



CATIA V5 Administration Customizing Standards



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Executive Summary

It is possible to operate CATIA V5 with the default Standards delivered with the code, but many companies customize these Standards to best meet their needs and the needs of their suppliers. Supply Chain companies may also have to manage a multitude of Standards from other companies along with their own. This session discusses the steps required to customize the different Standards, and how to share the customized Standards across multiple environments and workstations. This session is intended for the beginner to intermediate System Administrator, in order to create a working knowledge of the setup of custom Standards.

Introduction

The administration of your CAD/CAM systems is an important part of the overall ability to keep your workstations effective in your business. Because many companies create their own custom Standards to best meet their needs and the needs of their suppliers, a clear understanding of how to customize the Standards is required. This paper focuses on the information that is necessary for maintaining CATIA's Standards. Having this knowledge allows the Administrator to easily set up specific company Standards, and to manage Standards received from other companies.

This document is intended for Systems Administrators, Application Engineers, and Technical Support Members who require an understanding of V5 Client Administration. The information was generated from the experiences gained through many years of experience of administering CATIA V5 and on many installs of various levels of the V5 code. Although this paper is written based on a V5R21, it is applicable to all levels of V5.

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1. What are the CATIA V5 Standards?

The CATIA V5 Standards define default values for the element properties within CATIA. Although CATIA can be used out of the box with the Standards defined by Dassault Systèmes, many companies do customize these Standards, to meet their needs and the needs of their suppliers. Because of this, it is important to understand the CATIA V5 Standards and how to manage them.

1.1. Types of Standards

There are two types of Standards available with CATIA V5. The first type is the **Default Standards**, which are provided by Dassault Systèmes. These files are stored in the location defined by the environment variable **CATDefaultCollectionStandard**. The default location of the default Standards is set by Dassault Systèmes to be the “install folder”\resources\standard. Figure 1 shows an example for the default Windows folder for a Windows 64-bit installation:

C:\Program Files\Dassault Systemes\B21\win_b64\resources\standard

The second type of Standards available is the user Standards, which are also known as the **Customized Standards**. These Standards are created and managed by the CATIA V5 Administrator. They are stored in the location defined by the environment variable **CATCollectionStandard**. By default, a value for this environment variable is not defined, as shown in Figure 1.

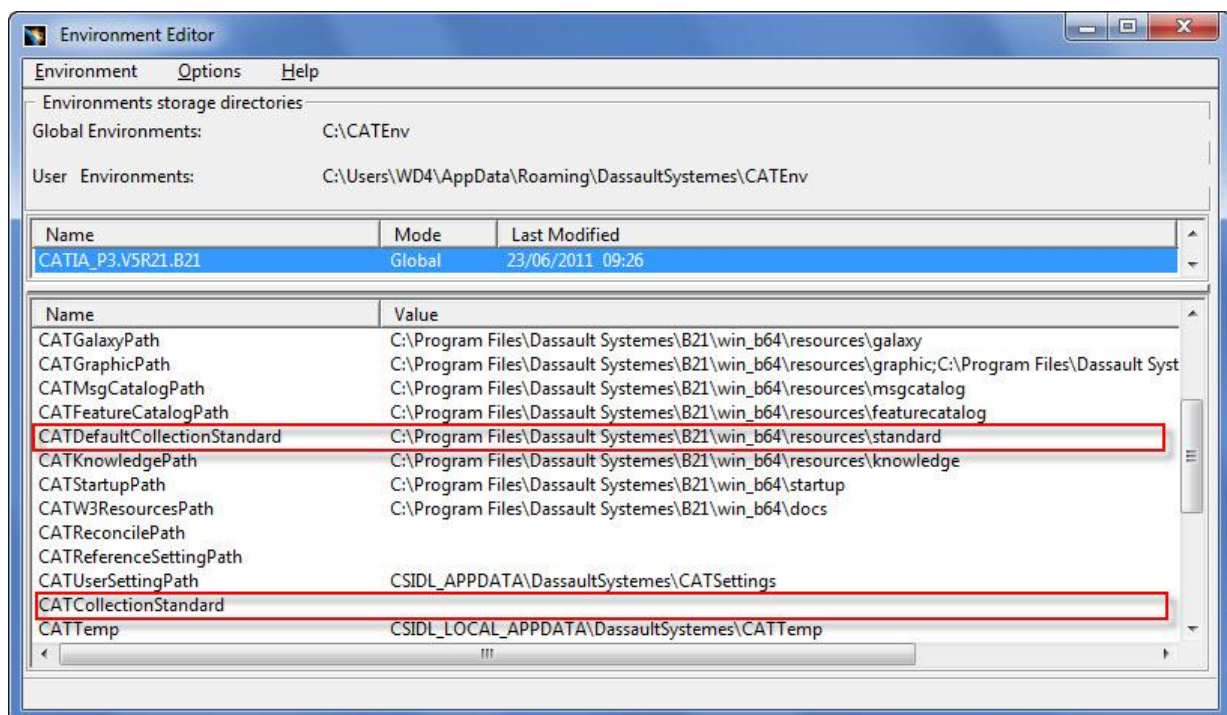


Figure 1 – V5R21 Environment Editor

1.2. Categories of Standards

There are four different categories of Standards. Each of the Standards affects different CATIA documents and workbenches. All of the CATIA V5 Standards are defined in XML (Extensible Markup Language) files, with specific names, and are stored in specific locations in the Standards directory structure.

The XML file names of the Standards cannot be modified or customized with the exception of the Drafting Standards. Only Drafting Standards can have customized XML file names. All of the other Standards must keep use the names as defined by Dassault Systèmes.

The following sections describe the four Standards categories that are used to define CATIA's attributes.

1.2.1. General Parameters

The General Parameters Standards include the following four areas:

Default Graphic Attributes

These Standards define the general graphic attributes for Lines, Points, Planes, Surfaces, Solids and OpenBodies. They are defined in the file **CATStdMechanicalDesign.xml**. This file by default is not provided in the "install folder"\resources\standard directory. To be able to modify it, it must be copied from the "install_folder"\startup\templates directory into the "install_folder"\resources\standard directory.

Attributes for Layers and Filters

These Standards define attributes for Layers and Filters in CATParts and CATDrawings. For CATParts, the Standards are defined files **CATStdLayersAndFilters.xml** and **CATStdTypeLayerAndFilter.xml**.

For CATDrawings, the Standards are defined in the files **CATDftStdLayersAndFilters.xml** and **CATDftStdTypeLayerAndFilter.xml**.

These files are located in the directory "install_folder"\resources\standard

Note: Even though the xml files **CATStdTypeLayerAndFilter.xml** and **CATDftStdTypeLayerAndFilter.xml** are located in the directory "install_folder"\resources\standard (as shown in Figure 2), they are not editable as they are used only to define the layer and definition types used in the **CATStdLayersAndFilters.xml** and **CATDftStdLayersAndFilters.xml** files.

