

CMI CAD Standards Manager User Training Manual

*CAD Masters, Inc.
1111 Civic Drive, Suite 130
Walnut Creek, CA 94596
(925) 939-1378 sales
(925) 939-1399 support*

Table of Contents

CONVENTIONS USED IN THIS MANUAL	1
INTRODUCTION	2
EXPLANATION OF STANDARDS	2
THE LAYER STANDARD.....	3
LAYER COMMANDS MENU / TOOLBAR / RIBBON	4
DATABASE	5
SET DESIRED STANDARD.....	5
LAYER CREATION	6
OVERVIEW OF THE LAYER CREATION PROCESS.....	6
LAYERS PALETTE	6
LAYER SELECTION BY CATEGORY.....	8
LAYER SELECTION BY GROUP	10
IMPORT LAYERS FROM THE CURRENT DRAWING.....	12
LAYER PREFIX/SUFFIX	12
STYLE BASED LAYER SETUPS (USE OF <i>.STB</i> PLOT FILES)	13
SAVE AND RESTORE LAYER SETS.....	14
OTHER LAYER FUNCTIONS	16
LAYER ID.....	16
LIST LAYERS	17
<i>Loading Additional Drawings</i>	17
<i>Sorting</i>	18
<i>Filter</i>	18
<i>Column Display</i>	18
<i>Exporting Layer List</i>	19
RESET COLORS/LINETYPES/LINEWEIGHTS	19
LAYER GROUP TOOLS	20
LOADING LINETYPES.....	21
SAVE LAYER SETTINGS	21
RESTORE LAYER SETTINGS	22
SAVE VP LAYER SETTINGS	22
RESTORE VP LAYER SETTINGS	23
SYMBOLS	24
SYMBOL SETTINGS OPTIONS	25
SYMBOL INSERTION PARAMETERS	26
SYMBOL QUERY	27
DETAILS	28
DETAIL INSERTION SETTINGS	29
DETAIL QUERY.....	29
SHEET BORDERS	30
SHEET BORDER INSERTION SETTINGS	31
SHEET BORDER QUERY	31
MULTILEADER STYLES	32
PAGE SETUPS	33

TEXT STYLES	34
INSERTING A TEXT STYLE	34
DIMENSION STYLES	36
STANDARDS COMPLIANCE TOOLS	37
LAYER CONVERSION	37
TEXT STYLE CONVERSION.....	42
DIMENSION STYLE CONVERSION.....	43
SUPPORT COMMANDS	44
REPORTING	45
LOAD CUSTOM PROGRAMS	47
CIVIL 3D STYLES	48

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 CAD Standards Manager* include:

- Layer, Color, and Line Pattern Management
- Symbol, Detail, and Sheet Border Management
- Text, Multi-leader Dimension Style Management
- Standard conversion capabilities
- Custom Program Management

This manual was written to facilitate the use of the *User Module for CMI CAD 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 CAD 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.

 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. Example 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 CAD 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. For example, National CAD Standards is a 5 category system.

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.

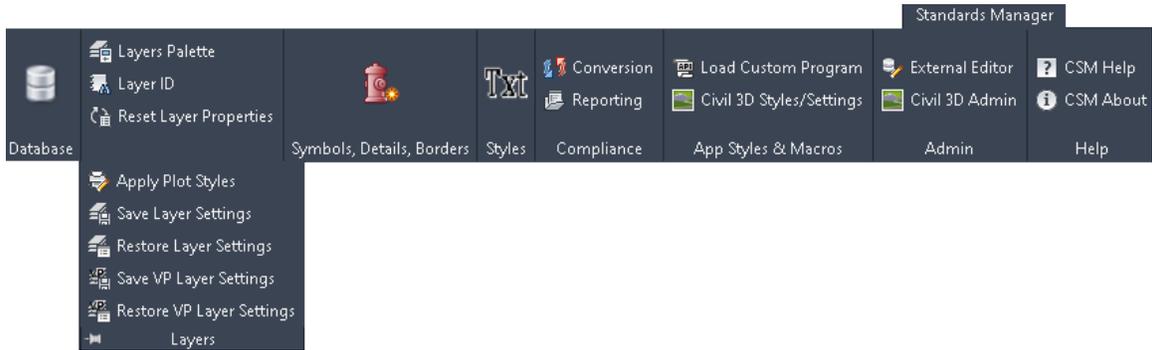
The Layer Standard is an agreed-upon structured naming convention for AutoCAD layers. Each layer should be assigned base properties of color, linetype, and optionally, lineweight. In the past most organization’s layer standards were published in a manual that was given to its CAD users. Each user was 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 CAD 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 Standards Manager ribbon is shown in the figure below. Quick keys can easily be made using the command line: all CAD Standard Manager command begin with *CMI_CSM*.



Database

Set Desired 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: **Database** from the given ribbon. You must select the database type, Standards Library Path, and database to ensure the correct setting.

Standards Manager Connection

Database Type
SQL Server Compact

Standards Library Path
C:\ProgramData\CMI\Standards Library

Database
 Use DefaultStandard.txt
NCS

Connect Cancel

Available Database Types

1. Microsoft SQL Server
2. SQL Server Compact Database File (*.sdf)
3. SQLite Database File (*.sqlite)

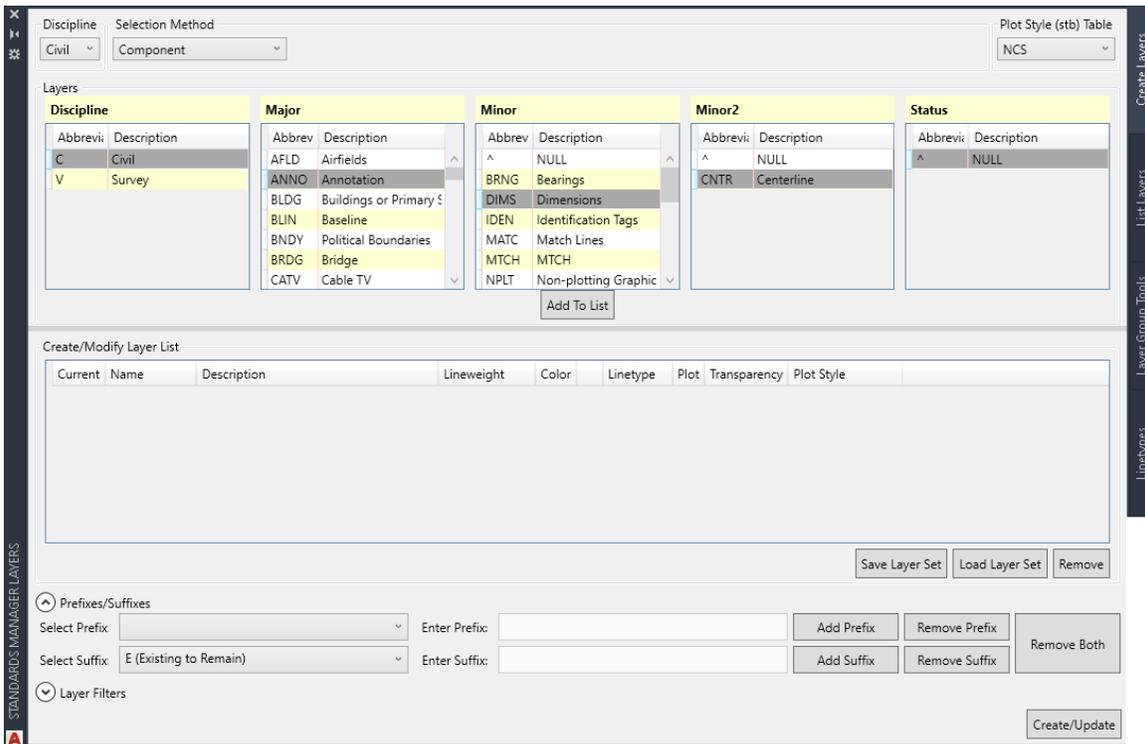
Layer Creation

Overview of the Layer Creation Process

Standard layers are created within the *Standards Manager Layers* palette. The process of adding layers begins by selecting an appropriate discipline. You may then select layers by one of four options: **Component**, **Group**, **Search**, and **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 **Group** option allows you to accomplish this quickly. The **Component** option allows you to select layers by individual category components. Once you have selected layer components within each displayed category you select Add to List that adds them to the Create/Modify Layer list. You can continue to add layers and/or make modifications such as 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.

Layers Palette

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 **Layers Palette** command from the **Standards Manager** ribbon or type **CMI_CSM_LAYERS** at the command line.



Standards Manager Layers Palette commands / functions overview:

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

Layer Selection Method

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

Group Allows selection from a defined layer group.

Drawing Imports layers from the current drawing.

Search Search layers in the current discipline. Please note that the syntax for the search is separated by an exclamation point. For example:

Contours ! Existing

This would search for layers with names or descriptions containing Contours, but exclude layers with name or descriptions that contain Existing.

The **Create/Update** button is used to create the new layers or make any changes to existing layers in the current AutoCAD drawing.

Optional Layer Prefix / Suffix Additions

The *CMI CAD 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 Layers Palette* dialog will show additional commands at the bottom of the dialog as shown in the figure below.

Prefixes/Suffixes

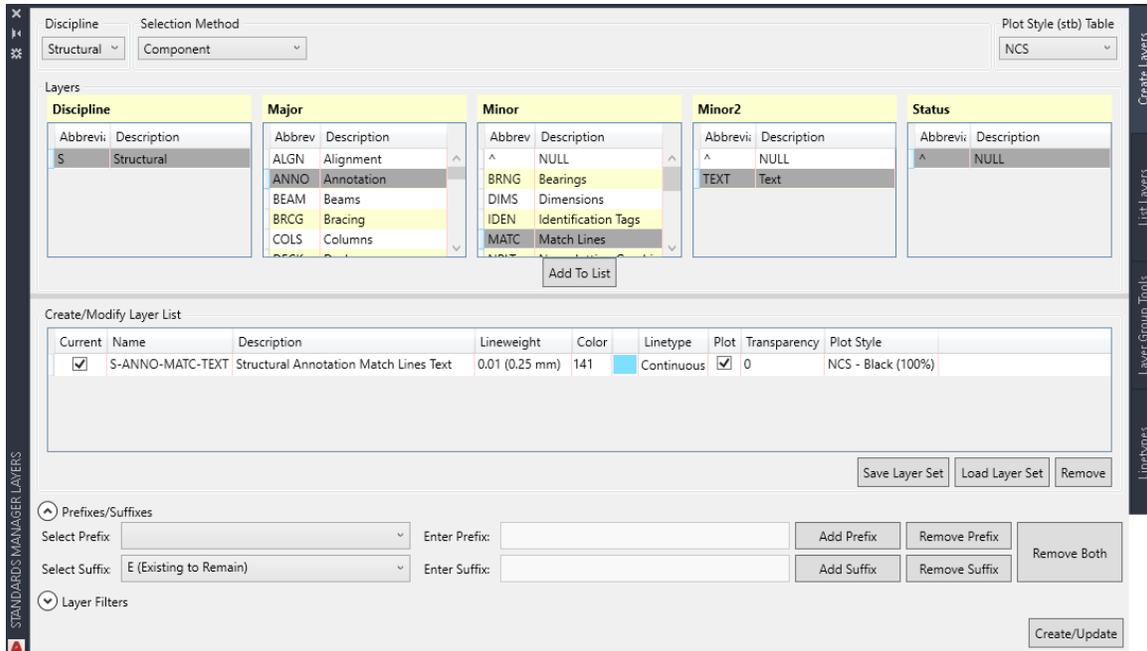
Select Prefix	<input type="text"/>	Enter Prefix:	<input type="text"/>	Add Prefix	Remove Prefix	Remove Both
Select Suffix	E (Existing to Remain)	Enter Suffix:	<input type="text"/>	Add Suffix	Remove Suffix	

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.

