

Part I

AutoCAD Customization: Increasing Productivity through Personalization

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RECEPTION/
ADMIN

LECTURE THEATRE/
SECONDARY GALLERY
140 SEATS

KITCHEN

CAFE

70 seats inside

CAFE patio

court yard

RETAIL 3/
PROFESSIONAL

FFL 0.040

FFL 0.420

FFL 0.555

RL 0.000

RL 0.515

RL 0.515

RL 0.000

RL 0.750



Chapter 1

Establishing the Foundation for Drawing Standards

Drawing standards, also known as *CAD standards*, are guidelines that help you name the files that are created inside or outside of the Autodesk® AutoCAD® software for a project, the named objects that are used within a drawing file, and the file formats that you might accept. Much like the marketing or management teams use the same logos and memo templates, all drafters or professionals using AutoCAD software in your company should follow a set of company standards.

Without having a well-defined set of standards, you will find it harder to share and output files within a company, and in turn this can lead to delays and make it nearly impossible to achieve a consistent look to all the drawings that your client receives. In addition, it is difficult to customize AutoCAD to help enforce your company's standards if there really are no standards.

Can you imagine what a client might think when a company sends them a set of drawings that contain different fonts or title blocks, or the inconsistent use of lineweights?

Well-established drawing standards ensure that your drawings all look the same when they are presented to the client, and they can make it easier to

- ◆ Train new drafters and other professionals on your company's standards that use AutoCAD
- ◆ Identify which drawing and externally referenced files are associated with a project
- ◆ Determine the purpose of a named object in a drawing
- ◆ Share project files with clients and contractors because your standards are well defined

Naming Standards for Projects and Files

As you might have gathered, it is not in your company's best interest to let everyone define their own drafting standards; this same approach applies to naming standards for projects or the files associated with a project, and how files should be stored. At the end of the day, the files created are owned by the company, and there is a pretty good chance that more than one individual will be working on a project over its entire lifetime. There is nothing more frustrating than when changes to a project are requested and the files can't be located because they are missing or no one understands how the files were saved.

The first step your company should consider if you have no current file-naming standard, or if you are considering a change to your existing system, is a way to log and name a project.

Project logging can be as simple as posting a spreadsheet on the network drive or using a project-collaboration site on Microsoft SharePoint to ensure everyone is logging projects using the same system. Once the logging system is determined, you can determine how projects and files should be named. As with the project-logging system, all files should be stored in a central location on a networked drive or a system that allows you to check files in and out, such as Autodesk Vault.

You can take two approaches to the way you name projects and files: you can establish a system yourself using the guidelines that I offer in the next few sections, or you can use the standards set by a consortium or other professional governing body. Based on your industry or the country you work in, you might consider the guidelines established by the American Institute of Architects (www.aia.org), National Institute of Building Sciences (www.nibs.org), Royal Institute of British Architects (www.architecture.com), or American National Standards Institute (www.ansi.org).

Project Names

The project-naming structure you choose to use should be short and sequential. For example, you might consider just a basic numeric value such as 000001, 000002, and so on. I do not recommend that you use the year as part of the numbering system since projects can span multiple years.

Project names commonly include an alphabetic prefix with one or more letters to make it easy to start a secondary naming system if you decide to organize your projects by business type or some other classification. For example, say your company works on residential, commercial, and government projects. In this case, you might consider prefixing the project's number with R, C, or G, so they would be R00001, C00002, or G00003.

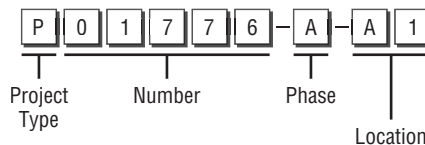
Other information you might want to represent as part of a project name could be

Phase Often a numeric value of one alphabetic letter or number (for example, -A or -1 to represent the first phase of a project).

Location Optional; often a combination of alphabetic letters and numbers to help identify multiple locations on a single and very large job site (for example, -A1 to represent the first location and -B1 to represent a second location). Using a Building attribute in a filename might be a better choice for you since all work is being done under a single project instead of multiple projects for a single job.

If a project has only a single phase, it is up to you to determine whether you want to indicate that as part of a project name. A project always has a first phase, but it might never have a second phase, based on the size of the project. Information such as floor, discipline, file type, and sheet type should be reserved for use by files within a project and not included as part of a project's name. Figure 1.1 shows what a structure for a project number might look like.

FIGURE 1.1
Possible project numbering structure



The project-naming information I've presented is only one of many possible ways you can define the numbers for your projects. Just keep in mind that project numbers should be short

and sequential. No matter how you choose to name your projects, be sure to document your system and follow it for every project you create. Documenting the system will be important for those inside and outside your company.

After you have defined the requirements and structure for your project name, assign that name to the folder on the network in which all files related to that project should be stored. Alternatively, you can use the name as an attribute in your file-management system of choice.

Filenames

There are two schools of thought when it comes to naming the files that are part of a project: one is to include the project name as part of the filename and the other is not to include it, since it is already the project folder's name. I subscribe to the school of thought that a project name should be part of a file's name, and my main reason is that a file can accidentally be placed in the wrong project folder. If the project name is not part of the filename, the file in a way could be lost forever. No matter which approach you choose, you will want to be consistent. Either prefix all your files with the project name, or don't.

Similar to a project name, the files in a project should use consistent, short, and meaningful names. A basic filename might contain the following information:

Discipline Often a single letter that represents the main discipline that the drawing is used by (for example, A for Architecture, C for Civil, M for Mechanical, or S for Structural).

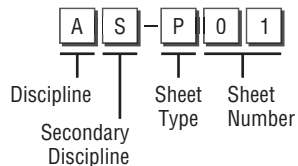
Secondary Discipline Often a single alphabetic letter that helps provide an additional level of classification for the file based on the designated main discipline (for example, D for details, G for grading, L for lighting grid, or S for sitework).

Sheet/File Type Often a single alphabetic letter used to identify the contents of a file—for example, -P for plans, -G for columns grid, and -I for images. The same can also be represented by numbers—for example, -1 for plans, -0 for columns grid, and -9 for images. You might want to consider also adding an -X when the file should be used only as an external reference across multiple drawing files.

Sheet Number Often a sequential numeric value of two numbers that range from 00 through 99 to uniquely identify a file from other files that might contain all the same file attributes in the project (for example, 01 or 76).

Figure 1.2 shows a possible structure for a filename based on the file information described.

FIGURE 1.2
Possible file-naming structure



The following is some additional information you might want to consider adding to your filenames:

Sheet Size Optional; often a single alphabetic letter used in combination with the sheet number to represent the paper size that the file should be output on (for example, A for an ANSI A-size [8 1/2" × 11"] or D for an Architectural D-size [24" × 36"]).

Building Often a single alphabetic letter used in combination with *floor* to indicate which building the file belongs to (for example, -A or -B).

Floor Often two numbers to indicate which floor of a building the file belongs to (for example, -01 or -22).

Area Optional; a single alphabetic letter used in combination with *floor* to help identify a specific area on a floor when a floor is broken up into one or more files (for example, A or C).

Revision Often the letter R or RV followed by a numeric value to represent the current revision level of a file (for example, -R1 or -RV03). Using revision numbers in a filename has its pros and cons. One of the benefits is that you can go back to an earlier revision of a design if you do not use a system that supports version tracking. The downside is that it affects the other drawings that might reference the drawing; to avoid this, you could create a copy of the drawing being revised and rename the copied file to include the revision number in its name.

Not all of the attributes will make sense for each discipline. For example, a civil drawing will most likely not contain a Building or Floor attribute, but it might contain an Area. So, it is possible that you might use different file-naming structures for different disciplines in your company. If you use different naming structures for different disciplines, be sure the attribute values have the same meaning across the entire company. For instance, do not use numbers to indicate Areas for civil, and alphabetic letters for the architectural drawings.

Managing Standards with Drawing Templates

When you create a new drawing, you have two choices: start from scratch or use a drawing template (DWT) file. Starting from scratch, or using the default drawing that is created when AutoCAD first starts up, is not ideal as it most likely does not conform to your company's standards and the settings can change from release to release. The same is true for the drawing template files that come with AutoCAD: you can use them, but they are not tailored to your company's standards. The default drawing templates make for an excellent starting point, but you should create your own drawing template files so that you know what's in them.

What Is a Drawing Template?

A drawing template is a file that contains the objects, styles, unit of measurement, and other settings that should be used when creating a new drawing with the `new` or `qnew` commands. It has a file extension of `.dwt`. Prior to drawing templates, they were called prototype drawings; you should know that just in case you hear that term come up in a conversation with an AutoCAD veteran.

