

CMI Standards Manager User Training Manual

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Conventions Used in this Manual

This section describes the format of the notations used in this manual.

The following fonts are used throughout this guide:

Standard Font	Times New Roman is used as the standard font for the body of this document.
Bold Font	A bolded font represents a command name as shown on a pull-down menu, toolbar, or dialog box.
<i>Italic Font</i>	Italic Font is used for narrative instructions regarding user input, for file and folder names in the body of the text, for dialog box titles, and to highlight software product names.
Command Line Font	Courier New font is used to delineate commands shown within the AutoCAD command window.

Format	Description
<Name>	Represents keys found on the keyboard
AutoCAD LINE command	AutoCAD commands and variables are capitalized, and are commands preceded with the word AutoCAD.
* ?	Represent wildcard characters "*" and "?"
<i>symbols</i>	Italic type indicates the name of a directory or file as shown in the body of the text or for dialog box titles. File and folder names are typically shown as lowercase letters.
NAME	Uppercase type indicates a variable, layer name, or command name as shown in the body of the text.
{ <i>File Name</i> }	Italic uppercase type enclosed in fancy braces indicates an item which varies. This example indicates that the item varies by file name.
Suggested Exercise:	Training manual exercises or important reference notes are denoted by shaded backgrounds.

Introduction

CMI Standards Manager (CSM) automates the implementation of CAD standards within AutoCAD® by maintaining your firm's standard layers, symbols, details, sheet borders, text styles, line patterns, and custom programs in a shared database. This product does not replace existing fundamental AutoCAD functions, but provides productivity enhancements that by default follow an organization's computer aided drawing standard.

CMI Standards Manager is comprised of two software modules: The "User Module" is a simple, user-friendly interface designed for straightforward application of your drawing related standards by all of your AutoCAD® users. The *CSM Administration Module* provides easy access by management personnel for development and maintenance of the company/agency standard.

Some of the features of *CMI Standards Manager* include:

- Layer, Color, and Line Pattern Management
- Symbol, Detail, and Sheet Border Management
- Text and Dimension Style Management
- Custom Program Management

This manual was written to facilitate the use of the *User Module for CMI Standards Manager*. For information on the use of the *Administration Module*, please see the second manual included with this software package: *CMI Standards Manager Administration Module*.

Some of the commands in *CMI Standards Manager* work closely with existing AutoCAD commands. This training manual was written with the assumption that users have basic knowledge of the use of AutoCAD. For detailed information on related AutoCAD commands, please refer to the AutoCAD User's Guide or AutoCAD Command Reference.

 Please note: For the purposes of this manual we are assuming the use of a right-handed mouse

Explanation of Standards

The *CSM* uses a system of standards, disciplines, groups and categories to organize the procedures CAD designers and drafters are expected to follow.

A **Standard** is a set of company/agency layers, colors, linetypes, symbols, details, sheet borders, text styles, dimension styles and custom programs. *CSM* controls standards setups via a relational database and organized folder structures containing symbol, detail and sheet border drawings. The root folder of each standard setup is named identical to the name used for the Standard setup. The *CSM* allows the use of an unlimited number of standards for agencies/companies/clients need to be followed.

A **Discipline** is a subset of the company/agency standard layers, symbols, details, sheet borders, and text styles. Each department within the company/agency can define their own discipline within a standard in order segment their project work from other departments. Examples discipline names would be “Architectural”, “Civil Engineering”, and “Mechanical.”

A **Group** is a subset of the CAD Department’s discipline-specific standard CAD elements. A discipline’s layers, symbols, details, and sheet borders drawings can be subdivided into different groups according to type or use. Examples groups could be walls, doors, roads, and sewers.

Layer Categories are used to as the basic building blocks for the creation of layer names within a discipline. *CMI Standards Manager* allows use of one to five category layer setups that can be assigned to a given discipline. A single category system is simply a complete layer name as a category component. In a multi-category setup a selection from each category is combined to create a layer name.

The Layer Standard

Layers are the basic organizational element within AutoCAD. As such, an organization’s layering standard is the most important guideline in their plan set development. The larger the organization the more important to it is to impose a system of standards upon the CAD users that facilitates communication. The effective implementation of a layering standard is the first step in establishing this communication.

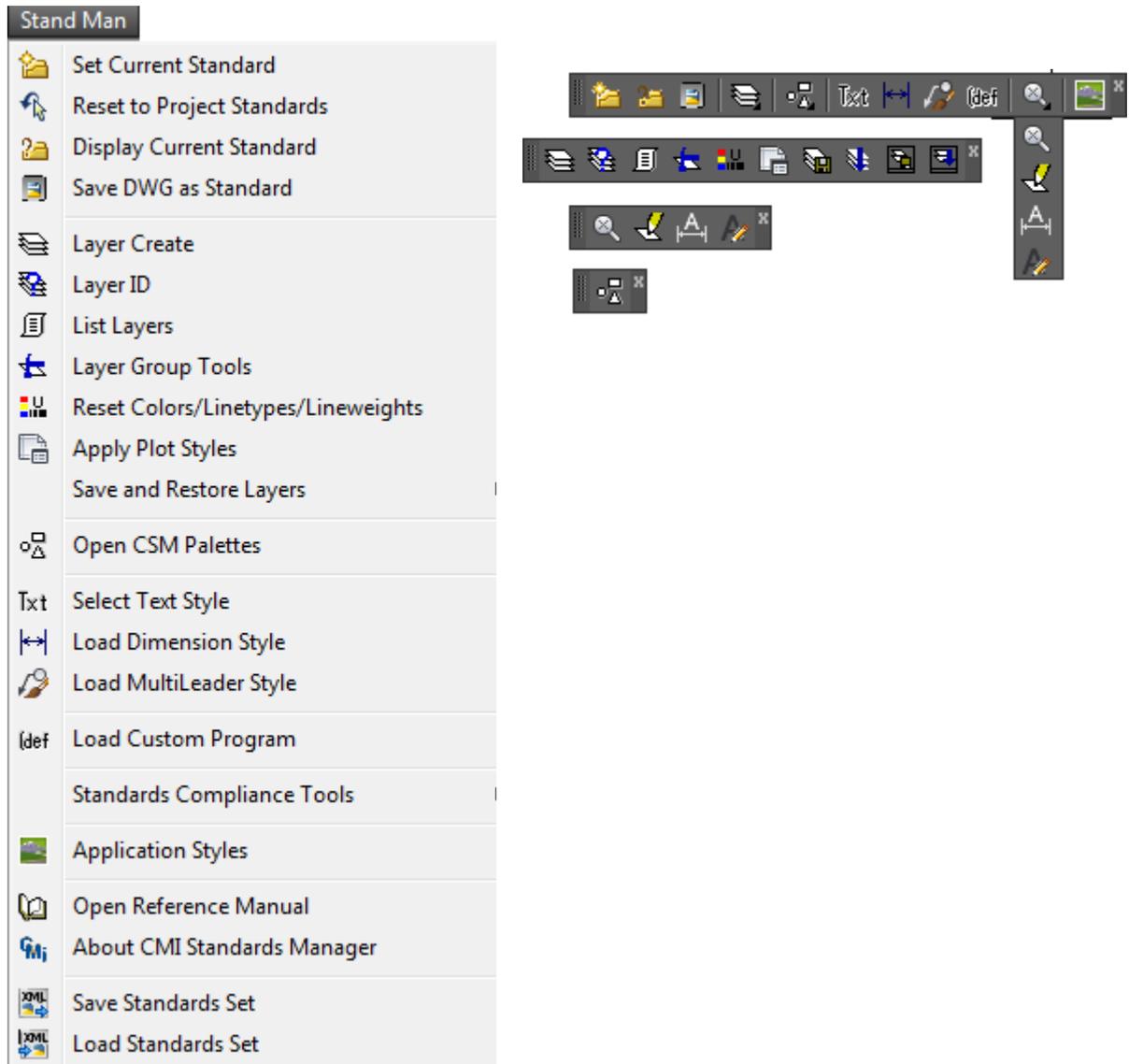
The Layer Standard is an agreed-upon structured naming convention for AutoCAD layers. Each layer should be assigned base properties of color, linetype, and optionally, linewidth. In the past most organization’s layer standards were published in a manual that was given to its CAD users. Each user was expected to create the layers he/she requires while following the guidelines of the Standards manual. This method has the inherent problems of compliance and introduction of user error. A standard is of no use unless it is followed.

The *CSM* is designed to simplify the implementation and management of CAD Standards while helping ensure user compliance. This software employs the concept of “the path of least work.” With an intuitive user interface, *CSM* reduces the effort required to create layers within CAD drawings while following a prescribed layering standard. By making the layer creation process easier than base AutoCAD commands a path is created that CAD users will choose to follow because it simplifies and speeds up their work.

☞ Please note: Although the user interface makes it easier for standards to be followed, they are still “passive” standards. It is ultimately your responsibility to use *CMI Standards Manager* layer functions to create new layers when required and to set the currently active layer.

Layer Commands Menu / Toolbar / Ribbon

A view of the CSM menu is shown in the figure below. The layer commands available within the software include layer creation; identify, list, and print layers found within a drawing; update layer properties based on your CAD Standards; change the layer states of layer groups; convert existing layers to your selected CAD Standard; as well as save and restore layer states.



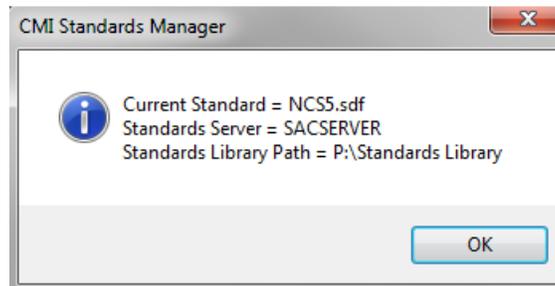
Standards Setup

Set Current Standard

Your CAD manager will have preset a default standard for your use. If this is your organizations only standard setup within *CSM* then you can simply start using the *CSM* functions since every drawing file created and opened will be automatically linked to this standard database. If you link the current drawing to a different standard than that of the default then you can use the function: **Stds Manager > Set Current Standard** from the pull-down. You must find the desired *standards.mdb* file in the search list. Please note that by default the file selection dialog opens in what is referred to as the “Standards Library” root folder.

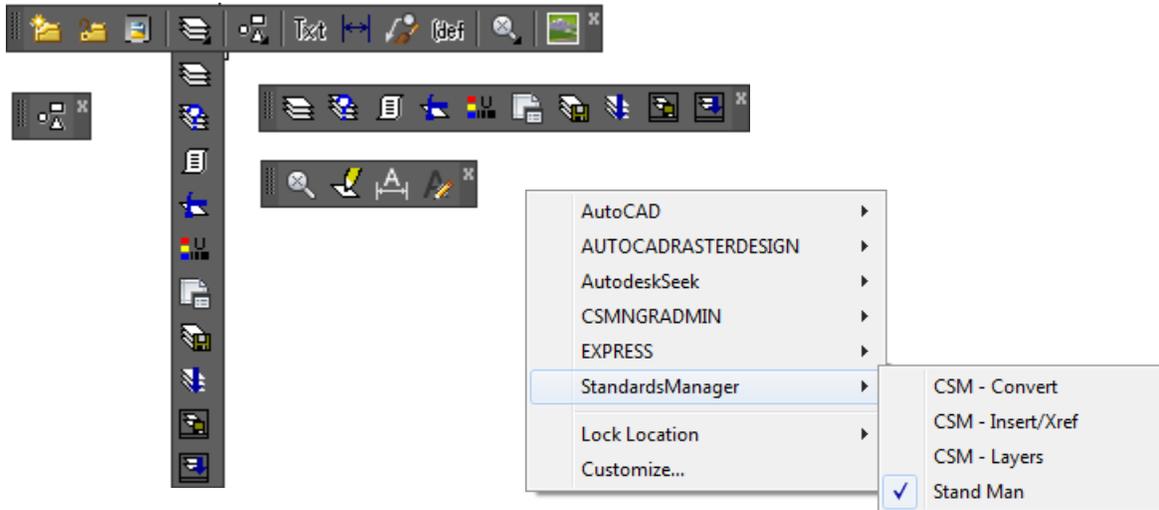
Display Current Standard

If you are unsure about which standard is currently in use, simply use the command **Stds Manager > Display Current Standard**, and a dialog box will open displaying the currently assigned.



Standards Toolbars

The leftmost button on the *CSM* toolbar is a fly-out containing all available **layer** commands. Click and hold the mouse on top of the toolbar button to see all available functions as shown in the images below. Drag the mouse to the desired command and release the mouse button. This will activate the command and leave that command's button as the current button on the **CMI Standards** toolbar. An alternative to using the Layer Commands as a fly-out toolbar you can activate the toolbar separately by right clicking on the CMI Standards toolbar and selecting **Standards Manager – Layers** from the now active context menu.



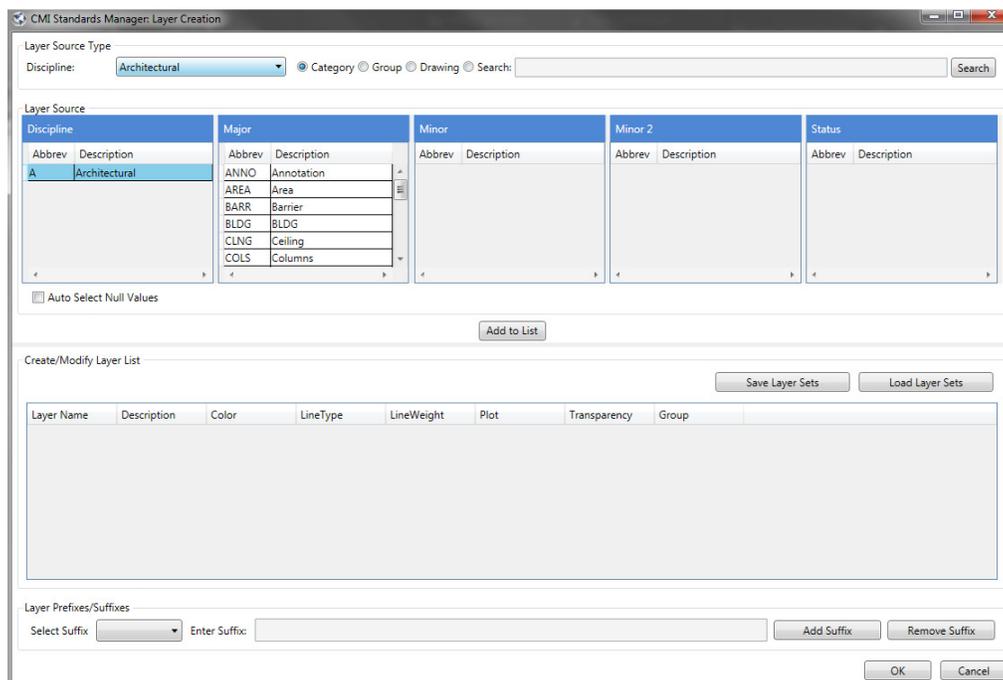
Layer Creation

Overview of the Layer Creation Process

Standard layers are created within the *CSM Layer Creation / Management* dialog. The process of adding layers begins by selecting an appropriate discipline. You may then select layers by one of three options: **By Category**, **By Group**, and **From Drawing**. The options may be used more than once or interchangeably during the same session. For example, you might want to include an entire group of layers from a discipline. The **By Group** option allows you to accomplish this quickly. The **By Category** option allows you to select layers by individual category components. Once you have selected layer components within each displayed category you created the layers immediately or you can add them to the dialog's layer list and return the main dialog by pressing **Ok**. You can continue to add layers and/or make modifications such as changing the color and screening or adding a prefix and/or suffix to the layers shown in the list. Modifications can be made to individual layers or in certain cases to the entire list of layers. Finally, once the appropriate layers have been selected and any necessary modifications have been made, the **Create/Update** button adds or updates these layers in the current drawing.

