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CAD
MANAGEMENT

CHAPTER OBJECTIVES

After completing this chapter you should:

1. know the importance of establishing and enforcing CAD standards for a project or company;
2. understand what a drawing standards (.DWS) file is and how to create one;
3. be able to use the *Check Standards* dialog box to verify that the current drawing is using the correct layers, linetypes, dimension styles and text styles that meet the project or company standards;
4. be able to use the *Batch Standards Checker* to validate multiple drawings for compliance of the established standards;
5. be able to use the *Layer Translator* to correct layer properties that do not match the project or company standards; and
6. be aware of the additional tools available to improve the quality of CAD standards for a project or company.

CONCEPTS

Since the first use of Computer Aided Drawing, standards were established based on the standards originated from board drafting. Standards are necessary for everything that is mass-produced, beginning with the initial design and following through the final product. Standards are needed to ensure that quality, safety, and consistency are maintained. Standards can be international (such as International Standards Organization [ISO] standards), national (such as American National Standards Institute [ANSI]), industry-wide (such as American Institute of Architects [AIA] and Construction Standards Institute [CSI]), or company-wide (such as using a company-standard CAD template).

It is crucial to be able to communicate with everyone collaborating on a project. The CAD data is the central source used to collaborate and share information on a project. Therefore, standards such as consistent layer names, colors, text styles, and procedures may be necessary to share information accurately and consistently with all members of a project, including designers, managers, marketing and sales associates, production or construction staff, contractors, vendors, and clients.

It is typically the job of the CAD manager or a senior member of the design group or department to develop and implement CAD standards. These standards ensure that all drawings within a project or company use the same layer names, dimension styles, linetypes, text styles, and so on. Imagine the frustration and inefficiency of working with a set of drawings for one project created by several CAD operators, each drawing having its own set of layer names, dimension styles, linetypes, and text styles! CAD standards provide better coordination among teams, a more efficient design process, and ultimately increased production and profits.

Although CAD standards are increasingly important and are used by many companies to some extent, no tools existed in AutoCAD to help assist in the process of developing and maintaining CAD standards until AutoCAD 2002. Autodesk improved the CAD standards process by introducing real-time notification and control for plug-ins with AutoCAD 2004. You can now develop CAD standards simply by creating layers, dimension styles, linetypes, and text styles, and then saving them to a drawing standards (.DWS) file.

Two CAD standards tools found in AutoCAD simplify the process of maintaining standards using .DWS files. The *Layer Translator* and the *Standards Manager* are used to audit a drawing against the defined standards in a .DWS file for any types of violations. Corrections can then be made automatically or through user input. Additionally, .DWS files can be accessed from DesignCenter to add named objects defined in the file (such as layer names, dimension styles, and text styles) to an existing drawing. AutoCAD also comes with an external utility called the *Batch Standards Checker* to audit CAD standards against a set of drawings in a project.

Establishing Standards

Even though this chapter focuses on using the CAD standards tools that are provided with AutoCAD 2006, a good set of CAD standards should include additional components. Several considerations are listed below that can improve the quality of your drawings and improve collaboration on projects. These items should be standardized for drawings within one project or company.

- drawing naming standards and drawing properties summaries
- format (version) and locations for saving drawings
- drawing units and other drafting settings
- layouts, page setups, and plot styles
- blocks, names, and layers

layer names and properties
 text styles, names, and properties
 table styles
 dimension styles and tolerances, names, and properties
 multiline styles

Creating a Drawing Standards (.DWS) File

Drawing standards (.DWS) files are similar to drawing (.DWG) or drawing template (.DWT) files in that they can store layer, dimension style, linetype, and text style information; however, .DWS files do not necessarily contain drawing geometry. A .DWS file typically contains only the following:

layer names and related properties for each layer (color, linetype, lineweight, and plot style settings)
 dimension styles and all related settings
 loaded linetypes
 text styles and associated fonts and properties

Four methods can be used to create a drawing standards (.DWS) file.

1. Begin a *New* drawing from scratch or using an existing template. Create the desired layers and set all related properties. Create all needed dimension styles and related settings. Create all needed text styles and make all font assignments and related settings. Load any additional linetypes that may be needed (other than those already loaded for the existing layers). Finally, *Save* the drawing and select *AutoCAD Drawing Standards (*.dws)* from the *Files of type* drop-down. This action saves the drawing with the file extension .DWS and contains the layers, dimension styles, linetypes, text styles, and so on that you specified.
2. *Open* an existing drawing (.DWG) or drawing template (.DWT) file that already contains the project or company standard layers, dimension styles, linetypes, and text styles. Then *Save* the drawing and select *AutoCAD Drawing Standards (*.dws)* from the *Files of type* drop-down. This method saves all the layer, dimension style, linetype, and text style information.
3. Use Windows Explorer to make a copy of an existing drawing (.DWG) or drawing template (.DWT) file and rename the file extension to .DWS. This method is identical to opening the file and saving it as an *AutoCAD Drawing Standards (*.dws)* file type.
4. Use the *Save* option from the *Layer Translator* to create a drawing standards (.DWS) file that contains only layer translation mappings for the current drawing. However, with this method, no dimension styles, text styles or drawing geometry information is added to the .DWS file.

The new .DWS file can contain layer, dimension style, linetype, and text style information, depending on the option used to create the .DWS file. Creating a .DWS file using methods 1 through 3 above also saves any drawing geometry in the file, although the drawing geometry is not needed for auditing standards with the tools described in this chapter. Once the .DWS file has been created and set up to include the appropriate named objects, it can then be used by the *Layer Translator*, *Standards Manager*, or *Batch Standards Checker* to compare and/or correct violations against the settings contained in the .DWS file.

If you need to include standards in an existing drawing, it is possible to retroactively import layers, dimension styles, linetypes, and text styles from other existing drawings using DesignCenter.

NOTE: Layer setting such as on/off, freeze/thaw, lock/unlock, and plot/no plot are ignored in the standards file. In addition, any attached *Xrefs* are ignored when standards are being audited.

NOTE: When creating a .DWS file, use the same type of Plot Style Table (color-dependent or named) in the .DWS file that you intend to use in the drawing (.DWG) files that you audit. The *Layer Translator*, *Standards Manager*, or *Batch Standards Checker* cannot correct the plot style property of layers when the Plot Style Table types are different. To remedy this situation retroactively, you could convert a drawing's Plot Style Table type from color-dependent to named or vice versa using the *Convertpstyles* command. If your standards are changing to support named Plot Style Tables, the command *Convertctb* can be helpful in creating a named plot style that is based on an existing color-dependent plot style.

AUDITING DRAWINGS USING A STANDARDS FILE

In a typical office environment, you should perform periodic audits of all the drawings for current projects to ensure that they conform to the project or company standards. To be competitive in the growing global economy, the design process must be much faster paced than in years past. This change has led to a need to interact and collaborate at a more complex level, especially when working on projects with multiple colleagues, vendors, contractors, or subcontractors.

For example, it is likely that you may have to access drawings from a vendor or subcontractor that uses a different set of standards from yours based on the standards of the vendor's primary client. Therefore, you may have to work with drawings that contain different layers, dimension styles, linetypes, or text styles that do not comply with the standards defined for the specific project or by your company.

Using the tools in AutoCAD makes it possible to perform periodic audits that identify these nonstandard named objects and give you the control to ignore them, temporarily change them, or permanently fix them. This process is rather simple since a drawing that has been audited against your standard, then saved, also saves the fixed and ignored status of the violations. Therefore, checking the drawing again provides you the opportunity to automatically audit the same violations using the previous mappings. It also helps to quickly identify any new named objects that might have been created that do not match the standards.

In general, the standards auditing function first compares the name of the drawing with that found in the standards file. If the names match, the properties for each item are compared to the properties of the matching named object saved in the standards file. If the name or the properties don't match, the violations are reported.

Since the standards file is retained as a separate file and only linked to the drawing, any changes that take place in the drawing do not affect the standards file. This distinction also allows you to easily change or update your standards by making changes to the .DWS file, then you can audit and update the project drawings to the latest set of standards.

