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Managing CAD Standards with AutoCAD®

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GD215-2 This class is for the CAD manager and the CAD operator looking for consistency among drawing files. We'll examine and utilize the AutoCAD CAD Standards tools, creating a CAD Standards file and then using it to check drawings for conformance to standards. We will also use the Layer Translator to further promote consistency and conformance to company standards.

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About the Speaker:

Craig both manages the Autodesk Authorized Training Center and teaches Mechanical Design at Fox Valley Technical College (FVTC) in Appleton, Wisconsin. He is a Certified Instructor in AutoCAD, and has previous certification in Mechanical Desktop (now AutoCAD Mechanical) and Architectural Desktop (now AutoCAD Architecture). He teaches at ATCs across the United States, and has been elected twice to the Autodesk Training Center Executive Committee, serving as chairperson in 2001. Craig is coauthor of the popular book "AutoCAD and its Applications – Advanced," and contributing author to "AutoCAD and its Applications – Basics," and Republic Research Training Center's "R14 AutoLISP" Autodesk Certified Courseware. Craig is no stranger to AU, having taught AutoCAD training classes at this event for several years.

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Learning Objectives

After completing this course, you will be able to:

- Understand the need for CAD standards
 - Create a CAD standards file (*.dws)
 - Understand what aspects of an AutoCAD drawing can be maintained via a CAD standards file
 - Associate a CAD standards file with other drawings
 - Check a drawing for adherence to the associated CAD standards file
 - Use the Layer Translator command to “map” a layering scheme or portions of a layering scheme from one drawing to another
 - Set up and use the Batch Standards Checker
-

Using AutoCAD to create a CAD drawing file is one thing, but maintaining consistent drawing file output from user to user and from project to project has to be the ultimate goal of the person or persons in charge of “the system”. This paper introduces you to just some of the concepts that need virtually constant monitoring to maintain that consistency.

INTRODUCTION TO CAD STANDARDS

AutoCAD drawings have four areas that lend themselves to rather easy customization: layer schemes, dimension styles, text styles and linetypes. Because of this ease of customization, company standards and drawing consistency can sometimes take a backseat to personal preference. Standardization of these concepts needs to be developed and maintained within any organization using AutoCAD.

When working in a professional environment it is very desirable to have consistency from one drawing to the next, no matter who created the drawing. When the next person sits down to work on a drawing, whether it is to continue the design process or make some revisions, that person needs to know what to expect when he or she opens that drawing. The four types of information mentioned above need to be consistent throughout the company. A CAD operator should spend more time doing the actual design or layout, rather than trying to figure out which layer hidden objects are to be placed on, which text style should be used to label a section view, or what dimension style should be used to locate a drilled hole.

For a number of years AutoCAD has provided the concept of a template file. The template file holds all the initial standards like those mentioned previously. Starting a new drawing by using a template file allows the CAD operator to have all these standards in place when he or she begins the drawing. But what if a CAD operator gets a little rebellious or strays from the standards by creating a few non-standard layers or linetypes, or makes up his or her own dimension style? How does the drawing get brought back to the consistency of the company standards? That is exactly where two commands, introduced in AutoCAD 2002, come into play: STANDARDS and CHECK STANDARDS.

Creating a CAD Standards File

The first step in maintaining CAD standards is to create them in the first place. This is done by creating a new drawing, establishing all of the desired standard settings for layers, dimstyles, text styles and linetypes, then saving it as a Drawing Standards file. Do this by simply using the SAVEAS command and changing the “Files of type:” drop down list to “AutoCAD Drawing Standards (*.dws)”.

See Figure 1.