When a new drawing is created using a DWT file, the DWT file is copied into memory as a new drawing and the DWT file remains unchanged. A DWT file is identical to a drawing (DWG™) file. While you commonly use the `saveas` command to save a DWG file as a DWT file, you could also just change the file extension of a DWG file from `.dwg` to `.dwt` and achieve the same results.

The following steps explain how to save a DWG file as a DWT file using AutoCAD on Windows:

1. In the AutoCAD software, open the DWG file that you want to save as a DWT file and click the Application button \triangleright Save As (or at the command prompt, you could enter **saveas** and press Enter).



2. In the Save Drawing As dialog box, click the Files Of Type drop-down list and choose AutoCAD Drawing Template (*.dwt).

The AutoCAD software changes the Save In location to the location specified by the Drawing Template File Location node on the Files tab of the Options dialog box (options command).

3. In the File Name box, enter a name for the new drawing template file. Click Save.

If you are using AutoCAD on Mac OS, use the following steps:

1. With AutoCAD as the active application, open the DWG file that you want to save as a DWT file and click File > Save As (or at the command prompt, you could enter **saveas** and press Enter).
2. In the Save Drawing As dialog box, click the File Format drop-down list and choose AutoCAD Drawing Template (*.dwt).

AutoCAD changes the current Save To location in the Where drop-down list to the location specified by the Drawing Template File Location node on the Application tab of the Application Preferences dialog box (options command).

3. In the Save As box, enter a name for the new drawing template file. Click Save.

NOTE Drawing standards (DWS) files are always saved in the latest file format. If you support multiple releases of AutoCAD in your company, you will want to instead save your standards to a DWG file with the oldest file format required. Once the DWG file is created, you can then just change the file's extension through the operating system.

Units of Measurement and Format

The drawings that you create in AutoCAD are based on one of two *systems* of measurement: *Imperial* or *English*, and *metric*. Imperial measurement is based on inches, and metric measurement is based on meters. The system of measurement that a drawing currently uses is stored in the measurement system variable; 0 (Imperial units) or 1 (metric units). Changing the value of the measurement system variable does not affect the objects that are already in a drawing. Resizing objects in a drawing to fit the new system of measurement can be done with the `scale` command.

The measurement system does not affect the formatting of linear and angular units, but it does control the following:

- ◆ The default drawing template used for the initial drawing that is created when AutoCAD first starts up: `acad.dwt` for Imperial measurement and `acadiso.dwt` for metric measurement
- ◆ Which hatch pattern and linetype definition files AutoCAD looks for in its support-file search paths: `acad.pat` and `acad.lin` for Imperial measurement and `acadiso.pat` and `acadiso.lin` for metric measurement
- ◆ The current measurement choice of the Scale List area in the Default Scale List dialog box of the Options dialog box (Windows) and Application Preferences dialog box (Mac OS)

The current measurement system does not affect the way linear and angular drawing units are accepted or displayed. Drawing-unit formatting is controlled by several settings, which you can change by using the Drawing Units dialog box (units command); see Figure 1.3.

FIGURE 1.3
Drawing Units dialog box. The Windows version of the dialog box appears on the top; the Mac OS version appears on the bottom.



Each of the settings shown in the Drawing Units dialog box can also be accessed using system variables. Understanding the relationship between many of the AutoCAD dialog boxes and the system variables they represent will help you automate tasks related to drawing setup, configure drafting aids, and control the default behavior of commands. Table 1.1 explains the system variables that can be changed using the Drawing Units dialog box.

TABLE 1.1: System variables

VARIABLE	CONTROLS
lunits	Current linear unit mode
luprec	Number of decimal places or the precision in which linear units should be displayed
aunits	Current angular unit mode
auprec	Number of decimal places or the precision with which angular units should be displayed
angdir	Positive direction for angles relative to the active User Coordinate System (UCS)
angbase	Direction of Angle 0 relative to the active UCS
insunits	Units used to properly scale an image or block being attached or inserted into the drawing
lightingunits	Controls the use of generic or photometric lighting along with the lighting units that should be used when rendering the drawing

You can learn more about system variables, including how to change their values, later in this chapter in the section “Working with System Variables.” For now, just understand that system variables play an intricate part in being able to customize AutoCAD.

Graphical and Named Objects

As I previously mentioned, a drawing template (DWT) file is the same as a drawing (DWG) file; it can contain both graphical and nongraphical objects. The graphical and nongraphical objects in a DWT file should be limited in number and common to all the new drawings you create. If you have named objects that might be less frequently used, you could store them in a drawing file and then insert that drawing into your drawing to inherit the additional named objects. Or you may choose to automate the creation process of the named objects using one of the available customization methods mentioned later in this book. I cover the creation and management of named objects in Chapter 2, “Working with Named Objects.”

Most DWT files have no objects in model space but do have objects placed on one or more of the named layouts in the file. Named layouts commonly have viewports as well as a number of objects or a single block that defines a title block used to frame your design when outputting. The title block could be an external reference (xref) or a block that has already been defined within the DWT file. I cover defining blocks in Chapter 3, “Building the Real World One Block at a Time.” General notes, callouts, symbols, and a revision table are other objects that you might place on a layout as well in your DWT file.

The named objects that you should add to your DWT files are as follows:

- ◆ Blocks
- ◆ Detail view styles
- ◆ Dimension styles
- ◆ Layers
- ◆ Linetypes
- ◆ Multileaders
- ◆ Multiline styles
- ◆ Section view styles
- ◆ Table styles
- ◆ Text styles

If you work on 3D models, you might also want to consider including UCSs, model-space viewports, saved views, materials, and visual styles. I cover the creation and management of named objects in Chapter 2.

Organize Output with Layouts

Each drawing template (DWT) file contains at least one named layout, and it is a good practice to make sure that the layout is ready to be plotted after a drawing is created using the drawing template. A drawing template can contain more than one layout, but each layout should add some value. If you need additional layouts in your drawing, you could use the `layout` command to duplicate a layout that already exists in the drawing or to import a layout from another drawing file.

The layouts in a drawing template should contain the following:

- ◆ Page-setup settings that define the output device, paper size, and other settings that impact the layout when it is plotted or published. I cover configuring plotter devices and plot styles in the section “Defining Plotter Configurations and Plot Styles” later in this chapter.
- ◆ A title block that contains all the informational fields needed to identify the drawing once it is plotted or published. A title block might include your company name and address, project name and address, part number, scale, and other information that helps identify the drawing. I discuss creating blocks and attributes in Chapter 3.
- ◆ One or more floating viewports that define which areas of model space should be plotted as part of the layout, along with the scale the objects in the viewports should be viewed at.

You might also want to consider adding the following to a drawing for use with or on a layout:

- ◆ General notes or disclaimers, and viewport labels that each or most of the drawings created with the drawing template might contain.
- ◆ A border around the margins of the paper. The viewports, title block, and other objects on the layout should be inside the border.
- ◆ A plot stamp that indicates when the drawing was plotted and the name of the source drawing file.
- ◆ Dimension and text styles set up for adding dimensions and annotation to a layout. Remember that layouts are plotted at a 1:1 scale, so the styles need to be defined correctly.

Working with System Variables

System variables in many ways are like the keys hiding under the doormat at the front door of a house. They allow you to directly access many of the settings that can be used to manipulate the AutoCAD environment and control how commands work, instead of you resorting to a dialog box or using options with a command. The values of system variables are stored in one of two places: with the drawing or as part of the user's AutoCAD profile. The user's AutoCAD profile is stored in the Windows Registry, or in several Plist files on Mac OS.

You use the `setvar` command to list and change the current value of a system variable. After you start the `setvar` command, you can enter the name of the system variable to work with or `?` (question mark) to return a list of all values or a filtered list using a wildcard match of the system variables that are supported.

The following steps explain how to list and change the value of a system variable:

1. At the command prompt, enter **setvar** and press Enter.
2. At the Enter *variable name* or `[?]`: prompt, enter the name of the system variable whose value you want to view or change.

For example, enter **cursorsize** and press Enter. The `cursorsize` system variable controls the size of the crosshairs in the drawing window.

3. At the Enter *new value for system_variable_name* `<current_value>`: prompt, enter a new value for the system variable.

The current value is displayed between the angle brackets. Pressing Enter without providing a new value ends the command and does not change the value of the system variable.

TIP The name of a system variable can also be entered directly at the command prompt just like any other command. Doing so avoids having to first start the `setvar` command.

You can list all supported variables by doing the following:

1. At the command prompt, enter **setvar** and press Enter.
2. At the Enter *variable name* or `[?]`: prompt, enter `?` and press Enter.
3. At the Enter *variable(s) to list* `<*>`: prompt, press Enter to list all supported system variables or enter a wildcard search string.

