



CATIA V5 Training Exercises

V5 Administration

Version 5 Release 18
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EDU_CAT_EN_ADM_FX_V5R18

Student Notes:

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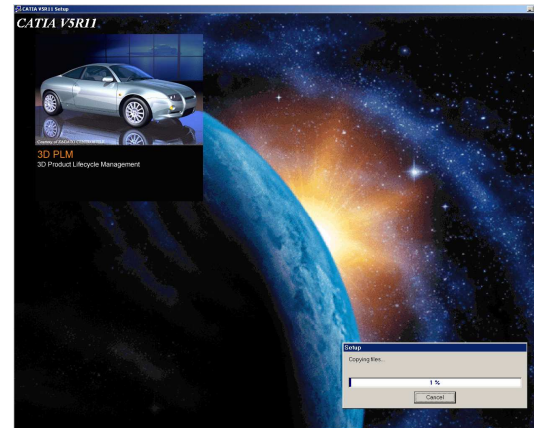
Exercise 1

CATIA V5 Installation on Windows



This exercise explains you how to :

- Install CATIA V5
- Install CATIA V5 documentation
- Start CATIA V5
- Have access to CATIA V5 documentation
- Check CATIA V5 installation



Do It Yourself

What you will have to do :

1. Install CATIA V5R17 on Windows from CD-ROM

with default values, except for :

- Destination code folder : C:\DS\CATIA_V5R17
- Environment folder : C:\DS\CATEnv
- AL2 configuration

2. Install the CATIA V5R17 documentation from CD-ROM

- Destination documentation folder : C:\DS\CATIA_V5R17doc

3. Launch CATIA V5R17 in different ways

- Shortcut, Run, command line, ...

4. Launch CATIA V5R17 Documentation

- Search the item “CATIA Infrastructure Hardware requirements”
- Try to launch CATIA online documentation from CATIA session, with F1 and Shift-F1 keys

5. Check CATIA V5 integrity with Software Management V5R17

6. Check the installation with the log file






Exercise 2

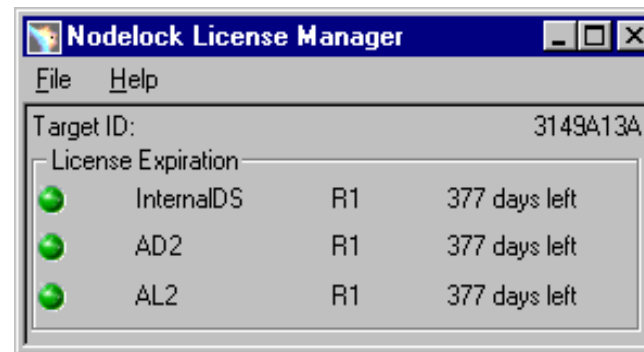
Licensing



60 min

This exercise explains you how to :

-  **Check nodelock keys**
-  **Configure a LUM client**
-  **Check concurrent keys**
-  **Try the dynamic licensing**
-  **Define an offline license server**



Do It Yourself (1/2)

What you will have to do :

1. Nodelock licenses

- Check nodelock keys validity with Nodelock Key Management (CATNodelockMgt)

2. Concurrent licenses

- Define a license server
- Enroll concurrent license
- Define a LUM client with “**LUM → Configuration Tool**”
- Check Network License Client
- In “direct binding”, add the name of a LUM server
- Check all available concurrent licenses with “**LUM → Basic License Tool**”
- Check all available concurrent licenses with LUM command lines in a DOS shell, using : C:\ifor\WIN\BIN\i4blt -lp -l

3. Dynamic licenses

- From a CATIA session reserve a product license like GSD
- Check the workbench and toolbar changes
- Free the license

Do It Yourself (2/2)

What you will have to do :

4. Concurrent offline Licensing

- Migrate concurrent licenses to concurrent offline on the server
- Define the authorizations and the rules on the licenses
- Extract a license from a Windows machine
- Try the license with the machine disconnected
- Restitute the license
- Check the license server

Exercise 3

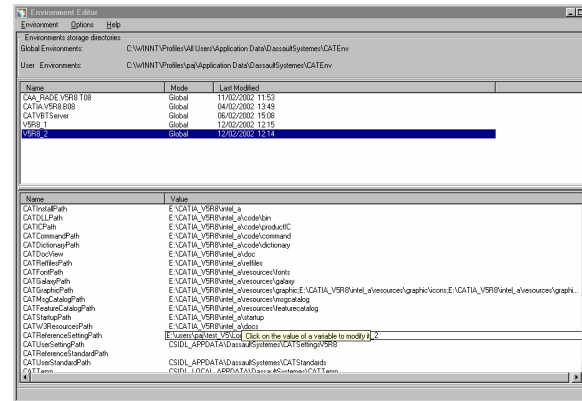
Environment Management



20 min

This exercise explains you how to :