Layer Creation / Management

The **Layer Create** command allows you to quickly add layers from the appropriate standard or easily “modify” layers working with existing drawings. To begin the layer creation process, select the **Layer Create** command from the **Stds Manager** pull-down menu or click on the **Layer Create** button  on the *CMI Standards* toolbar.



CSM Layer Creation / Management dialog commands / functions overview:

Select Discipline This drop-down sets the current discipline within which layers have been defined.

Layer Selection Options

By Category Allows selection of layers by multiple category/component naming system.

By Group Allows selection from a defined layer group.

From Drawing Imports layers from the current drawing.

Search Search layers in the current discipline.

Layer List Functions (Right-Click)

Set Current Sets the highlighted layer as the current layer upon creation.

Remove from List Removes selected (highlighted) layers from the list.

Screen Set the highlighted layers to the default screening color per lineweight. You be in a color-based standard setup (.ctb) in order to utilize this feature.

Change Color Changes the highlighted layer to a new color.

Load Linetype Load a linetype without actually creating a layer

The **OK** button is used to create the new layers or make any changes to existing layers in the current AutoCAD drawing. That is, no new layers are created or existing layers modified until you close the layer creation session by clicking the **OK** button.

Optional Layer Prefix / Suffix Additions

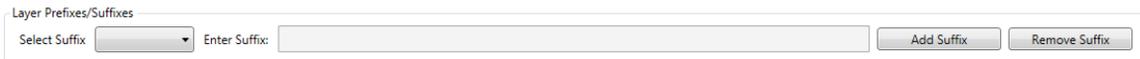
The *CMI Standards Administration* module allows the addition of a prefix and/or suffix to extend layer names. If this feature has been enabled within a discipline the *CSM Layer Creation / Management* dialog will show additional commands at the bottom of the dialog as shown in the figure below.



Select Prefix / Suffix Adds the selected prefix/suffix to the selected layer(s) and optionally changes the color, lineweight or linetype based on the standard's setup. Consult your CAD Manager as to the intended uses.

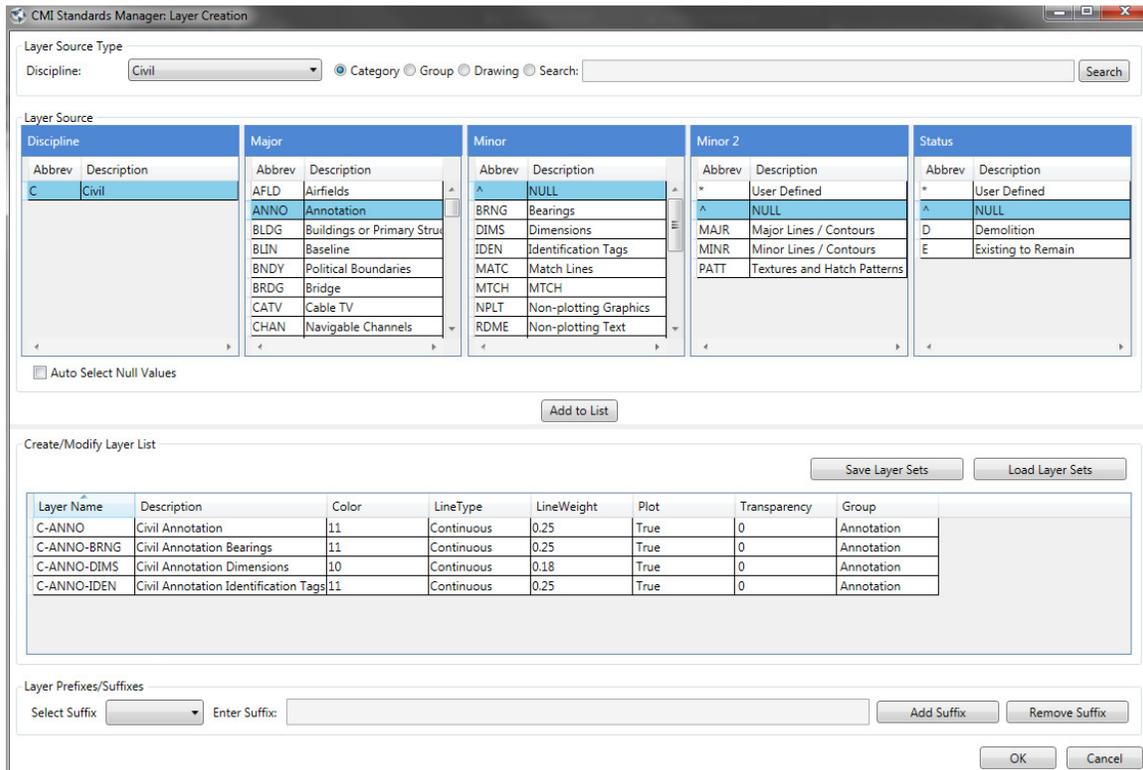
Enter Prefix / Suffix This field allows the user to enter an arbitrary prefix / suffix if enabled from *CAD Standards Administration* module.

Remove Suffix Removes the prefix and/or suffix from the selected layer(s) and restores them to their original layer properties.



Layer Selection by Category

Choose this option to select layers by a combination of category components. The *CSM Layer Selection (By Category)* dialog box is shown in the figure below.



Category names are defined within the *CSM Administration* module. In the example above a five-category system has been defined. The five categories have been named (by the Administrator), in order, Discipline, Major, Minor 1, Minor 2, and Status. Each layer category contains layer components (the basic building block of layer names) defined

within the chosen discipline. A layer component is defined as a layer name abbreviation (the part that will appear in AutoCAD's layer manager) with a description as shown in the **Abbrev** and **Description** portion of each category column in the figure.

☞ Please note: The *CMI Standards Administration* software allows the use of one to five category layer naming. If a discipline uses a one-category layer setup (i.e. simply a complete layer name), layer selection **By Category** will be disabled in the *CSM Layer Creation / Management* dialog.

Begin the selection process by picking a component in the first category. Possible component selections are then displayed in the second category allowing the user to select the next layer component(s). Continue this process until at least one component is selected in each category.

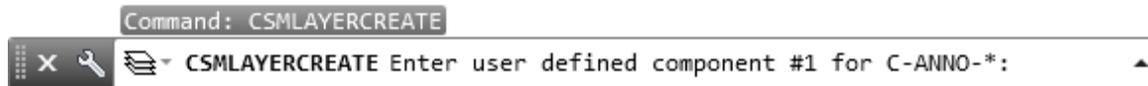
Individual or multiple components can be selected in each category. If multiple component selection is desired, standard windows selection techniques can be used (i.e. <SHIFT> and <CONTROL> keys) or simply click and drag the mouse over a group of components to select them. Selecting multiple components in a category displays components in the next category that are common to the selected set of components. If no components are displayed in the next category after multiple components have been selected then there are no components in the next category that will link to all the components selected. Reduce the selection until components appear in the next category. This method of selection allows quick isolation of layers even within a very large layer set.

Once the desired layers are highlighted you have the option to place them in the list at the bottom of the dialog, **Add to List** button, or simply press the **Create** button to immediately create the layers constructed. If **Create** is pressed with only one layer selected (one component has been selected in each category) that layer will immediately be set current.

Use **Add to List** to create a set of layers not selectable in one pass or if you desire to return to the main layer creation dialog. If undesired layers have been included in the *Selected Layers* list, highlight the layer(s) and right click and choose **Remove from List**. If additional layers need to be added to the list, simply repeat the selection steps above and click **Add to List**. Once all the desired layers are shown under *Selected Layers* click **Ok** to import these layers into the *Layer Creation / Management* dialog layer listing.

Optional Layer Component Manual Input

Assuming the CAD administrator created a user defined input field (an asterisk “*”) for a layer component, you may select this wildcard component to create a layer that better describes your intent. When you add the layer to the *Selected Layers* list or when you create the layer(s) you will be prompted to enter the desired layer component from the command line as shown below.



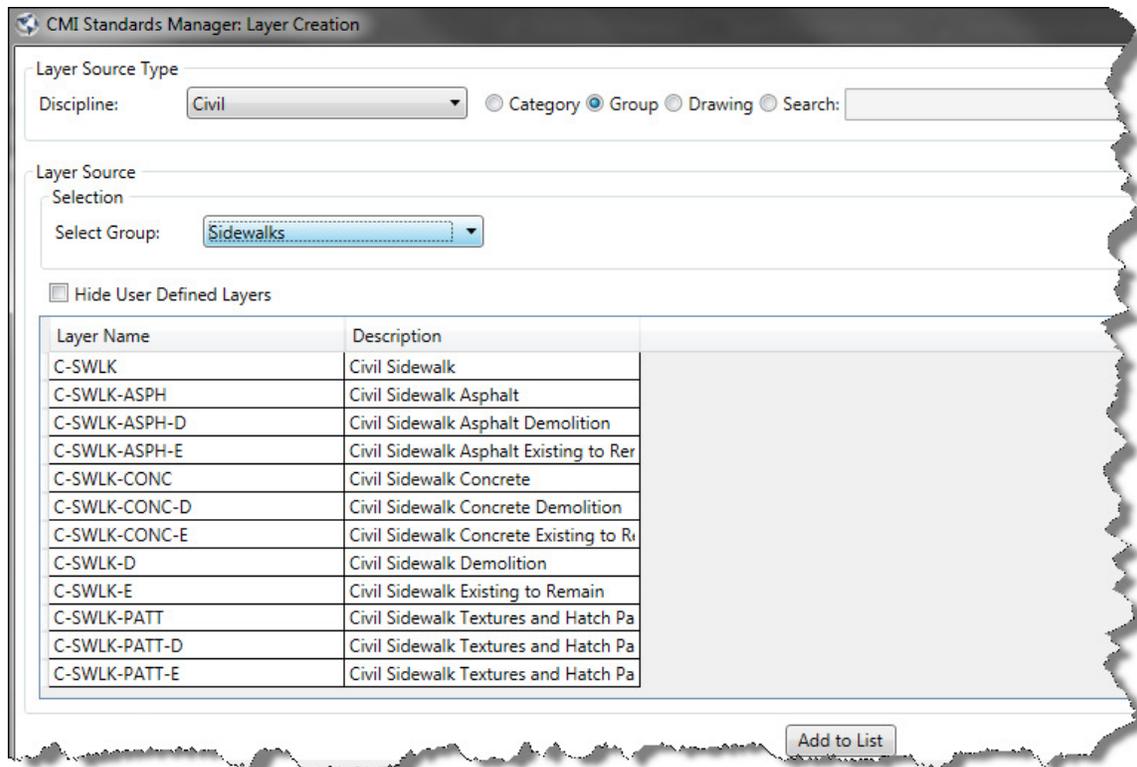
Suggested Exercise:

(For discipline setups configured with multiple layer categories)

1. From the **Stds Manager** pull-down menu select **Layer Create**. Once Layer Creation / Management dialog appears, select a desired discipline and then press the **By Layer Category** button.
2. Select at least one component in each category.
3. Press the **Add to List** button. Note that the **Create** button has vanished and is replaced by an **OK** button.
4. Highlight the items in the *Selected Layers* list and then press the **Remove from List** button. Note that the **Create** button has reappeared.
5. Select one component once more in each category and then press the **Create** button. Note that the layer has been created and is set current within AutoCAD.

Layer Selection by Group

Choose this option to select layers from a defined layer group. You can select all layers, a subset of layers, or individual layers from a group as illustrated in the *CSM Layer Selection (By Group)* dialog shown in the figure below.



Begin the process by selecting the appropriate group in the **Select Group** drop-down list. Only those groups within the discipline selected at the top of the *CSM Layer Creation / Management* dialog are displayed. You can also use standard Windows selection techniques to select layers or simply click and drag with your mouse to highlight the desired layers. Once the desired layers have been highlighted click **Add to List** to add them to the layer list, and finally click **OK** to import them to the drawing.

Suggested Exercise:

1. Press **Layer Create** from the **Stds Manager** pull-down menu.
2. Pick a desired discipline and then press the **By Group** button.
3. Inside the *CSM Layer Selection (By Group)* dialog pick a desired group from **the Select Group** list.
4. Using the various Windows multiple selection methods pick a sampling of layers from the list.
5. Press **OK** to return this selection to the main layer creation dialog.
6. Repeat steps 2 through 5.
7. Finally, press the Create/Update > button found in the lower right corner of the *CSM Layer Creation / Management* dialog box.
8. Now open AutoCAD's Layer Manager and note the layers and associated properties that have been added to the drawing.

Import Layers from the Current Drawing

Choosing the **From Drawing** option activates The *Import Layers from Current Drawing* dialog shown in the figure below. You may wish to import layers from a drawing in order to apply quick modifications to layers such as changing colors or adding a prefix and/or suffix.

Suggested Exercise:

1. Open the Layer Creation dialog and click the **From Current Drawing** radio button.
2. Highlight the desired layer and click **Add to List**.
3. Highlight all the layer in the Create/Modify Layer List, then choose a layer prefix or suffix, finally click Add Suffix to add to the existing layers.
4. Finally, press the **OK** button to update the layer properties in the drawing.
5. Check within AutoCAD's Layer Manager that the new layers have been created.

Layer Prefix/Suffix

Layer Name	Description	Color	LineType	LineWeight	Plot	Transparency	Group
C-CHAN-A	Civil Navigable Channels	32	Continuous	0.35	True	0	Navigable Channels
C-COMM-A	Civil Communications	112	Continuous	0.35	True	0	Communications
C-CTRL-A	Civil Control: Points / System	3	Continuous	0.35	True	0	Control Points
C-DETL-A	Civil Detail	3	Continuous	0.35	True	0	Details

 Please note: This portion of the *CSM Layer Creation* dialog box is only visible in those disciplines for which a prefix or suffix has been defined via the *CMI Standards Administration* module.

This option allows the addition of a prefix and/or suffix to the selected (highlighted) layers (more than one layer may be selected for modification). A prefix or suffix is often used to add additional description to a layer. For example, different phases of a project drawing could be identified with the addition of a prefix of the form PH1-, where PH1- stands for “Phase-1”. An example of a suffix might be “-DEMO” for “to be demolished”. You may select from the list of available prefixes or suffixes in their respective drop-down box.

Adding a prefix and/or suffix not only changes a layer's name but also can cause a layer's properties to be modified. That is, a layer's color, lineweight, linetype and named plot style can be altered by the addition of the prefix or suffix. These modifications are set when the prefix/suffix definition is created in the *CMI Standards Administration* module. (For further information please see the *CMI Standards Administration* documentation or talk with your CAD Manager.)

It is also possible to create a wildcard (*) definition. If the wildcard character, "*", is available in the drop-down list selecting this option will activate the **Enter Prefix/Suffix** edit box. This provides you the option of creating your own prefix or suffix name.

Once you have selected or entered the desired prefix /suffix, clicking the **Add** button will modify all of the highlighted layers and optionally update the layer's properties (i.e. color, linetype, etc.) according to the prefix/suffix definition. Note: these changes do not take effect in the drawing until you exit the layer creation session by clicking the **Create/Update** button.