For example, you can enter the wildcard search string **DIM*** to list all the system variables that start with the letters DIM.

All the system variables that match the previous entered value are returned.

If the message `Press ENTER to continue:` is displayed in the command-line window, it indicates that there are additional results that have not yet been displayed. Press Enter to continue paging through all the returned system variables. Continue pressing Enter until you reach the last page of the results returned, or press Esc when you are done paging through the system variables.

MANAGING DRAWING TEMPLATE FILES

After you spend the time creating and customizing your drawing template (DWT) files, you will want to make sure that all users in your company have access to them. By default, AutoCAD looks in a local folder on each workstation for the DWT files it can use. This local folder is specified in the Options dialog box (Windows) or Application Preferences dialog box (Mac OS).

Using a local folder is not ideal for maintaining these files; it is best to change the location AutoCAD looks in to a folder on a networked drive. A networked folder makes it easy to update your drawing template files; post the files once, and then everyone always has access to the latest version.

If you need to use a local folder for your DWT files—when you have remote users, for example—it is good practice to keep your customized files separate from those that come with AutoCAD. Doing so makes it easy to back up your custom files and also removes the temptation to stray from your company's CAD standards. The recommended process for managing local DWT files is to first create a company folder on the local drive, and then create a Template subfolder and add your DWT files to the Template folder. Keeping the files synchronized from a network or remote server can be done using a batch (BAT) or BASH script (SH), Windows Task Scheduler event, or login script to synchronize the files from a network or remote server.

Follow these steps to specify a different folder for AutoCAD on Windows to look in for DWT files:

1. In Windows Explorer or File Explorer, browse to the folder that you created for this book under My Documents (or Documents) or create a new folder that you will use to store your DWT files.
 - ◆ On the keyboard, press the Windows+E key combination, or right-click in the lower-left corner of the screen (not the AutoCAD application window) and click Windows Explorer or File Explorer, based on your operating system.
 - ◆ In Windows Explorer or File Explorer, navigate to the folder where you want to store your DWT files.
 - ◆ Right-click in an empty area in the Folders/Files list and click New Folder. Enter the name **Templates** or the name you want to use. Press Enter to accept the new name.
2. Click the Application button > Options.
3. In the Options dialog box, click the Files tab.
4. Expand the Templates Settings node, and then expand the Drawing Template File Location node.
5. Double-click the folder path listed under the Drawing Template File Location node.
6. In the Browse For Folder dialog box, browse to and select the folder that contains or will contain your DWT files. Click OK.
7. Click OK to return to your drawing.
8. Click the Application button > New. Click Cancel.

You should notice that the Select Template dialog box now opens to the location you specified in step 6.

If you are using AutoCAD on Mac OS, use these steps:

1. In Finder, browse to the folder that you created for this book under Documents or create a new folder that you will use to store your DWT files.
 - ◆ In the Mac OS Dock, click Finder or from the desktop click File > Computer.
 - ◆ In Finder, navigate to the folder where you want to store your DWT files.
 - ◆ Ctrl-click or secondary-click in an empty area in the Folders/Files list and click New Folder. Enter the name **TempLates** or the name you want to use. Press Enter to accept the new name.
2. Click AutoCAD <release> menu > Preferences.
3. In the Application Preferences dialog box, click the Application tab.
4. Expand the Templates Settings node, and then expand the Drawing Template File Location node.
5. Double-click the folder path listed under the Drawing Template File Location node.
6. In the Open dialog box, browse to and select the folder that contains or will contain your DWT files. Click Open.
7. Click OK to return to your drawing.
8. Click File > New Drawing. Click Cancel.

You should notice that the Select Template dialog box now opens to the location you specified in step 6.

In addition to specifying the location of the Select Template dialog box, you can specify the default DWT file that is used with the `qnew` (Quick New) command. You specify the DWT file for the `qnew` command using the Options dialog box (Windows) or Application Preferences dialog box (Mac OS).

Use these steps to specify which DWT file should be used when the `qnew` command is executed in AutoCAD on Windows:

1. Click the Application button > Options.
2. In the Options dialog box, click the Files tab.
3. Expand the Templates Settings node, and then expand the Default Template File Name for QNEW node.
4. Double-click the filename listed under the Default Template File Name For QNEW node.
5. In the Select A File dialog box, browse to and select the DWT file that you want to use with the `qnew` command. Click Open.
6. Click OK to return to your drawing.
7. On the Quick Access toolbar, click New. A new drawing file is created based on the DWT file you selected in step 5.

If you are using AutoCAD on Mac OS, use the following steps:

1. Click AutoCAD <release> menu > Preferences.
2. In the Application Preferences dialog box, click the Application tab.
3. Expand the Templates Settings node, and then expand the Default Template File Name For QNEW node.
4. Double-click the filename listed under the Default Template File Name For QNEW node.
5. In the Open dialog box, browse to and select the DWT file that you want to use with the qnew command. Click Open.
6. Click OK to return to your drawing.
7. At the command prompt, enter **qnew** and press Enter. A new drawing file is created based on the DWT file you selected in step 5.

Choosing a File Format for Your Drawings

Out of the box, AutoCAD saves all the drawings you create or open to the latest file format. This is done to ensure that any of the objects you create are properly stored and can be restored when the drawing is opened later. Using the current drawing file format is not always the best choice; perhaps you are working with another department that needs access to the drawings you create and they are still on an older release, or the contract you are bidding on requires all the files for a project to be submitted in a specific format.

No matter the reasoning, AutoCAD allows you to set the default file format it uses when saving a drawing instead of requiring you to change to an earlier format from the Files Of Type (Windows) or File Format (Mac OS) drop-down list in the Save Drawing As dialog box. The default file format for a drawing can be set in the Options dialog box (Windows) or Application Preferences dialog box (Mac OS).

Follow these steps to change the default file format used when saving a drawing file in AutoCAD on Windows:

1. Click the Application button > Options.
2. In the Options dialog box, click the Open And Save tab. In the File Save area, click the Save As drop-down list and then choose the default file format you want to use.
3. Click OK.

If you are using AutoCAD on Mac OS, use the following steps:

1. Click AutoCAD <release> menu > Preferences.
2. In the Application Preferences dialog box, click the General tab. Then choose the default file format you want to use from the Save As drop-down list in the File Save area.
3. Click OK.

TIP Before you save a drawing file to an earlier file format, you will want to know which releases your users are working in. I don't recommend just saving all your drawing files to the oldest possible release, as that could result in some undesired results. Some objects in a drawing could disappear or be exploded into individual objects.

Table 1.2 shows which file format you should use to save your drawings so that users on an older release can open your files.

TABLE 1.2: AutoCAD releases and drawing file formats

AUTOCAD RELEASE(S)	DRAWING FILE FORMAT
AutoCAD 2013 and AutoCAD 2014	AutoCAD 2013
AutoCAD 2010, AutoCAD 2011, and AutoCAD 2012	AutoCAD 2010
AutoCAD 2007, AutoCAD 2008, and AutoCAD 2009	AutoCAD 2007
AutoCAD 2004, AutoCAD 2005, and AutoCAD 2006	AutoCAD 2004
AutoCAD 2000, AutoCAD 2000i, and AutoCAD 2002	AutoCAD 2000
AutoCAD Release 14	AutoCAD R14
AutoCAD Release 13	AutoCAD R12 DXF
AutoCAD Release 12	AutoCAD R12 DXF

If you want to make sure that all the files in a project are saved in a specific file format, you can use one of the following tools:

Autodesk DWG TrueView A drawing file conversion program that is standalone from AutoCAD. It's available on Windows only and can be downloaded from www.autodesk.com/dwg.

AutoCAD 360 Online collaboration website that allows you to view, edit, and share DWG files with others on a project. After a drawing has been uploaded, it can be downloaded in a different file format. The AutoCAD 360 website can be found at <https://www.autocadws.com/>.

ETRANSMIT command Command that can be used to resave all the drawing files in a sheet set, or in the drawing files you add to the Create Transmittal dialog box. It's available on Windows only.

Script Pro Utility that allows you to run script files on selected drawing files. The utility is available for Windows only and can be downloaded from here:

<http://usa.autodesk.com/adsk/servlet/item?siteID=123112&id=4091678&linkID=9240618>

Script Files and SCRIPT command You can create a script file that opens, saves, and closes multiple drawing files. I cover creating script files in Chapter 8, "Automating Repetitive Tasks."

