters: Row As Long, Column As Long, Scale As Double.
SetBlockTableRecordId		Sets the block table record ID of the cell. Parameters: Row As Long, Column As Long, BlockId As Long, AutoFit As Boolean.
SetCellAlignment		Sets the cell alignment for the specified row and column. Parameters: Row As Long, Column As Long, CellAlignment As AcCellAlignment. For a list of possible values for the AcCel-lAlignment enumeration, see Appendix B.
SetCellBackgroundColor		Sets the background true color value for the specified row and column. Parameters: Row As Long, Column As Long, Color As AcCmColor. For a list of possible values for the AcCmColor enumeration, see Appendix B.
SetCellBackgroundColorNone		Sets the value specifying that there is no background color for the specified row and column. Parameters: Row As Long, Column As Long, Value As Boolean. No background color used when Value is True.
SetCellContentColor		Sets the true color value for the content of the specified row and column. Parameters: Row As Long, Column As Long, Color As AcCmColor.
SetCellGridColor		Sets the grid color value for an edge of the specified row and column. Parameters: Row As Long, Column As Long, Color As AcCmColor.
SetCellGridLineWeight		Sets the grid lineweight value for an edge of the specified row and column. Parameters: Row As Long, Column As Long, Edges As Long, Lineweight As AcadLWeight. For a list of possible values for the AcadLWeight enumeration, see Appendix B.
SetCellGridVisibility		Sets the grid visibility value for an edge of the specified row and column. Parameters: GridLineTypes As Long, RowTypes As Long, Value As Boolean. Grid cell will be visible when value is True.
SetCellTextHeight		Sets the text height for the specified row and column. Parameters: Row As Long, Column As Long, TextHeight As Double.

Continued

Name	Returns	Description
SetCellTextStyle		Sets the text style name for the specified row and column. Parameters: Row As Long, Column As Long, TextStyle As String.
SetCellType		Sets the cell type for the specified row and column. Parameters: Row As Long, Column As Long, CellType As AcCellType. For a list of possible values for the AcCellType enumeration, see Appendix B.
SetColumnWidth		Sets the column width for the column at the specified column index in the table. Parameters: Column As Long, Width As Double.
SetContentColor		Sets the true color value for the specified row types. Parameters: RowTypes As Long, Color As AcCmColor. For a list of possible values for the AcCmColor enumeration, see Appendix B.
SetFieldId		Sets the field object ID of the specified cell. Parameters: Row As Long, Column As Long, FieldId As Long.
SetGridColor		Sets the grid color value for the specified grid linetypes and row types. Parameters: GridLineTypes As Long, RowTypes As Long, Color As AcCmColor. For a list of possible values for the AcCmColor enumeration, see Appendix B.
SetGridLineWeight		Sets the grid lineweight value for the specified grid linetypes and row types. . Parameters: GridLineTypes As Long, RowTypes As Long, Lineweight As AcadLWeight. For a list of possible values for the AcadLWeight enumeration, see Appendix B.
SetGridVisibility		Sets the grid visibility value for the specified grid linetype and row types. Parameters: GridLineTypes As Long, RowTypes As Long, Value As Boolean. Grid will be visible when value is True.
SetRowHeight		Sets the height for the row at the specified row index in the table. Parameters: Row As Long, Height As Double.
SetSubSelection		Sets the row and column indices of the cells in a subselection set. This method sets the row and column indices of the cells in the subselection set. The total number of selected cells is equal to (rowMax - rowMin + 1) * (colMax - colMin + 1). Parameters: RowMin As Long, RowMax As Long, ColumnMin As Long, ColumnMax As Long.
SetText		Sets the text value for the specified row and column. Row As Long, Column As Long, Text As String.
SetTextHeight		Sets the text height for the specified row types. Parameters: RowTypes As Long, Height As Double.
SetTextRotation		Sets the text rotation for the specified row and column. Parameters: Row As Long, Column As Long, Rotation As AcRotationAngle. For a list of possible values for the AcRotationAngle enumeration, see Appendix B.
SetTextStyle		Sets the text style name for the specified row types. Parameters: RowTypes As Long, TextStyle As String.
UnmergeCells		Splits merged cells in a table. Parameters: RowMin As Long, RowMax As Long, ColumnMin As Long, ColumnMax As Long.

`AcadTable` Properties

The `AcadTable` object inherits all the properties of the `AcadEntity` and `AcadObject` objects as well as the common `Application` property. It also supports the following properties.

Name	Returns	Description
`Columns`	Long	Gets or sets the number of columns in the table.
`ColumnWidth`	Double	Gets or sets the uniform width of all columns in the table.
`Direction`	Variant	Gets or sets the direction vector of the table. The vector (X-axis, in WCS coordinates) defines the horizontal plane containing the table.
`FlowDirection`	`AcTableDirection`	Gets or sets whether the title and header rows are at the bottom or the top of the table. For a list of possible values for the `AcTableDirection` enumeration, see Appendix B.
`HasSubSelection`	Boolean	Gets whether the table contains a subselection set. Read-only.
`HeaderSuppressed`	Boolean	Gets or sets whether the header of the table is suppressed.
`Height`	Double	Gets or sets the height of the table.
`HorzCellMargin`	Double	Gets or sets the value of the horizontal margin of cells.
`InsertionPoint`	Variant	Gets or sets insertion point for a table
`MinimumTableHeight`	Double	Gets the minimum height of the table. Read-only.
`MinimumTableWidth`	Double	Gets the minimum width of the table. Read-only.
`RegenerateTableSuppressed`	Boolean	Enables or disables regeneration of the table block. All methods that change the `Table` object do the following: open the `Table` object in write mode, modify the `Table` object based on the input parameters, close the `Table` object, which recomputes the `Table`. Recomputing large tables consumes a lot of time and memory because the `Table` object is reconstructed from scratch. To avoid performance problems, you should temporarily disable the regeneration of the `Table` object when modifying large tables through the VBA APIs and then apply the modifications and re-enable regeneration. To access this property, use the `IAcadTable2` interface.
`RowHeight`	Double	Gets or sets a uniform height for all rows in the table.
`Rows`	Long	Gets or sets the number of rows in the table.
`StyleName`	String	Gets or sets the name of the table style.
`TableStyleOverrides`	Variant	Gets the table style overrides. Read-only.
`TitleSuppressed`	Boolean	Gets or sets whether the title of the table is suppressed.
`VertCellMargin`	Double	Gets or sets the value of the vertical margin of cells.
`Width`	Double	Gets or sets the width of the table.

Note Because this object inherits from `AcadObject`, it supports the `Modified` event.

AcadTableStyle Object

You can use the TableStyle object to store table formatting, such as grid visibility, lineweight, and color. The object controls the initial formatting of a newly created Table object.

AcadTableStyle Methods

The AcadTableStyle object inherits all the methods of the AcadObject object. It also supports the following methods.

Name	Returns	Description
GetAlignment	AcRowType	Returns the cell alignment for a row type. For a list of the values for the AcRowType enumeration, see Appendix B.
GetBackgroundColor	AcCmColor	Returns the background color value for the specified row type. Parameter: RowType As AcRowType. For a list of possible values for the AcRowType enumeration, see Appendix B.
GetBackgroundColorNone	Boolean	Returns the value specifying that there is no background color for the specified row type. For a list of the values for the AcRowType enumeration, see Appendix B.
GetColor	AcCmColor	Returns the true color value for the specified row type. Parameter: RowType As AcRowType. For a list of the values for the AcCmColor and AcRowType enumerations, see Appendix B.
GetGridColor	AcCmColor	Returns the grid color value for a grid linetype and row type. Parameters: GridLineType As AcGridLineType, RowType As AcRowType. For a list of possible values for the AcCmColor, AcGridLineType, and AcRowType enumerations, see Appendix B.
GetGridLineWeight	Acad_LWeight	Returns a grid lineweight value for a grid linetype and row type. Parameters: GridLineType As AcGridLineType, RowType As AcRowType. For a list of possible values for the Acad_LWeight, AcGridLineType, and AcRowType enumerations, see Appendix B.
GetGridVisibility	Boolean	Returns a grid visibility value for the specifed grid linetype and row type. Parameters: GridLineType As AcGridLineType, RowType As AcRowType. For a list of possible values for the AcGridLineType, and AcRowType enumerations, see Appendix B.
GetTextHeight	Double	Returns the text height for the specified row type. Parameter: RowType As AcRowType. For a list of possible values for the AcRowType enumeration, see Appendix B.
GetTextStyle	String	Returns the text style name for the specified row type. Parameter: RowType As AcRowType. For a list of possible values for the AcRowType enumeration, see Appendix B.
SetAlignment		Sets a cell alignment for the specified row types. Parameters: RowTypes As Long, CellAlignment As AcCellAlignment. For a list of possible values for the AcCellAlignment enumeration, see Appendix B.
SetBackgroundColor		Sets a background color value for the specifed row types. Parameters: RowTypes As Long, Color As AcCmColor. For a list of possible values for the AcCmColor enumeration, see Appendix B.

Name	Returns	Description
SetBackgroundColorNone		Sets the value specifying whether there is no background color for the specified row types. Parameters: RowTypes As Long, Value As Boolean. No background color used when Value is True.
SetColor		Sets the true color value for the specified row types. Parameters: RowTypes As Long, Color As AcCmColor. For a list of possible values for the AcCmColor enumeration, see Appendix B.
SetGridColor		Sets the grid color value for the specified grid linetypes and row types. Parameters: GridLineTypes As Long, RowTypes As Long, Color As AcCmColor. For a list of possible values for the AcCmColor enumeration, see Appendix B.
SetGridLineWeight		Sets the grid lineweight value for the specified grid linetypes and row types. Parameters: GridLineTypes As Long, RowTypes As Long, Lineweight As AcadLWeight. For a list of possible values for the AcadLWeight enumeration, see Appendix B.
SetGridVisibility		Sets the grid visibility value for the specified grid linetype and row types. Parameters: GridLineTypes As Long, RowTypes As Long, Value As Boolean. Grid will be visible when value is True.
SetTextHeight		Sets the text height for the specified row types. Parameters: RowTypes As Long, Height As Double.
SetTextStyle		Sets the text style name for the specified row types. Parameters: RowTypes As Long, TextStyle As String.

AcadTableStyle Properties

The AcadTableStyle object inherits all the properties of the AcadObject objects as well as the common Application property. It also supports the following properties.

Name	Returns	Description
BitFlags	Long	Gets and sets the bit flag values of a table style.
Description	String	Gets and sets the description of an object.
FlowDirection	AcTableDirection	Gets or sets whether the title and header rows are at the bottom or the top of the table. For a list of possible values for the AcTableDirection enumeration, see Appendix B.
HeaderSuppressed	Boolean	Gets or sets whether the header of the table is suppressed.
HorzCellMargin	Double	Gets or sets the value of the horizontal margin of cells.
Name	String	Gets or sets the name of the table style.
TitleSuppressed	Boolean	Gets or sets whether the title of the table is suppressed.
VertCellMargin	Double	Gets or sets the value of the vertical margin of cells.

Note Because this object inherits from AcadObject, it supports the Modified event.

AcadText Object

The AcadText object represents a single line of alphanumeric characters. This object differs from the AcadMText object in that it creates only a single line of text, whereas the AcadMText object creates a paragraph of text. The AcadText object is created using the AddText method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadText Object Methods

The AcadText object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadText Object Properties

The AcadText object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Alignment	AcAlignment	Gets or sets both the vertical and horizontal alignments for the text. For a full list of the values of the AcAlignment enumerated type, see Appendix B. See also the InsertPoint and TextAlignmentPoint properties.
Backward	Boolean	Specifies the direction of text. True is for backward; False is for forward.
Height	Double	Use to set the height of uppercase text, measured in the current units. This property is used as a scale factor for both the height and width of the text.
HorizontalAlignment	AcHorizontalAlignment	Gets or sets the horizontal alignment for the text. For a full list of the values of the AcHorizontalAlignment enumerated type, see Appendix B. See also the InsertPoint and TextAlignmentPoint properties.
InsertionPoint	Variant	Gets or sets the insertion point for text. This property's value is read-only except for text whose Alignment property is set to acAlignmentLeft, acAlignmentAligned, or acAlignmentFit. To position text whose justification is other than left, aligned, or fit, use the TextAlignmentPoint property. Also, text with the HorizontalAlignment property set to acHorizontalAlignmentLeft, acHorizontalAlignmentAligned, or acHorizontalAlignmentFit uses this property to position the text. All other settings for HorizontalAlignment use the TextAlignmentPoint property.
Normal	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the text object.
ObliqueAngle	Double	Gets or sets the oblique angle of the attribute, and is measured from the vertical axis. The units are radians within the range of –85 to +85 degrees. A positive value denotes a lean toward the right.

Name	Returns	Description
`Rotation`	Double	Gets or sets the rotation angle (in radians) for the text object, relative to the X-axis with positive values going counterclockwise when viewed along the Z-axis toward the origin. The rotation angle is read-only for text whose `Alignment` property is set to `acAlignmentAligned` or `acAlignmentFit`.
`ScaleFactor`	Double	Gets or sets the scale factor for the object. It has to be a value greater than 0.0. A scale factor greater than 1 enlarges the object. A scale factor between 0 and 1 shrinks the object. The initial value for this property is 1.0000.
`StyleName`	String	Gets or sets the name of the style used with the object, the default being the current style. Use the `TextStyle` object to change the attribute of a given text style. Note that the name given must already be defined in the drawing.
`TextAlignmentPoint`	Variant	Gets or sets the alignment point for text, returning a three-element array of Doubles. If the `Alignment` property is set to `acAlignmentLeft`, this property will be reset to (0,0,0) and will become read-only. Also, you can't use this property to position text with the `HorizontalAlignment` property set to `acHorizontalAlignmentLeft`. In both cases, use the `InsertPoint` property.
`TextGenerationFlag`		Gets or sets the attribute text generation flag. Possible values: `acTextFlagBackward` and `acTextFlagUpsideDown`.
`TextString`	String	Gets or sets the text string for the entity.
`Thickness`	Double	Gets or sets the distance the `AcadText` object is extruded above or below its elevation, in a Z-axis direction. The default is 0.0.
`UpsideDown`	Boolean	Specifies the direction of text. `True` is for upside down; `False` is for the right way up.
`VerticalAlignment`	`AcVerticalAlignment`	Gets or sets vertical alignment for the text. For a list of the values of the `AcVerticalAlignment` enumerated type, see Appendix B.

Note Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadTextStyle` Object

The `AcadTextStyle` object represents a named, saved collection of settings that determine the appearance of text characters. The `ActiveTextStyle` property sets the active text style, which is used to format new text created in the drawing and also existing text that has no individual text style specified. If changes are made to the active text style, the new `AcadTextStyle` object must be reset as the active text style, and the drawing must be regenerated for the changes to appear. To specify a distinct individual text style for an object so that it won't change along with the active text style, use the StyleName property for that particular object.

AcadTextStyle Object Methods

The AcadTextStyle object inherits all the methods of the AcadObject object. It also supports the following methods.

Name	Description
GetFont	Gets the definition data of the font for the text style. Parameters: TypeFace As String, Bold As Boolean, Italic As Boolean, Charset As Integer, and PitchAndFamily As Integer. The TypeFace parameter specifies the font name of the text style. The Bold parameter is set to True for text to be bold. The Italic parameter is set to True for text to be italic. The Charset parameter specifies the character set for the font. The default value is 1. The PitchAndFamily parameter combines several constants (by a Boolean OR operation) to give the pitch and family values for the font. The default value is 0. A full list of constants can be found in the win32api.txt file supplied with the Visual Basic IDE.
SetFont	Sets the definition data of the font for the text style. Parameters: TypeFace As String, Bold As Boolean, Italic As Boolean, Charset As Integer, and PitchAndFamily As Integer.

AcadTextStyle Object Properties

The AcadTextStyle object inherits all the properties of the AcadObject object, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
BigFontFile	String	Gets or sets the name of the Asian language big font file associated with the text or attribute. The only valid file type is .shx, and the property can't be set to NULL or contain an empty string.
FontFile	String	Gets or sets the primary font file path and name. Fonts define the shapes of the text characters that make up each character set. Note that once this property has been set, you have to regenerate the drawing to see the changes to the text.
Height	Double	Sets the height of the text in current units.
LastHeight	Double	Gets or sets the last text height value used.
Name	String	Gets the name of the object. This property's value is read-only.
ObliqueAngle	Double	Gets or sets the oblique angle of the attribute and is measured from the vertical axis. The units are radians within the range of –85 to +85 degrees. A positive value denotes a lean toward the right.
TextGenerationFlag	Integer	Gets or sets the attribute text generation flag. Possible values are acTextFlagBackward and acTextFlagUpsideDown.
Width	Double	Gets or sets the width of the text boundary and sets the character spacing. Entering a value of less than 1.0 condenses the text. Entering a value of greater than 1.0 expands it. The maximum value is 100.

Note Because this object inherits from `AcadObject`, it supports the `Modified` event.

`AcadTextStyles` Collection

The `AcadTextStyles` collection contains all the text styles in the drawing. To add a new member to the collection, use the `Add` method. To select a specific text style, use the `Item` method. Although this collection inherits a `Delete` method, you can't actually delete this collection. If you need to delete a specific text style, use the `Delete` method found in the `AcadTextStyle` object. There is no limit to the number of text styles you can create in your drawing. However, there can be only one instance of the `AcadTextStyles` collection, which is predefined for each drawing. You can make multiple references to it by using the `TextStyles` property.

`AcadTextStyles` Collection Methods

The `AcadTextStyles` collection inherits all the methods of the `AcadObject` object. It also supports the following methods.

Name	Returns	Description
`Add`	`AcadTextStyle`	Creates a member object and adds it to the collection. Parameter: `Name As String`.
`Item`	`AcadTextStyle`	Gets the member object at a given index in a collection. Parameter: `Index As Variant` (an Integer or a String). If the `Index` value is a String, it must match an existing text style definition.

`AcadTextStyles` Collection Properties

The `AcadTextStyles` collection inherits all the properties of the `AcadObject` object, the `Count` property, and the common `Application` property. It supports no other properties.

Note Because this collection inherits from `AcadObject`, it supports the `Modified` event.

`AcadTolerance` Object

The `AcadTolerance` object represents a geometric tolerance contained in a feature control frame. Tolerances are influenced by several system variables, which can be set and queried using the `SetVariable` and `GetVariable` methods. The color of the feature control frame is controlled by `DIMCLRD`. The color of the tolerance text is controlled by `DIMCLRT`. The gap between the feature control frame and the text is controlled by `DIMGAP`. The size of the tolerance text is controlled by `DIMTXT`. Finally, the style of the tolerance text is controlled by `DIMTXTSTY`.

The `AcadTolerance` object is created using the `AddTolerance` method of the `AcadBlock`, `AcadModelSpace`, or `AcadPaperSpace` objects.

AcadTolerance Object Methods

The AcadTolerance object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadTolerance Object Properties

The AcadTolerance object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Arrowhead1Type	AcDimArrowheadType	Gets or sets the type of arrowhead for the tolerance object. It overrides the value of the DIMBLK1 system variable. For the values of the AcDimArrowheadType enumerated type, see Appendix B.
DecimalSeparator	String	Gets or sets the character used as the decimal separator in tolerance values. The initial value for this property is the period (.), though any character will be accepted as a valid value for this property. It overrides the value of the DIMSEP system variable.
DimensionLineColor	AcColor	Gets or sets the color of the tolerance object. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in the default color) or acByLayer (where AutoCAD draws objects in the color specified for the layer). For a list of possible values for the AcColor enumerated type, see Appendix B. This property overrides the value of the DIMCLRD system variable for the given object.
DirectionVector	Variant	Gets or sets the direction for the tolerance through a vector. It returns a three-element array of Doubles.
FractionFormat	AcDimFractionType	Gets or sets the formats of fractional values. It overrides the value of the DIMFRAC system variable. Possible values are acHorizontal, acDiagonal, and acNotStacked.
InsertionPoint	Variant	Represents an insertion point in the tolerance object as a set of 3-D WCS coordinates. It returns a three-element array of Doubles.
Normal	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the tolerance object.
PrimaryUnitsPrecision	AcDimPrecision	Gets or sets the number of decimal places displayed for the primary units. For a list of possible values for the AcDimPrecision enumerated type, see Appendix B.
ScaleFactor	Double	Gets or sets the scale factor for the object. It has to be a value greater than 0.0. A scale factor greater than 1 enlarges the object. A scale factor between 0 and 1 shrinks the object. The initial value for this property is 1.0000. This property overrides the DIMSCALE system variable.
StyleName	String	Gets or sets the name of the style used with the object, the default being the current style. Use the TextStyle object to change the attribute of a given text style. Note that the name given must already be defined in the drawing.

Name	Returns	Description
TextColor	AcColor	Gets or sets the color of the tolerance text. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock (where AutoCAD draws objects in the default color), acByLayer (where AutoCAD draws objects in the color specified for the layer), and the AcColor enumerated type (see Appendix B). This property overrides the value of the DIMCLRD system variable for the given object.
TextHeight	Double	Gets or sets the height for the tolerance text, the value of which overrides the value of the DIMTXT system variable. This property is ignored if the current text style has a fixed text height. The initial value is 0.1800.
TextString	String	Gets or sets the text string for the entity.
TextStyle	String	Gets or sets the text style for the tolerance text, the value of which overrides the value of the DIMTXSTY system variable. The initial value is STANDARD.

Note Because this object inherits from AcadObject, it supports the Modified event.

AcadToolbar Object

The AcadToolbar object represents an AutoCAD toolbar, which provides access to frequently used or custom commands, settings, and macros. You can add tools to or remove tools from the AutoCAD default toolbars and reposition them. You can also create your own toolbars.

You create a new toolbar with the Add method. AddToolbarButton inserts a new button to the toolbar, and AddSeparator adds a separator. Use the ToolbarItem object to add functionality to the toolbar button or to create flyout toolbar buttons. You can change the position of toolbars by docking them along the top, bottom, or sides of the AutoCAD screen, or you can float toolbars anywhere on the screen.

AcadToolbar Object Methods

The AcadToolbar object supports the following methods.