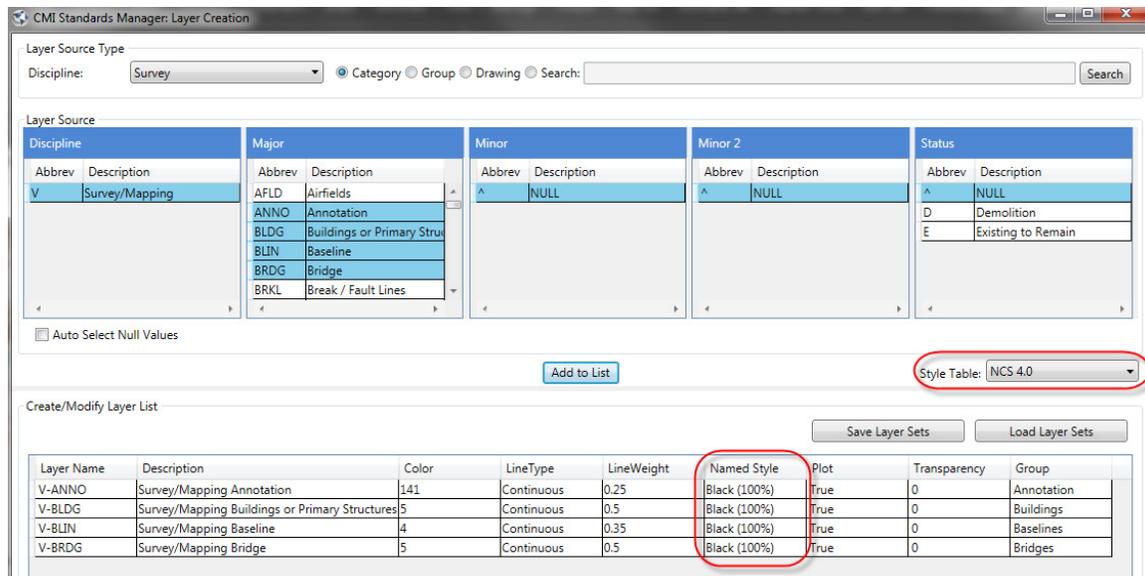
If a layer already exists in a drawing and you wish to add a prefix or suffix you can import the layer into the *Layer Creation* dialog using the **From Drawing** method and make the desired modifications. Note that the original layer is renamed in the drawing at the successful completion of this process. If the original layer and modified layer are both required in a drawing, you should import the original layer from your standards, add the prefix or suffix, and click the **Create/Update** button to create this new layer in the drawing.

 Please note, if you have two layers in a drawing created through this process, that is, a standard layer and that same layer root with a prefix and/or suffix added you can **not** remove the prefix or suffix from the extended layer name since this would attempt to rename the layer to a layer name that already exists in the drawing.

If a layer name has been extended with a prefix and/or suffix and you wish to restore it to its root name and properties use the **Remove Prefixes and Suffixes** button. First import the layer(s) from the drawing, highlight the layer(s) that need to be restored and click the **Remove Prefixes and Suffixes** button. Once again, these changes will not take affect in the drawing until you click **OK**.

Style Based Layer Setups (use of *.stb* plot files)

If CSM determines that the current drawing is set to use named plot style tables (*.stb*) the *Layer Creation / Management* dialog displays an additional command **Set Plot Style Table** in the upper right portion of the dialog. The drawing's default standard plot style table should be displayed. If this list is blank please consult with your CAD Manager so that you AutoCAD setup can be corrected. The dialog in this configuration is shown below.



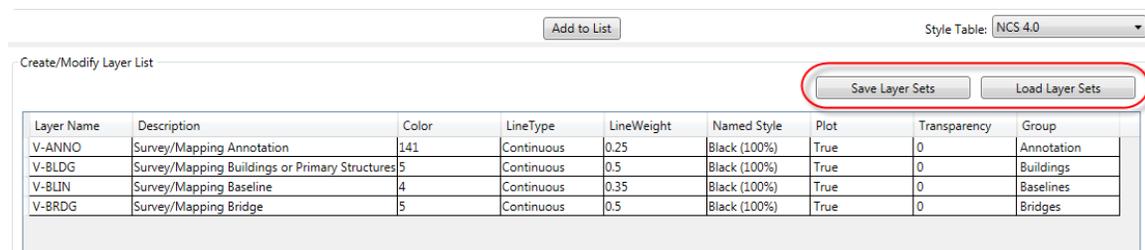
You should also note the additional column “PlotStyle” that is now displayed within the layer list window. The software refers to the current standard database to determine the named plot style for each layer. If you have more than one plot style table available for your CAD standard then the additional table names should be listed in the drop-down list.

Suggested Exercise: (for setups that employ use of named plot styles)

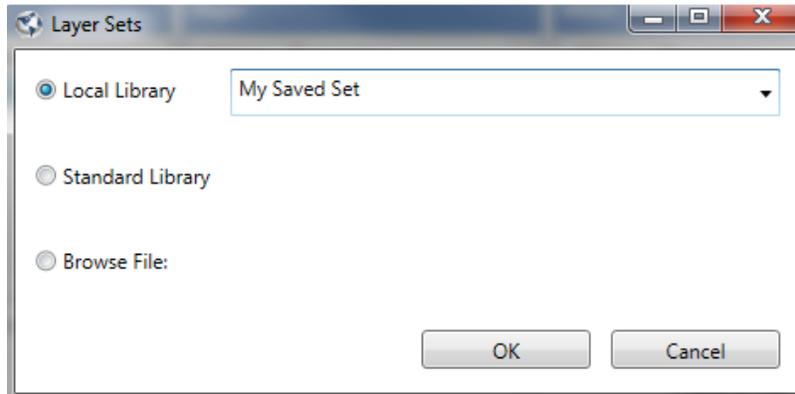
1. Open the Layer Creation dialog and select a desired discipline.
2. Bring in layers to the main dialog using **By Layer Category** or **By Group**.
3. Note the Plot Style names assigned to the layers
4. Press the Create/Update button to create these layers in your currently active drawing.
5. Finally, open AutoCAD’s Layer Manager and verify the Plot Style settings for the layers just created.

Save and Restore Layer Sets

The layer creation command allows the user to save a list of layers for retrieval at a later time. This can be convenient for layers that are created often. The user chooses the desired layers and clicks the **Add to List** button. Once layers are in this list the user can use the **Save Layer Sets** button to save the list.



The user has the ability to save the set to the local computer, the standard library location (permissions required), or to an external file.



The default saved locations for these options are as follows.

Local Library:

C:\Users\<login name>\AppData\Roaming\CAD Masters\CSM\LocalLibrary <version>\<Standard Name>\LayerSets

Standard Library:

<Standard Library Location>\<Standard Name>\LayerSets

Other Layer Functions

Layer ID

The **Layer ID** command allows you to quickly identify an entity's layer. Run the command by selecting **Layer ID** from the **Stds Manager** menu or by clicking on the **Layer ID** button  on the CMI Standards toolbar.

Once the **Layer ID** tool is active the cursor now displays as the standard AutoCAD pick-box. Hold the pick-box over an entity and click with the left mouse button. This command will also work with XREF layers as well, however you must use the Xref option to do so. The entity's layer will be identified in the AutoCAD Command line as illustrated in the figure below. If the layer is not within the current project's standard, it will return the layer name but show "Non-Standard Layer" for the description.

```
Command: CSMLAYERID
Select object [Xref]:
V-TOPO-MAJR: "Survey/Mapping Topography Major Lines / Contours"
```

```
Command: CSMLAYERID
Select object [Xref]:
C-EXMP: "Non-Standard Layer"
```

The **Layer ID** command remains active as long as you continue selecting entities. To cancel, press the <Enter> key or your right mouse button. A left mouse click within a blank area of the drawing will also terminate the command.

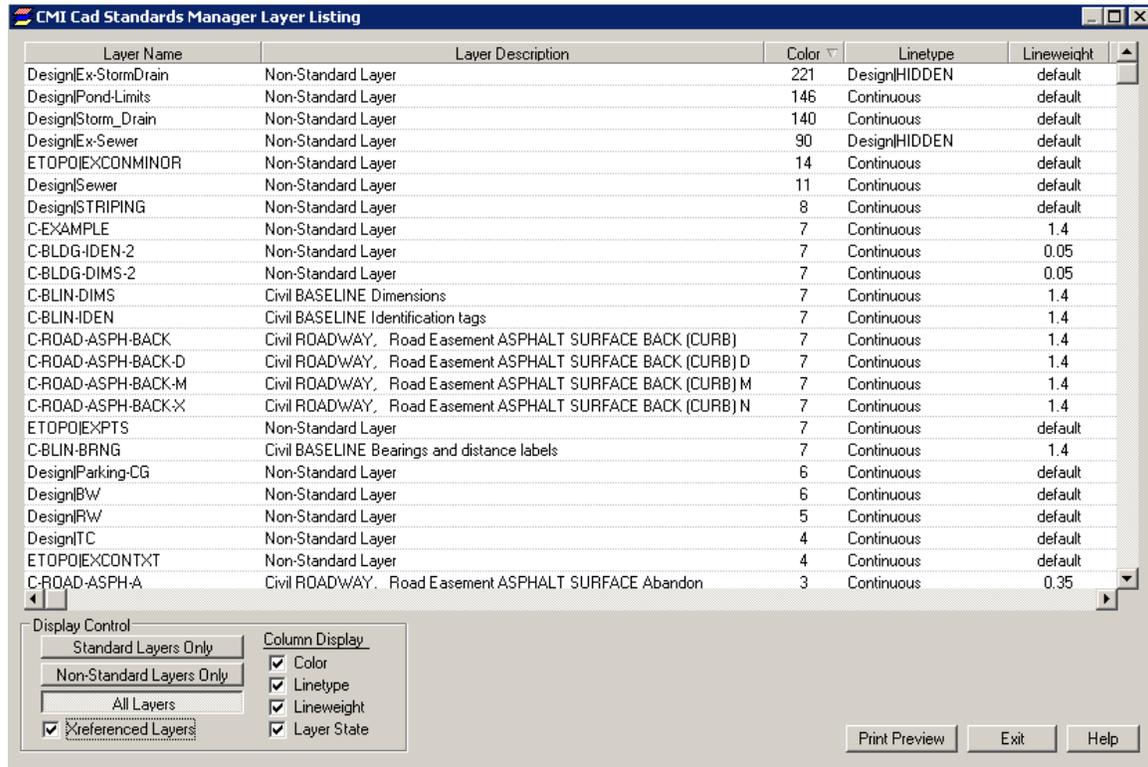
 Important note: The Layer ID command will identify individual Xreference entity layers using the Xref option.

Suggested Exercise:

1. Zoom window from 0,0 to 0, 500.
2. Create a few standard layers and draw AutoCAD entities on each layer.
3. Save the drawing to your training folder as prescribed by your instructor.
4. Start a new drawing and repeat steps 1 and 2 (you don't have to use the same layer names).
5. Xreference (overlay) the first drawing into the current drawing.
6. Run Layer ID and select entities (one by one that exists in the current drawing or in the Xreference). Note that the software identifies the layers equally well whether found in the current drawing or in the Xreferenced drawing.

List Layers

This command lists all layers in the current drawing and provides a facility to print the list in a table. To run the command, select **List Layers** from the **Stds Manager** menu, or select the **List Layers** button  from the *CMI Standards* toolbar. The command displays the *CSM Layer Listing* dialog (displayed below).



If a layer is defined in the current standard, then its full description will appear in the layer description column. If a layer is not in the current standard, it will have the description “Non-Standard Layer”.

The CMI Standards Manager Layer Listing dialog is resizable allowing you to set column and page widths for printing. Resizing the dialog vertically will not affect the printed output but horizontal resizing will adjust the width of all columns in the table. You can also resize individual columns by sliding the column divider bars (the lines separating column names) in the header at the top of the listing.

Display Control

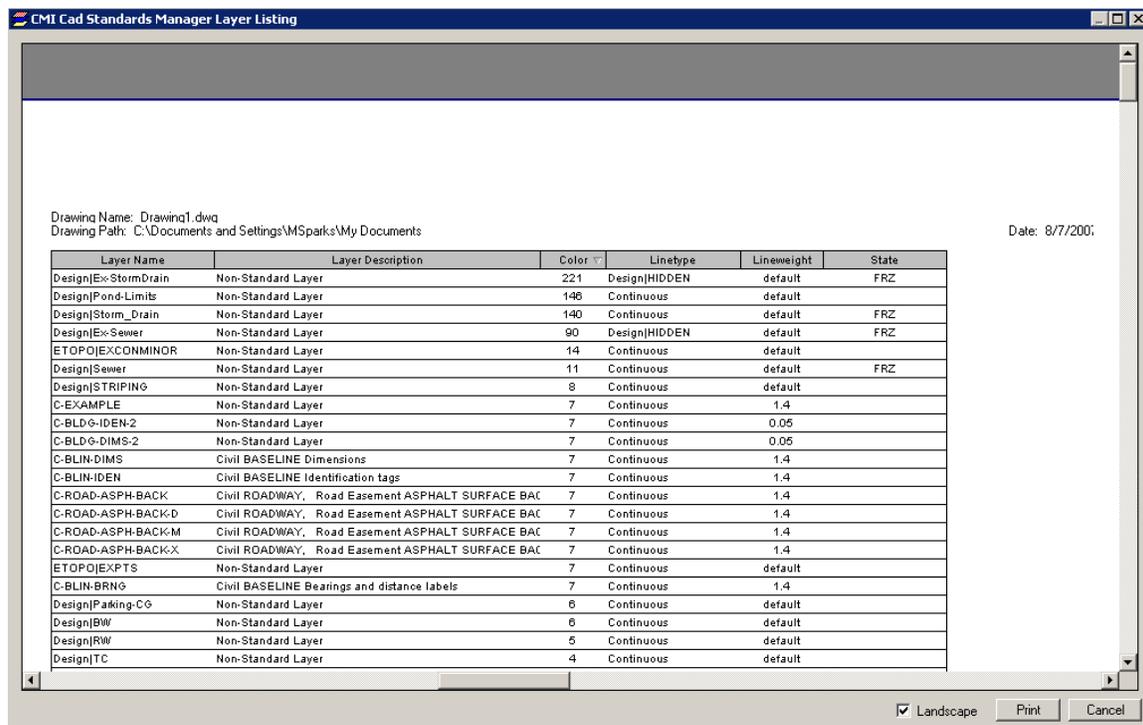
In this portion of the dialog you can select which layers to included in the listing as well as what layer information to display. Making a selection in this area immediately updates the table to display the desired information.

The radio buttons and check box to the left identify which layers will be shown in the listing. **Standard Layers** refers to layers that are defined in the current project's assigned standard. **Non-Standard** refers to any layer not in this definition. Selecting All Layers displays both standard and non-standard layers. Use the **Xreferenced Layers** check box to select whether Xreferenced layers are included in the list.

With the check boxes under Column Display you can choose to show layer Color, Linetype, Lineweight and State information in the table. The Layer State column indicates if a layer is OFF, FROZEN or LOCKED. If a layer in the listing is currently ON, THAWED, and UNLOCKED this field is blank.

Print Preview

The **Print Preview** button opens a dialog similar to the one shown below.



This dialog allows you to preview exactly how the layer listing will print before clicking the print button. At the top of the page the drawing name, path and date are shown. By default the page layout is portrait but can be changed to landscape by selecting the Landscape check box at the bottom of the dialog. In this example the listing is small and the column widths have been adjusted to fit on one page. If the layer list spans multiple pages a slider bar will appear on the right side of the dialog allowing you to view the additional pages.

The Print Preview dialog has two options for changing or zooming a view of the page. First the dialog is resizable. Making the dialog bigger expands the size of the page. Second you can use the mouse as a magnifying tool. Holding the mouse over the image

of the page displays a magnifying glass. If you are in full-page view (as in the figure above) the magnifying glass shows a plus symbol (+). Clicking with the left mouse button zooms in to a partial view of the page. The dialog will update to include horizontal and vertical slider bars allowing you to adjust your view. Once you are in the zoomed view the magnifying glass switches to a minus sign (“-“), clicking again with the mouse button zooms out to the full-page view.

Suggested Exercise:

1. Create some non-standard layers in the current drawing using AutoCAD commands or Layer Manager. Assign color, linetype, linewidth, and optionally plot style (for style based plotting) to the layers.
2. Create some standard layers using the **By Group** and **By Layer Category** methods. **Where are these layer creation options found?**
3. Now create and set current your organization’s defined Xreference insertion layer. **The instructor should provide or suggest this layer for the class.**
4. Xreference (overlay) the drawing created as part of an earlier exercise or any drawing available.
5. Locate and run the **List Layers** command within *CMI Standards* toolbar > *Layers* flyout.
6. Toggle the **Display Control** options and note the results of each selection.
7. Press the **Print Preview** button to open the preview dialog. Are all columns that are checked displayed on the first page? If not, **Cancel** from the preview window and “shrink” the CSM Layer Listing dialog by gripping it with your mouse. Now adjust column widths as need using your mouse. Press the **Print Preview** button again and determine if the print layout is now acceptable. Repeat until you are satisfied with the results.
8. Finally, if a printer is available, press the **Print** command from within the preview window.