Remove Prefix/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 Method (Component)* dialog box is shown in the figure below.



Component 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 CAD 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 **Component** will be disabled in the *CSM Layers Palette* 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

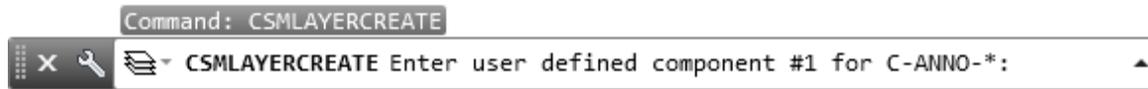
components to select them. Selecting multiple components in a category displays components in the next category that are common to the selected set of components. If no components are displayed in the next category after multiple components have been selected then there are no components in the next 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. If **Create/Update** 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.



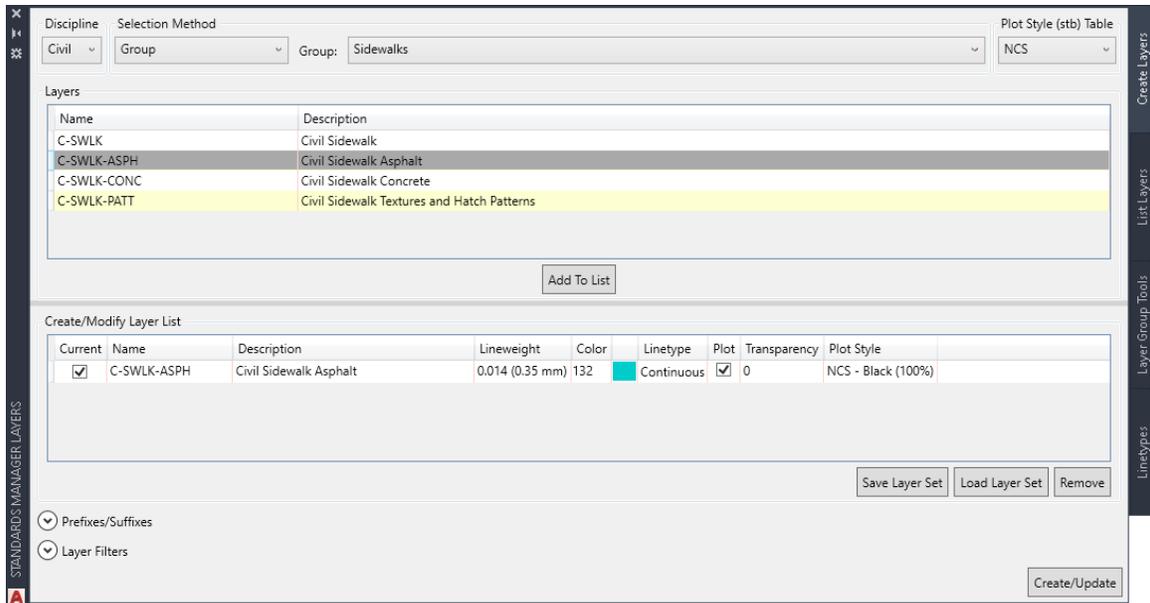
Suggested Exercise:

(For discipline setups configured with multiple layer categories)

1. From the **Standards Manager** ribbon tab menu select **Layers Palette**. Once the palette appears, select a desired discipline, and then select the **Component** option from selection method drop down list.
2. Select at least one component in each category.
3. Press the **Add to List** button.
4. Highlight the items in the *Create/Modify Layer List* and then press the **Remove** button.
5. Select one component once more in each category and then press the Add to List button, and then select the **Create/Update** 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 Layers Palette (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 Layers Palette* 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 **Create/Update** to import them to the drawing.

Suggested Exercise:

1. Press **Layers Palette** from the **Standards Manager** ribbon tab.
2. Pick a desired discipline and then choose the **Group** selection method.
3. Inside the *CSM Layer Selection (By Group)* dialog pick a desired group from the **Group** list.
4. Using the various Windows multiple selection methods pick a sampling of layers from the list.
5. Finally, press the **Create/Update** > button found in the lower right corner of the *CSM Layers Palette* dialog box.
6. 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 **Drawing** selection method option activates The layer list of current standard layers in the drawing. You may wish to import layers from a drawing in order to apply quick modifications to layers such 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

Layer Prefixes/Suffixes

Select Suffix: A(Abandoned) Enter Suffix:

Add Suffix Remove Suffix

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.

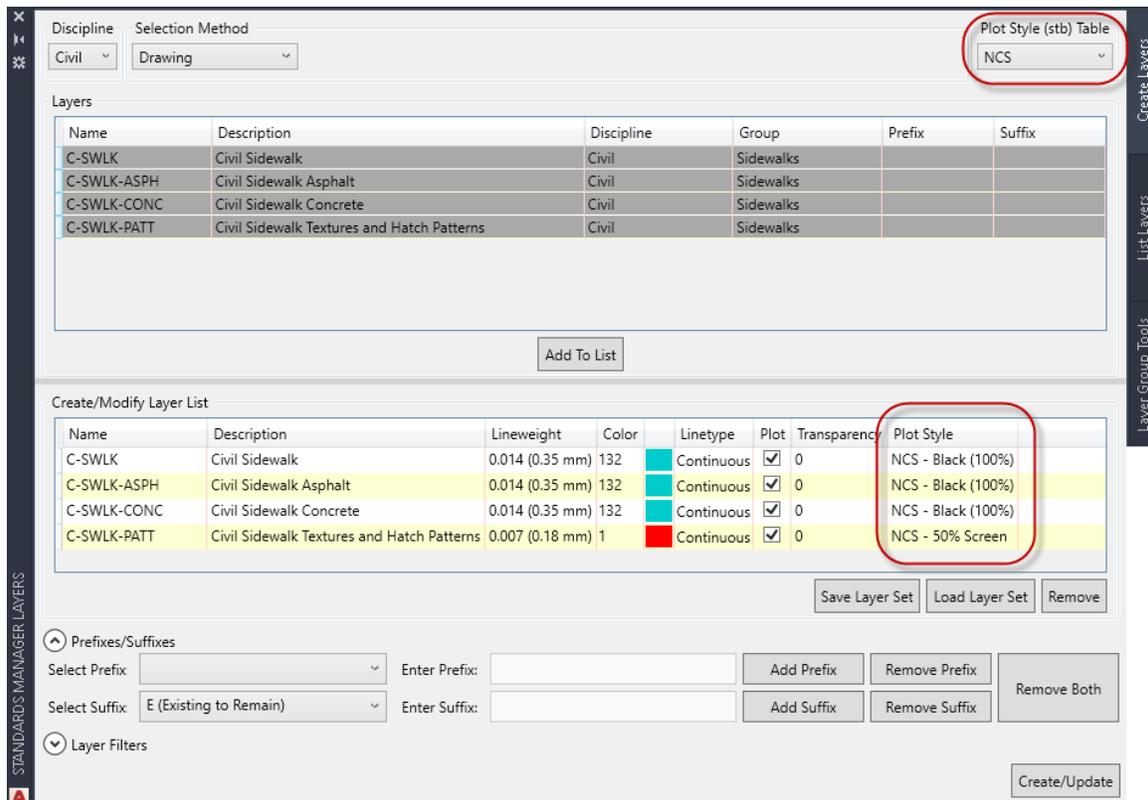
Once you have selected or entered the desired prefix /suffix, clicking the **Add** button will modify all of the highlighted layers and optionally update the layer’s properties (i.e. color, linetype, etc.) according to the prefix/suffix definition. Note: these changes do not take effect in the drawing 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 *Layers Palette* dialog using the **Drawing** method and make the desired modifications.

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 Conversions Palette to rename existing layers.

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 *Layers Palette* 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.



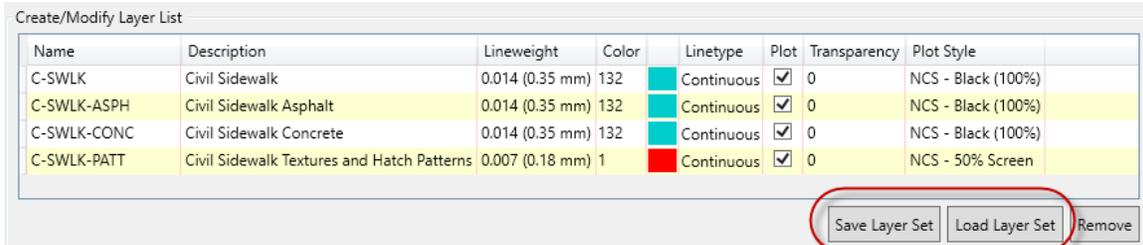
You should also note the column “PlotStyle” that is now displayed within the layer list window. The software refers to the current standard database to determine the named plot style for each layer. If you have more than one plot style table available for your CAD standard then the additional table names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

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

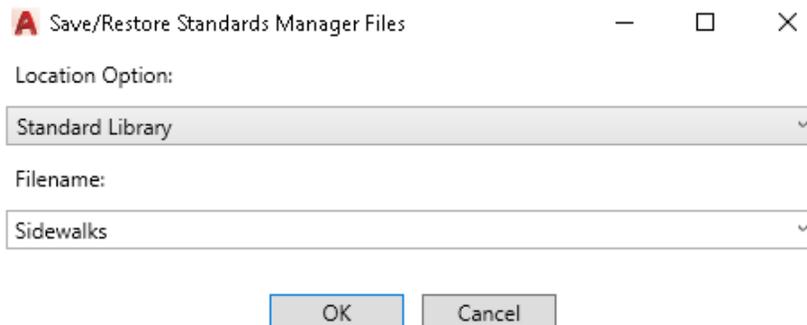
1. Open the Layers Palette and select a desired discipline.
2. Bring in layers to the main dialog using **Component** or **Group**.
3. Note the Plot Style names assigned to the layers
4. Press the Create/Update button to create these layers in your currently active drawing.
5. Finally, open AutoCAD’s 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 container 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 Set** 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 **Standards Manager** ribbon tab.

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 also, 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: CMI_CSM_LAYERID
Select object [Xref]:
Multiple standard layers found for layer C-TOPO-MAJR:
C-TOPO-MAJR: Description = Civil Topography Major Lines / Contours, Discipline = Civil, Group = Surfaces
C-TOPO-MAJR: Description = Civil Topography Major Lines / Contours, Discipline = Civil, Group = Topography
C-TOPO-MAJR: Description = Civil Topography Major Lines / Contours, Discipline = Survey, Group = Topography-Civil
Select object [Xref]: x
Select Xref object:
riolosub\C-ESMT-UTIL: Description = Civil Easements Utilities, Discipline = Civil, Group = Easements
```