Attributes for Line Thickness

These Standards define Line Thickness attributes for CATDrawings. The Standards are defined in the file **CATStdLineThickness.xml**, which is located in the directory "install_folder"\resources\standard

Aerospace SheetMetal Design Parameters

These Standards define the graphic properties and visibility characteristic curves used in the Aerospace SheetMetal Design Workbench. The Standards are defined in the file **AerospaceSheetMetalDesign.xml**, which is located in the directory **“install_folder”\resources\standard**

1.2.2. Drafting Parameters

These Standards define the attributes for drafting elements used in CATDrawings for the Interactive and Generative Drafting Workbenches.

Dassault Systemes provides eleven Drafting Standard files. One for each of the four International Drafting Standards, two Drafting Standards and five for use with other workbenches.

The International Drafting Standards files provided with the CATIA V5 code are:

ANSI.xml, ASME.xml, ISO.xml and JIS.xml

The Drafting Standards based on the ISO 3098 standard defining text representation are:

ISO_DS.xml and JIS_DS.xml

The remaining five Drafting Standards are specifically intended for the 2D Layout for 3D Design, and the Functional Tolerancing and Annotation workbenches. They are based on the Drafting Standards and are suffixed with **_3D** (for Drafting) or with **_3D_DS** (for Functional Tolerancing and Annotation). In these specific Standards, the colors, for example, have been customized for optimized display. The five Standards are provided in the files:

ASME_3D.xml, ISO_3D.xml, ISO_3D_DS.xml, JIS_3D.xml, and JIS_3D_DS.xml

All of the Drafting Standard files are located in the directory **“install folder”/resources/standard/drafting**

1.2.3. DFX/DWG Interface Parameters

These Standards define the import mapping parameters of element properties for DXF/DWG files coming into CATIA V5.

They are defined in the **DXF.xml** file. This file is located in the directory:

“install folder”\resources\standard\dxdf

1.2.4. Generative Parameters

These Standards define the Generative View Styles in the Generative Drafting Workbench.

They are defined in the file **DefaultGenerativeStyle.xml**. This file is located in the directory

“install folder”\resources\standard\generativeparameters

Figure 2 shows the default **standard** directory for Windows workstations. All of the General Parameters Standards XML files are shown in this directory, as well as the sub-directories **drafting**, **dxf** and **generativeparameters**.

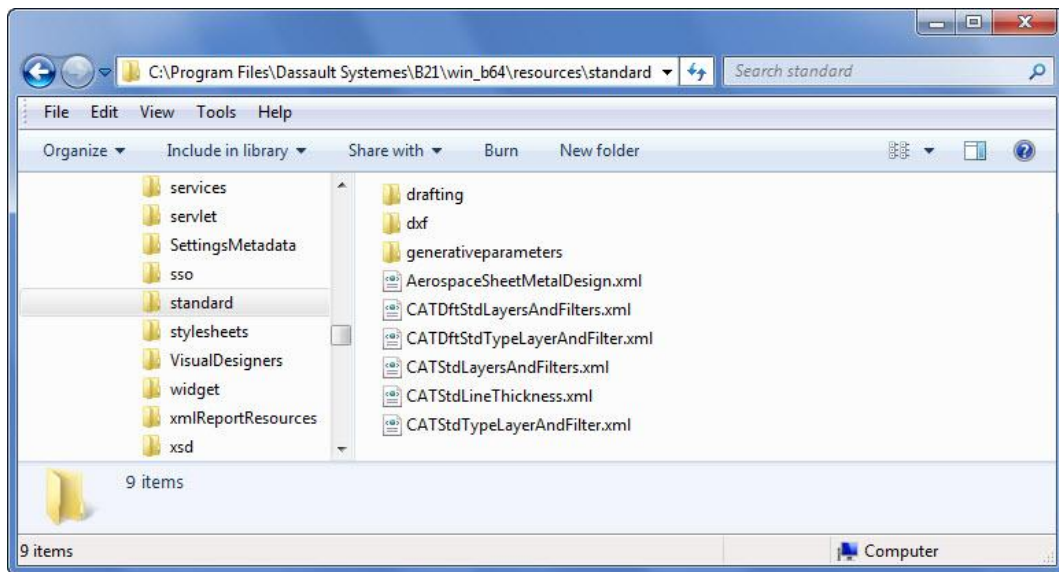


Figure 2 - Default standard directory - Windows

Figure 3 shows the contents of the default **drafting** sub-directory for Windows. All of the Drafting Standards are stored in this sub-directory.

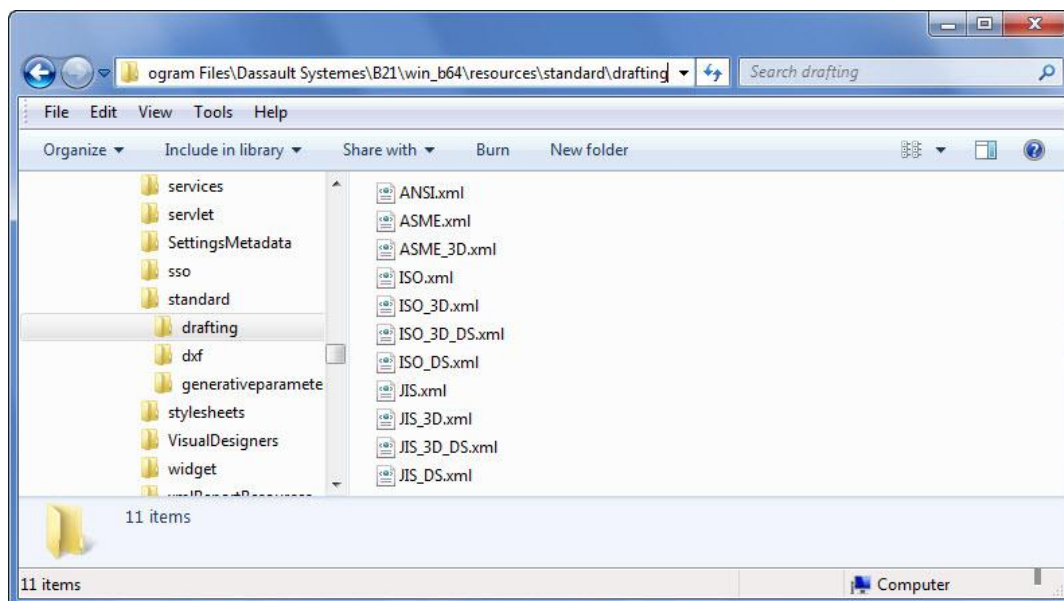


Figure 3 - Default drafting sub-directory - Windows

Figure 4 shows the contents of the default **dxf** sub-directory for Windows. The DXF/DWG Standards are stored in this sub-directory.



Figure 4 - Default dxf sub-directory - Windows

Figure 5 shows the contents of the default **generativeparameters** sub-directory for Windows. The Generative View Styles Standards are stored in this sub-directory.

