

PDM Workbench Release 3.9 for Aras Innovator

User Manual

Version 1

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Your Comments are Welcome

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Preface

About this Manual

This manual describes the main functionality delivered by the PDM Workbench.

The functionality of the PDM Workbench as described in this manual uses Aras Innovator as backend PDM system for CATIA V5.

Other PDM systems might support the functionality of the PDM Workbench with their own types of objects. Layout of dialogs as well as object and relation types or classes might vary accordingly.

This manual is intended for end users of the PDM Workbench.

It assumes that the reader is familiar with the CATIA application and with Aras Innovator.

Related Documents

The following manuals contain information about installation, administration, usage, and customization of the PDM Workbench:

Manual Title	Version
PDM Workbench Installation & Administration Manual	3.9
PDM Workbench User Manual	3.9

Organization

This manual contains the following chapters:

Chapter 1 provides basic information about the PDM Workbench and describes some features of this application.

Chapter 2 describes the supported data models.

Chapter 3 describes the functionality which is implemented in the CATIA V5 workshop.

Glossary contains the PDM Workbench terminology.

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CHAPTER 1

Overview

This chapter provides basic information about the *PDM Workbench* and lists some features of this application.

Introducing PDM Workbench

The *PDM Workbench* is developed by T-Systems as a high-end integration between the CAD system *CATIA V5* and several PDM systems. Those PDM systems might be customized or virtual (a combination of several PDM systems). No matter which type they are the PDM Workbench will integrate them into CATIA V5. How those PDM object types are to present their dialogs and functionalities within the PDM Workbench is defined in a configuration file (called *PDM Workbench Schema File*).

The PDM Workbench workshop works with two different windows. The first window type presents the results of your queries. There you also might expand your product structure. Or you can create new objects in this window.

You can load this product structure into a CATIA V5 native window, and you can modify the content of this CATIA V5 native window (see *Picture 1: PDM Workbench workshop in CATIA V5*).



Picture 1: PDM Workbench workshop in CATIA V5

CHAPTER 2

Supported Data Models

The PDM Workbench supports two different data models.

BOM Part Structure data model

In the BOM Part Structure data model the PDM structure is represented by Parts (Assembly or Component). The relation "Part BOM" is used.

Each Part is described by a CAD Document which includes the CATIA file for a CATProduct, CATPart, or CATDrawing (see *Picture 2: Structure in the BOM Part Structure data model*).



Picture 2: Structure in the BOM Part Structure data model

Document data model

In the Document data model the PDM structure is represented by CAD Documents. The relation "CAD Structure" is used.

Each CAD Document includes the CATIA file for a CATProduct, CATPart, or CATDrawing (see *Picture 3: Structure in the Document data model*).



Picture 3: Structure in the Document data model

CHAPTER 3

Getting Started

This chapter describes the functionality of the PDM Workbench which is implemented in the CATIA V5 workshop.

We suppose that you have installed CATIA V5 with the PDM Workbench workshop on your computer. All configurations for the PDM Workbench (including the configurations for the PDM system) are done properly.

Note: All user actions described below are based on the corresponding configuration of the data model and the actions to be provided by the PDM system.

Login

In order to access the PDM Workbench functionality you must log in into the PDM system.

You select the "Login" icon within the PDM Workbench toolbar (see *Picture 4: The PDM Workbench toolbar before the login*) in CATIA V5. The other PDM Workbench icons remain deactivated.



Picture 4: The PDM Workbench toolbar before the login

The following dialog (see *Picture 5: "Login" dialog*) prompts you for all information necessary to identify yourself in the PDM system. In our example you are asked to enter your "User", "Password", and "Database". The identification items marked with an asterisk are defined as necessary for Login in the PDM Workbench configuration file.

PDM Login	<u>? ×</u>
	ormation
User	admin 💌 *
Password	
Database	InnovatorSolutions 💌 *
	Login Gancel

Picture 5: "Login" dialog

Click the "Login" button.

Once the login action was successful most of the icons in the PDM Workbench toolbar will turn active while some will remain inactive as they depend on further user actions to be done to get them available (see *Picture 6: The PDM Workbench toolbar after the login*).



Picture 6: The PDM Workbench toolbar after the login

For CATPart and CATDrawing as top level object of the CATIA V5 window you have to use the actions "PDM Properties", "Unlock", "Lock", and "Duplicate" of the "PDM Workbench context commands" toolbar (see *Picture 7: The PDM Workbench context commands toolbar*). The icons in this toolbar are only repainted (e.g. switch from "Lock" to "Unlock") when you newly activate the CATIA V5 window.



Picture 7: The PDM Workbench context commands toolbar

Query

You can query for any object stored in the PDM system.

Once you click the "Query" icon within the PDM Workbench toolbar the query dialog opens. On the left side of the dialog you can define the query criteria. On the right side you will see the query results (see *Picture 8: "PDM Query" dialog*).

PDM Query				<u>? ×</u>
PDM object type	Part Number	Major Rev.	Generation	Name
Part 💌				
PDM object attributes				
Part Number				
Major Rev.				
Generation				
Name				
State				
Unit				
Make / Buy				
Description				
Created on				
Modified on				
Locked by				
•	•			
	<u> </u>			
				Query Close

Picture 8: "PDM Query" dialog

The query criteria are defined by the type and the attributes for this type. A default type is selected and its attributes are shown.

You can select a type in the single selector list in the first line. The attributes will be shown automatically.

Specify your selection criteria in the lines below.

You can use the wildcard "*" in this dialog. All attributes visible in this dialog are attributes of the PDM system for the selected object type. (These attributes as well as their adherence to the "Query" dialog of this type are defined in the PDM Workbench configuration file).

Some attribute values can be keyed in as free text while for others a value might be selected from a list.

PDM Query				<u>?</u> ×
PDM object type	Part Number	Major Rev.	Generation	Name
Part 💌				
PDM object attributes				
Part Number Cr*				
Major Rev. A				
Generation				
Name				
State				
Unit				
Make / Buy				
Description				
Created on				
Modified on				
Locked by				
	•			
				Query Close

Picture 9: "PDM Query" dialog – enter query criteria

When you have specified the selection criteria you can start the query for PDM objects with a click on the "Query" button.

In the settings of the PDM Workbench you can define the columns for the query result on the right side of the dialog (see chapter *Options* - *Query Mode*).

The found objects will be presented on the right side of the dialog. By clicking on the column name you can sort the query result (see *Picture 10: "PDM Query" dialog – found objects*).

1 1 1 1 1 1 1	CrankBolt CrankBoltModel CrankShaft CrankShaftLeft CrankShaftLeftModel CrankShaftRight CrankShaftRight CrankShaftRightModel
1 1 1 1 1 1	Crankshaft CrankShaftLeft CrankShaftLeftModel CrankShaftRight
1 1 1 1	CrankShaftLeft CrankShaftLeftModel CrankShaftRight
1 1 1	CrankShaftLeftModel CrankShaftRight
1 1 1	CrankShaftRight
1 1	
1	CrankShaftRightModel
	D
	Download CATDrawings related to BOM Parl

Picture 10: "PDM Query" dialog – found objects

There you can select the object or the objects to be opened in a new PDM Workbench window. Click on the right mouse button and select "Open in New PDM Window" (see *Picture 11: "PDM Query" dialog – Action "Open in New PDM Window"*).

PDM Query (Logged in as admin)				? ×
PDM object type	Part Number 💧	Major Rev.	Generation	Name
Part 💌	🔅 CrankBolt	A	1	CrankBolt
	🔅 CrankBoltModel	A	1	CrankBoltModel
PDM object attributes	🔯 Crankshaft	· · · · · · · · · · · · · · · · · · ·	<u> </u>	Crankshaft
Part Number	CrankShaftLeft	Open in new PDM Window		CrankShaftLeft
Major Rev.	CrankShaftLeftModel	🐴 Unlock		CrankShaftLeftModel
Generation	CrankShaftRight	📙 Lock		CrankShaftRight
Name	🔅 CrankShaftRightModel	Promote		CrankShaftRightModel
State		/ Promoce		
Unit				
Make / Buy				
Description				
Created on				
Modified on				
Created by				
Modified by				
Locked by				
	 €			
			🧧 Dowr	nload CATDrawings related to BOM Part
				Query Close

Picture 11: "PDM Query" dialog – Action "Open in New PDM Window"

A double click on a single object opens it in a new PDM Workbench window.

Now you can close the query dialog by clicking on the "Close" button.

The found objects are opened in the PDM Workbench window now (see *Picture 12: Query result in PDM Structure window*).

📉 Assembly (Logged in as admin)	- O ×
CrankShaftLeft, A, 1, CrankShaftLeft, Preliminary	
🕸 Crankshaft, A, 1, Crankshaft, Preliminary	
CrankShaftRight, A, 1, CrankShaftRight, Preliminary	
	7
	Ĺ
	×~~~y

Picture 12: Query result in PDM Structure window

Expand Single Level

You can expand from the selected object to other related objects via several relations in two directions.

Select the object from which you want to expand and open the context menu by clicking on the right mouse button. Select the context action "Expand". The "Expand" sub menu offers the possible relation directions for expansion from the selected object.

In the example in *Picture 13: Action "Expand Single Level"* you select the direction "Uses Part BOM" for the selected "Assembly" object "Engine".

🋐 Engine, A, 1, Engine, Prelimi	inary (Logged in as a	dmin)	
	Center graph Reframe On Hide/Show Properties Open Sub-Tree	Alt+Enter	
	Cut <u>Copy</u> <u>Baste</u> Paste <u>Special</u>	Ctrl+X Ctrl+C Ctrl+V	
	Delete Engine, A, 1, Engine	Del , Preliminary object 🔹 🕨	
1	Expand Exp <u>a</u> nd Multiple Leve De <u>-</u> Expand	els 🕨	Uses Part BOM Attaches Is Used in Part BOM
	👩 Open in new PDM Wi	indow	

Picture 13: Action "Expand Single Level"

The related objects for the selected "Assembly" object "Engine" are shown in *Picture 14: Result of expand single level.*

In the product structure tree you can see the objects and relations. The type of the relation is displayed in braces in the line of the relation.



Picture 14: Result of expand single level

When you have switched off the display of the relations in the options (see *Options - PDM Relations*) then the expand tree looks like displayed in *Picture 15: Result of expand single level without relations*.

Nengine, A, 1, Engine, Preliminary (Logged in as admin)	_ 🗆 🗵
Engine, A, 1, Engine, Preliminary	
Uses Part BOM (Part BOM, A, 1) / FixedParts, A, 1, FixedParts, Preliminary	
See Part BOM (Part BOM, A, 1) / Cylinder, A, 1, Cylinder, Preliminary	
Uses Part BOM (Part BOM, A, 1) / ConnectionRod, A, 1, ConnectionRod, Preliminary	
Uses Part BOM (Part BOM, A, 1) / Crankshaft, A, 1, Crankshaft, Preliminary)	Ĩ
	×⁄~y

Picture 15: Result of expand single level without relations

The type of the relation, the revision, and the generation of the relation are displayed in the braces in the line of the relation. The attributes can be configured in the PDM Workbench configuration file.

Expand Multiple Levels

You also might expand a relation direction in multiple levels from the selected object.

Select the object from which you want to expand and open the context menu by clicking on the right mouse button. Select the context action "Expand Multiple Levels". The "Expand Multiple Levels" sub menu offers the possible relation directions for expansion from the selected object.

In the example in *Picture 16: Action "Expand Multiple Levels"* you select the direction "Uses Part BOM" for the selected "Assembly" object "Engine".

🛐 Engine, A, 1, Engine, Preliminary (Logged in as admin)				
🔅 Engine, A, 1, Em	aipa Draliminary		1	
	C <u>e</u> nter graph			
	<u>R</u> eframe On			
	🚰 Hide/Show			
	Properties	Alt+Enter		
	Open Sub-Tree			
2	🔓 Cu <u>t</u>	Ctrl+X		
	≧ ⊆ору	Ctrl+C		
	Paste	Ctrl+V		
	Paste <u>S</u> pecial,			
	<u>D</u> elete	Del		
	Engine, A, 1, Engine,	Preliminary object 🔹 🕨		
	Expand	•		
	Exp <u>a</u> nd Multiple Levels	; •	<u>U</u> ses Part BOM	
	De <u>-</u> Expand		Is Used in Part BOM	
<u> </u>	🛃 Open in new PDM Win	dow	1	

Picture 16: Action "Expand Multiple Levels"

The related objects in multiple levels are shown for the selected "Assembly" object "Engine" in *Picture 17: Result of expand multiple levels*.

In the product structure tree you can see the objects and relations. The type of the relation, the revision, and the generation are displayed in braces in the line of the relation.

When you have switched off the display of the relations in the options (see *Options - PDM Relations*) then the relation objects will not be displayed in the expanded product structure tree.



Picture 17: Result of expand multiple levels

De-Expand

When you want to reduce the displayed PDM structure then you can de-expand parts of the structure.

In *Picture 18: PDM structure before the De-Expand* you can see the PDM structure before the de-expand.



Picture 18: PDM structure before the De-Expand

First you select the root element of the sub structure which you want to de-expand. You also might de-expand a single object. Then you open the context menu by clicking the right mouse button and you select the action "De-Expand".

In the example in *Picture 19: Action "De-Expand"* you select the object "FixedParts_Assm" as root of the sub structure to be de-expanded.



Picture 19: Action "De-Expand"

In *Picture 20: PDM structure after the De-Expand* you see that the selected sub structure with the root element "FixedParts_Assm" is de-expanded as it is no longer displayed in the PDM structure.

But remember that the PDM structure will not be modified by this de-expand. The sub structure will not be deleted from the PDM structure.



Picture 20: PDM structure after the De-Expand

Properties

The PDM properties of the objects in the PDM Workbench window are transferred from the PDM system when the objects are displayed in the PDM Workbench window for the first time, e.g. when they are shown as a result of a query or an expand action.

You take a look at those properties and/or change these properties when opening the "Properties" dialog. Therefore click on the right mouse button. Now the context menu opens and you select the action "Properties" (see *Picture 21: Action "Properties*").

🋐 Engine, A, 1, Engine, Preliminary (L	ogged in as admin)		
Engine, A, 1, Engine, Prelimir	larv		1
- 🗘 Uses Part BOM (Part I	C <u>e</u> nter graph		minary
Uses Part BOM (Pa	<u>R</u> eframe On		ock, Preliminary
L L C Uses Part BOM	Hide/Show		rblock, Preliminary
Uses Part BOM (Pa	Properties	Alt+Enter	
- Uses Part BOM	Ope <u>n</u> Sub-Tree		V
	Cu <u>t</u>	Ctrl+X	odel, Preliminary
	⊆ору	Ctrl+C	hary
L Uses Part BC	Paste	Ctrl+V	/Model, Preliminary
- Ouses Part BOM (Part I	Paste Special		rv
Uses Part BOM (Pa	Delete	Del	oltAssm, Preliminary
- OUses Part BOM	Engine, A, 1, Engine, Prelimina	ry object 🛛	ngRing, Preliminary
Uses Part BC	Expand		- 1, RetainingRingMode
	Expand Multiple Levels	1	stonBoltModel, Prelim
Uses Part BOM (Pa	De_Expand		
Log Uses Part BOM	Open in new PDM Window		pdel, Preliminary
Uses Part BOM (Part I	Unlock		nRod, Preliminary
	Lock		2
	Promote		onnectionRodModel,
	Re <u>vi</u> se		

Picture 21: Action "Properties"

The "Properties" dialog will be opened (see *Picture 22: "Properties" dialog – tab "Properties"*). It consists of two tabs.

In the tab "Properties" the values of the attributes are shown on a grey background. This indicates that the values cannot be changed. All properties that are shown within this dialog can be specified within the PDM Workbench configuration file.

Properties		<u>? ×</u>
Current selection :	Engine, A, 1, Engine, Preliminary	v
Properties Up	date Item	
- Assembly		
Part Number	Engine	
Major Rev.		
Generation	1	
Name	Engine	
State	Preliminary	
Unit	EA 🔻	
Make / Buy	Make	
Cost		
Description		
New Version		
Туре	Assembly	
Assigned Creator		
Designated User		
Created by	Innovator Admin	
	2012-03-28T14:01:16	
	Innovator Admin	
	2012-03-28T14:01:17	
	Innovator Admin	
Release Date		
Effective Date		
· · · · · · · · · · · · · · · · · · ·		
		More
		OK Apply Close

Picture 22: "Properties" dialog - tab "Properties"

In the tab "Update Item" those attribute values shown on a grey background cannot be changed. Those presented with the white background can be changed (see *Picture 23: "Properties" dialog – tab "Update Item"*).

Required attributes are marked with an asterisk (*) on the right side of the input field.

There are several types of attributes presented on corresponding widgets:

- Free text can be inserted.
- One value can be selected in a single selector list or combo box for instance or any other widget.
- Multiple values can be selected in a multi selector list or other widget types supporting this feature.
- The value can be marked or unmarked in a check box for instance.

Properties				<u>? ×</u>
Current selection	on : Engine, A, 1, Engine, Preliminary			~
Properties	Update Item			
-Assembly -				
Part Number	Engine			
Major Rev.	A			
Generation	1			
	Engine			
Unit	EA			
Make / Buy	Make			
Cost				
Description				
Туре	Assembly			
			M	lore
		🎱 ок	Apply	Close

Picture 23: "Properties" dialog – tab "Update Item"

Please use the "OK" button to close the "Properties" dialog.

In the tab "Update Item" the required attributes get checked for being satisfied. If not you will get an error window that describes your fault.

The object gets updated in the PDM system database according to the Update dialog values. When the update is not possible in the PDM system (because of a wrong value of an attribute or different reasons) then you will get an error window.

The PDM system properties of a CATPart and CATDrawing as top level object of the CATIA V5 window cannot be inspected with the context menu. You have to use the "PDM

Properties" button within the "PDM Workbench context commands" toolbar. The values of the attributes are in read-only mode. You are not able to make changes on the PDM system attributes in this dialog.

Lock

Lock of the Part in the PDM Workbench window

You have to lock the PDM objects prior to changing and updating them.

So you select the object in the PDM Workbench window and click the right mouse button. In the context menu you select the context action "Lock" (see *Picture 24: Action "Lock"*). This action is only active in the context menu when it is possible for the selected object. Otherwise it will be deactivated in the context menu.

A multi-select of objects is also supported.

🎦 Engine, A, 1, Engine, Preliminary	(Logged in as admin)	
Engine, A, 1, Engine, Prolim	ain an I	
- Uses Part BOM (Par	C <u>e</u> nter graph	eliminary
Uses Part BOM (F	<u>R</u> eframe On	Block, Preliminary
L L Uses Part BON	Hide/Show	derblock, Preliminary
=- Uses Part BOM (F	Properties Alt+Ent	
- Uses Part BON	Ope <u>n</u> Sub-Tree	()
	Cu <u>t</u> Ctrl-	
Uses Part E	<u>C</u> opy Ctrl-	
=- 🗘 Uses Part BON 🔒	Paste Ctrl-	
Uses Part E	Paste Special	ewModel, Preliminary
🗧 📿 Uses Part BOM (Par —		hary
🗧 🏳 Uses Part BOM (F	Delete	^{Del} BoltAssm, Preliminary
🝦 🌣 Uses Part BON	Engine, A, 1, Engine, Preliminary object	iningRing, Preliminary
Uses Part E		, 1, RetainingRingMode
🖕 🗘 Uses Part BON	Expand	lt, Preliminary
📕 📙 🖵 🔆 Uses Part E 📻	Exp <u>a</u> nd Multiple Levels	PistonBoltModel, Prelim
- Ouses Part BOM (F	De <u>-</u> Expand	ary
L Uses Part BON	Open in new PDM Window	Model, Preliminary
- Ouses Part BOM (Par	Unlock	ionRod, Preliminary
Uses Part BOM (Lock	ConnectionRodModel.
	Promote	× Y
12	Re <u>v</u> ise	

Picture 24: Action "Lock"

The confirmation message for the Lock opens (see *Picture 25: Confirm the Lock of the part*). If it is the correct object that you want to lock then just press the "Yes" button.



Picture 25: Confirm the Lock of the part

The selected object and the attached documents will be locked by the user (see *Picture 26: Object is locked*).

M-Workbench	?
Information	
PDM Messages:	
Locked PDM object "Engine (Mechanical/Assembly)". Locked PDM object "Engine (Assembly)".	
Action succeeded.	
	Close

Picture 26: Object is locked

The background color of the locked object changed to green in the PDM Workbench window (see *Picture 27: Locked object*).

🛐 Engine, A, 1, Engine, Preliminary (Logged in as admin)
Engine, A, 1, Engine, Preliminary
🗣 😳 Uses Part BOM (Part BOM, A, 1) / FixedParts, A, 1, FixedParts, Preliminary
🚽 🖓 Uses Part BOM (Part BOM, A, 1) / CylinderBlock, A, 1, CylinderBlock, Preliminary
Uses Part BOM (Part BOM, A, 1) / Zylinderblock, A, 1, Zylinderblock, Preliminary
🖶 🗘 Uses Part BOM (Part BOM, A, 1) / Rack, A, 1, Rack, Preliminary
- 🗘 Uses Part BOM (Part BOM, A, 1) / Case, A, 1, Case, Preliminary
Uses Part BOM (Part BOM, A, 1) / CaseModel, A, 1, CaseModel, Preliminary
- Oliver Service Content and C
Uses Part BOM (Part BOM, A, 1) / ScrewModel, A, 1, ScrewModel, Preliminary
🗣 😳 Uses Part BOM (Part BOM, A, 1) / Cylinder, A, 1, Cylinder, Preliminary
🚽 📮 💭 Uses Part BOM (Part BOM, A, 1) / PistonBoltAssm, A, 1, PistonBoltAssm, Preliminary
- Uses Part BOM (Part BOM, A, 1) / RetainingRing, A, 1, RetainingRing, Preliminary
📕 📕 🖣 🖧 Uses Part BOM (Part BOM, A, 1) / RetainingRingModel, A, 1, RetainingRingMode
- 🗘 Uses Part BOM (Part BOM, A, 1) / PistonBolt, A, 1, PistonBolt, Preliminary
🗖 📔 🖕 🔆 Uses Part BOM (Part BOM, A, 1) / PistonBoltModel, A, 1, PistonBoltModel, Prelim
📼 🗘 Uses Part BOM (Part BOM, A, 1) / Piston, A, 1, Piston, Preliminary
📙 🖓 Uses Part BOM (Part BOM, A, 1) / PistonModel, A, 1, PistonModel, Preliminary
🜩 💭 Uses Part BOM (Part BOM, A, 1) / ConnectionRod, A, 1, ConnectionRod, Preliminary
Uses Part BOM (Part BOM, A, 1) / ConnectionRodModel, A, 1, ConnectionRodModel,

Picture 27: Locked object

The green background color of the locked object indicates that the object in the PDM system can be changed by the user.