Standards

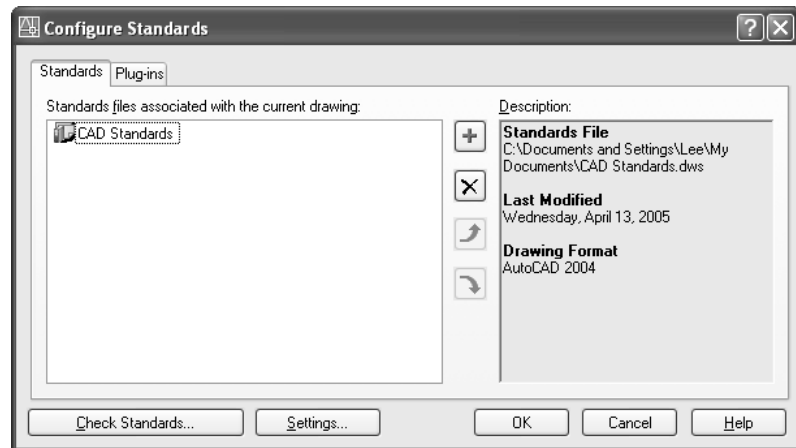


Pull-down Menu	Command (Type)	Alias (Type)	Short-cut	Screen (side) Menu	Tablet Menu
Tools > CAD Standards > Configure	Standards	STA

The *Standards* command, *Configure* button on the CAD Standards toolbar, or any other method given in the command table above produces the *Configure Standards* dialog box (also called the *Standards Manager*) shown in Figure 46-1 on the next page.

The *Configure Standards* dialog box does not do anything specific with the drawing standards (.DWS) file and does not perform any of the auditing of the standards. Its role is to specify which plug-ins are to be used during the auditing of the drawing against the specified standard files and which standard files are to be used to validate that the current drawing follows the established standards. The dialog box does allow for you to start the audit process with the *Check Standards* button located in the lower-left corner. Selecting the *Check Standards* button produces the *Check Standards* dialog box where the named objects between the current drawing and the standards files specified in the *Configure Standards* dialog box are compared (see “*Check Standards*”).

FIGURE 46-1



Standards Tab

Add Standards File



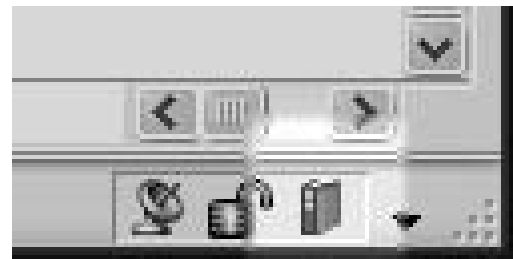
Use the *Add Standards Files* button to locate a drawing standards (.DWS) file and add it to the list titled *Standard files associated with the current drawing*. Selecting this button produces the *Select standards file* dialog box which allows you to select a previously created .DWS file that might be stored locally or on the network. It is possible to associate multiple .DWS files with the current drawing file. Each of the standards files must be added separately, since the *Select standards file* dialog box does not allow multiple .DWS files to be selected at one time.

If more than one standards file is listed, the highlighted file's description is displayed on the right under *Description*. The order in which standards files are listed is important. The order of the standards files in the list determines the priority used when violations occur in the current drawing (when *Check Standards* is used). For example, if a layer's properties (in the current drawing) do not match the standards files, the properties would be changed to match those of the standards file highest in the list. To change the order in which standard files are used during auditing, use the *Move Up* and *Move Down* buttons located in the middle of the *Configure Standards* dialog box.

Once the *OK* button is selected, the standards files become associated with the current drawing. Therefore, when the *Check Standards* dialog box is later used to audit the drawing, you are notified of any potential violations of the drawing against the associated standards files. When a .DWS file is associated with a drawing an icon appears in the AutoCAD Status bar (Fig. 46-2).

NOTE: When a .DWS file is associated to a drawing file, the path to the .DWS is stored with the drawing. Therefore, if the .DWS file is moved to another folder, it may not be found next time you attempt to perform a standards audit for the drawing. AutoCAD searches the stored path, the *Support File Search Path* and *Working Support File Search Path* (set in the *Options* dialog box, *Files* tab), and the current directory. It is suggested to keep associated .DWS files in one location (preferably on a network) or, when exchanging files with your subcontractors, move or copy the .DWG and associated .DWS files together (in the same folder).

FIGURE 46-2



Remove Standards File



Use the *Remove Standards Files* button to remove one of the previously associated drawing standards files. Select the drawing standards file from the list then use this button. If all standards files are removed, the icon that appears in the status bar area is also removed.

Move Up



Use the *Move Up* button to change the priority order that the standards files are searched. The standards file at the top has the highest priority and the one at the bottom has the lowest priority. Select the desired drawing standards file in the list and use this option to move the file up in the priority order.

Move Down



The *Move Down* button changes selected standards files to a lower priority search order.

Check Standards

Selecting *Check Standards* produces the *Check Standards* dialog box. This dialog box is where the current drawing is compared against the associated standards files for any violations. Selecting this button is the same as using the *Checkstandards* command (see “*Checkstandards*”). Selecting the button also saves any changes that are made under the *Standards* or *Plug-ins* tabs of the *Configure Standards* dialog box.

Settings

Selecting *Settings* produces the *CAD Standards Settings* dialog box. This dialog box allows you to control the notification balloon that is displayed near the Status bar when a violation occurs during the drawing process. The balloon appears when a layer, dimension style, linetype or text style is changed and it no longer matches the standards file. The dialog box also contains some options that determine how the audit process is performed. See “*CAD Standards Settings*” for more information.

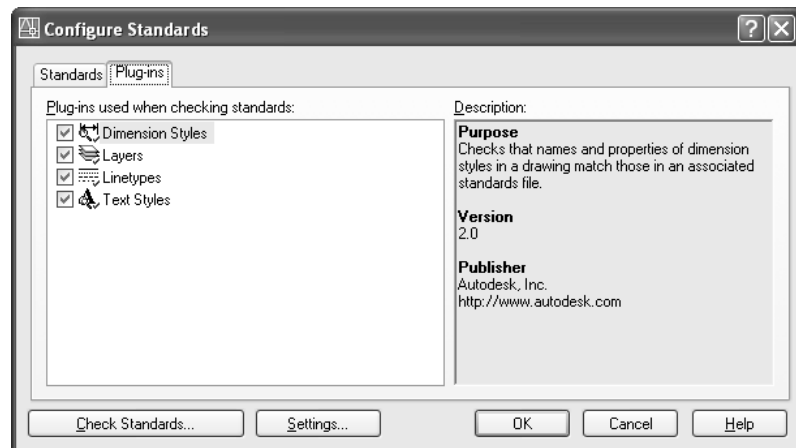
OK

Selecting *OK* associates the standards files that are in the list with the current drawing file. Also, any changes under the *Plug-ins* tab are saved with the current profile to the Windows registry. No standards auditing is performed at this time but can be accomplished later using the *Checkstandards* command.

Plug-ins Tab

The *Plug-ins* tab (Fig. 46-3) is used to evaluate the loaded plug-ins that are used during the standards auditing process. From this tab you can see which plug-ins have been installed on the computer. You can choose from four default plug-ins: *Dimension Styles*, *Layers*, *Linetypes*, and *Text Styles*. It is possible to obtain other plug-ins from third party developers that might be used to verify geometry or other named objects. It is possible to control which of the plug-ins are used during the standards auditing process. Uncheck the plug-ins from the *Plug-ins used when checking standards* list that you do not want to use. If a plug-in is highlighted in the list, a description of the plug-in is displayed under *Description* on the right side of the dialog box.