```
Command: CMI_CSM_LAYERID
Select object [Xref]:
Non-Standard Layer: Trees
```

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.

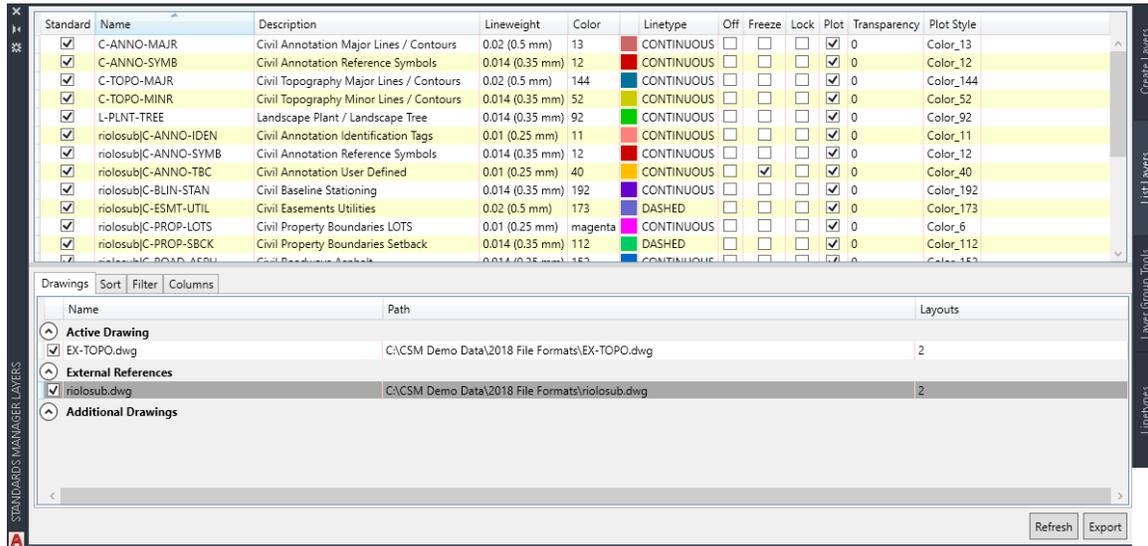
 Important note: The Layer ID command will identify individual Xreference entity layers using the Xref option.

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

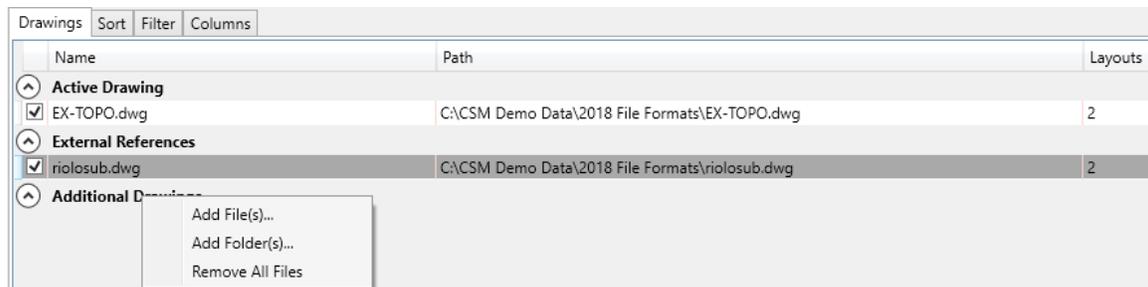
This command lists all layers in the current drawing and provides a facility to print the list in a table. To run the command, select **Layers Palette** from the **Standards Manager** ribbon. The command displays the *List Layers* tab from the Layers Palette (displayed below).



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, the description will be blank.

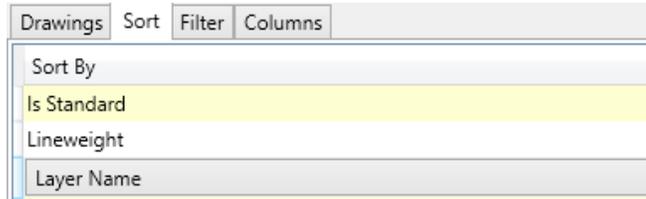
Loading Additional Drawings

In addition to the active drawing and external references, this can load any number of additional drawings. Simply right click on additional drawings and add drawings by selecting files or folders.



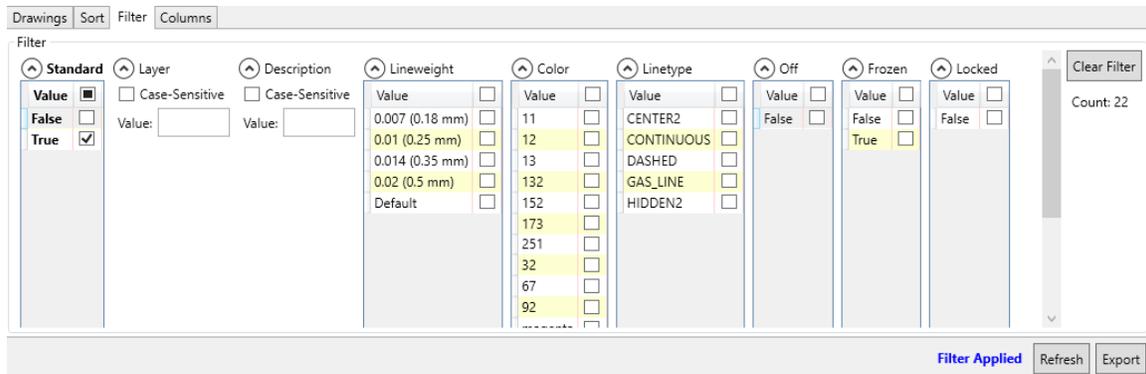
Sorting

This tab allows the user to sort by any number of columns. Simply click the row you wish to edit or add a sorting option. To remove a sort simply highlight the row and hit the delete key on the keyboard.



Filter

To isolate certain layers in the list, simply activate the *Filter* tab and select the desired options. By checking any of the options, this will isolate that layer's properties while filtering out the layers that are not selected in the filter. If any filters are used, a "Filter Applies" text will display in the lower right corner of the palette.



Column Display

The end user can turn on and off any column for display. This will affect the display when exporting the layer list to an Excel spreadsheet.



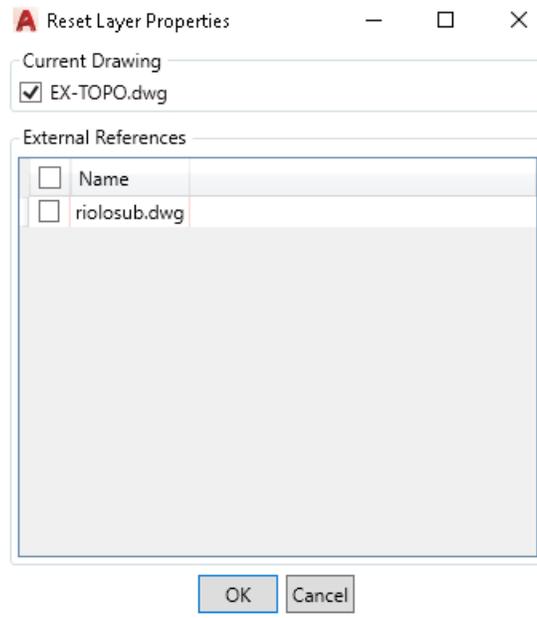
Exporting Layer List

Once all filters, sorting, and column options are set, hitting the **Export** button will connect to Microsoft Excel and create a new file for printing or saving for later use.

Reset Colors/Linetypes/Lineweights

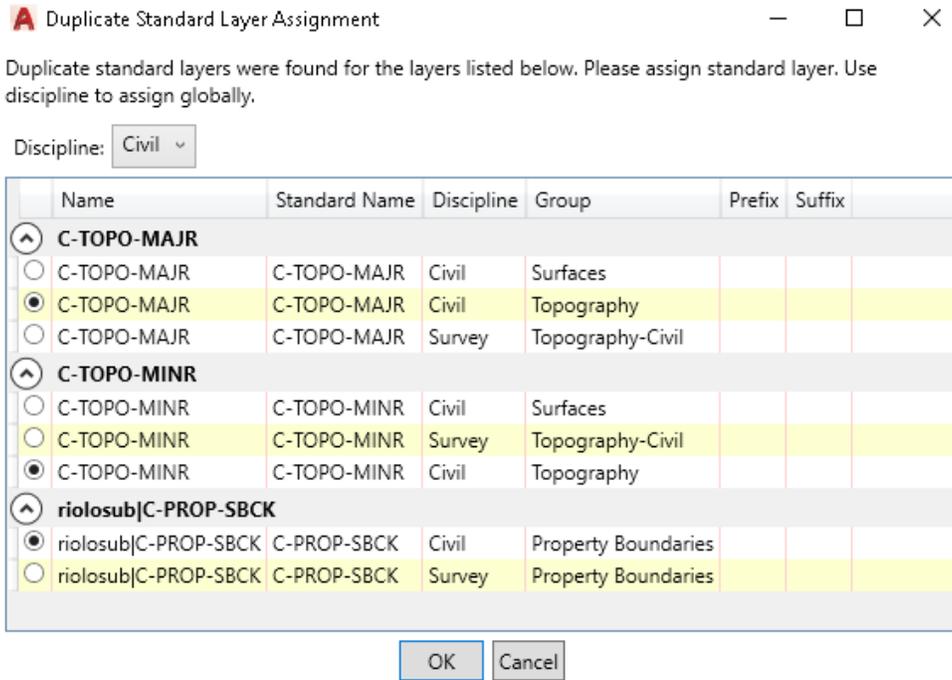
This command resets the layer display properties of all standard layers found within the current drawing to their respective standard values. If you are using named plot styles then this command updates all layers to reflect the named styles for the currently set plot style table (.stb file). In this 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 Layer Properties** from the **Standards Manager** ribbon tab.



Additional functionality of the Reset Colors/Linetypes/Lineweights

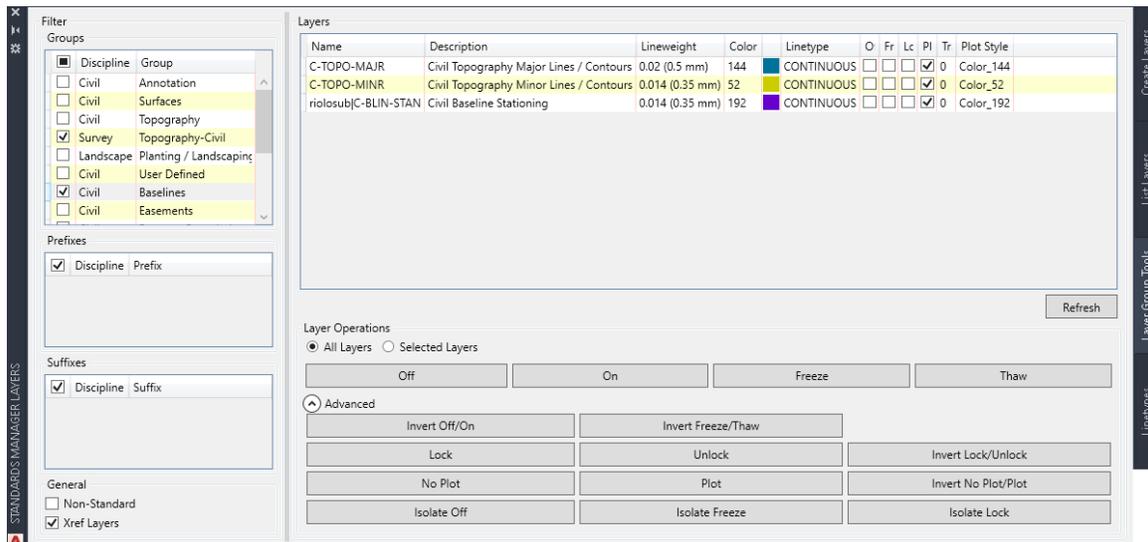
Optionally, your CAD standards administrator can place the exact same layer name in more than one discipline. In this case, if the software determines that a layer name has more than one set of properties in the database (duplicate layers exist) then you will be prompted to select the controlling discipline.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

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** palette is shown below.



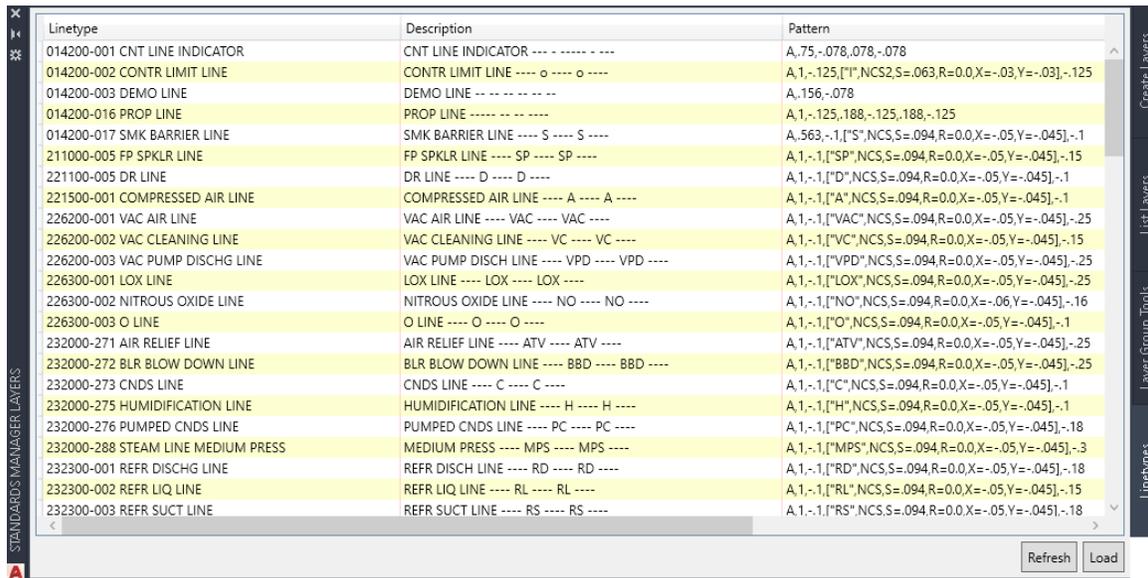
Execute the command from the Standards Manager tab by selecting the *Layers Palette* command. Then select the *Layer Group Tools* tab on the palette,

This tab will scan the current drawing for all standard layers and display all found layer groups within the database. Simply check the groups you wish to modify, and then choose the layer operation to use.

Once you have selected the desired layer groups use one of the **Layer Operations** or **Advanced** commands to apply a command to the entire selection set at once. The **Layer Group Tools** tab 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.

Loading Linetypes

There are times when the user needs access to the defined linetypes from the database, and the *Layers Palette* command allows for this. Simply open the **Layers Palette** from the Standards Manager ribbon tab and go to the Linetypes tab. From there, select the desired layers and click the **Load** button.



Linetype	Description	Pattern
014200-001 CNT LINE INDICATOR	CNT LINE INDICATOR ---- - - - - -	A,75,-.078,.078,-.078
014200-002 CONTR LIMIT LINE	CONTR LIMIT LINE ---- o ---- o ----	A,1,-.125,[1],NCS2,S=.063,R=0.0,X=-.03,Y=-.03],-.125
014200-003 DEMO LINE	DEMO LINE -- - - - - -	A,156,-.078
014200-016 PROP LINE	PROP LINE ---- - - - - -	A,1,-.125,188,-.125,188,-.125
014200-017 SMK BARRIER LINE	SMK BARRIER LINE ---- S ---- S ----	A,.563,-.1,[S],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.1
211000-005 FP SPKLR LINE	FP SPKLR LINE ---- SP ---- SP ----	A,1,-.1,[SP],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.15
221100-005 DR LINE	DR LINE ---- D ---- D ----	A,1,-.1,[D],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.1
221500-001 COMPRESSED AIR LINE	COMPRESSED AIR LINE ---- A ---- A ----	A,1,-.1,[A],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.1
226200-001 VAC AIR LINE	VAC AIR LINE ---- VAC ---- VAC ----	A,1,-.1,[VAC],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.25
226200-002 VAC CLEANING LINE	VAC CLEANING LINE ---- VC ---- VC ----	A,1,-.1,[VC],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.15
226200-003 VAC PUMP DISCHG LINE	VAC PUMP DISCH LINE ---- VPD ---- VPD ----	A,1,-.1,[VPD],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.25
226300-001 LOX LINE	LOX LINE ---- LOX ---- LOX ----	A,1,-.1,[LOX],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.25
226300-002 NITROUS OXIDE LINE	NITROUS OXIDE LINE ---- NO ---- NO ----	A,1,-.1,[NO],NCS,S=.094,R=0.0,X=-.06,Y=-.045],-.16
226300-003 O LINE	O LINE ---- O ---- O ----	A,1,-.1,[O],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.1
232000-271 AIR RELIEF LINE	AIR RELIEF LINE ---- ATV ---- ATV ----	A,1,-.1,[ATV],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.25
232000-272 BLR BLOW DOWN LINE	BLR BLOW DOWN LINE ---- BBD ---- BBD ----	A,1,-.1,[BBD],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.25
232000-273 CNDS LINE	CNDS LINE ---- C ---- C ----	A,1,-.1,[C],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.1
232000-275 HUMIDIFICATION LINE	HUMIDIFICATION LINE ---- H ---- H ----	A,1,-.1,[H],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.1
232000-276 PUMPED CNDS LINE	PUMPED CNDS LINE ---- PC ---- PC ----	A,1,-.1,[PC],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.18
232000-288 STEAM LINE MEDIUM PRESS	MEDIUM PRESS ---- MPS ---- MPS ----	A,1,-.1,[MPS],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.3
232300-001 REFR DISCHG LINE	REFR DISCH LINE ---- RD ---- RD ----	A,1,-.1,[RD],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.18