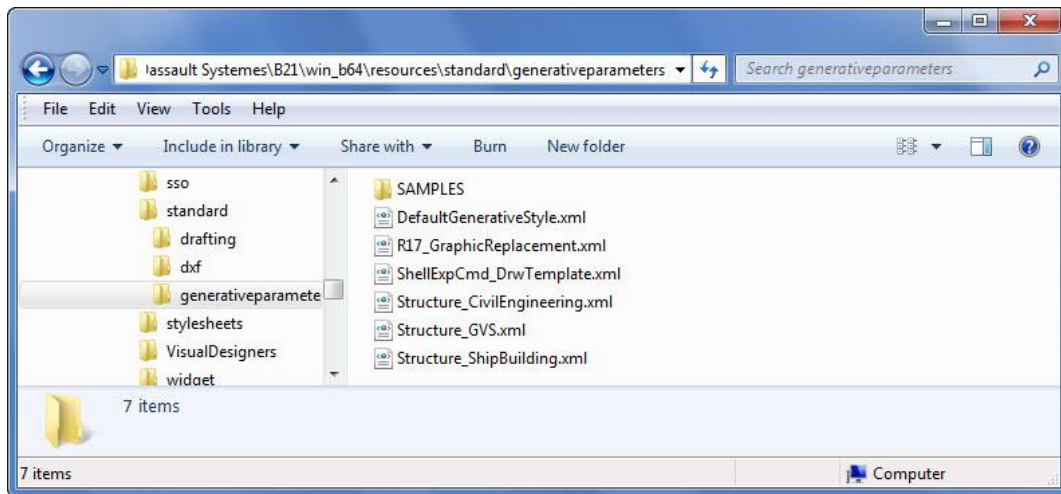


Figure 5 - Default generativeparameters sub-directory - Windows

The AIX directory structure is the same as that of Windows except for the default path name. Since V5R21 is only supported on UNIX (for servers only), this directory is: **/usr/DassaultSystemes/B21/aix_a/resources/standard.**

2. Customizing the CATIA V5 Standards

In order to modify the CATIA V5 Standards, four steps must be taken. The first step is to define the environment variables needed to edit the Standards. After these variables are defined, then their supporting directory structure needs to be created. Once these first two steps are done, the Standard Definitions must be accessed in CATIA V5 admin mode. Finally, the default Standards must be selected, modified, and saved in the appropriate customized Standards directory.

The following sections provide detailed procedures for performing these four steps.

2.1. Define Environment Variables

There are two environment variables that must be set up in the CATIA Environment Editor: **CATCollectionStandard** (discussed earlier) and **CATReferenceSettingPath**.

The **CATReferenceSettingPath** variable defines the directory that stores the Administrator's CATSettings files that are created when modifying CATIA settings in admin mode. This is also the location for the storage of the locked CATSettings.

To define the values for the **CATCollectionStandard** and **CATReferenceSettingPath** environment variables, use the Environment Editor, as follows:

- For Windows, as a user with Administrator authority, the Environment Editor is started using: **Start - All Programs - CATIA - Tools - Environment Editor**
- For AIX, as root, the Environment Editor is started by issuing the following command: **/usr/DassaultSystemes/B21/aix_a/code/command/catstart -run CATIAENV**
- Once the Environment Editor Window opens, select **OK** for the Administrator Warning Message.
- If there is more than one environment defined, select the one that you will work with, and select **OK** again for the Administrator Warning Message.
- For the environment variable that needs to be modified (**CATCollectionStandard** and **CATReferenceSettingPath**), right-mouse-click on the variable name to access its contextual menu, and then select **Edit**.
- Assign values to the environment variables, and make their path names meaningful. For example, define **CATCollectionStandard** as **C:\CATIAV5R21\standard** (for Windows) or **/CATIA/V5R21/standard** (for AIX). For **CATReferenceSettingPath**, define it as **C:\CATIAV5R21\catlock** (Windows) or **/CATIA/V5R21/catlock** (AIX).
- After making the modifications to the environment, save the changes by right-mouse-clicking in the lower pane of the Environment Editor Window to access its contextual menu, then select **Save**.

Figure 6 shows the Environment Editor with these two variables added.

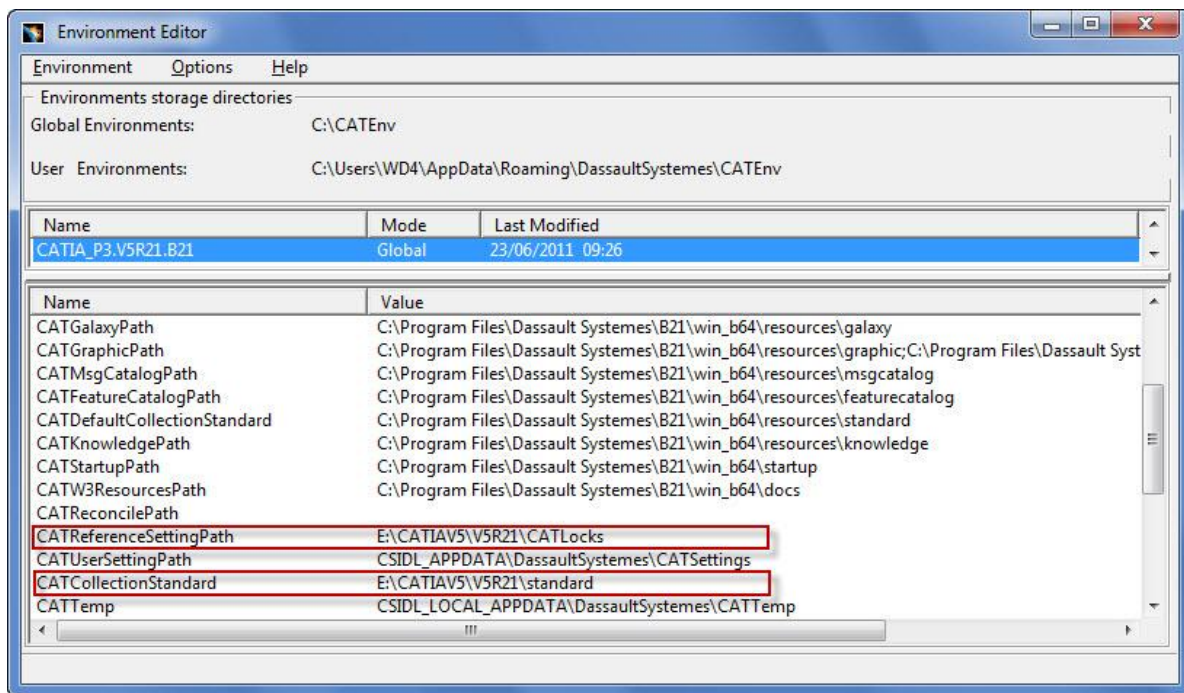


Figure 6 - V5R21 Environment Editor: variables defined (Windows)

2.2. Create the Required Directories

Once the environment variables **CATReferenceSettingPath** and **CATCollectionStandard** have been defined, their directories need to be created. The structure for **CATCollectionStandard** should mimic the structure of the default Standards directory **CATDefaultCollectionStandard**.

Create the directory **E:\CATIAV5\V5R21\CATLock** (for Windows) or **/CATIAV5/V5/CATLock** (for AIX) to support the environment variable **CATReferenceSettingPath**.