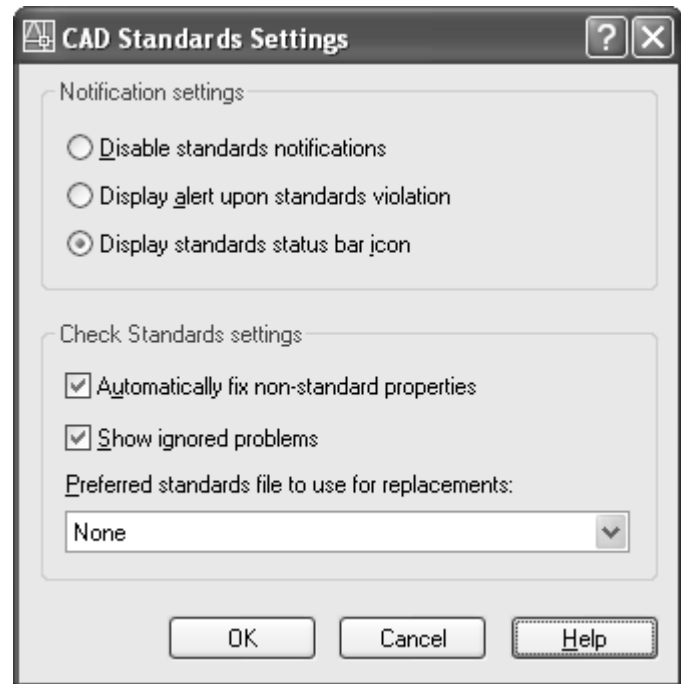
FIGURE 46-3



CAD STANDARDS SETTINGS

The *CAD Standards Settings* dialog box (Fig. 46-4) is used to control the behavior of the real-time notification system for CAD standards. The CAD standards notifications can be customized under the *Notification settings* section, giving you some control over how the notification system works. The notification settings can also be controlled with the *STANDARD-SVIOLATION* system variable. See AutoCAD 2006 Instructor Appendix A, for the valid values you can use. The *Check Standards settings* options are used to control the behavior of how violations are handled by the *Check Standards* dialog box.

FIGURE 46-4



Notification settings

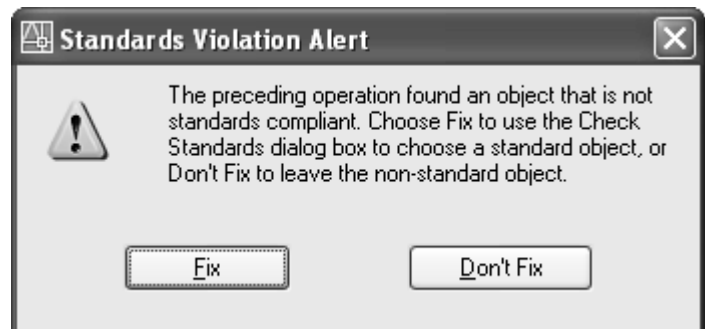
Disable standards notifications

Choosing this option disables the notification balloon in the Status bar. In this case the balloon is not displayed when a violation to the standards has been identified and for missing standards files. This option also removes the icon from the Status bar.

Display alert upon standards violation

This option forces an alert box to appear (instead of the notification balloon) in the status bar area when a standards violation occurs (Fig. 46-5). You have the option to immediately fix or not fix the standards violation. If this setting is active and an AutoLISP or script routine is running in the drawing, the notification is not displayed until after the routine has completed. If you are using AutoLISP or scripts to automate parts of the drawing process, you might want to disable the use of notifications using the *STANDARD-SVIOLATION* system variable.

FIGURE 46-5



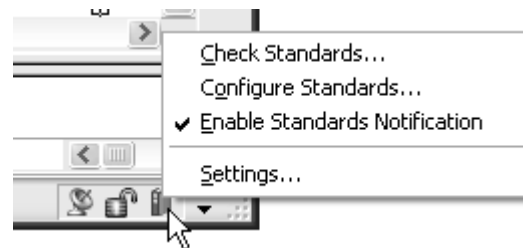
Display standards status bar icon

An icon is displayed in the status bar when a drawing with associated standards files is opened. This icon serves three purposes. First, it indicates that standards files are associated with the drawing. Second, when a standards violation occurs a notification balloon is displayed instead of an alert dialog box (Fig. 46-6). Third, right-clicking on the icon produces a shortcut menu of the commands related to CAD standards (Fig. 46-7). The shortcut menu allows you to display the *Check Standards*, *Configure Standards*, and *CAD Standards Setting* dialog boxes. It also allows you to disable the icon in the status bar.

FIGURE 46-6



FIGURE 46-7

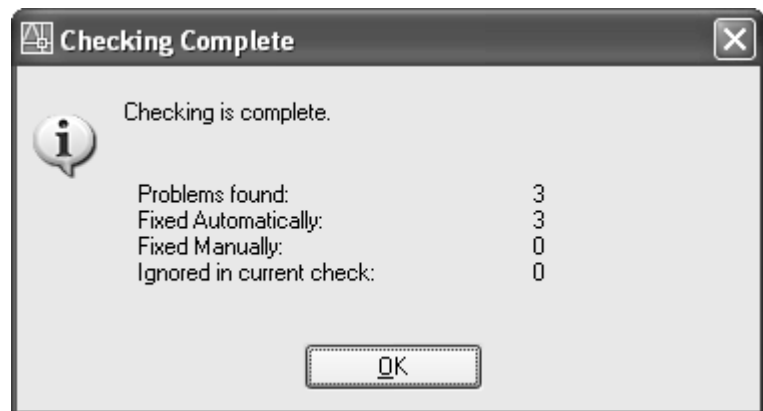


Check Standards Settings

Automatically fix non-standard properties

When you check for standards violations (using *Check Standards*), each named object in the current drawing is examined to ensure it has the same name and properties as in the standards file. When this option is checked, any of the standards violations that are found are automatically fixed without being prompted when a recommended fix is available. Recommended fixes are available only when a nonstandard object has a name that matches a standard object but has different properties. In this case, the properties of the standard object are applied to the nonstandard object. The *Checking Complete* alert box then displays the number of items that were automatically fixed (Fig. 46-8). If the option is not checked, the user is prompted to fix the violations.

FIGURE 46-8



NOTE: In an automatic fix, you must accept the AutoCAD-recommended fix. In such a case, you are not able to review the *Preview of changes* in the *Check Standards* dialog box to make sure the change occurring to the object's properties is appropriate. This option should be used with caution since manually undoing the results of a fix can be a difficult process.

Show ignored problems

When this option is selected, any previously ignored violations appear again in the *Check Standards* dialog box during future checks. If the option is not selected, ignored violations are not displayed again in the *Check Standards* dialog box during future checks. See "Mark this problem as ignored" in the *Check Standards* dialog box.

Preferred standards file to use for replacements

Use this drop-down list to specify the default standards file to use during audits. This list is composed of .DWS files associated with the drawing (those previously specified in the *Configure Standards* dialog box). If no recommended replacement is found when a standards file is selected in this list, the default selection in the *Replace With* list is the first replacement item from the selected standards file. If this setting is set to *None* and no recommended replacement is found, no items will be selected in the *Replace With* list.

CHECK-STANDARDS



Pull-down Menu	Command (Type)	Alias (Type)	Short-cut	Screen (side) Menu	Tablet Menu
Tools > CAD Standards > Check	Checkstandards	CHK

The *Checkstandards* command, *Check* button on the CAD Standards toolbar, or any other method given in the command table above produces the *Check Standards* dialog box (Fig. 46-9). The *Check Standards* dialog box uses the standards files and plug-ins specified in the *Configure Standards* dialog box along with the options specified in the *Check Standards settings* section of the *CAD Standards Settings* dialog box to carry out the standards auditing process. This tool reads the named objects and their properties from the current drawing and compares them to the associated standards files. Based on the settings of the *CAD Standards Setting* dialog box, the user is either prompted to specify a fix for the violation or the violation is automatically fixed.