232300-002 REFR LIQ LINE	REFR LIQ LINE ---- RL ---- RL ----	A,1,-.1,[RL],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.15
232300-003 REFR SUCT LINE	REFR SUCT LINE ---- RS ---- RS ----	A,1,-.1,[RS],NCS,S=.094,R=0.0,X=-.05,Y=-.045],-.18

Save Layer Settings

The **Save Layer Settings** command saves all layers properties 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 Manager** ribbon by first selecting the Layers dropdown from the Layers, and then **Save Layer Settings**.

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 Manager** ribbon by first selecting the Layers dropdown from the Layers, and then **Restore Layer Settings**.

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.

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 Manager** ribbon by first selecting the Layers dropdown from the Layers, and then **Save VP Layer Settings**.

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 a 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/Layout (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 Manager** ribbon by first selecting the Layers dropdown from the Layers, and then **Restore VP Layer Settings**.

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)*

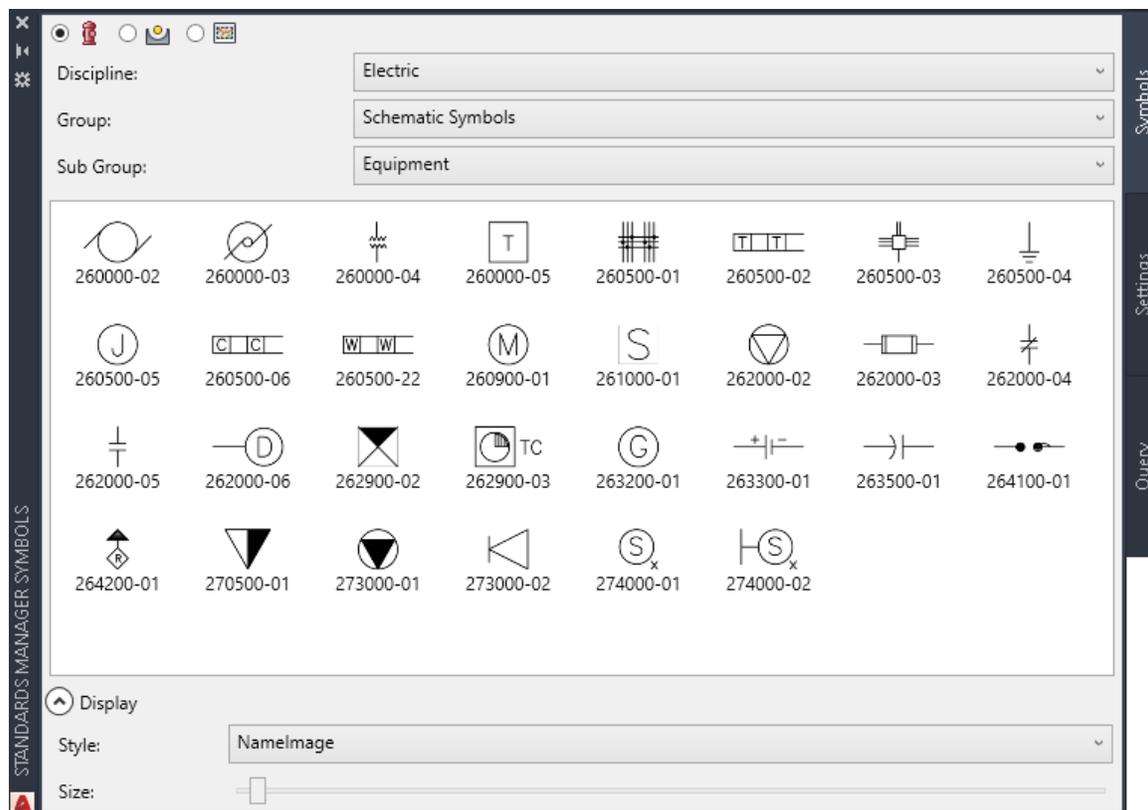
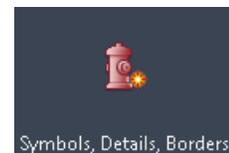
The “freeze in active viewport” setting will be set for those layers stored in the selected .vpt file.

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

Symbols

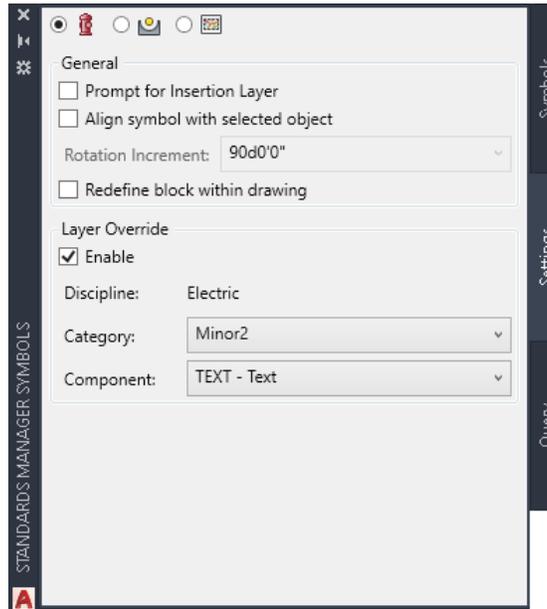
Symbols, by typical definition, are blocks (whose entities are created on layer “0”) that represent a physical object or design element. They may appear multiple times in one drawing and may sometimes be used on multiple layers. By creating the block’s entities on layer “0” they then take on the properties 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 *CMI 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 **Symbols Details Sheet Borders Palette** command from the **Standards Manager** ribbon tab.



The palette is resizable and image size can be adjusted. Use the Display area of the symbols tab to set the desired image display size and type as described in the **Style**.

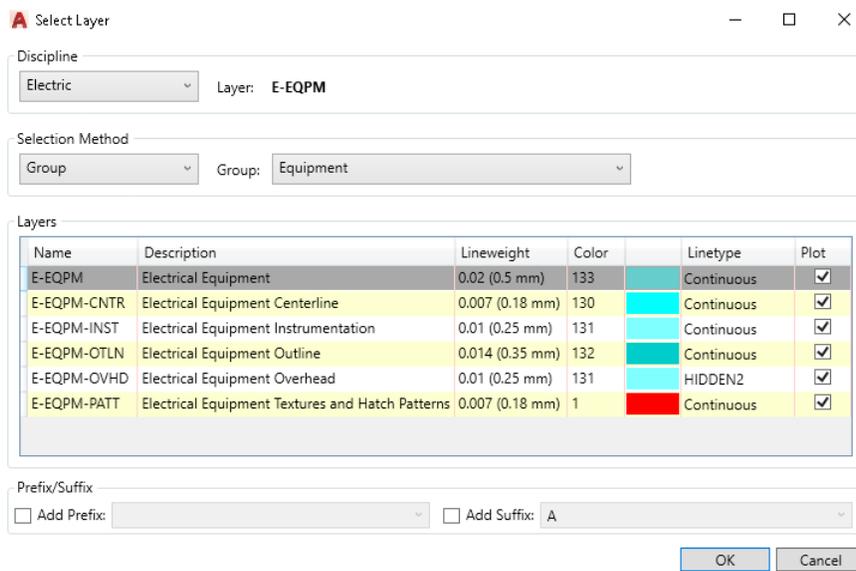
Symbol Settings Options



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

Select Layer [From standards/Select entity/Default] <Default>:

The standard layer as defined for the symbol is the default layer option, hitting <Enter> will choose this option. Typing “s” allows you to select an entity from the drawing to establish the insertion layer. Alternatively, typing “F” brings a Select Layer interface to choose the desired layer any number of ways.



Select a layer name and press “Ok”. 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, Arc or Polyline. 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. Once the symbol rotation is as desired, left click to mouse to complete the insertion.

 The rotation option will also work on an entity selected through and Xref.

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.

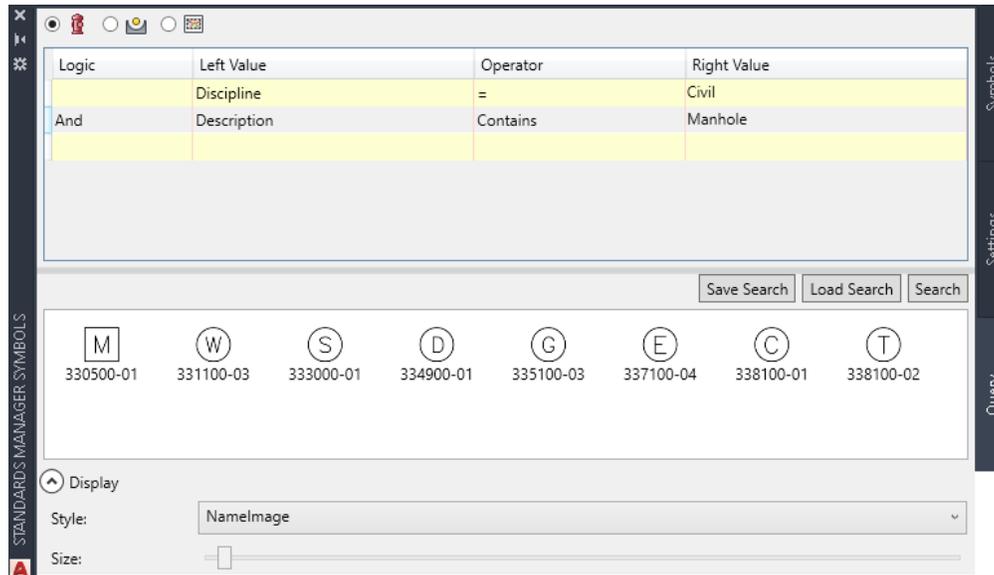
With **Align symbol with selected object** set in the *Symbols Settings Tab* dialog you will first be prompted to choose an insertion point. You may align the symbol with a line, arc or polyline, and must use object snaps to pick the point of insertion on the desired object. Once you have chosen the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

At this point the symbol’s angle is aligned to the entity (tangent or normal to the object) and the user can then move the mouse to display the different available angles depending on the rotation increment selected in the *Settings* tab for symbol options.

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

Symbol Query

A company's symbol library can be large and difficult to location the desired block. Therefore, the CAD Standards Manager allows the user to search for a particular symbol to make it quicker to find.

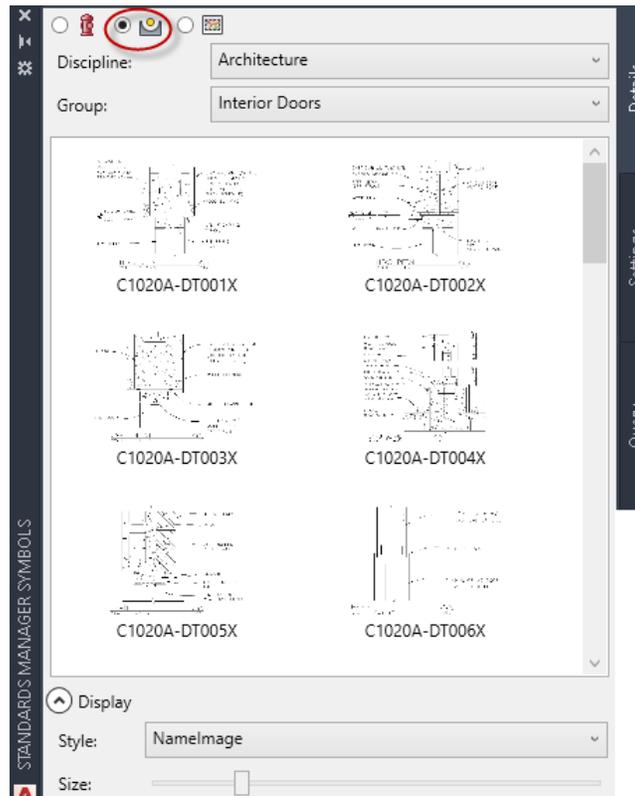


The figure above displays a few of the options allowed for the query to find the desired symbol. The user can save this query for later use with the **Save Search** button. The user can select the desired symbol from the windows to insert it to the current drawing.

☞ To remove a row from the query simply highlight the row and hit the *Delete* key on the keyboard.

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 **Symbols Details Sheet Borders Palette** command from the **Standards Manager** ribbon tab.



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.

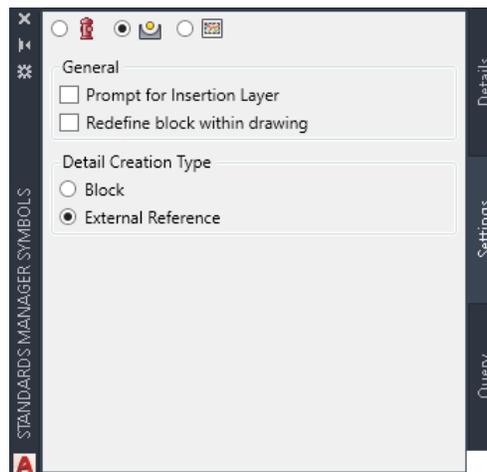
Before inserting or attaching the detail select the desired method of insertion in the **Settings** tab. Your choices are as a **Block** or **External Reference**. Choosing **Block** will insert the detail as a single entity.

Selecting **External Reference** 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. The detail will be placed in a folder of the users choosing with the default location being the same folder and the current drawing. A dialog will be displayed giving the user the capability of choosing any folder to place the copied detail. The software uses overlay as the default option for external reference type and a relative path (if possible) for the saved path option.

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** during the insertion of the detail.

Detail Insertion Settings

The only available options when inserting details is **Prompt for insertion layer** and **Redefine block within drawing** as shown in the figure below.



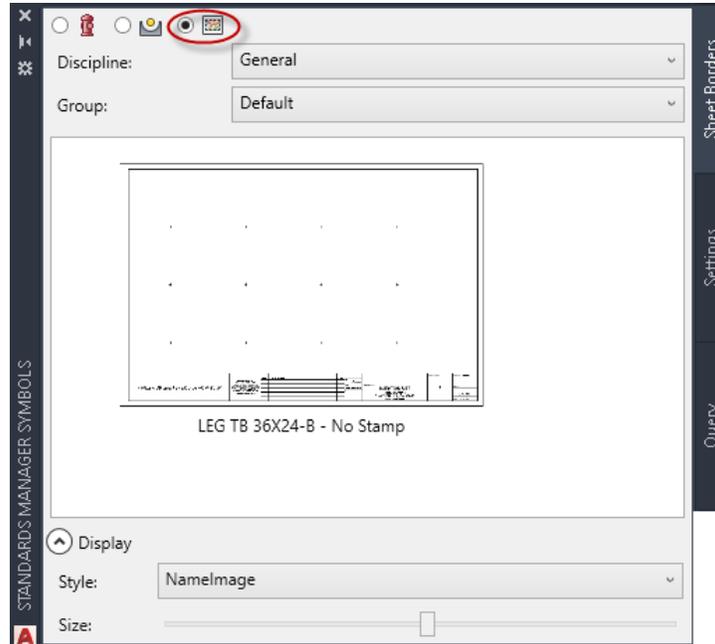
The Layer Selection Process is described in **Symbols** section of this manual.

Detail Query

The exact same feature exists for detail query as for symbol query. Please refer to the symbol section of the manual for further explanation.

Sheet Borders

Sheet Borders (title blocks) can be accessed from the **Symbols Details Sheet Borders Palette** command from the **Standards Manager** ribbon tab.



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.

If the CAD Manager has the database option to External Reference, the software will prompt the user for the desired folder to place the titleblock. If the drawing file already exists in this folder, the software will not overwrite it, instead it will Xreference the drawing already in the folder.

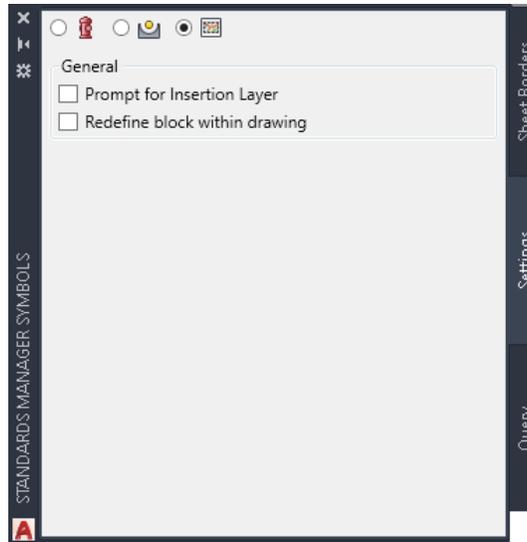
If you have “checked” the **Prompt for insertion layer** option you will be prompted for the insertion layer. If not, you will be immediately prompted for the insertion point and the default layer set in the database will be used. By default this is 0,0 as shown here:

```
Insertion Point <0,0,0>: <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*.

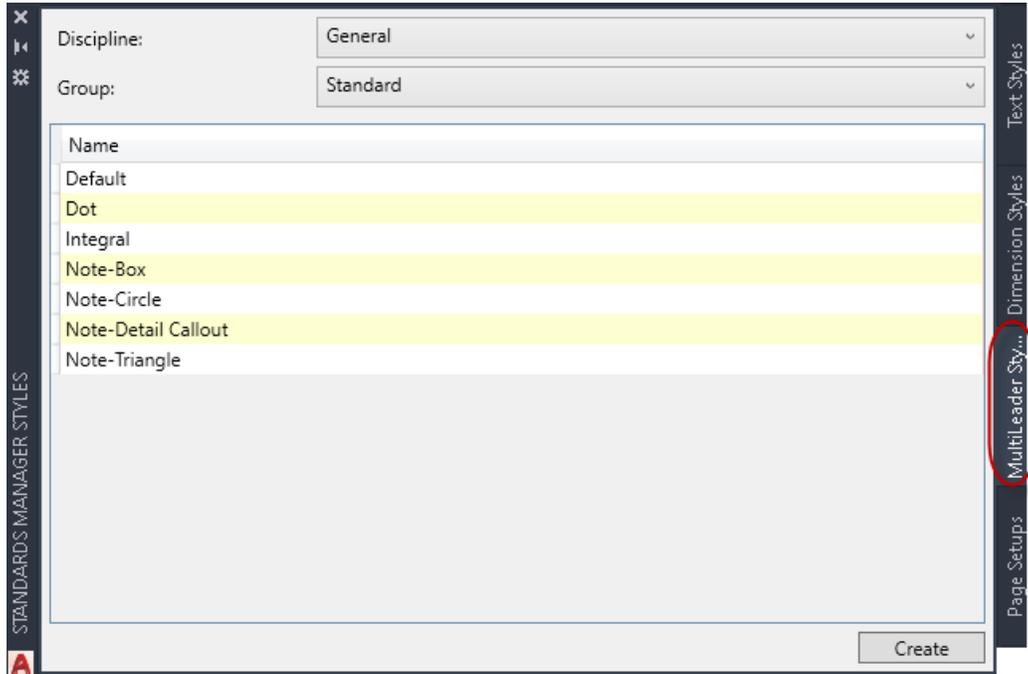


Sheet Border Query

The exact same features exists for sheet border query as for symbol query. Please refer to the symbol section of the manual for further explanation.