Defining Plotter Configurations and Plot Styles

Plotting, printing, and publishing are all forms of outputting a drawing file to a hardcopy (a physical sheet of paper) or an electronic representation of a hardcopy. Doing so helps you keep

your intellectual property in your drawings secure, and it also gives those that do not have access to or know how to use AutoCAD the ability to review and approve your drawings.

Before you can output a drawing, you need to

- ◆ Set up an output device
- ◆ Define the plot styles that control the way drawing objects appear in the output
- ◆ Configure the settings of model space or a named layout

Configuring Output Devices

Setting up an output device is commonly handled by installing a system printer in Windows or Mac OS, but AutoCAD on Windows also supports a second option that allows you to configure a nonsystem printer using custom device drivers. Nonsystem printers are stored in PC3 files. Once a device is set up, additional settings can be specified inside AutoCAD to control the output being sent to the device. After a device is configured, you can then assign the device directly to a layout or page setup, or set it as the current device when using the `plot` command.

You can configure a nonsystem printer for use with AutoCAD on Windows by doing the following:

1. Click the Application button > Print > Manage Plotters.
2. In the Plotters window, double-click Add-A-Plotter Wizard.
3. In the Add Plotter wizard, click Next on the Introduction page.
4. On the Begin page, select one of the three types of plotters that can be added:
 - ◆ My Computer: The device is configured to use a port on your local computer, to plot to a file, or to use an AutoSpool utility (an application that controls how the plot file is handled).
 - ◆ Network Plotter Server: The device is configured by posting plot files to a network location where the plotter checks for new files that need to be plotted.
 - ◆ System Printer: The device is configured to use an installed system printer and allows you to control the properties of AutoCAD-specific output settings.

Then click Next.

5. You will see a different page based on which option you selected:
 - ◆ If you chose My Computer, the Plotter Model page is displayed. Select a manufacturer and one of the supported plotter models. Click Next.
 - ◆ If you chose Network Plotter Server, the Network Plotter page is displayed. Enter or select the name of the network server (UNC) that you want to use, and click Next. The Plotter Model page is displayed. Select a manufacturer and one of the supported plotter models. Click Next.
 - ◆ If you chose System Printer, the System Printer page is displayed. Select a printer that is installed under the operating system and click Next.
6. If the Import PCP Or PC2 page is displayed, click Import if you have a PCP or PC2 file from AutoCAD Release 14 or earlier that contains plotter-specific information. Select the PCP or PC2 file and click Open. Click Next to move to the next page in the wizard.

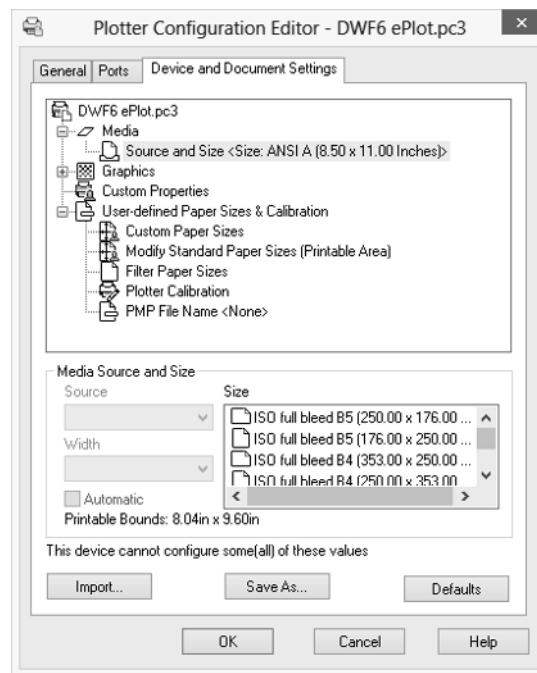
7. If you chose System Printer in step 4, the Ports page is displayed. Select a port on your local computer, plot to a file, or use an AutoSpool utility. Click Next.
8. On the Plotter Name page, enter a name in the Plotter name text box and click Next.
9. On the Finish page, click Edit Plotter Configuration to make changes to the device's settings in the Plotter Configuration Editor (see Figure 1.4) or click Calibrate Plotter to adjust the output size compared to that of a test drawing in the Calibrate Plotter wizard. Both of these options are optional, and availability is based on the type of device you are adding along with the plotter model you specified. Click Finish.

The new plotter is added to the Plotter window and is ready for use by AutoCAD.

Do the following to edit a nonsystem printer in AutoCAD on Windows:

1. Click the Application button > Print > Manage Plotters.
2. In the Plotters window, double-click the PC3 file that contains the nonsystem printer configuration you want to edit. The Plotter Configuration Editor (see Figure 1.4) is displayed.

FIGURE 1.4
Editing a plotter
configuration file



3. Optionally, in the Plotter Configuration Editor on the General tab, enter a description for the plotter configuration in the Description text box.
4. Click the Ports tab and change the current port, plot to file, or AutoSpool utility settings.
5. Click the Device And Document Settings tab and change device settings as needed. Click OK.

After a system printer is installed or a plotter configuration is created, you can use it to output the objects in model space or on a named layout. The following steps explain how to assign a plot configuration to a layout or page setup and how to use it with the plot command:

- ◆ For a layout or page setup, click the Application button > Print > Page Setup (Windows) or click File > Page Setup Manager (Mac OS). In the Page Setup Manager, select the layout or page setup you want to change, and then click Modify on Windows or click the Action menu (gear icon) > Edit on Mac OS to display the Page Setup dialog box. In the Page Setup dialog box, in the Printer/Plotter area, choose a device from the Name (Windows) or Printer (Mac OS) drop-down list. Click OK. Click Close to exit the Page Setup Manager.
- ◆ For plotting or printing with the plot command, click the Application button > Print (Windows) or click File > Print (Mac OS). In the Plot dialog box (Windows) or Print dialog box (Mac OS) in the Printer/Plotter area, choose a device from the Name (Windows) or Printer (Mac OS) drop-down list. Click OK (Windows) or Print (Mac OS) to output the drawing.

Using and Creating Plot Styles

Plot styles are used to control the way drawing objects appear onscreen and when they are output to hardcopy (a physical sheet of paper) or an electronic representation of a hardcopy. AutoCAD supports two types of plot styles: *color dependent* and *named*.

Color-dependent plot styles, stored in CTB files, are the most common of the two styles that AutoCAD supports. They are the way object properties were applied during plotting in AutoCAD prior to the introduction of named plot styles in AutoCAD 2000. With color-dependent plot styles, as the name reflects, the color assigned to an object in a drawing controls the object properties that are assigned during plotting. Even though your drawings can use true colors, plot styles are based on the AutoCAD Color Index (ACI) system of 255 colors, and true color values are therefore mapped to their nearest ACI value when plotting.

Named plot styles, stored in STB files, are the newest way to control object properties when plotting. First introduced in AutoCAD 2000, this style uses a name-based system instead of being dependent on the color of an object. Although this might sound ideal, there are a few places where color can be assigned only to objects in a drawing or style. For example, you can only set the color of the grid lines in a table or the dimension line of a dimension style.

TIP You can convert a drawing from the color-dependent plot style to the named plot style, and vice versa, by using the `convertplotstyles` command.

Both types of plot styles have their advantages. Color dependent is based on the legacy system and is thus compatible with decades of drawings, making it the clear choice for most companies. It is also the style that provides the most control over complex objects and many styles. Switching to named plot styles requires a bit of planning and time, which at the end of the day keeps it from being an option for many companies. Here are a few of the issues you need to consider if you plan on adopting named styles:

- ◆ Existing block libraries need to be updated to ensure they display correctly using named plot styles.
- ◆ Dimension and table styles might need to be updated to ensure they use a single color and that they use lineweights and linetypes to control the way they output.

- ◆ Color-dependent drawings need to be updated to use named plot styles with the `convertplotstyles` command. Having a mix of plot styles in the same project does not give your drawings a consistent look when they are plotted.

You can create a plot style on Windows by doing the following:

1. Click the Application button > Print > Manage Plot Styles.
2. In the Plot Styles window, double-click Add-A-Plot Style Table Wizard.
3. In the Add Plot Style Table wizard, click Next on the Introduction page.
4. On the Begin page, select one of these four options to create a plot style:
 - ◆ **Start From Scratch:** Use this option when you want to create a new color-dependent (CTB) or named (STB) plot style file.
 - ◆ **Use An Existing Plot Style Table:** This option creates a copy of an existing CTB or STB file, and then allows you to edit the copied file.
 - ◆ **Use My R14 Plotter Configuration (CFG):** If upgrading from AutoCAD Release 14, you can import your settings to create a new plot style file.
 - ◆ **Use A PCP Or PC2 File:** If you have a PCP or PC2 file that was exported from an earlier release, you can import the pen table properties and create a new plot style file.