Also create **E:\CATIAV5\V5R21\standard** (for Windows) or **/CATIAV5/V5R21/standard** (for AIX) to support the environment variable **CATCollectionStandard**. Add the sub-directories **drafting**, **dxf** and **generativeparameters** as needed based on the Standards that you plan to customize. For example, this customized Standards structure for Windows would look like Figure 7.

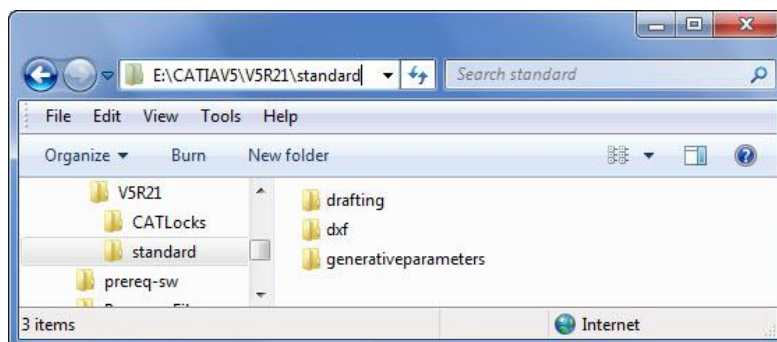


Figure 7 - Customized Standard directory Structure - Windows

Tip: If the default Standard directory (defined by **CATDefaultCollectionStandard**) and the Customized Standard directory (defined by **CATCollectionStandard**) both contain an identically-named Standard, the Customized Standard will take precedence and will be used.

2.3. Access of Standard Definitions in CATIA V5 admin mode

Log into CATIA and access the Standard Definitions using **Tools - Standards**. Because you didn't log into CATIA in admin mode, you will only be able to view the Standards. The Standard values are not editable (they are "grayed-out"), and the **Save As New** icon is not selectable. This is shown in Figure 8.

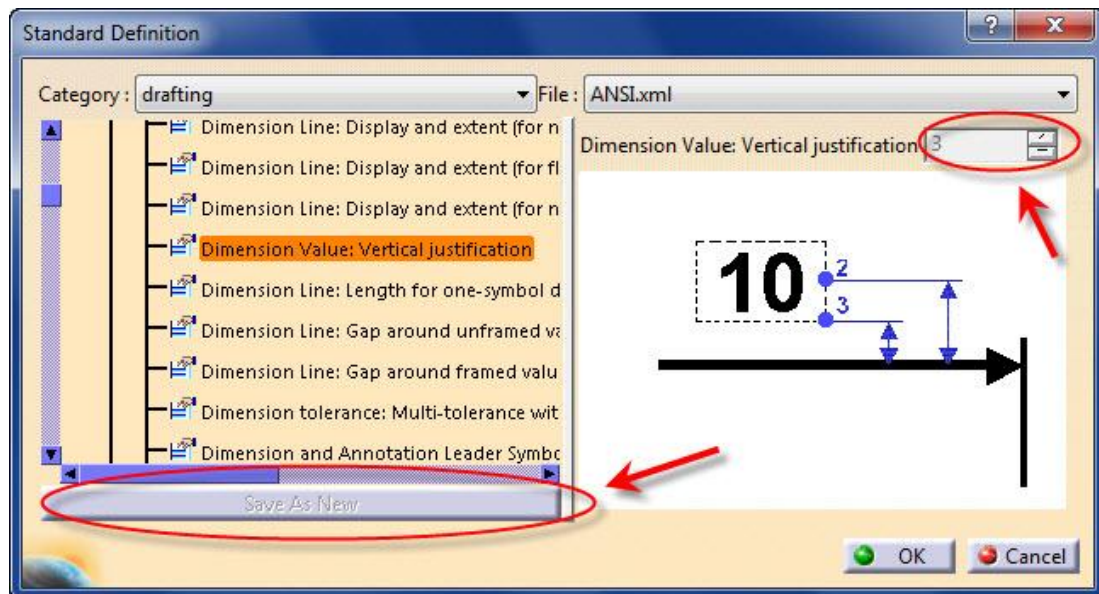


Figure 8 - Standard Definition Window - View Mode

To be able to edit (and save) the Customized Standards, you must log onto CATIA in admin mode. Since starting CATIA in admin mode has generated many questions at the Technical Support Center, the following steps show the requirements for successfully doing this.

To start CATIA in the admin mode, you must have Administrator/root authority, and you must have the environment variable **CATReferenceSettingPath** set to a directory owned by the Administrator/root.

Windows:

Open a command window and **cd** to the "**unload_directory**"\code\bin directory. For example, for Windows 64-bit, **cd** to **C:\Program Files\Dassault Systemes\B21\win_b64\code\bin** and issue the command:

cnext -admin

If you have multiple environments, the **-env** flag must also be used to specify the correct environment:

```
cnext -env CATIA.V5R21.B21 -admin
```

Additionally, if the directory path of the environment file is not the default

(**C:\ProgramData\DassaultSystemes\CATEnv** for Windows 7 64-bit), the **-direnv** flag must be used:

```
cnext -env CATIA.V5R21.B21 -direnv "C:\CATEnv" -admin
```

AIX:

In a UNIX window, **cd** to the **"unload_directory"/code/command** directory. For example, **cd** to **/usr/DassaultSystemes/B21/aix_a/code/command** and issue the command:

```
./catstart -run "CNEXT -admin"
```

Be sure to include the **-env** or **-direnv** flags if required, as noted above in the Windows example:

```
./catstart -run CNEXT -env CATIA.V5R21.B21 -direnv /CATEnv -admin
```

Once you log into CATIA in admin mode, the message shown in Figure 9 will appear:

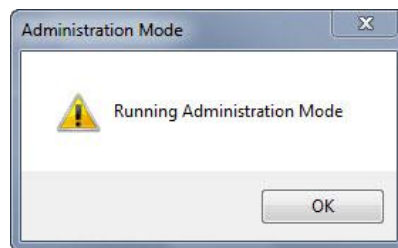


Figure 9 - Admin mode acknowledgement

After logging into CATIA in admin mode, re-access the Standard Definitions (**Tools - Standards**) and you will see that the Standard values are now editable, and the **Save As New** icon is now selectable. This is shown in Figure 10.