Lock of the object in the Query dialog

It is also possible to lock the object from the query result list of the "Query" dialog. In the example in *Picture 28: Action "Lock" in the query result list* you select the object and click on "Lock" in the context menu. This action is only active in the context menu when it is possible for the selected object. Otherwise it will be deactivated in the context menu.

A multi-select of objects is also supported.

PDM Query (Logged in as admin)						
	lame					
Part Case A 1 Ca	ase					
CaseModel A 1 Ca	aseModel					
	ConnectionRod					
	ConnectionRodModel					
	irankBolt					
	irankBoltModel					
Name	irankshaft					
CrankShaftLeft A 1 Cr	irankShaftLeft					
CrankShartLeftModel A 1 Cr	rankShaftLeftModel					
	irankShaftRight					
	irankShaftRightModel					
	ylinder					
	ylinderBlock					
The Unlock						
Created op						
Modified on						
Created by						
Locked by						
📮 Download C	CATDrawings related to BOM Part					
	Query Close					

Picture 28: Action "Lock" in the query result list

Lock of the Document in the CATIA V5 window

You have to lock the PDM objects prior to changing and updating them.

You select the object in the CATIA V5 window and click the right mouse button. In the context menu you select the context action "PDM Workbench \rightarrow Lock" (see *Picture 29: Action "Lock" in the CATIA V5 window*).

Engine.CATProduct					
🔊 Engine				4	
+- [•] FixedParts_Assm (EivedDorto_too	m 1)		(Λ)	
🕂 😽 Cylinder_Assm (Cy	C <u>e</u> nter graph			×{*	
	Kendine on				
+-🔖 ConnectionRod (Co	🔗 Hide/Show				
🕂 🐝 Crankshaft_Assm (Properties	Alt+Enter			
+- Constraints	📔 Ope <u>n</u> Sub-Tree				
	🔀 Cu <u>t</u>	Ctrl+X			
Applications	Сору	Ctrl+C			
	🚰 Paste	Ctrl+V			
	Paste <u>S</u> pecial				
	Delete	Del			
	EixedParts_Assm.	1 object 🕨			
	PD <u>M</u> -Workbench)	PDM Properties		
	Components	,	Highlight PDM Nodes		
	Representations	,	Lulock.		7
	Selection Mode		🚇 Lock		¥ L×
			Duplicate		

Picture 29: Action "Lock" in the CATIA V5 window

For further details of the lock dialogs please refer to the chapter *Lock of the Part in the PDM Workbench window*.

The lock of a CATPart and CATDrawing as top level object of the CATIA V5 window cannot be done with the context menu. You have to use the "Lock" button of the "PDM Workbench context commands" toolbar. The icons in this toolbar are only repainted (e.g. switch from "Lock" to "Unlock") when you newly activate the CATIA V5 window.

A multi-select of objects is also supported.

Unlock

Unlock of the Part in the PDM Workbench window

When an object is locked by you then you have to unlock it in the PDM system to make it available for all other users.

You select the object in the PDM Workbench window and click the right mouse button. In the context menu you select the "Unlock" context action (see *Picture 30: Action "Unlock"*). This action is only active in the context menu when it is possible for the selected object. Otherwise it will be deactivated in the context menu.

A multi-select of objects is also supported.

E 🛐	ingine, A, 1, Engine, Prelimina	iry (Logged in as admin)		_ 🗆 🗡
	C Engine, A, 1, Engine, P C Uses Part BOM (Part C Uses Part BOM (F		C <u>e</u> nter graph <u>R</u> eframe On			eliminary Block, Preliminary
	=-↓ Uses Part BOM (F		<u>H</u> ide/Show Properties Ope <u>n</u> Sub-Tree	Alt+Enter		lerblock, Preliminary ,
	=-○ Uses Part BON	% ••• •••	Cu <u>t</u> Copy Paste	Ctrl+X Ctrl+C Ctrl+V		ary Model, Preliminary hinary
•	- ♀ Uses Part B Uses Part BOM (Part - ♀ Uses Part BOM (F		Paste <u>S</u> pecial Delete	Del		ewModel, Preliminary hary BoltAssm, Preliminary
	- Ouses Part BON LOUses Part B - Ouses Part BON		Engine, A, 1, Engine, E⊻pand		•	ningRing, Preliminary , 1, RetainingRingMode It, Preliminary
	Uses Part B OUses Part BOM (F)		Exp <u>a</u> nd Multiple Leve De <u>-</u> Expand Open in new PDM Wir			PistonBoltModel, Prelim ary 40del, Preliminary
•	Uses Part BOM (Part Control Uses Part BOM (F		Unlock Lock Pro <u>m</u> ote Revise			onRod, Preliminary ConnectionRodModel,

Picture 30: Action "Unlock"

The confirmation message for the "Unlock" opens (see *Picture 31: Confirm the Unlock of the part*). If it is the correct object that you want to unlock then just select the "Yes" button.

🛐 Engine, A, 1, Engine, Preliminary (Logged in as admin)					
Engine, A, 1, Engine, Preliminary					
- 🔆 Uses Part BOM (Part BOM, A, 1) / FixedParts, A, 1, FixedParts, Preliminary					
🚽 📮 🍄 Uses Part BOM (Part BOM, A, 1) / CylinderBlock, A, 1, C	ylinderBlock, Preliminary				
Uses Part BOM (Part BOM, A, 1) / Zylinderblock, A, 1	, Zylinderblock, Preliminary				
🔹 🖓 Uses Part BOM (Part BOM, A, 1) / Rack, A, 1, Rack, Preli	iminary				
- 🖓 Uses Part BOM (Part BOM, A, 1) / Case, A, 1, Case, P	reliminary				
Uses Par Unlock PDM Objects?	CaseModel, Preliminary				
=- 🗘 Uses Part B	Preliminary				
Lie	, ScrewModel, Preliminary				
📮 🖓 Uses Part BOM (F 🛛 🔍 Engine, A, 1, Engine, Preliminary?	reliminary				
- O Uses Part BOM	istonBoltAssm, Preliminary				
Uses Part B	RetainingRing, Preliminary				
	del, A, 1, RetainingRingMode				
- Ouses Part BOM (Part BOM, A, 1) / PistonBolt, A, 1, Pis	stonBolt, Preliminary				
Uses Part BOM (Part BOM, A, 1) / PistonBoltModel	, A, 1, PistonBoltModel, Prelim				
- Uses Part BOM (Part BOM, A, 1) / Piston, A, 1, Piston, P					
Uses Part BOM (Part BOM, A, 1) / PistonModel, A, 1, PistonModel, Preliminary					
- Uses Part BOM (Part BOM, A, 1) / ConnectionRod, A, 1, ConnectionRod, Preliminary					
Uses Part BOM (Part BOM, A, 1) / ConnectionRodModel, A, 1, ConnectionRodModel,					

Picture 31: Confirm the Unlock of the part

The selected object and the attached documents will be unlocked by the user (see *Picture 32: Object is unlocked*).

2DM-Workbench	×
_ Information	_
PDM Messages:	
Unlocked PDM object "Engine (Mechanical/Assembly)". Unlocked PDM object "Engine (Assembly)".	
Xction succeeded.	
	0

Picture 32: Object is unlocked

The background color of the unlocked object changed to blank in the PDM Workbench window (see *Picture 33: Unlocked object*).

Engine, A, 1, Engine, Preliminary (Logged in as admin)
Engine, A, 1, Engine, Preliminary
- 🔆 Uses Part BOM (Part BOM, A, 1) / FixedParts, A, 1, FixedParts, Preliminary
🖕 🖓 Uses Part BOM (Part BOM, A, 1) / CylinderBlock, A, 1, CylinderBlock, Preliminary
Uses Part BOM (Part BOM, A, 1) / Zylinderblock, A, 1, Zylinderblock, Preliminary
🖕 🖓 Uses Part BOM (Part BOM, A, 1) / Rack, A, 1, Rack, Preliminary
- 🖓 Uses Part BOM (Part BOM, A, 1) / Case, A, 1, Case, Preliminary
Uses Part BOM (Part BOM, A, 1) / CaseModel, A, 1, CaseModel, Preliminary
- Ouses Part BOM (Part BOM, A, 1) / Screw, A, 1, Screw, Preliminary
Uses Part BOM (Part BOM, A, 1) / ScrewModel, A, 1, ScrewModel, Preliminary
🗧 🖓 Uses Part BOM (Part BOM, A, 1) / Cylinder, A, 1, Cylinder, Preliminary
= 😳 Uses Part BOM (Part BOM, A, 1) / PistonBoltAssm, A, 1, PistonBoltAssm, Preliminary
- Uses Part BOM (Part BOM, A, 1) / RetainingRing, A, 1, RetainingRing, Preliminary
Uses Part BOM (Part BOM, A, 1) / RetainingRingModel, A, 1, RetainingRingMode
= 😳 Uses Part BOM (Part BOM, A, 1) / PistonBolt, A, 1, PistonBolt, Preliminary
Lises Part BOM (Part BOM, A, 1) / PistonBoltModel, A, 1, PistonBoltModel, Prelim
 Uses Part BOM (Part BOM, A, 1) / Piston, A, 1, Piston, Preliminary
Sector (Part BOM (Part BOM, A, 1) / PistonModel, A, 1, PistonModel, Preliminary
- Cuses Part BOM (Part BOM, A, 1) / ConnectionRod, A, 1, ConnectionRod, Preliminary
Uses Part BOM (Part BOM, A, 1) / ConnectionRodModel, A, 1, ConnectionRodModel,

Picture 33: Unlocked object

The blank background color of the unlocked object indicates that the object in the PDM system cannot be changed by the user.

Unlock of the object in the Query dialog

It is also possible to unlock the object from the query result list of the "Query" dialog. In the example in *Picture 34: Action "Unlock" in the query result list* you select the object and click on "Unlock" in the context menu. This action is only active in the context menu when it is possible for the selected object. Otherwise it will be deactivated in the context menu.

A multi-select of objects is also supported.	

	-				
	Part Number 🖉		Major Rev.	Generation	Name
¢	Case		A	1	Case
1	CaseModel		А	1	CaseModel
¢	ConnectionRod		A	1	ConnectionRod
	ConnectionRodModel		A	1	ConnectionRodModel
0	CrankBolt		A	1	CrankBolt
	CrankBoltModel		A	1	CrankBoltModel
•	Crankshaft		A	1	Crankshaft
¢	CrankShaftLeft		А	1	CrankShaftLeft
	CrankShaftLeftModel		A	1	CrankShaftLeftModel
0	CrankShaftRight		A	1	CrankShaftRight
	CrankShaftRightModel		A	1	CrankShaftRightModel
¢	Cylinder			1	Cylinder
¢	CylinderBlock 🛛 🔞	Open in new PDI	M Window	1	CylinderBlock
	<u></u>	Unlock			
		LOCK			
	7	Promote			
	1.0	Device			
	÷Q	Relate active File	e to Part		
	_				
•					Þ
				🖾 Downlow	ad CATDrawings related to BOM Part
					Query Close
	x0000000000000000000000000000000000000	CansettonRod ConnectionRod ConnectionRod ConnectionRod CanisBolt CrankBolt C	Case CaseModel ConnectionRod ConnectionRod CrankBotModel CrankBotModel CrankBotModel CrankShaftLeft CrankShaftLeft CrankShaftLeft CrankShaftRight CrankShaftRi	Case A CaseModel A ConnectionRod A ConnectionRodModel A CrantBolt A CrantBoltModel A CrantSoltModel A CrantSoltModel A CrantSoltModel A CrantSoltModel A CrantSoltLeftModel A CrantSoltLeftModel A CrantSoltLeftModel A CrantSoltLeftModel A CrantSoltLeftModel A CrantSoltLeftModel A CrantSoltLeftModel A CylinderBlock Promote 102 Revise Solt Revise Solt Relate active File to Part	Case A 1 CaseModel A 1 ConnectionRod A 1 ConnectionRod A 1 ConnectionRod A 1 CrankBolt A 1 CrankBolt A 1 CrankShaft A 1 CrankShaft A 1 CrankShaftLeftModel A 1 CrankShaftRight A 1 CrankSh

Picture 34: Action "Unlock" in the query result list

? X

Unlock of the Document in the CATIA V5 window

When an object is locked by you then you can unlock it in the PDM system to make it available for all other users.

You select the object in the CATIA V5 window and click the right mouse button. In the context menu you select the context action "PDM Workbench \rightarrow Unlock" (see *Picture 35: Action "Unlock" in the CATIA V5 window*).

Engine.CATProduct				
Engine				、 、
FixedParts_Assm	Center graph)
🕂 🐴 Cylinder_Assm (C				
← 🗞 ConnectionRod (C				-
🕂 🕂 Crankshaft_Assm	Properties	Alt+Enter		
+- Constraints	Dpen Sub-Tree			
+-Applications		Ctrl+X		
, ppreddono	Li ⊆opy Copy Baste	Ctrl+C Ctrl+V		
	Paste Special	Culty		
	Delete	Del		
	EixedParts_Assm.	1 object		
	PD <u>M</u> -Workbench		PDM Properties	
	Components		Highlight PDM Nodes	
	Represent <u>a</u> tions			7
	Selection Mode		Duplicate	
	Sejection Mode			

Picture 35: Action "Unlock" in the CATIA V5 window

For further details of the "Unlock" dialogs please refer to the chapter Unlock of the Part in the PDM Workbench window.

The unlock of a CATPart and CATDrawing as top level object of the CATIA V5 window cannot be done with the context menu. You have to use the "Unlock" button within the "PDM Workbench context commands" toolbar. The icons in this toolbar are only repainted (e.g. switch from "Lock" to "Unlock") when you newly activate the CATIA V5 window.

A multi-select of objects is also supported.

"Lock All" / "Unlock All"

It is possible lock or unlock all the items in a structure.

~	<u>U</u> nlock	
A	Lock	
Ъ	Unloc <u>k</u> all	
	Lock all	1.1



Promote

The PDM objects can be promoted.

You select the object in the PDM Workbench window and click the right mouse button. In the context menu you select the "Promote" context action (see *Picture 37: Action "Promote"*).

The part and the document have to be unlocked in order to be promoted.

Engine, A, 1, Engine, Preliminary	(Logged in as admin)		
- Uses Part BOM (Par - Uses Part BOM (Par - Uses Part BOM (F - Uses Part BOM (F)) (F - Uses	(Logged in as admin) Center graph Reframe On Hide/Show Properties Alt+Enter Open Sub-Tree Cut_ Ctrl+X Copy Ctrl+C Paste Ctrl+V Paste Special	r derblock _x ary Model, P v ninary	
Uses Part BOM (Par Uses Part BOM (F Uses Part BOM (F Uses Part BON	Delete Del Engine, A, 1, Engine, Preliminary object Expand	 BoitAssri ningRing , 1, Reta 	n, Preliminary g, Preliminary iiningRingMode
Uses Part BON Uses Part BOM (F Uses Part BOM (F Uses Part BOM (Par Uses Part BOM (Par Uses Part BOM (F	Expand Multiple Levels De_Expand Open in new PDM Window Unlock Lock Promote Revise	ary Model, Pi ionRod,	oltModel, Prelim reliminary Preliminary tionRodModel,

Picture 37: Action "Promote"

You will be asked if you really want to promote the objects. Please confirm with "Yes" (see *Picture 38: Confirm the "Promote" action*).
🛐 Engine, A, 1, Engine, Prelin	iinary (Logged in as admin)					
A 🗘 Engine, A, 1, Engine,	Preliminary					
🚽 🔆 Uses Part BOM (P	- Uses Part BOM (Part BOM, A, 1) / FixedParts, A, 1, FixedParts, Preliminary					
🚽 🍦 🌣 Uses Part BOM	(Part BOM, A, 1) / CylinderBlock, A, 1, 0	CylinderBlock, Preliminary				
Uses Part B	OM (Part BOM, A, 1) / Zylinderblock, A,	1, Zylinderblock, Preliminary				
🗧 🗘 Uses Part BOM	(Part BOM, A, 1) / Rack, A, 1, Rack, Pre	eliminary				
🔷 🗘 Uses Part B	DM (Part BOM, A, 1) / Case, A, 1, Case,	Preliminary				
Uses Par	Promote PDM Objects?	🔀 aseModel, Preliminary				
🗧 🗘 Uses Part B		teliminary				
📕 🖵 🖓 Uses Par	Are you sure you want to promote	ScrewModel, Preliminary				
🖕 🏳 Uses Part BOM (P	Engine, A, 1, Engine, Preliminary?	iminary				
🚽 🗧 🖓 Uses Part BOM		conBoltAssm, Preliminary				
🚽 🖕 🌣 Uses Part B	Yes No	etainingRing, Preliminary				
Uses Part	BOM (Part BOM, A, 1) / RetainingRingN	Nodel, A, 1, RetainingRingMode				
😑 🗘 Uses Part B	DM (Part BOM, A, 1) / PistonBolt, A, 1, F	PistonBolt, Preliminary				
Uses Part	BOM (Part BOM, A, 1) / PistonBoltMode	el, A, 1, PistonBoltModel, Prelim				
🗧 🗢 🗘 Uses Part BOM	Uses Part BOM (Part BOM, A, 1) / Piston, A, 1, Piston, Preliminary					
Uses Part BOM (Part BOM, A, 1) / PistonModel, A, 1, PistonModel, Preliminary						
Uses Part BOM (P	art BOM, A, 1) / ConnectionRod, A, 1, C	ConnectionRod, Preliminary				
Uses Part BOM	(Part BOM, A, 1) / ConnectionRodMode					

Picture 38: Confirm the "Promote" action

The selected object and the attached documents will be promoted. In this example from the life cycle state "Preliminary" to the state "In Review" (see *Picture 39: Object is promoted*).



Picture 39: Object is promoted

Revise

The PDM objects can be revised if the item is in released mode.

You select the object in the PDM Workbench window and click the right mouse button. In the context menu you select the "Revise" context action (see *Picture 40: Action "Revise"*).

🛐 Engine, A, 1, Engine, Released (Logged in as admin)					
Engine, A, 1, Engine, R		Center graph			[
🖕 🍄 Uses Part BOM (Par		Reframe On			reliminary
🚽 🝦 🆓 Uses Part BOM (F		Hide/Show			rBlock, Preliminary
Uses Part BON		Properties	Alt+Enter		iderblock, Preliminary
😑 🗘 Uses Part BOM (F		Open Sub-Tree			Y
🗧 🗘 Uses Part BON		Cut	Ctrl+X		nary
📕 🖌 🛶 Uses Part B		⊆ору	Ctrl+C		eModel, Preliminary
🚽 🗘 🚽 🗘 Uses Part BON		Paste	Ctrl+V		iminary
📕 🚽 🖓 Uses Part B		Paste Special			rewModel, Preliminary
Uses Part BOM (Par		Delete	Del		linary
- Uses Part BOM (F		Delete	Der		nBoltAssm, Preliminary
Uses Part BON		Engine, A, 1, Engine	Released object	+	ainingRing, Preliminary
Uses Part B		Expand		•	A, 1, RetainingRingMode
🚽 🗧 🗘 Uses Part BON		Expand Multiple Leve	ls	•	olt, Preliminary
Uses Part B		De <u>-</u> Expand			, PistonBoltModel, Prelim
Uses Part BOM (F		Open in new PDM Wi	ndow		hary
Uses Part BON		Unlock			nModel, Preliminary
- OUses Part BOM (Par		Lock			tionRod, Preliminary
Uses Part BOM (F	7	Pro <u>m</u> ote			, ConnectionRodModel,
	1,2	Re <u>v</u> ise			

The part and the document have to be released in order to be revised.

Picture 40: Action "Revise"

You will be asked if you really want to revise the object. Please confirm with "Yes" (see *Picture 41: Confirm the "Revise" action*).



Picture 41: Confirm the "Revise" action

The selected object will be revised (see Picture 42: Object is revised).



Picture 42: Object is revised

The new object was added in the window on the top.

Update structure relations

This functionality is only available for the document data model.

You have the possibility to update an object with a new revision of an already used object.

In this example the document "FixedParts_Assm" has been revised from "A" to "B". The revision "A" was already used by the document "Engine".

You have to select the CATIA Product "Engine" and choose "Update structure relations" (see *Picture 43: Action "Update structure relations"*).

CATProduct (Logged in	as admin)	
FixedParts_As	sm, B, 2, , Preliminary, Mechanical/Assembly,(version:2)
Engine, A, 🛀	Center graph	
- 🚽 🐼 CAD Suł	Center graph Reframe On	Parts_Assm, A, 1, , F
	Hide/Show	erBlock, A, 1, , Prelim
	Properties Alt+Ente	
	Open_Sub-Tree	
	CutCtrl+3	
▎▎▕▔▓▁▞॒હ	Ctrl+0	, Preliminary, Mechar
🖓 🕰	Paste Ctrl+1	ckModel, A, 1, , Prelii
	Paste Special	L, , Preliminary, Mech
	<u>D</u> elete De	rewModel, A, 1, , Pro
	Engine, A, 1, , Preliminary, Mechanical/Assembly, (version:1) object	L, , Preliminary, Mech
	Expand	• rewModel, A, 1, , Pro
	Expand Multiple Levels	L, , Preliminary, Mech
	De_Expand	rewModel, A, 1, , Pre
	Open in new PDM Window	L, , Preliminary, Mech
	Lock	crewModel, A, 1, , Pro
- CAD Sul	2 Revise	_Assm, A, 1, , Prelim
📘 🕴 🗧 🙀 📥	Update structure relations	onPin_Assm, A, 1, , F
📕 🚽 🚽 🖏 CA 🐔	, Load	ng, A, 1, , Preliminary
📘 L 🔊 🖏	Highlight CATIA Nodes	/ SnapRingModel, A,
🔽 🚣 🖓 🛱 🕼	Copy element attributes	stonPincomp, A, 1,

Picture 43: Action "Update structure relations"

Now the new revisions of the used objects are related to this object and the relations to the old revisions are deleted.