MultiLeader Styles

MultiLeader styles can be accessed from your standard via the **Styles Palette** command from the **Standards Manager** ribbon tab.

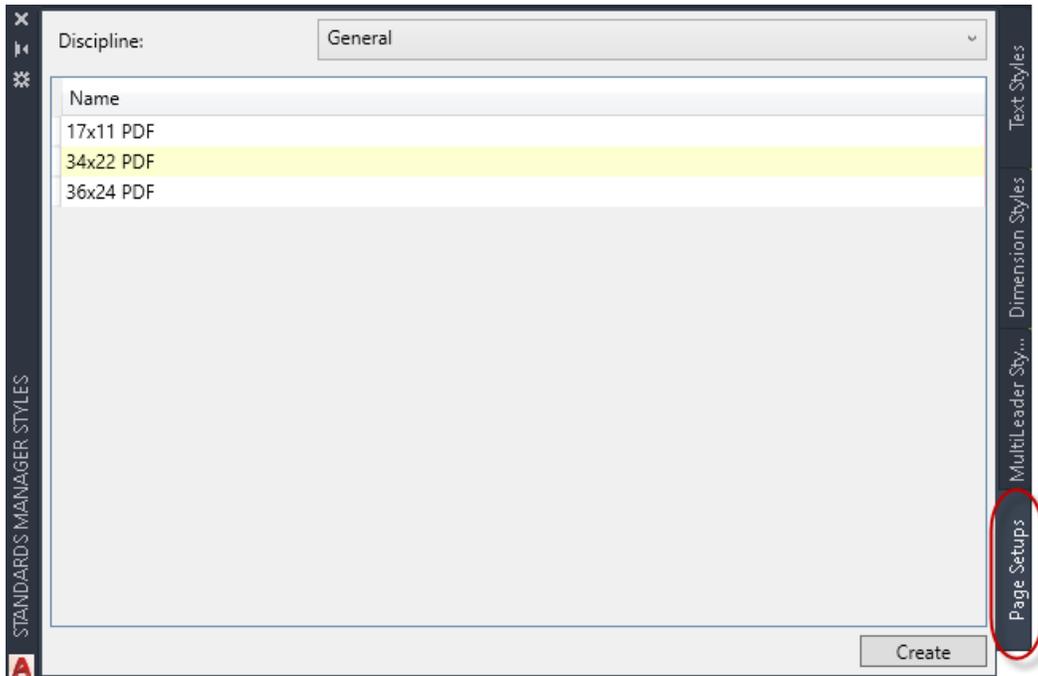


Choose the desired Discipline and Group from the drop-down list, and then select the desired multileader style(s) to load into the drawing. Finally, click **Create** to add the selected styles to the current drawing.

➤ 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 **Styles Palette** command from the **Standards Manager** ribbon tab.



Choose the desired Discipline from the drop-down list, and then select the desired page setup(s) to load into the drawing. Finally, click **Create** to add the selected page setups to the current drawing.

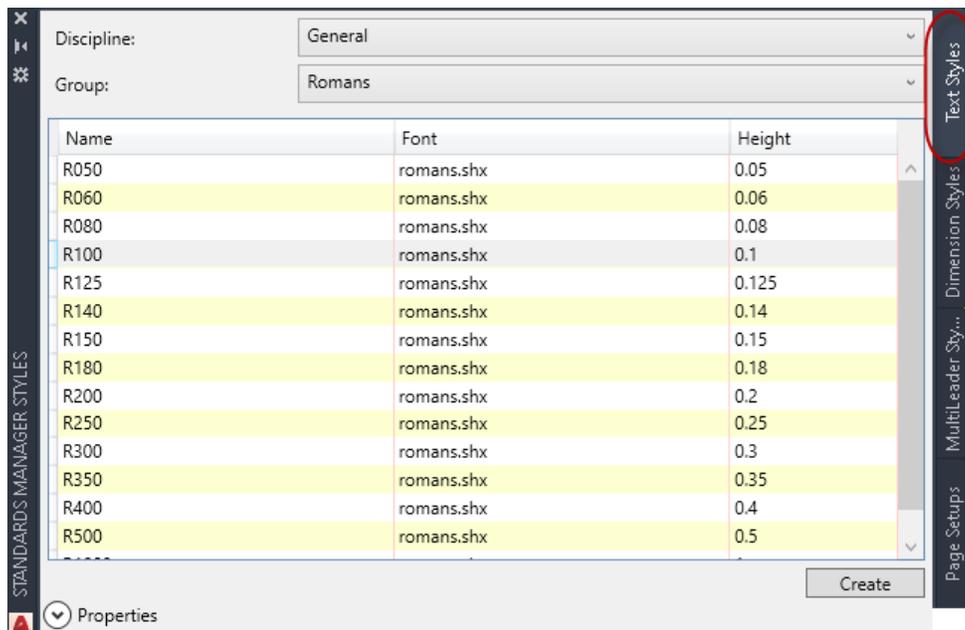
Text Styles

Text style standards ensure that text entities will have a consistent appearance from drawing to drawing. The *CMi CAD 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 CAD 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. If you were to insert a text style defined as annotative, then it will remain the unit height regardless of the scale of the current drawing.

Inserting a Text Style

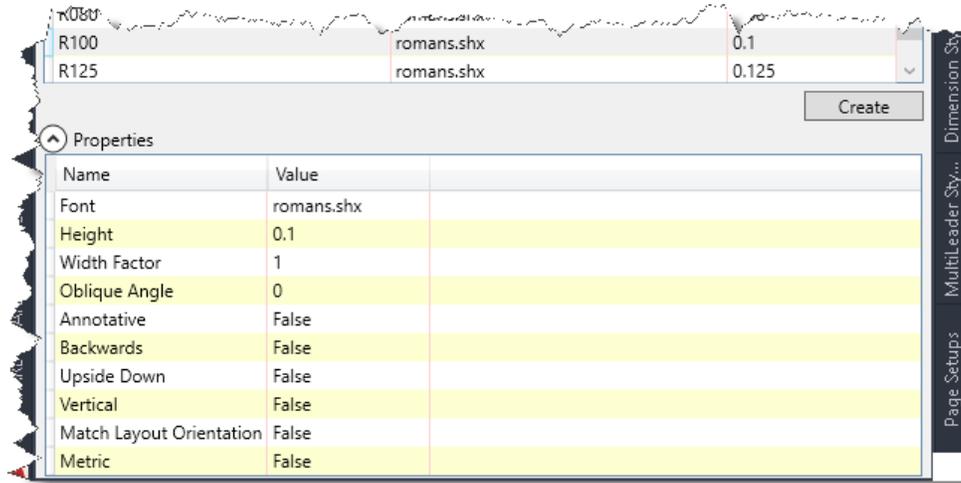
Before inserting a text style to the drawing be sure that the drawing scale is set properly by setting the annotative scale from with AutoCAD. The drawing scale will be used in the creation of text in Modelspace (and Modelspace viewports) to ensure proper text height.

Text styles can be accessed from your standard via the **Styles Palette** command from the **Standards Manager** ribbon tab.



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 list. Left mouse click on a name to select a style. The

attributes for the selected style can be displayed by expanding the properties list at the bottom of the palette.

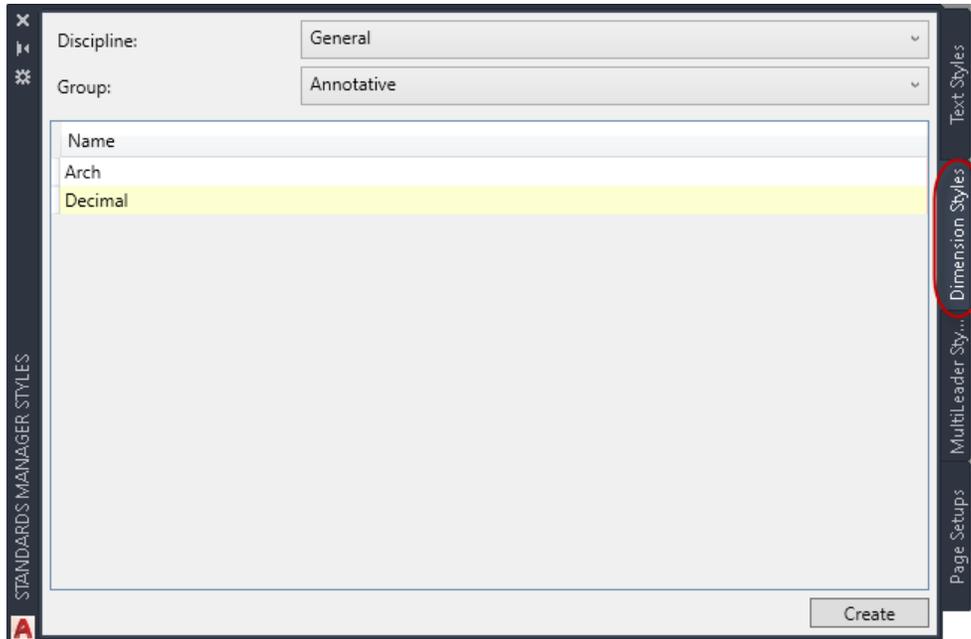


Once the proper style(s) are selected click the **Create** button. This will load the selected text and set it as the current style in the drawing. Alternatively, double clicking on the desired text style will load and set it current. You can also load multiple styles at once either by selecting a style one at a time and clicking **Create** or highlighting multiple styles in the list before clicking **Create**. You can select multiple styles using standard windows techniques (i.e. <Shift> or <Ctrl> keys).

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.

MultiLeader styles can be accessed from your standard via the **Styles Palette** command from the **Standards Manager** ribbon tab.



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 list. Left click on a name(s) to select a style. Click the **Create** button to load the style and set it current. Alternatively, double clicking on the desired dimension style will load and set it current.

Standards Compliance Tools

CAD Standards Manger (CSM) 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 4 separate tools: (1) Layer Conversion, (2) Text Style Conversion, (3) Dimension Style Conversion, and (4) MultiLeader Conversion

As you navigate through the tabs of the conversion palette, all settings are saved between the tabs even though it is refreshing the properties as you change tabs.

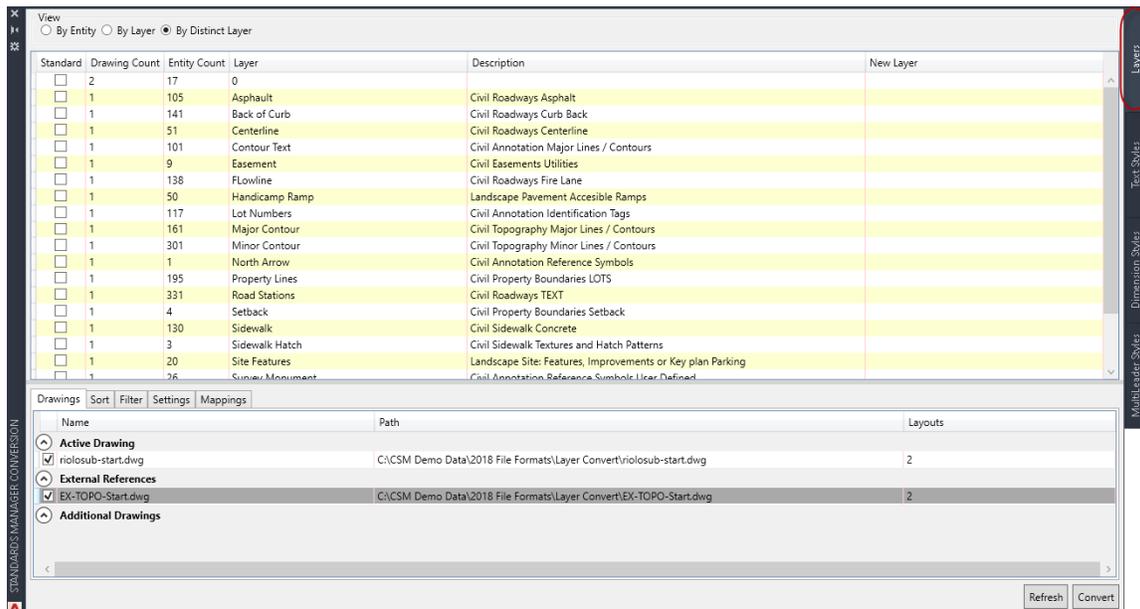
Layer Conversion

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.