Name	Returns	Description
AddToolbarButton	AcadToolbarItem	Adds an item to the toolbar immediately before the position specified by the index. To associate a toolbar to a flyout button, use the AttachToolbarToFlyout method on the returned AcadToolbarItem object. Note that to add a button to a toolbar, it has to be visible. Parameters: Index As Variant (Integer or String), Name As String, HelpString As String, Macro As String, and [FlyoutButton As Variant]. The Index parameter for this property, and the one that follows, must be between 0 and N minus 1 (if it is an Integer), where N is the number of objects in the pop-up menu. The new item will be added immediately before the specified index location. To add the new menu item to the end of a menu, set the index to be greater than N. If a String is specified and the indexed item doesn't exist, then the new menu item is added at the end of the menu. The Name must be composed of alphanumeric characters with no punctuation other than a dash (-) or an underscore (_). This will show up as the tooltip. The FlyoutButton parameter is a Boolean variable determining whether the new button is to be a flyout button (True) (False). False is the default value returned if this option is ignored.
AddSeparator	AcadToolbarItem	Adds a separator to the toolbar immediately before the position specified by the index. The first item in a menu can't be a separator. You can't add a separator immediately next to another separator. Parameter: Index As Variant (Integer or String).
Delete		Deletes an item from the toolbar. Note that the item must have no Index parameter.
Dock		Docks the toolbar to its owning frame window. Parameter: Side As AcToolbarDockStatus. For a list of possible values for the AcToolbarDockStatus enumeration, see Appendix B.
Float		Floats the toolbar. Parameters: Top As Integer, Left As Integer, and NumberFloatRows As Integer. The Top and Left parameters specify the pixel location for the top and left edges of the toolbar, respectively, and are screen coordinates. Get the position of the document window to float in the drawing limits. The NumberFloatRows parameter designates the number of rows to create in the horizontal toolbar. The buttons on the toolbar will be distributed automatically across the number of rows designated.
Item	AcadToolbarItem	Returns the item from the collection specified by the index. Parameter: Index As Variant (Integer or String). If the Index value is a String, it must match an existing object name in the collection.

AcadToolbar Object Properties

As well as the common Application property and the Count property, the AcadToolbar object supports the following properties.

Name	Returns	Description
`DockStatus`	`AcToolbarDockStatus`	Gets whether and where the toolbar is docked. This property's value is read-only. For a list of possible values for the `AcToolbarDockStatus` enumerated type, see Appendix B.
`FloatingRows`	Integer	Gets or sets the number of rows if the toolbar is floating.
`Height`	Integer	Gets the height of the toolbar. This property's value is read-only.
`HelpString`	String	Gets or sets the toolbar's help string.
`LargeButtons`	Boolean	Specifies whether the toolbar buttons are large. This property's value is read-only.
`Left`	Integer	Gets or sets the position in pixels of left edge of the toolbar from the left side of the screen.
`Name`	String	Gets or sets the toolbar's name.
`Parent`	`AcadToolbars`	Gets the object to which the toolbar object belongs. This property's value is read-only.
`TagString`	String	Gets the menu's tag string, which can consist of alphanumeric and underscore (_) characters. This string uniquely identifies the item within a given menu file. This string is automatically assigned when the object is created and is used internally by AutoCAD for toolbar and menu identification. This property's value is read-only.
`Top`	Integer	Gets or sets the position in pixels of the top edge of the toolbar from the top of the screen.
`Visible`	Boolean	Specifies whether the toolbar is visible.
`Width`	Double	Gets the width of the toolbar. This property's value is read-only.

`AcadToolbarItem` Object

The `AcadToolbarItem` object represents a single button or separator on an AutoCAD toolbar. A toolbar button can contain a macro to be executed when it is selected by the user, or it can contain a nested toolbar called a flyout. Separators can't contain macros or flyouts. The `Macro` property is used to add or change the button's associated macro.

`AcadToolbarItem` Object Methods

The `AcadToolbarItem` object supports the following methods.

Name	Description
`AttachToolbarToFlyout`	Attaches the toolbar to a toolbar button defined as a flyout. To create a new button as a flyout, you must first use the `AddToolbarButton` method of the `AcadToolbar` object with the `FlyoutButton` parameter set to `True`. Only then can you attach the toolbar to the flyout using this method. Parameters: `MenuGroupName As String` and `ToolbarName As String`.
`Delete`	Deletes a specified toolbar item. Note that you can only add or remove toolbar items when the toolbar is visible.

Continued

Name	Description
GetBitmaps	Gets the large and small bitmaps used as icons for the toolbar item. Parameters: SmallIconName As String and LargeIconName As String. The parameters require both the path and file name of the bitmaps.
SetBitmaps	Sets the large (24×22 pixels) and small (15×16 pixels) bitmaps used as icons for the toolbar item. Parameters: SmallIconName As String and LargeIconName As String. The parameters require both the path and file name of the bitmaps.

AcadToolbarItem Object Properties

As well as the common Application property, the AcadToolbarItem object supports the following properties.

Name	Returns	Description
Flyout	AcadToolbar	Gets the toolbar associated with the item, if it's a flyout toolbar item. This property's value is read-only.
HelpString	String	Gets or sets the help string for the menu item, which appears in the AutoCAD status line when a user highlights a menu item.
Index	Integer	Gets the index for the menu item. The first position in the index is 0. This property's value is read-only.
Macro	String	Gets or sets the macro associated with the toolbar item.
Name	String	Gets or sets the name of the toolbar item.
Parent	AcadToolbar	Gets the parent object of the toolbar item. This property's value is read-only.
TagString	String	Gets the menu's tag string, which can consist of alphanumeric and underscore (_) characters. This string uniquely identifies the item within a given menu file. This string is automatically assigned when the object is created and is used internally by AutoCAD for toolbar and menu identification.
Type	AcToolbarItemType	Gets the toolbar item's type. This property's value is read-only. Possible values for the AcToolbarItemType enumeration are acToolbarButton, acToolbarFlyout, and acToolbarControl.

AcadToolbars Collection

The AcadToolbars collection contains all the toolbars loaded in the current AutoCAD session, some or all of which may be currently displayed in AutoCAD. To load an existing toolbar into the current session, load the menu group that contains the toolbar using the Load method of the AcadMenuGroups collection. To display large buttons for all toolbars, set the LargeButtons property to True.

AcadToolbars Collection Methods

The AcadToolbars collection supports just two methods.

Name	Returns	Description
Add	AcadToolbar	Creates an AcadToolbar object and adds it to the collection. Parameter: Name As String.
Item	AcadToolbar	Returns the member of the collection specified by the index. Parameter: Index As Variant.

AcadToolbars Collection Properties

As well as the common Application property and the Count property, the AcadToolbars collection supports the following properties.

Name	Returns	Description
LargeButtons	Boolean	Specifies whether the displayed toolbar buttons are large.
ParentAcadMenuGroup		Gets the collection's parent object. This property's value is read-only.

AcadTrace Object

The AcadTrace object represents a 2-D solid line of specified width. The end points of a trace are always positioned on the centerline and are cut square. AutoCAD automatically calculates the correct bevels when connecting adjacent trace segments. Traces are solid-filled when the fill mode is on. Otherwise, only the outline of the trace appears. The fill mode uses the system variable FILLMODE, which is set using the SetVariable method and queried using the GetVariable method. The system variable TRACEWID stores the current width used for an AcadTrace object.

The AcadTrace object is created using the AddTrace method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadTrace Object Methods

The AcadTrace object inherits all the methods of the AcadEntity and AcadObject objects. It supports no other methods.

AcadTrace Object Properties

The AcadTrace object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
`Coordinate`	Variant	Gets or sets the coordinate of a single vertex in the object. This will replace an existing vertex for the specified object. Use standard array-handling techniques to process the values contained in this property. It returns a three-element array of Doubles containing 3-D coordinates in WCS. Note that the Z coordinate will default to 0 on the active UCS. Parameter: `Index As Integer` (the index in the zero-based array of vertices for the vertex you want to set or query).
`Coordinates`	Variant	Gets or sets the coordinates for each vertex in the object. This will replace any existing coordinates for the specified object. Use standard array-handling techniques to process the coordinates contained in this property. Note that you can't change the number of coordinates in the object by using this property, you can only change the location of existing coordinates. It returns a three-element array of Doubles containing 3-D coordinates in WCS. Note that the Z coordinate will always default to 0 on the active UCS.
`Normal`	Variant	Gets or sets the 3-D (Z-axis) normal unit vector for the trace object.
`Thickness`	Double	Gets or sets the distance the `AcadTrace` object is extruded above or below its elevation, in a Z-axis direction. The default is 0.0.

■ **Note** Because this object inherits from `AcadObject`, it supports the `Modified` event.

AcadUCS Object

The `AcadUCS` object represents a User-defined Coordinate System (UCS) that determines the orientation of the X-, Y-, and Z-axes in 3-D space. With this object you can change the location of the origin and the orientation of the XY plane and the Z-axis, and the resulting UCS definition can be orientated anywhere within the 3-D layout. Many UCSs can be defined as required for each drawing. The `GetUCSMatrix` method returns the transformation matrix for a given UCS. To turn on the UCS icon for a given viewport, use the `UCSIconOn` property of the `AcadPViewport` and `AcadViewport` objects. Note that all coordinates in ActiveX Automation are entered in the WCS.

The `AcadUCS` object is created by the `Add` method of the `AcadUCSs` collection and accessed by the `Item` method of the `AcadUCSs` collection. UCSs can also be activated using the `ActiveUCS` property of the `Document` object. If any changes are made to the active UCS, the new `UCS` object must be reset as the active UCS for the changes to appear.

AcadUCS Object Methods

The `AcadUCS` object inherits all the methods of the `AcadObject` object. It also supports the following method.

Name	Returns	Description
`GetUCSMatrix`	Variant	Gets the transformation matrix consisting of UCS coordinate system data. It returns a 4×4 array of Doubles. This matrix is the passed as a parameter to the `TransformBy` method of a drawing object to apply a given UCS to that object.

AcadUCS Object Properties

The AcadUCS object inherits all the properties of the AcadObject object and the common Application property. It also supports the following properties.

Name	Returns	Description
Name	String	Gets or sets the name of the UCS object.
Origin	Variant	Gets or sets the origin of the UCS object in WCS coordinates, relative to the lower-left corner. It returns an array of Doubles.
XVector	Variant	Gets or sets the X direction of the given UCS. It returns a three-element array of Doubles, which is stored in the UCSXDIR system variable. If the X vector value is changed on the active UCS, you must reset the active UCS to see the change.
YVector	Variant	Gets or sets the Y direction of the given UCS. It returns a three-element array of Doubles, which is stored in the UCSYDIR system variable. If the Y vector value is changed on the active UCS, you must reset the active UCS to see the change.

Note Because this object inherits from AcadObject, it supports the Modified event.

AcadUCSs Collection

The AcadUCSs collection contains all the UCS definitions in the drawing. To add a new member to the collection, use the Add method. To select a specific UCS definition, use the Item method. Although this collection inherits a Delete method, you can't actually delete this collection. If you need to delete a specific UCS definition, use the Delete method found in the AcadUCS object. There is no limit to the number of UCS definitions you can create in your drawing. However, there can be only one instance of the AcadUCSs collection, which is predefined for each drawing. You can make multiple references to it by using the UserCoordinateSystems property.

AcadUCSs Collection Methods

The AcadUCSs collection inherits all the methods of the AcadObject object. It also supports the following methods.

Name	Returns	Description
Add	AcadUCS	Creates a member object and adds it to the collection. Parameters: Origin As Variant, XAxisPoint As Variant, YAxisPoint As Variant, and Name As String. The Origin parameter specifies where the UCS is to be added. The XAxisPoint parameter specifies a point on the positive X-axis of the UCS. The YAxisPoint parameter specifies a point on the positive Y-axis of the UCS. (The Z-axis follows by applying the right-hand rule.) These coordinates are all 3-D in WCS. Note that XAxisPoint and YAxisPoint together can't specify the same location as the Origin. The Name parameter specifies the name of the UCS to add to the collection.
Item	AcadUCS	Gets the member object at a given index in a collection. Parameter: Index As Variant (an Integer or a String). If the Index value is a String, it must match an existing object name in the collection.

AcadUCSs Collection Properties

The AcadUCSs collection inherits all the properties of the AcadObject object, the Count property, and the common Application property. It supports no other properties.

■Note Because this collection inherits from AcadObject, it supports the Modified event.

AcadUtility Object

The AcadUtility object represents a series of methods provided for utility purposes.

AcadUtility Object Methods

The following methods are supported.

Name	Returns	Description
AngleFromXAxis	Double	Returns the angle (in radians) of a line passing through two points from the X-axis. Parameters: Point1 As Variant and Point2 As Variant. The parameters are three-element arrays of Doubles.
AngleToReal	Double	Converts an angle as a string to a real number (Double). Parameters: Angle As String and Unit As AcAngleUnits. For the possible values of the AcAngleUnits enumerated type, see Appendix B.
AngleToString	String	Converts an angle from a real number (Double) to a string. Parameters: Angle As Double, Unit As AcAngleUnits, and Precision As Integer. For the possible values of the AcAngleUnits enumerated type, see Appendix B. The range of values for the third parameter is 0 to 8.
CreateTypedArray		Creates an array that contains the values specified by the parameters. The resulting array can be passed into any AutoCAD method or property that accepts an array of numbers. Parameters: VarArr As Variant, Type As (Visual Basic Constant), Value1 As Type, [Value2 As Type], …, and [ValueN As Type]. The values for Type are vbBoolean, vbInteger, vbLong, vbSingle, and vbDouble.
DistanceToReal	Double	Converts a distance from a string to a real number (Double). Parameters: Distance As String and Unit As AcUnits. For the possible values of the AcUnits enumerated type, see Appendix B.
GetAngle	Double	Asks the user to input an angle and then returns its value, taking into account the value in the ANGBASE system variable. The value of the angle returned is expressed in radians. The direction of angular increase is always counterclockwise. Parameters: [Point As Variant] and [Prompt As String]. The Point parameter in this method and the two that follow specifies 3-D WCS coordinates. The Prompt parameter in this method and the methods that follow specifies text string used to prompt the user for input.

Name	Returns	Description
GetCorner	Variant	Asks the user to input the corner of a rectangle and returns the point as an array of three Doubles. Note that a call to GetCorner will frequently follow a call to GetPoint, the first point selected being used as the first parameter for the GetCorner method. Parameters: Point As Variant and [Prompt As String].
GetDistance	Variant	Asks the user to input a linear distance from the keyboard or by specifying two locations on the graphics screen, and returns it as a Double number or as an array of Double numbers. Parameters: [Point As Variant] and [Prompt As String].
GetEntity		Asks the user to select a drawing object by picking a point on the screen. The object is then placed in the first parameter. Parameters: Object As Object, PickedPoint As Variant, and [Prompt As String]. The PickedPoint parameter specifies 3-D WCS coordinates.
GetInput	String	Asks the user to input text and returns it. A call to this method must follow a call to a user-input function returning the "User input is a keyword" error description. The input must be no longer than 511 characters.
GetInteger	Integer	Asks the user to input an integer and returns it. The AutoCAD user can enter any short integer in the range of –32,768 to +32,767. Parameter: [Prompt As String].
GetKeyword	String	Asks the user to input a keyword and returns it. The keyword must be no longer than 511 characters. If the user doesn't enter anything but presses the *Enter* key, this method returns an empty string. This method can be used only if InitializeUserInput is called first. Parameter: [Prompt As String].
GetOrientation	Double	Asks the user to input an angle and then returns its value in radians without regard to the values stored in the ANGBASE system variable. The 0 angle employed by this method is always to the right (east). Parameters: [Point As Variant] and [Prompt As String]. The Point parameter specifies 3-D WCS coordinates.
GetPoint	Variant	Asks the user to choose a point and returns it. Parameters: [Point As Variant] and [Prompt As String]. The Point parameter and the return value specify 3-D WCS coordinates.
GetReal	Double	Asks the user to input a real (Double) number and returns it. Parameter: [Prompt As String].
GetRemoteFile		Downloads the specified file from a URL. Parameters: URL As String, LocalFile As String, and IgnoreCache As Boolean. The third parameter specifies whether to download the file, even if it has already been transferred earlier in this session.
GetString	String	Asks the user to input a string and returns it. Parameters: HasSpaces As Boolean and [Prompt As String]. HasSpaces determines whether the return string can contain spaces. If True, the string can contain spaces but terminated by a carriage return only. If False, the string can't contain spaces. It can be terminated by a carriage return or a space.

Continued

Name	Returns	Description
GetSubEntity		Asks the user to select a drawing object by specifying the object and the selected in the first two parameters. Parameters: Object As Object, PickedPoint As Variant, TransMatrix As Variant, ContextData As Variant, and [Prompt As String]. The PickedPoint parameter specifies 3-D WCS coordinates. The TransMatrix parameter represents the translation matrix applied to this entity. The Variant parameter is a 4×4 array of Doubles. The ContextData parameter is an array of Longs, containing the object IDs for any nested objects in the selected object. More specifically, this argument returns an array holding the object ID(s) of the subentity's container block references. The first array element is the object ID of the block reference containing the picked entity. The last array element is the object ID of the block reference inserted directly in the drawing. The array of object IDs is therefore ordered from the most deeply nested block reference that contains the subentity to the object ID of the block reference that would have been picked with the GetEntity function. If the picked entity isn't a subentity (i.e., it isn't nested in a block reference), then the GetSubEntity function returns an empty array for the ContextData parameter.
InitializeUserInput		Initializes the GetKeyword method. Parameter: Bits As Integer and [Keyword As String]. For the values for the Bits parameter and their explanations, see the following table. The second parameter specifies the keywords, delimited by spaces, that the GetKeyword method will recognize. This method must be called before GetKeyword.
IsRemoteFile	Boolean	Returns whether the specified file was downloaded from a remote location. This method is the inverse of the IsURL method and provides a mapping from a local file to the corresponding URL that the file was downloaded from. Parameters: LocalFile As String and URL As String.
IsURL	Boolean	Returns whether a string is a valid URL for the application. The default functionality of this method for AutoCAD supports the FTP, HTTP, HTTPS, and FILE protocols. Parameter: URL As String.
LaunchBrowserDialog	Boolean	Runs the web browser dialog box that allows the user to navigate to any URL and select a URL. It returns whether the operation was successful. Parameters: SelectedURL As String, DialogTitle As String, OpenButtonCaption As String, StartPageURL As String, RegistryRootKey As String, and OpenButtonAlwaysEnabled As Boolean. The first four parameters specify the selected URL, the title to be displayed for the browser dialog box, the caption for the OK/Open button, and the URL that the web browser should use as its start page, respectively. The fifth parameter is used to specify where in the Registry persistent web browser dialog box information is stored. Input an empty string to disregard this functionality. The sixth parameter determines whether the user can select HTML links in addition to downloadable file. If True, the Open button is enabled, allowing a file or link to be selected. If False, the Open button is disabled and is only enabled when the user selects a file for download.

Name	Returns	Description
PolarPoint	Variant	Returns a point at the specified angle (in radians) and distance (in current units) from the specified point. Parameters: Point As Variant, Angle As Double, and Distance As Double. The value for Point and the return value are both 3-D WCS coordinates.
Prompt		Sends a message to the command line. Parameter: Message As String.
PutRemoteFile		Puts a local file at the location specified by a URL. This method is designed to complement GetRemoteFile method. When accessing a secure URL, a dialog box will be posted prompting the user for the necessary password information. Message boxes may also appear if the user hasn't suppressed this activity in the browser. Parameter: URL As String.
RealToString	String	Converts a real number into a properly formatted string. Parameters: Value As Double, Unit As AcUnits, and Precision As Integer. For a list of possible values for the AcUnits enumerated type, see Appendix B. The range of values for the third parameter is 0 to 8.
TranslateCoordinates	Variant	Changes a point from one coordinate system to another. You can't directly translate a set of coordinates from one OCS to another OCS. If you want to do this, you must first translate the OCS coordinates to WCS and then change these coordinates to the second OCS. Parameters: OriginalPoint As Variant, From As AcCoordinateSystem, To As AcCoordinateSystem, Disp As Boolean, and [OCSNormal As Variant]. OriginalPoint specifies the original 3-D WCS coordinates to be translated. This parameter can be treated as a point or a displacement vector depending on the value of Disp. For a list of possible values for the AcCoordinateSystem enumeration (the From and To parameters), see Appendix B. Disp is a displacement vector flag. If True, OriginalPoint is treated as a displacement vector. Otherwise it is treated as a point. The optional last parameter represents the normal for the OCS.

AcadUtility Object Properties

The AcadUtility object inherits the common Application property. It supports no other properties.

AcadView Object

The AcadView object represents a graphical representation of a 2-D drawing or 3-D model from a specific location (viewpoint) in space. The line of sight is drawn from the viewpoint, represented by the Center property, to the target point. The Height and Width properties crop the view to fit into the viewport.

AcadView Object Methods

The AcadView object inherits all the methods of the AcadObject object. It supports no other methods.

AcadView Object Properties

The AcadView object inherits all the properties of the AcadObject object, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Center	Variant	Gets or sets the center of the view as a set of 2D coordinates (as Doubles). The default center is (0,0).
Direction	Variant	Gets or sets the viewing direction for a 3-D visualization of the drawing. This property puts the viewer in a position to look at the drawing as if looking back at the origin (0, 0, 0) from a specified point in space. It is similar to AutoCAD's VPOINT command. It returns a three-element array of Doubles.
Height	Double	Gets or sets the height of the view, which is the Y-axis measurement of the area within a viewport that is used to display the model.
Name	String	Gets or sets the name of the view.
Target	Variant	Gets or sets the target point for the view as a set of 3-D WCS coordinates representing the target point. The line of sight is drawn from the center to the target point. It returns a three-element array of Doubles.
Width	Double	Gets or sets the width of the view, which is the X-axis measurement of the area within a viewport that is used to display the model.

Note Because this object inherits from AcadObject, it supports the Modified event.

AcadViewport Object

The AcadViewport object represents a bounded area that displays some portion of a drawing's model space. You can activate it by using the ActiveViewport property of the Document object. The ActiveSpace property, which is equivalent to the TILEMODE system variable, determines the type of viewport used. You can make changes to the view only while the viewport is active, and you can see any changes only once the viewport has been reactivated.

AcadViewport Object Methods

The AcadViewport object inherits all the methods of the AcadObject object. It also supports the following methods.