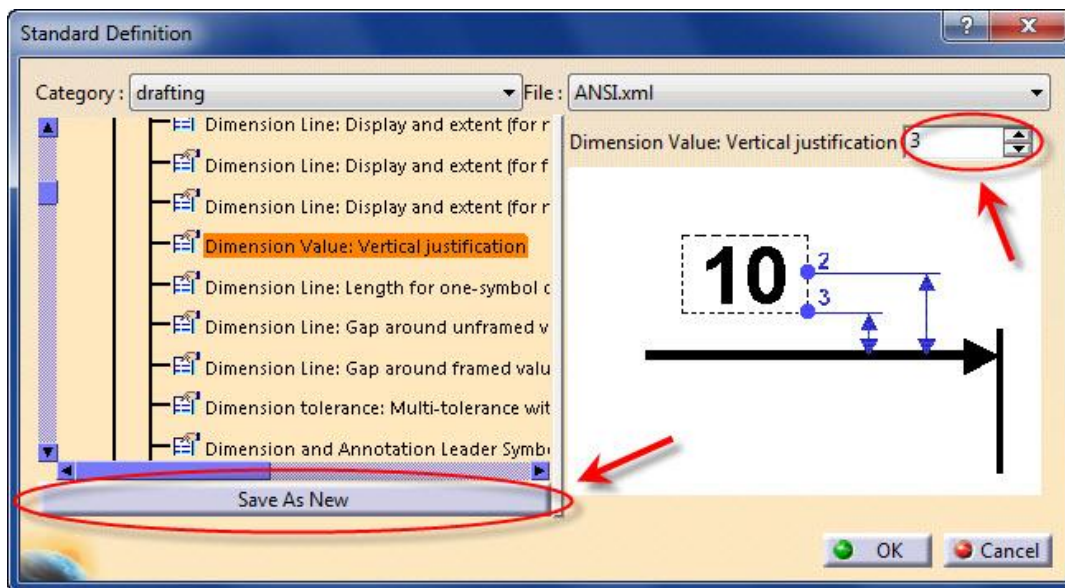


Figure 10 - Standard Definition Window - Edit Mode

2.4. Select a default Standard – Modify and Save

As mentioned earlier, the XML file names of the General Standards cannot be modified or customized. The Drafting Standards, DXF/DWG Interface Parameters and Generative Parameters can have customized XML file names.

If you want to change one of these standards, in the **Standard Definition** window (while logged into CATIA in admin mode) you would select category type, and start with an existing standard file. Make the required changes, and then select **Save As New** with a new name and it will be saved in the directory specified by the **CATCollectionStandard** variable. The new standard will be shown in the standard selection list as shown in Figure 11.

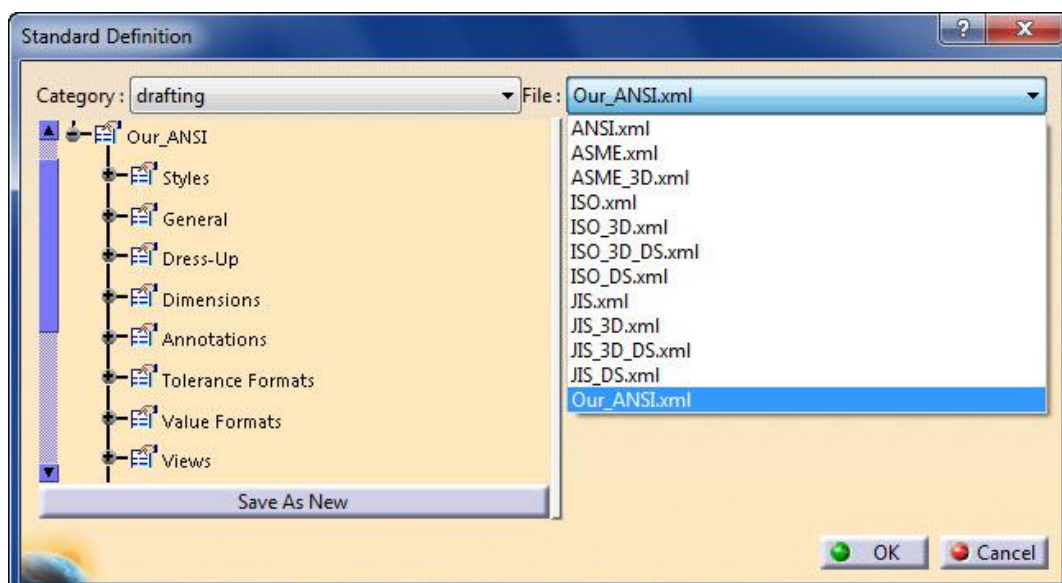


Figure 11 - New Customized Standard

If you want to change one of the General Standards, you will need to copy the file for the desired standard from the directory defined by **CATDefaultCollectionStandard** into the directory defined by **CATCollectionStandard**. If you modify the name, the file will not be recognized by CATIA, and the default one (without the modifications) will be used. Next, manually edit the copy of the XML file to make the desired changes. If the **CATStdMechanicalDesign.xml** file is not in the **CATDefaultCollectionStandard** directory, then you didn't copy it from the "install_folder"\startup\templates directory as discussed on Page 5 of this document.

Since the Drafting Standards are unique, the following steps are used to create a new company Drafting Standard:

- Log into CATIA in admin mode, and select **Tools - Standards** to open the Standard Definition window
- Under **Category** select **drafting**.
- Under **File**, select the International Standard that is closest in content to your company's requirements (**ANSI.xml**, **ASME.xml**, **ISO.xml**...)
- Expand the Standard Tree and edit the category that needs to be modified
- In the example shown in Figure 12, the **Dimension Line: Gap around unframed value** is being changed
- Continue through all categories in which your company varies from the selected International Standard and make the required modifications.
- Select the **Save As New** button and key a new name. This will save the new Standard in the **drafting** folder designated in the **CATCollectionStandard**.
- In this example, the Standard was saved as **Company_A.xml**

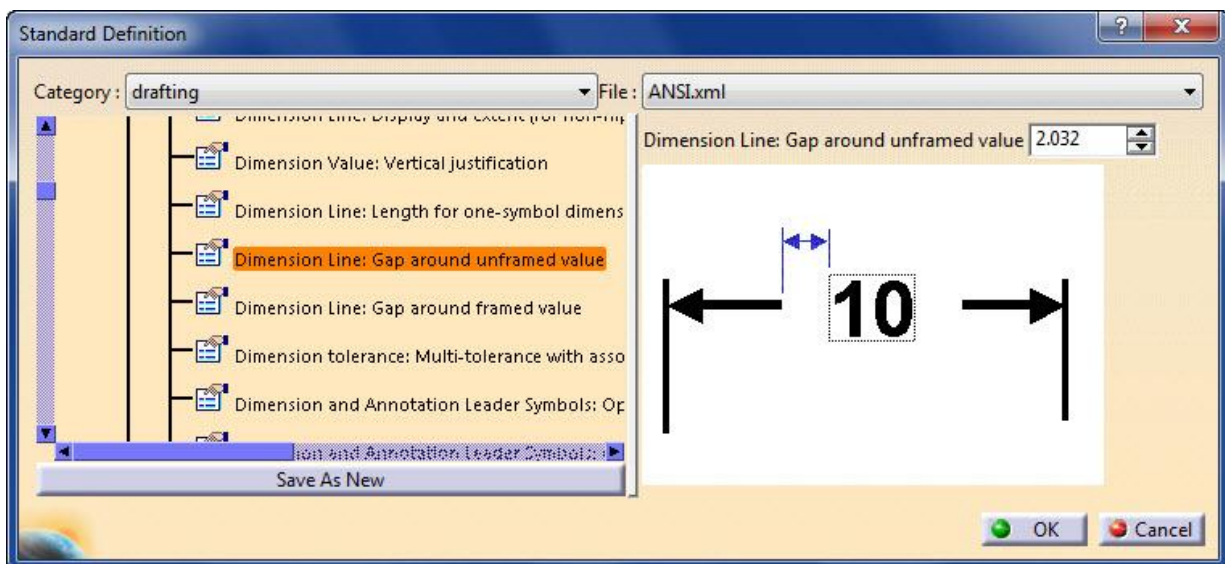


Figure 12 - New Drawing Window showing the New Drafting Standard

When a new Drawing is created, or when **File - Page Setup** is used to change the drawing's Standard, this new Standard will be available as one of the selectable Standards as shown in Figure 13.