FIGURE 46-9

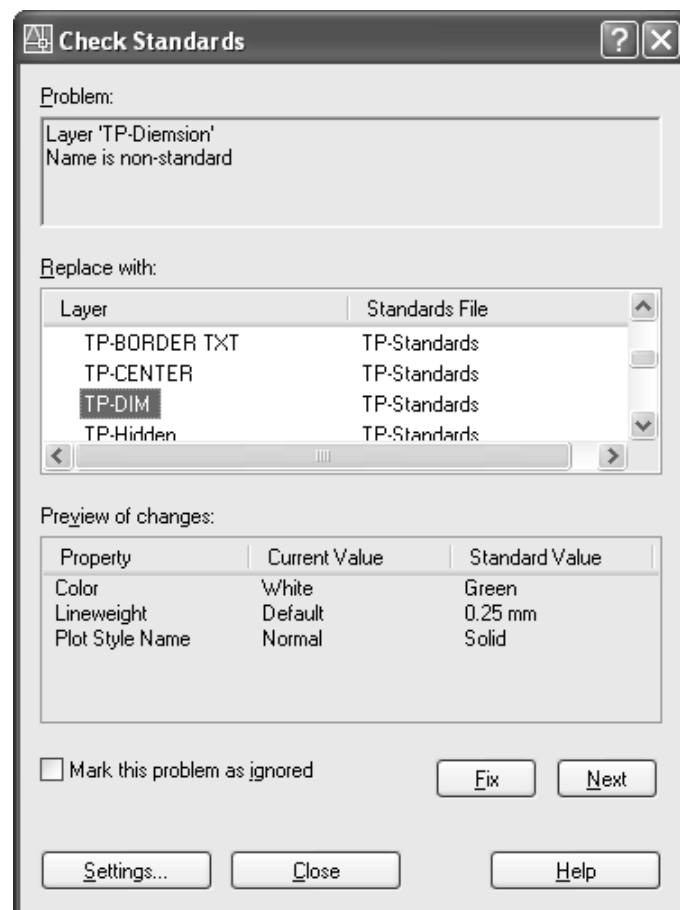
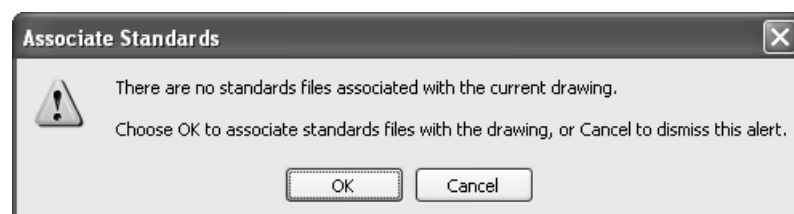


FIGURE 46-10

If no standards file has yet been associated, the *Associate Standards* alert message is displayed (Fig. 46-10). If *OK* is selected here, the *Configure Standards* dialog box is displayed. Selecting *Cancel* results in the command *Checkstandards* to end.



The *Check Standards* dialog box allows you to view the standards violations for the current drawing. The dialog box reports each non-standard layer, dimension style, linetype, and text style, along with any suggested fixes based on the specified plug-ins and designated *Check Standards* settings. As you proceed through the named objects, each non-standard object is listed under the *Problem:* area near the top of the dialog box. With each standards violation that is reported, you have the option to fix the problem (press *Fix*) or ignore it (press *Next*).

Fix

If you want to fix the standards violation that is shown in the *Problem:* area, first select the desired replacement in the *Replace with:* list, then select the *Fix* button. Changes to the properties of the problem item occur according to the *Preview of changes:* list. In other words, if a layer name appears in the *Problem:* list and you choose to fix it, the layer name is changed to the selected name in the *Replace with:* list and the properties of that layer change to the *Standard Value* in the *Preview of changes:* list.

If a small checkmark appears to the left of the named object in the *Replace with:* list, it indicates an AutoCAD-suggested mapping. This occurs when the name of a named object in the current drawing (listed in the *Problem:* area) matches a name from the .DWS file (listed in the *Replace with:* list), yet only the properties listed in *Preview of changes:* are different. For example, you may have a layer named WALLS in the current drawing that matches a layer by the same name in the standards file, but the color of the layer in the drawing (red, for example) is not the same as the standard (green, for example). Accepting this suggestion by selecting the *Fix* button results in changing only the properties of the named object to the standard properties.

NOTE: If *Automatically fix non-standard properties* is checked in the *CAD Standards Settings* dialog box these problems (as described in the previous paragraph) are automatically fixed, that is, when the name of a named object in the current drawing matches a name from the .DWS file, but the properties are different.

Next

Use the *Next* button to skip the current standards violation reported in the *Problem:* area. If the option *Mark this problem as ignored* is checked before the *Next* button is clicked, the standards violation will be automatically ignored again in future audits. When a standards violation is first specified as ignored, AutoCAD records the current CAD operator's login name. The login name is based on the *LOGIN-NAME* system variable, which is normally the Windows login name. For automatic ignoring to occur during the next audit, the *Show ignored problems* checkbox must also be cleared (not checked) in the *CAD Standards Setting* dialog box (see "*Check Standards Settings*").

Basic Steps for Checking and Correcting Standards

Below is an overview of the basic steps for using the *Configure Standards* and *Check Standards* dialog boxes.

1. Use the *Standards* command to display the *Configure Standards* dialog box.
2. Specify the desired standards file(s) to use for audits by pressing the *Add Standards Files* button. Change the priority order of the standards files with the *Move Up* and *Move Down* buttons as necessary.
3. Select the *Plug-ins* tab and select the plug-ins that you wish to use for checking the standards of the current drawing.
4. Select the *Settings* button and make any desired setting changes in the *CAD Standards Settings* dialog box, such as *Notification settings* or *Check Standards settings*, then select *OK*.
5. Select *OK* in the *Configure Standards* dialog box. This action associates the standards file(s) with the current drawing and exits the *Configure Standards* dialog box.
6. Use the *Checkstandards* command to display the *Check Standards* dialog box.

7. For each standards violation that appears in the *Problem:* area, select the desired named object from the *Replace with:* area and click *Fix*, or to skip the violation, *Check Mark this problem as ignored* and select *Next*.
8. Select OK in the *Checking Complete* dialog box.

AUDITING MULTIPLE DRAWINGS FOR STANDARDS VIOLATIONS

AutoCAD 2006 provides the *Batch Standards Checker* that runs externally from AutoCAD and allows you to audit standards for more than one drawing at a time. The batch checker automatically audits a list of drawings without your having to “manually” open and close each drawing individually. Using this utility, each drawing to be checked can be audited using the specific standards file(s) associated with that drawing or all the drawings can be audited using the standards files assigned under the *Standards* tab of the *Batch Standards Checker*.

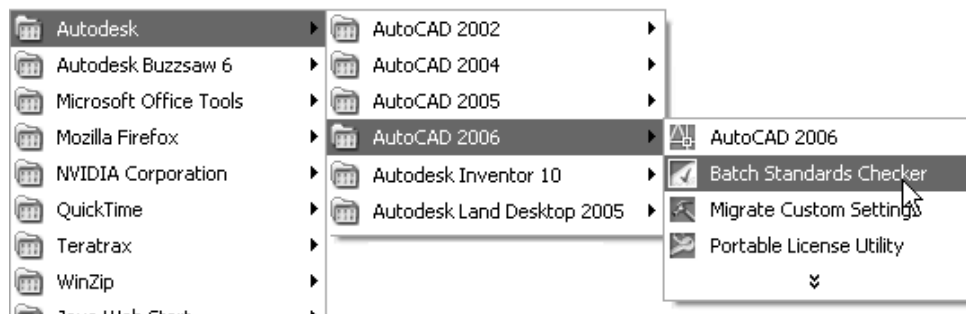
The *Batch Standards Checker* does not provide automatic correcting for standards violations, but it does generate a report of the violations. The report gives a summary of the violations for the listed drawing files including the named objects (layers, dimension styles, linetypes, and text styles) and the individual properties of the objects (such as color of a layer and variable settings of a dimension style) that do not match the specified standard. Once a set of drawings and related standards files for auditing have been specified, the configuration can be saved to a Standard Check (.CHX) file for future audits.

Since the *Batch Standards Checker* does not provide automatic correcting for standards violations, you should examine the report, then use the *Check Standards* dialog box to correct any drawings that have reported standards violations. Therefore, the *Batch Standards Checker* is most useful for performing periodic audits of an entire set of project drawings during the design process, specifically to identify violations that may have occurred during design updates.

Starting the *Batch Standards Checker*

The *Batch Standards Checker*, similar to the *Reference Manager*, runs externally from the current AutoCAD session. More specifically, the *Batch Standards Checker* runs as a separate application and does not require AutoCAD to be running. You can start the *Batch Standards Checker* two ways.

FIGURE 46-11



1. Using the *Start* menu in Windows, select *(All) Programs*, then *Autodesk*, then *AutoCAD 2006*, then *Batch Standards Checker* (Fig. 46-11).
2. If AutoCAD is running, you can type *Shell* at the Command line. At the “OS Command:” prompt that appears, type *Dwgcheckstandards*, as shown below.