Name	Description
GetGridSpacing	Gets the grid spacing for the viewport. Parameters: XSpacing As Double and YSpacing As Double.
GetSnapSpacing	Gets the snap spacing for the viewport. Parameters: XSpacing As Double and YSpacing As Double.
SetGridSpacing	Sets the grid spacing for the viewport. Parameters: XSpacing As Double and YSpacing As Double.
SetSnapSpacing	Sets the snap spacing for the viewport. Parameters: XSpacing As Double and YSpacing As Double.
SetView	Sets the view in a viewport to a saved view in the AcadViews collection. Parameter: View As AcadView.
Split	Splits a viewport into the given number of views. Parameter: NumWins As AcViewportSplitType. For a list of possible values for the AcViewportSplitType enumerated type, see Appendix B.

AcadViewport Object Properties

The AcadViewport object inherits all the properties of the AcadObject object, as well as the common Application property. It also supports the following properties. Note that many of these properties are identical to those of the AcadPViewport object. See the AcadPViewport object for more details for the ArcSmoothness, Center, Direction, GridOn, SnapBasePoint, SnapOn, and SnapRotationAngle properties.

Name	Returns	Description
ArcSmoothness	Integer	Gets or sets the smoothness of circles, arcs, and ellipses. The valid range of values is from 1 to 20000. The initial value for this property is 100.
Center	Variant	Gets or sets the center of the viewport as a set of 2-D coordinates (as Doubles). The default center is (0,0).
Direction	Variant	Gets or sets the viewing direction for a 3-D visualization of the drawing. This property puts the viewer in a position to look at the drawing as if looking back at the origin (0, 0, 0) from a specified point in space.
GridOn	Boolean	Specifies the status of the viewport grid. The value of this property is stored in the GRIDMODE system variable.
Height	Double	Gets or sets the height of the viewport, which is the Y-axis measurement of the area within a viewport that is used to display the model.
LowerLeftCorner	Variant	Gets the lower-left corner of the current active viewport, which with the UpperRightCorner property represents the graphic placement of the viewport on the display. It returns a set of 2-D coordinates as an array of Doubles. This property's value is read-only.
Name	String	Gets or sets the name of the viewport.
OrthoOn	Boolean	Specifies status of the Ortho mode for the viewport. You can use Ortho mode to restrict the cursor to the horizontal or vertical axis, which helps you draw parallel lines or move objects.

Continued

Name	Returns	Description
SnapBasePoint	Variant	Gets or sets the snap base point for the viewport, the value of which is stored in the SNAPBASE system variable. The snap base point can't be changed for the active paper space viewport. Any changes to this property aren't reflected in the display until the drawing is regenerated.
SnapOn	Boolean	Specifies the status of snap mode. The value of this property is stored in the SNAPMODE system variable.
SnapRotationAngle	Double	Gets or sets the snap rotation angle (in radians) of the viewport relative to the current UCS. The value of this property is stored in the SNAPANG system variable. The valid range is 0 to 6.28.
Target	Variant	Gets or sets the target point for the viewport as a set of 3-D WCS coordinates representing the target point. The line of sight is drawn from the center to the target point. It returns a three-element array of Doubles.
UCSIconAtOrigin	Boolean	Specifies whether the UCS icon is displayed at the origin or at the WCS coordinate defined by the UCSORG system variable.
UCSIconOn	Boolean	Specifies whether the UCS icon is on.
UpperRightCorner	Variant	Gets the upper-right corner of the current active viewport, which with the LowerLeftCorner property represents the graphic placement of the viewport on the display. It returns a set of 2-D coordinates as an array of Doubles. This property's value is read-only.
Width	Double	Gets or sets the width of the viewport, which is the X-axis measurement of the area within a viewport that is used to display the model.

Note Because this object inherits from AcadObject, it supports the Modified event. There is no programmatic means to directly create or manipulate polygonal viewports.

AcadViewports Collection

The AcadViewports collection contains all the viewports in the drawing. Although this collection inherits a Delete method, you can't actually delete this collection. If you need to delete a specific viewport, use the DeleteConfiguration method or the Delete method of the AcadViewport object. The number of viewports you can have active at one time is controlled by the MAXACTVP system variable. However, there can be only one instance of the AcadViewports collection, which is predefined for each drawing. You can make multiple references to it by using the Viewports property of the Document object.

AcadViewports Collection Methods

The AcadViewports collection inherits all the methods of the AcadObject object. It also supports the following methods.

Name	Returns	Description
Add	AcadViewport	Creates a member object and adds it to the collection. Parameter: Name As String.
DeleteConfiguration		Deletes a viewport configuration, which consists of a single viewport that has been split using the Split method. Once a viewport has been split, the resulting viewports are considered a viewport configuration and have the same name as the original viewport before the split. Parameter: Name As String.
Item	AcadViewport	Gets the member object at a given index in a collection. Parameter: Index As Variant (an Integer or a String). If the Index value is a String, it must match an existing viewport in the collection.

AcadViewports Collection Properties

The AcadViewports collection inherits all the properties of the AcadObject object, the Count property, and the common Application property. It supports no other properties.

■ **Note** Because this collection inherits from AcadObject, it supports the Modified event.

AcadViews Collection

The AcadViews collection contains all the views in the drawing. Although this collection inherits a Delete method, you can't actually delete this collection. If you need to delete a specific view, use the Delete method of the AcadView object. There is no limit to the number of views you can create in your drawing. However, there can be only one instance of the AcadViews collection, which is predefined for each drawing. You can make multiple references to it by using the Views property of the Document object.

AcadViews Collection Methods

The AcadViews collection inherits all the methods of the AcadObject object. It supports no other methods.

AcadViews Collection Properties

The AcadViews collection inherits all the properties of the AcadObject object, the Count property, and the common Application property. It supports no other properties.

■ **Note** Because this collection inherits from AcadObject, it supports the Modified event.

AcadXline Object

The AcadXline object represents a construction line that is infinite in both directions, unlike the AcadRay object, which is infinite in only one direction. It is created using the AddXline method of the AcadBlock, AcadModelSpace, or AcadPaperSpace object.

AcadXline Object Methods

The AcadXline inherits all the methods of the AcadEntity and AcadObject objects. It also supports the following method.

Name	Returns	Description
Offset	Variant	Creates a new line by offsetting the current Xline by a specified distance, which can be positive or negative but can't be zero. If the offset is negative, this means that the line is drawn closer to the WCS origin. Parameter: Distance As Double.

AcadXline Object Properties

The AcadXline object inherits all the properties of the AcadEntity and AcadObject objects, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
BasePoint	Variant	Gets or sets the point through which the Xline passes. It returns a set of 3-D coordinates as a three-element array of Doubles.
DirectionVector	Variant	Gets or sets the direction for the Xline through a vector. It returns a three-element array of Doubles.
SecondPoint	Variant	Gets or sets a second point on the Xline. It returns a set of 3-D WCS coordinates as a three-element array of Doubles.

■ **Note** Because this object inherits from AcadObject, it supports the Modified event.

AcadXRecord Object

The AcadXRecord object is used to store and manage arbitrary data. XRecords are similar in concept to XData but are not limited by size or order. They use standard AutoCAD group codes, the values of which are all below 1000, which means in addition to all the normally used data types, XRecords are capable of storing object IDs, allowing ownership of many other objects. For a list of supported group codes, see the AutoCAD documentation.

XRecord objects are saved with the drawing and can be directly accessed by other ObjectARX and LISP programs. For this reason, you should take care when storing security information in XRecords. XRecords are created using the AddXRecord method of the Dictionary object and accessed programmatically through the Item method of the Dictionary object.

The only basic operations that can be performed on XRecords are Add, Get, and Delete. There is no means to update (modify) an XRecord other than to delete and re-create it.

AcadXRecord Object Methods

The AcadXRecord object inherits all the methods of the AcadObject object. It also supports the following methods.

Name	Description
GetXRecordData	Gets the extended record data associated with a dictionary. Parameters: XRecordDataType As Variant and XRecordDataValue As Variant.
SetXRecordData	Sets the extended record data associated with a dictionary. Parameters: XRecordDataType As Variant and XRecordDataValue As Variant.

AcadXRecord Object Properties

The AcadXRecord object inherits all the properties of the AcadObject object, as well as the common Application property. It also supports the following properties.

Name	Returns	Description
Name	String	Gets or sets the name of the object within the dictionary. This name doesn't represent the class name of the object.
TranslateIDs	Boolean	Specifies whether translation of any contained object IDs occurs during deepClone or wblockClone operations.

■ **Note** Because this object inherits from AcadObject, it supports the Modified event.

SecurityParams Object

The SecurityParams object stores settings for drawing security. The SecurityParams object encrypts and digitally signs drawing files.

The object properties included provide information about cryptography providers and other settings for drawing security. Additional information about cryptography providers is available on MSDN (http://msdn.microsoft.com).

Name	Returns	Description

Action	Long	Gets or sets the desired security operation to be performed. Use one or more of the following constants for drawing encryption, drawing properties encryption, a digital signature, or a timestamp:
		ACADSECURITYPARAMS_ENCRYPT_DATA = 0×00000001
		ACADSECURITYPARAMS_ENCRYPT_PROPS = 0×00000002
		ACADSECURITYPARAMS_SIGN_DATA = 0×00000010
		ACADSECURITYPARAMS_ADD_TIMESTAMP = 0×00000020

Continued

Name	Returns	Description
`Algorithm`	Long	Gets or sets the identifier of the encryption algorithm. This value can only be the following constant: `ACADSECURITYPARAMS_ALGID_RC4` = 0×00006801
`Comment`	String	Gets or sets the comment to be included with a digital signature.
`Issuer`	String	Gets or sets the issuer of a digital certificate.
`KeyLength`	Long	Gets or sets the length of the encryption key, which is based upon the value in the `ProviderName` property.
`Password`	String	Gets or sets the encryption password.
`ProviderName`	String	Gets or sets the encryption provider name. More information about cryptography providers is available on MSDN.
`ProviderType`	Long	Gets or sets the encryption provider type. More information about cryptography providers is available on MSDN.
`SerialNumber`	String	Gets or sets the serial number of the digital certificate.
`Subject`	String	Gets or sets the subject name of the digital certificate.
`TimeServer`	String	Gets or sets the name of the time server to be used for a digital signature. If you don't set this property, the time from the local machine is used as the timestamp for the digital signature.

`AcCmColor` Object

The `AcCmColor` object represents colors within AutoCAD. You use the `AcCmColor` object to set colors and perform other color-related operations on objects.

`AcCmColor` Object Methods

Unlike the other AutoCAD objects, the `AcCmColor` object doesn't inherit methods from any other AutoCAD objects. It supports the following methods.

Name	Description
`SetColorBookColor`	Sets the color name from an existing color book. Parameters: `ColorName As String` and `ColorBook As String`.
`SetNames`	Sets the color name and book name of the color. Parameters: `ColorName As String` and `BookName As String`.
`SetRGB`	Sets the RGB values of the True Color. Parameters: `Red As Long`, `Green As Long`, and `Blue As Long`.

`AcCmColor` Object Properties

Unlike the other AutoCAD objects, the `AcCmColor` object doesn't inherit properties from any other AutoCAD objects. It supports the following properties.

Name	Returns	Description
Blue	Long	Gets the blue component, from 0 to 255, of AcCmColor. This property's value is read-only.
BookName	String	Gets the name of the color book that the color came from. This property's value is read-only.
ColorIndex	acColor enum	Gets or sets the default color designation in acByLayer. Use a color index number from 0 to 256 or one of the constants listed here: acByBlock acByLayer (not valid for the Layer object) acRed acYellow acGreen acCyan acBlue acMagenta acWhite
ColorMethod	acColorMethod enum	Gets or sets the default color method in acColorMethodByLayer. To directly determine how a color is set, use the constants listed here: acColorMethodByACI acColorMethodByBlock acColorMethodByLayer acColorMethodByRGB acColorMethodForeground
ColorName	String	Gets the name (if any) of the color. This property's value is read-only.
EntityColor	Long	Gets or sets the 32-bit AcCmEntityColor portion of the color.
Green	Long	Gets the green component, from 0 to 255, of AcCmColor. This property's value is read-only.
Red	Long	Gets the red component, from 0 to 255, of AcCmColor. This property's value is read-only.

LayerStateManager Object

The LayerStateManager object provides a set of functions for working with saved layer settings. The LayerStateManager object manipulates XRecord objects that define the properties of a layer. These XRecords are stored in the ACAD_LAYERSTATE dictionary, which is an extension dictionary in the drawing's Layers collection. (An extension dictionary is a mechanism for attaching data to objects. Every AutoCAD object can have an extension dictionary.)

After you retrieve the LayerStateManager object and associate a database with it, then you can access the object's methods. Use the SetDatabase method to associate a database with the LayerStateManager object.

LayerStateManager Object Methods

Unlike the other AutoCAD objects, the LayerStateManager object doesn't inherit methods from any other AutoCAD objects. It supports the following methods.

Name	Description
`Delete`	Name representing the layer state to be deleted. Parameter: `Name As String`.
`Export`	Exports an AutoCAD drawing or a group of saved layer settings to a file. Parameters: `Filename As String`, `Extension As String`, and `SelectionSet As SelectionSet`.
`Import`	Imports a drawing or a group of saved layer settings from a file. Parameters: `Filename As String`, `InsertionPoint As Variant`, and `ScaleFactor As Double`.
`Rename`	Renames a set of saved layer settings. Parameters: `OldName As String` and `NewName As String`.
`Restore`	Restores a group of layer property settings. Parameter: `Name As String`.
`Save`	Saves a group of layer property settings. Parameters: `Name As String` and `Mask` (set the `Mask` property as described in the next table).
`SetDatabase`	Gets AutoCAD database associated with the `LayerStateManager` object.

`LayerStateManager` Object Properties

Unlike the other AutoCAD objects, the `LayerStateManager` object doesn't inherit properties from any other AutoCAD objects. It supports the following properties.

Name	Returns	Description
`Mask`	Enum	Specifies the layer properties to be restored as a number representing the layer properties to be restored. The following table represents acceptable values for this property.

Constant	Layer Property	Numeric Value
`AcLsAll`	All layer properties	–65535
`AcLsColor`	Color	–32
`AcLsFrozen`	Frozen or thawed	–2
`AcLsLineType`	Linetype	–64
`AcLsLineWeight`	Lineweight	–128
`AcLsLocked`	Locked or unlocked	–4
`AcLsNewViewport`	New viewport layers frozen or thawed	–16
`AcLsNone`	None	–0
`AcLsOn`	On or off	–1
`AcLsPlot`	Plotting on or off	–8
`AcLsPlotStyle`	Plot style	–256

APPENDIX B

AutoCAD Constants Reference

This appendix lists all the enumerated types used in conjunction with the AutoCAD objects. The methods and properties (presented in Appendix A) that use them have been noted.

Ac3DPolylineType

The following constants can be values for the `Type` property of the `Acad3DPolyline` object.

Constant	Value	Description
`acSimple3Dpoly`	0	A simple polyline
`acQuadSpline3Dpoly`	1	A quadratic B-spline polyline
`acCubicSpline3Dpoly`	2	A cubic B-spline polyline

AcActiveSpace

The following constants can be values for the `ActiveSpace` property of the `AcadDocument` object.

Constant	Value	Description
`acModelSpace`	1	The active space is model space.
`acPaperSpace`	2	The active space is paper space.

ACAD_LWEIGHT

The following constants can be values for the `GetGridLineWeight` and `SetGridLineWeight` properties of an `AcadTable` and `AcadTableStyle` object. Additionally, the following constants can be values for the `GetCell` and `SetCellGridLineWeight` methods of an `AcadTable` object.

Constant	Value	Constant	Value
acLnWt000	0	acLnWt070	70
acLnWt005	5	acLnWt080	80
acLnWt009	9	acLnWt090	90
acLnWt013	13	acLnWt100	100
acLnWt015	15	acLnWt106	106
acLnWt018	18	acLnWt120	120
acLnWt020	20	acLnWt140	140
acLnWt025	25	acLnWt158	158
acLnWt030	30	acLnWt200	200
acLnWt035	35	acLnWt211	211
acLnWt040	40	acLnWtByLayer	–1
acLnWt050	50	acLnWtByBlock	–2
acLnWt053	53	acLnWtByLwDefault	–3
acLnWt060	60		

AcAlignment

The following constants can be values for the Alignment property of the AcadAttribute, AcadAttributeReference, and AcadText objects.

Constant	Value	Description
acAlignmentLeft	0	The attribute, attribute reference, or text is aligned with the insertion point.
acAlignmentCenter	1	The attribute, attribute reference, or text is aligned at the center.
AcAlignmentRight	2	The attribute, attribute reference, or text is aligned at the right.
acAlignmentAligned	3	The attribute, attribute reference, or text is aligned according to the points held in the InsertionPoint and TextAlignmentPoint properties.
acAlignmentMiddle	4	The attribute, attribute reference, or text is aligned at the middle, at a position slightly below middle center.
acAlignmentFit	5	The attribute, attribute reference, or text is aligned to fit to the points held in the InsertionPoint and TextAlignmentPoint properties.
acAlignmentTopLeft	6	The attribute, attribute reference, or text is aligned at the top left.
acAlignmentTopCenter	7	The attribute, attribute reference, or text is aligned at the top center.
acAlignmentTopRight	8	The attribute, attribute reference, or text is aligned at the top right.

Constant	Value	Description
acAlignmentMiddleLeft	9	The attribute, attribute reference, or text is aligned at the middle left.
acAlignmentMiddleCenter	10	The attribute, attribute reference, or text is aligned at the middle center.
acAlignmentMiddleRight	11	The attribute, attribute reference, or text is aligned at the middle right.
acAlignmentBottomLeft	12	The attribute, attribute reference, or text is aligned at the bottom left.
acAlignmentBottomCenter	13	The attribute, attribute reference, or text is aligned at the bottom center.
acAlignmentBottomRight	14	The attribute, attribute reference, or text is aligned at the bottom right.

AcAlignmentPointAcquisition

The following constants can be values for the `AlignmentPointAcquisition` property of the `AcadPreferencesDrafting` object.

Constant	Value	Description
acAlignPntAcquisitionAutomatic	0	Autoalignment points are acquired automatically.
acAlignPntAcquisitionShiftToAcquire	1	The user must use the Shift key to acquire autoalignment points.

AcAngleUnits

The following constants can be values for the `AngleFormat` property of the `AcadDim3PointAngular` object, and they can be supplied as the value for the second parameter of the `AngleToReal` and `AngleToString` methods of the `AcadUtility` object.

Constant	Value	Description
acDegrees	0	Angle units are degrees.
acDegreeMinuteSeconds	1	Angle units are degrees, minutes, and seconds.
acGrads	2	Angle units are grads.
acRadians	3	Angle units are radians.

AcARXDemandLoad

The following constants can be values for the `DemandLoadARXApp` property of the `AcadPreferencesOpenSave` object.

Constant	Value	Description
`acDemandLoadDisable`	0	Turns off demand loading.
`acDemandLoadOnObjectDetect`	1	Demand-loads the source application when you open a drawing that contains custom objects. This setting doesn't demand-load the application when you invoke one of the application's commands.
`acDemandLoadCmdInvoke`	2	Demand-loads the source application when you invoke one of the application's commands. This setting doesn't demand-load the application when you open a drawing that contains custom objects.

AcAttachmentPoint

The following constants can be values for the `AttachmentPoint` property of the `AcadMText` object.

Constant	Value	Description
`acAttachmentPointTopLeft`	1	Left-justified, spills down
`acAttachmentPointTopCenter`	2	Center-justified, spills down
`acAttachmentPointTopRight`	3	Right-justified, spills down
`acAttachmentPointMiddleLeft`	4	Left-justified, spills up and down
`acAttachmentPointMiddleCenter`	5	Center-justified, spills up and down
`acAttachmentPointMiddleRight`	6	Right-justified, spills up and down
`acAttachmentPointBottomLeft`	7	Left-justified, spills up
`acAttachmentPointBottomCenter`	8	Center-justified, spills up
`acAttachmentPointBottomRight`	9	Right-justified, spills up

AcAttributeMode

The following constants can be values for the `Mode` property of the `AcadAttribute` object, and they can also be supplied as the value for the second parameter of the `AddAttribute` method of the `AcadBlock`, `AcadModelSpace`, and `AcadPaperSpace` objects.

Constant	Value	Description
acAttributeModeNormal	0	The default mode. In other words, it lacks any of the other modes listed here.
acAttributeModeInvisible	1	When the block is inserted, the attribute's values won't be visible. The system variable ATTDISP overrides this mode setting.
acAttributeModeConstant	2	Attribute values will have a fixed value for each block inserted. When set, prompting for the attribute is disabled.
acAttributeModeVerify	4	When you insert the block, AutoCAD will ask you to verify that the attribute value is correct.
acAttributeModePreset	8	The block is inserted using its default attribute values. You can't edit these values.

AcBlockScaling

The following constants can be values for the BlockScaling property of an AcadBlock object. To access this property, use the IAcadBlock3 interface.

Constant	Value	Description
AcAny	0	The X, Y, Z scale values may be different.
acUniform	1	The X, Y, Z scale values are equal.

AcBooleanType

The following constants are values supplied as the value for the first parameter of the Boolean methods of the Acad3DSolid and AcadRegion objects.

Constant	Value	Description
acUnion	0	Performs a union operation
acIntersection	1	Performs an intersection operation
acSubtraction	2	Performs a subtraction operation

AcCellAlignment

The following constants are possible return values for the GetAlignment and GetCellAlignment methods. Also, they can be values for the CellAlignment parameter of the SetAlignment and SetCellAlignment methods.

Constant	Value	Description
acBottomCenter	8	Centers text at a point based on the bottom of the text
acBottomLeft	7	Left-justifies text at a point based on the bottom of the text
acBottomRight	9	Right-justifies text at a point based on the bottom of the text
acMiddleCenter	5	Centers text at a point based on the middle of the text
acMiddleLeft	4	Left-justifies text at a point based on the middle of the text
acMiddleRight	6	Right-justifies text at a point based on the middle of the text
acTopCenter	2	Centers text at a point based on the top of the text
acTopLeft	1	Left-justifies text at a point based on the top of the text
acTopRight	3	Right-justifies text at a point based on the top of the text

AcCellEdgeMask

The following constants can be values for the Edge parameter of the GetCellGridColor, GetCellGridLineWeight, and GetCellGridVisibility methods of an AcadTable object.