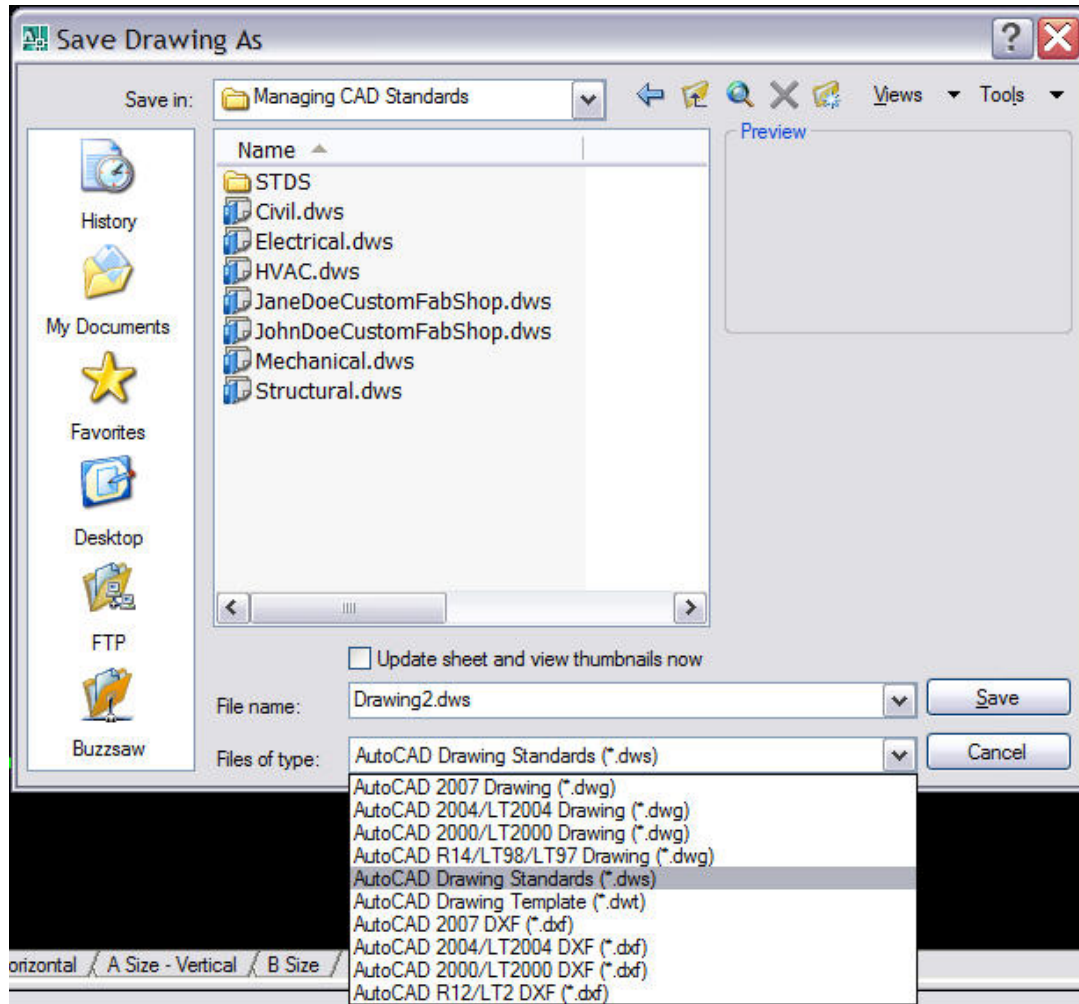


Figure 1. Changing the File type to Drawing Standard (.dws)

Again, the four standards areas that AutoCAD allows the CAD user to maintain control over are layers, dimension styles, text styles and linetypes. The properties associated with each of these standards and which subsequent drawings are checked against for adherence to these standards are as follows:

Layers: The names, color assignments, linetype assignments and lineweight assignments are all stored in the drawing standard file. The layer “states” (on, off, frozen, thawed, locked, unlocked, plotted, non-plotted) are *not* stored.

Dimension Styles: The dimension style names and all of their variable settings are stored in the drawing standard file. Any dimension variable overrides are *not* stored – though they do stay with the drawing standard file, they are not used in the checking process.

Text Styles: The text style names and all attributes of the style are stored in the drawing standard file, including fonts, font styles, height and effects (upside down, backwards, vertical, width and oblique angle).

Linetypes: The linetype names and their segment length and spacing definitions are stored in the drawing standards file.

Exercise 1: Layer Properties Manager

Begin by opening the AutoCAD drawing file “mechanical1.dwg”

Take a moment to familiarize yourself with the Layer Scheme. Add the following layers to complete the development of a mechanical CAD Standards File:

Layer scheme:

Name	Color	Linetype	Lineweight
Construction	252	Continuous	Default
SectionLine	white	Continuous	Default

The following figure shows the completed Layer Scheme:

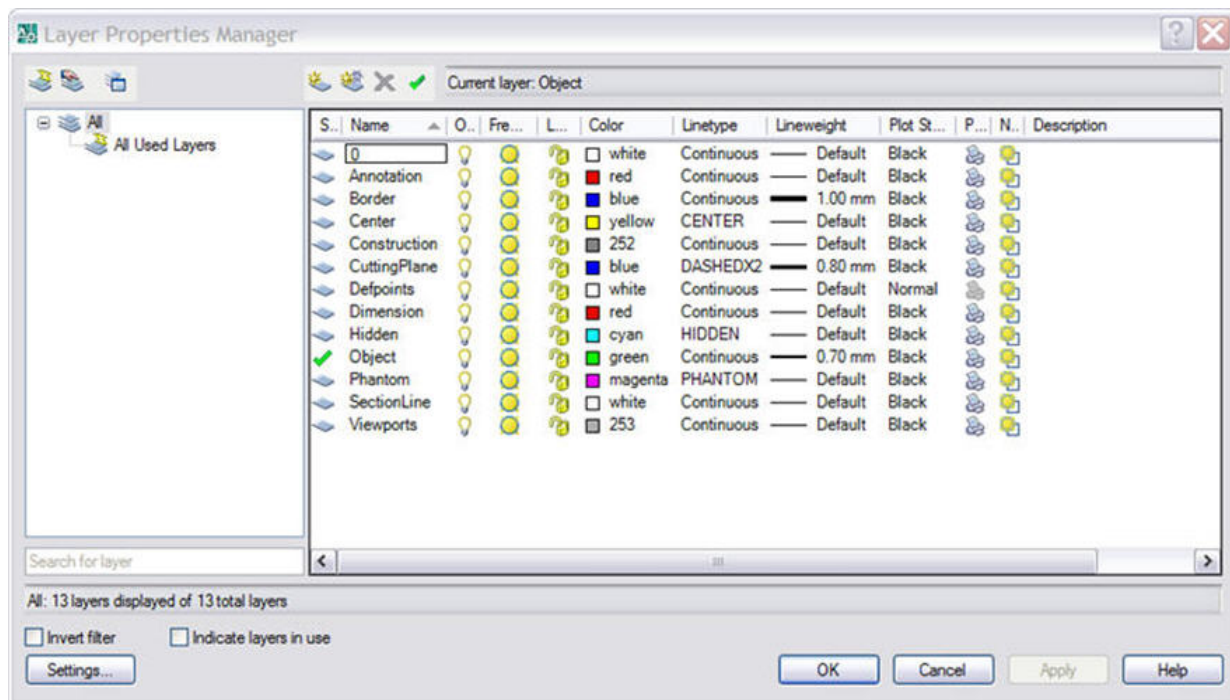


Figure 2. Completed Layer Scheme for a Mechanical Drawing Standard file (.dws)



Exercise 2: Dimension Style Manager

Take a moment to familiarize yourself with the current Dimension Styles. Create a new style based on the “DEC -2PL” style and another based on the “FRAC-16” style, making the following changes to complete the development of a mechanical CAD Standards File:

Dimension Styles:

Section Tab	Item	Dimension Style Name	
		DEC -3PL	FRAC-32
		Start with DEC -2PL	Start with FRAC-16
Primary Units	Precision	.000	1/32

The following figure shows the completed Dimension Styles:

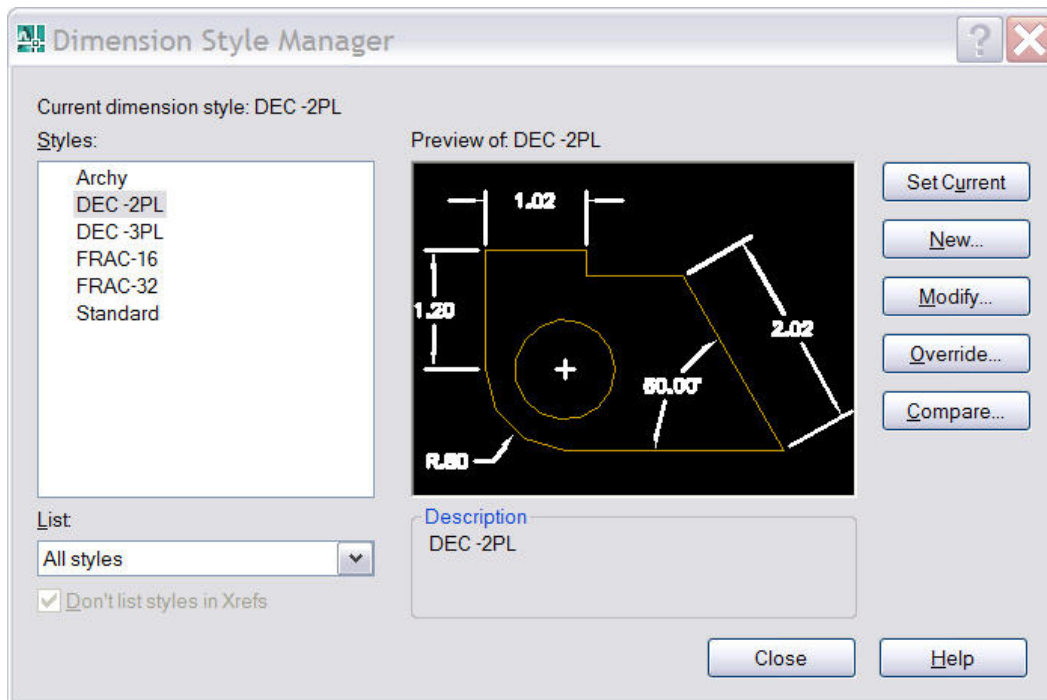


Figure 3. Completed Dimension Styles for a Mechanical Drawing Standard file (.dws)

Exercise 3: Text Style Manager

Take a moment to familiarize yourself with the current Text Styles. Create a new text style as follows to complete the development of a mechanical CAD Standards File:

Name	Font	Width
Label	romand.shx	1.5

The following figure shows the completed Text Styles:

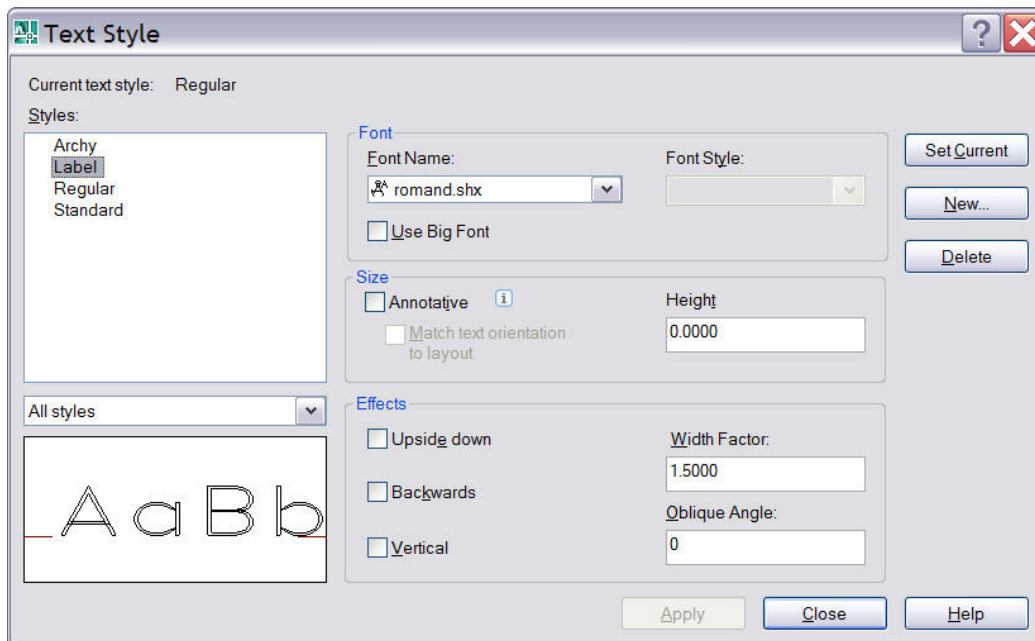


Figure 4. Completed Text Styles for a Mechanical Drawing Standard file (.dws)

Save the drawing keeping the same name. "mechanical1.dwg"

Save the drawing as "STD-Mechanical1.dws" and close all drawing files.

Configuring CAD Standards

Before you can start checking whether the desired CAD standards are being adhered to, you must first associate a CAD standards file with the current drawing. This is done with the STANDARDS command. You can launch this command by typing STANDARDS at the Command: prompt, using the pull down menus (Tools → CAD Standards → Configure...) or by using the first (left-most) button on the CAD Standards toolbar.

When the Configure Standards dialog box opens you will notice two tabs: *Standards* and *Plug-ins*. See Figure 5.