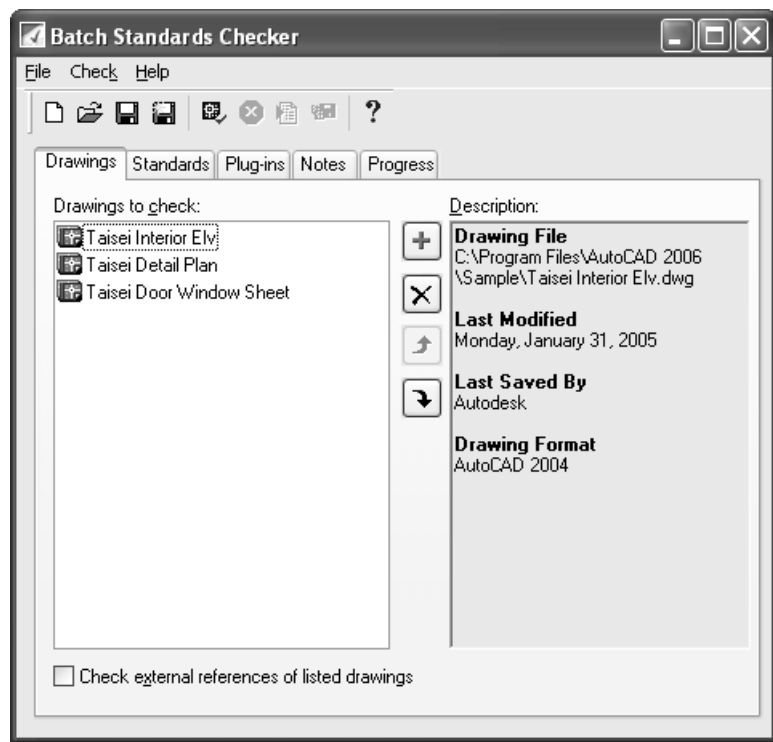
Command: **shell**

OS Command: **dwgcheckstandards**

Auditing a Set of Drawings Using the *Batch Standards Checker*

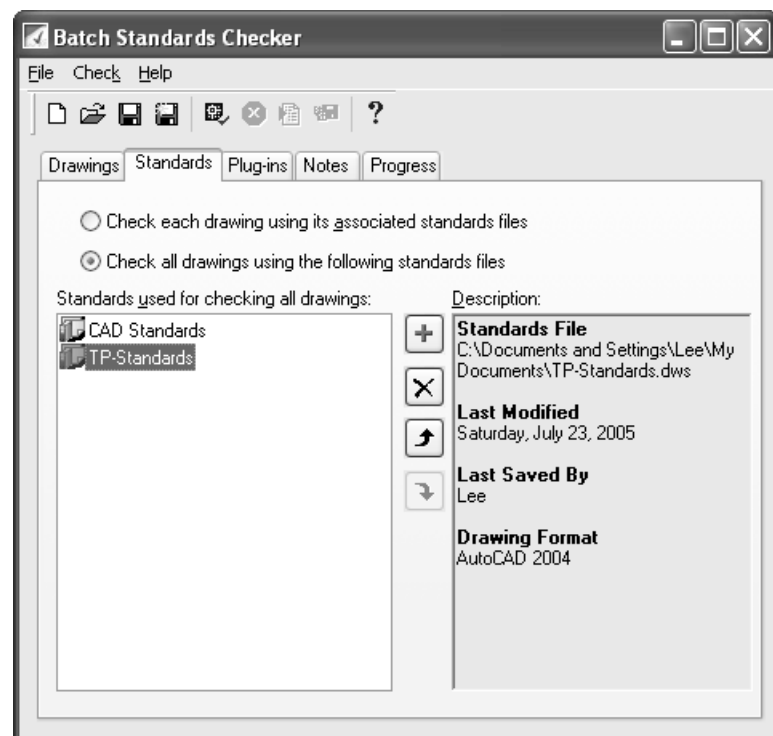
1. Start the *Batch Standards Checker* by either method described in “Starting the *Batch Standards Checker*.” The *Batch Standards Checker* (Fig. 46-12) appears.
2. The *Drawings* tab allows for adding drawings to the batch process. To add drawings, click the *Add* button to display the *Batch Standards Checker – File Open Dialog* box. Locate the drawing (.DWG) files or drawing template (.DWT) files that you want to audit. More than one drawing can be selected at a time using standard Windows selection methods (holding the Ctrl or Shift key to select multiple files or a range of files, respectively). After the drawing and/or drawing template files have been added, change the order of the files in the list or remove files from the list as needed using the *Move Up*, *Move Down*, or *Remove Drawing* buttons (see “Standards” for a description of these functions). The *Check external references of listed drawings* checkbox forces all attached Xrefs to the *Drawings to check:* list to be included in the audit.

FIGURE 46-12



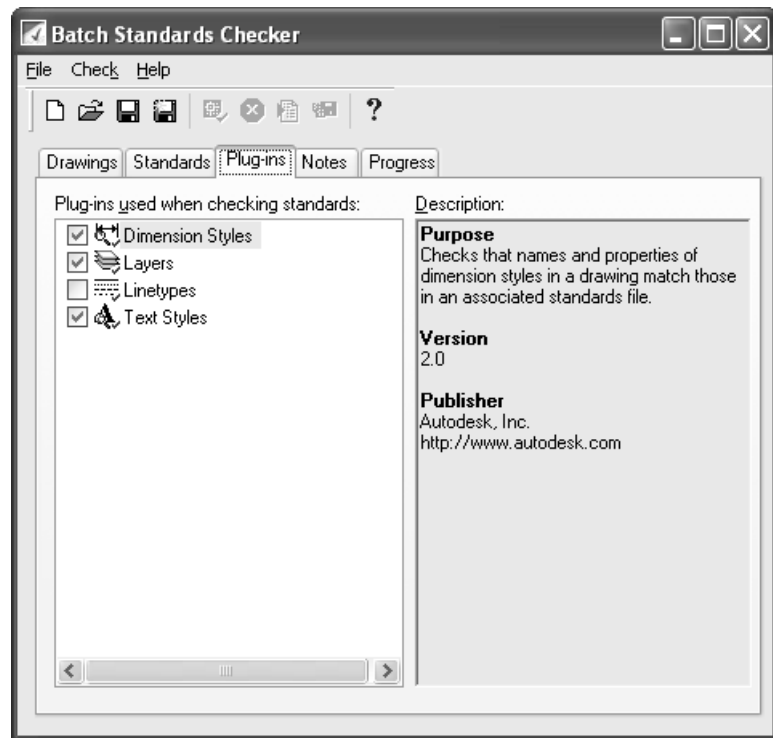
3. Use the *Standards* tab (Fig. 46-13) to specify the drawing standards (.DWS) files you want to use during the audit. Select from either *Check each drawing using its associated standards file* or *Check all drawings using the following standards files*. Use the first option if you want to audit each drawing in the list against its own .DWS files and you are sure that each drawing has at least one .DWS file associated with it (previously specified using the *Configure Standards* dialog box). In this case, selection of the associated .DWS files is automatic, so the list and buttons are disabled. If you use the second option, use the *Add Standards File* button to select the .DWS file(s) to use for globally auditing all the listed drawings. Use the *Move Up* and *Move Down* buttons to prioritize the .DWS files if needed.