Constant	Value	Description
acBottomMask	4	Bottom-edge index of the cell
acLeftMask	8	Left-edge index of the cell
acRightMask	2	Right-edge index of the cell
acTopMask	1	Top-edge index of the cell

AcCellType

The following constants are possible return values for the GetCellType method of an AcadTable object. Also, they can be values for the CellType property or the SetCellType method of an AcadTable object.

Constant	Value
acBlockCell	2
acTextCell	1
acUnknownCell	0
AcColor (and the ColorIndex property of the AcCmColor pbject)	

The following constants can be values for these properties:

- The AutoSnapMarkerColor property of the AcadPreferencesDrafting object
- The Color property of the AcadEntity object (and all derived objects) and the AcadGroup and AcadLayer objects

- The `DimensionLineColor` property of the dimension objects that support it (all except `AcadDimOrdinate`) and the `AcadLeader` and `AcadTolerance` objects
- The `ExtensionLineColor` property of the dimension objects that support it (all except `AcadDimDiametric` and `AcadDimRadial`)
- The `GripColorSelected` and `GripColorUnselected` properties of the `AcadPreferencesSelection` object
- The `TextColor` property of the `AcadDimension` object (and all derived objects) and the `AcadTolerance` object

Constant	Value	Description
`acByBlock`	0	Objects are drawn in the default color until they're grouped into a block. When the block is inserted into a drawing, the objects inherit the value of the block's `Color` property.
`acRed`	1	Red
`acYellow`	2	Yellow
`acGreen`	3	Green
`acCyan`	4	Cyan
`acBlue`	5	Blue
`acMagenta`	6	Magenta
`acWhite`	7	White
`acByLayer`	256	Object assumes color of layer on which it is drawn.

AcColor

The following constants can be values for these properties:

- `AutoSnapMarkerColor` property of the `PreferencesDrafting` object
- The `Color` property of all `Drawing Objects`, `Group`, `Layer`, and `LabelTemplate` objects
- The `DimensionLineColor` property of the `Dim3PointAngular`, `DimAligned`, `DimAngular`, `DimArcLength`, `DimDiametric`, `DimRadial`, `DimRadialLarge`, `DimRotated`, `Leader`, and `Tolerance` objects
- The `ExtensionLineColor` property of the `Dim3PointAngular`, `DimAligned`, `DimAngular`, `DimArcLength`, `DimOrdinate`, and `DimRotated` objects
- The `GripColorSelected` and `GripColorUnselected` properties of the `PreferencesSelection` object
- The `TextColor` property of the `Dim3PointAngular`, `DimAligned`, `DimAngular`, `DimArcLength`, `DimDiametric`, `DimOrdinate`, `DimRadial`, `DimRadialLarge`, `DimRotated`, and `Tolerance` objects
- The `ColorIndex` property of the `AcCmColor` object

Constant	Value	Description
acByBlock	0	ByBlock
acRed	1	Red
acYellow	2	Yellow
acGreen	3	Green
acCyan	4	Cyan
acBlue	5	Blue
acMagenta	6	Magenta
acWhite	7	White
acByLayer	8	ByLayer

AcColorMethod

The following constants can be supplied as values for the AcColorMethod object as it relates to the TrueColor object. Only those constants marked with an asterisk (*), however, apply to the GradientColor1 and GradientColor2 properties for Hatch objects.

Constant	Value	Description
acColorMethodByACI*	195	Use traditional color index value.
acColorMethodByBlock*	193	
acColorMethodByLayer	192	
acColorMethodByRGB	194	Use RGB index values to define color.
acColorMethodForeground	197	

AcCoordinateSystem

You can supply the following constants as values for the second and third parameters of the TranslateCoordinates method of the AcadUtility object.

Constant	Value	Description
acWorld	0	World Coordinate System (WCS)
acUCS	1	User Coordinate System (UCS)
acDisplayDCS	2	Display Coordinate System (DCS)
acPaperSpaceDCS	3	Paper Space Display Coordinate System (PSDCS)
acOCS	4	Object Coordinate System (OCS)

AcDimArcLengthSymbol

The following constants can be values for the SymbolPosition property of an AcadDimArcLength object.

Constant	Value
acSymAbove	1
acSymInFront	0
acSymNone	2

AcDimArrowheadType

The following constants can be values for these properties:

- The Arrowhead1Type and Arrowhead2Type properties of the dimension objects that support them (all except AcadDimOrdinate and AcadDimRadial)
- The Arrowhead1Type property of the AcadTolerance object
- The ArrowheadType property of the AcadDimRadial and AcadLeader objects

Constant	Value	Description
acArrowDefault	0	The arrowhead is the default determined by the dimension style.
acArrowClosedBlank	1	The arrowhead is a closed empty triangle.
acArrowClosed	2	The arrowhead is a closed triangle with the dimension line inside.
acArrowDot	3	The arrowhead is a large, filled circle.
acArrowArchTick	4	The arrowhead is a bold, oblique line.
acArrowOblique	5	The arrowhead is an oblique line.
acArrowOpen	6	The arrowhead is a narrow, open arrow.
acArrowOrigin	7	The arrowhead is a circle with the dimension line inside.
acArrowOrigin2	8	The arrowhead comprises two circles with the dimension line inside.
acArrowOpen90	9	The arrowhead is an open arrow with an internal angle of 90 degrees.
acArrowOpen30	10	The arrowhead is an open arrow with an internal angle of 30 degrees.
acArrowDotSmall	11	The arrowhead is a small, filled circle.
acArrowDotBlank	12	The arrowhead is a large, empty circle.
acArrowSmall	13	The arrowhead is a small arrow.
acArrowBoxBlank	14	The arrowhead is an empty box.
acArrowBoxFilled	15	The arrowhead is a filled box.

Continued

Constant	Value	Description
`acArrowDatumBlank`	16	The arrowhead is a closed, empty, inward-pointing triangle.
`acArrowDatumFilled`	17	The arrowhead is a filled, inward-pointing triangle.
`acArrowIntegral`	18	The arrowhead is an integral sign.
`acArrowNone`	19	There is no arrowhead.
`acArrowUserDefined`	20	The arrowhead is defined by the user.

AcDimCenterType

The following constants can be values for the `CenterType` property of the `AcadDimDiametric` and `AcadDimRadial` objects.

Constant	Value	Description
`acCenterMark`	0	Marks the center with a small cross (+)
`acCenterLine`	1	Marks the center as the intersection of two perpendicular axes
`acCenterNone`	2	No center mark

AcDimFit

The following constants can be values for the `Fit` property of the dimension objects that support it (all except `AcadDimOrdinate`).

Constant	Value	Description
`acTextAndArrows`	0	Text and arrowheads are placed inside the extension lines.
`acArrowsOnly`	1	Arrowheads are placed inside the extension lines. The text is placed outside the arrowheads.
`acTextOnly`	2	Text is placed inside the extension lines. The arrowheads are placed outside the extension lines.
`acBestFit`	3	This option places the text and arrowheads in the best-fit location given the space available.

AcDimFractionType

The following constants can be values for the `FractionFormat` property of the `AcadDimension` object (and all derived objects) and the `AcadTolerance` object.

Constant	Value	Description
`acHorizontal`	0	Fraction displayed as “$\frac{1}{2}$” (with a horizontal divider)
`acDiagonal`	1	Fraction displayed as “½” (with a diagonal divider)
`acNotStacked`	2	Fraction displayed as “1/2”

AcDimHorizontalJustification

The following constants can be values for the `HorizontalTextPosition` property of the dimension objects that support it (all except `AcadDimOrdinate`).

Constant	Value	Description
`acHorzCentered`	0	The text is centered along the dimension line between the extension lines.
`acFirstExtensionLine`	1	The text is next to the first extension line.
`acSecondExtensionLine`	2	The text is next to the second extension line.
`acOverFirstExtension`	3	The text is above and aligned with the first extension line.
`acOverSecondExtension`	4	The text is above and aligned with the second extension line.

AcDimLUnits

The following constants can be values for the `UnitsFormat` property of the dimension objects that support it (all except `AcadDim3PointAngular` and `AcadDimAngular`).

Constant	Value	Description
`acDimLScientific`	1	Displays measurements in scientific notation
`acDimLDecimal`	2	Displays measurements in decimal notation
`acDimLEngineering`	3	Displays measurements in feet and decimal inches
`acDimLArchitectural`	4	Displays measurements in feet, inches, and fractional inches
`acDimLFractional`	5	Displays measurements in mixed-number (integer and fractional)
`acDimLWindowsDesktop`	6	Displays measurements in Windows Desktop format

AcDimPrecision

The following constants can be values for these properties:

- The `AltTolerancePrecision` and `AltUnitsPrecision` properties of the dimension objects that support them (all except `AcadDim3PointAngular` and `AcadDimAngular`)
- The `PrimaryUnitsPrecision` property of the dimension objects that support them (all except `AcadDim3PointAngular` and `AcadDimAngular`) and the `AcadTolerance` object
- The `TextPrecision` property of the `AcadDim3PointAngular` and `AcadDimAngular` objects
- The `TolerancePrecision` property of the `AcadDimension` object (and all derived objects)

Constant	Value	Description
`acDimPrecisionZero`	0	Accurate to no decimal places (0.)
`acDimPrecisionOne`	1	Accurate to one decimal place (0.0)
`acDimPrecisionTwo`	2	Accurate to two decimal places (0.00)
`acDimPrecisionThree`	3	Accurate to three decimal places (0.000)
`acDimPrecisionFour`	4	Accurate to four decimal places (0.0000)
`acDimPrecisionFive`	5	Accurate to five decimal places (0.00000)
`acDimPrecisionSix`	6	Accurate to six decimal places (0.000000)
`acDimPrecisionSeven`	7	Accurate to seven decimal places (0.0000000)
`acDimPrecisionEight`	8	Accurate to eight decimal places (0.00000000)

AcDimTextMovement

The following constants can be values for the `TextMovement` property of the `AcadDimension` object (and all derived objects).

Constant	Value	Description
`acDimLineWithText`	0	The dimension line follows the text.
`acMoveTextAddLeader`	1	The text moves independently of the dimension line, with a leader drawn from the text to the dimension line.
`acMoveTextNoLeader`	2	The text moves independently of the dimension line, with no leader drawn from the text to the dimension line.

AcDimToleranceJustify

The following constants can be values for the `ToleranceJustification` property of the `AcadDimension` object (and all derived objects).

Constant	Value	Description
`acTolBottom`	0	Displays the bottom tolerance value at the same level as the dimension text
`acTolMiddle`	1	Displays tolerance values half-size level with the dimension text
`acTolTop`	2	Displays the top tolerance value at the same level as the dimension text

AcDimToleranceMethod

The following constants can be values for the `ToleranceDisplay` property of the `AcadDimension` object (and all derived objects).

Constant	Value	Description
acTolNone	0	No tolerance values to be displayed with dimension text.
acTolSymmetrical	1	Use this constant to display tolerances when the positive and negative values are the same (e.g., 5.0 ± 0.2).
acTolDeviation	2	Use this constant to display tolerances when the positive and negative values are not the same. The positive value is displayed over the negative value.
acTolLimits	3	This constant incorporates tolerance values into the dimension value and displays the maximum dimension over the minimum.
acTolBasic	4	Basic draws a box around the dimension text, which is often used to indicate theoretically exact dimensions.

AcDimUnits

The following constants can be values for the AltUnitsFormat property of the dimension objects that support it (all except AcadDim3PointAngular and AcadDimAngular).

Constant	Value	Description
acDimScientific	1	Displays measurements in scientific notation
acDimDecimal	2	Displays measurements in decimal notation
acDimEngineering	3	Displays measurements in feet and decimal inches
acDimArchitecturalStacked	4	Displays measurements in feet, inches, and stacked (e.g., ½) fractional inches
acDimFractionalStacked	5	Displays measurements in mixed-number notation, integers, and stacked (e.g., ½) fractions
acDimArchitectural	6	Displays measurements in feet, inches, and fractional inches
acDimFractional	7	Displays measurements in mixed-number (integer and fractional) notation
acDimWindowsDesktop	8	Displays measurements in Windows Desktop format

AcDimVerticalJustification

The following constants can be values for the VerticalTextPosition property of the AcadDimension object (and all derived objects) and the AcadLeader object.

Constant	Value	Description
acVertCentered	0	Centers the dimension text between the extension lines
acAbove	1	Places the dimension text above the dimension line except when the dimension line isn't horizontal and text inside the extension lines is forced horizontal
acOutside	2	Places the dimension text on the side of the dimension line farthest from the defining points
acJIS	3	Places the dimension text to conform to Japanese Industrial Standards (JIS)

AcDrawingAreaSCMCommand

The following constants can be values for the `SCMCommandMode` property of the `AcadPreferencesUser` object.

Constant	Value	Description
`acEnter`	0	Disables the Command shortcut menu. As a result, right-clicking in the drawing area when a command is in progress issues Enter.
`acEnableSCMOptions`	1	Enables the Command shortcut menu only when options are currently available from the command-line prompt. In a command-line prompt, options are enclosed in square brackets. If no options are available, right-clicking issues Enter.
`acEnableSCM`	2	Enables the Command shortcut menu.

AcDrawingAreaSCMDefault

The following constants can be values for the `SCMDefaultMode` property of the `AcadPreferencesUser` object.

Constant	Value	Description
`acRepeatLastCommand`	0	Disables the Default shortcut menu. As a result, right-clicking in the drawing area when no objects are selected and no commands are in progress issues Enter, which repeats the last issued command.
`acSCM`	1	Enables the Default shortcut menu.

AcDrawingAreaSCMEdit

The following constants can be values for the `SCMEditMode` property of the `AcadPreferencesUser` object.

Constant	Value	Description
`acEdRepeatLastCommand`	0	Disables the Edit shortcut menu. As a result, right-clicking in the drawing area when one or more objects are selected and no commands are in progress issues Enter, which repeats the last issued command.
`acEdSCM`	1	Enables the Edit shortcut menu.

AcDrawingDirection

The following constants can be values for the `DrawingDirection` property of the `AcadMText` object.

Constant	Value	Description
`acLeftToRight`	1	Text written from left to right
`acRightToLeft`	2	Reserved for future use
`acTopToBottom`	3	Text written from top to bottom
`acBottomToTop`	4	Reserved for future use
`acByStyle`	5	Reserved for future use

AcDynamicBlockReferenceProperty

The following constants can be values for the `UnitsType` property of an `AcadDynamicBlockReferenceProperty` object.

Constant	Value
`acAngular`	1
`acArea`	3
`acDistance`	2
`acNoUnits`	0

AcEntityName

The following constants can be values for the `EntityName` property of the `AcadEntity` object.

Constant	Value	Description
`ac3dFace`	1	`Acad3DFace` object
`ac3dPolyline`	2	`Acad3DPolyline` object
`ac3dSolid`	3	`Acad3DSolid` object
`acArc`	4	`AcadArc` object
`acAttribute`	5	`AcadAttribute` object
`acAttributeReference`	6	`AcadAttributeReference` object
`acBlockReference`	7	`AcadBlockReference` object
`acCircle`	8	`AcadCircle` object
`acDimAligned`	9	`AcadDimAligned` object
`acDimAngular`	10	`AcadDimAngular` object
`acDimDiametric`	12	`AcadDimDiametric` object

Continued

Constant	Value	Description
acDimOrdinate	13	AcadDimOrdinate object
acDimRadial	14	AcadDimRadial object
acDimRotated	15	AcadDimRotated object
acEllipse	16	AcadEllipse object
acHatch	17	AcadHatch object
acLeader	18	AcadLeader object
acLine	19	AcadLine object
acMtext	21	AcadMText object
acPoint	22	AcadPoint object
acPolyline	23	AcadPolyline object
acPolylineLight	24	AcadLWPolyline object
acPolymesh	25	AcadPolygonMesh object
acRaster	26	AcadRasterImage object
acRay	27	AcadRay object
acRegion	28	AcadRegion object
acShape	29	AcadShape object
acSolid	30	AcadSolid object
acSpline	31	AcadSpline object
acText	32	AcadText object
acTolerance	33	AcadTolerance object
acTrace	34	AcadTrace object
acPViewport	35	AcadPViewport object
acXline	36	AcadXline object
acGroup	37	AcadGroup object
acMInsertBlock	38	AcadMInsertBlock object
acPolyfaceMesh	39	AcadPolyfaceMesh object
acMLine	40	AcadMLine object
acDim3PointAngular	41	AcadDim3PointAngular object
acExternalReference	42	AcadExternalReference object
acTable	43	AcadTable object
acDimArcLength	44	AcadDimArcLength object
acDimRadialLarge	45	AcadDimRadialLarge object

AcExtendOption

The following constants can be supplied as the value for the third parameter of the IntersectWith method of the AcadEntity object and those derived objects that support it (all except AcadPolygonMesh and AcadPViewport).

Constant	Value	Description
acExtendNone	0	Doesn't extend either object
acExtendThisEntity	1	Extends the base object
acExtendOtherEntity	2	Extends the object passed as an argument
acExtendBoth	3	Extends both objects

AcGradientPatternType

The following constants can be values for the HatchObjectType property of the Hatch object. They may also be used as the PatternType parameter value of an AddHatch method.

Constant	Value
acPredefinedGradient	0
acUserDefinedGradient	1

AcGridLineType

The following can be values for the GridLineType parameter of the GetGridColor, GetGridLineWeight, and GetGridVisibility methods of an AcadTable or AcadTableStyle object. Also, they can be values for the GridLineTypes parameter of the SetGridColor, SetGridLineWeight, and SetGridVisibility methods of an AcadTable or AcadTableStyle object.

Constant	Value	Description
acHorzBottom	4	Top or bottom horizontal grid line, based on the flow direction
acHorzInside	2	All horizontal grid lines, excluding the top and bottom lines
acHorzTop	1	Top or bottom horizontal grid line, based on the flow direction
acInvalidGridLine	0	An invalid grid line
acVertInside	16	All the vertical grid lines, excluding the farthest-left and farthest-right grid lines
acVertLeft	8	Farthest-left grid line
acVertRight	32	Farthest-right grid line

AcHatchObjectType

These constants specify the type of Hatch.

Constant	Value	Description
acHatchObject	0	Classic
acGradientObject	1	Gradient

AcHatchStyle

The following constants can be values for the `HatchStyle` property of the `AcadHatch` object.

Constant	Value	Description
`acHatchStyleNormal`	0	Specifies standard style or normal. This option hatches inward from the outermost area boundary. If AutoCAD encounters an internal boundary, it turns off hatching until it encounters another boundary.
`acHatchStyleOuter`	1	Fills the outermost areas only. This style also hatches inward from the area boundary, but it turns off hatching if it encounters an internal boundary and doesn't turn it back on again.
`acHatchStyleIgnore`	2	Ignores internal structure. This option hatches through all internal objects.

AcHorizontalAlignment

The following constants can be values for the `HorizontalAlignment` property of the `AcadAttribute`, `AcadAttributeReference`, and `AcadText` objects.

Constant	Value	Description
`acHorizontalAlignmentLeft`	0	Aligns the attribute, attribute reference, or text at the left, the bottom edge of the text being on the baseline.
`acHorizontalAlignmentCenter`	1	Aligns the attribute, attribute reference, or text at the center, the bottom edge of the text being on the baseline.
`acHorizontalAlignmentRight`	2	Aligns the attribute, attribute reference, or text at the right, the bottom edge of the text being on the baseline.
`acHorizontalAlignmentAligned`	3	The attribute, attribute reference, or text is aligned according to the points held in the `InsertionPoint` and `TextAlignmentPoint` properties.
`acHorizontalAlignmentMiddle`	4	Aligns text at the middle. This isn't the same as `acHorizontalAlignmentCenter`, because the middle is defined as the midpoint between the top and bottom limits, the bottom referring to a position below the baseline.
`acHorizontalAlignmentFit`	5	The attribute, attribute reference, or text is aligned to fit to the points held in the `InsertionPoint` and `TextAlignmentPoint` properties.

AcInsertUnits

The following constants can be values for the `ADCInsertUnitsDefaultSource` and `ADCInsertUnitsDefaultTarget` properties of the `AcadPreferencesUser` object.

Constant	Value	Description
`acInsertUnitsUnitless`	0	No units
`acInsertUnitsInches`	1	Inches
`acInsertUnitsFeet`	2	Feet
`acInsertUnitsMiles`	3	Miles
`acInsertUnitsMillimeters`	4	Millimeters
`acInsertUnitsCentimeters`	5	Centimeters
`acInsertUnitsMeters`	6	Meters
`acInsertUnitsKilometers`	7	Kilometers
`acInsertUnitsMicroinches`	8	Microinches
`acInsertUnitsMils`	9	Mils
`acInsertUnitsYards`	10	Yards
`acInsertUnitsAngstroms`	11	Angstrom units
`acInsertUnitsNanometers`	12	Nanometers
`acInsertUnitsMicrons`	13	Microns (micrometers)
`acInsertUnitsDecimeters`	14	Decimeters
`acInsertUnitsDecameters`	15	Decameters
`acInsertUnitsHectometers`	16	Hectometers
`acInsertUnitsGigameters`	17	Gigameters
`acInsertUnitsAstronomicalUnits`	18	Astronomical units
`acInsertUnitsLightYears`	19	Light years
`acInsertUnitsParsecs`	20	Parsecs

AcISOPenWidth

The following constants can be values for the `ISOPenWidth` property of the `AcadHatch` object.

Constant	Value	Description
`acPenWidthUnk`	–1	Unknown or nonstandard pen width
`acPenWidth013`	13	13/100th of a millimeter
`acPenWidth018`	18	18/100th of a millimeter
`acPenWidth025`	25	25/100th of a millimeter
`acPenWidth035`	35	35/100th of a millimeter
`acPenWidth050`	50	50/100th of a millimeter
`acPenWidth070`	70	70/100th of a millimeter
`acPenWidth100`	100	1 millimeter
`acPenWidth140`	140	1.4 millimeters
`acPenWidth200`	200	2 millimeters

AcKeyboardAccelerator

The following constants can be values for the KeyboardAccelerator property of the AcadPreferencesUser object.

Constant	Value	Description
acPreferenceClassic	0	Uses the AutoCAD classic keyboard
acPreferenceCustom	1	Uses the Windows standard keyboard

AcKeyboardPriority

The following constants can be values for the KeyboardPriority property of the AcadPreferencesUser object.