The Plug-ins tab

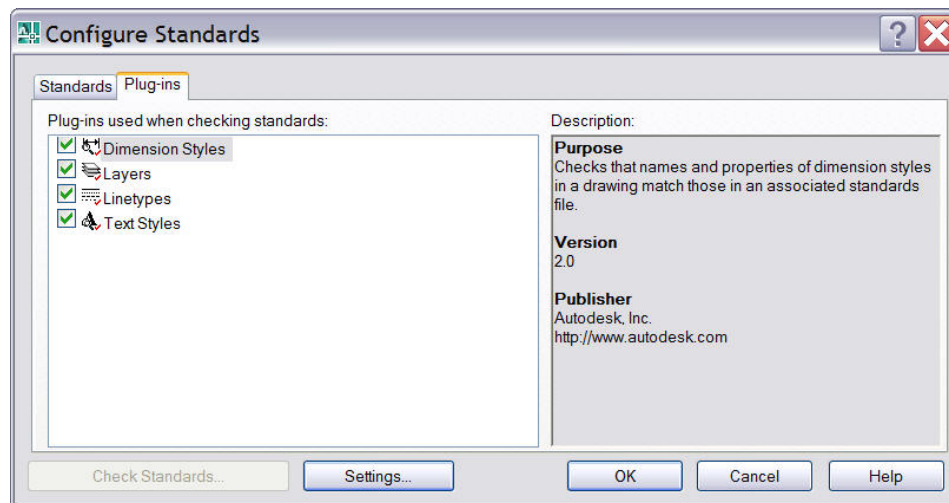


Figure 5. The Configure Standards dialog box displaying the Plug-ins tab

This tab shows the currently supported objects included in the checking process.

The Standards tab

The Standards tab has two main areas. The left side shows any standards files currently associated with the current drawing, and the right side shows a description of the highlighted standards file from the left side of the dialog tab. See Figure 6.

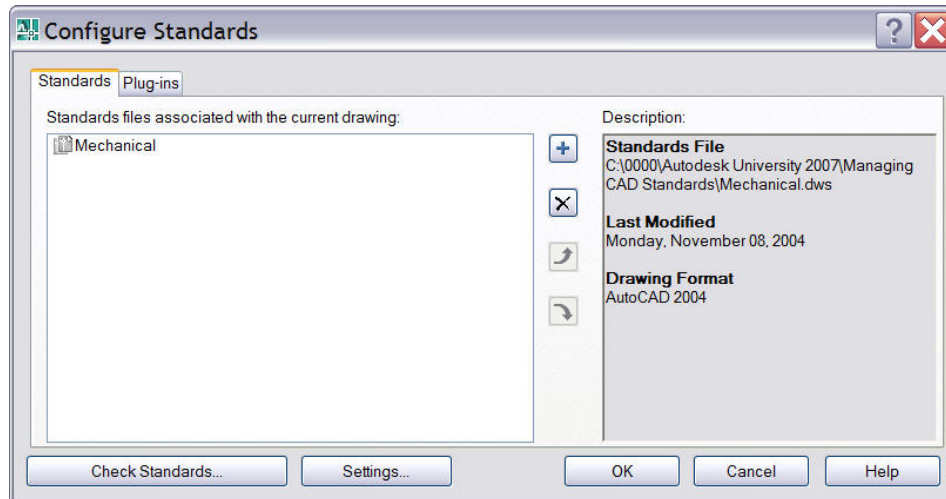


Figure 6. The Configure Standards dialog box displaying the Standards tab

The description area consists of the standard file name and its path, the date the standards file was last modified and the drawing format of the standards file.

Adding a Standards File



To add a standards drawing to the list of files on the left, use the *Add Standards File* button. It appears at the top of the column of buttons in the middle of the dialog box. (You can also press the F3 key). Picking this button brings up the standard Windows “Select file” dialog box. Simply navigate to the location of the file you want to use as a standard and select it. It will then be added to the list.

Ordering the list of Standards Files



Multiple drawings can be added to the list, but the order that they appear in the list is important. When conflicts arise between multiple standard files such as identical layer names but differing properties (color, linetype, etc.) the standard file listed first takes precedence. To rearrange the listed standard files highlight the file to be re-ordered and use the *Move Up* and *Move Down* buttons (or the F4 and F5 keys, respectively) to rearrange the list.

Removing a Standards File



Standards files can also be deleted from the list by highlighting the intended file and picking the *Remove Standards File* button, or pick the Del key. Pick the *OK* button to exit the dialog and save the established standards.

Exercise 4: Non-standard drawing

Begin by opening the AutoCAD drawing file “mechanical1.*dwg*”

You are going to make changes to the Layers, Dimension Styles, and Text Styles. Make the changes indicated in each area below:

Layer scheme:

Rename the following layers:

“Dimensions” to Dims; “Object” to Visible; “Annotation” to Notes

Change the color of the following layers:

“Phantom” to Cyan; “Center” to 50; “Hidden” to 254

Change the linetype of the following layers:

“Hidden” to Dashed2; “CuttingPlane” to Phantom; “Phantom” to Phantom2

Dimension Styles:

Make changes to the “DEC -2PL” and “FRAC-32” dimension styles as shown in the table below:

Section Tab	Item	Dimension Style Name	
		DEC -2PL	FRAC-32
Lines	Offset from Origin	.06	.06
Text	Text Height	.13	.13

Text Styles:

Make the changes to the Text Styles as shown in **bold** in the table below:

Name	Font	Height	Width
Regular	Arial	0.125	0.9
Label	romant.shx	0.25	1.0

Associate the drawing standards file “STD-Mechanical1.dws”, created in Exercises 1 through 3. Save the drawing as “Mechanical2.dwg”

Notice that an “Alert Balloon” pops up from the lower right area of AutoCAD’s status bar. We will ignore it for now, but cover it in the very next section.

Checking CAD Standards

Once a standards file or set of standards files has/have been associated with the current drawing, the process of checking the current drawing for adherence to these standards can begin. This process can be done immediately from the *Configure Standards* dialog box. Pick the *Check Standards...* button at the bottom left of the dialog. You can also type CHECKSTANDARDS at the Command: prompt, or select Tools → CAD Standards → Check, or pick the middle button on the CAD Standards toolbar.

The Check Standards dialog is divided into three main areas: Problem:, Replace with: and Preview of changes. See Figure 7. All three of these areas interact with one another, but let’s look at them one at a time.