Conversion commands can be accessed from your standard via the **Conversion Palette** command from the **Standards Manager** ribbon tab.

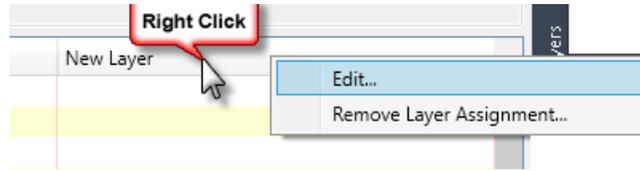


The layer conversion utility can be a very powerful tool to assist you in converting drawing to the company standard. The command allows the user to change and update attached X References, as well as multiple drawings at the same time. The batch processing allows entire projects to be converted very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

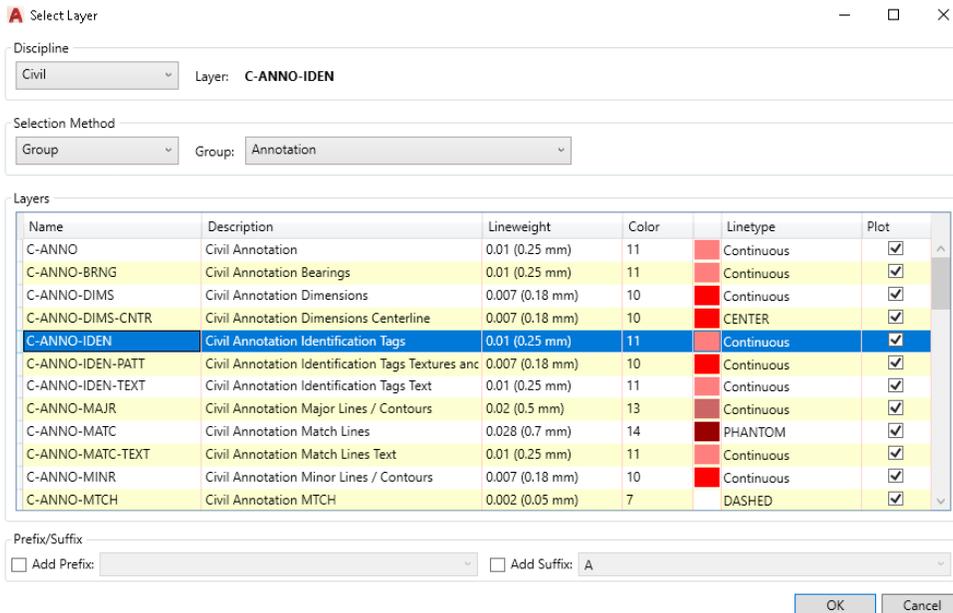


The process of conversion starts by selecting layers in the list. You can select multiple layers using windows standard selection keys or simply click and drag with the mouse.

You can modify the new layer information one at a time, or multiple but right-clicking on the new layer column header and choosing **Edit**.

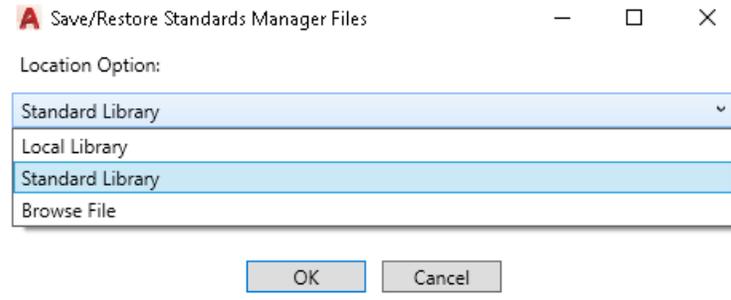


The dialog displayed below shows the interface to select the correct layer.



To make the assignment, first select the appropriate discipline in the **Discipline** drop-down list. Then choose the Select Method and finally the layer you wish to assign to the selected layers in the drawing. You may continue the selection and assignment process until all the necessary assignments are made.

Under the Mappings tab within the palette the **Save Mappings** button is useful if you have several drawings with the same layers that need conversion. Once you have made all the necessary layer assignments as described above you can save the non-standard to standard layer assignments by clicking **Save Mapping**. This opens the dialog shown below:

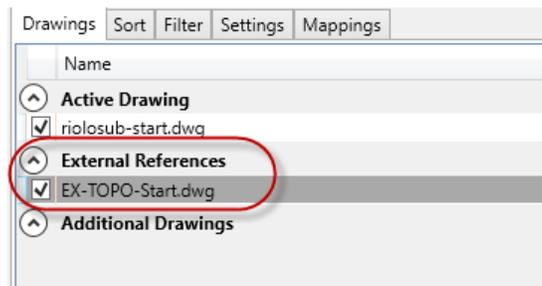


You have the option of saving the conversion assignments to the database or to a text file. The local library option will save the file to the local computer that can only be used by the current user. The Standard Library option will save the file to the standards library path and can be used for all users. Finally, the Browse file option allows the user to place the external file wherever they desire.

The **Load Mappings** 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.

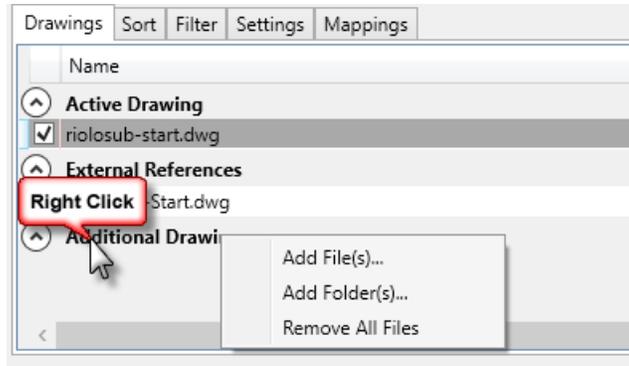
Convert X-Reference Layers

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



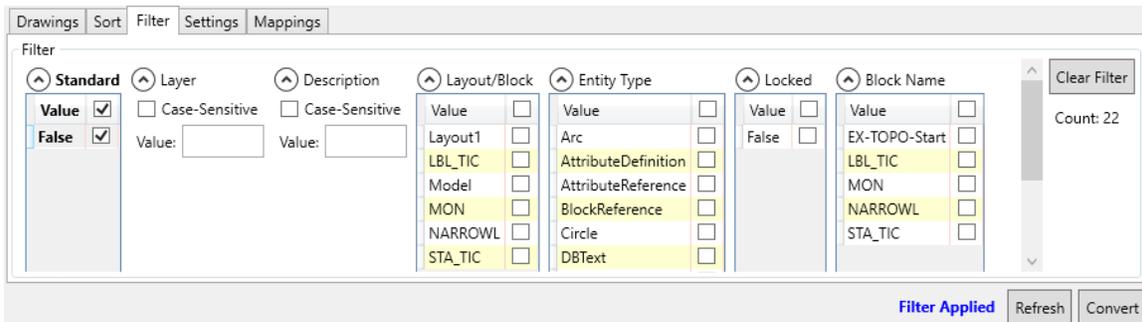
Convert Multiple Drawings (batch process)

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. An excellent filter choice would be to *By Distinct Layer*; therefore if the same layer name exists in multiple drawing, only one will appear.



Filter Options

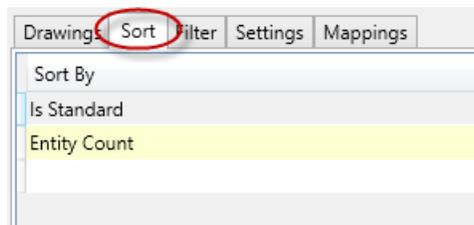
The tab allows the user to filter by entity type. This function only can work within the current drawing and not any attached X-References.



For example, if a user is looking for only non-standard layers, they could easily filter for those entities. The current filter is remembered by the software and will only be removed once the **Clear Filter** button is pressed.

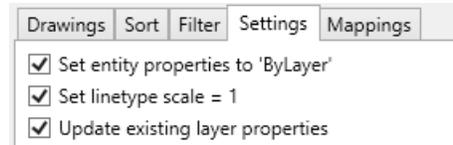
Sorting

To find the desired layers quickly, the user can set multiple sorting columns to display as desired. The user can add as many columns to the sort, and by and the desired order. If you wish to remove any sorting column simple highlight the row and hit the delete key on the keyboard.



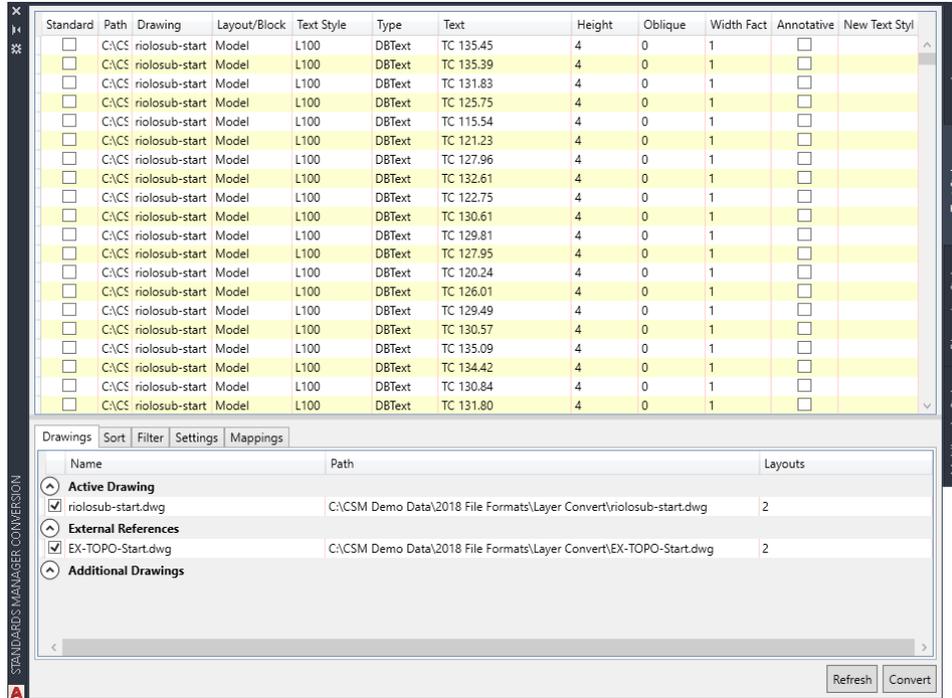
Miscellaneous Settings

When finally ready to process the layers to convert, this function has a few items to redefine the layers in the drawing. These are in the Setting tab.



Text Style Conversion

Conversion commands can be accessed from your standard via the **Conversion Palette** command from the **Standards Manager** ribbon tab.



The conversion process can scan through any attached Xref's or additional drawings selected. Simply check which XREF(s) you wish to add to the list. This will add any text entities from the Xref in the list.

The process of converting text styles 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. Right-click the New Text Style column header and select **Edit**.