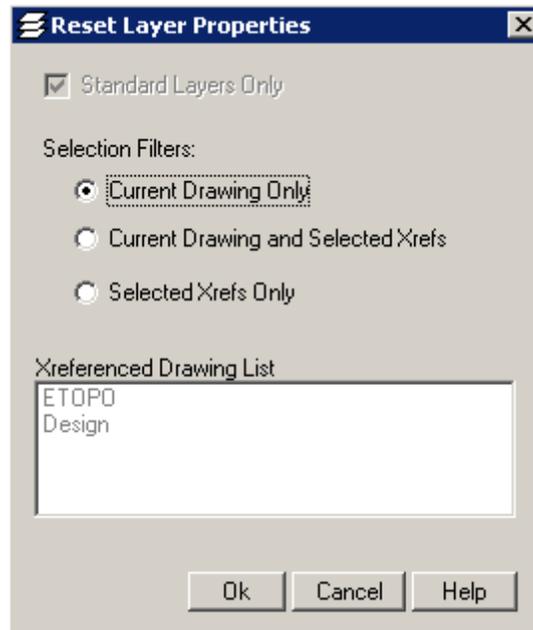


Important Fact: Most dialog boxes with *CMI Standards Manager User Module* remember their size, shape and screen position from the last use. Use this feature to set the *CSM Layer Listing* dialog size so that your printouts are adjusted properly for your desired print layout for each successive use of the command.

Reset Colors/Linetypes/Lineweights

This command resets the layer display properties of all standard layers found within the current drawing to their respective standard values. If you are using named plot styles then this command updates all layers to reflect the named styles for the currently set plot style table (.stb file). In this case, the software requires that the current default style table be referenced within the assigned standard. Your AutoCAD profile must be able to “see” the organization’s style tables (.stb files) and these style tables must be imported and assigned with the *CSM Administration Module*. If you have any problems please consult with your CAD manager regarding you AutoCAD setup in combination with the current standard database assigned to the drawing.

Run the command by selecting **Reset Colors/Linetypes/Lineweights** from the **Stds Manager** pull-down menu or by clicking on the **Reset Colors/Linetypes/Lineweights** button  on the *CMi Standards > Layers* flyout toolbar.



Suggested Exercise:

1. Start with the drawing from the last exercise or create some standard layers using the **By Group** and **By Layer Category** methods in the current drawing
2. Open AutoCAD’s Layer Manager and change the properties on a few standard layers.
3. Optionally, draw entities on the standard layers that were modified in the Layer Manager.
4. Execute the **Reset Colors/Linetypes/Lineweights** command and pick the **Current Drawing Only** option. Press Ok to reset the layer properties.
5. Did that layers change back to their original assigned properties?

6. Optionally, if your drawing contains Xreferenced files with standard layers, change some of the Xreference layer properties. Run the command again, this time choosing an Xreference option.

Additional functionality of the Reset Colors/Linetypes/Lineweights

Optionally, your CAD standards administrator can place the exact same layer name in more than one discipline. In this case, if the software determines that a layer name has more than one set of properties in the database (duplicate layers exist) then you will be prompted to select the controlling discipline. This dialog prompt is shown below.



An example use of this feature would be the ability to instantly “screen” layers for one discipline while another discipline “sees” the layers at full weight. Remember that CSM can “reset” Xreferenced layers just as easily as the layers in the current drawing.

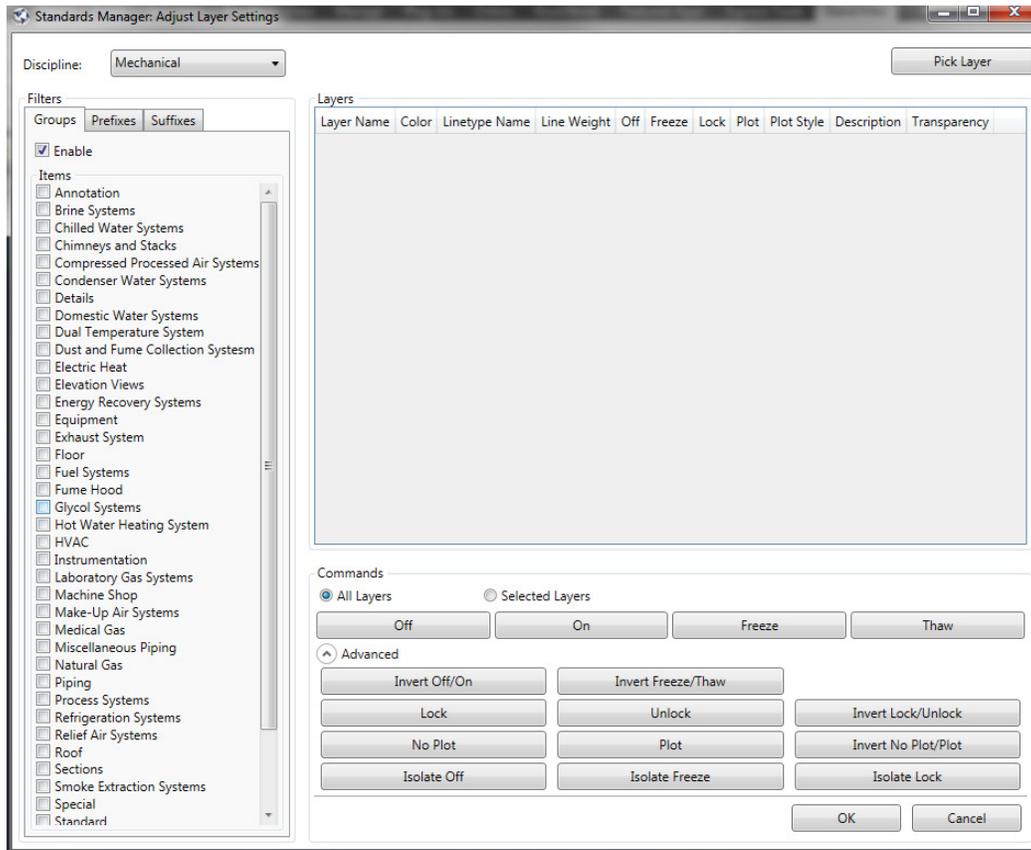
Suggested Exercise:

1. Create some standard layers that are used in more than one discipline (duplicate layers). **Your instructor will provide you with the layer names.**
2. Draw entities on these layers.
3. Execute the **Reset Colors/Linetypes/Lineweights** command and pick the **Current Drawing Only** option. Press Ok.
4. Once the *Reset Layer Properties* dialog opens, select a controlling discipline from the drop-down list.
5. Note the layer properties and then run the reset function once more and pick a different controlling discipline. Did the properties change?

Try this same exercise with a drawing containing Xreferenced duplicate layer names.

Layer Group Tools

The **Layer Group Tools** give you the ability to select and modify the status of entire groups of layers specific to a discipline. Once these layers are selected you can rapidly apply layer commands to the entire selection set such as On/Off, Freeze/Thaw etc. The **Layer Group Tools** dialog is shown below.



Execute the command from the Standards menu by first selecting **Layer Group Tools** or alternatively, pick the **Layer Group Tools** toolbar button  from the *CMI Standards > Layers* flyout toolbar.

First choose the appropriate discipline in the **Select Discipline** dropdown list. The central column lists all the layer groups in that discipline. If the discipline contains prefixes or suffixes the side columns will display these as in the example above. Each column has an **On/Off** check box to allow selection filtering. For example if you wanted to pick all layers with the prefix PH1, first enable the **Prefix Filter** column, select that prefix and disable the **Group Filter** and **Suffix Filter** columns. Using the Layer commands will affect all of the layers in the drawing with the prefix PH1. You can select multiple layers from one or all of the columns using windows standard selection keys or simply click and drag with the mouse.

Use the **Pick from Drawing** button to select layers from entities within the drawing. If you pick an entity that is **not** on a layer in the selected discipline in the **Layer Group Tools** dialog, a message similar to the following will display in the command line:

```
Layer "NG-SHT-TTL" is not a member of the current Discipline.
```

You may only select layers from the currently selected discipline to add those layers to the selection set in the **Layer Group Tools** dialog. The pick box remains active allowing you to select multiple layers. Return to the **Layer Group Tools** dialog by pressing *<Enter>* or clicking the mouse in a blank area of the drawing.

Once you have selected the desired layer groups use the **Execute Layer Command** buttons to apply a command to the entire selection set at once. Upon execution of any command the dialog closes and ends the **Layer Group Tools** function. The **Layer Group Tools** dialog will remember the selection set the next time you execute the function. This saves you from having to reselect a group of layers if you need to apply multiple commands.

Suggested Exercise:

1. Start with a drawing from a previous exercise or create a new drawing, add standard layers from at least two layer groups within the same discipline. Draw some entities on those layers. You may work in Modelspace or Paperspace.
2. Open the **Layer Group Tools** dialog. The discipline shown at the top should be the same discipline used to create the layer groups in step 1. If not, switch to that discipline.
3. Press the **< Pick from Drawing** button and select an entity drawn in step 1. The software should respond by telling you the group selected. Press *<Enter>* once you have an entity on a standard layer.
4. Note that the layer group is now highlighted in the **Group Filter** list.
5. Press the **Freeze** button and all layers in the selected group are immediately frozen.
6. Execute **Layer Group Tools** once more. Note that the interface remembers the last group you worked on (even with interceding commands).
7. Press the **Thaw** button and the layer group is immediately thawed.

Apply Plot Styles

The **Apply Plot Styles** function is useful for those companies that use named plot styles (.*stb* files) to control the look of their plotted drawings. The command opens the dialog shown below allowing you to select from a list of predefined plot styles in your organization's standard(s). The **Apply Named Styles to Layers** button attaches that .*stb* file to your drawing. You can update the appearance of the drawings layers to reflect the styles defined within the file by using the **Reset Colors/Linetypes/Lineweights** command. Please consult your CAD Manager with regards to the plot style tables in use within your organization.

Save Layer Settings

The **Save Layer Settings** command saves all layer states from the current drawing. A text file is generated containing layer names, colors, linetype, and layer states (on/off, freeze/thaw, locked/unlocked). Execute the command from the **Standards** menu by first selecting the cascading menu item **Save and Restore Layers**, and then **Save Layer Settings**. Alternatively, pick the **Save Layer Settings** button  on the *CMI Standards > Layers* flyout toolbar.

The command opens a standard file selection/entry dialog box. By default, the current folder is set to the same folder location as the current drawing. Enter a file name (without file extension) or select an existing file. The mandatory file extension “.*gbl*” is automatically appended to any new file name entered. After specifying a file location and name press the **Save** button, the following statement is displayed on the command line after execution.

```
Global layer settings saved to <filename>.
```

Restore Layer Settings

The **Restore Layer Settings** command takes a previously saved global layer states file (“.*gbl*” file extension) and restores the current drawing's layer states to those saved in the file. Execute the command from the Standards menu by first selecting the cascading menu item **Save and Restore Layers**, and then **Restore Layer Settings**. Alternatively, pick the **Restore Layer Settings** toolbar button  from the *CMI Standards > Layers* flyout toolbar.

The command opens a file dialog for the selection of the desired global layer states (“*.*gbl*”) file. By default, the contents of the current drawing folder is displayed. After selecting a file press the **Open** button located in the lower right corner of the dialog. The following command line prompts will appear:

```
Restore linetypes and colors? No/<Yes>: <Enter>
```

```
Freeze layers not found in layer status file? No/<Yes>:  
<Enter>
```

Press the <Enter> key to accept the default responses.

Suggested Exercise:

1. Start with the drawing from the last exercise or any other drawing.
2. Freeze some layers and turn off other layers.
3. Execute the **Save Layer Settings** command and without changing folders, enter a file name for the global layer states file.
4. Thaw and turn on the layers previously frozen.
5. Execute the **Restore Layer Settings** command and select the file that was just created with the “save” function. Respond affirmatively to any prompts.
6. Determine if the “restore” function reset the layer states as set in step 2 of this exercise.
7. Try the command again with a drawing containing Xreferences. Are the results consistent?

Save VP Layer Settings

Within AutoCAD’s Paperspace (Layout) environment, individual layers can be frozen in one viewport (VP) and thawed in another, allowing different views of Modelspace entities to be displayed on one layout. The **Save VP Layer Settings** command saves the freeze/thaw settings for a selected viewport. Execute the command from the **Standards** menu by first selecting the cascading menu item **Save and Restore Layers**, and then **Save VP Layer Settings**. Alternatively, pick the **Save VP Layer Settings** toolbar button  from the *CMI Standards > Layers* flyout toolbar.

The command opens a standard file selection/entry dialog box. By default, the current folder is set to the same folder location as the current drawing. Enter a file name (without file extension) or select an existing file. The mandatory file extension “.vpt” is automatically appended to any new file name entered. After specifying a file location and name and press the **Save** button found in the lower right corner of the dialog. Select the desired viewport edge when prompted as follows:

```
Select Paperspace viewport: (select the edge of desired viewport)  
Viewport layer settings saved to <"filename ">.
```

Any layers frozen (“Current viewport freeze” set) within the selected viewport are written to the file.

 Please note: The **Save VP Layer Settings** command can only be run in Paperspace (tilemode = 0).

Restore VP Layer Settings

The Restore VP Layer Settings command restores the freeze/thaw settings of layers in a selected viewport from a previously saved VPT file. Execute the command from the Standards menu by first selecting the cascading menu item **Save and Restore Layers**, and then **Restore VP Layer Settings**. Alternatively, pick the **Restore VP Layer Settings** toolbar button  from the *CMI Standards > Layers* flyout toolbar.

The command begins by opening a file dialog for the selection of the desired viewport layer states (“*.vpt”) file. By default, the content of the current drawing folder is displayed. After selecting a file press the **Open** button located in the lower right corner of the dialog. The following command line prompt will appear:

Select Paperspace viewport: (select the edge of desired viewport)

The “freeze in active viewport” setting will be set for those layers stored in the selected .vpt file.

 Please note: The **Restore VP Layer Settings** command can only be run in PaperSpace (tilemode = 0).

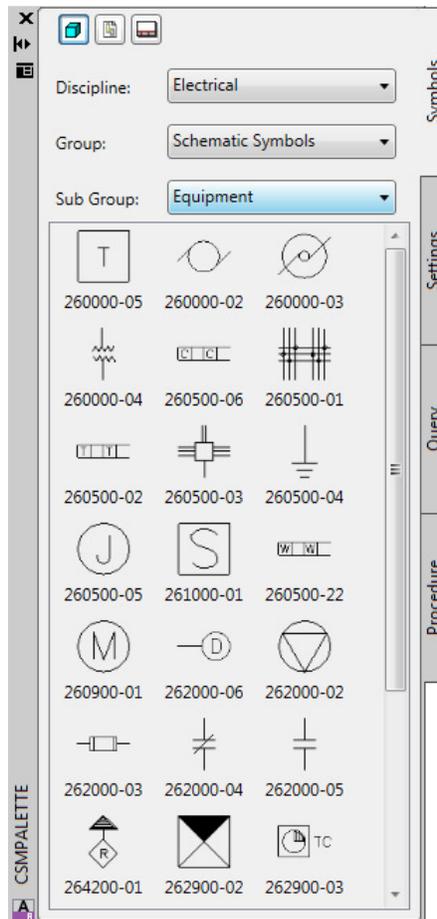
Suggested Exercise:

1. Start with a drawing from a previous exercise or any other drawing containing entities.
2. Switch to a layout tab and create a rectangular viewport if one does not already exist. Double-click inside the viewport to switch to Modelspace.
3. Create some entities (LINEs, ARCs, TEXT, etc.) on a few different standard layers. Write down the layers used.
4. Open AutoCAD’s Layer Manager and in the VP Freeze column, click to freeze the layers you wrote down. Press **Ok**, to apply the status change.
5. Execute the **Save VP Layer Settings** command and enter a file name of your choice. When prompted pick the edge of the viewport.
6. Now, from within Paperspace create another rectangular viewport. Do not copy the first viewport.
7. Execute the **Restore VP Layer Settings** command and select the file that was just created with the “save” function. Pick the edge of the new viewport when prompted. Did the layer states change within the new viewport?