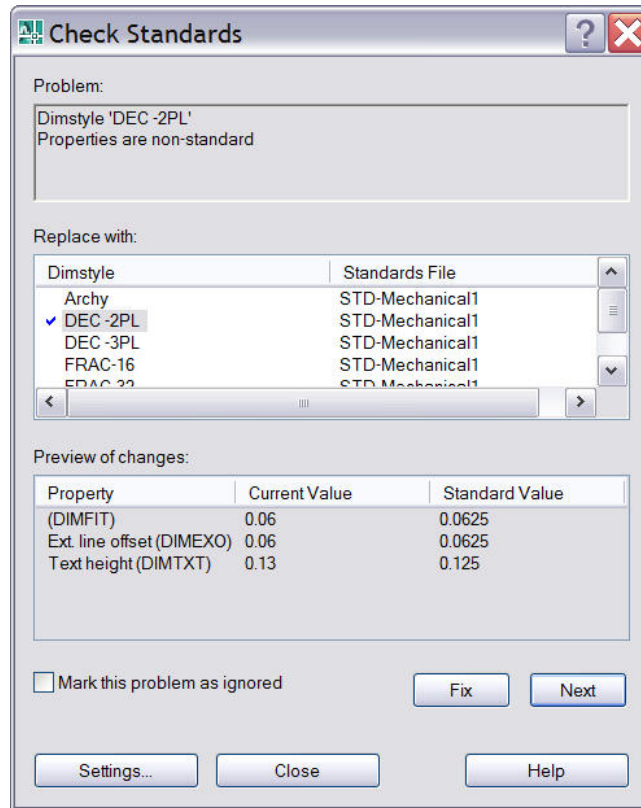


Figure 7. The Check Standards dialog box

Problem:

This area of the dialog will display the problems, one at a time, that are in non-compliance with the associated standard file(s).

The first problem that may appear in this area is the fact that “There are no standards files associated with the current drawing,” Figure 8.

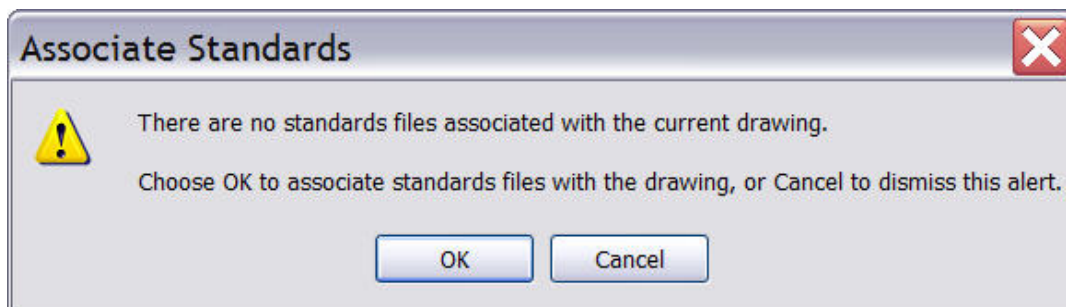


Figure 8. The “no standards file...” error message

If you select “OK”, AutoCAD will launch the **Configure Standards** dialog so that you can associate a CAD standards file with the drawing. Choosing “Cancel” will return you to the Command prompt.

If a standards file has been associated with the drawing, the *Problem:* area of the dialog box will point out a specific problem with the drawing. The *Replace with:* area will allow you to make a change that conforms to the standard file(s) associated with the drawing. And the *Preview:* area will show you the changes that will take effect if the change in the *Replace with:* area is approved.

Replace with:

Here, the possible choices for correcting the non-compliance will be shown. To use this area all you need to do is select one of the items in the list and pick the “Fix” button with the checkmark on it that is located next to the list of choices.

Preview of changes:

This area displays the property and value change that will take place should the currently selected item in the *Replace with:* area be used to fix the problem, and bring the drawing into compliance with the standard file(s).

The “Fix” button

Use this button to update the drawing according to the changes shown in the *Preview of changes:* area, which are based on the highlighted item in the list of choices in the *Replace with:* area. When a “fix” is made, the non-standard named items are purged from the drawing.

Mark this problem ignored

This toggle allows a problem to be marked as ignored. The login name of the user will be recorded with the fact that the problem has been ignored.

Next button

This button is used to go on to the next problem without initiating a fix. The current problem will be shown in subsequent standards checks if it is not marked ignored, unless the *Show ignored problems* toggle is checked in the Settings dialog box. Examples are shown of some of the

commonly encountered type of problems, some possible “fixes” shown in the Replace with: area, and the preview of the changes that will be made. Some of the types of problems that can occur are shown in the following figures:

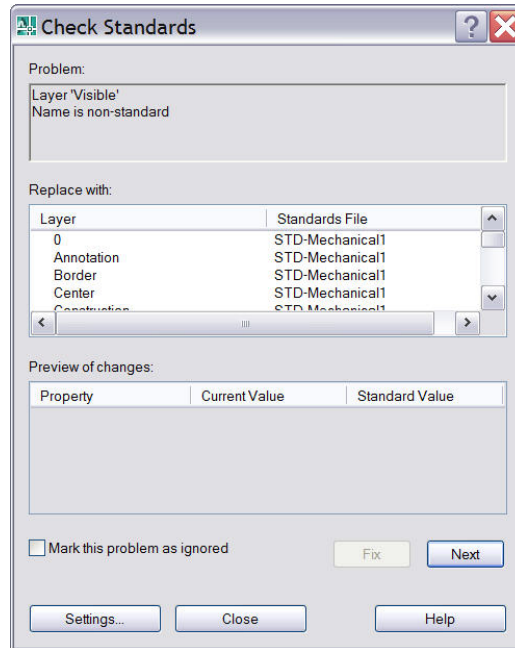


Figure 9. a non-standard layer name problem

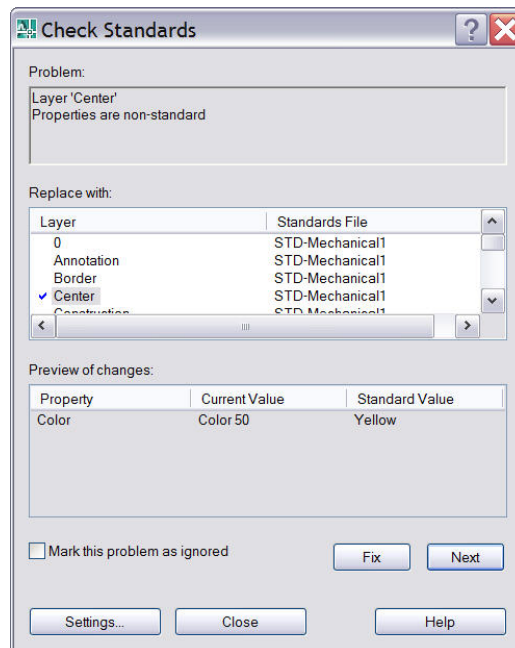


Figure 10. a single non-standard layer property problem

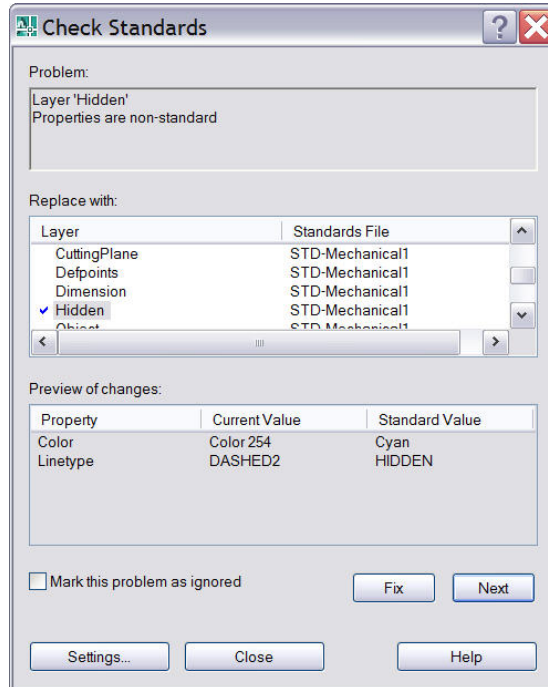


Figure 11. a multiple non-standard layer properties

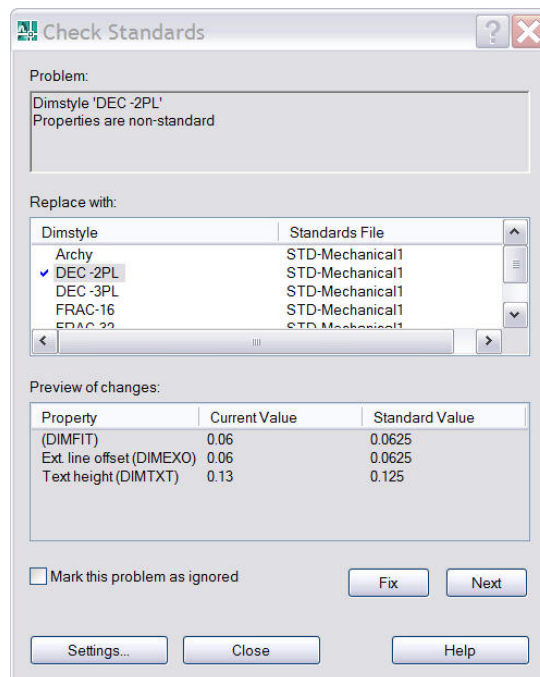


Figure 12. a non-standard dimension properties problem

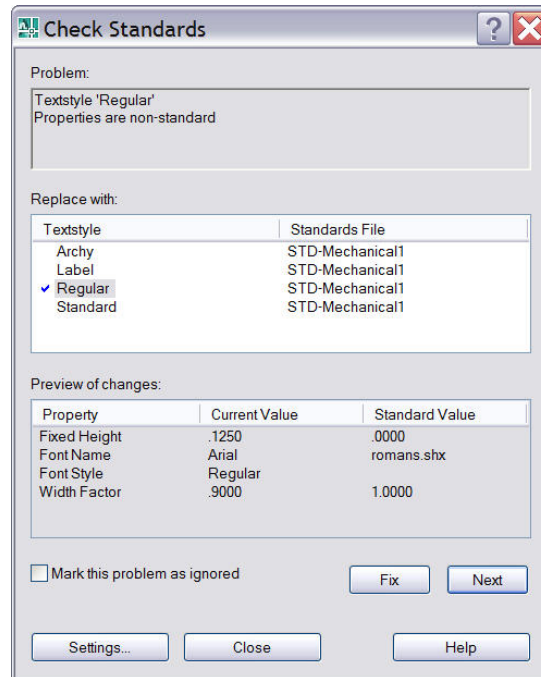


Figure 13. a non-standard text style properties problem

Settings...

This button calls up the Check Standards - Settings dialog box that lets you adjust the way certain standard checking events occur. Select from either the Standards or Check Standards Dialog boxes.

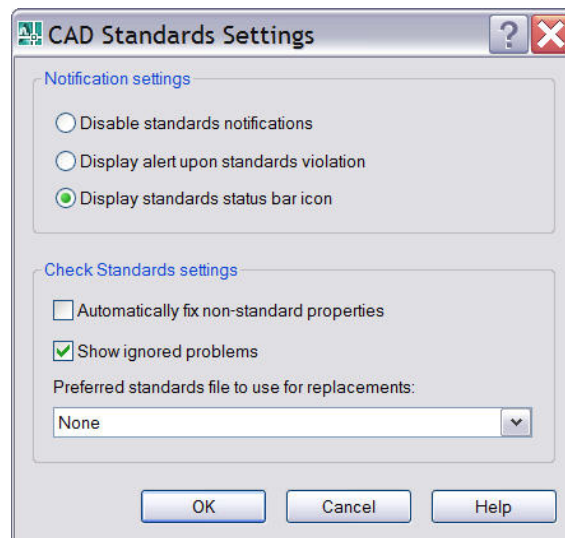


Figure 14. CAD Standards Settings dialog box



Automatically fix non-standard properties

“Automatic” fixes can only take place if the drawing being checked has a standard object with an identical name to a standard object found in the standards file listed in the Preferred standards file to use for replacements: drop-down list. When this toggle is checked all the properties of the identically named objects will be changed to match the properties settings in the preferred file.