You can see that the revision "B" of the "FixedParts_Assm" is used by the "Engine" now (see *Picture 44: Updated structure relations*).

CATProduct (Logged in as admin)	
FixedParts_Assm, B, 2, , Preliminary, Mechanical/Assembly, (version:2)	
Engine, A, 1, , Preliminary, Mechanical/Assembly, (version:1)	
CAD Sub-Structure (CAD Structure, Crankshaft_Assm.1) / Crankshaft_Assm, A, 1, , Pr	
ench	<u>? ×</u>
)	
adon from FixedParts_Assm to FixedParts_Assm	Close
2	
2	FixedParts_Assm, B, 2, , Preliminary, Mechanical/Assembly, (version:2) Engine, A, 1, , Preliminary, Mechanical/Assembly, (version:1) CAD Sub-Structure (CAD Structure, FixedParts_Assm.1) / FixedParts_Assm, B, 2, , Pre CAD Sub-Structure (CAD Structure, Cylinder_Assm.1) / Cylinder_Assm, A, 1, , Prelimin CAD Sub-Structure (CAD Structure, ConnectionRod.1) / ConnectionRod, A, 1, , Prelimin CAD Sub-Structure (CAD Structure, Crankshaft_Assm.1) / Crankshaft_Assm, A, 1, , Prelimin CAD Sub-Structure (CAD Structure, Crankshaft_Assm.1) / Crankshaft_Assm, A, 1, , Prelimin CAD Sub-Structure (CAD Structure, Crankshaft_Assm.1) / Crankshaft_Assm, A, 1, , Prelimin Rob End

Picture 44: Updated structure relations

Update parent relation

This functionality is only available for the document data model.

Like "Update structure relations", this functionality updates the structure relations of a used document to the latest version of that document. The difference is that in this case, only the selected child node (all instances of the document) is updated, not all the direct child nodes of the parent document.

The context action is only available for child nodes in a structure, not for the root node.

🋐 Engine, A, 1, Engin	e - Nomenclature, Preliminary, Mechanical/Assembly, (Logged in as adm
🔯 Engine, A, 1, Er	ngine – Nomenclature, Preliminary, Mechanical/Assembly,
- 🔊 CAD Sub-S	tructure (/CAD Structure/Structure, PistonAssm.1) / PistonAssm,
	tructure (/CAD Structure/Structure, ConnectionRod.1) / Connect
CAD Sub-	
	C <u>e</u> nter graph
CAD Sub-	<u>R</u> eframe On
	🔗 Hide/Show
	Properties
	는 Ope <u>n</u> Sub-Tree
	Cu <u>t</u>
	📄 Сору
	Paste
	Paste <u>S</u> pecial
	<u>D</u> elete
	CrankShaft, A, 1, CrankShaft - Nomenclature, Preliminary, Mechanical
	Expand
	Expand <u>M</u> ultiple Levels
· · · ·	📙 De_Expand
<u> </u>	🕫 Open <u>i</u> n new PDM Window
1	🖳 Unlock
· · · · · · · · · · · · · · · · · · ·	🖀 Lock
· · · · · · · · · · · · · · · · · · ·	192 Update structure relations
	1+2 B Update parent relation
1	Load
1	🖌 Add Temp
	🨼 Highlight CATIA Nodes

Picture 45: Action "Update parent relation"

The relation has been updated (see Picture 47: Updated structure relation).

En	gine, A, 1, Engine - Nomenclature, Preliminary, Mechanical/Assembly, (Logged in as admi
	ngine, A. 1, Engine - Nomenclature, Preliminary, Mechanical/Assembly,
P	DM-Workbench
Г	Information
	PDM Messages:
	Moved relation from CrankShaft (A.1) to CrankShaft (A.2)
	Action succeeded.

Picture 46: Updated relation in PDM

The structure relations will have to be expanded again to show the current status (see *Picture 47: Updated structure relation*):

Engine, A, 1, Engine - Nomenclature, Preliminary, Mechanical/Assembly, (Logged in as admin)
Engine, A. 1, Engine - Nomenclature, Preliminary, Mechanical/Assembly,
CAD Sub-Structure (/CAD Structure/Structure, FixedParts.1) / FixedParts, A, 1, FixedParts - N
CAD Sub-Structure (/CAD Structure/Structure, PistonAssm.1) / PistonAssm, A, 1, PistonAssr
CAD Sub-Structure (/CAD Structure/Structure, ConnectionRod.1) / ConnectionRod, A, 1, Co
CAD Sub-Structure (/CAD Structure/Structure, CrankShaft.1) / CrankShaft, A, 2, CrankShaft

Picture 47: Updated structure relation

Delete

Existing PDM objects can be deleted from the PDM Workbench. Delete is a set based operation, which means multiple objects can be deleted in one action.

Select objects in the PDM Workbench window and from the contextual menu choose "Delete" (see *Picture 48: Action "Delete"*).

Same and the second sec	lmin)		
 NewProduct1, A, 1, NewProduct1, NewProduct2, A, 1, NewProduct2, NewPart1, A, 1, NewPart1, Prelimin. NewPart2, A, 1, NewPart2, Prelimin. NewPart3, A, 1, NewPart3, Prelimin. 		Center graph Reframe On Hide/Show Properties Open Sub-Tree Cut: Copy Paste Paste Special	Alt+Enter Ctrl+X Ctrl+C Ctrl+V	Ļ
		<u>D</u> elete	Del	

Picture 48: Action "Delete"

A confirmation message box is displayed listing the objects which will be deleted. When you confirm this dialog with "Yes" the objects will be deleted.

🛐 Assembly, Componen	t (Logged in as admin)	_ 🗆 ×
 NewProduct NewProduct NewPart1, A NewPart2, A NewPart3, A 	Are you sure you want to delete NewPart3, A, 1, NewPart3, Preliminary NewPart2, A, 1, NewPart2, Preliminary NewPart1, A, 1, NewPart1, Preliminary NewProduct2, A, 1, NewProduct2, Preliminary NewProduct1, A, 1, NewProduct1, Preliminary? CATIA files related to BOM parts will also be deleted.	7
	Yes No	γ,∫_×

Picture 49: Confirm Delete objects

The operation result dialog is displayed containing error or success messages.

PDM-Workbench	? ×
_ Information	
Deleted PDM object NewPart1 (Mechanical/Part), Deleted PDM object NewPoduct2 (Assembly), Deleted PDM object NewPoduct2 (Mechanical/Assembly), Deleted PDM object NewPoduct1 (Mechanical/Part), Deleted PDM object NewPoduct1 (Mechanical/Part), Deleted PDM object NewPoduct1 (Assembly), Deleted PDM object NewPoduct1 (Mechanical/Assembly), Deleted PDM object NewPort3 (Component), Deleted PDM object NewPart3 (Component), Deleted PDM object NewPart3 (Mechanical/Part),	×
Action succeeded.	
	Close

Picture 50: Delete result window

When you delete a part object the attached documents will be deleted, too.

Delete newest version

You can delete the newest version of the file when there exist more than one version for the file and you see that you do not need this version anymore because you want to design the geometry a different way.

You have to select the last version of the document and click on the right mouse button. The context menu will be opened. There you select "Delete newest version" (see *Picture 51: Action "Delete newest version"*).

ScrewModel, A, 1, ScrewModel, Preliminary	(Logged in as admin)	_ 🗆 2
ScrewModel, A, 1, ScrewModel, Prelimin	ary	
Attaches (Part CAD) / ScrewModel, A	, 2, Carbon and Carbon and Carbo	
	Center graph	
	Reframe On	
	Hide/Show	
	Properties	Alt+Enter
	Open Sub-Tree	
	Cu <u>t</u>	Ctrl+X
	⊆ору	Ctrl+C
	🖺 Paste	Ctrl+V
	Paste Special	
	Delete	Del
	Scre <u>w</u> Model, A, 2, ScrewModel, Preliminary, Med	:hanical/Part, Test object 🔹 🕨
	E⊻pand	•
	Te_Expand	
	Open in new PDM Window	
	The Unlock	
	Euck	
	Promote	
	1+2 Revise	
	1+2 B	
	AR Open File with Link	
	Open File temporarily	
	Highlight CATIA Nodes	
	Update Item	
	Relation	
	T Duplicate	

Picture 51: Action "Delete newest version"

Then you are asked to confirm the delete of the newest version. You have to click the "Yes" button (see *Picture 52: Confirm Delete newest version*).

ScrewModel, A, 1, ScrewModel, Preliminary (Logged in as admin)	_ 🗆 X
ScrewModel, A, 1, ScrewModel, Preliminary	
Attaches (Part CAD) / ScrewModel, A, 2, ScrewModel, Preliminary, Mechanical/Part, Test	
Delete PDM Relations?	
Are you sure you want to delete	
Attaches (Part CAD) / ScrewModel, A, 2, ScrewModel, Preliminary, Mechanical/Part, Test?	
nichanicay raky rake	
Yes No	

Picture 52: Confirm Delete newest version

The newest version will be deleted. The document will be removed from the window (see *Picture 53: Delete newest version result window*).

ScrewModel, A, 1, ScrewModel, Preliminary (Logged in as admin)	
ScrewModel, A, 1, ScrewModel, Preliminary	
PDM-Workbench	? ×
_ Information	
PDM Messages:	
Deleted PDM object ScrewModel (Mechanical/Part).	
Action succeeded.	
	Close

Picture 53: Delete newest version result window

You have to re-expand the document in order to see the document that is attached to the component (see *Picture 54: Re-Expand of the document*).

ScrewModel, A, 1, ScrewModel, Preliminary (Logged in as admin)	
ScrewModel, A, 1, ScrewModel, Preliminary Attaches (Part CAD) / ScrewModel, A, 1, , Preliminary, Mechanical/Part,	

Picture 54: Re-Expand of the document

Open File

This functionality is only available for the document data model.

You can open a single CATIA V5 Drawing file existing in the PDM data base with the PDM Workbench in the native CATIA V5 window.

You can also open a single CATIA V5 Part or CATIA V5 Product file existing in the PDM data base in the native CATIA V5 window.

To open the file in CATIA V5 you select the PDM file object in the PDM Workbench window and click the right mouse button to open the context menu. There you select the context action "Open File" (see *Picture 55: Action "Open File"*).

EngineDrawing, A, 1, , Preliminary, N	lechanical/Drawing, (version:1) (Logged in as admin)		- D ×
EngineDrawing, A, 1, 🖻	liminary Machanical/Drawing Averaiant	•	1
	C <u>e</u> nter graph		
	<u>R</u> eframe On		
	Hide/Show		
	Properties	Alt+Enter	
	Open Sub-Tree		
8	Cut	Ctrl+X	
	⊆ору	Ctrl+C	
	Paste	Ctrl+V	
	Paste Special		
	Delete	Del	
	EngineDrawing, A, 1, , Preliminary, Mechanical/Drawing, (version	on:1) object 🔹 🕨	
	Expand	•	
	De <u>-</u> Expand		
1	Open in new PDM Window		
	Unlock		
	Lock		
7	Promote		
1-2	Re <u>v</u> ise		
	Open <u>F</u> ile		
1	Open File Tempor <u>a</u> ry		ſ
	Copy element attributes		X~Y

Picture 55: Action "Open File"

The PDM Workbench downloads the required CAD file to the client's special PDM Workbench exchange map. You can watch the download progress on the "Open File" progress bar (see *Picture 56: Open File - progress bar*).

Open File	×
AB	Downloading selected file object from PDM system
	Downloading CAD files
)II	
Status : Estimated time remaining	:

Picture 56: Open File - progress bar

The geometry opens in its corresponding CATIA V5 native window (see *Picture 57: Split window after Open File – PDM Workbench node and CATIA drawing*).

In the above window (PDM Workbench window) you see the selected PDM file object. The window on the bottom shows the loaded CATDrawing.



Picture 57: Split window after Open File – PDM Workbench node and CATIA drawing

Open File Temporary

The action "Open File Temporary" allows the user to visualize a temporary file of a different version together with the working version.

In the first step you load the current version of the file. In this example you open the version "2" of the "ScrewModel".

You select the object and click on the right mouse button. In the context menu you select the action "Open File" (see *Picture 58: Action "Open File"*).



Picture 58: Action "Open File"



The current version of the file is loaded in CATIA V5, now (see Picture 59: Current file).

Picture 59: Current file

Then you query for a different version (in this case version "1") and open the file temporary.

You select the object and click on the right mouse button. In the context menu you select the action "Open File Temporary" (see *Picture 60: Action "Open File Temporary"*).



Picture 60: Action "Open File Temporary"

The version "1" of the file is opened temporary in CATIA V5 (see *Picture 61: Temporary opened file*).

The Part Number and the File Name of the temporarily opened geometry are prefixed with "TMP#_", where "#" is a counter in CATIA V5, beginning with 1. Every action "Open File Temporary" will increase the counter. This prefix is customisable by the customer. For details please refer to the *PDM Workbench Installation & Administration Manual.*



Picture 61: Temporary opened file

Now you can compare the both versions of the file.

Load

It is possible to load geometry corresponding to an expanded PDM product structure into a native CATIA V5 window in order to work on it, e.g. perform geometric transformations, geometry changes and so on.

To load the geometry in CATIA V5 you select the root PDM object wherefrom downward you want to get the geometry and click the right mouse button to open the context menu and you select the context action "Load" (see *Picture 62: Action "Load"*).

🛐 Engine, B, 2, Engine, Preliminary	(Logged in as admin)		_ 🗆 🗵
Engine, B, 2, Engine, F	C <u>e</u> nter graph <u>R</u> eframe On		reliminary
=-○ <u>Uses Part BOM (</u> Uses Part BOM (=-○ <u>Uses Part BOM (</u> E	Hide/Show Properties Open Sub-Tree	Alt+Enter	rBlock, Preliminary derblock, Preliminary y
Contraction of the second seco	Cu <u>t</u>	Ctrl+X	hary Model, Preliminary
e-O Uses Part BO Lo Uses Part BO	⊆opy Paste	Ctrl+C Ctrl+V	minary wModel, Preliminary
- Ouses Part BOM (Par	Paste Special	Del	nary
Uses Part BOM (Engine, B, 2, Engine, Prelimin	ary object 🔹 🕨	BoltAssm, Preliminar iningRing, Preliminar
=-☆ <u>Uses Part F</u> =-☆ <u>Uses Part BO</u>	Expand Expand Multiple Levels	•	1, 1, RetainingRingMe olt, Preliminary
uses Part I -☆ Uses Part BOM (De <u>-</u> Expand		PistonBoltModel, Pre
	Open in new PDM Window <u>U</u> nlock		Model, Preliminary
Uses Part BOM (Par Uses Part BOM (Par Uses Part BOM (Lock Pro <u>m</u> ote		tionRod, Preliminary ConnectionRodMoc
	Re <u>vi</u> se Load		Preliminary IkShaftRight, Prelimir
	Load <u>w</u> ith Links		1. GrankSĥaftPi

Picture 62: Action "Load"

The PDM Workbench downloads the corresponding CAD files (CATParts, CATProducts, and CATDrawings) to the client's PDM Workbench exchange map. The download progress is indicated by the "Load" progress bar (see *Picture 63: Load - progress bar*).

Load	2	×
1	Loading selected object from PDM system	
	Downloading files	
Status :	50% completed	
Estimated time remaining	: Osec	
	Cancel	

Picture 63: Load - progress bar

The geometry downloaded opens in a CATIA V5 native window (see *Picture 64: Split window after Load – PDM Workbench and CATIA V5 nodes*).

In the left window (PDM Workbench window) you see the expanded product structure wherefrom you opened the CATIA V5 native window presenting the geometry on the right. In the right window you see the loaded geometry.



Picture 64: Split window after Load – PDM Workbench and CATIA V5 nodes

Add Temp

The action "Add Temp" allows the user to visualize a temporary structure together with the working one.

In the first step you load the current version of the structure. In this example you open the revision "B" of the "Engine".

You select the object and click on the right mouse button. In the context menu you select the action "Load" (see *Picture 65: Action "Load"*).



Picture 65: Action "Load"

The current version of the structure is loaded in CATIA V5, now (see *Picture 66: Loaded geometry for revision "B"*).



Picture 66: Loaded geometry for revision "B"

Then you query for a different version (in this case revision "A") and load the structure temporary.

You select the object and click on the right mouse button. In the context menu you select the action "Add Temp" (see *Picture 67: Action "Add Temp"*).

🛐 Engine, B, 2, Engine, Preliminary (Lo	gged in as admin)	
 Engine, B, 2, Engine, Prelimination Uses Part BOM (Part BC Uses Part BOM (Part BOM (Par	Properties Alt+Enter	inary :k, Preliminary lock, Preliminary
- OUses Part BOM (P) - OUses Part BOM (P) - OUses Part BOM (P) - OUses Part BOM (P) - OUses Part BOM (P)	Open_Sub-Tree Cut_ Ctrl+X Copy Ctrl+C Paste Ctrl+V Paste_Special Ctrl+V	el, Preliminary ry odel, Preliminary
- Uses Part BOM (Part BC Uses Part BOM (Part - Uses Part BOM (Part - Uses Part BOM (Part - Uses Part BOM (Part - Uses Part BOM -	Delete Del Engine, B, 2, Engine, Preliminary object Expand	Assm, Preliminary , Ring, Preliminary , RetainingRingModel, Prel
÷- [©] Uses Part BOM (Pa [©] Uses Part BOM [©] Uses Part BOM (Part [©] Uses Part BOM (Part [©] Uses Part BOM (Part	Expand Multiple Levels De_Expand Open in new PDM Window Unlock	reliminary pnBoltModel, Preliminary el, Preliminary
- OUses Part BOM (Part)	☐ Lock ▶ Promote •2 Revise	kod, Preliminary inectionRodModel, Prelim hinary affRight, Preliminary
Uses Part BOM (Part	Load Load with Links Add Temp Highlight CATIA Nodes	CrankShaftRightModel, F minary

Picture 67: Action "Add Temp"

The CATProducts will not be loaded. Only the CATParts are loaded and positioned correctly (see *Picture 68: Loaded geometry for revision "A"*).

All Part Numbers and File Names in the temporarily added structure are prefixed with "TMP#_", where "#" is a counter in CATIA V5, beginning with 1. Every action "Add Temp" will increase the counter. This prefix is customisable by the customer. For details please refer to the *PDM Workbench Installation & Administration Manual.*



Picture 68: Loaded geometry for revision "A"

Now you can compare both geometry structures.

Duplicate

It is possible to duplicate existing CATProduct and CATPart objects.

You have to open the objects in CATIA V5 and click on the right mouse button in order to open the context menu. There you have to select the action "Duplicate" (see *Picture 69: Action "Duplicate"*).

For CATParts you have to use the "Duplicate" action in the "PDM Workbench context commands" toolbar.

Engine.CA1	Product				<u>_0×</u>
Engine - S Fixe - S Cyli - S Con - S Cra - S Cra	Center graph Beframe On Elide/Show Popenties Cote Cote	Alt+Enter Ctrl+X Ctrl+C Ctrl+V Del	s_Assm.1) sm.1) Rod.1) ft_Assm.1)		
	PD <u>M</u> -Workbench	•	PDM Properties	1. And Car)
	Components Represent <u>a</u> tions	•	Highlight PDM Nodes		×
	Selection Mode	•	Duplicate		

Picture 69: Action "Duplicate"

The "PDM Create" dialog will be opened. The correct type is already selected. The identifying name of the document is filled, too.

You have to fill or change the attributes (see *Picture 70: "PDM Create" dialog for duplicate*). The CATIA object has the temporary suffix "-DUPLICATE".

Engine.CATProduct	PDM Create ?	X	<u> </u>
Engine-DUPLICATE	Create dialog for type Assembly:		
🕂 😽 FixedParts_Assm (Fixe	Part Number Engine *		
🕂 🐝 Cylinder_Assm (Cylinde	Name Engine Unit EA		
ConnectionRod (Conne	Make / Buy Make		
🕂 😽 Crankshaft_Assm (Cra			
+- Constraints	Description		
 Applications 			
	Create dialog for type CATProduct:	<u>1</u>	
	Doc. Number Engine	*	
	Name Description		
		. ((0	
		0	
			7
	OK Apply Cance		¥_Ľx

Picture 70: "PDM Create" dialog for duplicate

When you close the dialog with "OK" (see *Picture 71: Filled "PDM Create" dialog for duplicate*) the CATIA document will be renamed to the new part number and created in the PDM system (see *Picture 72: Duplicated CATProduct object*).

Engine.CATProduct	PDM Create	? X	_ 🗆 🗵
Pagine-DUPLICATE	Create dialog for type Assembly:		
🕂 🐝 FixedParts_Assm (Fixe		*	$\langle \rangle$
🕂 🐝 Cylinder_Assm (Cylinde	Name EngineNew		
+-😽 ConnectionRod (Conne	Unit EA Make / Buy Make		
+-🐝 Crankshaft_Assm (Crar	Cost		
+- Constraints	Description		
 Applications 			
- Applications			
	Create dialog for type CATProduct:		
	Doc. Number EngineNew	*	
	Name EngineNew Description		
		0	
	1	L.C.	7
	OK Apply	Cancel	¥-Ĺ×

Picture 71: Filled "PDM Create" dialog for duplicate



Picture 72: Duplicated CATProduct object

When a CATDrawing object is related to the CATProduct and the file name and the document name of the CATDrawing ("Product-No-100-Drawing.CATDrawing") includes the part number of the CATProduct ("Product-No-100") as prefix then the CATDrawing can be duplicated and related to the CATProduct.

You load the product structure with the drawing. They are opened in two CATIA windows. Then you select the top product and open the context menu and select the action "Duplicate". The product will be renamed with the suffix "-DUPLICATE" and the "PDM Create" dialog will be opened. The document name is filled (see *Picture 73: "PDM Create" dialog for Assembly and CATProduct*).

	100, A, 1, Product100, Preliminary) (Product100, A, 1, Product100, Preliminary)	
Product100.CATProduct	PDM Create ? X	
Product100-DUPLICATE	Create dialog for type Assembly: Part Number Product100 Name Product100 Unit Wake / Buy Cost Cost Description	
	Create dialog for type CATProduct: Doc. Number Product100 Name Product100 Description OK Apply Cancel	Ļ,

Picture 73: "PDM Create" dialog for Assembly and CATProduct

You change and fill the dialog and select the "OK" button (see *Picture 74: Filled "PDM Create" dialog for Assembly and CATProduct*).

