

CATIA ADMIN MODE

1. Copy and paste the CATIA icon on the desktop.
2. Right click and go into properties on this new icon.
3. Go to the General tab and rename “CATIA V5R19” to “CATIA V5R19 ADMIN”
4. Go to the Shortcut tab.
5. Change the Target by changing the original path of the Target field from (only change the red bold text as shown below):

"C:\Program Files\Dassault Systemes\B19\intel_a\code\bin\CATSTART.exe" -run
"CNEXT.exe" -env CATIA.V5R19.B19 -direnv "C:\Documents and Settings\All
Users\Application Data\DassaultSystemes\CATEnv" -nowindow
to

"C:\Program Files\Dassault Systemes\B19\intel_a\code\bin\CNEXT.exe" -admin -env
CATIA.V5R19.B19 -direnv "C:\Documents and Settings\All Users\Application
Data\DassaultSystemes\CATEnv" -nowindow

6. Next step is to create two directories where the admin settings and your company drafting standards will reside. Create two directories similar in name to:
 - a. c:\CATIA_admin\settings and
 - b. c:\CATIA_admin\standards
7. Now you need to set the path to these two directories in CATIA. In windows go to Start/All Programs/CATIA/Tools/Environment Editor
8. In this program, in the bottom window right click on the variable “CATReferenceSettingsPath” choose edit and add the path to the directory as the Value : c:\CATIA_admin\settings
9. In the same way change the variable “CATCollectionStandard” to the directory path : c:\CATIA_admin\standards
10. Exit from this program and save the change
11. Now finally you should be able to start CATIA in Admin mode : Double click the CATIA Admin icon on the desktop to start in this mode.

1. Start CATIA up in Admin mode by double clicking on “CATIA V5R19 ADMIN”
2. Go to Tools/Standards
3. Under Category choose “drafting”
4. Under File choose ISO.xml or the appropriate base standard which you would like to use.
5. In the tree in this window expand the ISO heading and go for the setting to change.
6. Open the online help file to show you what each variable will do : C:\Program Files\Dassault Systemes\B19doc\English\online\CATIAfr_C2\driugCATIAfrs.htm.
7. For example to change the default arrow type go to:
ISO/Styles/DistanceLengthDimension/Default/Symbols/Symbol1/Type then in the right window choose the preferred type of angle.
8. When you are finished customising to your delight, and exit, CATIA will automatically save this modified standard in the directory, CATIA will tell you where it saves the file.