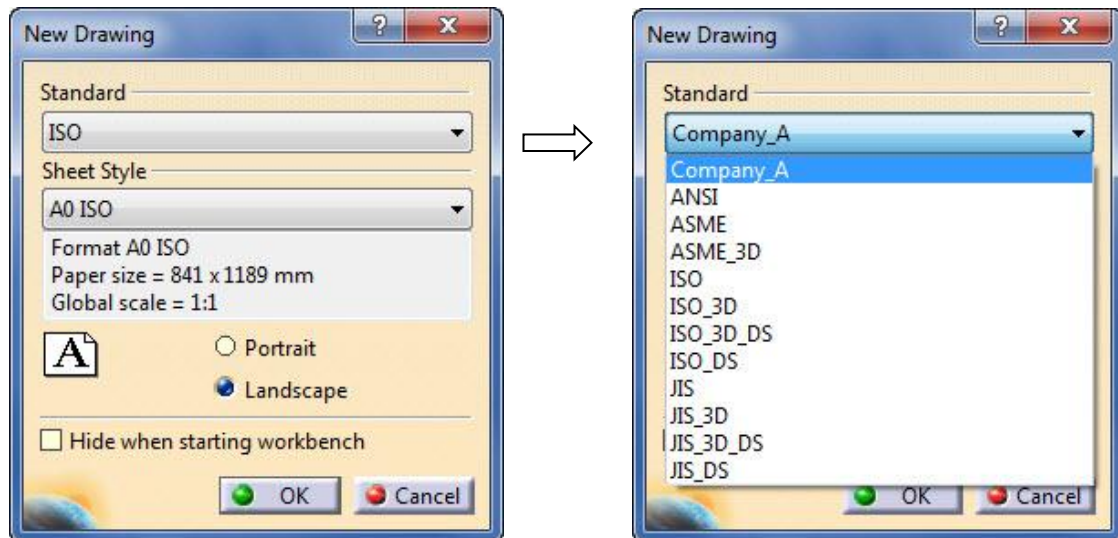


Figure 13 - New Drawing Window showing the New Drafting Standard

3. Managing the CATIA V5 Standards

3.1. Tips for Defining Drafting Standards Directories

Many customers who modify the CATIA Standards only modify the Drafting Standards. A typical mistake that is made when attempting to perform this modification is an incorrect definition of the environment variable **CATCollectionStandard**. The problem occurs when the variable is set up to point to the **drafting** sub-directory instead of to the **standard** directory: e.g., it has been set to **E:\CATIAV5\V5R21\standard\drafting** when it should be set to **E:\CATIAV5\V5R21\standard**

3.2. Tips for Controlling Multiple Customized Drafting Standards

If you have multiple CATIA environments in order to support different projects or customers, each having a unique requirement for Drafting Standards, you will likely want a method for enforcing use of the correct Drafting Standard for each environment. To do this, the following three steps should be taken:

1. Set unique values for the variables **CATReferenceSettingPath** and **CATCollectionStandard** in each environment, and create the directory structures to support these variables.
2. Create new Drafting Standards for each environment, and save the XML files into **CATCollectionStandard**, in the **drafting** sub-directory. An alternative to building your own customized Drafting Standard is to obtain a copy of your customer's Drafting Standard XML file, and place it into **CATCollectionStandard drafting** sub-directory.

3. Make the Default Drafting Standard XML files unavailable in the default **drafting** sub-directory:

On Windows 64-bit: Move all the XML files in the directory

C:\Program Files\Dassault Systemes\B21\win_b64\resources\standard\drafting

to:

C:\Program Files\Dassault Systemes\B21\win_b64\resources\standard\drafting\original

Doing this keeps the original Standard files available for future reference, but makes them unavailable to interactive CATIA.

Once these three steps have been followed, the user will **only** see the Customized Standards that are available in the **CATCollectionStandard** directory when creating a new CATIA Drawing. This is shown in Figure 14. The only selectable Standard available in this example is the new Standard **Company_A.xml**. This prevents an incorrect Drafting Standard from being used.

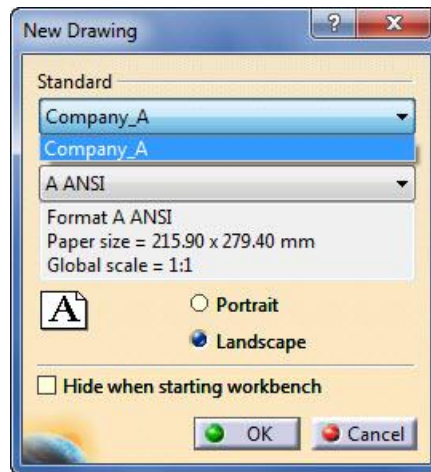


Figure 14 - New Drawing Window showing ONLY the New Drafting Standard

3.3. Editing the Standard XML files using a Text Editor

All standard XML files can be edited using a text editor instead of using the CATIA interactive editor. But, whenever possible, edit the standards using the CATIA interactive editor instead of a text editor because it is an easy-to-use graphical editor that allows the values to be edited correctly.

To start, place a copy the required XML file from the directory defined by **CATDefaultCollectionStandard** into the directory defined by **CATCollectionStandard**. Edit this file to show the contents. Search through the large file and find the attribute and change the value to the desired quantity. Save the file, and the change will be applied to the next CATIA document that is created. Figure 15 shows the ANSI.xml file with the attributes for value of the Dimension Value: Vertical Justification (as shown back in Figure 10).

```

<?xml version="1.0" ?>
- <std:node name="drafting" xmlns:std="http://www.dsweb.com/std">
- <std:enumdef name="YesNo">
  <std:strval>No</std:strval>
  <std:strval>Yes</std:strval>
</std:enumdef>
- <std:enumdef name="UpDown">
  <std:strval>Up</std:strval>
  <std:strval>Down</std:strval>
</std:enumdef>
- <std:enumdef name="ThreadRep">
  <std:strval>Circle</std:strval>
  <std:strval>Arc circle</std:strval>
</std:enumdef>
<std:floatval>0</std:floatval>
</std:node>
- <std:node name="DIMTxtJustif">
  <std:intval>3</std:intval>
</std:node>
- <std:node name="DIMLTail">
  <std:floatval>0</std:floatval>

```

Figure 15 - CATStdMechanicalDesign.xml File (Partially shown)

3.4. Updating Standards from previous CATIA Levels

With each new release of CATIA, the default Standards may change, with the addition and modification of some parameters, as well as new functionality being introduced. Because of these changes, the Standards XML files might not be upward compatible. If you have customized your Standards in previous levels of CATIA and want to use the customization in the current level, the XML files may need to be upgraded.