N Pr	oduct100-Drawing.CATDrawing (Product10)		nary)
<u>r</u>	Product100-Drawing	1	
	Product100.CATProduct		-0×
	Product100-DUPLICATE	PDM Create ? 🗙	
	+-• Product100-DopEleATE	Create dialog for type Assembly:	${\sim}$
		Part Number Product100New *	
	-Applications	Name Product100New	γ V
		Unit	
		Make / Buy	\sim
		Cost	
		Description	
		Create dialog for type CATProduct:	
		Doc. Number Product100New *	
		Name Product100New	
		Description	
		and the second	
			Z
		OK Apply Gancel	×

Picture 74: Filled "PDM Create" dialog for Assembly and CATProduct

The CATProduct will be renamed to the desired name and created in the PDM system. The "PDM Create" dialog for the CATDrawing will be opened. The part number and the name of the assembly are filled. The action is set to "Duplicate Part" (see *Picture 75: "PDM Create" dialog for Assembly and CATDrawing*).

	uct100, A, 1, Product100, Preliminary) (Product100, A, 1, Product100, Preliminary)	×
🖕 Product100-Drawing-Dl	Sheet.1 PDM Create ? X	
🖵 🗖 <mark>Sheet.1</mark>	Create dialog for type Assembly:	٦×
	Part Number Product 100New *	
	Name Product100New	
	Unit Y Make / Buy	_
	Cost	
	Description	-11
		-11
	Create dialog for type CATDrawing:	
	Duplicated Part 💌 *	
	Doc. Number Product100-Drawing *	
	Name Product100-Drawing Description	
		- []:
-	OK Apply Gancel	—у
I I I I I I I I I I I I I I I I I I I		

Picture 75: "PDM Create" dialog for Assembly and CATDrawing

You change and fill the dialog. The Document Name should have the Part Number of the Assembly as prefix. Then you start the process with the "OK" button (see *Picture 76: Duplicated CATProduct object*).



Picture 76: Duplicated CATProduct object

The drawing document will be created in the PDM system and related to the assembly object. Analogous you can duplicate a CATPart with its CATDrawing.

Create Relation between windows

You might modify the product structure by adding existing objects from several PDM Workbench windows to the structure in another PDM Workbench window.

You select the object you want to copy and click the right mouse button to get the context menu. Then you select the context action "Copy" (see *Picture 77: Action "Copy" between windows*). Of course you also can use the short cut "CTRL+C".



Picture 77: Action "Copy" between windows

Then you select the object where you want to add the copied object to and click the right mouse button to open the context menu. You select the context action "Paste" (see *Picture 78: Action "Paste" between windows*). Of course you also can use the short cut "CTRL+V".

📲 Engine, B, 2, Engine, Preliminary	(Logged in as admin)	_ 🗆 🗙	NewProduct1, A, 1, NewProduct1, Preliminary (Logged in as admin)	
C Engine, B, 2, Engine, Pre	Center graph		NewProduct1, A, 1, NewProduct1, Preliminary	
Uses Part BOM (Part E	Reframe On	minary		
	Hide/Show	ock, Pr		2
	Properties Alt+Enter	rblock,		9
Service Part BOM (Pa	Open Sub-Tree			7
- Ouses Part BOM	Cu <u>t</u> Ctrl+X	У		
Uses Part BO	⊆opy Ctrl+C	odel, Pr		17
	Paste Ctrl+V	hary		1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
Uses Part BO	Paste Special	/Model		
- 🖓 Uses Part BOM (Part F - 🖓 Uses Part BOM (Pa	Delete Del	iry oltAssm		50
Uses Part BOM	Engine, B, 2, Engine, Preliminary object			\$0
Uses Part BO		, 1, Reta		
- Uses Part BOM	Expand	Prelim		A. K.
2 Line Date Do	Expand Multiple Levels	stonBc		1
=- Uses Part BOM (Pa	De_Expand	V		1
- Uses Part BOM (Pa	Open in new PDM Window	, pdel, Pr		
Uses Part BOM (Part F	Uniock	nRod,		
Louises Part BOM (Pa	Dramata	onnect		
Uses Part BOM (Part I		eliminar		
- Uses Part BOM (Pa		ShaftRig		
Uses Part BOM		1, Cran		
- O Uses Part BOM (Pa		eliminar		
Uses Part BOM		kBoltM		
🗸 🗘 Uses Part BOM (Pa 🖓		haft_eft		
L 🔆 Uses Part BOM 👸		Crank:	ľ ľ	
×	Duplicate	Y	×~~ Y	
	🔊 🗠 🙀 🖢 for 🔗 🔋 🔳 i			

Picture 78: Action "Paste" between windows

You specify the relation you want to create in the structure between the two objects (see *Picture 79: Select the new relation*) once there is more than one relation type available. In the dialog window you see all relations possible between the two objects. In case you want to add the Assembly object in the structure to another Assembly object then you might choose the "Part BOM" relation for example. The dialog will not appear if there is only one relation type available.

Part BOM Part BOM 2	
	Close

Picture 79: Select the new relation

The instance object gets inserted into the existing product structure tree and the new relation gets created in the PDM system (see *Picture 80: Product Structure with inserted object*).

<mark>¶CATIAVS</mark> <mark>≩art E</mark> lle Edit <u>Vi</u> ew Insert Iools <u>W</u> indow <u>H</u> elp	
🛐 Engine, B, 2, Engine, Preliminary (Logged in as admin)	🔲 🗙 🛐 NewProduct 1, A, 1, NewProduct 1, Preliminary (Logged in as admin) 📃 🗖 🗙
Engine, B, 2, Engine, Preliminary	NewProduct1, A, 1, NewProduct1, Preliminary
- 🖓 Uses Part BOM (Part BOM, A, 1) / FixedParts, A, 1, FixedParts, Prel	iminary
🚽 💬 Uses Part BOM (Part BOM, A, 1) / CylinderBlock, A, 1, CylinderBl	lock, Pr
Uses Part BOM (Part BOM, A, 1) / Zylinderblock, A, 1, Zylinde	rblock,
Uses Part BOM (Part BOM, A, 1) / Rack, A, 1, Rack, Preliminary	
- 🔆 Uses Part BOM (Part BOM, A, 1) / Case, A, 1, Case, Preliminal	'y
Uses Part BOM (Part BOM, A, 1) / CaseModel, A, 1, CaseM	odel, Pr
- 🖓 Uses Part BOM (Part BOM, A, 1) / Screw, A, 1, Screw, Prelimi	nary
L 🖓 Uses Part BOM (Part BOM, A, 1) / ScrewModel, A, 1, Screv	vModel,
🗝 😳 Uses Part BOM (Part BOM, A, 1) / Cylinder, A, 1, Cylinder, Prelimina	ary .
- 🖓 Uses Part BOM (Part BOM, A, 1) / PistonBoltAssm, A, 1, PistonBo	oltAssm
🚽 🔆 Uses Part BOM (Part BOM, A, 1) / RetainingRing, A, 1, Retain	ingRing
Uses Part BOM (Part BOM, A, 1) / RetainingRingModel, A,	1, Reta
🗢 😳 Uses Part BOM (Part BOM, A, 1) / PistonBolt, A, 1, PistonBolt	, Preim
🦳 🦾 Uses Part BOM (Part BOM, A, 1) / PistonBoltModel, A, 1, P	stonBc
= 😳 Uses Part BOM (Part BOM, A, 1) / Piston, A, 1, Piston, Preliminar	У
🦳 🤤 Uses Part BOM (Part BOM, A, 1) / PistonModel, A, 1, PistonM	odel, Pr
- 💭 Uses Part BOM (Part BOM, A, 1) / ConnectionRod, A, 1, Connectio	nRod,
└── 🗘 Uses Part BOM (Part BOM, A, 1) / ConnectionRodModel, A, 1, C	onnect
- 🌣 Uses Part BOM (Part BOM, A, 1) / Crankshaft, A, 1, Crankshaft, Pre	eliminar
😑 😳 Uses Part BOM (Part BOM, A, 1) / CrankShaftRight, A, 1, Cranks	ShaftRi
└ CrankShaftRightModel, A, 1) / CrankShaftRightModel, A,	1, Cran
= 🗘 Uses Part BOM (Part BOM, A, 1) / CrankBolt, A, 1, CrankBolt, Pr	eliminar
South Contract South Contract South South Contract	kBoltM
Ü Uses Part BOM (Part BOM, A, 1) / CrankShaftLeft, A, 1, CrankSh	naftLeft
└ CrankShaftLeftModel, A, 1, 1) / CrankShaftLeftModel, A, 1,	Crank! 7
Uses Part BOM (Part BOM, A, 1) / NewProduct1, A, 1, NewProduct	
🗋 🗃 🖶 🛃 💃 🛅 🖺 ဟု ભု 🕵 🕇 👘 🔗 🕴 📑 🖷	
ect an object or a command	

Picture 80: Product Structure with inserted object

Update

The Update functionality can be used to create, to complete, or to update the product structure in the PDM system based on the geometry in the CATIA V5 window.

You can start the Update process by clicking on the "Update" icon

A dialog opens and asks to confirm the described actions. In this example the CATIA documents will be created (see *Picture 81: Confirm the Update (with Create) action*).

ile name	PDM Display	Locked By	PLM State	Operation	Result	
ewCylinderBlockModel.CAT				Create		
ewRackModel.CATPart				Create		
ewScrewModel.CATPart				Create		
ewRack.CATProduct				Create		
ewScrew.CATProduct				Create		
ewCylinderBlock.CATProdu				Create		
ewRack_Assm.CATProduct				Create		
ewSnapRingModel.CATPar	t			Create		
ewPistonPincompModel.CA	TPart			Create		
ewSnapRing.CATProduct				Create		
ewPistonPincomp.CATProd	uct			Create		
						Update

Picture 81: Confirm the Update (with Create) action

The progress of the Update will be shown with the progress bars (see *Picture 82: Progress bars for Update action*).

	×
8	"NewCylinderBlock"
111	
Status : Estimated time remaining :	4% completed 5min 42sec
Update in PDM	×
Updating change:	s to PDM system
	Uploading files to PDM
Status : Estimated time remaining :	40% completed 3sec

Picture 82: Progress bars for Update action

When the Update process has finished you are informed about the actions that have been performed. The related instances and the created objects are reported in the information window (see *Picture 83: Information window when Update (with Create) is finished*).

File name	PDM Display	Locked By	PLM State	Operation	Result	
lewCylinderBlockModel.CATPart				Create	SUCCESS	
lewRackModel.CATPart				Create	SUCCESS	
NewScrewModel.CATPart				Create	SUCCESS	
lewRack.CATProduct				Create	SUCCESS	
lewScrew.CATProduct				Create	SUCCESS	
lewCylinderBlock.CATProduct				Create	SUCCESS	
lewRack_Assm.CATProduct				Create	SUCCESS	
lewSnapRingModel.CATPart				Create	SUCCESS	
lewPistonPincompModel.CATPart				Create	SUCCESS	
vewSnapRing.CATProduct				Create	SUCCESS	
NewPistonPincomp.CATProduct				Create	SUCCESS	
Treated '/Part/Assembly' with name 'NewEr Io main file PDM item exists yet, creating it Treated '/CAD/Assembly Model' with name' Io relation exists between part and main fil Treated 'Part BOM' relation between 'NewE	NewEngine' e yet, creating it.					

Picture 83: Information window when Update (with Create) is finished

If there are no objects to be created then it will only be asked if you want to update (see *Picture 84: Confirm the Update action*).

-ile name	PDM Display	Locked By	PLM State	Operation	Result	
lewCylinderBlockModel.CATPart	NewCylinderBlockModel, , Part Model, ,	admin	Preliminary	Update		
lewRackModel.CATPart	NewRackModel, , Part Model, , Prelimin	admin	Preliminary	Update		
ewScrewModel.CATPart	NewScrewModel, , Part Model, , Prelimi	admin	Preliminary	Update		
wRack.CATProduct	NewRack, , Assembly Model, , Prelimin	admin	Preliminary	Update		
wScrew.CATProduct	NewScrew, , Assembly Model, , Prelimi	admin	Preliminary	Update		
wCylinderBlock.CATProduct	NewCylinderBlock, , Assembly Model, ,	admin	Preliminary	Update		
wRack_Assm.CATProduct	NewRack_Assm, , Assembly Model, , P	admin	Preliminary	Update		
wSnapRingModel.CATPart	NewSnapRingModel, , Part Model, , Pr	admin	Preliminary	Update		
wPistonPincompModel.CATPart	NewPistonPincompModel, , Part Model,	admin	Preliminary	Update		
wSnapRing.CATProduct	NewSnapRing, , Assembly Model, , Prel	admin	Preliminary	Update		
wPistonPincomp.CATProduct	NewPistonPincomp, , Assembly Model,	admin	Preliminary	Update		
						Update Cl

Picture 84: Confirm the Update action

When the Update process has finished you are informed about the actions that have been performed (see *Picture 85: Information window when Update is finished*).

File name	PDM Display	Locked By	PLM State	Operation	Result	
NewCylinderBlockModel.CATPart	NewCylinderBlockModel, , Part Model, ,	admin	Preliminary	Update	SUCCESS	
NewRackModel.CATPart	NewRackModel, , Part Model, , Prelimin	admin	Preliminary	Update	SUCCESS	
VewScrewModel.CATPart	NewScrewModel, , Part Model, , Prelimi	admin	Preliminary	Update	SUCCESS	
NewRack.CATProduct	NewRack, , Assembly Model, , Prelimin	admin	Preliminary	Update	SUCCESS	
VewScrew.CATProduct	NewScrew, , Assembly Model, , Prelimi	admin	Preliminary	Update	SUCCESS	
vewCylinderBlock.CATProduct	NewCylinderBlock, , Assembly Model, ,		Preliminary	Update	SUCCESS	
NewRack_Assm.CATProduct	NewRack_Assm, , Assembly Model, , P	admin	Preliminary	Update	SUCCESS	
NewSnapRingModel.CATPart	NewSnapRingModel, , Part Model, , Pr	admin	Preliminary	Update	SUCCESS	
NewPistonPincompModel.CATPart	NewPistonPincompModel, , Part Model,	admin	Preliminary	Update	SUCCESS	
NewSnapRing.CATProduct	NewSnapRing, , Assembly Model, , Prel		Preliminary	Update	SUCCESS	
NewPistonPincomp.CATProduct	NewPistonPincomp, , Assembly Model,	admin	Preliminary	Update	SUCCESS	
Uploaded NewFixedParts_Assm.CATP Uploaded NewCylinder_Assm.CATProc						

Picture 85: Information window when Update is finished

Select Type of additional Parts in Document mode

If PDM Workbench is configured to work in the CAD document structure mode it is possible to create an additional part item in Aras Innovator while creating a CAD document during the PDM update. By default there is a configured part type that will be created in this case. The new functionality allows you to select a specific part type to be created:

20151026_1.CATProduct	
20151026_1	
Applications	Center graph <u>B</u> eframe On <u>B</u> Hide/Show <u>Centers</u> Alt+Enter
	Image: Sub-Tree Imag
	Delete Del 20151026_2.1 object +
	PDM Workbench
	Components Set PDM Type to Component (BOM) Representations Replace Node Std. Part (BOM) Selection Mode CATPart (Non-BOM) *

Picture 86: "Set PDM Type" context menu

When updating a product structure the type of the additional part will be shown in the "Operation" column of the update dialog. The "Create Parts at Update" button will be hidden.

ile name	CATIA Part Num	PDM Display	File Status	Operation	Result
0151026_2.CATPart 0151026_1.CATProduct	20151026_2 20151026_1		Modified Modified	Create 1. "/Part/Component" PDM Object will be created additionally Create 1. "/Part/Assembly" PDM Object will be created additionally: 2	

Picture 87: Update dialog for CATProduct structure

When updating a single CATPart the "Create Parts at Update" button will be shown. If a BOM type was selected for the CATPart, the button will be deactivated, otherwise the "Create new Parts at Update" button will be active. In this case the default part type will be created for the CATPart when setting the button to "YES"

date to PDM?						
File name	CATIA Part Number	PDM Display	File Status	Operation	Result	
20151026_2.CATPart	20151026_2		Modified	Create 1. "/Part/Component" PDM Object will be created additional		
eate new CAD document ge	neration? <u>No</u>	Create new part	s at update?	Yes		Update Cl

Picture 88: Update dialog for CATPart document

Constraints:

- It is only possible to select a BOM type if the parent Product is marked to have an additional BOM Part or if the parent CATProduct was loaded from PDM.
- If you switch the type of a CATProduct from BOM to NON-BOM all children will be switched to NON-BOM.

As there is no context menu for a single CATPart you have either to use the default part type for single loaded CATParts, or you have to put the CATPart in a temporary CATProduct and set the type of the additional PDM part item before updating the single CATPart.

Reconnect at Update

This functionality can be used for an initial import of existing CAD data into Aras.

It is possible to reconnect CATParts and CATProducts inside a structure loaded from disc to already existing CADDocuments in Aras even if the CATIA files in Aras were renamed (rule based) during the first import. When a CATPart / CATProduct is reconnected, the external file is not saved to Aras.

Before using this functionality it may be necessary to clean up the existing CATIA data in that way, that the files can be renamed during import using a rule. T-Systems can provide a tool that checks if all file names, part numbers, ... fit to the naming convention of your company.

Usage

1. Import Product Structure

Open a cleaned structure from disc and use PWB Update to save the structure into Aras. If some of the files may be already stored in Aras, select "Import with Reconnect" before performing the update:

File name	CATIA Part Number			Locked By	PLM State	File Status	Operation	
100315424	100315425 A01 TUBES			Modified	Create			
100311342	. 100311342 C03 HEADER 68					Modified	Create	
100311345	100311345 C08 SIDE PLATE 68						Create	
100315424	100315424 100315424 C01 CORE 68MA CAC+						Create : 1. Create PDM File Objec	
100310635 100310635 C08 TANK AL UPPER							Create	
100311826 100311826 C14 TANK AL LOWER						Modified	Create	
100312822 100312822 B42 PROTECTION PLUG						Modified	Create	
6099197 L 6099197 F01 LABEL WITHOUT PN						Modified	Create	
20758814 20758814 P05 CAC VOLVO TRUCK P2285						Modified	Create : 1. Create PDM File Obje	
Create new file	versions at update?	No	Import wi	th Reconnect?	Yes	Create ne	w parts at update? No	

Picture 89: Update dialog with "Import with Reconnect" button

During update the files are renamed according to the installed rule and reconnected if already found in Aras:



Picture 90: "Reconnect" prompt



Picture 91: Messages about reconnected items

To check if the reconnected Documents in Aras use the same content like the imported files on disk, the user has to close all windows in CATIA and open the structure from Aras.

2. Import CATDrawing

CATDrawings itself are not reconnected in Aras, but the referenced CATProducts / CATParts may be renamed during import. Therfore the following procedure can be used to import a CATDrawing without braking the links.

Open the CATDrawing and use File \rightarrow Desk to open the directly referenced CATPart(s) / CATProduct(s)

Desk	
100315424 C01 CORE ASSY 68MA CAS® 100315424 C01 CO	100315424 TUBES+ FINS CAC+.CATPart
	Properties Alt+Enter t
	🚰 Open
4	CATDUAVS
100315424 C01 CORE ASSY 68MA CAC+.CATDrawing	
F 100315424 C01 CO	Launch Document Reconciliation
- a Parameters	
Front view	

Picture 92: Opening referenced 3D geometry files

Update the opened structure like described in "Import Product Structure". Do not close the renamed Product structure after update.

After save / reconnect the structure in Aras the CATDrawing activates the "Update current Sheet" button that indicats that the CATDrawing need an update:



Picture 93: Updating the current sheet

Use the "Update current Sheet" command to make sure the drawing is clean.

Use the PWB Update function to save the Drawing in Aras. To save a CATDrawing in PDM do not select "Import with Reconnect".

Show PDM Structure

When you have opened the geometry in CATIA V5 you have the possibility to show the corresponding PDM product structure.

For this you have to click on the "PDM Structure" icon . Then a PDM Workbench window with the PDM product structure will be opened (see *Picture 94: PDM Structure for geometry*).



Picture 94: PDM Structure for geometry

Please note that with BOM-part structures, the expanded part structure (window in the middle) usually does not contain the related CATIA files, but the structure displayed by "Show PDM Structure" does (right window).

Refresh PDM Structure

When you have made some actions on the geometry that have an impact on the status of the objects in the PDM product structure, that are not displayed automatically, then you have to update the status display manually.

After the load of the geometry the status of the objects of the PDM product structure is in the default state.

There are the following possible states:

- No background color: not owned by session user, no changes
- Green: owned by session user, no changes
- Yellow: owned by session user, dirty because of changes
- Red: not owned by session user, dirty because of changes
- Black: the object is finalized in the PDM (for example: released state)
- White: the object does not have the file in the PDM

The CATIA Part is owned by the session user and no changes. It is marked in green (see *Picture 95: PDM Structure and geometry in CATIA V5*).



Picture 95: PDM Structure and geometry in CATIA V5

In the CATIA V5 you can make some changes in the geometry (see *Picture 96: Making changes in the geometry*) that make the objects in the PDM product structure dirty. This state change will not be displayed automatically.



Picture 96: Making changes in the geometry

In order to display the changes you have to click on the "Refresh PDM Structure" icon

. Then the status of the changed objects in the PDM product structure will be updated.

Now e.g. the dirty object owned by the session user will be displayed in yellow in the PDM product structure. The dirty objects on the way to the root are displayed in red because they are not owned by the session user (see *Picture 97: Refreshed PDM Structure*).



Picture 97: Refreshed PDM Structure

Force Load CATPart

When you have loaded the CATIA node into the structure via the Desk command because the file could not be loaded then you can load the CATIA data with the required PDM Workbench information. For this you have to select the CATIA node and open the context menu. Then choose "PDM Workbench→Force Load CATPart" (see *Picture 98: Action* "Force Load CATPart").



Picture 98: Action "Force Load CATPart"

Highlight PDM Nodes

Sometimes it is important to find a PDM node when you are working on the corresponding object in the geometry, e.g. in order to lock the PDM object. Or you have selected an object in the PDM product structure and want to see the corresponding object in the geometry.