-  Create a new global environment
-  Customize an environment
-  Launch CATIA in admin mode
-  Create environments with command lines
-  Create new variables



Student Notes:

Do It Yourself (1/2)

What you will have to do :

1. Creating a new environment with the Environment Editor

- **Create a new global environment with the Environment Editor**
Name : global_env
Mode : global
Product Line : CATIA
- **Customize this new environment, modifying the following values:**
CATUserSettingPath = C:\DS\SettingsV5R17
CATDocView = C:\DS\CATIA_V5R17doc
- **Launch CATIA with the new environment**
Look at the folder : C:\DS\SettingsV5R17
- **Launch CATIA in admin mode with the new environment**
What happens ?
- **Launch again the Environment Editor and define the value**
CATReferenceSettingPath = C:\DS\LockV5R17
Launch CATIA in admin mode

Do It Yourself (2/2)

2. Creating a new environment with command lines

- Create a global environment by a command line

Name : global_env2

Mode : global

Product Line : CATIA

Environment folder : C:\DS\CATEnv

- Change the value of the CATDocView variable of the global_env2 environment by a command line
- Read the “global_env2” environment
- Delete the “global_env2” environment with command line

3. Creating a new variable

- Launch again the Environment Editor and add a new variable in “global_env”

Variable : Data1

Value : C:\DS\Data1






Exercise 4

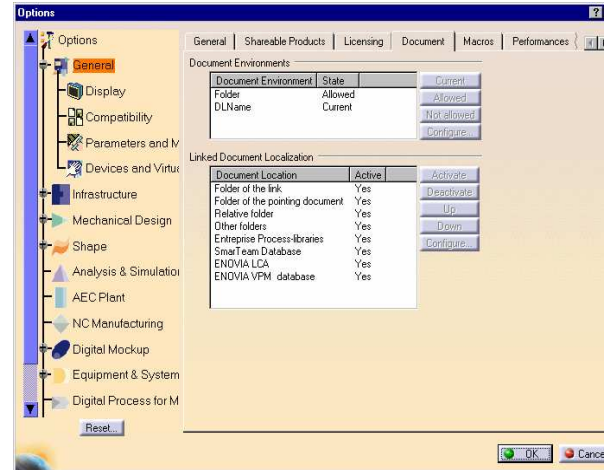
Settings Management



60 min

These exercises explain you how to :

-  Customizing general settings
-  Resetting settings
-  Locking the settings in a global environment
-  Creating a two levels company settings
-  Managing settings with VB script



Do It Yourself (1/4)

What you will have to do :

1. Locking settings in a global environment

- **Create a new global environment “env1” with the Environment Editor**
- **Customize this new environment, modifying the value:**
 - CATReferenceSettingPath = C:\DS\LockSettings1
 - Create the folder C:\DS\LockSettings1 with read/write access rights for the administrator and read for end users
- **Launch CATIA in admin mode with the environment env1:**
- **Lock some settings (padlock) for instance :**
 - General → Licensing tab : AL2
 - General → User Interface style : P2
- **Give default values (no padlock)**
 - General → Display → Visualization tab : Background
- **Launch CATIA V5 in normal mode**
 - Try to modify some options locked by administrator
 - Check that default values set by the administrator can be changed
 - Reset the values and check the setting set by default at previous item

Do It Yourself (2/4)

2. Creating a two levels company settings

- Create a new global environment “env2” with the Environment Editor
- Create in this environment two levels of administration:

1st: the level of admin created at the previous step C:\DS\LockSettings1

2nd: A new level of admin pointing to C:\DS\LockSettings2

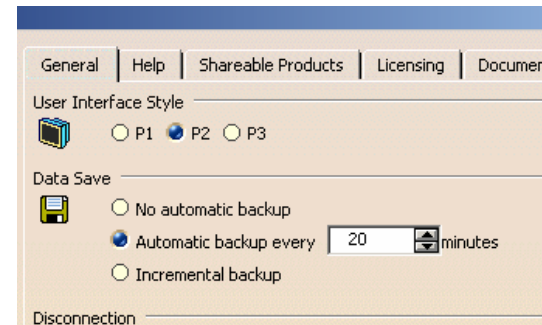
Don't forget to set the access rights accordingly