Constant	Value	Description
acKeyboardRunningObjSnap	0	Object snaps are strictly adhered to when typing in coordinates.
acKeyboardEntry	1	Keyboard entry values are strictly adhered to when typing in coordinates.
acKeyboardEntryExceptScripts	2	Keyboard entry values are strictly adhered to when typing in coordinates. However, when coordinates are entered through a script, running object snaps are adhered to.

AcLeaderType

The following constants can be values for the Type property of the AcadLeader object.

Constant	Value	Description
acLineNoArrow	0	A line with no arrow
acSplineNoArrow	1	A spline with no arrow
acLineWithArrow	2	A line with an arrow
acSplineWithArrow	3	A spline with an arrow

AcLineSpacingStyle

The following constants can be values for the LineSpacingStyle property of the AcadMText object.

Constant	Value	Description
acLineSpacingStyleAtLeast	1	Allows the spacing between different lines of text to adjust automatically, based on the height of the largest character in a line of text
acLineSpacingStyleExactly	2	Forces the line spacing to be the same size for all the lines in the text regardless of formatting overrides

AcLineWeight

The following constants can be values for these properties:

- The `Lineweight` property of the `AcadEntity` object (and all derived objects), the `AcadDatabasePreferences`, `AcadGroup`, and `AcadLayer` objects
- The `DimensionLineweight` property of the dimension objects that support it (all except `AcadDimOrdinate`) and the `AcadLeader` object
- The `ExtensionLineweight` property of the dimension objects that support it (all except `AcadDimDiametric`, `AcadDimOrdinate`, and `AcadDimRadial`)

Constant	Value	Description
`acLnWtByLwDefault`	–3	Draws lines in the default lineweight.
`acLnWtByBlock`	–2	Draws lines in the lineweight specified for the block (initial value).
`acLnWtByLayer`	–1	Draws lines in the lineweight specified for the layer.
`acLnWt000`	0	Plots with the thinnest lineweight available for the specified plotting device. In model space, this is set with a width of 1 pixel.
`acLnWt005`	5	5⁄100 millimeter.
`acLnWt009`	9	9⁄100 millimeter or ¼ point.
`acLnWt013`	13	13⁄100 millimeter or 5⁄1000 inch.
`acLnWt015`	15	15⁄100 millimeter.
`acLnWt018`	18	18⁄100 millimeter, ½ point, or Pen Size 0000.
`acLnWt020`	20	2⁄10 millimeter or Pen Size 000.
`acLnWt025`	25	¼ millimeter, 1⁄100 inch, or ¾ point.
`acLnWt030`	30	3⁄10 millimeter or Pen Size 00.
`acLnWt035`	35	35⁄100 millimeter, 1 point, or Pen Size 0.
`acLnWt040`	40	4⁄10 millimeter.
`acLnWt050`	50	½ millimeter or Pen Size 1
`acLnWt053`	53	53⁄100 millimeter or 1½ points.
`acLnWt060`	60	6⁄10 millimeter or Pen Size 2.
`acLnWt070`	70	7⁄10 millimeter, 2¼ point, or Pen Size 2½.
`acLnWt080`	80	8⁄10 millimeter or Pen Size 3.
`acLnWt090`	90	9⁄10 millimeter.
`acLnWt100`	100	1 millimeter or Pen Size 3½.
`acLnWt106`	106	3 point.
`acLnWt120`	120	1.2 millimeters or Pen Size 4.
`acLnWt140`	140	1.4 millimeters.
`acLnWt158`	158	4¼ point.
`acLnWt200`	200	2 millimeters.
`acLnWt211`	211	6 point.

AcLoopType

The following constants can be supplied as the value for the second parameter of the `InsertLoopAt` method of the `AcadHatch` object.

Constant	Value	Description
`acHatchLoopTypeDefault`	0	The loop type hasn't been specified.
`acHatchLoopTypeExternal`	1	The loop consists of external entities.
`acHatchLoopTypePolyline`	2	The hatch loop consists of a polyline.
`acHatchLoopTypeDerived`	4	The loop was derived by AutoCAD's boundary tracer from a picked point.
`acHatchLoopTypeTextbox`	8	The loop consists of a box around an existing text object.

AcMenuFileType

The following constants can be supplied as the value for the parameter of the `Save` method of the `AcadMenuGroup` object or the second parameter for the property of the `SaveAs` method of the `AcadDocument` and `AcadMenuGroup` objects.

Constant	Value	Description
`acMenuFileCompiled`	0	A compiled menu file (`.mnc` file type)
`acMenuFileSource`	1	A source menu file (`.mns` file type)

AcMenuGroupType

The following constants can be values for the `Type` property of the `AcadMenuGroup` object.

Constant	Value	Description
`acBaseMenuGroup`	0	A base menu group
`acPartialMenuGroup`	1	A partial menu group

AcMenuItemType

The following constants can be values for the `Type` property of the `AcadPopupMenuItem` object.

Constant	Value	Description
acMenuItem	0	Specifies a pop-up menu item
acMenuSeparator	1	Specifies a separator in a pop-up menu
acMenuSubMenu	2	Specifies a pop-up submenu

AcMLineJustification

The following constants can be values for the `Justification` property of an `AcadMLine` object. To access this property, use the `IAcadMLine2` interface.

Constant	Value
`acBottom`	2
`acTop`	0
`acZero`	1

AcOleQuality

The following constants can be values for the `OleQuality` property of the `AcadPreferencesOutput` object.

Constant	Value	Description
`acOQLineArt`	0	Specifies line art quality (e.g., for an embedded spreadsheet)
`acOQText`	1	Specifies text quality (e.g., for an embedded Word document)
`acOQGraphics`	2	Specifies graphics quality (e.g., for an embedded pie chart)
`acOQPhoto`	3	Specifies photograph quality
`acOQHighPhoto`	4	High-quality photograph

AcPatternType

The following constants can be values for the `PatternType` property of the `AcadHatch` object, and they can also be supplied as the value for the second parameter of the `AddHatch` method of the `AcadBlock`, `AcadModelSpace`, and `AcadPaperSpace` objects and the `SetPattern` method of the `AcadHatch` object.

Constant	Value	Description `SetPattern` method
`acHatchPatternTypeUserDefined`	0	Defines a pattern of lines using the current linetype
`acHatchPatternTypePreDefined`	1	Selects the pattern name from those defined in the `acad.pat` file
`acHatchPatternTypeCustomDefined`	2	Selects the pattern name from a `.pat` file other than the `acad.pat` file

AcPlotPaperUnits

The following constants can be values for the `PaperUnits` property of the `AcadLayout` and `AcadPlotConfiguration` objects.

Constant	Value	Description
`acInches`	0	Units specified in inches
`acMillimeters`	1	Units specified in millimeters
`acPixels`	2	Units specified in pixels

AcPlotPolicy

The following constants can be values for the `PlotPolicy` property of the `AcadPreferencesOutput` object.

Constant	Value	Description
`acPolicyNamed`	0	No association is made between color and plot-style name. Plot-style name for each object is set to the default defined in the `DefaultPlotStyleForObjects` property.
`acPolicyLegacy`	1	An object's plot-style name is associated with its color per the naming convention ACI*x*, where *x* is the color number of the object according to the AutoCAD color index.

AcPlotRotation

The following constants can be values for the `PlotRotation` property of the `AcadLayout` and `AcadPlotConfiguration` objects.

Constant	Value	Description
`ac0degrees`	0	These values are read counterclockwise relative to the X-axis.
`ac90degrees`	1	
`ac180degrees`	2	
`ac270degrees`	3	

AcPlotScale

The following constants can be values for the `StandardScale` property of the `AcadLayout` and `AcadPlotConfiguration` objects.

Constant	Value	Description
acScaleToFit	0	Scale to fit
ac1_128in_1ft	1	1/128″ : 1′
ac1_64in_1ft	2	1/64″ : 1′
ac1_32in_1ft	3	1/32″ : 1′
ac1_16in_1ft	4	1/16″ : 1′
ac3_32in_1ft	5	3/32″ : 1′
ac1_8in_1ft	6	1/8″ : 1′
ac3_16in_1ft	7	3/16″ : 1′
ac1_4in_1ft	8	1/4″ : 1′
ac3_8in_1ft	9	3/8″ : 1′
ac1_2in_1ft	10	1/2″ : 1′
ac3_4in_1ft	11	3/4″ : 1′
ac1in_1ft	12	1″ : 1′
ac3in_1ft	13	3″ : 1′
ac6in_1ft	14	6″ : 1′
ac1ft_1ft	15	1′ : 1′
ac1_1	16	1 : 1
ac1_2	17	1 : 2
ac1_4	18	1 : 4
ac1_8	19	1 : 8
ac1_10	20	1: 10
ac1_16	21	1: 16
ac1_20	22	1 : 20
ac1_30	23	1 : 30
ac1_40	24	1 : 40
ac1_50	25	1 : 50
ac1_100	26	1: 100
ac2_1	27	2 : 1
ac4_1	28	4 : 1
ac8_1	29	8 : 1
ac10_1	30	10 : 1
ac100_1	31	100 : 1

AcPlotType

The following constants can be values for the `PlotType` property of the `AcadLayout` and `AcadPlotConfiguration` objects.

Constant	Value	Description
`acDisplay`	0	Prints everything that is in the current display.
`acExtents`	1	Prints everything that falls within the extents of the currently selected space.
`acLimits`	2	Prints everything that is in the limits of the current space.
`acView`	3	Prints the view named by the `ViewToPlot` property.
`acWindow`	4	Prints everything in the window specified by the `SetWindowToPlot` method.
`acLayout`	5	Prints everything that falls within the margins of the specified paper size with the origin being calculated from the 0,0 coordinate location in the Layout tab. This option isn't available when printing from model space.

AcPolylineType

The following constants can be values for the `Type` property of the `AcadPolyline` object.

Constant	Value	Description
`acSimplePoly`	0	Specifies a simple polyline
`acFitCurvePoly`	1	Specifies a fit curve polyline
`acQuadSplinePoly`	2	Specifies a quadratic B-spline polyline
`acCubicSplinePoly`	3	Specifies a cubic B-spline polyline

AcPolymeshType

The following constants can be values for the `Type` property of the `AcadPolygonMesh` object.

Constant	Value	Description
`acSimpleMesh`	0	Specifies a simple mesh with no surface fitting or smoothing
`acQuadSurfaceMesh`	5	Specifies a quadratic B-spline surface fit
`acCubicSurfaceMesh`	6	Specifies a cubic B-spline surface fit
`acBezierSurfaceMesh`	8	Specifies a Bezier surface fit

AcPreviewMode

The following constants can be values for the `DisplayPlotPreview` property of the `AcadPlot` object.

Constant	Value	Description
`acPartialPreview`	0	Shows an accurate representation of the effective plot area relative to the paper size. The final location of the affected area on the paper depends on the plotter.
`acFullPreview`	1	Displays the drawing on the screen as it will appear when plotted on paper. Because this requires a regeneration of the drawing, it's slower than a partial preview. It's faster than the normal plot regeneration because AutoCAD performs no vector sorting or optimization.

AcPrinterSpoolAlert

The following constants can be values for the `PrinterSpoolAlert` property of the `AcadPreferencesOutput` object.

Constant	Value	Description
`acPrinterAlwaysAlert`	0	Always alert and log errors.
`acPrinterAlertOnce`	1	Alert only the first error, and log all errors.
`acPrinterNeverAlertLogOnce`	2	Never alert, but log all errors.
`acPrinterNeverAlert`	3	Never alert, and don't log any errors.

AcProxyImage

The following constants can be values for the `ProxyImage` property of the `AcadPreferencesOpenSave` object.

Constant	Value	Description
`acProxyNotShow`	0	Don't show the proxy image.
`acProxyShow`	1	Show the proxy image.
`acProxyBoundingBox`	2	Show the proxy image in a bounding box.

AcRegenType

The following constants can be supplied as the value for the parameter of the `Regen` method of the `AcadDocument` object.

Constant	Value	Description
`acActiveViewport`	0	Regenerates drawing in active viewport only
`acAllViewports`	1	Regenerates drawing in all viewports

AcRotationAngle

The following constants can be values for the TextRotation parameter of the SetTextRotation property of an AcadTable object. Also, they can be return values of the GetTextRotation method of an AcadTable object.

Constant	Value
acDegrees000	0
acDegrees090	1
acDegrees180	2
acDegrees270	3
acDegreesUnknown	–1

AcRowType

The following constants can be values according to the following list.

Method	Parameter	Object(s)
GetAlignment	RowType	AcadTable, AcadTableStyle
GetBackgroundColor	RowType	AcadTable, AcadTableStyle
GetBackgroundColorNone	RowType	AcadTable, AcadTableStyle
GetColor	RowType	AcadTableStyle
GetGridColor	RowType	AcadTable, AcadTableStyle
GetGridLineWeight	RowType	AcadTable, AcadTableStyle
GetGridVisibility	RowType	AcadTable, AcadTableStyle
GetTextHeight	RowType	AcadTable, AcadTableStyle
GetTextStyle	RowType	AcadTable, AcadTableStyle
GetContentColor	RowType	AcadTable
SetGridColor	RowTypes	AcadTable, AcadTableStyle
SetGridLineWeight	RowTypes	AcadTable, AcadTableStyle
SetGridVisibility	RowTypes	AcadTable, AcadTableStyle

Also, they can be return values of the GetRowType method of an AcadTable object.

Constant	Value
AcUnknown	0
acDataRow	1
acTitleRow	2
acHeaderRow	4

AcSaveAsType

The following constants can be values for the SaveAsType property of the AcadPreferencesOpenSave object.

Constant	Value	Description
acUnknown	–1	This constant is read-only. The drawing type is unknown or invalid.
acR12_dxf	1	AutoCAD Release12/LT2 DXF (*.dxf).
acR13_dwg	4	AutoCAD Release13/LT95 DWG (*.dwg).
acR13_dxf	5	AutoCAD Release13/LT95 DXF (*.dxf).
acR14_dwg	8	AutoCAD Release14/LT97 DWG (*.dwg).
acR14_dxf	9	AutoCAD Release14/LT97 DXF (*.dxf).
acR15_dwg	12	AutoCAD 2000 DWG (*.dwg).
ac2000_dwg	12	AutoCAD 2000 DWG (*.dwg).
acR15_dxf	13	AutoCAD 2000 DXF (*.dxf).
ac2000_dxf	13	AutoCAD 2000 DXF (*.dxf).
acR15_Template	14	AutoCAD 2000 Drawing Template File (*.dwt).
ac2000_Template	14	AutoCAD 2000 Drawing Template File (*.dwt).
acR18	24	AutoCAD 2004 DWG (*.dwg)
ac2004_dwg	24	AutoCAD 2004 DWG (*.dwg)
acNative	24	A synonym for the current drawing release format. If you want your application to save the drawing in the format of whatever version of AutoCAD the application is running on, then use the acNative format.
acR18_dxf	25	AutoCAD 2004 DXF (*.dxf)
ac2004_dxf	25	AutoCAD 2004 DXF (*.dxf)
acR18_Template	26	AutoCAD 2004 Drawing Template File (*.dwt).
ac2004_Template	26	AutoCAD 2004 Drawing Template File (*.dwt).

■ **Note** The acR13_dwg, acR13_dxf, acR14_dwg, and acR14_dxf options aren't available in AutoCAD 2006.

AcSelect

The following constants can be supplied as the value for the first parameter in either of the Select or SelectByPolygon methods of the AcadSelectionSet object.

Constant	Value	Description
acSelectionSetWindow	0	Selects all entities completely inside a rectangular area whose corners are defined by Point1 and Point2 (Select method only).
acSelectionSetCrossing	1	Selects objects within and crossing a rectangular area whose corners are defined by Point1 and Point2 (Select method only).
acSelectionSetFence	2	Selects all entities crossing a selection fence. The fence is defined by coordinates in PointList. The fence mode is similar to CrossingPolygon mode except that AutoCAD doesn't close the last vector of the fence, and a fence can cross itself. The fence isn't affected by the PICKADD system variable (SelectByPolygon method only).
acSelectionSetPrevious	3	Selects the most recent selection set. This mode is ignored if you switch between paper and model space and attempt to use the selection set (Select method only).
acSelectionSetLast	4	Selects the most recently created visible entity (Select method only).
acSelectionSetAll	5	Selects all entities (Select method only).
acSelectionSetWindowPolygon	6	Selects objects within a polygon defined by PointList (SelectByPolygon method only).
acSelectionSetCrossingPolygon	7	Selects objects within and crossing an area defined by a polygon. Use PointList to define the coordinates of the polygon. AutoCAD will close the last vector of the polygon. A polygon definition can't cross itself (SelectByPolygon method only).

AcSelectType

The following constants can be values for the SelType parameter of the SelectSubRegion method of an AcadTable object.

Constant	Value	Description
acTableSelectCrossing	2	Cross-selection type
acTableSelectWindow	1	Window-selection type

AcShadePlot

The following constants can be values for the ShadePlot property of an Pviewport object.

Constant	Value	Description
acShadePlotAsDisplayed	0	Model space view plots the same way it is displayed.
acShadePlotWireframe	1	Model space view plots as wireframe regardless of display.
acShadePlotHidden	2	Model space view plots with hidden lines removed, regardless of display.
acShadePlotRendered	3	Model space view plots as rendered regardless of display.

AcTableDirection

The following constants can be values for the `FlowDirection` property of an `AcadTable` and `AcadTableStyle` object.

Constant	Value	Description
`acTableBottomToTop`	1	Table flow direction of bottom to top
`acTableTopToBottom`	0	Table flow direction of top to bottom

AcTableStyleOverrides

The following constants can be values for `TableStyleOverrides` property of an `AcadTable` object.

Constant	Value	Description
`acTitleSuppressed`	1	Indicates title-suppressed table style override for table object
`acHeaderSuppressed`	2	Indicates header-suppressed table style override for table object
`acFlowDirection`	3	Indicates flow direction table style override for table object
`acHorzCellMargin`	4	Indicates horizontal cell margin table style override for table object
`acVertCellMargin`	5	Indicates vertical cell margin table style override for table object
`acTitleRowColor`	6	Indicates title row color table style override for table object
`acHeaderRowColor`	7	Indicates header row color table style override for table object
`acDataRowColor`	8	Indicates data row color table style override for table object
`acTitleRowFillNone`	9	Indicates title row no fill table style override for table object
`acHeaderRowFillNone`	10	Indicates header row no fill table style override for table object
`acDataRowFillNone`	11	Indicates data row no fill table style override for table object
`acTitleRowFillColor`	12	Indicates title row fill color table style override for table object
`acHeaderRowFillColor`	13	Indicates header row fill color table style override for table object
`acDataRowFillColor`	14	Indicates data row fill color table style override for table object
`acTitleRowAlignment`	15	Indicates title row alignment table style override for table object
`acHeaderRowAlignment`	16	Indicates header row alignment table style override for table object

Continued

Constant	Value	Description
acDataRowAlignment	17	Indicates data row alignment table style override for table object
acTitleRowTextStyle	18	Indicates title row text style table style override for table object
acHeaderRowTextStyle	19	Indicates header row text style table style override for table object
acDataRowTextStyle	20	Indicates data row text style table style override for table object
acTitleRowTextHeight	21	Indicates title row text height table style override for table object
acHeaderRowTextHeight	22	Indicates header row text height table style override for table object
acDataRowTextHeight	23	Indicates data row text height table style override for table object
acTitleHorzTopColor	40	Indicates title row horizontal top gridline color table style override for table object
acTitleHorzInsideColor	41	Indicates title row horizontal inside gridline color table style override for table object
acTitleHorzBottomColor	42	Indicates title row horizontal bottom gridline color table style override for table object
acTitleVertLeftColor	43	Indicates title row vertical left gridline color table style override for table object
acTitleVertInsideColor	44	Indicates title row vertical inside gridline color table style override for table object
acTitleVertRightColor	45	Indicates title row vertical right gridline color table style override for table object
acHeaderHorzTopColor	46	Indicates header row horizontal top gridline color table style override for table object
acHeaderHorzInsideColor	47	Indicates header row horizontal inside gridline color table style override for table object
acHeaderHorzBottomColor	48	Indicates header row horizontal bottom gridline color table style override for table object
acHeaderVertLeftColor	49	Indicates header row vertical left gridline color table style override for table object
acHeaderVertInsideColor	50	Indicates header row vertical inside gridline color table style override for table object
acHeaderVertRightColor	51	Indicates header row vertical right gridline color table style override for table object
acDataHorzTopColor	52	Indicates data row horizontal top gridline color table style override for table object
acDataHorzInsideColor	53	Indicates data row horizontal inside gridline color table style override for table object
acDataHorzBottomColor	54	Indicates data row horizontal bottom gridline color table style override for table object
acDataVertLeftColor	55	Indicates data row vertical left gridline color table style override for table object

Constant	Value	Description
acDataVertInsideColor	56	Indicates data row vertical inside gridline color table style override for table object
acDataVertRightColor	57	Indicates data row vertical right gridline color table style override for table object
acTitleHorzTopLineWeight	70	Indicates title row horizontal top gridline lineweight table style override for table object
acTitleHorzInsideLineWeight	71	Indicates title row horizontal inside gridline lineweight table style override for table object
acTitleHorzBottomLineWeight	72	Indicates title row horizontal bottom gridline lineweight table style override for table object
acTitleVertLeftLineWeight	73	Indicates title row vertical left gridline lineweight table style override for table object
acTitleVertInsideLineWeight	74	Indicates title row vertical inside gridline lineweight table style override for table object
acTitleVertRightLineWeight	75	Indicates title row vertical right gridline lineweight table style override for table object
acHeaderHorzTopLineWeight	76	Indicates header row horizontal top gridline lineweight table style override for table object
acHeaderHorzInsideLineWeight	77	Indicates header row horizontal inside gridline lineweight table style override for table object
acHeaderHorzBottomLineWeight	78	Indicates header row horizontal bottom gridline lineweight table style override for table object
acHeaderVertLeftLineWeight	79	Indicates header row vertical left gridline lineweight table style override for table object
acHeaderVertInsideLineWeight	80	Indicates header row vertical inside gridline lineweight table style override for table object
acHeaderVertRightLineWeight	81	Indicates header row vertical right gridline lineweight table style override for table object
acDataHorzTopLineWeight	82	Indicates data row horizontal top gridline lineweight table style override for table object
acDataHorzInsideLineWeight	83	Indicates data row horizontal inside gridline lineweight table style override for table object
acDataHorzBottomLineWeight	84	Indicates data row horizontal bottom gridline lineweight table style override for table object
acDataVertLeftLineWeight	85	Indicates data row vertical left gridline lineweight table style override for table object
acDataVertInsideLineWeight	86	Indicates data row vertical inside gridline lineweight table style override for table object
acDataVertRightLineWeight	87	Indicates data row vertical right gridline lineweight table style override for table object
acTitleHorzTopVisibility	100	Indicates title row horizontal top gridline visibility table style override for table object
acTitleHorzInsideVisibility	101	Indicates title row horizontal inside gridline visibility table style override for table object