Symbols

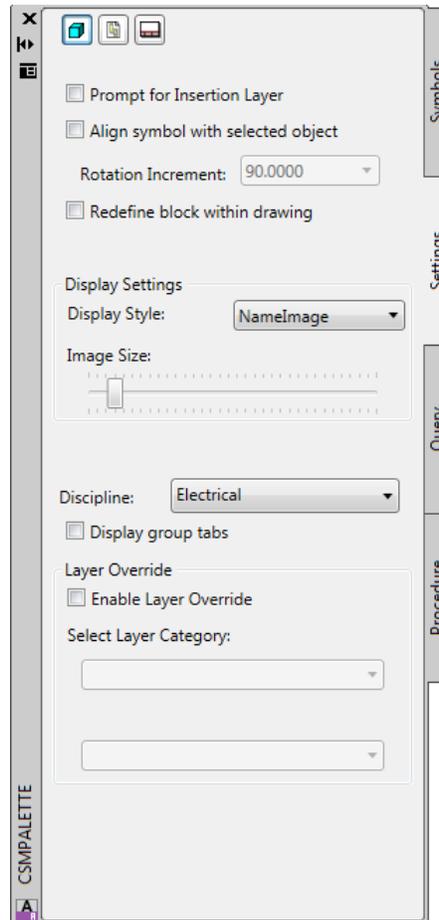
Symbols, by typical definition, are blocks (whose entities are created on layer “0”) that represent a physical object or design element. They may appear multiple times in one drawing and may sometimes be used on multiple layers. By creating the block’s entities on layer “0” they then take on the properties of the insertion layer (color, lineweight, linetype, and layer states). Symbols are typically scaled if they are a symbolic representation of a physical object and/or they contain text attributes that need to vary in size according the final scaled output. Symbols can also be defined for physical objects that are not simply symbolic, but are drafted at the actual size of the object. The *CFI Standards Manager* provides for both cases within the *Administration module* (via a scale symbol toggle) in combination with the visual interface for CAD users where a drawing scale can be entered. The symbol insertion dialog displays an image of the symbols, helping you quickly select the correct symbol for insertion.

To access the symbol insertion dialog, run the **Open CSM Palettes** command from the **Stds Manager** pull-down menu, or click on the **Symbols** button  on the *CFI Standards > Insert* toolbar flyout. The *CFI Standards Manager – Symbols* dialog is shown below.



The dialog is resizable and image size can be adjusted. Use the Settings tab to set the desired image display size and type as described in **Symbol Insertion & Display Options**.

Symbol Settings Options



With the **Prompt for insertion layer** check box marked you are prompted during the process of symbol insertion as follows:

```
Set current layer (Select entity/from List/<layer>):
```

The standard layer as defined for the symbol is the default layer selection and will be shown in the <layer> field above. Press <Enter> to set this layer current. Optionally, you may type an existing layer name at this prompt. Typing “s” allows you to select an entity from the drawing to establish the insertion layer. Alternatively, typing “L” brings up a layer list dialog displaying all the layers within the current drawing.

Select a layer name and press “Ok” or simply double clicking a layer will set that layer for detail insertion. The active layer in the drawing remains active and is not affected by the insertion process.

The **Align symbol with selected object** option allows you to rotate/align a symbol to a selected **Line** or **Arc**. An object snap (OSNAP) **must** be used to select the entity. The symbol will align itself tangent or perpendicular to the object at the selected point along the object. The initial symbol rotation will depend on how the symbol was drawn and saved. If the alignment is not correct you may use a left mouse click to align the symbol at chosen increments. Moving the mouse around the point of insertion snaps a reference line to the nearest selected increment, click the left mouse button to re-align the symbol to the reference line. Once the symbol rotation is as desired, press <Enter> to complete the insertion process.

The **Explode (to Standard Layer)** option causes the symbol to be exploded to its base AutoCAD entities, but alternative to the standard AutoCAD command it places these entities onto the assigned/selected insertion layer.

 The **Symbols - Insertion Options** settings are maintained from drawing session to drawing session.

Symbol Insertion Parameters

With no insertion options set within the *Symbols Options* dialog the following prompts will appear in the command line.

Select Insertion Point: *(pick insertion point)*

Pick an insertion point using the left mouse button or enter the coordinates directly at the command line.

Symbol rotation angle <0>: *(enter value or select a direction)*

Pick a rotation angle using the left mouse button or enter the angle directly at the command line.

With **Prompt for insertion layer** set in the *Symbols Settings Tab* dialog you will first be prompted to select the desired layer. Please refer to the Symbol Insertion & Display Options section for a complete description of the prompt. Once the layer has been selected you will return to the **insertion point** and **rotation angle** prompts above.

With **Align symbol with selected object** set in the *Symbols Settings Tab* dialog you will first be prompted to choose an insertion point. You may align the symbol with a line or arc and must use object snaps to pick the point of insertion on the desired object. Once you have chosen the point of insertion you will be prompted on the AutoCAD command line:

At this point the screen crosshairs align themselves to the selected object (tangent or normal to the object) and only angles in 90-degree increments from this initial alignment can be selected on screen.

☞ If you pick a point that is not on a line or arc the symbol will insert normally without the alignment or rotation restriction.

Additional Features

Depending on your standard setup, as defined in the *CMI Standards Administration* module, the symbol insertion process may allow you to select alternative category names within one layer category in a multiple category layering standard. Prior to inserting the symbol you would be prompted to select from a list of available layer category names in the command line. A typical use of this feature is to allow a change in the status of a symbol to be inserted (e.g. category /descriptions such as Proposed, Existing, or As-Built as shown in the figure below).

Command: Symbol Status category ... Existing/As-built/Proposed:

In this example the category name is **Status** and the available options are **Proposed**, **Existing**, and **As Built** (the default setting). Once the selection is made the symbol will be inserted on the proper layer. For example, the layer “AB-CO-SBE” for “As-Built Coax Box, Electrical” versus the layer “EX-CO-SBE” for “Existing Coax Box, Electrical”

Suggested Exercise:

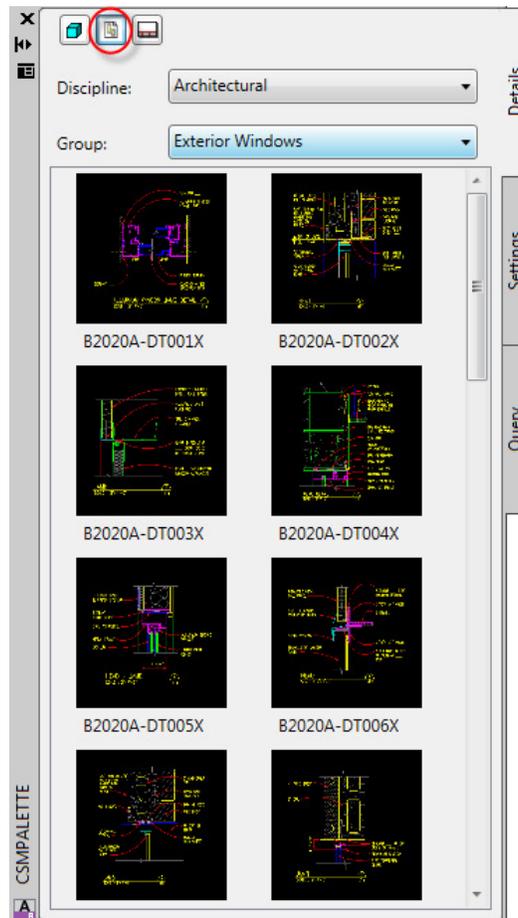
1. Start with a drawing from a previous exercise or create a new drawing and make sure to you are linked to the proper Standard.
2. Open the CSM Symbols palette (**Insert Symbols**). You may be prompted for drawing units and scale if these have not been previously set.
3. Try resizing the dialog and adjusting the *Display Options...*
4. Make sure the *Insertion Options* are all unchecked. Press **Ok** to close the *Symbol Options* dialog.
5. Explore what **Disciplines** and **Groups** are available in your Standard.
6. Pick (click) a symbol image or name and the press the **Insert >** button.
7. Pick the insertion point and rotation for the symbol.
8. Run *CSM's Layer ID* command pick the symbol you just inserted. Note if the layer used for the symbol seems appropriate. Your CAD Manager has assigned the layer in the *CSM Administrative module*.
9. Next, draw a LINE and an ARC on any layer.
10. Re-open the *CSM – Symbols* dialog and then press the **Options** button. Enable (check) the **Align symbol with selected object** option and press **Ok** to exit to the main dialog.
11. Double-click on a symbol image you wish to insert and then using a “nearest” osnap pick the line or arc you drew previous. Note the rotation angle prompt

and you crosshair orientation. Move your crosshairs around the insertion point and note the 90-degree angle increments. Finally, select a desired angle and the press <Enter>.

12. Repeat the insertion process trying out the additional insertion options.

Details

Details, by typical definition, are plans, elevations, or sections that provide more specific information about a portion of a project. To access the detail insertion dialog, run the **Insert/Xref Details** command from the **Stds Manager** pull-down menu, or click on the **Details** button  found on the *CFI Standards > Insert* toolbar flyout. The *CFI Standards Manager – Details* dialog is shown below.



The first step when inserting a detail is to select the proper discipline in the **Discipline Name** drop-down list. All the groups defined within that discipline appear in the **Group Name** drop-down list. Secondly, select the proper group where the desired symbol can be found. You may now select a detail by clicking on the detail name in the **Drawing Name** list or the thumbnail image.

Unless you have chosen the **Prompt for insertion layer** option, detail insertion/referencing will occur on the one of two layers. If your Standard defines a default layer for the selected detail it will insert on that layer. If that layer is not in the drawing it will be created at the time of insertion. This process does not affect the active layer. If there is no default layer specified in your Standard the detail will be inserted or xreferenced on the currently active layer.

Before inserting or attaching the detail select the desired method of insertion in the lower left hand corner of the dialog. Your choices are as a **Block**, **Xref Overlay**, or **Xref Attachment**. Choosing **Block** will insert the detail as a single entity. When **Block** is selected an **Explode** check box will appear at the bottom of the dialog. This option explodes the detail upon insertion into its individual AutoCAD entities.

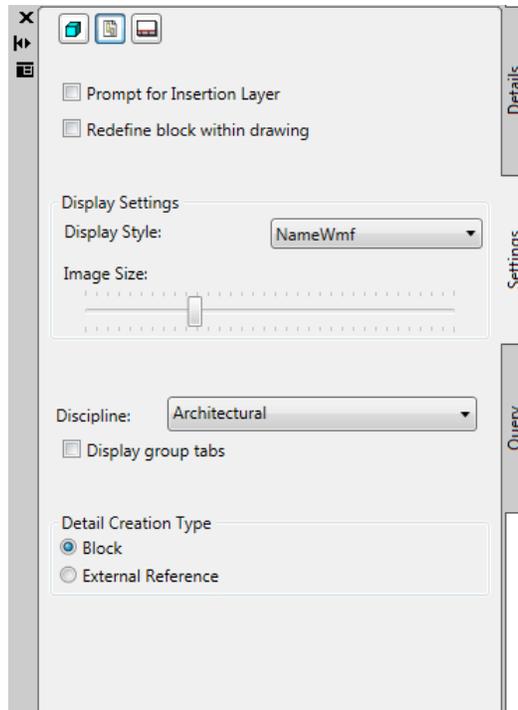
Selecting **Xref Overlay/Attachment** will place the detail as an external reference. The Xref will be attached in one of three ways: To the default layer, the current layer or the selected layer if **Prompt for insertion layer** is active. A copy of the detail drawing will automatically be saved in the current project folder structure. This will allow you to make changes to the drawing without changing the original in your standards. Additionally it is advantageous to include the drawing in your project for archiving purposes. If a CAD Standards folder was specified as part of the product installation and that folder exists within the current project folder structure, the detail will be placed beneath that folder in a subfolder indicating its originating discipline and group. If the CAD Standards folder cannot be found in the project folder structure the detail will be copied into the same folder as the current drawing file and then attached or overlaid as indicated.

In general, details are placed in a drawing on a one-to-one scale. You can optionally place the detail at some scale multiplier by entering a **Scale Factor** in the edit box at the bottom of the dialog.

Click the **Insert/Xref** button to complete the insertion process. You will be prompted to select the **Detail insertion point** in the command line. You may enter the coordinates directly in the command line or choose the insertion point with a left mouse click.

Detail Insertion Settings

The only available option when inserting details is **Prompt for insertion layer** as shown in the figure below.



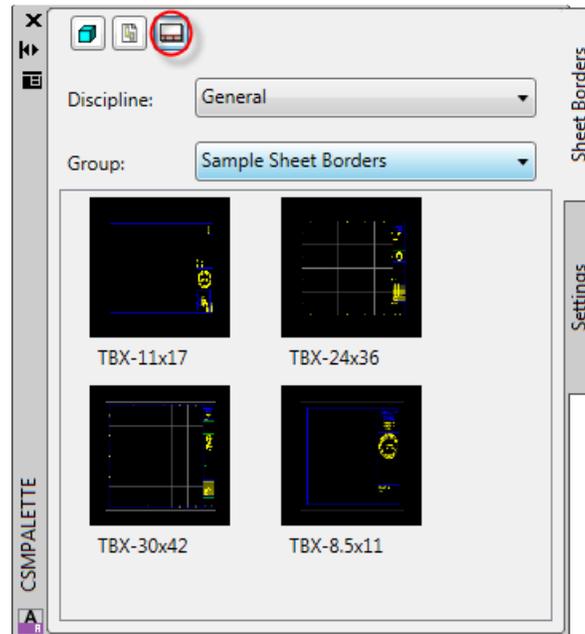
The Layer Selection Process is described in **Details** section of this manual.

Suggested Exercise:

1. Start with a drawing from a previous exercise or create a new drawing and make sure to you are linked to the proper Standard.
2. Open the CSM Details palette (**Insert/Xref Details**).
3. Try resizing the dialog and adjusting the *Display Options...*
4. Make sure the *Insertion Options* are all unchecked. Press **Ok** to close the *Details Options* dialog.
5. Explore what **Disciplines** and **Groups** contain details in your Standard.
6. Select the **Insert as: Block** option in the lower left-hand corner of the dialog.
7. Pick (click) a detail image or name from the list and the press the **Insert/Xref** button.
8. Pick the insertion point for the detail in the drawing.
9. Open the Details palette once more and this time “check” the **Explode** toggle and then insert the Detail. Verify that the detail has been exploded.
10. Run the AutoCAD UNDO command until the details are removed from the drawing.
11. Open the Details palette and select **Insert as: Overlay**. Place a detail in the drawing.
12. Run the AutoCAD XREF command to open *File References* dialog and check the “Saved Path” of the detail just added to the drawing. Please refer to the paragraph on the previous page regarding the how details are copied to a project folder structure.

Sheet Borders

Sheet Borders (title blocks) can be accessed from your standard via the **Insert/Xref Sheet Borders** command from the **Stds Manager** pull-down menu, or click on the **Sheet Borders** button  found on the *CMI Standards > Insert* toolbar flyout. The *CMI Standards Manager – Sheet Borders* dialog is shown below.



Sheet Borders can be inserted, overlaid, or attached. Your CAD Manager controls this setting within *the CSM Administration module*.

Once the dialog is open select the desired discipline in the **Discipline Name** drop-down list. All the groups defined within that discipline appear in the **Group Name** drop-down list. Secondly, select the proper group where the desired sheet border can be found. You may now select a sheet border by clicking on the name in the **Drawing Name** list or the thumbnail image.