- **Launch CATIA in admin mode in env2**
Lock some settings in env2, for instance :
 - General → Document Environment : DLName Allowed + currentSet some settings by default
Try to modify some settings locked in env1
- **Launch CATIA with mode normal**
Check the settings locked at two levels of admin
Reset the settings and look at the changes
- **Check the different cases described in the course**
(see Sharing settings – How CATIA reads the settings)

Do It Yourself (3/4)

3. Managing settings with VB Script

- Dump the General/General tab page with **Dump Macro** command.
Have a look on the result file.
Rename the file to compare it later
- Record a macro interactively to set the automatic Data Save at 20mn
- Alter manually this macro to set the delay of the automatic save to 15mn (parameter unit = ms)
- Add to the last macro the lock of the automatic save:
(SetAutoSaveLock=TRUE)
- Launch the macro:
→ in the admin mode
→ in batch
- Dump once again the General/General tab page with **Dump Macro** command
- Compare the 2 dump macros.



Do It Yourself (4/4)

4. Managing settings with VB Script (...)

- The macro to execute is:

```

Language="VBSCRIPT"

Sub CATMain()

Set settingControllers1 = CATIA.SettingControllers

Set generalSessionSettingAtt1 = settingControllers1.Item("CATCafGeneralSessionSettingCtrl")

generalSessionSettingAtt1.AutoSave = AutoBackupEvery

generalSessionSettingAtt1.TimeRoll = 900000

generalSessionSettingAtt1.SetAutoSaveLock True

generalSessionSettingAtt1.SaveRepository

End Sub

bSTR3 = ""
Dim bSTR4
bSTR4 = ""
Dim boolean2
boolean2 = generalSessionSettingAtt1.GetAutoSaveInfo(bSTR3, bSTR4)
'-----
' Parameter 1 : (String) "Set at Admin Level 0"
' Parameter 2 : (String) "Locked at Admin Level 0"
' Returned value : (Boolean) False
'-----

Dim long3
long3 = generalSessionSettingAtt1.TimeRoll
'-----
' Returned value : (Long) 900000
'-----

```

- The Dump result



Exercise 5

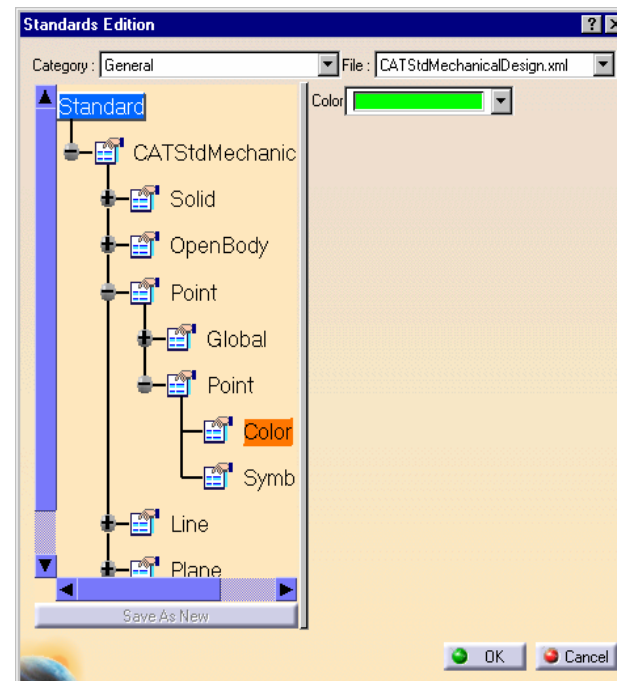
Standards



15 min

This exercise explains you how to :

-  **Customize a general standard**
-  **Customize a drafting standard**

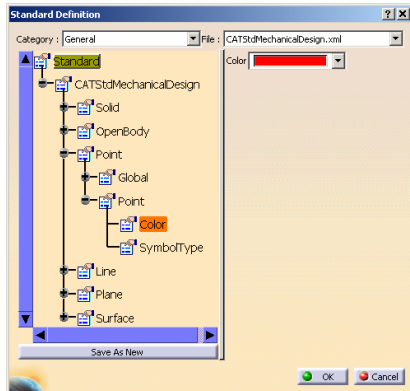


Do It Yourself (1/2)

What you will have to do :

1. Customizing a General Standard

- Set up the following variables in the env1 environment
CATReferenceSettingPath = C:\DS\LockSettings1
CATCollectionStandard = C:\DS\Standard
- Customize the General Standard CATStdMechanicalDesign