Continued

Constant	Value	Description
`acTitleHorzBottomVisibility`	102	Indicates title row horizontal bottom gridline visibility table style override for table object
`acTitleVertLeftVisibility`	103	Indicates title row vertical left gridline visibility table style override for table object
`acTitleVertInsideVisibility`	104	Indicates title row vertical inside gridline visibility table style override
`acTitleVertRightVisibility`	105	Indicates title row vertical right gridline visibility table style override for table object
`acHeaderHorzTopVisibility`	106	Indicates header row horizontal top gridline visibility table style override for table object
`acHeaderHorzInsideVisibility`	107	Indicates header row horizontal inside gridline visibility table style override for table object
`acHeaderHorzBottomVisibility`	108	Indicates header row horizontal bottom gridline visibility table style override for table object
`acHeaderVertLeftVisibility`	109	Indicates header row vertical left gridline visibility table style override for table object
`acHeaderVertInsideVisibility`	110	Indicates header row vertical inside gridline visibility table style override for table object
`acHeaderVertRightVisibility`	111	Indicates header row vertical right gridline visibility table style override for table object
`acDataHorzTopVisibility`	112	Indicates data row horizontal top gridline visibility table style override for table object
`acDataHorzInsideVisibility`	113	Indicates data row horizontal inside gridline visibility table style override for table object
`acDataHorzBottomVisibility`	114	Indicates data row horizontal bottom gridline visibility table style override for table object
`acDataVertLeftVisibility`	115	Indicates data row vertical left gridline visibility table style override for table object
`acDataVertInsideVisibility`	116	Indicates data row vertical inside gridline visibility table style override for table object
`acDataVertRightVisibility`	117	Indicates data row vertical right gridline visibility table style override for table object
`acCellAlignment`	130	Indicates alignment table style override for table cell
`acCellBackgroundFillNone`	131	Indicates background no fill table style override for table cell
`acCellBackgroundColor`	132	Indicates background color table style override for table cell
`acCellContentColor`	133	Indicates content color table style override for table cell
`acCellTextStyle`	134	Indicates text style table style override for table cell
`acCellTextHeight`	135	Indicates text height table style override for table cell
`acCellTopGridColor`	136	Indicates top-edge grid color table style override for table cell

Constant	Value	Description
acCellRightGridColor	137	Indicates right-edge grid color table style override for table cell
acCellBottomGridColor	138	Indicates bottom-edge grid color table style override for table cell
acCellLeftGridColor	139	Indicates left-edge grid color table style override for table cell
acCellTopGridLineWeight	140	Indicates top-edge grid lineweight table style override for table cell
acCellRightGridLineWeight	141	Indicates right-edge grid lineweight table style override for table cell
acCellBottomGridLineWeight	142	Indicates bottom-edge grid lineweight table style override for table cell
acCellLeftGridLineWeight	143	Indicates left-edge grid lineweight table style override for table cell
acCellTopVisibility	144	Indicates top-edge grid visibility table style override for table cell
acCellRightVisibility	145	Indicates right-edge grid visibility table style override for table cell
acCellBottomVisibility	146	Indicates bottom-edge grid visibility table style override for table cell
acCellLeftVisibility	147	Indicates left-edge grid visibility table style override for table cell

AcTextFontStyle

The following constants and their combinations can be values for the `TextFontStyle` property of the `AcadPreferencesDisplay` object.

Constant	Value	Description
acFontRegular	0	Specifies a normal font style
acFontItalic	1	Specifies italic style
acFontBold	2	Specifies bold style
acFontBoldItalic	3	Specifies both bold and italic styles

AcTextGenerationFlag

The following constants can be values for the `TextGenerationFlag` property of the `AcadAttribute`, `AcadAttributeReference`, and `AcadText` objects.

Constant	Value	Description
acTextFlagBackward	2	Text to be displayed backward
acTextFlagUpsideDown	4	Text to be displayed upside-down

AcToolbarDockStatus

The following constants can be values for the `DockStatus` property of the `AcadToolbar` object, or they can be supplied as the value for the parameter of the `Dock` method of the `AcadToolbar` object.

Constant	Value	Description
`acToolbarDockTop`	0	Toolbar docked at the top of the window
`acToolbarDockBottom`	1	Toolbar docked at the bottom of the window
`acToolbarDockLeft`	2	Toolbar docked at the left of the window
`acToolbarDockRight`	3	Toolbar docked at the right of the window
`acToolbarFloating`	4	Toolbar not docked at all

AcToolbarItemType

The following constants can be values for the `Type` property of the `AcadToolbarItem` object.

Constant	Value	Description
`acToolbarButton`	0	Specifies a generic button
`acToolbarSeparator`	1	Specifies a toolbar separator
`acToolbarControl`	2	Specifies a control button
`acToolbarFlyout`	3	Specifies a flyout button

AcUnits

The following constants can be supplied as the value for the second parameter of the `DistanceToReal` and `RealToString` methods of the `AcadUtility` object.

Constant	Value	Description
`acDefaultUnits`	–1	Displays measurements in default units
`acScientific`	1	Displays measurements in scientific notation
`acDecimal`	2	Displays measurements in decimal notation
`acEngineering`	3	Displays measurements in feet and decimal inches
`acArchitectural`	4	Displays measurements in feet, inches, and fractional inches
`acFractional`	5	Displays measurements in mixed-number (integer and fractional) notation

AcVerticalAlignment

The following constants can be values for the `VerticalAlignment` property of the `AcadAttribute`, `AcadAttributeReference`, and `AcadText` objects.

Constant	Value	Description
`acVerticalAlignmentBaseline`	0	Align on the baseline. Note that the baseline isn't the same as the bottom. The baseline is defined at a level partway between the bottom position and the middle position.
`acVerticalAlignmentBottom`	1	Align at the bottom.
`acVerticalAlignmentMiddle`	2	Align at the middle. This is midway between the bottom and the top positions.
`acVerticalAlignmentTop`	3	Align at the top.

AcViewportScale

The following constants can be values for the `StandardScale` property of the `AcadPViewport` object.

While these constants are still accepted as parameters for the `StandardScale` property of the `AcadPViewport` object (and are listed in AutoCAD's VBA Help file), they are not the constants shown in the VBA Object Browser. Also, they are not shown in the `AutoListMembers` tool. Those constants are named `acVp1` through `acVp100`, and their value is 1 greater than the numeral of their name. For values 1 through 29, the description scale corresponds to the scale listed.

Constant	Value	Description
`acVpScaleToFit`	0	Scale to fit
`acVpCustomScale`	1	Custom scale
`acVp1_1`	2	1 : 1
`acVp1_2`	3	1 : 2
`acVp1_4`	4	1 : 4
`acVp1_8`	5	1 : 8
`acVp1_10`	6	1: 10
`acVp1_16`	7	1: 16
`acVp1_20`	8	1 : 20
`acVp1_30`	9	1 : 30
`acVp1_40`	10	1 : 40
`acVp1_50`	11	1 : 50
`acVp1_100`	12	1: 100
`acVp2_1`	13	2 : 1
`acVp4_1`	14	4 : 1
`acVp8_1`	15	8 : 1

Continued

Constant	Value	Description
acVp10_1	16	10 : 1
acVp100_1	17	100 : 1
acVp1_128in_1ft	18	1⁄128″ : 1′
acVp1_64in_1ft	19	1⁄64″ : 1′
acVp1_32in_1ft	20	1⁄32″ : 1′
acVp1_16in_1ft	21	1⁄16″ : 1′
acVp1_32in_1ft	22	3⁄32″ : 1′
acVp1_8in_1ft	23	1⁄8″ : 1′
acVp1_16in_1ft	24	3⁄16″ : 1′
acVp1_4in_1ft	25	1⁄4″ : 1′
acVp1_8in_1ft	26	3⁄8″ : 1′
acVp1_2in_1ft	27	1⁄2″ : 1′
acVp1_4in_1ft	28	3⁄4″ : 1′
acVp1in_1ft	29	1″ : 1′
acVp29	30	1½″ : 1′
acVp3in_1ft	30	3″ : 1′
acVp6in_1ft	31	6″ : 1′
acVp1ft_1ft	32	1′ : 1′

AcViewportSplitType

The following constants can be supplied as the value for the parameter of the `Split` method of the `AcadViewport` object.

Constant	Value	Description
acViewport2Horizontal	0	Splits the viewport horizontally into two equal sections.
acViewport2Vertical	1	Splits the viewport vertically into two equal sections.
acViewport3Left	2	Splits the viewport into two vertical halves. The left half is split horizontally into two equal sections.
acViewport3Right	3	Splits the viewport into two vertical halves. The right half is split horizontally into two equal sections.
acViewport3Horizontal	4	Splits the viewport horizontally into three equal sections.
acViewport3Vertical	5	Splits the viewport vertically into three equal sections.
acViewport3Above	6	Splits the viewport into two horizontal halves. The top half will be a single viewport. The bottom half is split vertically into two equal sections.
acViewport3Below	7	Splits the viewport into two horizontal halves. The bottom half will be a single viewport. The top half is split vertically into two equal sections.
acViewport4	8	Splits the viewport horizontally and vertically into four equal sections.

AcWindowState

The following constants can be values for the `WindowState` property of the `AcadApplication` and `AcadDocument` objects.

Constant	Value	Description
`acNorm`	1	Neither maximized nor minimized
`acMin`	2	Minimized window
`acMax`	3	Maximized window

AcXRefDemandLoad

The following constants can be values for the `XrefDemandLoad` property of the `AcadPreference-sOpenSave` object.

Constant	Value	Description
`acDemandLoadDisabled`	0	Turns off demand loading. The entire drawing is loaded.
`acDemandLoadEnabled`	1	Turns on demand loading and improves AutoCAD performance. Other users can't edit the file while it's being referenced.
`acDemandLoadEnabledWithCopy`	2	Turns on demand loading but uses a copy of the referenced drawing. Other users can edit the original drawing.

AcZoomScaleType

The following constants can be supplied as the value for the second parameter of the `ZoomScale` method of the `AcadApplication` object.

Constant	Value	Description
`acZoomScaledAbsolute`	0	The scale factor is applied relative to the drawing limits.
`acZoomScaledRelative`	1	The scale factor is applied relative to the current view.
`acZoomScaledRelativePSpace`	2	The scale factor is applied relative to paper space units.

APPENDIX C

System Variables

AutoCAD provides several system variables that let the user control interface elements such as object snap mode, grids, and limits. This appendix explains all of AutoCAD's system variables. It also demonstrates how to use the `GetVariable` and `SetVariable` methods to set and check a system variable's value. Each successive release of AutoCAD had introduced new system variables, modified existing ones, and removed others. Therefore, before you use a system variable, research it to determine whether your code needs to check the AutoCAD version to use the right system variable.

You can also use certain properties of AutoCAD objects to access some of these variables. This appendix lists those properties.

The `GetVariable` Method

Use the `Document` object's `GetVariable` method to retrieve a system variable's value. This method has the following syntax:

```
CurrentValue = DocumentObject.GetVariable(VariableName)
```

This method returns the value of the system variable you specify in the `VariableName` parameter. This parameter is a String.

This example retrieves and displays the `ANGDIR` system variable's current value.

This variable's value determines whether AutoCAD measures angles counterclockwise or clockwise with respect to the current OCS.

```
Public Sub GetANGDIR()
Dim varAngleDirection As Variant

    varAngleDirection = ThisDrawing.GetVariable("ANGDIR")
    If varAngleDirection = 0 Then
        MsgBox "Angles are measured counterclockwise"
    Else
        MsgBox "Angles are measured clockwise"
    End If
End Sub
```

The `SetVariable` Method

Use the `Document` object's `SetVariable` method to set a system variable. It has the following syntax:

```
DocumentObject.SetVariable VariableName, VariableValue
```

Table C-1 explains this method's parameters.

Table C-1. *The* `SetVariable` *Method's Parameters*

Name	Data Type	Description
`VariableName`	String	The name of the system variable
`VariableValue`	Variant	The system variable's new value

This example lets the user set the `ANGDIR` system variable's value:

```
Public Sub SetANGDIR()
Dim strResponse As String

On Error Resume Next
    strResponse = InputBox("Please specify the direction that angles should " & _
    "be measured (Clockwise or Counterclockwise )", , "Counterclockwise")
    If UCase(strResponse) = "COUNTERCLOCKWISE" Then
        ThisDrawing.SetVariable "ANGDIR", 0
    ElseIf UCase(strResponse) = "CLOCKWISE" Then
        ThisDrawing.SetVariable "ANGDIR", 1
    ElseIf strResponse = "" Then
        MsgBox "You cancelled, the system variable setting has not been changed"
    Else
    MsgBox "The value you typed has not been recognized - " & _
        "the system variable setting has not been changed"
    End If
End Sub
```

A

Table C-2 explains the system variables that start with *A*.

Table C-2. *The* A *System Variables*

Name	Description
`ACADLSPASDOC`	Controls whether the `acad.lsp` file is loaded into every open drawing or just the first one in the session. This variable's value is also stored in the `LoadAcadLspInAllDocuments` property of the `AcadPreferencesSelection` object.
`ACADPREFIX`	Stores the directory path specified by the `ACAD` environment variable, including path separators where necessary.

Name	Description
ACADVER	Stores the AutoCAD version number. AutoCAD 2000 = 15.0, AutoCAD 2000i = 15.05, AutoCAD 2002 = 15.06, AutoCAD 2004 = 16.0, AutoCAD 2005 = 16.1, and AutoCAD 2006 = 16.2. This variable's value is also stored in the `Version` property of the `AcadApplication` object.
ACISOUTVER	Controls the ACIS version of `.sat` files created using the `ACISOUT` command.
ADCSTATE	Determines whether DesignCenter is active.
AFLAGS	Sets attribute flags for `ATTDEF` bit code. This variable's value is also stored in the `Mode` property of the `AcadAttribute` object.
ANGBASE	Sets the base angle 0 with respect to the current UCS.
ANGDIR	Sets the positive angle direction from angle 0 with respect to the current UCS.
APBOX	Turns the AutoSnap aperture box on or off. This variable's value is also stored in the `AutoSnapAperture` property of the `AcadPreferencesDrafting` object.
APERTURE	Sets object snap target height in pixels. This variable's value is also stored in the `AutoSnapApertureSize` property of the `AcadPreferencesDrafting` object.
AREA	Stores the last area computed by `AREA`, `LIST`, or `DBLIST`.
ASSISTSTATE	Indicates whether the Info palette that displays Quick Help is active.
ATTDIA	Controls whether `-INSERT` uses a dialog box when entering attribute values.
ATTMODE	Controls the display of attributes.
ATTREQ	Determines whether `INSERT` uses default attribute settings during the insertion of blocks.
AUDITCTL	Controls whether `AUDIT` creates an audit report (`.adt`) file.
AUNITS	Sets angle units.
AUPREC	Sets the number of decimal places for angular units.
AUTOSNAP	Controls the AutoSnap marker, tooltip, and magnet. This variable's value is also stored in the `AutoSnapMagnet`, `AutoSnapMarker`, `AutoSnapToolTip`, and `AutoTrackTooltip` properties of the `AcadPreferencesDrafting` object.

B

Table C-3 explains the system variables that start with *B*.

Table C-3. *The* B *System Variables*

Name	Description
BACKGROUNDPLOT	Controls whether background plotting is turned on or off for plotting and publishing. By default, background plotting is turned off for plotting and on for publishing.
BACKZ	Stores the current viewport's back clipping plane offset from the target plane.
BACTIONCOLOR	Sets the text color of actions in the Block Editor. Valid values include `BYLAYER`, `BYBLOCK`, and an integer from 1 to 255. Valid values for True Colors are a string of integers each from 1 to 255 separated by commas and preceded by RGB.

Continued

Table C-3. *Continued*

Name	Description
BDEPENDENCYHIGHLIGHT	Controls whether dependent objects are dependency highlighted when a parameter, action, or grip is selected in the Block Editor.
BGRIPOBJCOLOR	Sets the color of grips in the Block Editor. Valid values include BYLAYER, BYBLOCK, and an integer from 1 to 255. Valid values for True Colors are a string of integers each from 1 to 255 separated by commas and preceded by RGB.
BGRIPOBJSIZE	Sets the display size of custom grips in the Block Editor relative to the screen display. Valid values include an integer from 1 to 255. Use the REGEN command to update the display size of custom grips in the Block Editor.
BINDTYPE	Controls how Xref names are handled when binding them or when editing them in place.
BLIPMODE	Controls whether marker blips are visible.
BLOCKEDITLOCK	Disallows opening of the Block Editor and editing of dynamic block definitions. When BLOCKEDITLOCK is set to 1, double-clicking a dynamic block in a drawing opens the Reference Edit dialog box. If the block contains attributes, double-clicking the block reference opens the Enhanced Attribute Editor.
BLOCKEDITOR	Indicates whether the Block Editor is open.
BPARAMETERCOLOR	Sets the color of parameters in the Block Editor. Valid values include BYLAYER, BYBLOCK, and an integer from 1 to 255. Valid values for True Colors are a string of integers each from 1 to 255 separated by commas and preceded by RGB.
BPARAMETERFONT	Sets the font used for parameters and actions in the Block Editor. You can specify either a TrueType font or an SHX font (for example, Verdana or `Verdana.ttf`). You must add the `.shx` extension to specify an AutoCAD SHX font. When specifying an Asian Big Font, use the following naming convention: an SHX file followed by a comma (`,`), followed by the Big Font file name (for example, `Simplex.shx,Bigfont.shx`).
BPARAMETERSIZE	Sets the size of parameter text and features in the Block Editor relative to the screen display. Valid values include an integer from 1 to 255.
BTMARKDISPLAY	Controls whether value set markers are displayed for dynamic block references.
BVMODE	Controls how objects that are made invisible for the current visibility state are displayed in the Block Editor.

C

Table C-4 explains the system variables that start with *C*.

Table C-4. *The* C *System Variables*

Name	Description
CALCINPUT	Controls whether mathematical expressions and global constants are evaluated in text and numeric entry boxes of windows and dialog boxes.
CDATE	Sets the calendar's date and time.
CECOLOR	Sets the color of new objects.

Name	Description
CELTSCALE	Sets the linetype scaling factor for the current object.
CELTYPE	Sets the linetype of new objects.
CELWEIGHT	Sets the lineweight of new objects.
CENTERMT	Controls how grips stretch multiline text that is centered horizontally. CENTERMT does not apply to stretching multiline text by using the ruler in the In-Place Text Editor.
CHAMFERA	Sets the first chamfer distance.
CHAMFERB	Sets the second chamfer distance.
CHAMFERC	Sets the chamfer length.
CHAMFERD	Sets the chamfer angle.
CHAMMODE	Sets the input method by which chamfers are created.
CIRCLERAD	Sets the default radius for a circle.
CLAYER	Sets the current layer.
CLEANSCREENSTATE	Stores a value that indicates whether AutoCAD is in Clean Screen mode.
CLISTATE	Stores a value that indicates whether the command window is hidden or displayed.
CMDACTIVE	Stores the bit code that indicates whether an ordinary command, a transparent command, a script, or a dialog box is active.
CMDDIA	Controls whether the command line or dialog boxes are used in certain commands.
CMDECHO	Controls whether prompts and input are echoed during the AutoLISP Command function.
CMDINPUTHISTORYMAX	Sets the maximum number of previous input values that are stored for a prompt in a command. Display of the history of user input is controlled by the INPUTHISTORYMODE system variable.
CMDNAMES	Displays the names of active and transparent commands.
CMLJUST	Sets the justification of a multiline.
CMLSCALE	Sets the width of a multiline.
CMLSTYLE	Sets the style of a multiline.
COMPASS	Controls whether the 3-D compass is switched on or off for the current viewport.
COORDS	Controls when coordinates are updated on the status line.
CPLOTSTYLE	Sets the current plot style for new objects.
CPROFILE	Stores the current profile’s name.
CROSSINGAREACOLOR	Controls the color of the selection area during crossing selection. The valid range is 1 to 255. The SELECTIONAREA system variable must be on.
CTAB	Returns the name of the current Model tab or Layout tab in the drawing.
CTABLESTYLE	Sets the name of the current table style.
CURSORSIZE	Determines the size of the crosshairs as a percentage of the screen size. This variable’s value is also stored in the CursorSize property of the AcadPreferencesDisplay object.
CVPORT	Sets the current viewport’s identification number.

D

Table C-5 explains the system variables that start with *D*.

Table C-5. *The* D *System Variables*

Name	Description
`DATE`	Stores the current date and time.
`DBCSTATE`	Stores the state of the dbConnect Manager, active or not active.
`DBMOD`	Indicates the status of drawing modifications using bit code.
`DCTCUST`	Displays the current custom dictionary's path and file name.
`DCTMAIN`	Displays the current main dictionary's file name.
`DEFLPLSTYLE`	Specifies the default plot style for new layers. This variable's value is also stored in the `DefaultPlotStyleForLayer` property of the `AcadPreferencesOutput` object.
`DEFPLSTYLE`	Specifies the default plot style for new objects. This variable's value is also stored in the `DefaultPlotStyleForObjects` property of the `AcadPreferencesOutput` object.
`DELOBJ`	Controls whether any objects used to create other objects are retained or deleted from the drawing database.
`DEMANDLOAD`	Specifies the conditions that cause AutoCAD to demand load a third-party application, if a drawing contains objects created in that application. This variable's value is also stored in the `DemandLoadARXApp` property of the `AcadPreferencesOpenSave` object.
`DIASTAT`	Stores the exit method of the most recently used dialog box.
`DIMADEC`	Controls the number of places of precision displayed for angular dimensions.
`DIMALT`	Controls the display of alternate units in dimensions.
`DIMALTD`	Controls the number of decimal places in alternate units.
`DIMALTF`	Controls the multiplier for alternate units.
`DIMALTRND`	Determines the rounding of alternate units.
`DIMALTTD`	Sets the number of decimal places for tolerance values in a dimension's alternate units.
`DIMALTTZ`	Toggles suppression of zeros in tolerance values.
`DIMALTU`	Sets the units format for alternate units of all dimension style family members except angular.
`DIMALTZ`	Controls the suppression of zeros for alternate unit dimension values.
`DIMAPOST`	Specifies a text prefix and/or suffix to the alternate dimension measurement for all types of dimensions except angular.
`DIMASO`	Controls the associativity of dimension objects (replaced by `DIMASSOC` in AutoCAD 2002 and later versions).
`DIMASSOC`	Controls the associativity of dimension objects.
`DIMASZ`	Controls the size of dimension line and leader line arrowheads.
`DIMATFIT`	Determines how dimension text and arrows are arranged when space is not sufficient to place both within the extension lines.
`DIMAUNIT`	Sets the units format for angular dimensions.
`DIMAZIN`	Suppresses zeros for angular dimensions.