Then click Next.

5. If the Table Type page is displayed, click Color-Dependent Plot Style Table or Named Plot Style Table. Click Next.

If the Browse File Name page is displayed, browse to and select the type of file that is expected and any additional information that is needed for the type of file you selected. Click Next.
6. On the File Name page, enter a name in the File Name text box for the new plot style file. Click Next.
7. On the Finish page, click Plot Style Table Editor to make changes to the new plot style table in the Plot Style Table Editor (see Figure 1.5). Click Finish.

If you are using AutoCAD on Mac OS, use the following steps to create a new plot style file:

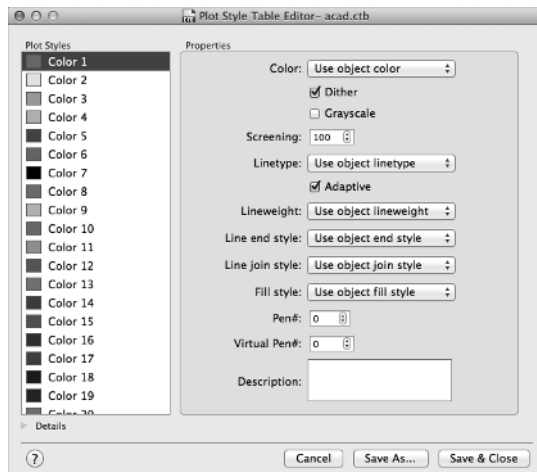
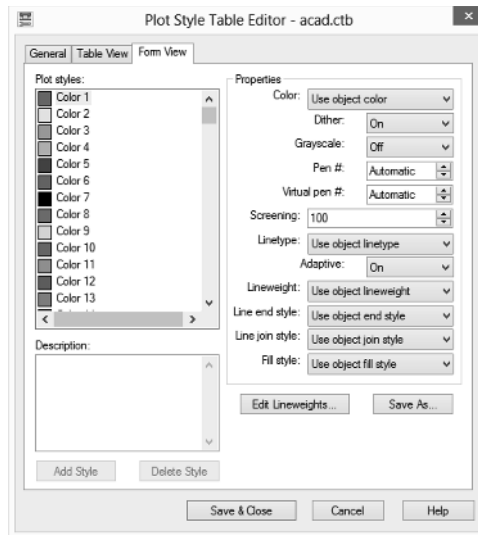
1. Click File > Plot Styles.
2. In the Plot Styles window, double-click one of the plot styles list. The Plot Style Table Editor (see Figure 1.5) is displayed.
3. From the AutoCAD Plot Style Editor menu bar, click File > New Color-Based Plot Style Table (CTB) or New Named Plot Style Table (STB).
4. In the Plot Style Table Editor, click Save As.
5. In the Save dialog box, enter a name for the new file in the Save As text box and specify a location to save the file to. Click Save.

6. In the Plot Style Table Editor, make the changes to the plot styles properties and click Save & Close.

If you want to make changes to an existing plot style file, do the following:

1. Click the Application button > Print > Manage Plot Styles (Windows) or click File menu > Plot Styles (Mac OS).
2. In the Plot Styles window, double-click the plot style you want to edit. The Plot Style Table Editor (see Figure 1.5) is displayed.

FIGURE 1.5
Editing a plot style file. The Windows version of the dialog box appears on the top and the Mac OS version on the bottom.



3. In the Plot Style Table Editor, edit the properties of the plot style as needed. If you are working with a named plot style, create any new named styles that you need. In Windows, click the Form View tab; it is the easiest way to work with the plot styles in the file.

4. Click Save & Close to save the changes you made.

After a plot style has been created, you can use it control how the objects of your drawing appear when output. The following steps explain how to assign a plot style to a layout or page setup, and how to use it with the plot command:

- ◆ For a layout or page setup, click the Application button > Print > Page Setup (Windows) or click File > Page Setup Manager (Mac OS). In the Page Setup Manager, select the layout or page setup you want to change, and then click Modify on Windows or click the Action menu (gear icon) > Edit on Mac OS to display the Page Setup dialog box. In the Page Setup dialog box, in the Plot Style Table area, choose a plot style from the drop-down list. If prompted to apply the plot style to all layouts, click Yes. On Mac OS only, you might need to click the More Options button in the lower-right corner of the dialog box to see the Plot Style Table area. Click OK. Click Close to exit the Page Setup Manager.
- ◆ On Windows, for plotting with the plot command, click the Application button > Print. In the Plot dialog box, click the More Options button in the lower-right corner of the dialog box. In the Plot Style Table area, choose a plot style from the drop-down list. Click OK to output the drawing.
- ◆ On Mac OS, for printing with the plot command, click File > Print. In the Print dialog box, in the Page Setup area, click Edit Page Setup. In the Page Setup dialog box, in the Plot Style Table area, choose a plot style from the drop-down list. You might need to click the More Options button in the lower-right corner of the dialog box to see the Plot Style Table area. Click Print to output the drawing.

Managing Plotter Configuration and Plot Style Files

After you have added your output devices and created your plot styles, you will want to make sure that all users in your company have access to them. By default, AutoCAD looks in local folders on each workstation for the PC3, CTB, and STB files that it might need to output your drawing files. These local folders are specified in the Options dialog box (Windows) or Application Preferences dialog box (Mac OS). As with other custom files, such as DWT files, you should place all your common plotter configuration and plot style files on a network location.

Follow these steps to specify different folders for AutoCAD on Windows to look in for PC3 and CTB/STB files:

1. Click the Application button > Options.
2. In the Options dialog box, click the Files tab.
3. Expand the Printer Support File Path node, and then expand the Print Configuration Search Path node to change the location for PC3 files or expand the Plot Style Table Search Path node to change the location for CTB/STB files.
4. Double-click the folder path listed under the expanded node.
5. In the Browse For Folder dialog box, browse to and select the folder that contains your PC3 or CTB/STB files. Click OK.

You can also select the Print Configuration Search Path or Plot Style Table Search Path node and click the Add button to add an additional location so you have access to your custom files and those that come with AutoCAD.

6. Click OK to return to your drawing.

If you are using AutoCAD on Mac OS, use the following steps:

1. Click AutoCAD <release> menu > Preferences.
2. In the Application Preferences dialog box, click the Application tab.
3. Expand the Printer Support File Path node, and then expand the Print Configuration Search Path node to change the location for PC3 files or expand the Plot Style Table Search Path node to change the location for CTB/STB files.
4. Double-click the folder path listed under the expanded node.
5. In the Open dialog box, browse to and select the folder that contains your PC3 or CTB/STB files. Click Open.

You can also select the Print Configuration Search Path or Plot Style Table Search Path node and click the + (plus) button to add an additional location so you have access to your custom files and those that come with AutoCAD.

6. Click OK to return to your drawing.

Enforcing CAD Standards

CAD standards enforcement is a gradual process if you are not doing it today. The best strategy you have is positive reinforcement and coaching. At the end of the day, your drafters and professionals need to want to follow all the established standards. It can be tempting to take the quick and easy route to get a job done, but shortcutting a process can have unexpected consequences, such as a custom tool not working in a drawing and resulting in manual steps or a drawing not plotting correctly. After all, a process works each and every time only when each step is completed properly.

There are steps you can take to help make following CAD standards easier for the drafters and professionals you support. You can use these methods to apply and enforce your company's standards:

User-Interface Customization AutoCAD offers an interface that can be customized to the way your company works. You can create ribbon buttons or menu items that use a custom macro to set a layer or style as current before starting a command. Customizing the user interface is covered in Chapter 5, "Customizing the AutoCAD User Interface for Windows," and Chapter 6, "Customizing the AutoCAD User Interface for Mac."

Scripts Script files are a great way to execute multiple commands and options in a specific order using predefined values without needing to understand a programming language. You can create layers and styles, insert a title block on a specific layer, or even change your current drafting settings. Creating script files is discussed in Chapter 8.

Action Macros Created using the Action Recorder, action macros are a modern take on script files. They allow you to execute multiple commands and options in a specific order, but they are created interactively while you use AutoCAD. After an action macro is created, it can

be executed in other drawing files. Recording action macros is covered in Chapter 8. (Action macros are not available on AutoCAD for Mac OS.)

Custom Programs The programming languages that AutoCAD supports give you the most flexibility in enforcing CAD standards. Custom programs can be created to make sure a specific layer is current before a command is started, whether the user starts the command from the user interface or from the command prompt.

You can also use custom programs to step through the objects in a drawing and make sure they conform to your standards, and much more.