For this you can select an object in the geometry and click on the right mouse button. The context menu opens and you have to choose "PDM Workbench→Highlight PDM Nodes" (see *Picture 99: Action "Highlight PDM Nodes"*).



Picture 99: Action "Highlight PDM Nodes"

The PDM Workbench window will be displayed in the foreground and the objects that correspond to the selection in the geometry will be highlighted (marked) (see *Picture 100: Highlighted nodes in PDM Structure*). If there is no PDM Workbench window opened you will get a warning that you have to open the PDM structure first. It is important that you have only one PDM Structure window for this Part Number.



Picture 100: Highlighted nodes in PDM Structure

This works in the opposite direction, too (see Highlight CATIA Nodes).

Highlight CATIA Nodes

Sometimes it is important to find a CATIA node when you are working on the corresponding object in the product structure, e.g. in order to lock the PDM object. Or you have selected an object in the geometry and want to see the corresponding object in the PDM product structure.

For this you can select an object in the PDM product structure and click on the right mouse button. The context menu opens and you have to choose "PDM Workbench→Highlight CATIA Nodes" (see *Picture 101: Action "Highlight CATIA Nodes*").



Picture 101: Action "Highlight CATIA Nodes"

The CATIA V5 geometry window will be displayed in the foreground and the objects that correspond to the selection in the PDM product structure will be highlighted (marked) (see *Picture 102: Highlighted nodes in CATIA geometry*). If there is no CATIA V5 geometry window opened you will get a warning that you have to open the CATIA V5 geometry window.



Picture 102: Highlighted nodes in CATIA geometry

This works in the opposite direction, too (see *Highlight PDM Nodes*).

Open in New Window

When you have a lot of objects in one window, e.g. received by query or expand then it can be necessary to open a subset of them in a new window.

You select the objects you want to open in a new window. In the context menu you select the action "Open in New Window" (see *Picture 103: Action "Open in New Window"*).

🛐 Engine, B, 2, Engine, Prelii	mina	ry (Logged in as admin)									
Engine, B, 2, Engine, Preliminary											
Uses Part BOM (Part BOM, A, 1) / FixedParts, A, 1, FixedParts, Preliminary											
🗧 🗧 🔅 Uses Part BOM (Pa		C <u>e</u> nter graph									
Uses Part BOM		<u>R</u> eframe On									
🖨 🏹 Uses Part BOM (Pa	8	Hide/Show									
🗧 🏹 Uses Part BOM	1	Properties	Alt+Enter								
L 😳 Uses Part B		Open Sub-Tree									
🖨 💭 Uses Part BOM	dia.	Cut	Ctrl+X								
Uses Part B	ĥ	Сору	Ctrl+C	Y							
Uses Part BOM (Part I		Paste	Ctrl+V								
Uses Part BOM (Pa		Paste Special		1							
Uses Part BOM		Paste guetian		У							
Uses Part B		<u>D</u> elete	Del	odel, Prelimina							
Uses Part BOM		FixedParts, A, 1, FixedParts, P									
Uses Part B		Executaics, A, I, Hitedraics, P	eliminary								
Uses Part BOM (Pa		Expand	•								
Uses Part BOM		Expand Multiple Levels	•	7							
Uses Part BOM (Part I		De_Expand		Ĺ							
Uses Part BOM (Pa	* <mark>8</mark>	Open in new PDM Window		l, Preliminary							

Picture 103: Action "Open in New Window"

The selected objects will be opened in a new window (see *Picture 104: The selected objects in the new window*).



Picture 104: The selected objects in the new window

Attribute mapping functionality

CATIA standard and user-defined properties can be mapped to PDM attributes.

In the following example the standard CATIA attributes "Nomenclature" and "Description" are mapped to the attributes "name" and "description" of the Aras Innovator part object (see *Picture 105: Standard attributes in the "Properties" dialog* and *Picture 106: Configuration of standard attributes in Aras Innovator*).
Properties	? ×
Current selection	n: NewProduct1
Product G	raphic Mechanical Drafting
	· · · · ·
Product	
Part Number	NewProduct1
Revision	
Definition	
Nomenclature	NewProduct1 - Nomenclature
Description	NewProduct1 - Description
Define other	properties
a de la Marriera	
•	>
	More
	OK Apply Close

Picture 105: Standard attributes in the "Properties" dialog

PWB Configuration - 77F7BC4E8	322E43E4A3DE6327	1A2AE279 - Windows In	iternet Explorer		
File Edit Views Search Actions Reports Tools Help					
🕂 🗎 🗙 🖸 🖶 🖾		ち ご ? 、	/		
PWB	Name				
Configuration	Configuration				
	Beschreibung				
			×		
Settings Standard Attribute	es User Attribut	es Drawing Attribu	tes		
Actions	+ 🖻 🗙 🛛	I 🛯 🔒 💼 🕻 🐔	A A A		
Cad Name	Innovator Name	Mapping Direction	ItemType		
CadNomenclature	name	Cad < - > Innovator	Part		
CadReferenceDescription	description	Cad < - > Innovator	Part		

Picture 106: Configuration of standard attributes in Aras Innovator

After creating the part with Update the defined CATIA attribute values have been written to the PDM part object (see *Picture 107: Standard attributes in the "Properties" dialog of the PDM node* and *Picture 108: Standard attributes in Aras Innovator window*).

Properties Up	date Item	
-Assembly		
Part Number	NewProduct1	
Major Rev.	A	
Generation	1	
	NewProduct1 - Nomenclature	
	Preliminary	
Unit		
Make / Buy	Make	
Cost		
Description	NewProduct1 - Description	
s de manana		

Picture 107: Standard attributes in the "Properties" dialog of the PDM node

Part - NewProduct1 - Windows I	Internet Explorer	
File Edit Views Search	Actions Reports Tools Help	
	💵 🔒 💼 🍤 🕼 🛅 🥐	
Part	Part Number Revision State	Assigned Creator
	NewProduct1 A Preliminary	Innovator Admin
-0-	Name	Designated User
**	NewProduct1 - Nomenclature	
Created By: Innovator Admin	Type Unit Make / Buy Cost	Effective Date
Created On: 9/23/2015	Assembly EA Make	
Modified By: Innovator Admin Modified On: 9/23/2015	Long Description	Select an image
Locked By: Innovator Admin	NewProduct1 - Descritpion	
Major Rev: A Release Date:		
Release Date: Effective Date:	V	
Generation: 1		
State: Preliminary		
BOM BOM Structure Alte	mates AML Documents CAD Documents Goals	Changes Part Submiss -
Actions Pick Related	i 🗛 🗶 🖾 🖬 🔒 💼 🖉 🗛 😣 🛛	Hide Search Criteria 💌
Sequence Part Numb	er Revision Name	Type Quantity S
Ready	Items 1-2 of 2. Page 1 of 1	Aras Inn

Picture 108: Standard attributes in Aras Innovator window

User-defined CATIA properties can also be mapped (see *Picture 109: Configuration of user-defined attributes in Aras Innovator*).

EPWB Configuration - 77F7BC4E822E43E4A3DE63271A2AE279 - Windows Internet Explorer				
File Edit Views Searc	h Actions Rep	orts Tools Help		
- B				
PWB	Name			
Configuration	Configuration			
comguration	Beschreibung			
			-	
Settings Standard Attribu	tes User Attribute	es Drawing Attribute	S	_ 🗆
Actions	🕂 🎋 🗙 🕸	🛯 🔒 💼 🕻	A A A	Hide Search Criteria 💌 💌
Cad Label	Innovator Name	Mapping Direction	ItemType	
+ CAD Name nam	ie	Cad < - > Innovator	CAD Document	
CAD Description desc	cription	Cad < - > Innovator	CAD Document	•
Ready	Item	s 1-2 of 2. Page 1 of 1		Aras Innovator

Picture 109: Configuration of user-defined attributes in Aras Innovator

While the structure is imported the values are written to the defined attributes of the Aras Innovator CAD document object (see *Picture 110: User-defined attributes in the "Properties" dialog of the PDM node* and *Picture 111: User-defined attributes in Aras Innovator window*).

	on : NewProduct2, A, 1, NewProduct2, Preliminary, Mechanica	•
Properties	Update Item	
CATProduc		^
	nber NewProduct2	
-	Rev. A	
Genera		
	ame NewProduct2	
	itate Preliminary	
Descrip	tion own description	_
		E
NI 17		
New Ver		
	Mechanical/Assembly	
	ator Innovator Admin	
Designated		μ
	d by Innovator Admin	
	d on 2015-10-21T11:59:39	
Create		
Creater Modifie	d by Innovator Admin	
Creater Modifie Modifie	d on 2015-10-23T12:42:36	
Creater Modifie Modifie	d on 2015-10-23T12:42:36 d by Innovator Admin	

Picture 110: User-defined attributes in the "Properties" dialog of the PDM node

CAD Docume	nt - NewProduct2 -	Windows Internet E	xplorer	
File Edit	Views Search	Actions Repor	rts Tools Help	
🕂 🗎 🔍	∽ 🖶 🔯	🕼 🔒 🕤 🏷	ℰ [*] 🗸 🗸	
CAD Docur	ment	Document Number	Revision State	Select an image
		Name NewProduct2		
Created By: Created On:	Innovator Admin 10/21/2015	Type Assembly	Authoring Tool	Version V5R24
	Innovator Admin	Description		VJICT
Modified On: Locked Bv:	10/23/2015 Innovator Admin	own description		×
Major Rev:	A			-
Release Date:		Assigned Creator	Designated User	Native File
Effective Date: Generation:	1	Innovator Admin	•••	···· NewProduct2.C
State:	Preliminary	Part	From Template	Viewable File
				1. Select and uploa
Structure F	Parents Files	Changes CAD E	DesignTable	
Actions • Pic	ck Related	- 🔊 🗙 🖾	🛯 🔒 💼 🕼 🔗	Hide Search Criteria 🔽
Sequence	Document Numb	er Revisi	Name	Type State
•				

Picture 111: User-defined attributes in Aras Innovator window

After the import or after loading the structure it can be shown that the values are written from the PDM attributes into the CATIA files (see *Picture 112: User-defined attributes in the "Properties" dialog*).

Pro	operties			2	x	
0	Current selection : NewProduct2					
1	Product Graphic Mechanical Drafting					
	- Product					
	Part Number	NewProduct2				
	Revision	,				
	Definition					
	Nomenclature	2				
	Source	Unknown	•			
	Description	ded Descention				
		ded Properties	N D I 12			
	CAD Name		NewProduct2			
	CAD Descript	tion	own description			
	<u>، ، ، ، ، ، ، ، ، ، ، ، ، ، ، ، ، ، ، </u>				F I	
	More					
100	100		OK Apply		USE	

Picture 112: User-defined attributes in the "Properties" dialog.

Because of the mapping direction of the properties it is also possible to change the values in the Catia properties. After the update of the structure the values are written from the CATIA files into the PDM attributes.

Internal CATIA information can be written to user-defined CATIA properties

If they are defined this way then the corresponding current values are written into the user-defined CATIA properties (see *Picture 113: User-defined attributes with internal CATIA information in the "Properties" dialog*).

roperties		?
Current celection	n : NewProduct1	
Product G	iraphic Mechanic	al Drafting
Product		
Part Number	NewProduct1	
Revision		
Definition		
5 (3) (3) (3) (3) (3) (3) (3) (3) (3) (3)	NewProduct1 - Nor	henclature
Source	Unknown	
Description	NewProduct1 - Des	cription
Draductu Ada	ded Properties	
CAD Documer		
-		NewProduct1
CATIA Enviro		V5R20
The Mass		0,576847
The Volume		0,000576847
The Density		1000
The Area		0,0900361
Define other	properties	
•		
		More
		OK Apply Close

Picture 113: User-defined attributes with internal CATIA information in the "Properties" dialog

If these values are configured correspondingly, like for instance "CATIA Environment" in the picture below, then the values are automatically written to the mapped attributes of the PDM object (see *Picture 114: Configuration of user-defined attributes in Aras Innovator*).

PWB Configuration - 77F7BC	C4E822E43E4A3DE6327	1A2AE279 - Windows Inte	ernet Explorer	
File Edit Views Sea	arch Actions Rep	orts Tools Help		
🕂 🗎 🔍 🖸 🖶 I	🛛 🗐 🔒 💕	ך ויי יין או	,	
PWB	Name			
Configuration	Configuration Beschreibung			
			×	
Settings Standard Attrib	butes User Attribute	es Drawing Attribute	S	_ 🗆
Actions Create Related	🕂 🚯 🗙 🚳		A A A	Hide Search Criteria 💌 💌
Cad Label	Innovator Name	Mapping Direction	ItemType	
CATIA Environment pv	wb_catia_environment	Cad -> Innovator	CAD Document	
+ CAD Document Name pv	wb_name	Cad < - > Innovator	CAD Document	
Ready		0 Items found.		Aras Innovator

Picture 114: Configuration of user-defined attributes in Aras Innovator

Template-based CAD document and part creation

In the dialog which appears after the user clicks on the toolbar action "Create" the type of the new object can be selected (see *Picture 115: Select a PDM type for the "Create" dialog*).

s	Select a PDM type f	or perfo	ormi	? ×
	PDM Object Types			
	Assembly Component Std. Part			
	CATPart			
	CATProduct CATDrawing			
	<u> </u>	OK	🥥 Ca	ancel

Picture 115: Select a PDM type for the "Create" dialog

If one of those types is selected then the next dialog opens. You have to select the template for the new object from a dropdown list. The template names are defined in the PDM Workbench configuration file. If you do not select a template an empty CATIA file is opened (see *Picture 116: "Create" dialog for CATPart – Select Template*).

PDM Create		<u>?</u> ×
Create dialog for ty	pe CATPart:	
CATPart Template		•
_	TemplateCatPart1.CATPart TemplateCatPart2.CATPart	

Picture 116: "Create" dialog for CATPart – Select Template

When you confirm the dialog with "OK" the template file or the empty file is opened in CATIA and the "Update to PDM?" dialog is opened. You have to type in the attribute values of the item object and the document object to be created (see *Picture 117: "Create" dialog for CATPart in BOM Part Structure*).

New.CATPart		
Update to PDM?	PDM Create	
File name PDM Display	Create dialog for type Component:	Operation Result
New.CATPart	Part Number *	Create
	Name Unit	
	Make / Buy	
	Cost	
	Description	
	Create dialog for type CATPart:	
	Doc. Number *	
	Name	
	Description	
]		
Create new file versions at update? No	1	
	OK Apply Cancel	Update. Close
		x.Ľ×

Picture 117: "Create" dialog for CATPart in BOM Part Structure

I	Jpdate to PDM?					1		
	File name New.CATPart	CATIA Part Number New	PDM Display	Locked By	PLM State	File Status Modified	Operation Create : 1. Create PDM	Result File Object of typ
			PDM Create Create dialog for ty New Doc. Number New Name Description Is Template _ Standard Part _		ancel			
	Create new CAD document gene	eration? No	Create new parts at update?	No				Update Close

Picture 118: "Create" dialog for CATPart in Document Structure

Then you have to change the name and click on the "OK" button. A normal update is performed, which creates a part with the corresponding CAD document in the PDM system and uploads the file (see *Picture 119: Created Part*).

NewPart1.CATPart	Properties	<u>? ×</u>	NewPart1, A, 1, , Preliminary (Logged in as admin)
NewPart1	Current selection : NewPart1	7	NewPart1, A, 1, , Preliminary
- xy plane	Product Graphic Mechanical Mass Color Management		Attaches (Part CAD) / NewPart1, A, 1, , Prelminary, Mechanical/Part
– 🖉 yz plane			
– zx plane			
🕂 🔓 Parameters			
🕂 🏇 PartBody			
	Product		
	Part Number NewPart1		
	Revision 1		
	Definition		
	Nomenclature NewPart1		
	Source Unknown		
	Description		
	[Define other properties]		
		Þ	
		tore	
	OK Apply	Close	

Picture 119: Created Part

Using standard parts

When CATParts which are defined as corresponding to a standard part are added to a CATProduct structure the update process will not try to create the corresponding part and CAD document objects in PDM. Instead the standard part object which has the same part number as the CATPart's CATIA part number will be queried and added to the part structure instead. Using this functionality standard part geometry can be added to a structure without having to first load the standard parts into the CATIA session.

CATIA documents are set to read-only if corresponding PDM node is not modifiable

tate v	Name	Location	Action Ac	cess	Save
pen Read Only	Piston.CATProduct	C:\PWB_XMAP		ead Only	Save As
oen Read Only	Engine.CATProduct	C:\PWB_XMAP		ead Only	Propagate director
pen Read Only	ScrewModel.CATPart	C:\PWB_XMAP		ead Only	
pen Read Only	Crankshaft_Assm.CATProduct	C:\PWB_XMAP		ead Only	Reset
pen Read Only	ConnectionRod.CATProduct	C:\PWB_XMAP		ead Only	
pen Read Only	Cylinder_Assm.CATProduct	C:\PWB_XMAP		ead Only	
pen Read Only	FixedParts_Assm.CATProduct	C:\PWB_XMAP		ead Only	
pen Read Only	CrankshaftLeft.CATProduct	C:\PWB_XMAP		ead Only	
pen Read Only	CrankPin.CATProduct	C:\PWB_XMAP		ead Only	
pen Read Only	CrankshaftRight.CATProduct	C:\PWB_XMAP		ead Only	
pen Read Only	RackModel.CATPart	C:\PWB_XMAP		ead Only	
pen Read Only	PistonPin_Assm.CATProduct	C:\PWB_XMAP		ead Only	્યાદ્રસ્ય
pen Read Only	Rack_Assm.CATProduct	C:\PWB_XMAP		ead Only	÷.
pen Read Only	CylinderBlock.CATProduct	C:\PWB_XMAP		ead Only	
pen Read Only	SnapRing.CATProduct	C:\PWB_XMAP		ead Only	
pen Read Only	PistonPincomp. CATProduct	C:\PWB_XMAP		ead Only	
pen Read Only	Screw.CATProduct	C:\PWB_Path: Path of the document		ead Only	
pen Read Only	Rack.CATProduct	C:\pwp_state: State of the document		ead Only	
pen Read Only	CylinderBlockModel.CATPart	C:\PWB_XMAP		ead Only	
pen	11.PWBDoc	C:\PWB_XMAP		ead Write	
pen	12.PWBDoc	C:\PWB_XMAP	Re	sad Write	
tern Name: *	Apply Pattern				
nsaved File(s) L	eft	Enable independent saves			

Loading a structure sets the corresponding CATIA files to read only (see *Picture 120: Save Management*).

Picture 120: Save Management

Added check whether CATIA structure is valid before update

Before the structure will be created in the PDM system it will be checked if it is valid (see *Picture 121: Check if CATIA structure is valid*).



Picture 121: Check if CATIA structure is valid

Thumbnails

In the properties dialog for part and drawing documents a thumbnail will be shown (see *Picture 122: CAD Document properties in Aras Innovator*).

CAD Docume	nt - Part1 (read on	ly) - Windows Inte	rnet Explorer		_ 0	×
File Edit	Views Search	Actions Rep	orts Tools Help			
🕂 🗎	😏 🖶 🖾	🗐 🔒 🔐 🖞	っ 🖍 📩 ? 🗸			
CAD Docui	ment	Document Numbe ScrewModel Name	r Revision State	0		
Created By: Created On: Modified By: Modified On: Locked By:	Innovator Admin 9/24/2015 Innovator Admin 10/7/2015	Type Part Description		rersion /SR24		
Major Rev: Release Date: Effective Date: Generation:	A 1	Assigned Creator	Designated User	Native File	.CATPart	
State:	Preliminary	Part	From Template	Viewable File		ŀ
Structure F	Parents Files	Changes CAE) DesignTable		_	_
Actions • Pic	ck Related	2 IA 46 I	I 💵 🔒 🖆 🦨 🗸	😣 🔊 Hide Search C	Driteria 💌	Ŧ
Sequence	Document Num	ber Revisi	Name	Туре	State	N
Ready			0 Items found.		Aras Innova	to

Picture 122: CAD Document properties in Aras Innovator

Basic Drawing Link Support

When a CATDrawing CAD document is created the user can decide which of the related 3D geometry documents is defined to be the main source geometry document. A link of the type "/CAD Structure/Drawing" is created from the CATDrawing to that 3D document when the drawing document is created.

If the source 3D geometry documents of the drawing are loaded from PDM and the PDM Workbench session contains information about the corresponding CAD documents in PDM then the linked 3D geometry document will be selected in the "Main 3D File" combo box if a single 3D document is linked (see *Picture 123: Creating a CATDrawing document with a link to 3D geometry*).

Engine.CATProduct	Drawing1
Finance FixedParts (FixedParts.1) PistonAssm (PistonAssm.1) PistonAssm (PistonAssm (PistonAssm.1) PistonAssm (PistonAssm (PistonAssm.1)) PistonAssm (PistonAssm	Sheet1 Sh
Creation of a new object of type 'CATDrawing': Doc. Number Name Description Main 3D File Engine.CATProduct	

Picture 123: Creating a CATDrawing document with a link to 3D geometry

When the user creates the drawing CAD document a PDM relation of the type "/CAD Structure/Drawing" will be created in the PDM system after the CATDrawing PDM document has been created (see *Picture 124: PDM message about created drawing link*).

Created relation '/CAD Structure/Drawing' between 'Engine-Drw1' and 'Engine'
PDM Messages:

You can expand to this document by selecting the CATDrawing object. Then click the right mouse button and select "Expand \rightarrow Is derived from Model" (see *Picture 125: Expanding newly created drawing link*).

	, 1, Engine drawing 1, Preliminary, Mechanical/Drawing, (Logged in as I	Jsr1)	
Engine-D	1, A. 1, Engine drawing 1, Preliminary, Mechanical/Drawing,		
	C <u>e</u> nter graph		
	<u>R</u> eframe On		
	Hide/Show		
	Properties	Alt+Enter	
	Ope <u>n</u> Sub-Tree		
	🖁 Cu <u>t</u>	Ctrl+X	
	⊆ору	Ctrl+C	
	Paste	Ctrl+V	
	Delete	Del	-
	Engine-Drw1, A, 1, Engine drawing 1, Preliminary, Mechanical/Drawi	ng, object 🕨 🕨	
	Expand	۱.	Is derived from Model
	De_Expand		Is attached by
	Open in new PDM Window		is attached by



The document will be displayed in the window (see *Picture 126: Displaying newly created drawing link*).