Show ignored problems

This toggle, when checked, will allow previously ignored problems to be displayed when subsequent standards checking takes place. The problem, and the login name of the person who previously checked it as ignored, will be shown in the *Check Standards* dialog.

Preferred standards file to use for replacements

This drop down list is used to designate which standards file is used when the *Automatically fix non-standard properties* toggle is checked.

Professional Tips:

- No Undo available within Check Standards dialog – upon exiting either all the “fixes” will be undone or none.
- Do not forget to hit the “next” arrow after checking the “ignore” toggle
- You can get to the “Check Standards” dialog from “Configure Standards”, but not vice-versa
- To avoid an unnecessary flag be sure to include the Defpoints layer needs in the standards file

Exercise 5: Checking for CAD Standard Adherence

Use the CHECKSTANDARDS command and fix the non-standard objects in the drawing created in Exercise 4. (non-standard test.*dwg*)

Ignore one of the problems, then re-use the CHECKSTANDARDS command to see what happens the second time around.

Ignore the problem once again. Use the Settings button to remove the check mark from “Show ignored problems”, then re-use the CHECKSTANDARDS command to see what happens this time.

USING THE LAYER TRANSLATOR

The Layer Translator is another tool that allows you to manage your drawings. Unlike the STANDARDS command, the LAYTRANS command only allows you to manage your layer scheme. LAYTRANS allows you to map the layers in the current drawing to those in another drawing or to a whole new layer scheme. This is extremely useful when two companies are collaborating on a project. Each company can translate the layers from the other company to match the layering scheme that they are comfortable with.

Loading a Drawing To Use For Translating

You can launch the Layer Translator by typing LAYTRANS at the command prompt, via the pull down menu by selecting Tools > CAD Standards > Layer Translator..., or by picking the last button on the CAD Standards tool bar.

Once the LAYTRANS command has been launched a dialog box will appear that has the current drawing's layer scheme shown. See Figure 15. A "layer" icon will appear to the left of each layer name. A "white" icon means that the layer has not yet been referenced (drawn on) in this drawing. A "colored" or filled-in icon means that it is referenced (there are object drawn on that layer). There are a couple other list boxes and a few buttons also in this dialog. Let's take a look at each of these items individually.

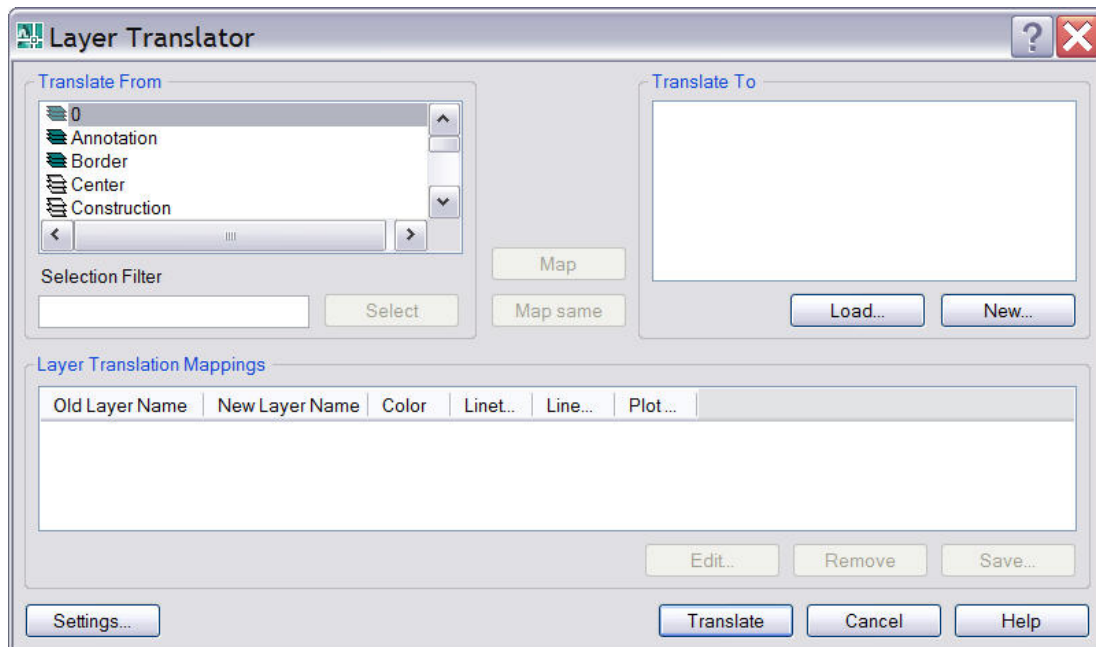


Figure 15. The Layer Translator dialog box

The Load... Button

This button calls up the standard Windows file selection dialog. Using this dialog you can load any drawing file (*.dwg), CAD Standards file (*.dws), or drawing template file (*.dwt). Use the “Look in:” area and the folder list to find the drawing that you would like to use for translating.

The New... Button

The New... button launches a dialog box that lets you create a layering scheme from scratch. You can enter a new layer name and assign a color, linetype and linewidth to the layer – and a plot style if you are using named plot style tables. See Figure 16.