Tool Palettes An arrangement of tools in the Tool Palettes window allows you to create objects using specific property settings. You can create a Note tool that uses a specific combination of text style and layer for general notes and define another tool for disclaimers; access and insert commonly used blocks at a set scale; or fill an enclosed area with a hatch pattern, using a specific lineweight and color. I discuss tool palettes in Chapter 7, “Creating Tools and Tool Palettes.” (The Tool Palettes window is not available on AutoCAD for Mac OS.)

DesignCenter™ DesignCenter allows you to access the named objects defined in a drawing and add them to your current drawing. It also allows you to insert blocks as well as attach external drawings and raster images. I do not cover using DesignCenter in this book; use the AutoCAD Help to learn more about this feature. (DesignCenter is not available on AutoCAD for Mac OS.)

Content Explorer™ Content Explorer is a modernized version of DesignCenter that allows you to locate named objects from other drawing files on a network and add them to your current drawing. I do not cover configuring and using Content Explorer in this book; use the AutoCAD Help to learn more about this feature. (Content Explorer is not available on AutoCAD for Mac OS.)

AutoCAD on Windows offers multiple tools that are designed to help you validate named objects and their properties in a drawing against those defined in a drawing standards (DWS) file. These tools are as follows:

Standards Manager Used to configure and validate a drawing file for standards violations. Violations found can be fixed to conform to the standards you are validating against.

Batch Standards Checker Validates the standards in a set of drawing files and generates a report of all the violations found and which drawing files they were found in. You must open each drawing file manually in order to fix the violations found.

Layer Translator Used to create layer mappings and align the layers of a drawing with those defined in your standards.

Drawing Standards (DWS) Files (Windows Only)

A drawing standards (DWS) file is the same as a drawing or a drawing template file, with the exception of the content it contains and its file extension. A DWS file commonly does not contain any graphical objects, but it does contain all of the named objects that are used to define your CAD standards and your layer translation mappings for the Layer Translator. For more

information on named objects, see Chapter 2. I cover the Layer Translator in the “Translating Layers (Windows Only)” section later in this chapter.

At a minimum, your DWS files should contain the following named objects:

- ◆ Dimension styles
- ◆ Layers
- ◆ Linetypes
- ◆ Text styles

These steps explain how to save a DWG file to a DWS file using AutoCAD on Windows:

1. In AutoCAD, open or create a DWG file that contains the named objects that represent your CAD standards. Click the Application button > Save As.
2. In the Save Drawing As dialog box, click the Files Of Type drop down-list and choose AutoCAD Drawing Standards (*.dws).
3. Browse to a common location on the network to store the DWS file so that you and others in your company can access it.
4. In the File Name box, enter a name for the new drawing template file. Click Save.

Drawing standards (DWS) files are always saved in the latest file format. If you support multiple releases of AutoCAD in your company, you will want to instead save your standards to a DWG file with the oldest file format required. Once the DWG file is created, you can then just change the file’s extension through the operating system.

Configuring, Checking, and Fixing Drawing Standards (Windows Only)

The AutoCAD Standards Manager provides you with the ability to check several of the named object types in a drawing against those defined in a drawing standards (DWS) file. Using the Standards Manager requires the completion of two distinct processes: first, you associate one or more DWS files that contain your CAD standards with the current drawing, and second, you check and fix any standards violations found.

CONFIGURING STANDARDS

Configuring a drawing to be checked for standards violations requires you to associate the DWS files that contain your CAD standards and specify which plug-ins you want to use. A plug-in defines the comparison rules that should be used to find any standards violations in the current drawing against the approved CAD standards in the DWS files. You can choose from one of four plug-ins that come with AutoCAD, you can obtain plug-ins from third-party developers, or you can develop your own plug-ins using the CAD Standards Plug-in API. You associate DWS files with a drawing using the standards command, which displays the Configure Standards dialog box.

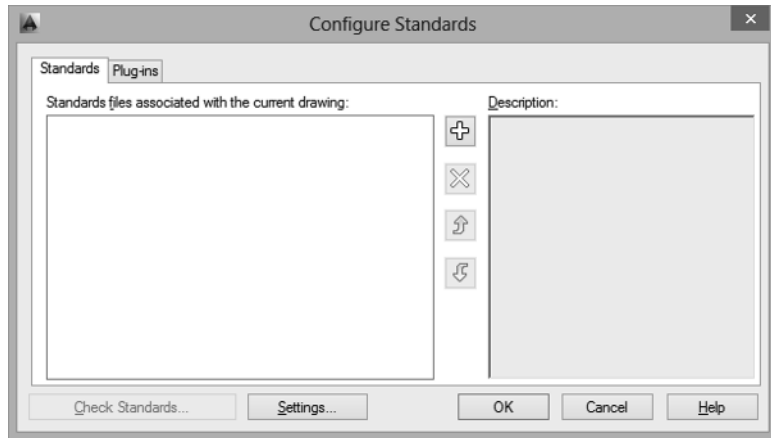
TIP You can associate DWS files with a drawing template (DWT) file. Any new drawings created from the drawing template will maintain the associations so that you do not have to go through the configuration process for each new drawing file you create.

The following steps explain how to associate a DWS file with a drawing and specify which standards plug-ins to use:



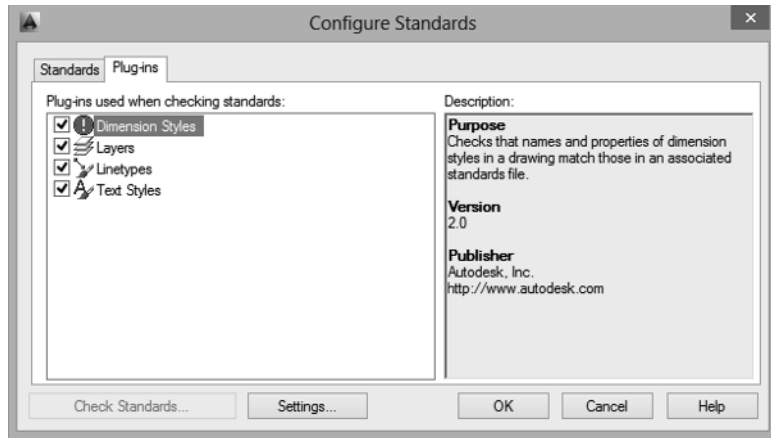
1. On the ribbon, click Manage tab > CAD Standards panel > Configure (or at the command prompt, enter **standards** and press Enter). The Configure Standards dialog box (see Figure 1.6) is displayed.

FIGURE 1.6
Configuring drawing standards files



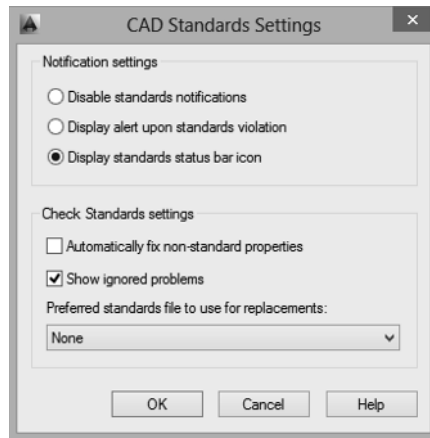
2. In the Configure Standards dialog box, click the Standards tab.
You use the Standards tab to associate DWS files with the current drawing and to control the order in which AutoCAD should search the associated DWS files when validating standards. AutoCAD starts searching for named objects that match the names of those in the current drawing from the top of the list downward.
3. Click the + (plus) button to add a new DWS file association to the current drawing. The Select Standards File dialog box is displayed.
4. In the Select Standards File dialog box, browse to and select the DWS file that you want to associate with the drawing. Click Open.
Only one DWS file can be associated at a time. Click the + (plus) button and add all the DWS files you want to check against.
5. If you add more than one DWS file, select an associated DWS file and click Move Up /Move Down to change the search order of the DWS files.
Select a DWS file and click the X to remove an associated DWS file.
6. Click the Plug-ins tab. You use the Plug-ins tab (see Figure 1.7) to specify which plug-ins should be used when checking the standards of the current drawing against the DWS files that have been associated with the drawing.

FIGURE 1.7
Enabling the plug-ins to use when validating drawing standards



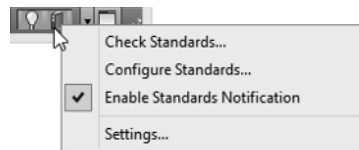
7. Specify the plug-ins you want to use when checking for standards violations.
8. Click the Settings button. The CAD Standards Settings dialog box (see Figure 1.8) is displayed.