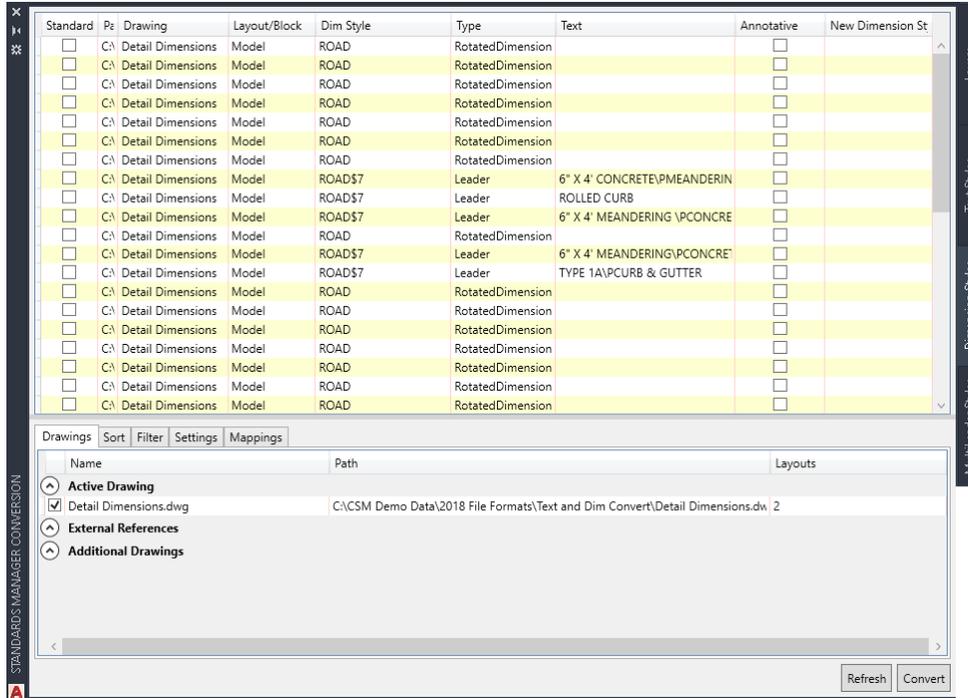
To make the text style assignment, first select the appropriate discipline in the **Discipline** drop-down list. Then select the desired **Group** from the drop down list. Finally select the text style, and then click the **Select** button.

Once you have the desired mappings set, simply click the **Convert** button to active the command.

Please see the layer conversion section of the manual for a description of the Sort, Filter, Settings, and Mappings tab.

Dimension Style Conversion

Conversion commands can be accessed from your standard via the **Conversion Palette** command from the **Standards Manager** ribbon tab.



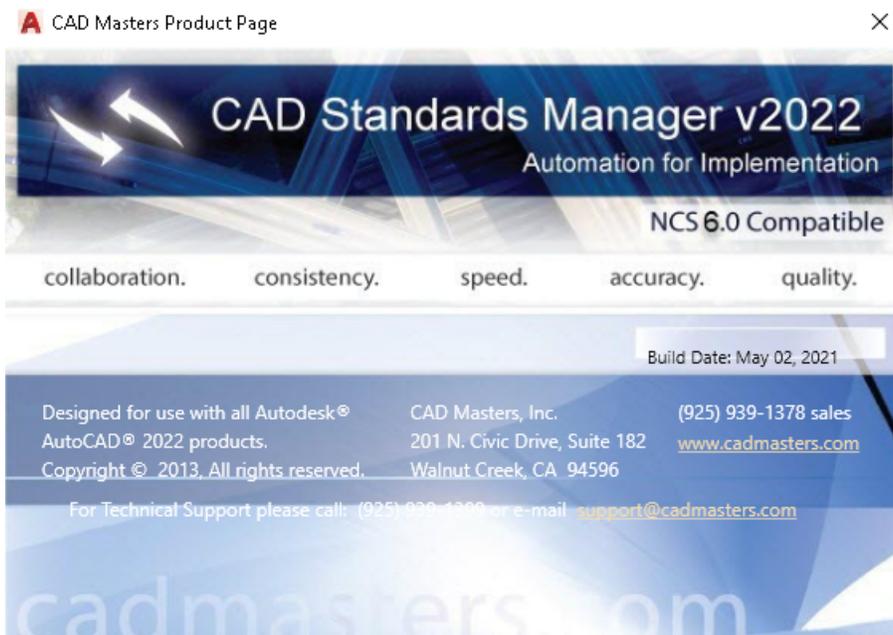
The conversion process can scan through any attached Xref's or additional drawings selected. Simply check which XREF(s) you wish to add to the list. This will add any dimension entities from the Xref in the list.

Support Commands

On the **Standards Manager** ribbon there reside commands for additional information for the user.



CSM About 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. Contact information for support is located here as well.



Reporting

CAD Standards Manager allows the user to perform quality checks on the current drawing, external references, or additional drawings at any time. This can then be exported to a spreadsheet so that it can be used for quality control. For example, before a company wishes to send a project to a client, they would like to guarantee the adherence to their standard practices. This can be done in a single command. If any deviations from standards exist, CAD Standards Manager also can correct those with the **Conversion** command.

Reporting can be accessed from your standard via the **Reporting Palette** command from the **Standards Manager** ribbon tab.



The screenshot shows the Reporting Palette interface. At the top, there are radio buttons for 'By Entity' and 'By Layer'. Below is a table with columns: Standard, Path, Drawing, Ent, Layer, Description, Lineweight, Color, Linetype, O, Fr, Lc, Pl, Tr, Plot Style, Pr, St, and Violations. The table lists various standards such as 'CACS EX-TOPO 8', 'CACS EX-TOPO 101', etc. Below the table is a 'Drawings' section with a table containing columns for Name, Path, and Layouts. The 'Drawings' table lists 'Active Drawing', 'EX-TOPO.dwg', 'External References', and 'riolosub.dwg'. At the bottom right, there are 'Refresh' and 'Export' buttons.

First determine if you wish to display the current drawings, external references, or add additional drawings to the reporting list. Any drawing(s) checked in the list with automatically be added to the reporting view.

This is a close-up of the 'Drawings' section from the screenshot above. It shows a table with three columns: Name, Path, and Layouts. The table contains the following entries:

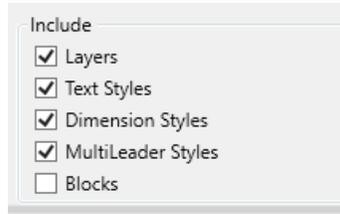
Name	Path	Layouts
Active Drawing		
<input checked="" type="checkbox"/> EX-TOPO.dwg	C:\CSM Demo Data\2018 File Formats\EX-TOPO.dwg	2
External References		
<input checked="" type="checkbox"/> riolosub.dwg	C:\CSM Demo Data\2018 File Formats\riolosub.dwg	2
Additional Drawings		

Next, you can cycle through the different tabs (layers, text styles, dimension styles, multileader styles, and blocks). Each tab can be individually sorted, filters, change

column display, and determine additional settings. For example, for the Layers tab, the settings are as follows.



Finally, to create the report, click the **Export** button on the bottom right of the palette in any particular tab. If you click on the **Export** button while in any tab other than the *All* tab, it will only export that particular tab. Using the **Export** button in the *All* tab will export every style checked in the include window.



An example of exporting layers, filtering out standard layers is as follows.

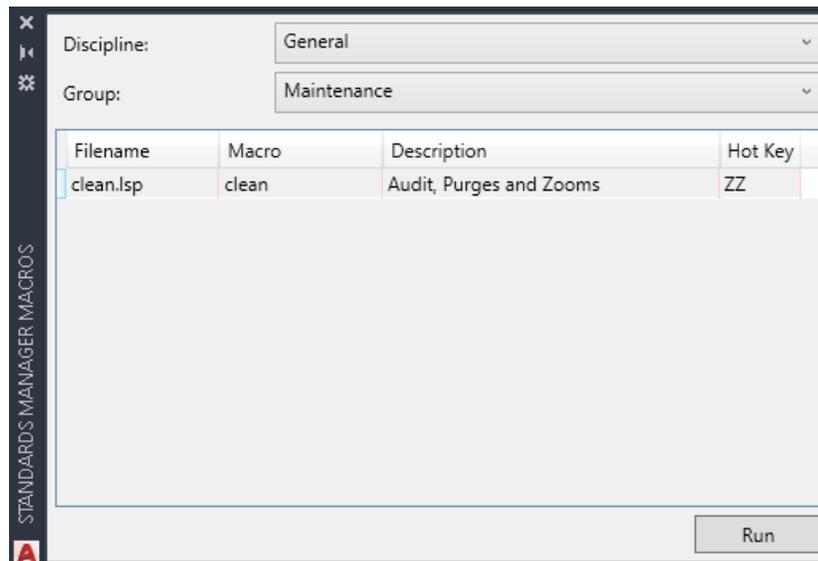
	A	B	C	D	E	F	G	H
1	Drawing	Entity Count	Layer	Description	Lineweight	Color	Linetype	Violations
2	EX-TOPO	161	C-TOPO-MAJR	Civil Topography Major Lines / Contours	0.02 (0.5 mm)	144	CONTINUOUS	Lineweight:LineWeight050 LineWeight025,Color:144 9
3	EX-TOPO	301	C-TOPO-MINR	Civil Topography Minor Lines / Contours	0.014 (0.35 mm)	52	CONTINUOUS	Lineweight:LineWeight035 LineWeight025,Color:52 11
4	riolosub	176	C-ANNO-TBC	Civil Annotation User Defined	0.01 (0.25 mm)	30	CONTINUOUS	Color:30 40
5	riolosub	26	C-BLIN-STAN	Civil Baseline Stationing	0.014 (0.35 mm)	32	CONTINUOUS	Color:32 192

Excel will automatically open with the entries listed. Any violations will be highlighted and described in the violation column in the spreadsheet. This file can be saved or discarded per the user.

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. Custom programs can be accessed from the **Load Custom Program** command from the **Standards Manager** ribbon tab.

The Standard Manager Macros palette appears.

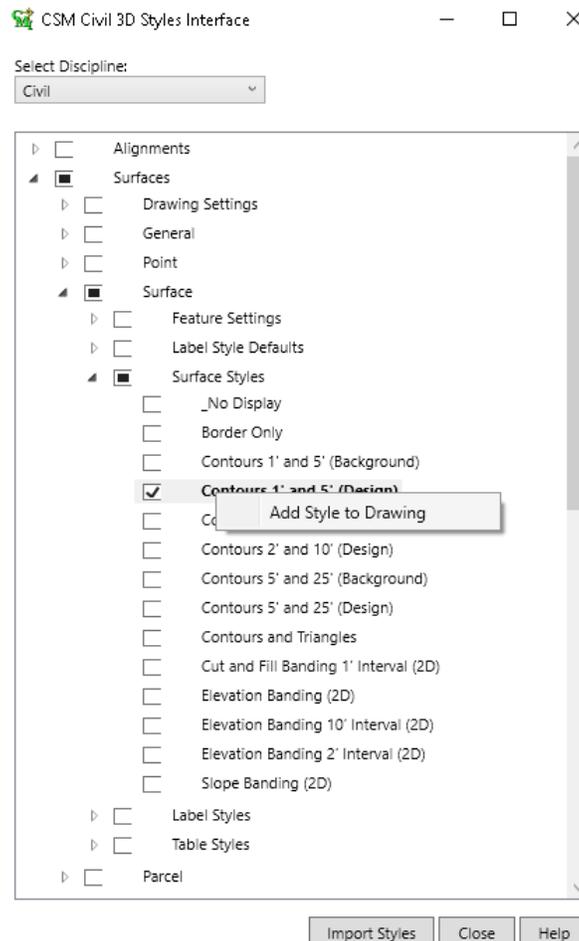


Load a custom macro/program into AutoCAD by first selecting the desired **Discipline** and **Group**. The programs associated with that discipline and group appear within the list below the group name. The program file name is listed in the left most column. If the program has internally defined commands 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 **Run** button.

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 **Civil 3D Styles/Settings** command from the **Standards Manager** ribbon tab 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.

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 bottom left corner of the dialog box will describe the styles being imported (see figure below).