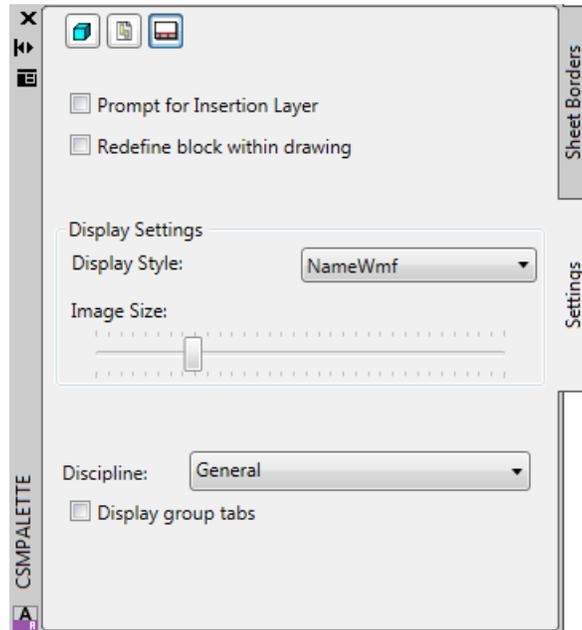
Finally, click the **Insert/Xref** button to complete the insertion process. If you have “checked” the **Prompt for insertion layer** option you will be prompted for the insertion layer. If not, you will be immediately prompted for the insertion point and the default layer set in the database will be used. By default this is 0,0 as shown here:

```
Border insertion point <0,0>: <Enter>
```

You may press *<Enter>* to accept the default, enter the coordinates directly in the command line or choose the insertion point with the mouse.

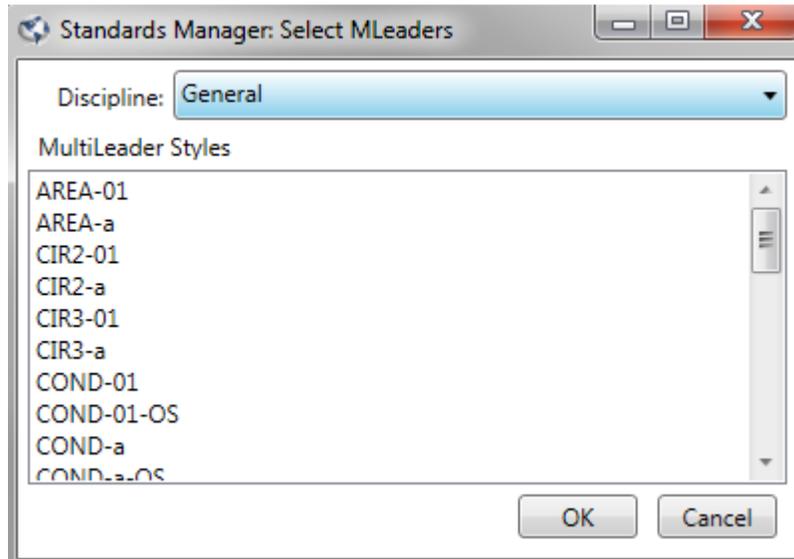
Sheet Border Insertion Settings

The Sheet Borders - Insertion options are shown below. The **Prompt for insertion layer** option is identical to the function described in *Symbol Options*.



MultiLeader Styles

MultiLeader styles can be accessed from your standard via the **Load MultiLeader Style** command from the **Stds Manager** pull-down menu, or click on the **Load MultiLeader Style** button  found on the *CMI Standards Manager (CSM)* toolbar. The *CSM Multileader Styles* dialog is shown below.

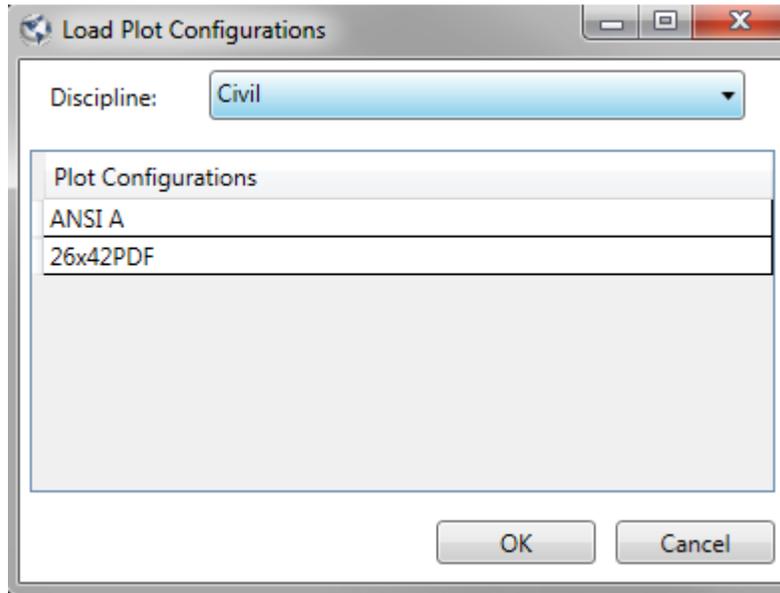


Choose the desired Discipline from the drop down list, and then select the desired multileader style(s) to load into the drawing.

 The last style selected becomes the current multileader style of the current drawing.

Page Setups

Page setups can be accessed from your standard via the **Load Page Setup** command from the **Stds Manager** pull-down menu, or click on the **Load Page Setup** button  found on the *CMI Standards Manager (CSM)* toolbar. The *Plot Configurations* dialog is shown below.



Choose the desired Discipline from the drop down list, and then select the desired page setup(s) to load into the drawing.

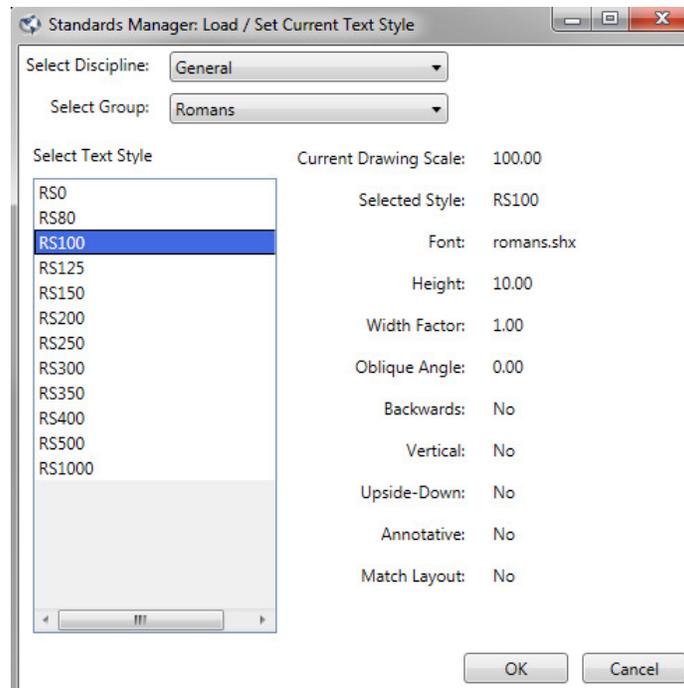
Text Styles

Text style standards ensure that text entities will have a consistent appearance from drawing to drawing. The *CMI Standards Manager* automatically adjusts text height when inserting text in Modelspace based on the current scale of the drawing. For example, if create a standard text style with a defined height of 0.1" in a drawing with the current drawing scale set to 40 (i.e. 1" = 40') *CMI Standards Manager* would multiply the standard text height by the drawing scale to create the style at the height of 4.0 AutoCAD units. This would ensure that the text appears at the proper height in the drawing in Modelspace. When inserting text in Paperspace a scale of 1 should be used regardless of the scale set in Modelspace. If you were to insert text with a height of 0.1" while in Paperspace the height would remain 0.1 AutoCAD units.

Inserting a Text Style

Before running the Text Style command be sure that the drawing scale is set properly with the **Set Current Drawing Scale** command. The drawing scale will be used in the creation of text in Modelspace (and Modelspace viewports) to ensure proper text height.

To execute the command choose **Select Text Style** from the CSM pull-down, or click on the **Select Text Style** button  in the toolbar. This will open the **Load / Set Current Text Style** dialog shown below. For reference the current drawing scale is displayed just below the group name field.



The first step in selecting a text style is to choose the appropriate discipline and group from their respective drop-down lists. All available text styles defined for that discipline

and group are displayed in the **Select Text Style** list. Left mouse click on a name to select a style. The attributes for the selected style are displayed to the right of the name list.

Once the proper style is selected click the **Load/Set Current** button. This will load the selected text and set it as the current style in the drawing. To return to the drawing click **Exit**, alternatively, double clicking on the desired text style will load, set it current and exit the dialog. You can also load multiple styles at once either by selecting a style one at a time and clicking **Load/Set Current** or highlighting multiple styles in the list before clicking **Load/Set Current**. You can select multiple styles using standard windows techniques (i.e. <Shift> or <Ctrl> keys).

☞ Once you have loaded a standard text style *CMI Standards Manager* will automatically adjust its text height as you move from Modelspace to Paperspace based on the scale of the drawing.

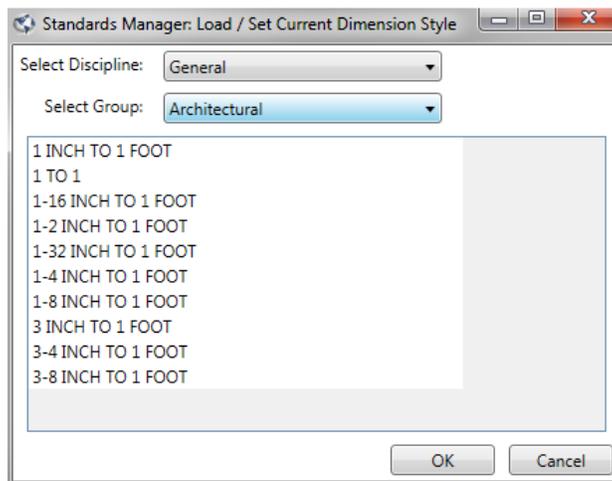
Suggested Exercise:

1. Start with a drawing from a previous exercise or create a new drawing and make sure to you are linked to the proper Standard.
2. Make the “Model” tab active and **Run Set Current Drawing Scale** and enter a desired scale.
3. Open the *Load/Set Current Text Style* dialog (**Select Text Style** command).
4. Load a single Text Style while noting its height in the right side of the dialog.
5. Run the AutoCAD STYLE command and verify the that Text Style you chose is set current and the height matches what you noted in the *CSM* dialog.
6. Switch to Paperspace and run the AutoCAD STYLE command again. What is the text height now?
7. Run the AutoCAD MVIEW command and create a viewport. Set the scale of the viewport to a different scale than you set earlier in this exercise.
8. Switch to Paperspace and then back into the Modelspace viewport. Check the text height each time you switch “spaces.”

Dimension Styles

Just as with standard text, dimension style standards ensure that dimensioned drawings will have a consistent appearance from drawing to drawing. The *CMI Standards Manager* allows you to quickly select and insert dimension styles from your company standards database.

To execute the command choose **Load Dimension Style** in the Standards menu or click on the **Load Dimensions** button  on the Standards toolbar. This will activate the **Load Dimension Style** dialog shown below.



The first step in selecting a dimension style is to choose the appropriate discipline and group from their respective drop-down lists. All available dimension styles defined for that discipline and group are displayed in the **Select Dimension Style** list. Left click on a name(s) to select a style. Click the **Load/Set Current** button to load the style, set it current and close the dialog.

Standards Compliance Tools

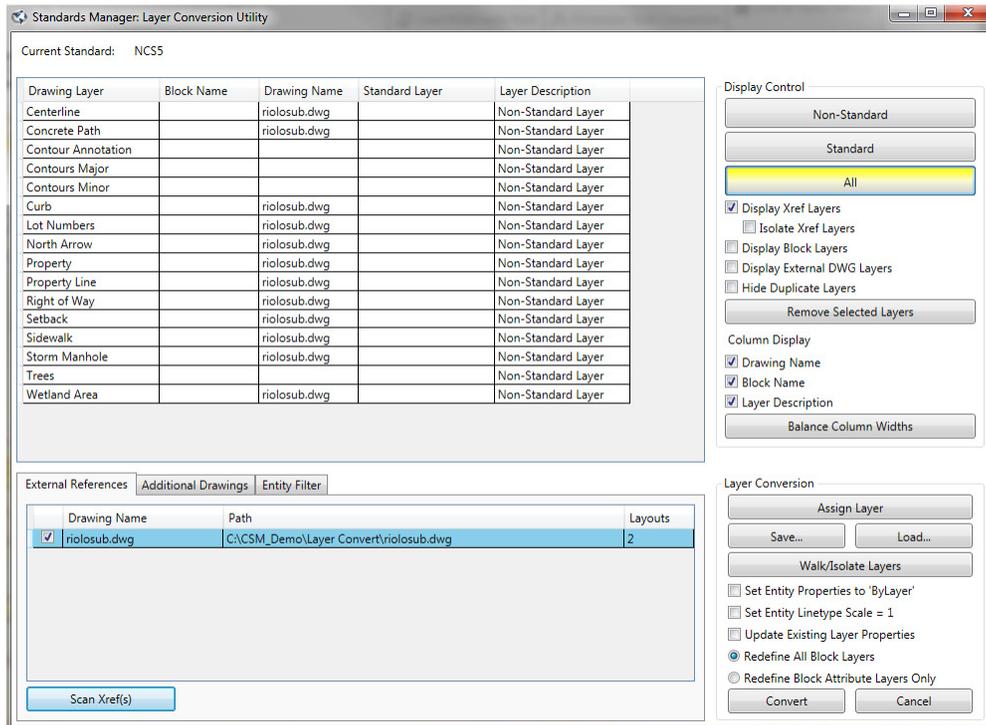
CSM Standards has a complete set of compliance tools to help the transition into standards, convert client drawing into the company standard, or modifying existing drawing that may have been done incorrectly in adherence to the company standard. It has 3 separate tools: (1) Layer Conversion, (2) Text Style Conversion, and (3) Dimension Style Conversion.

Layer Conversion

The **Layer Conversion** function allows you to rapidly convert non-standard layers in a drawing to standard layers. This function is useful, for example, in converting drawings from some outside source that may not be drawn in accordance with the appropriate standard or converting old drawings within your company to new standards. Run the command from the Standards menu by first selecting **Layer Conversion** or alternatively, pick the **Layer Conversion** toolbar button  from the *CSM Convert* toolbar.

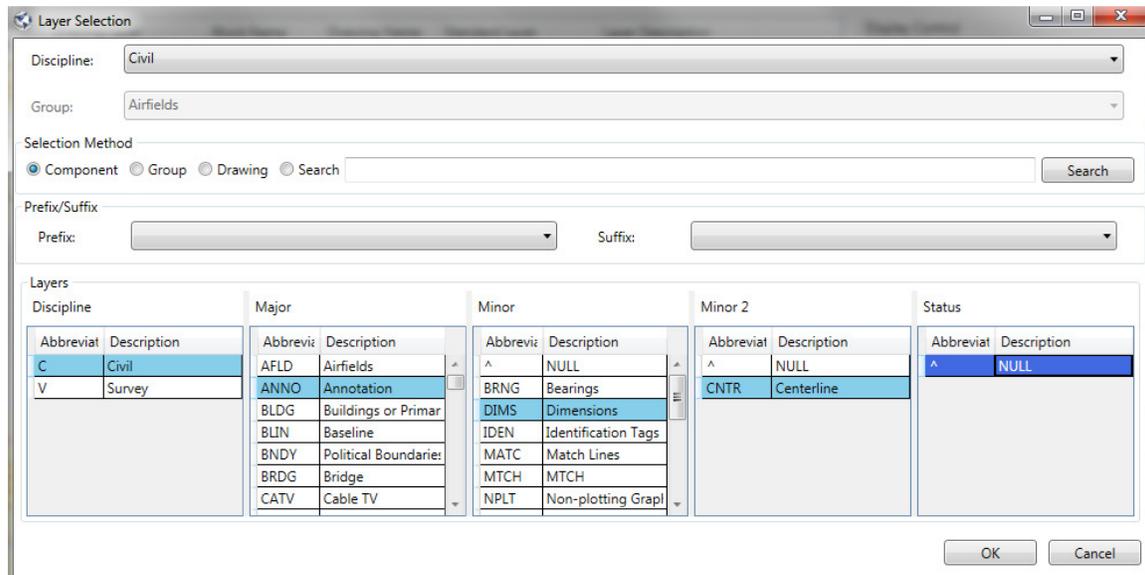
The layer conversion utility can be a very powerful tool to assist you in converting drawing to the company standard. The command allows the user to change and update attached X References, as well as multiple drawings at the same time. The batch processing allows entire projects to be converted very quickly and efficiently.