Picture 126: Displaying newly created drawing link

If the user creates a drawing with links to more than one 3D geometry file then both linked documents will be displayed in the window (see *Picture 127:*).

NewDrw_PWBBOM, A, 1, , Preliminary, Mechanical/Drawing (Logged in as admin):2	
NewDrw_PWBBOM, A. 1, , Preliminary, Mechanical/Drawing	
s derived from Model (/CAD Structure/Drawing) / Product_PWBBOM, A, 1, , Preliminary, Mechanical/Assembly	
🖵 🔊 is derived from Model (/CAD Structure/Drawing) / Product3BOMTest, A. 1, , Preliminary, Mechanical/Assembly	

Picture 127: Displaying all created drawing links.

Drawing links are not updated or deleted when a CATDrawing is updated, even if links are created or removed in CATIA. The related primary 3D document is not supposed to change during the lifetime of the CATDrawing.

Basic Multi-Model Link Support

When a CATPart CAD document is created or updated the geometry links of imported 3D geometry will be updated as PDM links of the type "/CAD Structure/Reference". Both reference links and instance links are supported.

If the functionality is switched on, when a CATPart contains geometry links, PDM relations of the type "/CAD Structure/Reference" which correspond to these links are created (see *Picture 128: Information when reference links are created*).



Picture 128: Information when reference links are created

If the links are removed from the CATIA file then the corresponding PDM relations are deleted (see *Picture 129: Information when reference links are deleted*).

Uploaded LinkTargetPart1.CATPart
PDM Messages:
Deleted relation '/CAD Structure/Reference' between 'LinkTargetPart1' and 'LinkSourcePart2'

Picture 129: Information when reference links are deleted

The created links can be expanded. Select the document and click the right mouse button. Select "Expand \rightarrow Uses Geometry from" (see *Picture 130: Expanding geometry links*).

🛐 Link	TargetPart1, A, 1, , Preliminary, Mechanical/Part, (Logged ir	n as Usr1)	
🔊 Lin	nkTargetPart1, A, 1, , Preliminary, Mechanical/Part,		_
	C <u>e</u> nter graph		
	<u>R</u> eframe On		
8	Bide/Show		
6	Pr <u>o</u> perties	Alt+Enter	
	Ope <u>n</u> Sub-Tree		
6	Cu <u>t</u>	Ctrl+X	
9	🗎 Сору	Ctrl+C	
Ľ.	Paste	Ctrl+V	
	Paste <u>S</u> pecial		
	Delete	Del	
	LinkTargetPart1, A, 1, , Preliminary, Mechanical/Part, ol	oject 🕨	
	E <u>x</u> pand	•	<u>U</u> ses Geometry from
	Exp <u>a</u> nd Multiple Levels	•	Is attached by
1	Ra De <u>-</u> Expand		
1	Open in new PDM Window		
ſ			Geometry is used in
	Open Sub-Tree Cut Copy Paste Paste: Special Delete LinkTargetPart1, A, 1, , Preliminary, Mechanical/Part, ol Expand Expand De_Expand Open in new PDM Window	Ctrl+X Ctrl+C Ctrl+V Del bject •	<u>I</u> s attached by U <u>s</u> ed in Structure <u>D</u> erives Drawing

Picture 130: Expanding geometry links

The document will be displayed in the window (see *Picture 131: Geometry link expansion result*).



Picture 131: Geometry link expansion result

Management of CATIA templates in Innovator

The existing template file functionality, where template CATIA files are stored on a local directory which is accessible from CATIA V5, is extended such that the file templates can be stored in PDM.

The template file functionality can be used in two ways:

1. When new CATIA document objects are created from the 'Create' command in the PDM Workbench toolbar.

Example:

Create CATPart (see Picture 132: Template file functionality: Creating a CATPart).

Sel	ect a PDM type for performing a Create	
	PDM Object Types Assembly Component Std. Part CATPart CATProduct CATDrawing	
	OK Scancel	

Picture 132: Template file functionality: Creating a CATPart

If templates are configured then the user gets to choose a template file name (see *Picture* 133: *Template file functionality: Selecting a template file*).

PDM Create	2	×
Create dialog for ty	pe CATPart:	
CATPart Template		•
	TemplateCatPart1.CATPart TemplateCatPart2.CATPart	

Picture 133: Template file functionality: Selecting a template file

If no template file is chosen a new CATIA CATPart file will be created.

2. When a part structure is loaded where one or more parts do not have a related CATIA file.

This use case is applicable when using BOM part structures ("UseBomPartStructure" is set to "true").

Example:

Create an Assembly (see Picture 134: Template file functionality: Creating an assembly).

Select a PDM type for performing a Create
PDM Object Types
Assembly Component Std. Part CATPart CATProduct CATDrawing
OK Cancel

Picture 134: Template file functionality: Creating an assembly

Fill all the necessary Assembly information on the Create dialog and click OK.

The created Assembly is opened in a new PDM window.

Right click on the Assembly node and chose "Load" from the context menu.

In this case the corresponding CATIA structure nodes are created on the fly using the first template file in the list, which is defined as the default template.

Insert from PDM

The CATIA file which corresponds to an existing Innovator item can be inserted directly into the CATIA structure.

The user right-clicks on an existing CATProduct node in a CATIA structure window and selects "Insert PDM Node" (see *Picture 135: "Insert PDM Node" context menu*).



Picture 135: "Insert PDM Node" context menu

A restricted query window opens. In CAD structure mode CATPart and CATProduct items can be queried, in BOM structure mode Part items can be selected (see *Picture 136: "Insert PDM Node" query dialog type selection*).

	PDM Query					
	PDM object ty	pe		Doc. Number	Major Rev.	Generation
	CATPart		-			
	CATPart					
I	CATProduct					
I	Doc. Number	-				
l	Major Rev.	•				
I	Generation					
l	Name					
I	State					
I	Description					
I						
	Created on					
	Modified on					
	Created by					
	Modified by					
	Locked by					
1		,				

Picture 136: "Insert PDM Node" query dialog type selection

After the query is performed all resulting items are displayed in the result list, like in the regular query dialog (see *Picture 137: "Insert PDM Node" query result*).

PDM Query				
PDM object type		Doc. Number	Major Rev.	Generation
CATPart	-	Screw	A	1
PDM object attributes				
Doc. Number Screw*	-			
Major Rev.	•			
Generation	•			

Picture 137: "Insert PDM Node" query result

Double-clicking on one of the result items in the list causes its corresponding file to be downloaded and inserted into the selected CATProduct node in the CATIA structure:



Picture 138: Item inserted in existing structure

It is possible to query for items which are already contained in the CATIA structure, as well as for items which do not exist in the structure yet.

The newly inserted CATIA node is not updated to PDM yet, the next Update process will create the corresponding structure relation.

Replace from PDM

The node selected in the CATIA structure can be replaced by the CATIA file which corresponds to an existing Innovator item.

Warning: This functionality can create broken links in the CATIA structure.

In order for CATIA links to be preserved, the link conditions of the replaced geometry and the replacing geometry has to be compatible.

If the new CATIA document is not compatible with the link conditions of the product structure the CATIA V5 Replace functionality presents a warning dialog.

This is the "Impacts on Replace" dialog as presented by the standard CATIA V5 Replace functionality (see *Picture 139: The "Impacts on Replace" standard CATIA dialog*).



Picture 139: The "Impacts on Replace" standard CATIA dialog

This "Impacts on Replace" dialog cannot be implemented by T-Systems due to lack of sufficient APIs.

If the inserted CATIA document is not compatible linkage information in the product structure gets lost. In this case some constraints are broken (see *Picture 140: Constraints destroyed by "Replace" operation*).



Picture 140: Constraints destroyed by "Replace" operation

This possibility of breaking CATIA links needs to be taken into account when this functionality is used. If in doubt please load the geometry which is supposed to replace existing geometry in the structure with "Load" and use the regular CATIA replace operation.

The functionality is used as follows:

The user right-clicks on an existing CATProduct or CATPart node in a CATIA structure window and selects "Replace Node" (see *Picture 141: "Replace Node" context menu*).

PD <u>M</u> Workbench	PDM Properties	
Components Represent <u>a</u> tions	Highlight PDM Nodes	
Selection Mode	Lock	
	<mark>የም <u>R</u>eplace Node</mark>	

Picture 141: "Replace Node" context menu

As in the "Insert PDM Node" case, a restricted query window opens. The user can query for existing items and double-click on one of the found items to select the one which should replace the selected CATIA structure node.

The user then gets to decide whether only the selected node or all instances of the document should be replaced (see *Picture 142: "Replace all instances" prompt*).



Picture 142: "Replace all instances" prompt

After that the selected instance or all of the instances of the selected CATIA node will be replaced by the CATIA file related to the queried and selected PDM item.

It is not possible to replace CATIA documents in a CATIA session when the new file has the same filename as the file to be replaced, because both files would be located in the same directory (PWB exchange map).

Replacing CATIA nodes does not change the instance names of the replaced nodes. Only nodes in the structure in the active window are affected by the replace operation.

CATDrawing attribute mapping

PDM attribute values of the drawing CAD document item can be mapped to CATDrawing attributes (see *Picture 143: CATDrawing attribute mapping*).

Engine-Drw1.CATDrawing	
Engine-Drw1 Engine-Drw1 Parameters Parameters Parameter	Sheet.1 Engine-Drw1 Name Engine-Drw1 A Properties Current selection : Engine-Drw1, A, 1, Engine-Drw1 Nam Properties Update Item
	CATDrawing
Ⅰ →	Doc. Number Engine-Drw1
	Major Rev. A
CATDrawing, CATProduc	Generation 1
	Name Engine-Drw1 Name
Engine-Drw1, A, 1,	State Preliminary

Picture 143: CATDrawing attribute mapping

After the drawing is created in PDM the CATDrawing file contains the attributes defined in the configuration attribute "CatiaDrawingCadFileAttributes", which in turn contain the values of the PDM attributes on the CATDrawing document defined in "CatiaDrawingPdmDocAttributes" (see *Picture 144: Derived CATDrawing containing attributes*).



Picture 144: Derived CATDrawing containing attributes

These drawing attributes' values can be assigned to text fields in the 2D drawing (see *Picture 145: Drawing attributes' values assigned to text fields*).





Edit Parameter	Engine-Drw1 New Name
Engine-Drw1.CATDrawing Fingine-Drw1 Fingine-Drw1 Fingine-Drw1 Fingine-Drw1 Fingine-Drw4tr1 Fingine-DrwAttr2 Fingine-DrwAttr3 Fingine-DrwAttr3 Fingine-DrwAttr3 Fingine-DrwAttr3 Fingine-DrwAttr3 Fingine-Drw4tr3 Fingi	Sheet.1 Engine-Drw1 New Name Engine-Drw1 A
	v1 Name, Preliminary, Mechanical/Drawing, (Logged in as ne-Drw1 Name, <mark>Preliminary, Mechanical/Drawing,</mark>

The user can modify one of the drawing attributes (see *Picture 146: Modified drawing attribute value*).

Picture 146: Modified drawing attribute value

After a PDM update, the new attribute value is written into its corresponding PDM attribute (see *Picture 147: Modified PDM attribute value*).

Engine-Drw1, A, 1, I	ngine-Drw1 New Name, Preliminary,	Mechanical/Drawing, (Logg	jed in a
Engine-Drw1, A	, 1, Engine-Drw1 <mark>New Name</mark> , Pre	eliminary, Mechanical/Dra	wing,
Properties		?	x
Current selection :	Engine-Drw1, A, 1, Engine-Drw1 New	Name, Preliminary, Mechan	
Properties Up	date Item		
CATDrawing —			-
Doc. Number	Engine-Drw1		
Major Rev.	A		
Generation	1		Ξ
Name	Engine-Drw1 New Name		
State	Preliminary		200

Picture 147: Modified PDM attribute value

If, on the other hand, the user changes one of the mapped PDM attributes, the new value will be written into the CATDrawing attribute the next time the CATDrawing is loaded to CATIA (see *Picture 148: PDM attribute value modified from Innovator*).

#2 N	1, Engine-Drw1 New Name, Preliminary, Mechanical/Drawing, (Logged in , A, 1, Engine-Drw1 New Name, Preliminary, Mechanical/Drawing
	Engine-Drw1, A, 1, Engine-Drw1 New Name, Preliminary, Mechani
Doc. Number Major Rev. Generation Name Description	A
Туре	Mechanical/Drawing
	More OK Apply Close

Picture 148: PDM attribute value modified from Innovator

You can see the PDM attributes in the parameters of the CATDrawing (see *Picture 149: Drawing attribute value changed to PDM attribute value*).

Engine-Drw1.CATDrawing	
Engine-Drw1 Parameters DrwAttr1 DrwAttr2 DrwAttr3 Front view Bottom view Right view	Sheet.1 Engine - Drw1 Name from PDM Engine - Drw1 A dit Parameter DrwAttr2 ne-Drw2 Name from PDM OK Cancel
	v1 Name from PDM, Preliminary, Mechanical/Drawing, (Logged in as
Engine-Drw1, A, 1, Engin	e-Drw1 Name from PDM, Preliminary, Mechanical/Drawing,

Picture 149: Drawing attribute value changed to PDM attribute value

Create new version

A new generation of a CAD document can be created by clicking on the "Create new version" context menu in the PDM window (see *Picture 150: "Create new version" PDM context menu*).



Picture 150: "Create new version" PDM context menu

Configurable CATIA components support

It is possible to load and update CATProduct structures which contain embedded CATIA components. Depending on the part number prefix the embedded component nodes can either be "skipped", that is, the node is treated as if it does not exist, but its child nodes are processed, or they can be "ignored", that is, the node and all its child nodes are treated as if they do not exist.

In the following example the two instance nodes of the CATPart "NewPart3" are treated as if they were both directly under the CATProduct "NewProduct3", and the two instance nodes of the CATPart "NewPart4" are completely ignored, that is, the structure is treated as if they do not exist (see *Picture 151: Embedded CATIA component nodes*).



Picture 151: Embedded CATIA component nodes

Support Electrical / Tubing

With this functionality it is possible to use functions like "Electrical Harness", "Electrical Wire Routing", "Piping Design", "Tubing Design", ... of the CATIA "Equipment & Systems Engineering" section.

The functions of the "Equipment & Systems Engineering" section often create embedded leaf components (without files) of special types like "ElecWireLight".

By default PDM Workbench does not support embedded components in the CATIA structure, because an embedded component does not have an own file, but is stored in the parent CATProduct. Therefore such a component cannot be reused under a different parent.

There is no need to map these leaf components to PDM documents / parts. The parent CATProduct of the embedded leaf components holds all information of the embedded leaf components.

This functionality allows two ways to use such embedded leaf components:

- a) Allow leaf components of any type in the CATIA structure. Leaf components are not mapped to PDM documents / parts
- b) Configure special types of like "ElecWireLight" to be allowed in the CATIA structure. Components of the configured types are not mapped to PDM documents / parts



Picture 152: Example document containing electrical components

To get the type of a component just use the PWB Update functionality. If there is an unsupported component, a message box shows the type of the first unsupported component:



Picture 153: Warning about unsupported CATIA component node

Download Drawing Option in Query Dialog

In the part structure mode the query result dialog contains a new check box with which the user can define whether CATDrawings related to the part being loaded should also be downloaded.

In this example two CATDrawing documents are related to the part item, in addition to the CATPart document:

B	OM BOM Structur	e Alte	ernates	AML Documents	CAD Docum	nents Characteristic
Act	ions 👻 Pick Related	-	1, 3	🖻 🔟 🔒 🔒	A 🔊 🗸	💫 🔊 🛛 Hide Search Crite
	Document Number	Revision	Name	Туре	State	Native File []
	NewPart1	А		Mechanical/Part	Preliminary	NewPart1.CATPart
Û	NewPart1-Drw1	А		Mechanical/Drawing	Preliminary	NewPart1-Drw1.CATDrawing
0	NewPart1-Drw2	Α		Mechanical/Drawing	Preliminary	NewPart1-Drw2.CATDrawing

Picture 154: CATDrawing documents related to part item

When the check box is checked then the related CATDrawings are downloaded and opened in the CATIA session when the part is loaded.

Part Number 🔺		Major Rev.	Generation	Name
🔚 NewPart1	-	A	1	NewPart1
i i i i i i i i i i i i i i i i i i i	Open in new Pl	OM Window		
) Unlock			
	Lock			
7	Promote			
1.	2 Revise			
	🔉 Load			
	🕽 Relate active F	ile to Part		
	Show Neighbor	hood		
	Colete newest	version		
	Duplicate			
-				
•				D
			🔎 Downl	load CATDrawings related to BOM Parl

Picture 155: Query result dialog with drawing download check box



Picture 156: CATDrawings opened in CATIA session

→

Support for relating a new CATIA file to an existing Part

The currently active CATIA document (only CATParts or CATDrawings) can be related to an existing BOM part item. If there is already a corresponding CAD document related to the part the document's file can be overwritten.

The CATPart or CATDrawing file which is the currently active document in the CATIA session can be related to a part in the query result list by a new context menu action:

	Part Number 🛛 🔻	Majo	or Rev. 🛛 🗍 Ge	neration	Name
¢	CylinderBlock	A	1	-	CylinderBlock
\$	Cylinder	Open in new PDI	M Window		Cylinder
\$	CrankShaftRightModel				CrankShaftRightModel
Q	CrankShaftRight				CrankShaftRight
\$	CrankShaftLeftModel	💾 Lock			CrankShaftLeftModel
2000	CrankShaftLeft	Promote			CrankShaftLeft
Q.	Crankshaft				Crankshaft
\$	CrankBoltModel	1+2 Revise			CrankBoltModel
0000	CrankBolt	५ 🛟 Relate active Fil	e to Part		CrankBolt
\$	ConnectionRodModel		<u>.</u>	-	ConnectionRodModel
Q.	ConnectionRod	А	1		ConnectionRod
\$	CaseModel	А	1		CaseModel
Q.	Case	А	1		Case
					F

Picture 157: "Relate active File to Part" context menu action

If there is already a CAD document related to the part the user is asked whether he wants to overwrite the corresponding file:



Picture 158: Overwrite prompt for "Relate active File to Part" context menu action

The file can only be overwritten if the BOM part item is not used in a Part BOM structure:



Picture 159: Information prompt for "Relate active File to Part" context menu action

"Delete relation" context menu action in the PDM structure window

PDM relations can be deleted in the PDM structure window with a single context menu action now, even if the PDM relations are not displayed in the structure.

The "Delete relation" action deletes the expanded parent relation of the selected PDM structure node in the PDM structure window (see *Picture 160: "Delete relation" context menu action*).



Picture 160: "Delete relation" context menu action

Support for CATIA Design Tables

It is possible to load and update text files or Microsoft Excel files which contain CATIA design table information.

The user can create a design table for a CATPart or a CATProduct.



Picture 161: CATPart with design table

Updating to PDM will create a Document item for the design table and upload the file:

Update to PDM?						
File name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation
NewPart1.CATPart	NewPart1				Modified	Create : 1. Create PDM File Object of type
DesignTable1.xls	DesignTable1				New	Create

Picture 162: Update dialog containing a design table

After the update the design table is related to the CAD document:

NewPart1, A, 1, NewPart1-Nomenclature, Preliminary, Mechanical/Part (Logged in as admin)
NewPart1, A, 1, NewPart1-Nomenclature, Preliminary, Mechanical/Part
Uses Values from Design Table (CAD DesignTable) / DesignTable1, A, 1, , Preliminary, Specification

Picture 163: Design table document related to CAD document

The design table file can be modified and uploaded to PDM again:

DesignTable.1 active, configuration row : 1		? <mark>— X</mark>
Design Table Properties		
Name : DesignTable.1		Activity
Comment : DesignTable created by ckessl	er 30.10.2013	
Configurations Associations		
📮 Filter :		Edit
Line PartBody\Pad.1\FirstLimit\Length	PartBody\Chamfer.1\ChamferRibbon.1\Length1	PartBody\Hole.1\Diameter NewPart1
<1> 15mm	3mm	10mm NewPart1
•	III	
Edit table		Duplicate data in CATIA model
		OK Apply Cancel

Picture 164: Editing a design table

	А	В	С	D
1	PartBody\Pad.1\FirstLimit\Length (mm)	PartBody\Chamfer.1\ChamferRibbon.1\Length1 (mm)	PartBody\Hole.1\Diameter (mm)	NewPart1\Nomenclature
2	15	3	10	NewPart1-Nomenclature
3	20	4	12	Nomenclature 2
4				

Picture 165: Adding a line to the design table excel sheet

DesignTable.1 active, configuration row : 1	5 X
Design Table Properties	
Name : DesignTable.1	General Activity
Comment : DesignTable created by ckessler 30.10.2013	
Configurations Associations	
🖾 Filter :	Edit
Line PartBody\Pad.1\FirstLimit\Length PartBody\Chamf	fer.1\ChamferRibbon.1\Length1 PartBody\Hole.1\Diameter NewPart1\Nomenclature
<1> 15mm 3mm	10mm NewPart1-Nomenclature
2 20mm 4mm	12mm Nomenclature 2
	Knowledge Report
	From Summary 1
	DesignTable.1 Design Table Synchronization I
Edit table	Message :
	The file of the design table DesignTable.1 has been modified.
	This design table has been synchronized with this file
	Close

Picture 166: The design table is updated in the CATIA session

Refreshing the PDM structure window shows that both the CATIA document and the design table are modified:

NewPart1, A, 1, NewPart1-Nomenclature, Preliminary, Mechanical/Part (Logged in as admin)
NewPart1, A. 1, NewPart1-Nomenclature, Preliminary, Mechanical/Part
Uses Values from Design Table (CAD DesignTable) / DesignTable1, A, 1, , Preliminary, Specification

Picture 167: Refreshed PDM structure window containing the design table

PDM update uploads both changed files.

When the design table functionality is switched on the design table files that are related to CAD documents are also downloaded when the CATIA files are downloaded.

Bounding Box Management / "Show Neighbor" functionality

PDM Workbench can be set up such that the bounding box values of the updated CATParts are saved in the PDM CAD document items. If that is done it is possible to use these values to find the neighbor geometry documents whose bounding boxes overlap with the bounding box of the selected CATPart.