FIGURE 46-13



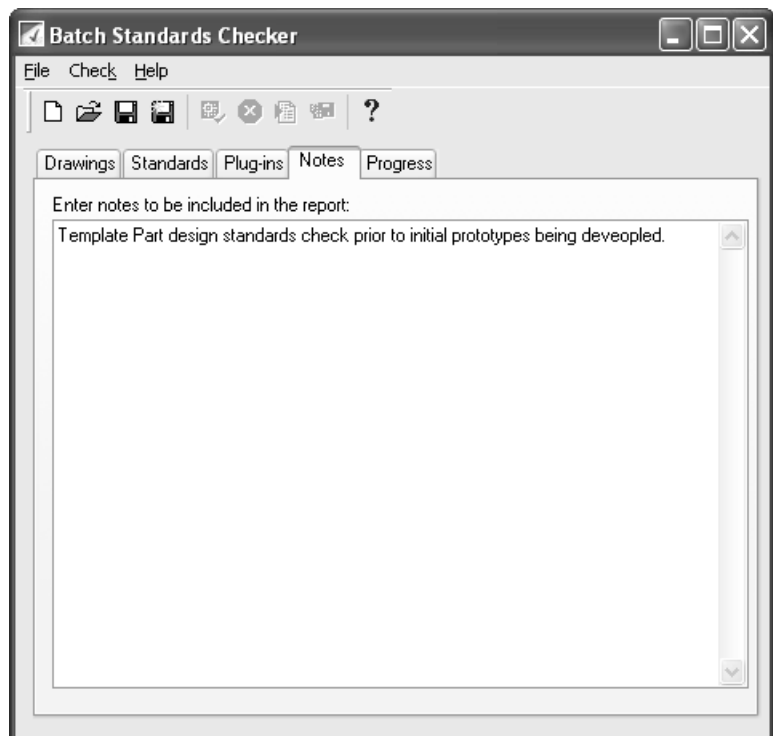
4. Use the *Plug-ins* tab (Fig. 46-14) to specify which types of named objects you want to audit when the batch process is run. These are the same plug-ins that are displayed in the *Configure Standards* dialog box. By default, the four available plug-ins are *Layers*, *Dimension Styles*, *Linetypes*, and *Text Styles*.

FIGURE 46-14



5. Use the *Notes* tab (Fig. 46-15) to add any notes that you want to appear at the top of the report once all the drawings have been audited against the standards files.

FIGURE 46-15



6. Use the *Save* button to save the current settings in the *Batch Standards Checker* (such as the list of drawing files to audit, list of standards files to use, plug-ins that should be used for auditing, and any notes that have been specified). This information is saved to a Standards Check (.CHX) file. The settings can be recalled any time in the future if you want to run an audit of the same set of drawings again. By saving the settings, you are able to efficiently audit the drawings periodically when design updates or changes are made.

7. Use the *Start Check* button to begin auditing the specified drawings. The *Batch Standards Checker* sequences through the list of drawings by opening each drawing in the background and comparing it to the specified standards file(s).
8. The *Progress* tab automatically opens when the auditing phase begins. During the auditing process, this tab lists the name of the drawing in the current audit as well as a list of completed audits. You can stop the audit at any time using the *Stop Check* button.
9. When the audit process is completed, an .XML-based report is generated. The report is automatically displayed in the default web browser for you to examine (see “Standards Audit Report”). The report is saved as part of the .CHX file, but can also be exported to a .HTM file using the *Export Report* button.

Additional Features of the *Batch Standards Checker*

In addition to the functions described in the list above, the following options are available in the *Batch Standards Checker*.

New

Use the *New* button to reset the values under the *Drawings*, *Standards*, and *Notes* tabs of the *Batch Standards Checker*. For example, if you just ran an audit but want to process a new set of drawings, use *New*.

Open

Use the *Open* button to load a previously created Standards Check (.CHX) file. Opening the file restores the settings that were previously saved (list of drawings, list of standards files, plug-ins, and the report).

View Report

After an audit has been performed, the report can be opened from the *Batch Standards Checker* (if not currently open). *Open* the .CHX file that contains the configuration information and report, then select the *View Report* button.

Viewing the Standards Audit Report

Remember that the *Batch Standards Checker* does not actually fix the standards violations in the audited drawings, it only reports them. If any problems have been identified, you should open the drawing(s) and use the *Check Standards* dialog box to correct the standards violations (see “Basic Steps for Checking and Correcting Standards”). Therefore, the report generated by the *Batch Standards Checker* is critical for identifying which drawings need to be fixed. This report contains several pages of information such as an *Overview*, the *Standards* (files) used, *Problems* that were found during the audit, and the *Ignored Problems*. The report is viewable using the *View Report* button.

The *Overview* page (Fig. 46-16, on the next page) displays general information about the audit such as the time, date, notes you added to the report, and a summary of drawings that were audited including problems found and ignored. Note that on each page you can select a radio button to display the following pages.

<i>Overview</i>	This page gives general information and a summary of problems found and problems ignored.
<i>Plug-ins</i>	This page lists the standard Autodesk plug-ins (<i>Layers</i> , <i>Dimension Styles</i> , <i>Linetypes</i> , and <i>Text Styles</i>) and any other plug-ins that might be installed that were used in the audit.

- Standards* Use this button to display all .DWS files used during the audit including all properties of all the layers, dimension styles, linetypes, and text styles.
- Problems* The *Problems* page lists the specific standard violations found during the audit of each drawing including all non-standard named objects and specific properties that did not comply for correctly named objects (see Fig. 46-17).
- Ignored Problems* This list reports only problems that were marked as ignored in previous audits using the *Check Standards* dialog box (see “*Checkstandards*”).

FIGURE 46-16

STANDARDS AUDIT REPORT

C:\Documents and Settings\Lee\My Documents\Check1.chx

Show:

Overview

Plug-ins

Standards

Problems

Ignored Problems

All

For:

All Drawings

Taisei Detail Plan.dwg

Taisei Door Window Sheet.dwg

Taisei Interior Elv.dwg

Overview

Created by:
Lee

Created on:
Saturday, July 23, 2005

Notes:
Template Part design standards check prior to initial prototypes being deveopled.

Summary:

Drawing	Problems	Ignored problems
Taisei Interior Elv.dwg	26	0
Taisei Detail Plan.dwg	43	0
Taisei Door Window Sheet.dwg	14	0

FIGURE 46-17

STANDARDS AUDIT REPORT

C:\Documents and Settings\Lee\My Documents\Check1.chx

Show:

Overview

Plug-ins

Standards

Problems

Ignored Problems

All

For:

All Drawings

Taisei Detail Plan.dwg

Taisei Door Window Sheet.dwg

Taisei Interior Elv.dwg

Problems

C:\Program Files\AutoCAD 2006\Sample\Taisei Detail Plan.dwg

The following problems were encountered in this drawing:

Name	Description
Dimension Styles	
B10	Name is non-standard
A10	Name is non-standard
A30	Name is non-standard
B30	Name is non-standard
A100	Name is non-standard
Layers	
A01	Name is non-standard

The *Problems* page (Fig. 46-17) is obviously the most important page of the report since it lists each drawing that contains standards violations as compared to the standards file. For each of these drawings, each named object (layer, dimension style, linetype, and text style) that violates the standards is listed. In cases where the named objects matched those in the standards but the properties were different, the standard value and the nonstandard value are listed.

USING THE LAYER TRANSLATOR

The *Layer Translator* is another tool included in AutoCAD 2006 to help manage CAD standards. The *Layer Translator* is used to convert specifically the layers (including names, colors, linetypes, and plot styles) in a drawing to a standard set of layers you specify. For example, if you open an architectural drawing from a client in which the layers containing the walls, fixtures, lighting, structural elements, etc. have different names and other properties from those used in your company, you can use the *Layer Translator* to “translate” the drawing’s layer names and properties from the current set of standards to one that matches your office without affecting the drawing geometry. The mapping process includes translating, or converting, layer names as well as changing properties of the layers such as color, linetype, lineweight, and plot style. You can change all the names of layers to other names that match a standard. Or, assuming that the layer names already matched the standard names, you could convert only the properties (color, linetype, lineweight, and plot style) to the standard. You can use a drawing (.DWG), drawing standards (.DWS), or drawing template (.DWT) file as the source for the layer properties you want to translate to. There are several added benefits to using the *Layer Translator* such as isolating and viewing the geometry on each layer, converting color and linetype layer properties to the *ByLayer* setting, and saving the layer mappings for future translations to a drawing standards (.DWS) or drawing (.DWG) file.