If you have customized non-drafting Standard files, you must re-customize them by starting with the new release level default Standards files, and editing the Standard to reflect your desired changes.

For the Drafting Standards, if you have customized only a small number the parameters, it may be easier to start with the new release level Standards files and re-customize them. If you have many customizations made for your Drafting Standards, then it may be easiest to use a batch utility to generate current XML files based on your customized Drafting XML files from the previous release. The batch utility **CATAnnStandardTools** provided with CATIA allows this conversion of the Drafting Standards only.

To upgrade the Drafting Standard XML files to the current version, issue one of the following commands:

CATAnnStandardTools UPGRADE_ALL "directory for upgraded XML files"

CATAnnStandardTools UPGRADE XXX "directory for upgraded XML files"

where **XXX** is the Drafting Standard name (such as **Company_A.xml**)

When running this batch tool with the **UPGRADE_ALL** argument, it searches for all of the Drafting Standard XML files in the drafting directories specified by the environment variables **CATDefaultCollectionStandard** and **CATCollectionStandard**. It then converts them to the upgraded XML file format, and places a copy in the directory specified.

When the **UPGRADE XXX** argument is used, the batch tool searches for the specific XML file in the Drafting Standard XML files in the drafting directories specified by the environment variables **CATDefaultCollectionStandard** and **CATCollectionStandard**. The tool then converts the XML file to the updated XML file format, and places a copy in the directory specified.

For Windows, this command is executed in a command window from the directory

C:\Program Files\Dassault Systemes\B21\win_b64\code\bin

For AIX, in a UNIX window cd to the directory

/usr/DassaultSystemes/B21/aix_a/code/command and execute one of the following commands:

./catstart -run CATAnnStandardTools UPGRADE_ALL "directory for upgraded XML files"

./catstart -run CATAnnStandardTools UPGRADE XXX "directory for upgraded XML files"

where **XXX** is the Drafting Standard name (such as **Company_A.xml**)

3.5. “Tools – Options” for controlling Standards

To further constrain the user’s ability to modify the Drafting and DXF Standards based on your company’s requirements, some of the Options under **Tools - Options** can be set and locked. To be able to lock the settings under **Tool - Options**, you must log onto CATIA in admin mode. The following two sections show the options available for these two Standards.

3.5.1. Drafting Standards Settings

These options are accessed under **Tools - Options - Mechanical Design - Drafting - Administration**, and are shown in Figure 16.

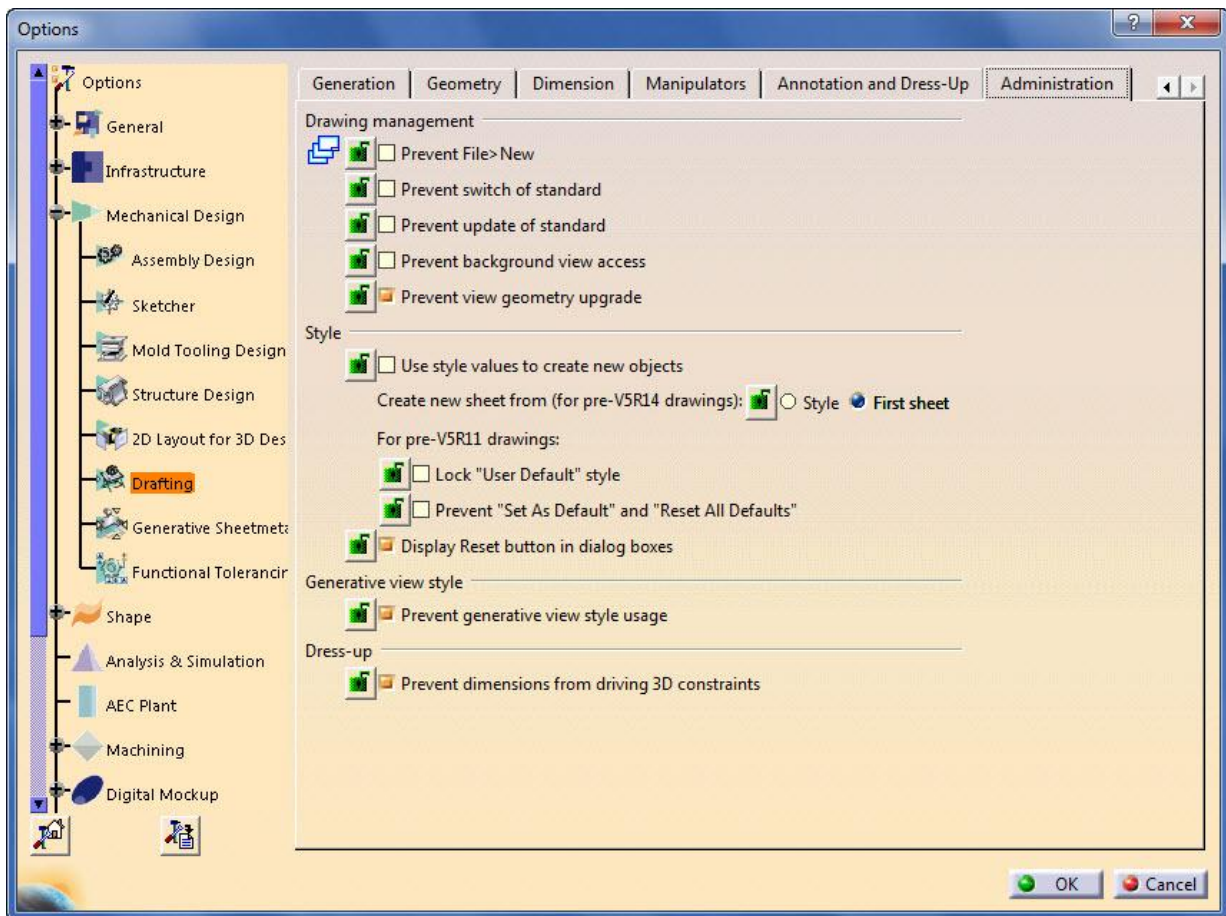


Figure 16 - Tools - Options for locking the Drafting Standards

3.5.2. DXF/DWG Standards Settings

These options are accessed under **Tools - Options - General - Compatibility - DXF**, and are shown in Figure 17.