Name	Description
DIMBLK	Sets the arrowhead block displayed at the ends of dimension lines or leader lines.
DIMBLK1	Sets the arrowhead for the first end of the dimension line when DIMSAH is on.
DIMBLK2	Sets the arrowhead for the second end of the dimension line when DIMSAH is on.
DIMCEN	Controls drawing of circle or arc center marks and centerlines by DIMCENTER, DIMDIAMETER, and DIMRADIUS.
DIMCLRD	Assigns colors to dimension lines, arrowheads, and dimension leader lines.
DIMCLRE	Assigns colors to dimension extension lines.
DIMCLRT	Assigns colors to dimension text.
DIMDEC	Sets the number of decimal places displayed for a dimension's primary units.
DIMDLE	Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn instead of arrowheads.
DIMDLI	Controls the spacing of dimension lines in baseline dimensions.
DIMDSEP	Specifies a single character decimal separator to use when creating dimensions whose unit format is decimal.
DIMEXE	Specifies how far to extend the extension line beyond the dimension line.
DIMEXO	Specifies how far extension lines are offset from origin points.
DIMFIT	Obsolete. Replaced by DIMATFIT and DIMTMOVE. Retained for backward compatibility only.
DIMFRAC	Sets the fraction format when DIMLUNIT is set to 4 or 5.
DIMGAP	Sets the distance around dimension text when the dimension line breaks to accommodate dimension text.
DIMJUST	Controls the horizontal positioning of dimension text.
DIMLDRBLK	Specifies the arrow type for leaders.
DIMLFAC	Sets a scale factor for linear dimension measurements.
DIMLIM	Generates dimension limits as the default text.
DIMLUNIT	Sets units for all dimension types except angular.
DIMLWD	Assigns lineweight to dimension lines.
DIMLWE	Assigns lineweight to extension lines.
DIMPOST	Specifies a text prefix and/or suffix to the dimension measurement.
DIMRND	Rounds all dimensioning distances to the specified value.
DIMSAH	Controls the display of dimension line arrowhead blocks.
DIMSCALE	Sets the overall scale factor applied to dimensioning variables that specify sizes, distances, or offsets.
DIMSD1	Controls suppression of the first dimension line.
DIMSD2	Controls suppression of the second dimension line.
DIMSE1	Suppresses display of the first extension line.
DIMSE2	Suppresses display of the second extension line.
DIMSHO	Controls redefinition of dimension objects while dragging.
DIMSOXD	Suppresses drawing of dimension lines outside the extension lines.

Continued

Table C-5. *Continued*

Name	Description
DIMSTYLE	Shows the current dimension style.
DIMTAD	Controls the vertical position of text in relation to the dimension line.
DIMTDEC	Sets the number of decimal places to display in tolerance values for a dimension's primary units.
DIMTFAC	Sets a scale factor used to calculate the height of text for dimension fractions and tolerances.
DIMTIH	Controls the position of dimension text inside the extension lines for all dimension types except ordinate.
DIMTIX	Draws text between extension lines.
DIMTM	When DIMTOL or DIMLIM is on, sets the minimum tolerance limit for dimension text.
DIMTMOVE	Sets dimension text movement rules.
DIMTOFL	Controls whether a dimension line is drawn between the extension lines even if text is placed outside.
DIMTOH	Controls the position of dimension text outside the extension lines.
DIMTOL	Appends tolerances to dimension text.
DIMTOLJ	Sets the vertical justification for tolerance values relative to the nominal dimension text.
DIMTP	When DIMTOL or DIMLIM is on, sets the maximum tolerance limit for dimension text.
DIMTSZ	Specifies the size of oblique strokes drawn instead of arrowheads for linear, radial, and diameter dimensioning.
DIMTVP	Controls the vertical position of dimension text above or below the dimension line.
DIMTXSTY	Specifies the text style of the dimension.
DIMTXT	Specifies the height of dimension text, unless the current text style has a fixed height.
DIMTZIN	Controls the suppression of zeros in tolerance values.
DIMUNIT	Obsolete. Replaced by DIMLUNIT and DIMFRAC. Retained for backward compatibility only.
DIMUPT	Controls options for user-positioned text.
DIMZIN	Controls the suppression of zeroes in the primary unit value.
DISPSILH	Controls display of silhouette curves of solid objects in Wireframe mode. This variable's value is also stored in the DisplaySilhouette property of the AcadDatabasePreferences object.
DISTANCE	Stores the distance computed by DIST.
DONUTID	Sets the default for the inside diameter of a torus (donut).
DONUTOD	Sets the default for the outside diameter of a torus (donut).
DRAGMODE	Controls the display of objects being dragged.
DRAGP1	Sets the sampling rate for regen-drag input.
DRAGP2	Sets the sampling rate for fast-drag input.
DRAWORDERCTL	Controls the display order of overlapping objects. Use this setting to improve the speed of editing operations in large drawings. The commands that are affected by inheritance are BREAK, FILLET, HATCH, HATCHEDIT, EXPLODE, TRIM, JOIN, PEDIT, and OFFSET.

Name	Description
DRSTATE	Determines whether the Drawing Recovery window is active.
DTEXTED	Specifies the user interface for editing single-line text.
DWGCHECK	Determines whether a drawing was last edited by a product other than AutoCAD.
DWGCODEPAGE	Stores the same value as SYSCODEPAGE (retained for compatibility reasons).
DWGNAME	Stores the drawing name as entered by the user.
DWGPREFIX	Stores the drive or directory prefix for the drawing.
DWGTITLED	Indicates whether the current drawing has been named.
DYNDIGRIP	Controls which dynamic dimensions are displayed during grip stretch editing. The DYNDIVIS system variable must be set to 2, which displays all dynamic dimensions. The DYNMODE system variable turns Dynamic Input features on and off.
DYNDIVIS	Controls how many dynamic dimensions are displayed during grip stretch editing. DYNDIGRIP controls which dynamic dimensions are displayed during grip stretch editing. The DYNMODE system variable turns Dynamic Input features on and off.
DYNMODE	Turns Dynamic Input features on and off. When all features are on, the context governs what is displayed. When DYNMODE is set to a negative value, the Dynamic Input features are not visible, but the setting is stored. Click the Dyn button in the status bar to set DYNMODE to the corresponding positive value. When DYNMODE is set to anything but 0, you can turn off all features temporarily by holding down the temporary override key, F12. Settings are on the Dynamic Input tab in the Drafting Settings dialog box.
DYNPICOORDS	Controls whether pointer input uses relative or absolute format for coordinates. The DYNMODE system variable turns Dynamic Input features on and off.
DYNPIFORMAT	Controls whether pointer input uses polar or Cartesian format for coordinates. This setting applies only to a second or next point. The DYNMODE system variable turns Dynamic Input features on and off.
DYNPIVIS	Controls when pointer input is displayed. The DYNMODE system variable turns Dynamic Input features on and off.
DYNPROMPT	Controls display of prompts in Dynamic Input tooltips. The DYNMODE system variable turns Dynamic Input features on and off.
DYNTOOLTIPS	Controls which tooltips are affected by tooltip appearance settings.

E

Table C-6 explains the system variables that start with *E*.

Table C-6. *The* E *System Variables*

Name	Description
EDGEMODE	Controls how TRIM and EXTEND determine cutting and boundary edges.
ELEVATION	Stores the current elevation relative to the current UCS for the current viewport.
ENTERPRISEMENU	Stores the CUI file name (if defined), including the path for the file name.
EXPERT	Controls whether certain prompts are issued.
EXPLMODE	Controls whether EXPLODE supports nonuniformly scaled (NUS) blocks.

Continued

Table C-6. *Continued*

Name	Description
EXTMAX	Stores the upper-right point of the drawing extents.
EXTMIN	Stores the lower-left point of the drawing extents.
EXTNAMES	Sets the parameters for named object names (such as linetypes and layers) stored in symbol tables. This variable's value is also stored in the AllowLongSymbolNames property of the AcadDataBasePreferences object.

F

Table C-7 explains the system variables that start with *F*.

Table C-7. *The* F *System Variables*

Name	Description
FACETRATIO	Controls the aspect ratio of faceting for cylindrical and conic ACIS solids.
FACETRES	Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed. This variable's value is also stored in the RenderSmoothness property of the AcadDatabasePreferences object.
FIELDDISPLAY	Controls whether fields are displayed with a gray background. The background is not plotted.
FIELDEVAL	Controls how fields are updated. The setting is stored as a bit code using the sum of the values.
FILEDIA	Suppresses display of file dialog boxes.
FILLETRAD	Stores the current fillet radius.
FILLMODE	Specifies whether multilines, traces, solids, hatches (including solid-fill hatches), and wide polylines are filled in. This variable's value is also stored in the SolidFill property of the AcadDatabasePreferences object.
FONTALT	Stores the alternate font that will be used when the specified font file can't be located.
FONTMAP	Specifies the font mapping file that will be used.
FRONTZ	Stores the front clipping plane offset from the target plane for the current viewport.
FULLOPEN	Indicates whether the current drawing is partially open or fully open.
FULLPLOTPATH	Controls whether the full path of the drawing file is sent to the plot spooler.

G

Table C-8 explains the system variables that start with *G*.

Table C-8. *The* G *System Variables*

Name	Description
`GFANG`	Specifies the angle for a gradient fill.
`GFCLR1`	Specifies the color for a one-color gradient fill or the first color for a two-color gradient fill.
`GFCLR2`	Specifies the second color for a two-color gradient fill.
`GFCLRLUM`	Makes the color a tint (mixed with white) or a shade (mixed with black) in a one-color gradient fill.
`GFCLRSTATE`	Specifies whether a gradient fill uses one color or two colors.
`GFNAME`	Specifies the pattern of a gradient fill.
`GFSHIFT`	Specifies whether the pattern in a gradient fill is centered or is shifted up and to the left.
`GRIDMODE`	Specifies whether the grid is turned on or off. This variable's value is also stored in the `GridOn` property of the `AcadPViewport` and `AcadViewport` objects.
`GRIDUNIT`	Specifies the grid spacing for the current viewport.
`GRIPBLOCK`	Controls the assignment of grips in blocks. This variable's value is also stored in the `DisplayGripsWithinBlocks` property of the `AcadPreferencesSelection` object.
`GRIPCOLOR`	Controls the color of nonselected grips, which are drawn as box outlines. This variable's value is also stored in the `GripColorUnselected` property of the `AcadPreferencesSelection` object.
`GRIPDYNCOLOR`	Controls the color of custom grips for dynamic blocks. The valid range is 1 to 255.
`GRIPHOT`	Controls the color of selected grips, which are drawn as filled boxes. This variable's value is also stored in the `GripColorSelected` property of the `AcadPreferencesSelection` object.
`GRIPHOVER`	Controls the fill color of a grip when the cursor pauses over the grip.
`GRIPOBJLIMIT`	Determines the maximum number of objects on which grips will be displayed.
`GRIPS`	Controls the use of selection set grips for the `Stretch`, `Move`, `Rotate`, `Scale`, and `Mirror` grip modes. This variable's value is also stored in the `DisplayGrips` property of the `AcadPreferencesSelection` object.
`GRIPSIZE`	Sets the size of the grip box in pixels. This variable's value is also stored in the `GripSize` property of the `AcadPreferencesSelection` object.
`GRIPTIPS`	Controls the display of grip tips when the cursor hovers over grips on custom objects that support grip tips.

H

Table C-9 explains the system variables that start with *H*.

Table C-9. *The* H *System Variables*

Name	Description
HALOGAP	Specifies the distance to shorten a haloed line.
HANDLES	Reports whether applications can access object handles.
HIDEPRECISION	Controls the accuracy of hides and shades.
HIDETEXT	Specifies whether text objects created by the TEXT, DTEXT, or MTEXT command are processed during a HIDE command.
HIGHLIGHT	Controls object highlighting. This variable's value does not affect objects selected with grips.
HPANG	Specifies the angle of the hatch pattern. This variable's value is also stored in the PatternAngle property of the AcadHatch object.
HPASSOC	Controls whether hatch patterns and gradient fills are associative.
HPBOUND	Controls the object type created by BHATCH and BOUNDARY.
HPDOUBLE	Specifies hatch pattern doubling for user-defined patterns. This variable's value is also stored in the PatternDouble property of the AcadHatch object.
HPDRAWORDER	Controls the draw order of hatches and fills. Stores the Draw Order setting from the Hatch and Fill dialog box.
HPGAPTOL	Treats a set of objects that almost enclose an area as a closed hatch boundary. The default value, 0, specifies that the objects enclose the area, with no gaps. Enter a value, in drawing units, from 0 to 5000 to set the maximum size of gaps that can be ignored when the objects serve as a hatch boundary.
HPINHERIT	Controls the hatch origin of the resulting hatch when using Inherit Properties in HATCH and HATCHEDIT.
HPNAME	Sets the default name for the hatch pattern. This variable's value is also stored in the PatternName property of the AcadHatch object.
HPOBJWARNING	Sets the number of hatch boundary objects that can be selected before displaying a warning message.
HPORIGIN	Sets the hatch origin point for new hatch objects relative to the current user coordinate system.
HPORIGINMODE	Controls how HATCH determines the default hatch origin point.
HPSCALE HPINHERIT	Specifies the scale factor for the hatch pattern. This variable's value is also stored in the PatternScale property of the AcadHatch object.
HPSEPARATE	Controls whether HATCH creates a single hatch object or separate hatch objects when operating on several closed boundaries.
HPSPACE	Specifies the hatch pattern's line spacing for user-defined simple patterns. This variable's value is also stored in the PatternSpace property of the AcadHatch object.
HYPERLINKBASE	Specifies the path used for all relative hyperlinks in the drawing.

I

Table C-10 explains the system variables that start with *I*.

Table C-10. *The* I *System Variables*

Name	Description
IMAGEHLT	Specifies whether the entire raster image or only the image frame is highlighted. This variable's value is also stored in the ImageFrameHighlight property of the AcadPreferencesDisplay object.
INDEXCTL	Controls whether layer and spatial indexes are created and saved in drawing files.
INETLOCATION	Stores the Internet location used by BROWSER and the Browse the Web dialog box. This variable's value is also stored in the DefaultInternetURL property of the AcadPreferencesFiles object.
INPUTHISTORYMODE	Controls the content and location of the display of a history of user input. The setting is stored as a bit code using the sum of the values.
INSBASE	Stores the insertion base point set by BASE.
INSNAME	Sets the default block name for INSERT.
INSUNITS	Specifies a value for the drawing units, when a block is dragged from the AutoCAD DesignCenter.
INSUNITSDEFSOURCE	Sets the source content's units value. This variable's value is also stored in the ADCInsertUnitsDefaultSource property of the AcadPreferencesUser object.
INSUNITSDEFTARGET	Sets the target drawing's units value. This variable's value is also stored in the ADCInsertUnitsDefaultTarget property of the AcadPreferencesUser object.
INTELLIGENTUPDATE	Controls the graphics refresh rate. The default value is 20 frames per second. If you encounter problems related to graphics generation or timing, turn off the variable by setting it to 0. INTELLIGENTUPDATE works by suppressing the graphics update until the timer expires. Subsequent updates reset the timer. The performance improvement significantly affects updates for scripts and AutoLISP graphics. Those using regular AutoCAD commands will not see a noticeable difference in performance.
INTERSECTIONCOLOR	Specifies the color of intersection polylines.
INTERSECTIONDISPLAY	Specifies the display of intersection polylines.
ISAVEBAK	Increases the speed of incremental saves for large drawings. This variable's value is also stored in the CreateBackup property of the AcadPreferencesSaveOpen object.
ISAVEPERCENT	Determines the amount of wasted space tolerated in a drawing file. This variable's value is also stored in the IncrementalSavePercent property of the AcadPreferencesOpenSave object.
ISOLINES	Specifies the number of isolines per surface on objects. This variable's value is also stored in the ContourLinesPerSurface property of the AcadDatabasePreferences object.

L

Table C-11 explains the system variables that start with *L*.

Table C-11. *The* L *System Variables*

Name	Description
`LASTANGLE`	Stores the end angle of the most recently drawn arc.
`LASTPOINT`	Stores the last point drawn.
`LASTPROMPT`	Stores the last string echoed to the command line.
`LAYERFILTERALERT`	Deletes excessive layer filters to improve performance. When a drawing has 100 or more layer filters, and the number of layer filters exceeds the number of layers, `LAYERFILTERALERT` provides a method for deleting layer filters to improve performance.
`LAYOUTREGENCTL`	Specifies how the display list is updated in the Model tab and Layout tabs.
`LENSLENGTH`	Stores the length of the lens (in millimeters) used in perspective viewing for the current viewport. This variable's value is also stored in the `LensLength` property of the `AcadPViewport` object.
`LIMCHECK`	Controls the creation of objects outside the drawing limits.
`LIMMAX`	Stores the upper-right drawing limits for the current space. This variable's value is also stored in the `Limits` property of the `AcadDocument` object.
`LIMMIN`	Stores the lower-left drawing limits for the current space. This variable's value is also stored in the `Limits` property of the `AcadDocument` object.
`LISPINIT`	In SDI mode, this specifies whether AutoLISP-defined functions and variables are preserved when a new drawing is opened.
`LOCALE`	Displays the ISO language code of the current AutoCAD version.
`LOCALROOTPREFIX`	Stores the full path to the root folder where local customizable files were installed.
`LOCKUI`	Locks the position and size of toolbars and windows such as DesignCenter and Properties palette. Locked toolbars and windows can still be opened and closed and items can be added and deleted. To unlock them temporarily, hold down Ctrl. A lock icon in the status bar tray indicates whether toolbars and windows are locked. Right-click the icon to display locking options. The setting is stored as a bit code using the sum of the values.
`LOGFILEMODE`	Specifies whether the contents of the text window are written to a log file. This variable's value is also stored in the `LogFileOn` property of the `AcadPreferencesOpenSave` object.
`LOGFILENAME`	Specifies the path and name of the log file.
`LOGFILEPATH`	Specifies the path for the log files for all drawings in a session. This variable's value is also stored in the `LogFilePath` property of the `PreferencesFiles` object.
`LOGINNAME`	Displays the user's login name when AutoCAD is loaded.
`LTSCALE`	Sets the scale factor for the global linetype.
`LUNITS`	Sets linear units.
`LUPREC`	Sets the number of decimal places displayed for linear units.
`LWDEFAULT`	Sets the lineweight's default value.
`LWDISPLAY`	Controls whether the lineweight is displayed in the Model or Layout tab.

Name	Description
LWSCALE	Controls whether the lineweight is scaled with the rest of the geometry when a layout is printed. This variable's value is also stored in the ScaleLineweights property of the AcadLayout and AcadPlotConfiguration objects.
LWUNITS	Controls whether lineweight units are displayed in inches or millimeters.

M

Table C-12 explains the system variables that start with *M*.

Table C-12. *The* M *System Variables*

Name	Description
MAXACTVP	Sets the maximum number of viewports that can be active at one time. This variable's value is also stored in the MaxActiveViewports property of the AcadDatabasePreferences object.
MAXSORT	Sets the maximum number of symbol names or block names that can be sorted.
MBUTTONPAN	Controls the behavior of the third button or wheel on the pointing device.
MEASUREINIT	Sets the initial drawing units as English or metric.
MEASUREMENT	Sets drawing units as English or metric for the current drawing only.
MENUCTL	Controls the page switching of the screen menu.
MENUECHO	Sets menu echo and prompt control bits.
MENUNAME	Stores the menu file name, including the path.
MIRRTEXT	Controls how MIRROR reflects text.
MODEMACRO	Displays a text string on the status line.
MSOLESCALE	Controls the size of an OLE object with text that is pasted into model space. MSOLESCALE controls only the initial size. If the scale factor value is changed, existing OLE objects in the drawing are not affected. Positive number scales by value. Zero (0) scales by DIMSCALE value.
MTEXTED	Sets the primary and secondary text editors to use for AcadMText objects.
MTEXTFIXED	Controls the appearance of the Multiline Text Editor.
MTJIGSTRING	Sets the content of the sample text displayed at the cursor location when the MTEXT command is started.
MYDOCUMENTSPREFIX	Stores the full path to the My Documents folder for the user currently logged on.

N

Table C-13 explains the system variables that start with *N*.

Table C-13. *The* N *System Variables*

Name	Description
`NOMUTT`	Controls whether message display (muttering) is suppressed in AutoCAD.

O

Table C-14 explains the system variables that start with *O*.