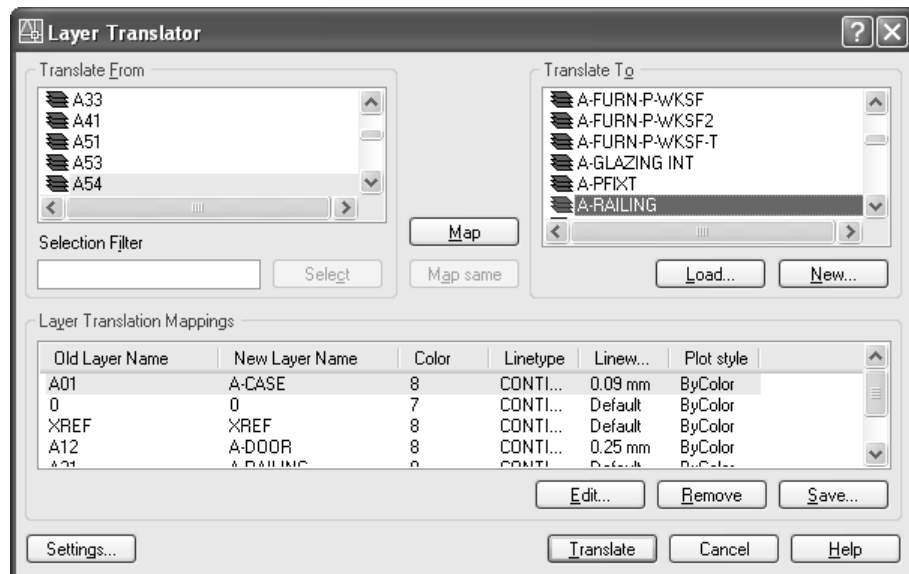
LAYTRANS



Pull-down Menu	Command (Type)	Alias (Type)	Short-cut	Screen (side) Menu	Tablet Menu
Tools > CAD Standards > Layer Translator	Laytrans

The *Laytrans* command, *Layer Translator* button on the CAD Standards toolbar, or any other method given in the command table above produces the *Layer Translator* dialog box shown in Figure 46-18. The *Layer Translator* dialog box is divided into three main sections: *Translate From*, *Translate To*, and *Layer Translation Mapping*. The basic process is to select a layer (in the current drawing) from the *Translate From* list, then select a layer from the *Translate To* list to convert it to, then press *Map*. The layer names and properties that will be converted, or translated, then appear in the *Layer Translation Mapping* area. Pressing *Translate* completes the process. For more details in this process, see “Basic Steps for Using the Layer Translator.”

FIGURE 46-18



Translate From

The *Translate From* area specifies the existing layers in the current drawing that are not yet “mapped.” Layers can be specified by selecting them individually or using a selection filter. The list includes both layers that contain geometry (shown with green icons) and layers that do not (shown with white icons). Layers without geometry can be purged from both the list and current drawing (see “*Purge layers*”).

Selection Filter

The *Selection Filter* textbox allows you to specify a naming pattern that can include wildcards to help select multiple layers at one time. For example, this function can be useful if a drawing has several layers with “text” in the name and you want to translate them all to the same layer. Below is a list of the valid wildcard characters that can be used in the selection filter value.

<u>Character</u>	<u>Description</u>
# (Pound)	Matches any numeric digit
@ (At)	Matches any alphanumeric character
. (Period)	Matches any non-alphanumeric character
* (Asterisk)	Matches any number of characters and can be used anywhere in the search string
? (Question mark)	Matches any single character; for example, ?23 matches 123, A23, and so on
~ (Tilde)	Matches anything but the pattern; for example ~*12* matches all strings that don't contain 12
[]	Matches any one character enclosed; for example, A[12]3 matches A13 and A23
[~]	Matches any character not enclosed; for example, A[~12]3 matches A33 but not A13
[-]	Specifies a range for a single characters; for example, A[1-9]3 matches A13, A23, and so on to A93, but not A03
` (Reverse quote)	Reads the next character literally; for example, `~12 matches ~12

Select

Use the *Select* button to select the layers in the *Translate From* area that match the filter specified in the *Selection Filter* textbox.

Purge layers

You can purge all layers that do not contain geometry from the current drawing using *Purge Layers*. This option is available only from the shortcut menu. To access this menu, place the pointer over the *Translate From* list and right-click.

Map

To “map” means to assign a nonstandard layer to a specific standard layer. Use the *Map* button to manually map the selected layer(s) in the *Translate From* area to the selected layer in the *Translate To* area. Note that multiple layers or a range of layers can be selected in the *Translate From* area by holding down the Ctrl or Shift keys, respectively, while selecting. When a mapping is specified, the layer names and properties to be changed are listed in the *Layer Translation Mappings* list below.

Map same

Use the *Map same* button to automatically map all layers that have the same name in the *Translate From* and *Translate To* areas. For example, layer “0” would always be automatically mapped since it exists in all drawings. This option is valuable to update only the layer properties (*Color*, *Linetype*, *Lineweight*, and *Plot style*) for layer names that already match the standard.

Translate To

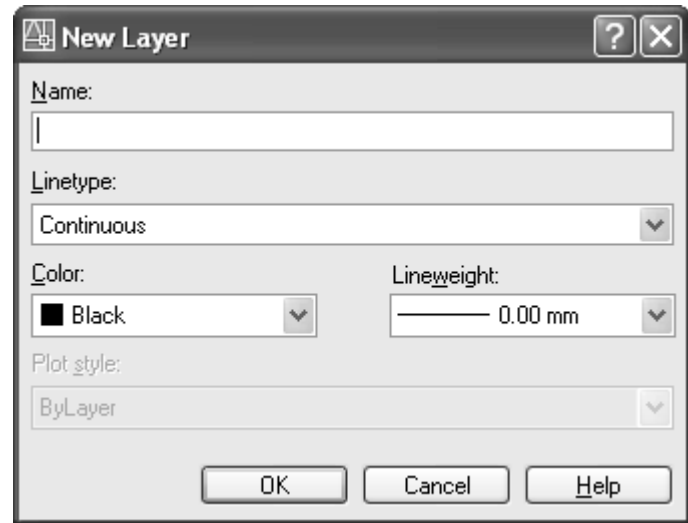
The *Translate To* area specifies the standard layers that have been loaded from either a drawing (.DWG), drawing standards (.DWS), or drawing template (.DWT) file. Different from the *Translate From* area, this list does not distinguish (by the icon appearance) which layers contain geometry and which layers do not.

Load

Use the *Load* button to locate a drawing (.DWG), drawing standards (.DWS), or drawing template (.DWT) file to load the layers that you want to use as the standard in the *Translate To* area. If the file contains previously saved layer maps, the layers in the *Translate From* list are automatically mapped and these mappings are added to the *Layer Translation Mappings* area.

New

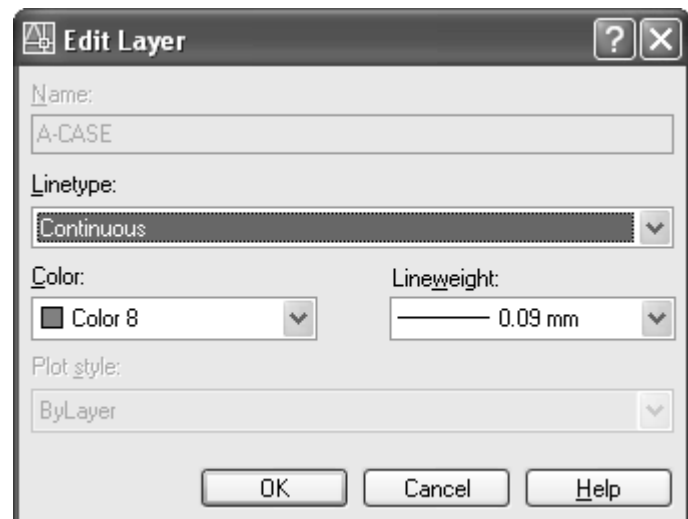
Use the *New* button if you want to create a new standard layer in the *Translate To* list that was not originally contained in the file that was loaded with the *Load* button. The *New Layer* dialog box (Fig. 46-19) appears where you can specify the new layer's name and properties. The new layer is not saved to the loaded .DWG, .DWS, or .DWT file unless *Save* (in the *Layer Translator*) is clicked.

FIGURE 46-19**Layer Translation Mappings**

The *Layer Translation Mappings* area (in the *Layer Translator*) gives the complete list of established layer mappings. The list includes both new mappings that have been created in the current session and any previously saved layer mappings from the loaded .DWG or .DWS file.