As an example, the user wants to find out which bounding boxes of other CATParts in the structure "Engine.CATProduct" overlap with the bounding box of the CATPart "CrankShaftLeft.CATPart":



Picture 168: CATPart geometry in the context of a CATProduct structure

This is the corresponding CAD structure in PDM:



Picture 169: CATPart document in CAD structure

The user first queries for the CATPart document which he wants the neighborhood of. Then he clicks on the context menu "Show Neighborhood":

	Doc. Number 📧	Major Rev.	Generation	Name	State
5 1	CrankBolt	A	1	CrankBolt - Nome	Preliminary
5	CrankShaft	A	1	CrankShaft - Nom	Preliminary
5	CrankShaftLeft			CrankShaftLeft - N	Preliminary
5	CrankShaftRight	ିଟ୍ଥ Open in new	PDM Window	CrankShaftRight	Preliminary
		🖳 Unlock			
		Cock			
		Open File			
		🗇 Show Neigh	borhood		
		🟒 Duplicate			



A query window appears where the user can search for a CATProduct document (or an assembly part in the BOM part structure mode):

PDM Quer	ry con	text for Neig	ghborhood (L	ogged in a	as admin)			
PDM obj		pe					Doc. Number 📧	Major Rev.
CATPro	duct				-	1	Engine	A
PDM obj								
Doc. Nu		Engine		•				
Majo	r Rev.			•				
Gener	ation			-				
1	Vame			•				
	State			•				
Descri	ntion					1		

Picture 171: Query dialog for context assembly node

The structure, of which the selected CATProduct is the root document, has to contain the previously selected CATPart.

When the user double-clicks on the selected CATProduct a specific multi-level structure expand is performed which only returns the parts of the structure where the CATPart's bounding boxes overlap with the bounding box of the previously selected CATPart. This is a sub-set of the complete structure:



Picture 172: Reduced structure containing only neighbor models

When this structure is loaded to CATIA the user can see the geometry where the bounding boxes overlap with the originally selected CATPart's bounding box:



Picture 173: Reduced structure loaded to CATIA

If the selected structure does not contain the selected CATPart the user will receive a warning message:

F	2DM Workbench
	_ Information
	PDM Messages:
	No item returned from multi expand. Selected CATPart for neighborhood search was not found under the selected context or has no bounding box. (Warning)
	Science extra in the neighborhood scaler was not found direct the science context of his no bounding box. (Marning)
	Warning.



Automatic Part Creation in CAD Structure Mode

It is possible to automatically create BOM part items when a new CAD document is created.

The user can define whether for new CATIA files, where new CAD document items will be created in the Update process, new BOM part items should also be created:

U	pdate to PDM?							X
Γ	File name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Result
	CrankShaftRight.CATPart	CrankShaftRight	Гомоврау	LOCKCO by	TENTState	Modified	Create	Result
	CrankBolt.CATPart	CrankBolt				Modified	Create	
L	CrankShaftLeft.CATPart CrankShaft.CATProduct	CrankShaftLeft CrankShaft				Modified Modified	Create Create : 1. Create PDM File Object of typ	
	CrankShart.CATProduct	CrankShart				woomed	Create : 1. Create PDIVI File Object of typ	
P								
L								
Ŀ								
								-
		_						
	Create new file versions at update	? No C	reate new parts at update? Yes					
							Up	date Close
L								

Picture 175: "Create new parts at Update" check box

After the update process has completed the part items are created in PDM, and the corresponding CAD documents are related to the parts with the "Part CAD" relation.

Expanding the Part CAD relation shows the part items in the PDM structure window:

CrankShaft, A. 1	1, CrankShaft - Nomenclature, Preliminary, Mechanical/Assembly
🔅 Is attached I	by (Part CAD) / CrankShaft A. 1, , Preliminary
- 🚮 CAD Sub-St	ructure (/CAD Structure/Structure, CrankShaftLeft.1) / CrankShaftLeft, A, 1, CrankShaftLeft - Nomenclature, Preliminary, Mechanical/Part
- 🔊 CAD Sub-St	ructure (/CAD Structure/Structure, CrankBolt.1) / CrankBolt. A, 1, CrankBolt - Nomenclature, Preliminary, Mechanical/Part
	ructure (/CAD Structure/Structure, CrankShaftRight.1) / CrankShaftRight. A. 1, CrankShaftRight – Nomenclature, Preliminary, Mechanical/F

Picture 176: CAD structure with related Part items

Synchronize CAD structure to BOM

It is possible to perform a synchronization of the CAD structure information to the corresponding Part BOM structure if every CAD document in the structure has a corresponding Part item (see "Automatic Part Creation in CAD Structure Mode").

The user clicks on the "Synchronize to BOM" context menu on the CATProduct document:



Picture 177: "Synchronize to BOM" context action

If all CAD documents in the structure are related to a part, and if all the parts are locked by the user, then the CAD structure instance information (instance name, instance description, and transformation matrix) is applied to the part structure.

The resulting expand structure can be expanded in the PDM structure window:



Picture 178: Created or updated part structure

"Current" and "Released" Expand Modes for "CAD Structure"

In addition to the default expand mode for the CAD structure ("As Saved") the modes "Current" and "Released" are supported.

Two new CAD structure expand modes are available, Current and Released.

The existing default mode is the AsSaved mode.

Expand	CAD Sub-Structure
Expand Multiple Levels	 CAD Sub-Structure (Current)
📴 De_Expand	CAD Sub-Structure (Released)
Open in new PDM Window	Uses Values from Design Table
🚡 Unlock	Is attached by
🔒 Lock	Used in Structure
Rromote	Derives Drawing
192 Revise	<u>Denics statning</u>

Picture 179: Three CAD Structure expand modes

In an example, a CATPart in a structure exists in two generations, generation 1 and generation 2:



Picture 180: Generation 1 of CATPart

The structure in PDM uses the generation 1 of the CAD document:

	, A, 1, Rack - Nomenclature, Preliminary, Mechanical/Assembly (Logged in as admin)	
Ra	ck, A. 1, Rack - Nomenclature, Preliminary, Mechanical/Assembly	
- 31	CAD Sub-Structure (/CAD Structure/Structure, Case.1) / Case, A. 1, Case - Nomenclature, Preliminary, Mechanical/Part, (newest versio	:2)
-🔊	CAD Sub-Structure (/CAD Structure/Structure, Screw:4) / Screw; A. 1, Screw - Nomenclature, Preliminary, Mechanical/Part	
-🔊	CAD Sub-Structure (/CAD Structure/Structure, Screw.3) / Screw, A. 1, Screw - Nomenclature, Preliminary, Mechanical/Part	
-🔊	CAD Sub-Structure (/CAD Structure/Structure, Screw.2) / Screw, A. 1, Screw - Nomenclature, Preliminary, Mechanical/Part	
-51	CAD Sub-Structure (/CAD Structure/Structure, Screw.1) / Screw, A, 1, Screw - Nomenclature, Preliminary, Mechanical/Part	

Picture 181: CAD structure containing generation 1 of CATPart

When a CATIA structure is loaded in one expand resolution it is not possible to expand the structure using a different expand mode. If the user attempts that he gets a warning:



Picture 182: Warning about different expand resolution

When all CATPart and CATProduct windows are closed it is possible to expand the structure using a different expand resolution, for instance "Current":

Rack, A,	1, Rack - Nor	menclatur	e, Preliminar	y, Mechanic	al/Assemb	<mark>ly</mark>				
🗿 CAD	Sub-Structur	e (/CAD S	tructure/Stru	ucture, Case	.1 - Currer	nt) / Case, A,	2, Case - No	menclature, P	reliminary, M	lechanical/Part
CAD S	Sub-Structur	e (/CAD S	tructure/Stru	ucture, Screv	м.4 - Curre	nt) / Screw,	A, 1, Screw -	Nomenclatur	e, Preliminary	, Mechanical/P
🗿 CAD	Sub-Structur	e (/CAD S	tructure/Stri	ucture, Screv	м.З - Curre	nt) / Screw,	A, 1, Screw -	Nomenclatur	e, Preliminary	, Mechanical/P
										Mechanical/P

Picture 183: CAD structure expanded as "Current"

The "Current" structure, which contains the latest versions of all CAD documents, can be loaded into the CATIA session:



Picture 184: CATIA structure containing the latest generations of the CATIA documents

When the user updates a CATIA structure that was not expanded as saved a confirmation dialog appears. If the user continues the update all the CAD structure relations in the loaded structure will be updated to the current generations of the documents.



Picture 185: Confirmation dialog at Update

Support for the new CAD structure instance handling introduced in Innovator 9.4 and 10.0

A new relation with the name "CAD Instance" has been introduced, which contains instance information for "CAD Structure" relations.

One visible difference in the usage is that the CAD Structure relations become multiquantity relations, that is, there is only one relation for any number instances on a CATProduct.

Here is an example with four instances of a child part:



Picture 186: Structure with four instances

The four instances are stored in one CAD Structure relation which contains four CAD Instance relations:

File	Edit Vi	ews Search	Actions Re	ports To	ols Help			
Ŧ.		o 🖶 🖾 🛛		5 (*	1 ? 🗸			
	D Documo	ent	Document Numl Rack Name	A Rev	sion State Preliminary	<u>Select an</u>	image	
Creat Modif Modif	ted On: 9 fied By: Ii fied On: 9	nnovator Admin /24/2015 nnovator Admin /24/2015 nnovator Admin	Type Assembly Description	Auti		ersion 5R24		
Relea Effec	r Rev: A ase Date: tive Date: eration: 1 e: P	reliminary	Assigned Creato Innovator Admin Part		Designated User	Native F	k.CATProduct	
						1 <u>Sel</u>	ect and uploa	
Stru	cture Par	rents Files (Changes CA	D DesignTa				
Strue Action	ns 🕶 🛛 Pick I	Related 🔽 🕂	N X I		Ê ∎ (° A /		Search Criteria 💌	•
			N X I			State	Search Criteria 💌 🛛 🚺	Authoring Tool
	Sequence	Related 🔽 🕂	N X I		Ê ∎ (° A /		h	•

Picture 187: One CAD Structure relation for each used CAD document

CAD Strue	cture - 33727394B019	45E892ABDF8FFD47F45B - Wi	indows Internet Explorer
File Edit	t Views Search	Actions Reports To	ols Help
F 🗎	× 🤣 🖶 🔯	🗐 🔒 🖬 🏷 🖍	ti 🖃 ? 🗸
CAD Str	ructure	Document Number	
Instances			
Actions 🔻	No Related 💌	🔊 🗙 🖾 🚮 🔒	📔 🦈 욙 🔎 Hide Search Criteria 🔽 🛄 🔲 🏗 🔻
Sequence	Name	Description	Transformation Matrix 🔺
128	Screw.1		1 0 0 0 0 1 0 0 0 1 0 -35.000000000016 -30.18101391196255 50.44
512	Screw.4		1 0 0 0 0 1 0 0 0 1 0 -35.000000000016 24.81898608803745 50.440
256	Screw.2		1 0 0 0 0 1 0 0 0 1 0 35.9999999999984 -30.18101391196255 50.440
384	Screw.3		1 0 0 0 0 1 0 0 0 1 0 35.9999999999984 24.81898608803745 50.440
Ready		Itomo	1-4 of 4. Page 1 of 1 Aras Innovator

The CATIA instance information is stored in the CAD Instance relations:

Picture 188: CAD Instance information

Standard Part Functionality

In part structure mode, it is possible to define part items and their corresponding CAD document items as standard parts. Standard parts are supposed to be parts which are used in a wide variety of different contexts and which are generally not modified by the designer, only used in the product structures that the designer works on.

The user can query for a standard part explicitly. Please note that regular users can not lock and modify standard parts, they can only use them in their structures:

 StdPart1, A, 1, Standard part 1, Preliminary (Logged in as pv StdPart1, A, 1, Standard part 1, Preliminary Attaches (Part CAD) / StdPart1, A, 1, , Prelimina 				
PDM Query (Logged in as pwbuser1) PDM object type				
Std. Part 👻	Part Number 🖄	Major Rev.	Generation	Name Standard part 1
PDM object attributes Part Number *				

Picture 189: Querying for a standard part

Standard parts can be used like regular parts. The exceptions are that regular users can not create or update standard parts, and it is possible to import CATProduct structures which contain standard parts which are already defined in PDM. In that case the existing standard part items are used for that structure.

Check for CAD document CATIA release at PDM update

A new functionality optionally asks the user before overwriting a file which has been created with a lower release of CATIA V5.

If the user is about to overwrite a file which has been created with a lower release of CATIA V5 he is asked whether he wants to continue:



Picture 190: Asking the user whether to continue the update process

Local Workspace Information

It is possible to check the status of the CATIA documents which are downloaded to the local working directory (PWB_XMAP). A list displays the local files and information about their corresponding CAD documents in PDM if they exist.

When the user clicks on the "Local Workspace" icon a window containing a list of CATIA files appears:



Picture 191: "Local Workspace" icon

	Modified 🛛	File Name	Part Number	Major Rev.	Generation	Name	State
-	Yes	FixedParts.CATProduct	FixedParts	A	1		Prelimina
5	Yes	Rack.CATProduct	Rack	А	1		Prelimina
(ک	Yes	Screw.CATPart	Screw	Α	1		Prelimina
5	Yes	Engine.CATProduct	Engine	Α	1		Prelimina
(ک	No	CrankShaftLeft.CATPart	CrankShaftLeft	Α	1		Prelimina
()	No	CrankShaftRight.CATPart	CrankShaftRight	Α	1		Prelimina
(ک	No	CylinderBlock.CATPart	CylinderBlock	Α	1		Prelimina
-	No	CrankShaft.CATProduct	CrankShaft	Α	1		Prelimina
(ک	No	CrankBolt.CATPart	CrankBolt	Α	1		Prelimina
(No	Piston.CATPart	Piston	Α	1		Prelimina
-	No	PistonAssm.CATProduct	PistonAssm	Α	1		Prelimina
@)	No	PistonBolt.CATPart	PistonBolt	Α	1		Prelimina
-	No	PistonBoltAssm.CATProduct	PistonBoltAssm	Α	1		Prelimina
(گ	No	ConnectionRod.CATPart	ConnectionRod	Α	1		Prelimina
(ک	No	RetainingRing.CATPart	RetainingRing	Α	1		Prelimina
(گ	No	Case.CATPart	Case	Α	1		Prelimina
(ک	-	NewPart3.CATPart					
(ک	-	NewPart4.CATPart					
(ک	-	NewPart1.CATPart					
(-	NewProduct1.CATProduct					
-	-	NewProduct2.CATProduct					
(ک	-	NewPart2.CATPart					
(-	NewProduct3.CATProduct					

Picture 192: "Local Workspace" window
Optional Load of linked CATPart Files

In the CAD structure mode it is possible to load a single CATPart file or a CATProduct structure with the CATPart files which are linked with the 'CAD Structure/Reference' relation (CATIA multi-model links).

Clicking on "Load with Links" for CATProduct documents or on "Open File with Link" for CATPart documents downloads and opens the selected files, plus the CATPart files which are related by the 'CAD Structure / Reference' links in PDM.



Picture 193: "Open / Load files with links" context menu action

Newest Version Info Context Menu

For performance reasons not all version data is retrieved for CAD documents in a structure which are not current. To retrieve additional information (which the newest version is, and which user has locked the newest version) a new context menu action has been added.

In the following example a sub-structure of a CAD document tree is not current. By default the information on the PDM tree node is "not current".

Engine, A, 1, Engine - Nomenclature, Preliminary, Mechanical/Assembly (Logged in as admin):2	_
Engine, A. 1, Engine – Nomenclature, Preliminary, Mechanical/Assembly	
CAD Sub-Structure (/CAD Structure/Structure) / CrankShaft, A. 1, CrankShaft - Nomenclature, Preliminary, Mechanical/Assembly	
🔊 CAD Sub-Structure (/CAD Structure/Structure) / CrankShaftLeft, A, 1, CrankShaftLeft - Nomenclature, Preliminary, Mechanical/Part	
🗐 CAD Sub-Structure (/CAD Structure/Structure) / CrankBolt, A, 1, CrankBolt - Nomenclature, Preliminary, Mechanical/Part	
GAD Sub-Structure (/CAD Structure/Structure) / CrankShaftRight, A, 1, CrankShaftRight - Nomenclature, Preliminary, Mechanical/Part	
CAD Sub-Structure (/CAD Structure/Structure) / ConnectionRod, A, 1, ConnectionRod - Nomenclature, Preliminary, Mechanical/Part	
CAD Sub-Structure (/CAD Structure/Structure) / PistonAssm, A, 1, PistonAssm - Nomenclature, Preliminary, Mechanical/Assembly	
CAD Sub-Structure (/CAD Structure/Structure) / Piston, A. 1, Piston - Nomenclature, Preliminary, Mechanical/Part	
CAD Sub-Structure (/CAD Structure/Structure) / PistonBoltAssm, A. 1, PistonBoltAssm - Nomenclature, Preliminary, Mechanical/Assembly	
CAD Sub-Structure (/CAD Structure/Structure) / RetainingRing. A, 1, RetainingRing - Nomenclature, Preliminary, Mechanical/Part	
CAD Sub-Structure (/CAD Structure/Structure) / PistonBolt, A. 1, PistonBolt - Nomenclature, Preliminary, Mechanical/Part	
CAD Sub-Structure (/CAD Structure/Structure) / FixedParts, A. 1, FixedParts - Nomenclature, Preliminary, Mechanical/Assembly, (not current)	
CAD sub-structure (CAD structure/structure) / Cylinderblock, A. 1, Cylinderblock - Nomenciature, Preliminary, Mechanical/Part, (not current) CAD sub-Structure (/CAD Structure/Structure) / Rack, A. 1, Rack - Nomenciature, Preliminary, Mechanical/Assembly, (not current)	2
CAD Sub-Structure (/CAD Structure/Structure) / Case, A, 1, Case - Nomenclature, Preliminary, Mechanical/Assembly, "Not current)	
CAD Sub-Structure (/CAD Structure/Structure) / Screw, A. 1, Screw - Nomenclature, Preliminary, Mechanical/Part, (not current)	
22 Ad Sub-Suddare (VSA) Suddare (SA) Suddare (Sa	

Picture 194: Default information "not current"

Click on the "Newest Version Info" context menu ...



Picture 195: "Newest Version Info" context menu action

... retrieves additional information about the latest version of the documents.



Picture 196: Additional version information

Non-BOM CATParts and CATProducts

In the part structure mode it is now possible to define CATParts and CATProducts in the CATIA structure to be defined as not BOM-relevant. In this case no corresponding part items will be created in PDM.

Before a CATProduct structure is created in PDM it is possible to change the wanted PDM type from a part type like Assembly or Component to the CATIA file type:



Picture 197: Setting a CATProduct to the non-BOM type

In that case all sub-nodes of the CATProduct also become non-BOM.

CATParts also can be changed to the non-BOM type:



Picture 198: Setting a CATPart to the non-BOM type

The result is a structure in PDM which contains both part structures and CATIA document structures:



Picture 199: Resulting PDM Structure

When nodes are added to or removed from the non-BOM CAD document structure "Update" synchronizes the changes in the CATProduct structure to the non-BOM CAD structure, just like to the BOM part structure.

Configuration of BOM Part Structure

e.

In the BOM part structure mode it is possible to create product configurations where, depending of the currently set configuration context, only a sub-set of the product structure is expanded and loaded. With this functionality it is possible to create and to work on different configurations of the same product.

This is a small example of the configuration functionality which shows the configuration management with options:

First a category, in this example named "Color", has to be created.

\equiv	Contents	٩	Properties		4	A	<mark>&</mark>	Simple	Search 🔹	50	↓ 目 •
			.				Name		mandatory	mutually exclus.	
Þ	Administration		Category			•					
►	Change Management					4		E			
►	Cloud Services		Created By:	Catego	ory - 2	C97F9	A45E3B45F1	BF1AD	8629127338D - 1	Windows Internet	Explorer
*	Config Management		Created On: Modified Bv:	File E	dit	View	s Search	Act	ions Report	s Tools Help	
	BOMConfigurations		Modified On:	F 🗎	×	0	🖶 🚺	W	i in 5		? 🗸
	Categories		Locked By: Major Rev:	Catho				Name			Mutually Exclusive
	ConfigurationExpressions		Generation:	Categ	јогу			Color		Mandatory	
	I Lots		State:								
	C Milestones										
	C Options										
	C ProductVariants										

Picture 200: Creating a new category "Color"

Then option items which refer to the category "Color" are created, in this case named "Blue", "Green", and "Yellow".

I.			1	ſ,	* 🖬 🗖 🖓	? ⊡
=	Contents	··· Properties	٩	Я	🔎 🔗 Si	imple Search
		0.11			Name	Category []
	Administration	Option				•••
►	Change Management			F	Green	Color
•	Cloud Services	Created By:				
-	Config Management	Created On: Modified By:			s Search	F56A70EFFADE - Windows I Actions Reports To
	E BOMConfigurations	Modified On:	🕂 🗎 🗴	0	🖶 🖾 🗹	
	Categories	Locked By: Major Rev:				
	ConfigurationExpressions	Generation:	Option			me een
	I Lots	State:				tegory
	C Milestones				Co	
	C Options					
	C ProductVariants					

Picture 201: Creating the options "Blue", "Green", and "Yellow"

Then BOMConfiguration items are created which refer to these color options. The names are "BlueConfig", "GreenConfig", and "YellowConfig".

≡	Contents	•	Properties	٩	Я	<mark>&</mark>	Simpl	e Search 💌		•
	Administration		BOMConfigu	uration		Classifica	tion	Lot []	Range from	Range to
			bonconnge	liación	•			•••		
Þ	Change Management				Ŧ	Option				
►	Cloud Services		Created By:	C PONCF		55025200	44054		35954 - Windows Inte	
-	Config Management		Created On: Modified By:			s Search				rnet Explorer
	5 BOMConfigurations		Modified On:		0					
	Categories		Locked By: Major Rev:	+ 🗎 🗙	•	8	V7		ℰ゚ロ╎■╎?	
	ConfigurationExpressions		Generation:	BOMConfi	gura	ation	Name BlueCo		Classification Option	ld 5583F
	5 Lots		State:				Range	From	Range To	
	C Milestones									
	C Options						Rank	From	Rank To	
	C ProductVariants									
۲	Dashboards						Date F	rom	Date To	
*	Design									
	🔅 Parts						Milest	one From	Milestone To	
	Products									••••
*	Documents						Lot		Option	
	CAD DesignTable								Blue	••••

Picture 202: Creating BOMConfiguration items

The next step is to create configuration expressions ("BlueConfigExpr", "GreenConfigExpr", "YellowConfigExpr"). Configuration expressions can be combined using the logical operators AND, OR, and NOT.