The dialog displays all layers in the drawing in the first column along with their description. The **Standard Layer** column is used during the assignment process to display the new standard layer selected to replace the old layer prior to conversion.



The process of conversion starts by selecting layers in the list. You can select multiple layers using windows standard selection keys or simply click and drag with the mouse. To make the selection process easier, use the Display Control section of the dialog box to filter the layer list.

Once you have selected the layer(s) that need conversion click the **Assign Layer...** button to open the layer selection dialog shown below.



To make the assignment, first select the appropriate discipline in the **Discipline Name** drop-down list. Then select the desired component from each category. Only one component can be selected in a category. Click **Ok** to return to the **Layer Convert** dialog. The selected standard layer will be displayed in the **Standard Layer** column. You may continue the selection and assignment process until all the necessary assignments are made. If you have a number of drawings that contain the same layers needing conversion, **prior** to converting use the **Save Layer Conversion** feature described below. Once the correct standard layers are assigned click the **Convert** button to complete the layer conversion process in the drawing.

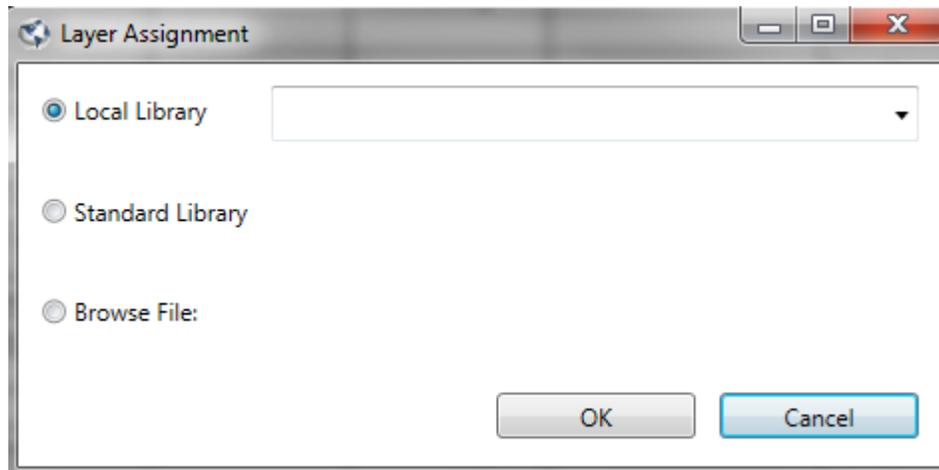
The **Walk Layers** button allows you to visually identify which entities reside on a given layer. Select the layer or layers that need clarification and click **Walk Layers**. This will take you to the drawing and walk through the selected layers isolating the entities one layer at a time. You are given the option in the command line to view the next layer or return to the dialog to make the necessary assignments before walking to the next layer. A typical example is shown below:

```
Isolate Layer = EP-GS-LT. Return to dialog / <Next>:
```

This process repeats until you have viewed all selected layers in the list or until you type "r" on the command line to return to the dialog.

The **Isolate Layers** button allows you to view only those layer which are highlight to quickly identify the layer contents to make it easier to determine the conversion layer.

The **Save** button is useful if you have a number of drawings with the same layers that need conversion. Once you have made all the necessary layer assignments as described above and **prior** to making the conversion you can save the non-standard to standard layer assignments by clicking **Save**. This opens the dialog shown below:



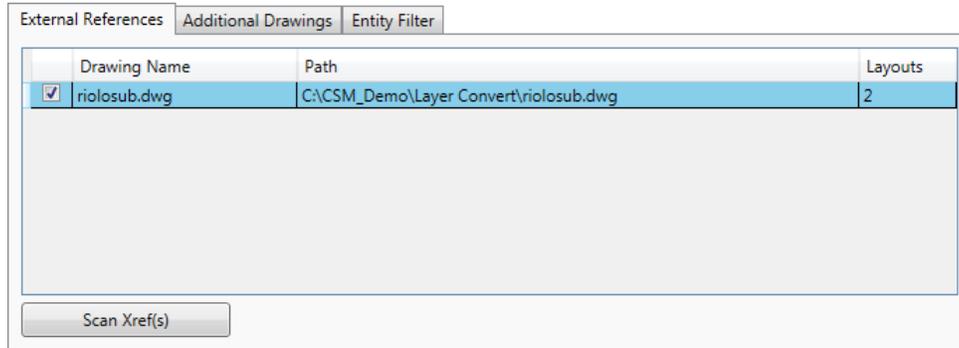
You have the option of saving the conversion assignments to the database or to a text file.

 If your installation of CSM is set to use local database caching then please use only the **Save to Text File** option. Please ask your CAD Manager if you are unsure.

The **Retrieve** button opens a similar dialog allowing you to select from the list of saved conversions or even browse for a previously saved text file. This feature allows for rapid conversion of layers without selecting them manually in the list each time and is especially useful for converting a large number of drawings containing the same layers.

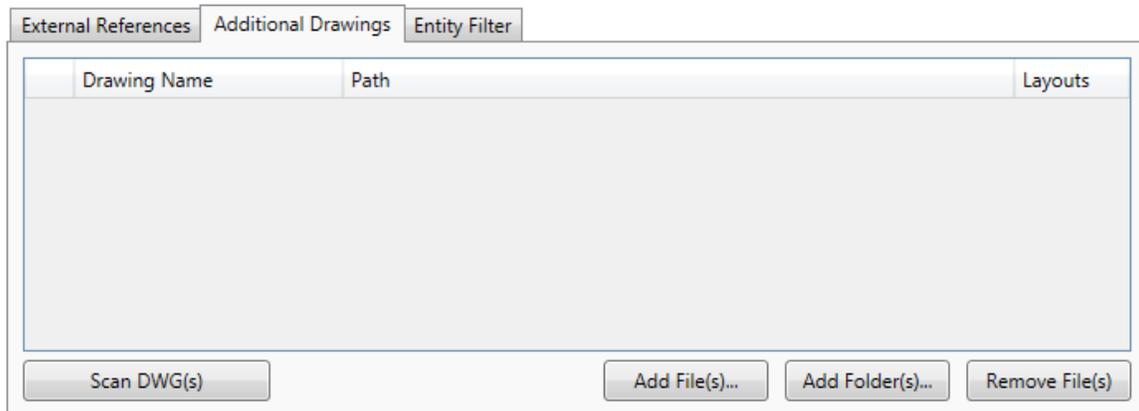
Convert X-Reference Layers

To convert X-Reference layers in the conversion process, simply check the box next to the desired X-Reference(s) name and then click **Scan Xref(s)**. This will add the layers from the selected Xrefs to the layer list. This allows the user to convert these layers as well.



Convert Multiple Drawings (batch process)

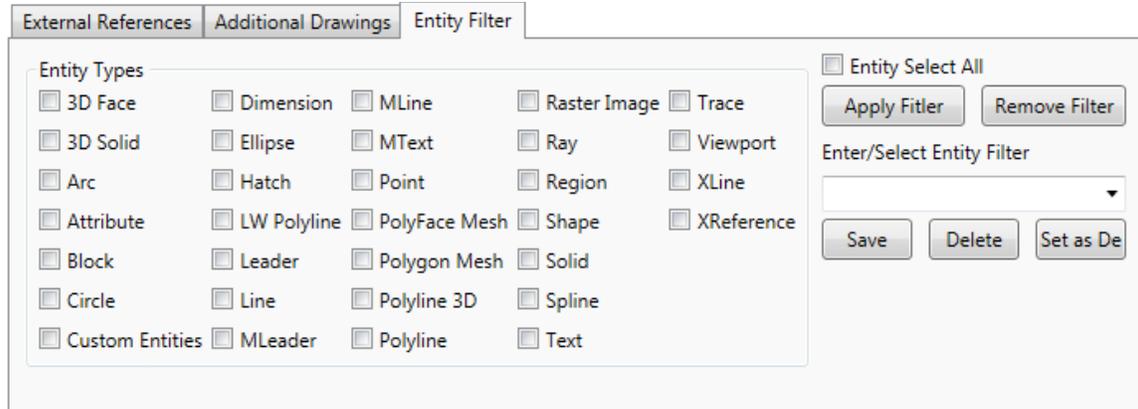
To perform a batch layer convert use the drawings tab located at the bottom of the main dialog box. Add individual files and/or folders to search for layers. Remove individual files as well. Then use the **Scan Dwgs** button to search through the entire drawing list for layers. An excellent filter choice would be to *Remove Duplicate Layers*; therefore if the same layer name exists in multiple drawing, only one will appear.



Due to the limitation of the software, batch processing a large amount of files at the same time, AutoCAD can run out of memory. If AutoCAD crashes, Standards will remember which drawing was saved and modified last so it can pick up where it left off before crashing.

Entity Filter

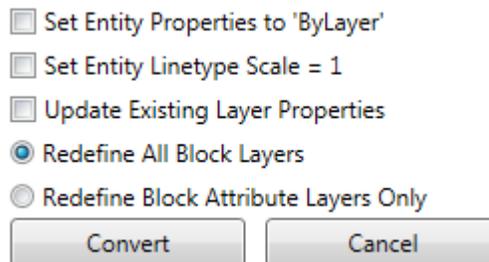
The tab allows the user to filter by entity type. This function only can work within the current drawing and not any attached X-References.



For example, if a user is looking for only text entities, they could easily filter for those entities. If this search becomes command, then saving the filter could become useful.

Miscellaneous Functions

When finally ready to process the layers to convert, this function has a few items to redefine the layers in the drawing. These are located just above the **Convert Layers** button.



Layers can be redefined inside the block definition as well as the attribute layers themselves. This can help you save time during the conversion process.

Suggested Exercise: (current drawing only)

1. Start with a drawing from a previous exercise or create a new drawing.
Instructor may wish to provide the users a drawing for this exercise.
2. Add some non-standard layers using basic AutoCAD LAYER commands or from with AutoCAD's Layer Manager same discipline. Optionally, draw some entities on those layers. You may work in Modelspace or Paperspace.
3. Run **Layer Conversion**.
4. Press the **Non-Standard** button if any layers shown in the list have a Layer Description other than "Non-Standard Layer."
5. Highlight a portion or all the remaining layers in the list. Use typical Windows multiple selection options (holding down <Ctrl> or <Shift>, etc).
6. Press the **Walk Layers** button and follow the command line prompting to isolate each layer selected in succession. Pause at any time to run a transparent zoom or pan function ("z" or "p") while "walking" through the layers. Note that by "returning" to the main dialog midstream the last layer shown in AutoCAD is highlighted in the CSM dialog.
7. Once back within the *Standards Manager – Layer Conversion Utility* dialog begin the process of assigning standard layers to a portion or all of the layers. Highlight one or more layers and press the **Assign...** button to bring up the *Select Individual Layer by Category* dialog. Note that the names of the layers you wish to convert are shown at the bottom left of this dialog.
8. Select the desired discipline containing the "intended" layer and then identify the layer by selecting the desired layer components. Press the **Ok** button to return to the main dialog and note the layer assignment for the original layer selection.
9. Add another layer assignment this time by double-clicking on a layer name in the left-hand column.
10. Once done with layer assignments press the **Save** button. In the dialog that appears, "check" the **Save to Text File** option. Press the Browse button and enter a desired file name for this conversion the **File Name** box near the bottom of the dialog. Please do not enter a file extension (i.e. *.txt*). Press the **Open** button and then the **Ok** button in the *Standards Manager Layer Conversion* dialog.
11. At this point please **Cancel** from the main layer conversion dialog.
12. Press your <Enter> key to repeat the last command (**Layer Conversion**).
13. Once again remove any standard layers and then press the **Retrieve** button. "Check" the **Retrieve Text File** option and **Browse** to select the file you previously saved. Press **Ok** to return to the main dialog. Note that your layer assignments have been retrieved.
14. Add more layer assignments if possible and "save" to the original text file from step 10. **What are the options you are presented with here?**
15. Finally, press the **Convert Layers** button to process the layer changes.

Suggested Exercise: (batch process only)

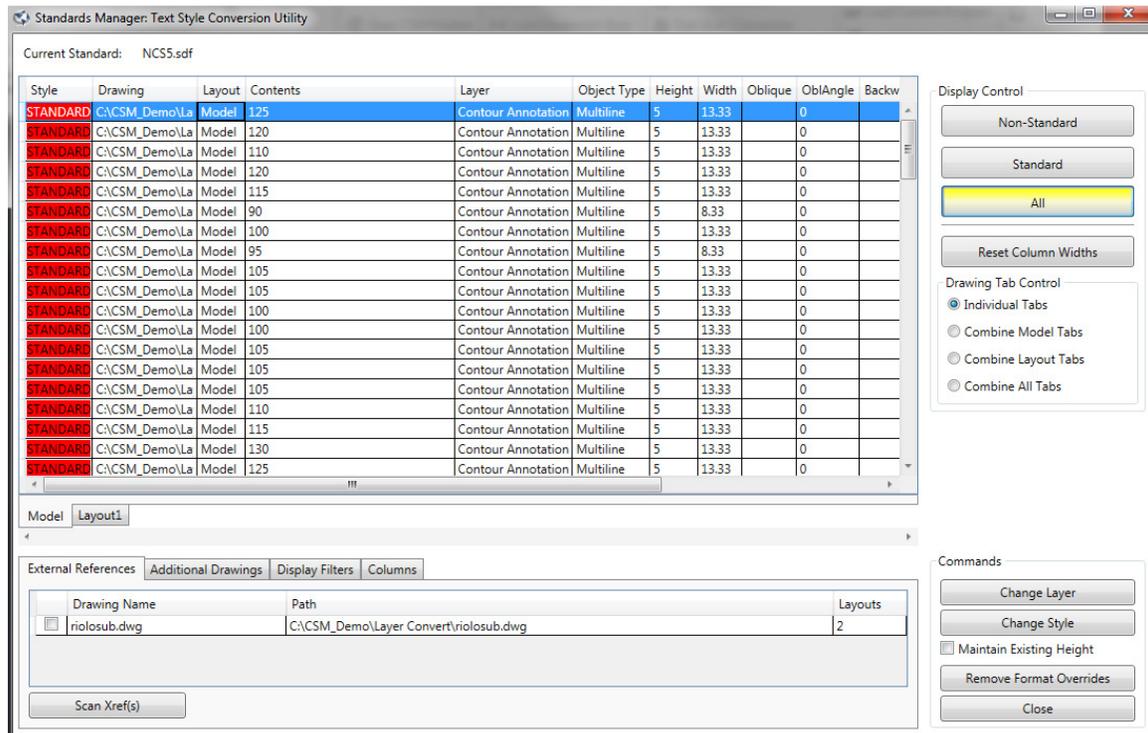
1. Start with a new drawing.
2. Run **Layer Conversion**
3. Under the Drawings tab, click on the Add Folder(s) button to add a folder with drawings. **Instructor should provide at least 5 drawings in the same directory to illustrate the flow process.**
4. Click **Scan Dwg**
5. Check the **Remove Duplicate Layers** check box
6. Within the *Standards Manager – Layer Conversion Utility* dialog begin the process of assigning standard layers to a portion or all of the layers. Highlight one or more layers and press the **Assign...** button to bring up the *Select Individual Layer by Category* dialog. Note that the names of the layers you wish to convert are shown at the bottom left of this dialog.
7. Select the desired discipline containing the “intended” layer and then identify the layer by selecting the desired layer components. Press the **Ok** button to return to the main dialog and note the layer assignment for the original layer selection.
8. Finally, press the **Convert Layers** button to process the layer changes.