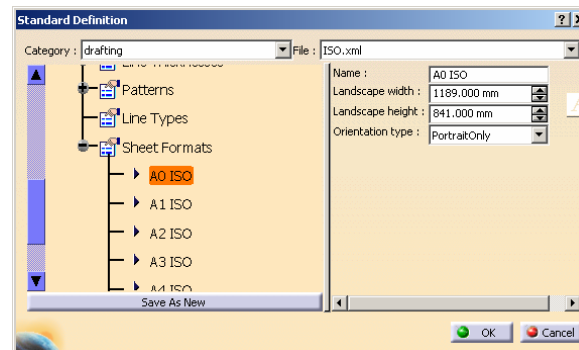
- Copy first
[InstallFolder]\B17\OS_a\startup\templates\CATStdMechanicalDesign.xml
in the following folder [Install Folder]\B09\OS_a\resources\standard
- Edit and customize the General Standard CATStdMechanicalDesign
- Set for example:
 - the point color to red
 - the SymbolType to 2
- Save the customized standard
- Launch a new CATIA session
- Check the new Standard, by creating points in Shape Design / Generative Shape Design

Do It Yourself (2/2)

What you will have to do :

2. Customizing a Drafting Standard

- **Create a new Drafting Standard from the ISO standard**
 - Launch the Standard Editor in Admin Mode
 - Open The ISO drafting standard
 - Change the Sheet Formats / A0 ISO:
Orientation type = "PortraitOnly"
 - Save As new Standard "ISOCusto"
 - Check that the new xml file has been created
 - Check the change, by opening a new drawing document
 - Select ISOCusto / ISO A0
- You cannot choose Landscape format !



Exercise 6

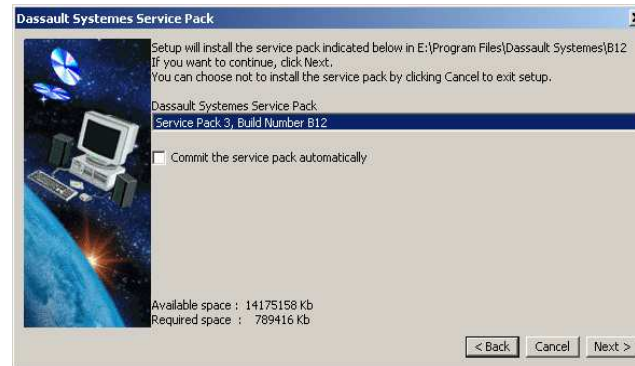
Software Management



60 min

This exercise explains you how to :

-  **Install Service Pack**
-  **Check SPK installation**
-  **Commit or Rollback a SP**
-  **Distributing CATIA code in batch mode**
-  **Distributing a service pack from an archive file**



Do It Yourself (1/2)

What you will have to do :

1. Installing a service pack

- Install the Service Pack without commit (Administrator log on)
- Check your installation with Software management tool
- Check the result in the file tree
- Commit or rollback the Service Pack

2. CATIA Installation in batch mode

On the server machine

- Create a folder C:\CDROM_V5R17 and share it.
- Copy the content of the cdrom of CATIA V5R17 code in the folder C:\CDROM_V5R17

On the client machine

- Log on with a network user which has administrator's right
- Create a batch file install.bat in C:\tmp
 - \\server\CDROM_V5R17\INTEL\StartB.exe -u C:\DS\CATIA_V5R17_BIS -newdir -D C:\DS\CATEnv -l "AL2.slt" -noreboot -v -ident BIS
- Launch the batch install.bat
- Look at the result in the file : C:\temp\cxinst.log

Do It Yourself (2/2)

What you will have to do :

3. Distributing a service pack from an archive file

- From an GA and SPn installation generate an archive file by means of CATDeltaInstall
- Copy and uncompress this archive on an other computer
- Check the installation

4. Uninstalling CATIA

This exercise must be done at the end of the course, to enable the next item exercises.

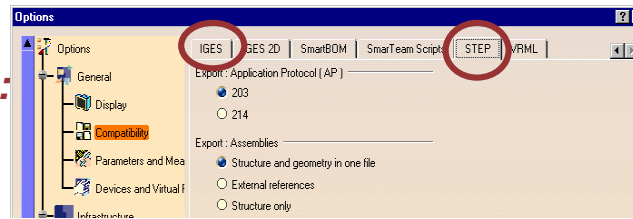
- Log as an administrator
- Remove Environments (delcatenv or Environment Editor)
- Uninstall CATIA
- Uninstall CATIA Documentation






Exercise 7

Tools



These exercises explain you how to :



-  Use the batch monitor
-  Use CATDMUUtility to convert data format
-  Change the minimum level of CATIA to open a CATPart
-  Use the batch monitor in the remote mode
-  Use the CATDLNameMigr

Do It Yourself (1/2)