Edit

To change an established layer mapping, select it from the *Layer Translation Mappings* list, then select *Edit* to produce the *Edit Layer* dialog box (Fig. 46-20). Similar to the *New Layer* dialog box, this dialog box allows you to edit all of the selected layer's properties except its name. You must use *Save* to save the property changes to the *Layer Translation Mappings* list.

FIGURE 46-20

Remove

Use the *Remove* button to remove a layer mapping from the *Layer Translation Mappings* list by first selecting the desired mapping and then selecting *Remove*. The old layer name then reappears in the *Translate From* list.

Save

Use the *Save* button to save the layer mappings in the *Layer Translation Mappings* area to a file for future use. Layer mappings can be saved to a .DWS or .DWG file format. The *Layer Translator* creates the referenced layers in the .DWS or .DWG file and stores the layer mappings of each layer. All linetypes used by those layers are also copied into the .DWS or .DWG file. Using *Save* does not translate the layers in the current drawing—use *Translate* to do so.

Translate

Select the *Translate* button to convert all *Layer Translation Mappings* to the current drawing and close the *Layer Translator*. If you did not use *Save* before selecting *Translate*, you are presented with the *Layer Translation Alert* dialog box prompting you to save the mapping information before translating. When the translation occurs, any related changes in color, linetype, and linewidth specified in the translation appear immediately in the current drawing.

NOTE: Using the *Layer Translator* to save layer mappings to either file type saves only the layers and related properties but no drawing geometry. Therefore, it is recommended to save layer mappings to the .DWS file type. Saving layer mappings to the .DWG file type can result in a collection of .DWG files on your computer or network, some containing drawing geometry (normal .DWG files) and some containing no geometry (those saved by the *Layer Translator*). Normal .DWG files (containing drawing geometry), however, can be used as a “template” with the *Load* button to provide a list of the needed standard layers. A .DWS file saved by other methods can contain other CAD standards information such as dimension styles and text styles as well as drawing geometry.

Settings

The *Settings* button produces the *Settings* dialog box (Fig. 46-21). Here you can set preferences of how the layer translation is performed.

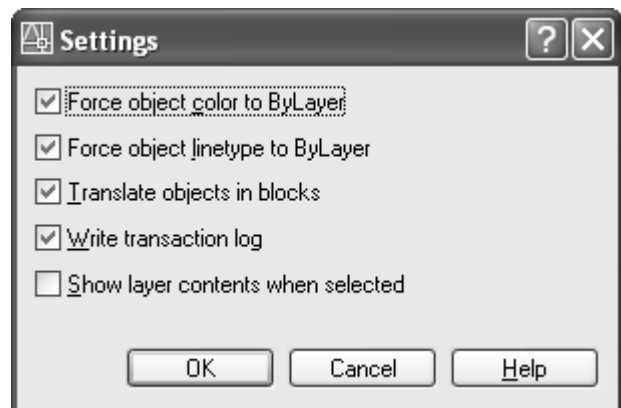
Force object color to ByLayer

When *Force object color to ByLayer* is checked, all layers listed in the *Layer Translation Mappings* area are converted to a *ByLayer* color assignment when the *Translate* button is selected. This option is especially helpful if your standards use colors assigned to layers (*Bylayer* assignment) and the drawings to convert have colors assigned to specific objects. In such a case, this one button would assign the proper color mappings for all layers.

Force object linetype to ByLayer

With this option, all layers in the *Layer Translation Mappings* list are changed to the *ByLayer* linetype assignment when the *Translate* button is selected. Similar to the previous option, this option is especially helpful if your standards also use linetypes assigned to layers (*Bylayer* assignment) and the drawings to convert have linetypes assigned to specific objects. Similarly, use this button to assign the proper linetype mappings for all layers.

FIGURE 46-21



Translate objects in blocks

When *Translate objects in blocks* is checked, all objects contained in the blocks in the drawing are translated and have their color and linetype properties reset based on the mapped properties.

Write transaction log

Use this option to create a log file in the directory of the current drawing that contains information about both the translated layers and those that were not translated. This information is saved to a .LOG file that can be opened with a text editor or word processor. An example of the .LOG file is shown below.

```

=====
Layer Translator Transaction Log
Sunday, July 24, 2005 4:03:01 PM
Source Dwg: c:\program files\autocad 2006\sample\taisei detail plan.dwg
Login Name: Lee
=====

Layers Translated
-----
A01 to A-CASE
A02 to A-COL
A03 to A-COL
A11 to A-STAIR-RAIL
A12 to A-DOOR
A21 to A-DOOR
A23 to A-STAIR-RAIL
A24 to A-STAIR-RAIL
A25 to A-STAIR-RAIL

Layers Not Translated
-----
0
A31
A32
A41

```

Show layer contents when selected

When *Show layer contents when selected* is checked, the selected layers in the *Translate From* area are displayed on screen while all others are turned off temporarily. This feature makes it possible to see which geometry is contained on the selected layer while specifying a layer mapping.

Basic Steps for Using the Layer Translator

Below is an overview of the basic steps for using the *Layer Translator*.

1. *Open* the drawing you want to convert to your layer standards.
2. Invoke the *Layer Translator*. The *Translate From* area in the upper-left corner of the dialog box displays the layers in the current drawing.

3. Select the *Load* button and locate the .DWG, .DWS, or .DWT file you want to use as your layer standards “template.” Select the file and click *Open* in the *Select Drawing File* dialog box. Once the desired file is loaded, the layers in the selected file appear in the *Translate To* area. If you load a .DWG or .DWS file, any previously saved layer mappings in the file are listed in the *Layer Translation Mappings* area. NOTE: It is recommended to load a .DWG or .DWS containing the same Plot Style Table type (color-dependent or named) as the current drawing to ensure the plot style properties for the layers can also be updated.
4. To make the match for conversion of one layer to another, select a layer from the *Translate From* list. If the *Show layer contents when selected* option is checked in the *Settings* dialog box (see “*Settings*”), only the objects on the selected layer appear in the drawing, confirming you have selected the desired layer. Next, select the desired match in the *Translate To* list that you want to convert the layer to.
5. Use the *Map* button to confirm the assignment between the selected layers in the two lists. The layer mappings then appear in the *Layer Translation Mappings* area near the bottom of the dialog box. This list gives the new and old layer names, the color, linetype, lineweight, and plot style of the new layer (layer to convert to).
6. Repeat steps 4 and 5 for each of the layers you want to translate. All mappings you specify should appear in the list near the bottom of the translator.
7. Select the *Translate* button to convert all *Layer Translation Mappings* in the current drawing and close the *Layer Translator*. If the *Layer Translation Alert* dialog box appears, select *OK* to save the mappings to the standards file if desired. When the translation occurs, changes to the layers such as name, color, linetype, and lineweight appear immediately in the drawing.

ADDITIONAL CAD STANDARDS TOOLS

Several additional tools are available to assist in implementing CAD standards (beyond those installed with AutoCAD 2006). These tools are available as free downloads from the Autodesk web site at <http://www.autodesk.com/migrationtools>. The tools range from a batch drawing converter to a utility that allows you to run a script file against a set of drawings. Below is a listing of the tools and a brief description of each.

Autodesk Batch Drawing Converter

This tool allows you to convert a set of drawing files to different AutoCAD file formats, such as AutoCAD Release 14, AutoCAD 2000 or AutoCAD 2004. The AutoCAD 2000 file format is used for AutoCAD 2000, 2000i and 2002. The AutoCAD 2004 file format is used for AutoCAD 2004, 2005, and 2006.

Autodesk Customization Conversion Tools

There are four different utilities in this set of tools that help convert customized menus and files, command aliases, layer states, and running scripts on a set of drawings.

AutoLISP Compatibility Analyzer

The utility is designed to help check the compatibility of the AutoLISP files that you might use to help automate command tasks. This utility can also assist in the migration process when updating to a newer release of AutoCAD.

Command Alias (PGP) Porter

The utility is designed to help migrate any custom command aliases from previous releases to newer releases of AutoCAD.

ScriptPro

ScriptPro can be used to run a script file (.SCR) on a set of drawings. One use of this utility is to use a script to change settings of multiple older drawings to match a current CAD standard.

Layer State Converter

The utility is designed to convert layer states files created with the older *Layer Manager Express Tool* (from AutoCAD 2000i and earlier) to the format of the *Layer* command used in current releases of AutoCAD.