FIGURE 1.8
Changing the notification and Check Standards settings



9. In the CAD Standards Settings dialog box, change the settings as needed and click OK.
You can change the display behavior of the CAD Standards notification balloon and icon in the drawing status bar tray (see Figure 1.9) in the Notification Settings section. The settings in the Check Standards Settings section control how errors are handled when a drawing is checked for standards violations.

FIGURE 1.9
Access to CAD Standards notifications and settings is just a click away.



- Click OK to close the Configure Standards dialog box and return to your drawing.

Based on your settings in the CAD Standards Settings dialog box, the CAD Standards icon might be displayed in the drawing status-bar tray. The tray icon, when displayed, can be used to indicate whether at least one DWS file is associated with the current drawing. Right-clicking the icon allows you to configure and check the drawing standards in your drawing.

CHECKING FOR AND FIXING STANDARDS VIOLATIONS

After you have associated at least one DWS file to a drawing and specified which plug-ins to use, you can check your drawing for standards violations. While you are checking for standards violations, each plug-in is executed one by one, and how a plug-in checks for standards violations can vary between plug-ins.

The standard four plug-ins that come with AutoCAD compare the name of an object in the current drawing to the names of objects in the associated DWS files. When a match is found, AutoCAD checks to see whether the properties of the two named objects are the same. If they are the same, AutoCAD moves on to the next named object and checks it. If the properties are different, you are asked to fix or ignore the standards violation. If no matching named object is found between the drawing and DWS files, you are prompted to select one of the approved named objects from the associated DWS files or ignore the standards violation.

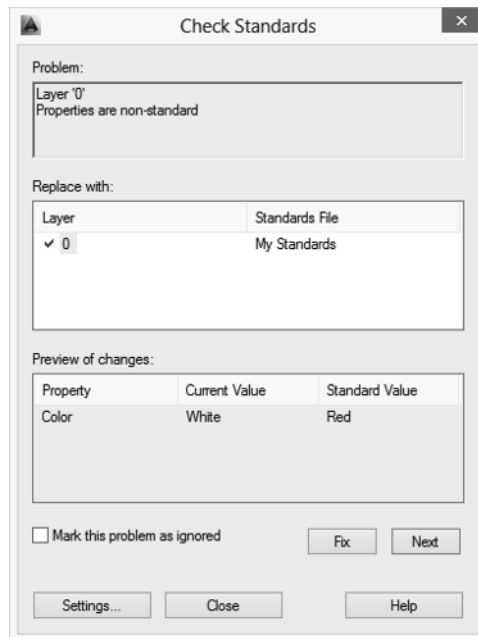
Follow these steps to check a drawing for any standards violations:



- On the ribbon, click Manage tab > CAD Standards panel > Check (or at the command prompt, enter **checkstandards** and press Enter).

The Check Standards dialog box (see Figure 1.10) is displayed. If no DWS file is associated with the drawing file, you will be presented with an error message. Click OK to configure the CAD standards settings for the drawing and then continue checking for standards violations.

FIGURE 1.10
Standards violation
found



2. In the Check Standards dialog box, in the Problem area review the first standards violation found. The Problem area provides you with a description of the standards violation.
3. In the Replace With area, choose one of the available fixes.

The Replace With area provides you with all the possible fixes that one of the plug-ins identified for the standards violation. A plug-in can recommend a fix that is indicated by a blue check mark to the left of the item. After selecting a possible fix for the standards violation, you can see an overview of the changes that will be made in the Preview Of Changes area. You can have AutoCAD automatically fix standards violations that have nonstandard properties by clicking the Settings button and enabling Automatically Fix Non-standard Properties in the CAD Standards Settings dialog box. If your DWS files offer more than one recommendation for a fix, you can specify the preferred DWS file that you want to use from the CAD Standards Settings dialog box as well.

4. Click Fix to correct or Next to skip the standards violation, and move to the next one found.

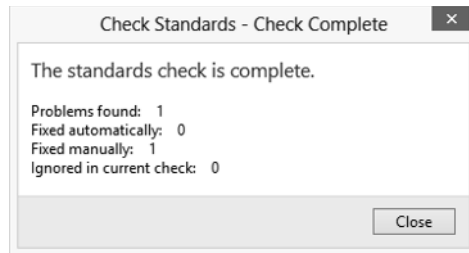
When you skip a standards violation, you can click Mark This Problem As Ignored to not have the violation show up the next time the drawing is checked. You can choose to display all ignored violations by clicking the Settings button and enabling Show Ignored Problems in the CAD Standards Settings dialog box.

5. Continue stepping through each standards violation until the Check Standards - Check Complete message box is displayed. Click Close.

The Check Standards - Check Complete message box (see Figure 1.11) provides you with a summary of the actions taken to resolve the standards violations found.

FIGURE 1.11

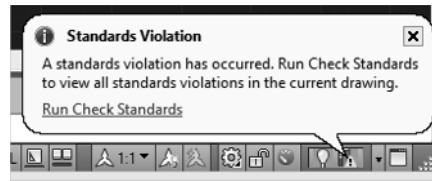
The check for standards violations is complete.



6. Click Close to close the Check Standards dialog box and return to your drawing.

While working on a drawing with CAD standards configured, you might see the Standards Violation balloon (see Figure 1.12) come up. This indicates that something in the drawing conflicts with your company's standards. A custom routine might have caused the standards violation, or maybe you inserted a drawing that contains named objects that were not updated with the latest standards. Click the Run Check Standards link in the balloon to fix or ignore the standards violation.

FIGURE 1.12
Standards violation
detected



CHECKING DRAWINGS IN BATCHES

The Check Standards dialog box is efficient for checking the standards of the current drawing, but it is not ideal if you have five, ten, or even hundreds of drawings in your project that need to be checked. When AutoCAD is installed, it also installs an external utility called the Batch Standards Checker. The Batch Standards Checker allows you to select a number of drawings and check them against the standards defined in a DWS file.

The Batch Standards Checker allows you to check the drawing files using the DWS files that are already associated with each file, or you can specify which DWS files should be used. This utility uses the same plug-ins that are available in the Configure Standards dialog box. The one feature that the Batch Standards Checker does not support is the ability to fix any of the violations found; that must be done by opening each drawing and using the `checkstandards` command.

The following steps explain how to use the Batch Standards Checker:

1. Do one of the following:

- ◆ (Windows XP and Windows 7) Click the Windows Start button > [All] Programs > Autodesk > AutoCAD <release> > Batch Standards Checker.
- ◆ (Windows 8) On the Start screen, right-click and click All Apps. Under the AutoCAD <release> category, click Batch Standards Checker.

The Batch Standards Checker dialog box (see Figure 1.13) is displayed.

- 2.** In the Batch Standards Checker, on the Drawings tab click the + (plus) button.
- 3.** In the Batch Standards Checker - File Open dialog, browse to and select the drawings and drawing templates you want to check for standards violations. Click Open.

Press and hold the Ctrl key when browsing to select more than one file.

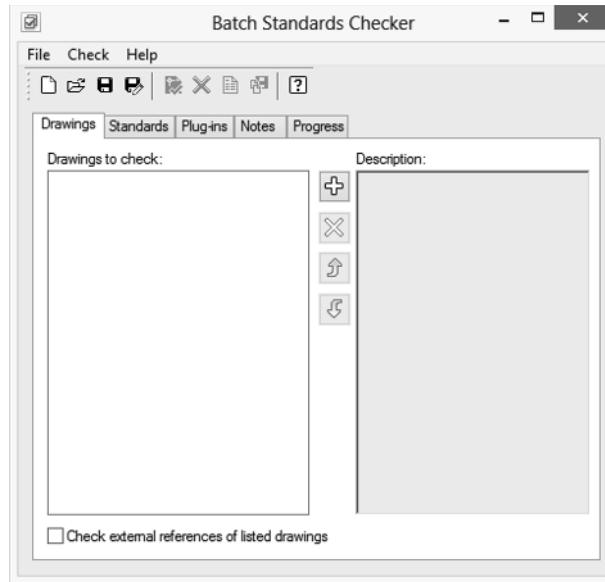
- 4.** Optionally, click the Check External References Of Listed Drawings check box.

When this option is enabled, the Batch Standards Checker will also check the external references attached to the listed files for any standards violations.

- 5.** Click the Standards tab.

This tab is similar to the Standards tab of the Configure Standards dialog box in AutoCAD. The only difference is the two options located along the top of the tab.

FIGURE 1.13
More than one drawing can be checked for standards violations with the Batch Standards Checker.