What you will have to do :

1. Use the batch monitor

- Through the batch monitor, check a CATPart document using CATDUAV5 tools in various ways:
 - ✓ Interactive mode
 - ✓ Real batch mode using a xml file

2. Use CATDMUUtility to convert data format

- Convert a CATPart to a cgr file using CATDMUUtility

3. View and change the minimum level of CATIA to open a CATPart

- Find the minimum level to open a CATPart with CATUIDownwardCompatibility or CATDownwardCompatibility
- Change this minimum level

Do It Yourself (2/2)

4. Use the batch monitor in the remote mode

- Start the server monitor on the remote machine (CATBatSrvMonitorExe).
- Through the batch monitor, find the minimum level to open a CATPart document using DownwardCompatibility tools.
- Control the communication by choosing a license on the server through the batch monitor
- Run the batch in batch mode (CATBatCliMonitorExe)
- Check the result

5. Use the CATDLNameMigr

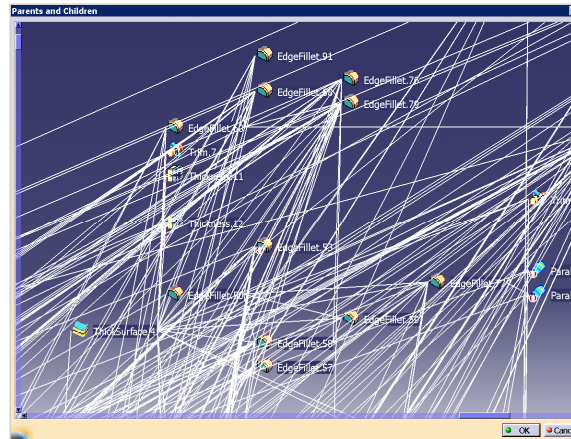
- Change the physical path links of a CATProduct onto DLNames
- Check the changes by means of Edit Links

Exercise 8






Data Management



60 min



This exercise explains you how to :

-  Define different document environment
-  View the documents links
-  Use and understand the search order strategies
-  Define a DLNames structure
-  Move CATIA data

Do It Yourself (1/3)

What you will have to do :

1. Define different document environment

- Authorize different documents environments
- Define DLnames (without structure)
- Check several file command and find where the document environments are used
 - File / open
 - Insert existing component
 - Edit / Links ...
- Define DLnames (with structure)

2. View the documents links

- Open a complex document
- Visualize the links with the file / desk command
- Visualize the links with the edit / links command

Do It Yourself (2/3)

3. Use and understand the search order strategies

- Set the “folder of the links” as the unique locator in the search order
- Create a CATProduct linked to a CATPart
- Move the CATPart into a folder different than the CATProduct folder, and re-open the CATProduct
- Repair the link with the Edit / Links
- Move the CATPart in the CATProduct folder, and change the search order to resolve the link
- Move the CATPart in a sub-folder of the CATProduct, set the “relative folder”, open the CATProduct, solve the link if necessary, close the link move the folder and its subfolder in an other place and reopen the folder.

4. Define a DLNames structure

- Activate the logical file tree setting

In admin mode

- Define a DLNames structure in admin mode, using the interactive command
- Export the list into a file
- Add a DLname into the list
- Import the list

In user mode

- Add a DLname in the existing tree, check the rules that are applied.

Do It Yourself (3/3)

5. Move CATIA data

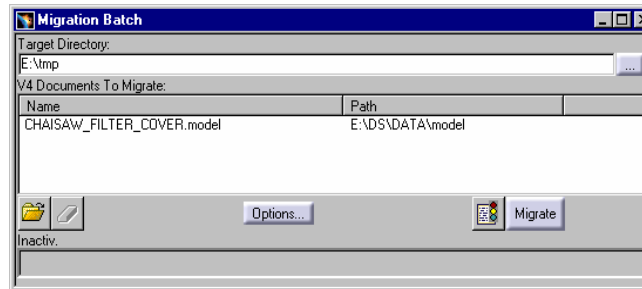
- Move a complex structure of documents with the SendTo command
- Rename documents
- Open the moved document (root document) and check the links

Exercise 9



Interoperability



15 min



This exercise explains you how to :

-  **Migrate CATIA V4 data to V5 with CATV4ToV5Migration interactive tool**
-  **Migrate CATIA V4 data to V5 with CATV4ToV5Migration batch tool**

Do It Yourself

What you will have to do :

- 1. Migrate CATIA V4 data to V5 with CATV4ToV5Migration interactive tool**
- 2. Migrate CATIA V4 data to V5 with CATV4ToV5 batch tool**