Picture 203: Creating Configuration Expression items

Then a sample CATIA product structure is imported, creating a part BOM structure in Innovator.



Picture 204: Sample CATIA product structure

The previously created ConfigurationExpression items can be related to either "Part BOM" or to "BOM Instance" relation items.

Part - NewProduct2 - Windows	Internet Explorer		
File Edit Views Search	Actions Reports	Tools Help	
📮 🗎 🗔 😏 🖶 🚳	🚳 🔒 💕 5 🤅	* 🖞 🔲 ? 🗸	
Part	Part Number NewProduct2 Name	Revision State	Assigned Creator Innovator Admin ••• Designated User
Created By: Innovator Admin Created On: 4/25/2014 Modified By: Innovator Admin Modified On: 4/28/2014 Locked By: Innovator Admin Major Rev: A Release Date: Effective Date:	Type Assembly Long Description	Unit Make / Buy Cost	Effective Date
BOM BOM Structure Alte	ernates AML Docur	nents CAD Documents Goals	Changes Part Submission Warrants
Actions Pick Related	🕂 🐼 🗙 🖾 🐠	🔒 💕 🥂 🔗 🔗 🗄	iide Search Criteria 🔽 📗 🔚 🎼 🚹 👘 👘
Sequence Part Number	Revision Name	Type Quantity State	Unit Reference Configuration flag Configuration Expression
128 NewPart1	Α	Component 2 Preliminary	EA
256 NewPart2	A	Component 2 Preliminary	EA GreenConfigExpr
C Part BOM - 57273D3667904			
File Edit Views Sear	rch Actions Reports		
	🕼 🚳 🗎 👘 🏷	C 🗋 🔲 ? 🗸	
Part BOM	Quantity Part Num 2 NewPart1 Reference Designator	•••	
		× ×	
Instances Substitutes			
Actions No Related	+ 🔊 🗙 🖉 🖬	🔒 🔐 🥂 🔑 😣 🖽	de Search Criteria 🔽 📗 🚹 🎦 🎼 📗
Sequence Reference X	Y Z Angle Sid	e CATIA Instance Name CATIA Tr	ansfor CATIA Instance De Configuration Expression []
128		NewPart1.1 1 0 0 0 0	1 0 0 0 YellowConfigExpr
256		NewPart1.2 1 0 0 0 0	1 0 0 0 BlueConfigExpr

Picture 205: Relating configuration expressions to PLM relations

In order to be able to set the configuration context ProductVariant items have to be created ("BlueVariant", "GreenVariant", and "YellowVariant").

Ŧ		W	≗ r ⊅ ±	¢ •	¢	° 🖞 🖃 구	⊡	
≡	Contents		··· Properties	٩	Я	🔎 🔊 Simp	e Search	50
						Product Varian	Serial Number	Date []
•	Administration		ProductVariant		•			
•	Change Management				Ŧ	BlueVariant		
•	Cloud Services		ProductVariant - 16	EEDCCC63	6848	B4BF6A0AA0391C6	579 - Windows Inte	ernet Explorer
-	Config Management					Actions Report		
	E BOMConfigurations		F 🗎 🗙 📀		W		£ 🖞 🗖	2 🗸
	Categories				Pro	oduct Variant Name		•
	ConfigurationExpressions		ProductVariant		Blu	ieVariant		
	I Lots				Ser	al Number Date		
	C Milestones							
	C Options				Lot		Rank	
	ProductVariants							
+	Dashboards				Mile	stone		
-	Design							
	🔅 Parts							
	Products		Option List					
-	Documents		Actions Pick Rela	ted 💌	Ŧ		💵 🔒 👘 🤇	·· 🔉 😣
	CAD DesignTable		Name	Categ				
	CAD Documents		Blue	Color	7.5			
					_			

Picture 206: Creating Product Variant items

Now the previously created part structure can be expanded and loaded in different configurations.





First, if no configuration is set, the complete structure is expanded and loaded.



Picture 208: Expanding and loading the complete structure

Then, if a particular product variant is set, expanded, and loaded, then only the configured parts are expanded and loaded.



Picture 209: Setting different product variant expand filters



Picture 210: Loaded the "Blue" variant (one BOM Instance)



Picture 211: Loaded the "Green" variant (one Part BOM with all instances)



Picture 212: Loaded the "Yellow" variant (one BOM Instance)

Archives

It is possible to compress a complete CATProduct sub-structure into one Zip file and to manage this compressed file in PDM. This makes it possible to hide a complicated CATProduct structure in one CAD document if it is not necessary to manage the structure information in PDM.

Any CATProduct sub-structure which has not been created in PDM can be defined as an archive. If this is done the subsequent Update process compresses this CATProduct structure into one single ZIP file and manages this ZIP file as a CAD document in PDM instead of the normal CATProduct structure.



Picture 213: Defining a CATProduct structure as an archive



Picture 214: The resulting archive CAD document in PDM

Standard Part functionality for CAD structure mode

The standard part functionality has been extended to work with CAD document structures.

After the standard part CAD document items have been created any regular CAD user can query for them, by checking the "Standard Part" check box in the query dialog.

Regular users can not lock or otherwise modify standard part CAD documents:

(CATPart (Logged in as testuser)		
	5)		
	StdPart2, A, 1, StdPart2, Preliminary, Mechanical/Part		C <u>e</u> nter graph
	StdPart3, A. 1, StdPart3, Preliminary, Mechanical/Part		<u>R</u> eframe On
	🛐 StdPart1, A. 1, StdPart1, Preliminary, Mechanical/Part	8	Hide/Show
		1	Pr <u>o</u> perties
1			Ope <u>n</u> Sub-Tree
		X	Cu <u>t</u>
		Þ	Сору
		2	<u>P</u> aste
			Paste <u>S</u> pecial
			Delete
			StdPart2, A, 1, StdPart2, Preliminary,
			Expand
			Expand Multiple Levels
		1 8	De_Expand
		1	Open in new PDM Window
		1	<u>Unlock</u>
		1	Lock
		7	Promote

Picture 215: Using standard parts as a regular user

They can use standard parts in the CATIA structures that they work on:



Picture 216: Using standard parts in CATIA structures

Adding standard parts to an existing structure at first does not seem different from adding other CATPart nodes ...

File name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Result
NewPart2.CATPart	NewPart2				Modified	Create	
StdPart1.CATPart	StdPart1				Modified	Create	
StdPart2.CATPart	StdPart2				Modified	Create	
NewProduct2.CATProduct	NewProduct2				Modified	Create : 1. Create PDM File Object of typ	

Picture 217: Update dialog with standard parts

... but the standard parts are not created by the update process, but the existing ones, which have been created by the standard part administrator, are used:



Picture 218: Existing standard parts being used in a new structure

Standard part CATParts can be inserted to a CATProduct structure manually or by the CATIA catalog functionality. For this the standard CATParts have to be added to a CATIA catalog file first.

The catalog file can reside in any client directory which is accessible to CATIA V5. It can be a network drive.



Picture 219: CATIA catalog containing standard part CATParts

Then the standard parts can be inserted to a CATProduct structure:



Picture 220: Standard part CATParts created from a catalog



Picture 221: Inserted standard parts

In the update process the standard part item from the database is taken.

file name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Result
StdPart3.CATPart	StdPart3				Modified	Create	
C00000119-AA_M000.CATPro	C00000119-AA_M0	C00000119-AA_M000, A, 1, NewProduct	testuser	Preliminary	Modified	Update : 1. Add sub-structure relation "	

→



Picture 223: Update result

It is important to make sure that the standard CATPart files in the Innovator vault and in the local directory are exactly the same.

As with regular CATParts, the new standard CATPart node is added to the CAD document structure:



Picture 224: "Show PDM Structure" icon



Picture 225: CAD Document structure containing standard parts

Check CAD Links

When CATIA documents with 3D links need to be imported this functionality helps the user to determine which documents have to be imported in which order, and which documents have to be in the CATIA session so the links are created correctly.

When a CATPart or a CATDrawing is the active CATIA document the user can click on the "Check CAD Links" icon to get information about which of the linked CATPart documents already exist in PDM, and which still have to be created:



Picture 226: "Check CAD Links" icon

In addition to this the functionality also opens a window containing all the CAD document items in PDM which should be opened in the CATIA session before the current CATIA document is imported to PDM.

Resul								? ×		
CAT	IA Documents in PD	м								
	Doc. Number	Major Rev.	Generation	Name	State	Description	Created on	Modified on		
1	LinkSourcePart1	Α	1		Preliminary		2014-10-30T16:23:08	2014-10-30T16:		
1	LinkSourcePart2	Α	1		Preliminary		2014-10-30T16:23:17	2014-10-30T16:		
ſ	PDM Workbench									? 💌
	Information									
	CATIA Link Cheo	k Result:								
	Please first load LinkSourcePart1. LinkSourcePart2. Then open "D:\D Please create the LinkSourcePart3. LinkSourcePart4.	(or close and CATPart, CATPart Data\CatiaFile an the followin CATPart, CATPart	re-load) the f s\Links\LinkT ng CATPart d	ollowing argetPar locumen	, CATPart doc t2.CATPart" a ts in PDM:	uments form gain.			o the correct documents.	
	V Action su	cceeded.								
										Close

Picture 227: Result of "Check CAD Links" action

Displaying part structure instances as separate nodes

The display of the part structure in the PDM structure window can be changed such that every part instance is shown as a separate node.

When this functionality is switched on and a part structure containing several instances of the same part is expanded then all the instances are shown as separate nodes:



Picture 228: Part structure showing every instance as a separate node.

Saving PDM Session Information

It is possible to save the content of a PDM structure window in a PWBDoc file, and to reload the content of that window later from that file. The window created by opening the PWBDoc file has the same properties as a PDM structure window opened by querying and expanding PDM nodes, except that the content may be out of date with the actual server database for a longer time.

Usage

The content of any PDM structure window can be saved to a PWBDoc file:



Picture 229: Example content of a PDM structure window

The content of this window can be saved by selecting "File / Save As" from the menu:



Picture 230: PWBDoc save dialog



Solution </th <th></th> <th>✓ 4 PWB_XMAP_TEST durchsuchen</th>		✓ 4 PWB_XMAP_TEST durchsuchen
Organisieren 🔻 Neuer Ordner		1000
🕽 Bibliotheken	▲ Name	Es wurden keine Suchergebnisse gefunden.
System (C:) Lokaler Datenträger (D:) Data Iicman20_ptf20_distrib_windows	E	
 Program Files Program Files (x86) Program Data 		
 temp VirtualBox VMs XMAP 		
PWB_XMAP PWB_XMAP_TEST		
Dateiname: Session1.PWBDoc		

Picture 231: Saving the window content under a specific name

After saving, the new PWBDoc file can be seen in the Windows Explorer:



Picture 232: Newly created PWBDoc file

In the same session, or in a later session, this file can be opened again:



Picture 233: Opening a PWBDoc file (1/2)

🕒 🗢 📕 « XMAP 🕨 PWB_XN	IAP_TEST2 🗸	PWB_XMAP_TEST	2 durchsuchen
Organisieren 🔻 Neuer Ordner		:==	- 🖬 🌘
🌗 PWB_XMAP (D:)	Name	Änderungsdatum	Тур
PWB_XMAP_TEST (D:) PWB_XMAP_TEST2 (D:)	Session1.PWBDoc	04.05.2015 15:30	PWBDOC-Da
4 🜉 Computer			
> 🚔 System (C:)			
4 🗐 Lokaler Datenträger (D:)			
🛛 퉲 Data			
🛛 퉬 licman20_ptf20_distrib_ 😑			
Program Files			
Program Files (x86)			
🛛 🎍 ProgramData			
🌗 temp			
🛛 🎍 VirtualBox VMs			
a 🌗 XMAP			
PWB_XMAP			
PWB_XMAP_TEST			
PWB_XMAP_TEST2			
Datei <u>n</u> ame:	•	All Files (*.*)	
	Open as read-only	Öffnen	Abbrechen

Picture 234: Opening a PWBDoc file (2/2)

The file can also directly be opened from the most recently file list:



Picture 235: Opening a PWBDoc file from the most recently used file list

The user has to be logged on to save or to load a PWBDoc file.

After the file is opened the PDM structure window can be used like any other opened PDM structure window.



Picture 236: PDM structure window opened from PWBDoc file

Options

Once you are logged in into the PDM Workbench you can set some options for the PDM Workbench.

You open the options dialog with *Tools* \rightarrow *Options* in CATIA V5. In the slider "PDM Workbench" you can set the options for the PDM Workbench (see *Picture 237: The PDM Workbench options*).

Options		<u>?</u> ×
Options General G	Graphics Formats IGES IGES 2D PDM-Workbench SMARTEAM (CAI) SMARTEA Version	
	окос	ancel

Picture 237: The PDM Workbench options

Query Mode

When you set "List View" for the Query Mode the query result will be opened in a list view window.

You can customize the columns to be used in the list view window.

Please click "Customize List View" to open the "Customize List View" dialog (see *Picture 238: "Customize List View" dialog*).

PD	M Session		?×
	Attributes PDM Types Part Assembly Component All CATIA Files CATPart CATProduct CATDrawing cgr	Attribute Settings Add -> <- Remove	Up Down
	Reset All	Preview	
			Close

Picture 238: "Customize List View" dialog

When you are using a new PDM Workbench configuration file then you have to reset the column settings by clicking on "Reset All".

In order to customize the columns for an object type you have to select the object type. In the example in *Picture 239: "Customize List View" dialog for "Assembly"* the object type "Assembly" has been selected. In the right part of the dialog the attributes to be shown as columns are displayed. In the middle part of the dialog the attributes not to be shown as columns are displayed. In this case no attribute is hidden.

You can select an attribute on the right and remove it by clicking on the "Remove" button. Or you can select an attribute in the middle and by clicking on the "Add" button you can add it to the columns to be shown.

PD	M Session				? ×
	PDM Types Part Assembly Component	- Attribute Settings - Exclude-List of : /Pa	Add ->	/Part/Assembly item_number	Up
	Component All CATIA Files CATPart CATProduct CATDrawing cgr		<- Remove	name state major_rev generation	Down
	Reset All			Preview	
					Close

Picture 239: "Customize List View" dialog for "Assembly"

By clicking on the "Preview" button you can see a preview of the list view (see *Picture 240: Preview of the "List View" dialog*).



Picture 240: Preview of the "List View" dialog

PDM Relations

You have the possibility to hide or to show the PDM relations in the PDM product structure.

CATDrawings

CATDrawing files that are related to the root part of a part structure can be automatically downloaded when the part structure is loaded to CATIA.

Loading PDM Structures

When a PDM structure is loaded to CATIA sub-structures in the CATProduct tree can be hidden or deactivated if their corresponding PDM structures are not expanded.

PDM Session Configuration

No configuration has to be set currently from CATIA for the Aras integration.

The configuration has to be made directly in the Aras Innovator application. For details please refer to the *PDM Workbench Installation & Administration Manual.*

Logout

Once you finished your work in PDM Workbench you do a Logout from the PDM system.

You select the "Logout" icon within the PDM Workbench toolbar (see *Picture 241: The PDM Workbench toolbar after the login*) in CATIA V5 ...



Picture 241: The PDM Workbench toolbar after the login

... and the session in the PDM system will be closed.

All PDM Workbench windows get closed. Please consider that CATIA native windows resulting from a "Load" or "Open File" PDM Workbench context action remain opened but that they are now out of synchronization with the PDM system. So we recommend you to close them, too.

CHAPTER 4

Additional optional functionality

This chapter describes optional functionalities of the PDM Workbench which are able to add in the CATIA V5 workshop.

Copy element attributes

It is possible to copy the attributes from a PDM object in order to use them in a create dialog.

You can select a PDM object in the PDM window and click on the right mouse button. Then you select the action "Copy element attributes" (see *Fehler! Verweisquelle konnte nicht gefunden werden.*).



Picture 242: Action "Copy element attributes"

The attributes will be copied to the clipboard.

In the next step you select the action "Create" from the toolbar and select the corresponding class for the object to be created, in this case "Assembly" for the copied attributes of the "Engine". The create dialog will be opened. It has the button "Insert attribute values" (see *Fehler! Verweisquelle konnte nicht gefunden werden.*).

PDM Create	<u></u>	? ×
Create dialog	for type Assembly:	
Part Number		*
Name		
Unit	▼	
Make / Buy	•	
Cost		
Description		
a de l'Altres		
		_
	Insert attribute valu	es
	lan 🤇 🕹 Apply	cel
CONCERNMENT OF THE OWNER		

Picture 243: "Create" dialog for Assembly

When you click on the button "Insert attribute values" the attributes of the dialog will be filled (see *Fehler! Verweisquelle konnte nicht gefunden werden.*).

PDM Create		? ×
Create dialog	for type Assembly:	
Part Number	Engine	*
Name	Engine	
Unit	EA	1
Make / Buy	Make 💌	İ
Cost		
Description	Engine description	
	Insert attribute valu	les.
	lan 🔰 🕒 Apply	cel

Picture 244: "Create" dialog for Assembly - Inserted attribute values

You can change the attribute values and start the create process by clicking on the "OK" button.

Autoname Support using Innovator Sequence Items

It is possible to optionally use Innovator sequence items to rename CATIA structures or single CATIA documents when they are created (see *Fehler! Verweisquelle konnte nicht gefunden werden.*).



Picture 245: CATIA structure before and after import to PDM

In the login dialog the user can select one of the autoname rule (Innovator sequence item) names (see *Fehler! Verweisquelle konnte nicht gefunden werden.*).

PDM Login		? ×
User information	on	
User	admin	•
Password	******	
Database	InnovatorSolutions	•
Autoname Rule		-
	PwbTestSequence1 PwbTestSequence2 CAD Document	el

Picture 246: "Login" dialog with autoname rule

If none of the names are selected then the autoname functionality is not used.

Later in the session the user can change the selected autoname rule by clicking on the "PDM Session Configuration" icon in the PDM Workbench toolbar and selecting one of the sequence item names. This dialog can also be used to switch off the autoname functionality by selecting the entry containing the empty string (see *Fehler! Verweisquelle konnte nicht gefunden werden.* and *Fehler! Verweisquelle konnte nicht gefunden werden.*

ſ	Set PDM Configuration	
	Set PDM Configuration : PDM Session Con	figuration
	Autoname Rule	
	OK Apply Ocancel	

Picture 247: PDM session configuration dialog



Picture 248: Autoname rule combo box in PDM session configuration dialog

If an autoname rule is selected the update dialog will contain the information which autoname rule is selected (see *Fehler! Verweisquelle konnte nicht gefunden werden.*).



Picture 249: Selected autoname rule displayed in update dialog

After creating new PDM items which correspond to the new CATIA documents the CATIA files will be renamed. The CATIA instance names will not change.

The corresponding PDM items will also have the names created by the selected sequence item (see *Fehler! Verweisquelle konnte nicht gefunden werden.*).

NwbTst1	0000119Suff1, A, 1, , Preliminary, Mechanical/Assembly, (Logged in as Usr1)	
	100000119Suff1, A. 1, , Preliminary, Mechanical/Assembly,	
÷-🔊 <u>c</u>	Sub-Structure (/CAD Structure/Structure, CrankShaft.1) / PwbTst100000115Suff1, A. 1, , Preliminary, Mechanical/Assembly	
	CAD Sub-Structure (/CAD Structure/Structure, CrankShaftLeft.1) / PwbTst100000118Suff1, A, 1, , Preliminary, Mechanical/Pa	rt,
	CAD Sub-Structure (/CAD Structure/Structure, CrankBolt.1) / PwbTst100000117Suff1, A, 1, , Preliminary, Mechanical/Part	
_3	CAD Sub-Structure (/CAD Structure/Structure, CrankShaftRight.1) / PwbTst100000116Suff1, A, 1, , Preliminary, Mechanical/F	⁹ art
	Sub-Structure (/CAD Structure/Structure, ConnectionRod.1) / PwbTst100000120Suff1, A, 1, , Preliminary, Mechanical/Part,	
- 🔊 😋	Sub-Structure (/CAD Structure/Structure, PistonAssm.1) / PwbTst100000113Suff1, A, 1, , Preliminary, Mechanical/Assembly	<u>,,</u>

Picture 250: PDM structure named by sequence item

Further updates will not affect the names of the CAD documents and PDM items.

Possibility to call a server method for a PDM item

It is possible to call custom server methods with a PDM item and optionally with a dialog as input.

The user can right-click on a PDM item and select one of the custom server methods.

	Part Number	N.	Major Rev.	Generation	Name
Ċ	Connec			1	
₽	LinkSou 🖪	Open in new PDM Window		1	
₽	LinkSou 🖳	Unlock		1	
		Lock			
		Promote			
	1+2	Revise			
	e	Load			
	÷⇔	Relate active File to Part			
	a	Show Neighborhood			
	-××	Delete newest version			
	~	Duplicate			
	* *	Expand	•		
		Custom Action	• <u>C</u> ustor	m Server Method	

Picture 251: Selecting a custom method on a part item

If a dialog is configured it appears pre-filled with the attributes of the item:

PDM Workben	ch Attributes Dialog	
Part Number	ConnectionRod 👻	
Major Rev.	A 👻	
Generation	1 🔹	
Name	•	
State	Preliminary 👻	
Unit		
Make / Buy	Make 🔻	
Description		
Created on	2014-10-31T13:21:32 -	
The second second second	2014-10-31T13:21:32	
	Innovator Admin 👻	
	Innovator Admin 👻	
LOCKED by	Innovator Admin 👻	
2	OK Apply Cancel	

Picture 252: Dialog with pre-filled attributes

	Glossary
Unlock	Action withdrawing the right to update a work item. Normally this corresponds with publishing the work item to a larger number of people getting read access on this object.
Lock	Action giving the user the exclusive right to update a work item.
Context Menu	The menu that appears when the user selects an <i>icon</i> and holds the right mouse button pressed.
Dialog Window	Window in which the user enters information.
Gli	Graphics Interactive Interface. The GII is a powerful programming tool, which completes the Open System Access to the CATIA environment.
lcon	Graphical representation of an <i>object</i> .
Object	An item or a relationship.
Query	To search the database for <i>objects</i> that match specific criteria.