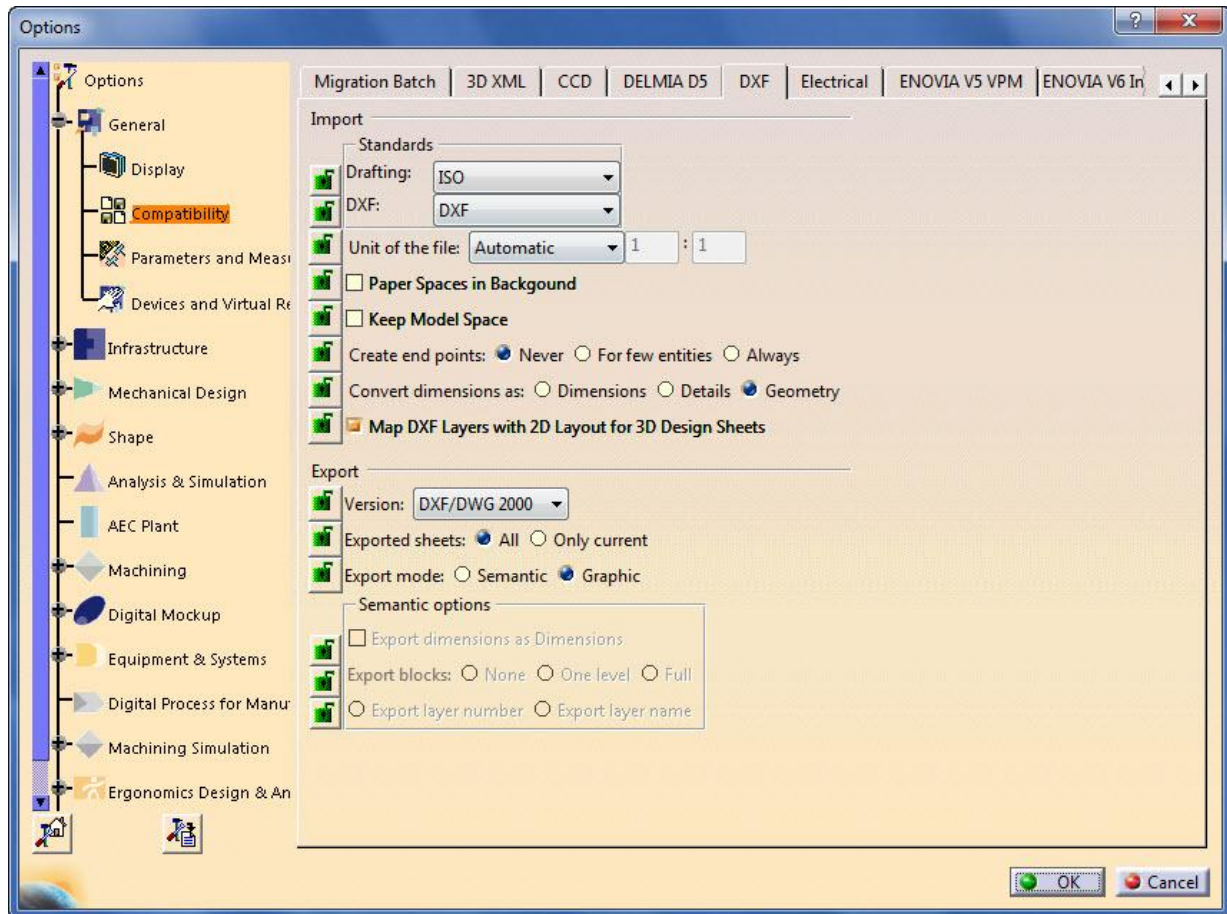


Figure 17 - Tools - Options for locking the DXF/DWG Standards

4. Sharing of Customized CATIA V5 Standards

When a CATIA document is created, the values of the parameters defined by the Standards in use at the time of the document creation are copied into the document. Since each document contains a copy of the Standards, the document is considered a standalone document, which allows it to be exchanged with other users, projects and companies, without having to send the Standards files along with the document.

If there is a need to create CATIA documents using a specific Standard, then using a new Standards file can be customized as outlined in this paper, or a copy of an already Customized Standard files can be obtained and placed into the directory structure defined by the environment variable **CATCollectionStandard**.

Once the directory structure for Standards has been created on a network accessible directory on one workstation, the directory structure can be set up as a Shared directory on Windows Workstations or a NFS exported directory on AIX workstations. An example of a directory structure for supporting multiple environments is shown in Figure 18. Here we see that each environment contains a sub-directory under the **V5R21** folder, and has a name that is based on the company name. Each of these Company directories contains a **CATLocks** sub-directory for holding the locked CATSettings for the environment, and a **standard** sub-directory, for holding the necessary Standards for the environment. In this example, **Company_A & Company_B** only have customized Drafting Standards, while **My_Company** has customized Standards for Drafting, DXF/DWG, and Generative Parameters

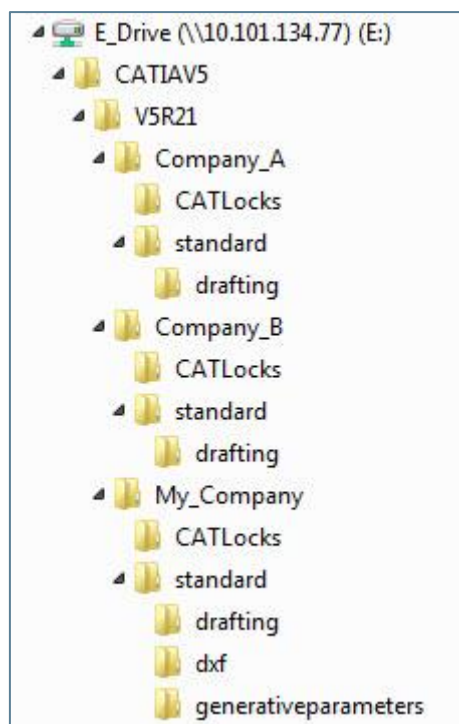


Figure 18 – XML Directory structure for Multiple Environments

On the Client workstations, this directory structure can be mapped for Windows, or NFS mounted for AIX. Each of the Clients must have different CATIA environments defined for each Company. In each environment, the environment variables **CATReferenceSettingPath** and **CATCollectionStandard** point to the appropriate mapped directory for Windows workstations, or to the appropriate NFS mounted directory for AIX workstations.

5. Conclusion

The ability to keep your company's CAD/CAM systems properly administered is an important part of your business. Learning how to modify and manage the default Standards provided with CATIA allows you to be able to work with your own Standards as well as the Standards of your customers.

It can be seen in this paper that there are several different types of Standards, which control the element properties within CATIA Documents. After understanding the different types, you were shown how to customize the default Standards to make you own customized versions, and then how to share your newly Customized Standards, and the Standards from your Customers across a network for use by all workstations.

6. References

CATIA V5 Documentation

Infrastructure – CATIA Infrastructure Users *Guide – Customizing Standards*

7. Revision History

August 2011 / Version 0.0 – original doc authored

March 2014 / Version 1.0 – updated to new template

8. Related Documentation

For additional information, refer to the following Q&A's from our Knowledge Base:

KA QA00000007852 - How to customize CATIA Mechanical Design Standard

KA QA00000004955 - Are Drafting Standards (.xml) files language/OS dependent

KA QA00000004094 - How to change ANSI dimension standard for drafting workbench

9. Document History

Document Revision	Date	Revised By	Changes/Notes
0.0	Aug 2011	A3R	Original
1.0	Mar 2014	A3R	Updated to new template

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