Text Style Conversion

The compliance tool allows the user to convert the layers and style of text entities in the current drawing and attached Xrefs.

Run the command from the Standards menu by first selecting **Text Style Conversion** under the Compliance Tools sub-heading or alternatively, pick the **Text Style**

Conversion toolbar button  from the *CSM Convert* toolbar. The following dialog box will appear.



 This command searches the entire drawing for all text entities. Therefore, in a drawing with a large amount of text, the dialog box may take a while before appearing

Simply clicking on any row will zoom to the particular entity in the drawing. Clicking on any layout tab will refresh the list to show only text entities in that particular space.

The conversion process can scan through any attached Xref's. Simply check which XREF(s) you wish to scan and then click **Scan Xref(s)**. This will add any text entities from the Xref in the list. The program is limited in the fact that it does not allow the user to zoom to a particular Xref entity.

If the desire is to locate text at a certain height, then using the text height filter can be useful.

Drawing Scale: 40

Text Height Filter:

Plotted Units Lower Limit .1

Plotted Units Upper Limit .125

Apply Clear

Enter the desired drawing scale and the lower limits, and then click the **Apply** button. This will filter the list currently displayed. Simply click the **Clear** button to show all the text entities again.

The process of conversion starts by selecting text entities in the list. You can select multiple rows using windows standard selection keys or simply click and drag with the mouse. Select the **Change Style** button.

To make the text style assignment, first select the appropriate discipline in the **Discipline Name** drop-down list. Then select the desired **Group** from the drop down list. Finally select the text style, and then click the **Select** button.

☞ If the desire is to convert the text entity to a new style, but not change the text height, simply check the box *Maintain Existing Text Height(s)*.

If the desired is to change the layer, simply click the Change Layer button after you select the row(s) to convert.

Suggested Exercise:

1. Start with a drawing from a previous exercise or create a new drawing.
Instructor may wish to provide the users a drawing for this exercise.
2. Run **Text Style Conversion**
3. Click the **Non Standard** button.
4. Click on a particular row to zoom to a particular piece of text
5. Click on the first layout tab. This shows the list will change to the text entities in that particular space.
6. Select a few rows, and then click **Change Style**.
7. Click on a row that has been converted, which should zoom to the entity.
8. Check the properties on the entity to determine to guarantee its success.

☞ When exiting the command it gives you an opportunity to restore the zoom, layer state, and close Xref(s) opened during the command.

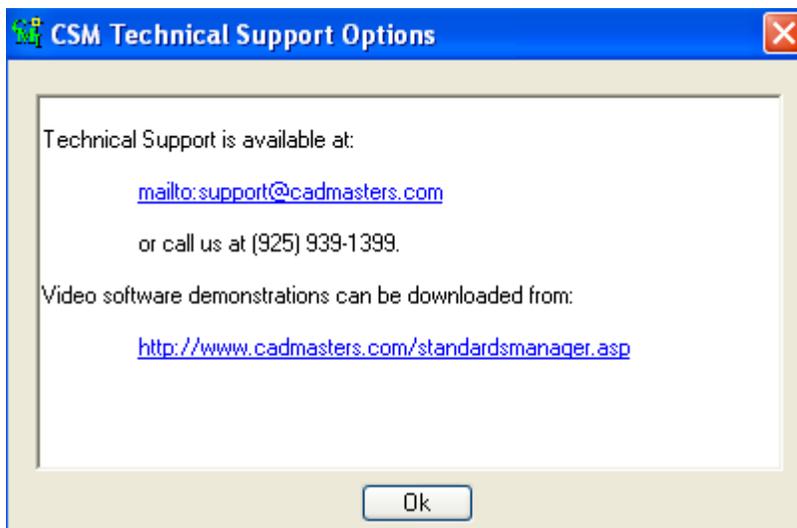
The **Removing Text Override** button makes the dimension associated to the measurement in AutoCAD.

Support Commands

At the bottom of the **Stds Manager** pulldown there reside commands for additional information for the user.

-  Open Reference Manual
- Technical Support Options
-  About CMI Standards Manager

Technical Support Options allows the CAD Manager to edit in order to provide email and/or web sites for user information.

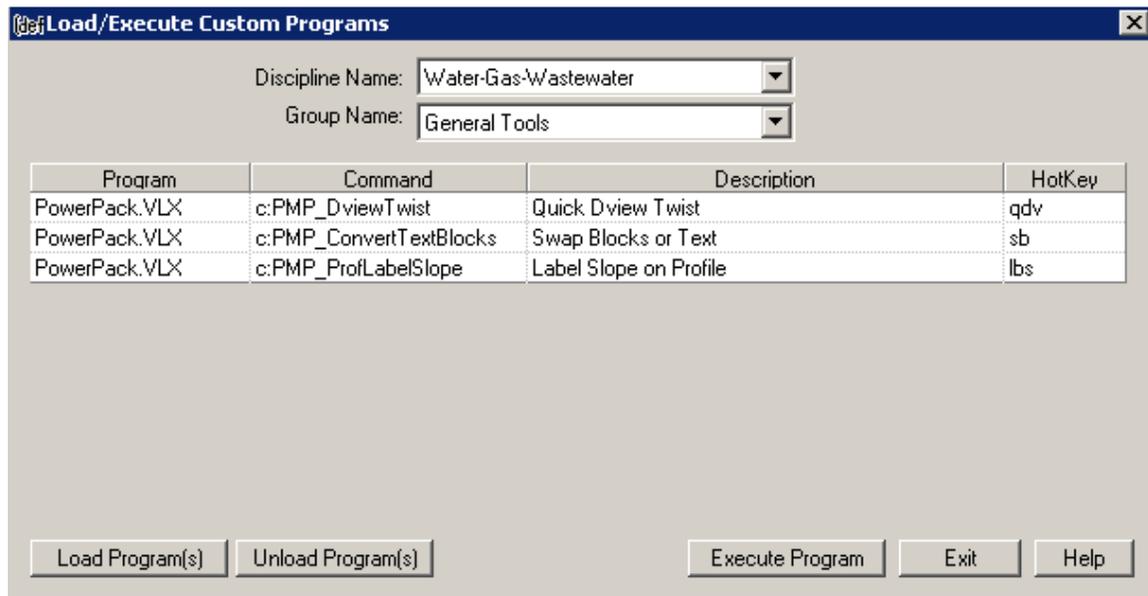


About CMI Standards Manager given the user the information about which version of standards manager is installed, as well as the build date for the latest update of the software.

Load Custom Programs

The *CMI Standards Manager* includes a tool that makes custom Lisp, VBA, and ARX programs available to all CAD users in the company. The routine allows you to load and run macros/programs defined within the current Standard. This interface is an alternative to use of the AutoCAD APPLOAD command and allows for better deployment and management of custom applications. To run the custom macros/programs, select the **Load Custom Program** command in the **Stds Manager** menu, or click on the **Custom Programs** toolbar button. (def

The Load/Execute Macros dialog box appears.



Load a custom macro/program into AutoCAD by first selecting the desired **Discipline Name** and **Group Name**. The programs associated with that discipline and group appear within the list below the group name. The program file name is listed in the left most column. If the program has internally defined commands then a command(s) may be listed adjacent to the program file name. If a Hot Key has been assigned within the Standards Administration module it will be displayed in the right-most column.

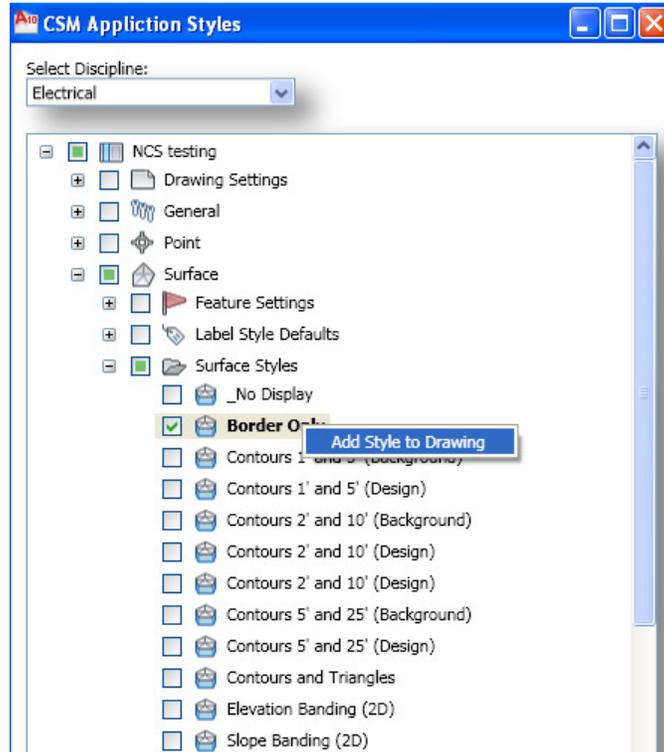
Immediately execute any program within the current list simply highlight the program and then click on the **Execute Program** button.

Those macros/programs that have assigned Hot Keys can be loaded within the current drawing session. To load program(s), highlight the desired programs, and click on the **Load Program(s)** button. The dialog box will close. Now, at the AutoCAD command line, execute a loaded program by typing its assigned Hot Key.

Programs previously loaded can be removed from the current drawing session by highlighting the desired programs, then clicking the **Unload Program(s)** button. Only programs with Hot Keys may be unloaded.

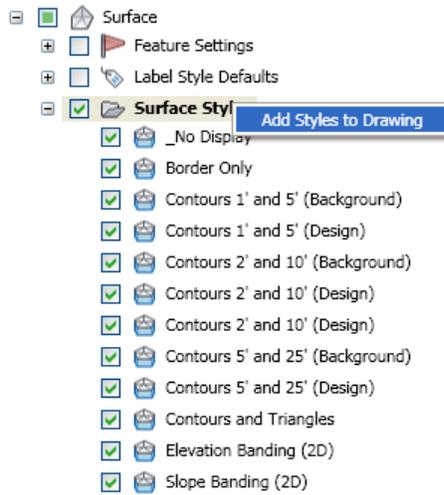
Civil 3D Styles

The *CMI Standards Manager* includes a tool to import Civil 3D styles and settings directly from the standards database. Simply execute the Application Styles command to invoke the import style utility.

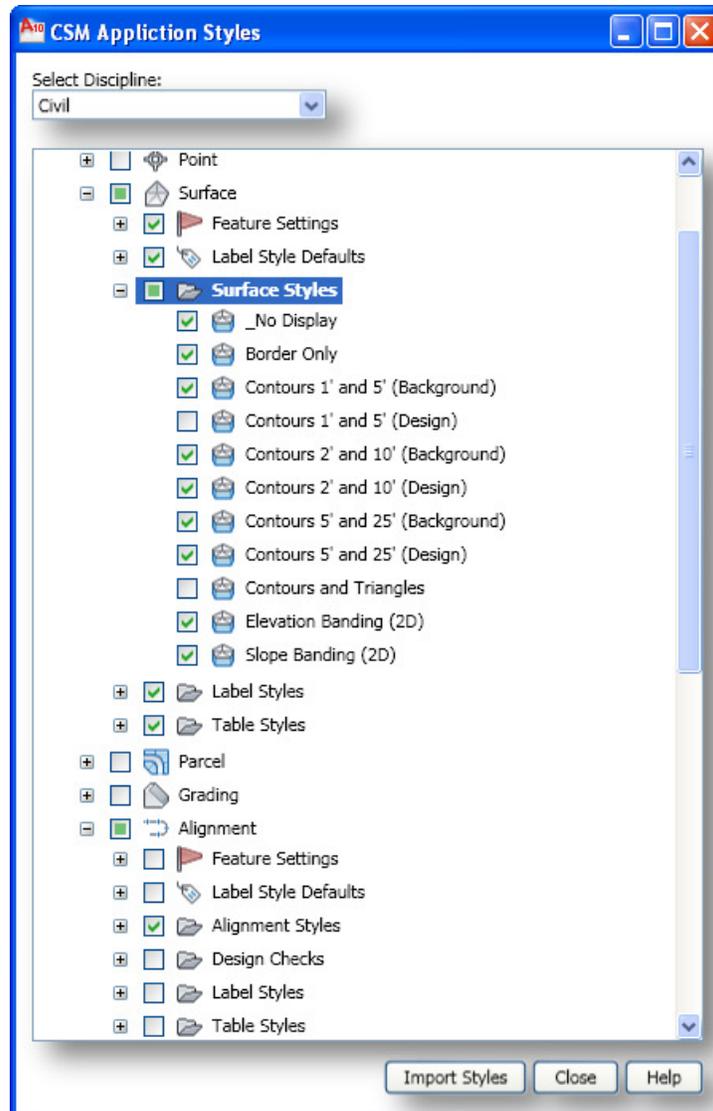


Choose the desired discipline from the drop down box located at the top of the dialog box. Then the software gives you 3 choices to import the styles to the drawing.

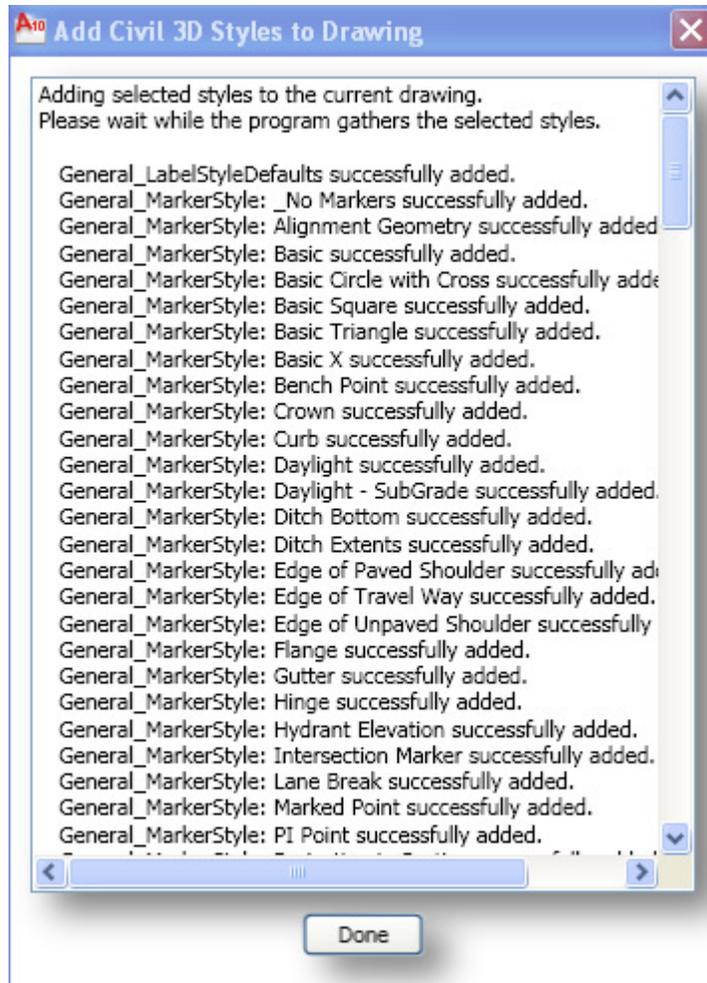
1. Navigate to the particular style, click the check box, and then right-click the style and choose *Add Style to Drawing*.
2. Right-click any style folder and choose *Add Styles to Drawing*. This will insert all the styles of that folder into the current drawing (see figure below).



3. Check the desired styles and settings in the interface and then click the Import Styles button located at the bottom of the dialog box. This will search the entire tree view and insert anything that is checked into the drawing.



While the program is inserting the styles to the drawing a progress window will appear describing the styles and settings that are importing. Also during the process, the AutoCAD status bar (where the coordinate display is shown) the current style being imported with be displayed.



After the completion of the process, the *Done* button will display giving the user control of the interface again.