6. Choose one of the two options along the top of the tab:
 - ◆ The Check Each Drawing Using The Associated Standards Files option instructs the Batch Standards Checker to use the DWS files associated with each drawing file; if no DWS file is associated with a drawing file, the drawing is not checked for standards violations.
 - ◆ Use the Check All Drawings Using The Following Standards Files option to specify which DWS files to use for validating the CAD standards. Click the + (plus) button to list which DWS files to use. If more than one DWS file is added, select an associated DWS file and click Move Up/Move Down to change the search order of the DWS files. Select a DWS file and click the X to remove an associated DWS file.
7. Click the Plug-ins tab.

This tab is identical to the Plug-ins tab of the Configure Standards dialog box in AutoCAD.
8. Specify which plug-ins to use when checking the drawings for standards violations.
9. Optionally, click the Notes tab and enter the text for the note.

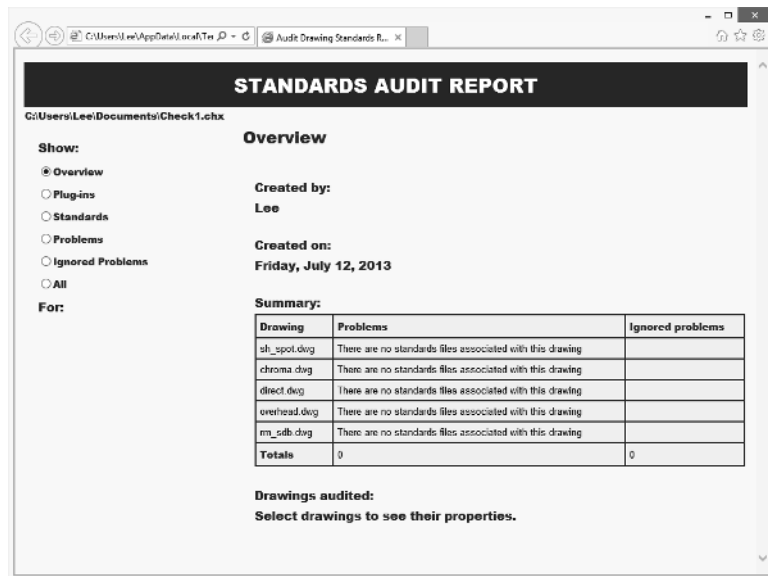
You might want to list the project's name, your name, and any other information that could be useful to the next person who views the report generated after checking the drawing files for standards violations.
10. On the Batch Standards Checker toolbar, click Save.
11. In the Batch Standards Checker - File Save dialog, browse to a location and enter a name for the Standards Check (CHX) file. Click Save.

The CHX file contains a reference to the drawings you added to the list and specifies which drawing standards (DWS) files and plug-ins to use, along with the notes and other settings you included. You might want to create a CHX file for each one of your projects and then store them with the drawing files in your project.

12. On the Batch Standards Checker toolbar, click Start Check.

The Progress tab is displayed with the current progress of the standards violation check for the drawing files that were listed on the Drawings tab. Once processing has been completed, the Standards Audit Report (see Figure 1.14) is displayed in your default web browser. Use this report to view any standards violations found. The most recent report is saved with the CHX and can be viewed by clicking Check > View Report on the Batch Standards Checker's menu bar.

FIGURE 1.14
Use this report to determine which files contain standards violations so they can be fixed.



Translating Layers (Windows Only)

Checking a drawing for standards violations is great for ensuring the files in the projects you create conform to your CAD standards, but working with drawings from a client or subcontractor can make this a bit more challenging, especially when it comes to layer standards. Over a set of drawings, you might use a few different dimension or text styles, but over those same drawings you could be working with dozens or hundreds of different layers.

AutoCAD provides a tool called the Layer Translator that allows you to map a single layer or a group of layers to a single layer based on your established CAD standards. Once a translation map is defined, it can be saved to a drawing or drawing standards file to reuse on other files from the client. This tool can also be useful in transitioning from an old to a new layer standard that your company is implementing. The Layer Translator can be displayed using the `laytrans` command.

Follow these steps to define a layer translation map and translate the layers in the current drawing to those defined by your CAD standards:



1. On the ribbon, click Manage tab > CAD Standards panel > Layer Translator (or at the command prompt, enter **laytrans** and press Enter). The Layer Translator dialog box (see Figure 1.15) is displayed.
2. In the Layer Translator, click Load in the Translate To area.
3. In the Select Drawing File dialog box, browse to and select the file that contains your CAD standards. Click Open. The layers in the file are populated in the Translate To list.
4. If your CAD standards file does not contain a layer you want to use as part of the layer translation map, click New. The New Layer dialog box (see Figure 1.16) is displayed.

FIGURE 1.15
Translating layers between CAD standards

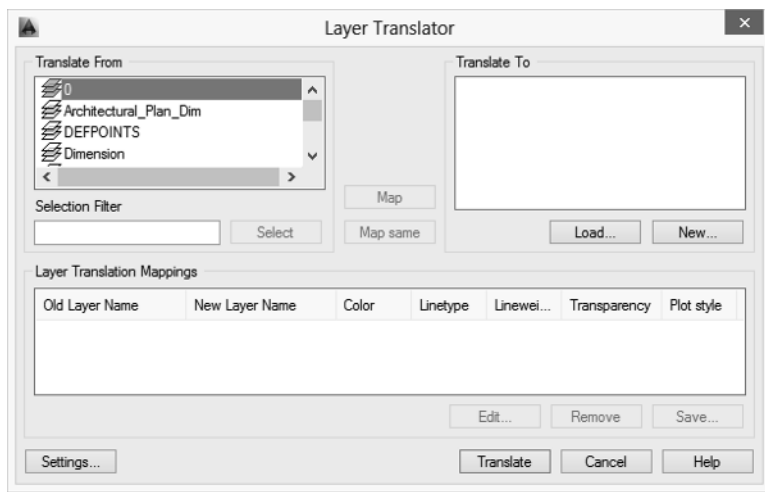
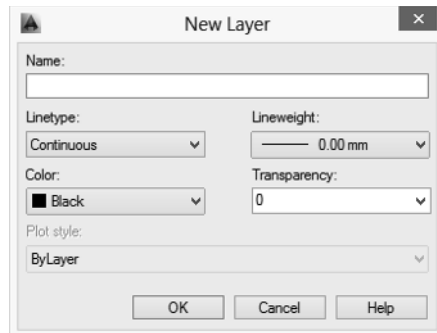


FIGURE 1.16
Defining a new layer to use as part of the layer translation map



5. In the New Layer dialog box, enter a name for the layer and define the new layer's properties. Click OK.

The new layer is added to the Translate To list, but it is not actually created in the drawing until it is used as part of the layer translation mapping and the Translate button is clicked.

6. Click Map Same, adjacent to the Translate To area. This creates layer mappings for the layers that are named the same between the Translate From and Translate To areas.
7. In the Translate From area, select the layers that you want to map to a layer listed in the Translate To area. Press and hold Ctrl to select more than one layer in the Translate From area.

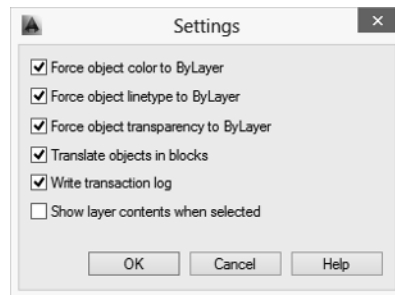
TIP If the layer list in the Translate From area contains a large number of layers, you can right-click the list and click Purge Layers to remove from the list the layers that are not being used in the current drawing. You can also enter a wildcard search in the Selection Filter text box to help you select layers listed in the Translate From area. For example, you can enter ***DIM*** to find all the layers that have a name containing the characters DIM some place in their names.

8. In the Translate To area, select the layer that the layers you selected in the Translate From list should be mapped to. Click Map.

One or more layer mappings are added to the Layer Translation Mapping list, and the layers selected from the Translate From list are removed from the list since they cannot be mapped to multiple layers.

9. Click Settings. The Settings dialog box (see Figure 1.17) is displayed.

FIGURE 1.17
Changing the settings to use when translating layers



10. In the Settings dialog box, enable the options you want to use when translating the mapped layers. Click OK.
11. In the Layer Translator dialog box, click Translate.
12. In the Layer Translator - Changes Not Saved message box, click Translate And Save Mapping Information or Translate Only.
 - ◆ If you clicked Translate And Save Mapping Information, the Save Layer Mappings dialog box is displayed. Browse to a location and enter a name for the file. If needed, choose Standards (*.dws) or Drawing (*.dwg) from the Files Of Type drop-down list, and then click Save. After the translation mappings are saved, the layers are updated in the current drawing according to the layer mappings you created.
 - ◆ If you clicked Translate Only, the layers are updated in the current drawing based on the layer mappings you created and then the layer translation mappings are discarded.