Table C-14. *The* O *System Variables*

Name	Description
`OBSCUREDCOLOR`	Specifies the color of obscured lines.
`OBSCUREDLTYPE`	Specifies the linetype of obscured lines.
`OFFSETDIST`	Sets the default offset distance.
`OFFSETGAPTYPE`	Controls how to fill in gaps resulting from offsetting individual polyline segments.
`OLEFRAME`	Controls whether a frame is displayed and plotted on all OLE objects in the drawing. The frame on an OLE object must be displayed in order for grips to be visible.
`OLEHIDE`	Controls the display of OLE objects in AutoCAD.
`OLEQUALITY`	Controls the default quality level for embedded OLE objects. This variable's value is also stored in the `OLEQuality` property of the `AcadPreferencesOutput` object.
`OLESTARTUP`	Controls whether to load an embedded OLE object's source application when plotting. This variable's value is also stored in the `OLELaunch` property of the `AcadDatabasePreferences` object.
`ORTHOMODE`	Constrains cursor movement to perpendicular axes.
`OSMODE`	Sets running object snap modes using bit codes. Specifies whether the line-weight is scaled with the rest of the geometry when a layout is printed.
`OSNAPCOORD`	Controls whether coordinates entered on the command line override running object snaps. This variable's value is also stored in the `KeyboardPriority` property of the `AcadPreferencesUser` object.
`OSNAPHATCH`	Controls whether object snaps ignore hatch objects. The default setting 0 improves performance.
`OSNAPZ`	Controls whether object snaps are automatically projected onto a plane parallel to the XY plane of the current UCS at the current elevation.

P

Table C-15 explains the system variables that start with *P*.

Table C-15. *The* P *System Variables*

Name	Description
`_PKSER`	Stores the AutoCAD license serial number (for example, 400-12345678).
`PALETTEOPAQUE`	Controls whether windows can be made transparent.
`PAPERUPDATE`	Controls the display of a warning dialog when you try to print a layout with a paper size other than that specified for the plotter configuration file.
`PDMODE`	Controls how to display point objects. See Chapter 8 for more details.
`PDSIZE`	Sets the display size for point objects. See Chapter 8 for more details.
`PEDITACCEPT`	Suppresses display of the Object Selected Is Not a Polyline prompt in `PEDIT`.
`PELLIPSE`	Controls the ellipse type created with `ELLIPSE`.
`PERIMETER`	Stores the last perimeter value computed by `AREA`, `LIST`, or `DBLIST`.
`PFACEVMAX`	Sets the maximum number of vertices per face.
`PICKADD`	Controls whether subsequent selections replace the current selection set or add to it. This variable's value is also stored in the `PickAdd` property of the `AcadPreferencesSelection` object.
`PICKAUTO`	Controls automatic windowing at the Select Objects prompt. This variable's value is also stored in the `PickAuto` property of the `AcadPreferencesSelection` object.
`PICKBOX`	Sets object selection target height. This variable's value is also stored in the `PickBoxSize` property of the `AcadPreferencesSelection` object.
`PICKDRAG`	Controls the method of drawing a selection window. This variable's value is also stored in the `PickDrag` property of the `AcadPreferencesSelection` object.
`PICKFIRST`	Controls whether you select objects before or after you issue a command. This variable's value is also stored in the `PickFirst` property of the `AcadPreferencesSelection` object.
`PICKSTYLE`	Controls the use of group selection and associative hatch selection.
`PLATFORM`	Indicates which platform (operating system) of AutoCAD is being used.
`PLINEGEN`	Sets how linetype patterns generate around the vertices of a 2-D polyline.
`PLINETYPE`	Specifies whether optimized 2-D polylines are used.
`PLINEWID`	Stores the default width of the polyline.
`PLOTID`	Obsolete in AutoCAD 2000 and later versions; retained for backward compatibility.
`PLOTLEGACY`	Controls whether legacy plot scripts can be run. This variable's value is also stored in the `PlotLegacy` property of the `AcadPreferencesOutput` object.
`PLOTOFFSET`	Controls whether the plot offset is relative to the printable area or to the edge of the paper.
`PLOTROTMODE`	Controls the orientation of plots.
`PLOTTER`	Obsolete in AutoCAD 2000 and later versions; retained for backward compatibility.

Continued

Table C-15. *Continued*

Name	Description
PLQUIET	Controls the display of optional dialog boxes and nonfatal errors for batch plotting and scripts.
POLARADDANG	Contains user-defined polar angles.
POLARANG	Sets the polar angle increment.
POLARDIST	Sets the snap increment when the SNAPSTYL system variable is set to 1 (polar snap).
POLARMODE	Controls the settings for polar and object snap tracking.
POLYSIDES	Sets the default number of sides for POLYGON.
POPUPS	Displays the status of the currently configured display driver.
PREVIEWEFFECT	Specifies the visual effect used for previewing selection of objects.
PREVIEWFILTER	Excludes specified object types from selection previewing. The setting is stored as a bit code using the sum of the values.
PRODUCT	Returns the product name (for example, AutoCAD).
PROGRAM	Returns the program executable name (for example, acad).
PROJECTNAME	Assigns a project name to the current drawing.
PROJMODE	Sets the current projection mode for trimming or extending.
PROXYGRAPHICS	Specifies whether to save images of proxy objects.
PROXYNOTICE	Displays a dialog box when a drawing is opened that contains custom objects created by an application that is not present on the system. This variable's value is also stored in the ShowProxyDialogBox property of the AcadPreferencesOpenSave object.
PROXYSHOW	Controls the display of proxy objects in a drawing.
PROXYWEBSEARCH	Specifies how AutoCAD checks for Object Enablers.
PSLTSCALE	Controls linetype scaling for the paper space.
PSPOOLALERT	Specifies whether to alert the user when output needs to be spooled through a system printer because of a conflict with an I/O port. Not used in AutoCAD 2004 and later. This variable's value is also stored in the PrinterSpoolAlertproperty of the AcadPreferencesOutput object.
PSPROLOG	Assigns a name for a prolog section to be read from the acad.psf file when you are using PSOUT. This variable's value is also stored in the PostScriptPrologFile property of the PreferencesFiles object.
PSQUALITY	Controls the rendering quality of PostScript images.
PSTYLEMODE	Indicates whether the current drawing is in a Color Dependent or Named Plot Style mode.
PSTYLEPOLICY	Controls whether an object's color property is associated with its plot style. This variable's value is also stored in the PlotPolicy property of the AcadPreferencesOutput object.
PSVPSCALE	Sets the view scale factor for all newly created viewports.
PUCSBASE	Stores the name of the UCS that defines the origin and orientation of orthographic UCS settings in paper space.

Q

Table C-16 explains the system variables that start with *Q*.

Table C-16. *The* Q *System Variables*

Name	Description
QAFLAGS	An undocumented, but useful variable. Bit coded: 1 causes ^C in menu macros to cancel grips, 2 causes screen listings to scroll past without pausing, 4 prevents alert dialogs from appearing.
QCSTATE	Determines whether the QuickCalc calculator is active.
QTEXTMODE	Controls how text is displayed. This variable's value is also stored in the `TextFrameDisplay` property of the `DatabasePreferences` object.

R

Table C-17 explains the system variables that start with *R*.

Table C-17. *The* R *System Variables*

Name	Description
RASTERDPI	Controls paper size and plot scaling when changing from dimensional to dimensionless output devices, or vice versa. Converts millimeters or inches to pixels, or vice versa. Accepts an integer from 100 to 32,767 as a valid value.
RASTERPREVIEW	Controls whether to save `.bmp` preview images. This variable's value is also stored in the `SavePreviewThumbnail` property of the `AcadPreferencesOpenSave` object.
RECOVERYMODE	Controls whether drawing recovery information is recorded after a system failure.
REFEDITNAME	Indicates whether a drawing is in a reference-editing state and stores the reference file's name.
REGENMODE	Controls automatic drawing regeneration.
RE-INIT	Reinitializes the digitizer, digitizer port, and `acad.pgp` file.
REMEMBERFOLDERS	Controls the default path for the Look In or Save In option in standard file selection dialog boxes.
REPORTERROR	Controls whether an error report can be sent to Autodesk if AutoCAD closes unexpectedly.
ROAMABLEROOTPREFIX	Stores the full path to the root folder where roamable customizable files were installed.
RTDISPLAY	Controls the display of raster images during real-time `ZOOM` or `PAN`. This variable's value is also stored in the `ShowRasterImage` property of the `AcadPreferencesDisplay` object.

S

Table C-18 explains the system variables that start with *S*.

Table C-18. *The* S *System Variables*

Name	Description
SAVEFILE	Stores the current autosave file name.
SAVEFILEPATH	Specifies the directory path for all automatic save files for the session. This variable's value is also stored in the AutoSavePath property of the AcadPreferencesFiles object.
SAVENAME	Stores the file name and directory path of a drawing after it is saved.
SAVETIME	Sets the automatic save interval in minutes. This variable's value is also stored in the AutoSaveInterval property of the AcadPreferencesOpenSave object.
SCREENBOXES	Stores the number of boxes in the drawing window's screen menu area.
SCREENMODE	Stores a bit code indicating the display's graphics and text state.
SCREENSIZE	Stores current viewport size in pixels.
SDI	Controls whether AutoCAD runs in single document interface (SDI) mode or in multiple-document interface (MDI) mode. This variable's value is also stored in the SingleDocumentMode property of the AcadPreferencesSelection object.
SELECTIONAREA	Controls the display of effects for selection areas. Selection areas are created by the Window, Crossing, WPolygon, and CPolygon options of SELECT.
SELECTIONAREAOPACITY	Controls the transparency of the selection area during window and crossing selection. The valid range is 0 to 100. The lower the setting, the more transparent the area. A value of 100 makes the area opaque. The SELECTIONAREA system variable must be on.
SELECTIONPREVIEW	Controls the display of selection previewing. Objects are highlighted when the pickbox cursor rolls over them. This selection previewing indicates that the object would be selected if you clicked. The setting is stored as a bit code using the sum of the values.
SHADEDGE	Controls the shading of edges in rendering.
SHADEDIF	Sets the ratio of diffuse reflective light to ambient light.
SHORTCUTMENU	Controls whether Default, Edit, and Command mode shortcut menus are available in the drawing area. This variable's value is also stored in the SCMCommandMode, SCMDefaultMode, SCMEditMode, and ShortCutMenuDisplay properties of the AcadPreferencesUser object.
SHOWLAYERUSAGE	Displays icons in the Layer Properties Manager to indicate whether layers are in use. Setting this system variable to Off improves performance in the Layer Properties Manager.
SHPNAME	Sets a default shape name.
SIGWARN	Controls whether a warning is presented when a file with an attached digital signature is opened.
SKETCHINC	Sets the record increment for SKETCH.
SKPOLY	Determines whether SKETCH generates lines or polylines.
SNAPANG	Sets snap and grid rotation angle for the current viewport. This variable's value is also stored in the SnapRotationAngle property of the AcadPViewport and AcadViewport objects.

Name	Description
SNAPBASE	Sets the snap and grid origin point for the current viewport relative to the current UCS. This variable's value is also stored in the SnapBasePoint property of the AcadPViewport and VAcadiewport objects.
SNAPISOPAIR	Controls the isometric plane for the current viewport.
SNAPMODE	Turns snap mode on and off. This variable's value is also stored in the SnapOn property of the AcadPViewport and AcadViewport objects.
SNAPSTYL	Sets the current viewport's snap style.
SNAPTYPE	Sets the current viewport's snap style.
SNAPUNIT	Sets the current viewport's snap spacing.
SOLIDCHECK	Turns solid validation on and off for the current session.
SORTENTS	Controls the OPTIONS command object sort order operations. This variable's value is also stored in the ObjectSortByPlotting, ObjectSortByRedraws, ObjectSortByPSOutput, ObjectSortByRegens, ObjectSortBySelection, and ObjectSortBySnap properties of the AcadDatabasePreferences object.
SPLFRAME	Controls the display of splines and spline-fit polylines.
SPLINESEGS	Sets the number of line segments to be generated for each spline-fit polyline. This variable's value is also stored in the SegmentPerPolyline property of the AcadDatabasePreferences object.
SPLINETYPE	Sets the type of curve generated by the Spline option of the PEDIT command.
SSFOUND	Displays the sheet set path and file name if a search for a sheet set is successful.
SSLOCATE	Locates and opens the sheet set associated with a drawing when the drawing is opened. SSMAUTOOPEN and SSLOCATE must both be set to 1 to open a sheet set automatically in the Sheet Set Manager.
SSMAUTOOPEN	Displays the Sheet Set Manager when a drawing associated with a sheet is opened. SSMAUTOOPEN and SSLOCATE must both be set to 1 to open a sheet set automatically in the Sheet Set Manager.
SSMPOLLTIME	Controls the time interval between automatic refreshes of the status data in a sheet set. The SSMPOLLTIME timer sets the time in seconds between automatic refreshes of the status data of sheets in a sheet set. Valid values are from 20 to 600. The SSMSHEETSTATUS system variable must be set to 2 for the timer to operate.
SSMSHEETSTATUS	Controls how the status data in a sheet set is refreshed. The status data for sheets in the current sheet set includes whether a sheet is locked and whether a sheet is missing (or found in an unexpected location). This status data can be updated automatically for all sheets. To refresh the sheet set manually, use the Refresh Sheet Status button on the Sheet List tab of the Sheet Set Manager.
SSMSTATE	Determines whether the Sheet Set Manager window is active.
STANDARDSVIOLATION	Specifies whether a user is notified of standards violations that exist in the current drawing.
STARTUP	Controls whether the Create New Drawing dialog box is displayed when starting a new drawing with the NEW and QNEW commands.
SURFTAB1	Sets the number of tabulations to be generated for RULESURF and TABSURF.

Continued

Table C-18. *Continued*

Name	Description
SURFTAB2	Sets the mesh density in the N direction for REVSURF and EDGESURF.
SURFTYPE	Controls the type of surface fitting to be performed by the Smooth option of the PEDIT command.
SURFU	Sets the surface density for PEDIT command's Smooth option in the M direction.
SURFV	Sets the surface density for PEDIT command's Smooth option in the N direction.
SYSCODEPAGE	Indicates the system code page specified in acad.xmf.

T

Table C-19 explains the system variables that start with *T*.

Table C-19. *The* T *System Variables*

Name	Description
TABLEINDICATOR	Controls the display of row numbers and column letters when the In-Place Text Editor is open for editing a table cell.
TABMODE	Controls the use of the tablet.
TARGET	Stores the location of the target point for the current viewport.
TBCUSTOMIZE	Controls whether toolbars can be customized.
TDCREATE	Stores the local time and date the drawing was created.
TDINDWG	Stores the total editing time.
TDUCREATE	Stores the universal time and date the drawing was created.
TDUPDATE	Stores the local time and date of the last update/save.
TDUSRTIMER	Stores the user-elapsed timer.
TDUUPDATE	Stores the universal time and date of the last update/save.
TEMPOVERRIDES	Turns temporary override keys on and off. A temporary override key is a key that you can hold down to temporarily turn on or turn off one of the drawing aids that are set in the Drafting Settings dialog box; for example, Ortho mode, object snaps, or Polar mode.
TEMPPREFIX	Contains the directory name for temporary files.
TEXTEVAL	Controls the method of evaluation for text strings.
TEXTFILL	Controls the filling of TrueType fonts while plotting, exporting with the PSOUT command and rendering.
TEXTQLTY	Sets the resolution of text outlines for TrueType fonts while plotting, exporting with the PSOUT command, and rendering.
TEXTSIZE	Sets the default height for new text objects drawn with the current text style.
TEXTSTYLE	Sets the name of the current text style.
THICKNESS	Sets the current 3-D solid thickness.

Name	Description
TILEMODE	Makes the Model tab or the Last Layout tab current. This variable's value is also stored in the ActiveSpace property of the AcadDocument object.
TOOLTIPMERGE	Combines drafting tooltips into a single tooltip. The appearance of the merged tooltip is controlled by the settings in the Tooltip Appearance dialog box.
TOOLTIPS	Controls the display of tooltips.
TRACEWID	Sets the default trace width.
TRACKPATH	Controls the display of polar and object snap tracking alignment paths. This variable's value is also stored in the FullScreenTrackingVector and PolarTrackingVector properties of the AcadPreferencesDrafting object.
TRAYICONS	Controls whether a tray is displayed on the status bar.
TRAYNOTIFY	Controls whether service notifications are displayed in the status bar tray.
TRAYTIMEOUT	Controls the length of time (in seconds) that service notifications are displayed.
TREEDEPTH	Specifies the number of times the tree-structured spatial index can divide into branches.
TREEMAX	Limits memory consumption during drawing regeneration by limiting the number of nodes in the spatial index (oct-tree).
TRIMMODE	Controls whether AutoCAD trims selected edges for chamfers and fillets.
TSPACEFAC	Controls the multiline text line spacing distance, measured as a factor of text height.
TSPACETYPE	Controls the type of line spacing used in multiline text.
TSTACKALIGN	Controls the vertical alignment of stacked text.
TSTACKSIZE	Controls the height of stacked text fraction, as a percentage relative to the height of the selected text.

U

Table C-20 explains the system variables that start with *U*.

Table C-20. *The* U *System Variables*

Name	Description
UCSAXISANG	Stores the default angle when rotating the UCS around one of its axes using the X, Y, or Z options of the UCS command.
UCSBASE	Stores the name of the UCS that defines the origin and orientation of orthographic UCS settings.
UCSFOLLOW	Generates a plan view whenever you change from one UCS to another.
UCSICON	Displays the UCS icon for the current viewport.
UCSNAME	Stores the name of the current coordinate system for the current viewport in the current space.
UCSORG	Stores the origin point of the coordinate system for the current viewport in the current space. This variable's value is also stored in the UCSIconAtOrigin property of the PViewport and Viewport objects.

Continued

Table C-20. *Continued*

Name	Description
UCSORTHO	Determines whether the related orthographic UCS setting is restored automatically when an orthographic view is restored.
UCSVIEW	Determines whether the current UCS is saved with a named view.
UCSVP	Determines whether the UCS in active viewports remains fixed or changes to reflect the currently active viewport.
UCSXDIR	Stores the X direction of the current UCS for the current viewport in the current space. This variable's value is also stored in the XVector property of the AcadUCS object.
UCSYDIR	Stores the Y direction of the current UCS for the current viewport in the current space. This variable's value is also stored in the YVector property of the AcadUCS object.
UNDOCTL	Stores a bit code indicating the state of the Auto and Control options of the UNDO command.
UNDOMARKS	Stores the number of marks that have been placed in the UNDO control stream by the Mark option.
UNITMODE	Controls the display format for units.
UPDATETHUMBNAIL	Controls updating of the thumbnail previews in the Sheet Set Manager. The setting is stored as a bit code using the sum of the values.
USERI1–5	Stores and retrieves integer values. There are five variables of this type: USERI1, USERI2, ..., USERI5.
USERR1–5	Stores and retrieves real numbers. There are five variables of this type: USERR1, USERR2, ..., USERR5.
USERS1–5	Stores and retrieves strings. There are five variables of this type: USERS1, USERS2, ..., USERS5.

V

Table C-21 explains the system variables that start with *V*.

Table C-21. *The* V *System Variables*

Name	Description
_VERNUM	Stores the internal build number of AutoCAD (such as V.0.86).
VIEWCTR	Stores the center of view in the current viewport.
VIEWDIR	Stores the viewing direction in the current viewport.
VIEWMODE	Controls the current viewport's view mode using a bit code.
VIEWSIZE	Stores the view's height in the current viewport.
VIEWTWIST	Stores the view's twist angle for the current viewport.
VISRETAIN	Controls the visibility, color, linetype, lineweight, and plot styles (if PSTYLEPOLICY is set to 0) of Xref-dependent layers and specifies whether nested Xref path changes are saved. This variable's value is also stored in the XrefLayerVisibility property of the AcadDatabasePreferences object.

Name	Description
VPMAXIMIZEDSTATE	Stores a value that indicates whether the viewport is maximized. The maximized viewport state is canceled if you start the PLOT command.
VSMAX	Stores the upper-right corner of the current viewport's virtual screen.
VSMIN	Stores the lower-left corner of the current viewport's virtual screen.
VTDURATION	Sets the duration of a smooth view transition, in milliseconds. The valid range is 0 to 5000.
VTENABLE	Controls when smooth view transitions are used. Smooth view transitions can be on or off for panning and zooming, for changes of view angle, or for scripts. The valid range is 0 to 7.
VTFPS	Sets the minimum speed of a smooth view transition, in frames per second. When a smooth view transition cannot maintain this speed, an instant transition is used. The valid range is 1.0 to 30.0.

W

Table C-22 explains the system variables that start with *W*.

Table C-22. *The* W *System Variables*

Name	Description
WHIPARC	Controls whether the display of circles and arcs is smooth.
WHIPTHREAD	Controls whether to use an additional processor (known as multithreaded processing) to improve the speed of operations such as ZOOM and PAN that redraw or regenerate the drawing.
WINDOWAREACOLOR	Controls the color of the transparent selection area during window selection. The valid range is 1 to 255. SELECTIONAREA must be on.
WMFBKGND	Controls the background of the output Windows metafile resulting from the WMFOUT command, and also the metafile format of objects placed on the clipboard or dragged and dropped into other applications.
WMFFOREGND	Controls the assignment of the foreground color of AutoCAD objects in other applications.
WORLDUCS	Indicates whether the UCS is the same as the WCS.
WORLDVIEW	Determines whether input to the 3DORBIT, DVIEW, and VPOINT commands is relative to the WCS (default), the current UCS, or the UCS specified by the UCSBASE system variable.
WRITESTAT	Indicates whether a drawing file is read-only, for developers who need to determine write status through AutoLISP.
WSCURRENT	Returns the current work space name in the command line interface and sets a work space to current.

X

Table C-23 explains the system variables that start with *X*.

Table C-23. *The* X *System Variables*

Name	Description
XCLIPFRAME	Controls the visibility of Xref clipping boundaries.
XEDIT	Specifies whether the current drawing can be edited in place when another drawing references it. This variable's value is also stored in the `XRefEdit` property of the `AcadDatabasePreferences` object.
XFADECTL	Controls the fading intensity for references being edited in place. This variable's value is also stored in the `XrefFadeIntensity` property of the `AcadPreferencesDisplay` object.
XLOADCTL	Turns Xref demand loading on and off and controls whether it opens the original drawing or a copy of the drawing. This variable's value is also stored in the `XrefDemandLoad` property of the `AcadPreferencesOpenSave` object.
XLOADPATH	Creates a path for storing temporary copies of demand-loaded Xref files.
XREFCTL	Controls whether AutoCAD writes external reference log (`.xlg`) files.
XREFNOTIFY	Controls the notification for changed or missing Xrefs.
XREFTYPE	Controls the default reference type when attaching or overlaying an external reference.

Z

Table C-24 explains the system variables that start with *Z*.

Table C-24. *The* Z *System Variables*

Name	Description
ZOOMFACTOR	Controls the incremental change in zoom with each IntelliMouse wheel action, whether forward or backward.

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