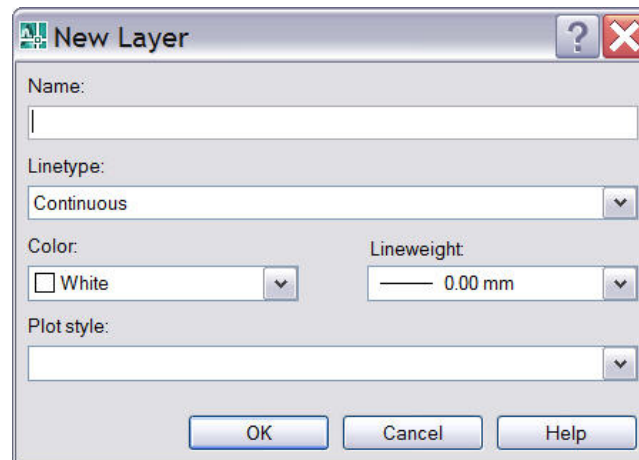


Figure 16. The New Layer dialog box

The Translate To: List Box

This list will show you all the layers that are available for translating to. The list can be created from more than one drawing or from scratch using either the Load... or the New... buttons described earlier.

The Map Button

You can pick a layer out of the “Translate From:” list and a layer out of the “Translate To:” list and have the layer that was listed in the “Translate From:” list take on all the properties, including the name, of the layer listed in the “Translate To:” list. You are allowed to pick more than one layer from the “Translate From:” list. You can select multiple layers with individual picks or by typing in a wildcard combination in the “Selection Filter” edit box located just below the “Translate From:” list and then picking the “Select” button which is located right next to the “Selection Filters:” edit box. See AutoCAD’s online HELP for more information on wildcard combinations.

The Map same button

This button allows you map all the layers that are identically named in the two lists and update them to the properties of the layer listed in the “Translate To:” list. No selecting needs to take place – the selections are automatic, based on identical layer names.

The Layer Translator Mappings List Box

This list shows all of the layer translations that have been mapped. You will see the Old Layer Name, the New Layer Name, along with columns displaying the Color, Linetype, Lineweight and Plot style that the objects on the old layer name will take on.

The Edit Button

This button becomes active when you select a line of information in the Layer Translation Mappings list. When the button is picked you can change the color, linetype, lineweight and plot style assignment using a dialog similar to the one shown in Figure 24-11 except that the Name: area will be grayed out. You cannot change the old or the new layer name. Once you change any of these assignments the change(s) will hold true for any and all mappings using that same new layer name.

The Remove Button

This button allows you to remove a proposed translation from the list. Just select the layer translation to be removed and pick the Remove button.

The Save... Button

The Save... button allows you to create a drawing file that contains all the layers and their properties from the “Translate To:” list. But that is all this new drawing will have – just the layer scheme. None of the drawn objects from the current drawing will become a part of the new drawing. This can be useful if you find yourself having to load 2 or 3 drawings quite often, or if you have had to create a whole new layer scheme from scratch. Save the list using this button and you will not have to go through quite so much work the next time around.

The Translate Button

Once all the mappings have been set up this button will “make it so”. The old layer names and their properties will be replaced by the mapped new layers and their properties. If you have not mapped some of your old layers to new layers, they will be left as they were. If you have not used the “Save...” button to save the mappings as they have been set up, you will be prompted to do so.

The Settings Button

This button will bring up a dialog box of toggles. See Figure 17. Most of these toggles are rather self descriptive but we'll discuss each of them to assure clarity.

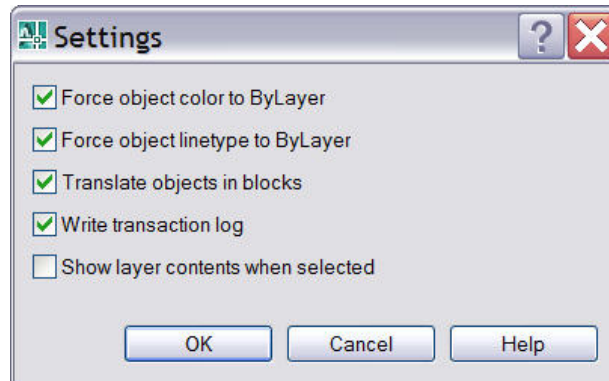


Figure 17. The Settings dialog box

Force object color to ByLayer

If checked, not only will the layer name and its properties be translated but also all objects found on that layer will have their color property forced to “ByLayer”. This is a very powerful feature.

Force objects linetype to ByLayer

Same as above except the linetypes of the objects on the translated layers will be forced to “ByLayer”.

Translates objects in blocks

If checked, objects that are nested within block definitions will translate also.

Write transaction log

If checked, a log file will be created in the same folder of the current drawing that has all the layer translation data recorded in it. The file will have the same name as the current drawing and will have “.log” extension. This is very useful in case you would like to “go back” to the layer scheme that was being used before the translation. There is no other record of “how things were”, so you can use this file to “re-translate”.

Show layer contents when selected

If checked, when the Layer Translator dialog is first called up the drawn objects of the current drawing disappear in the background. As layers are selected from the “Translate From:” list, the

objects that are on these layer(s) will show up in the background. If unchecked all objects in the current drawing that are not turned off or frozen will show up in the background. Objects that are on layers that are turned off will still show up when their layer name is selected if this box is checked. Objects that are on layers that are frozen will never show up.

Exercise 6: Using the Layer Translator

Begin by opening the AutoCAD drawing file: "LayerTrans-Arch.**dwg**"

Open the Layer Properties Manager and study the Layer Names, Colors, and Linetypes

Draw a couple objects on each of the layers listed here: A-Plumbing; A-Wall; A-Windows; Dim; Doors; Elevation; Section AA; Text

Freeze the "A-Electrical" layer

Turn Off the "Dim" layer

Load the CAD Standards file "Standard-Arch.**dws**"

Experiment with the Layer Translator command

Experiment with the "Settings" dialog inside the Layer Translator, especially with the last toggle and how it affects some of the drawn objects.