

PDM Workbench Release 19.0 for Aras Innovator

User Manual

Version 1



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This edition 19.0 of the manual obsoletes all previous editions.

Your Comments are Welcome

Please feel free to tell us your opinion; we are always interested in improving our publications. Mail your comments to:

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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	19.0
PDM Workbench User Manual	19.0

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.

Chapter 4 describes the additional optional functionality.

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 PDM structure. Or you can create new objects in this window.

You can load this PDM 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

The PDM Workbench maps the structure in the PDM system and the structure of the CATIA files. The CATProducts and CATParts are identified by their Part Number and file types like *.CATDrawing or *.cgr are identified by their File Name. The modified structure can be updated in the PDM system. The Part Numbers and File Names are controlled by the PDM system.

For new CATParts and CATProducts new Parts will be created in the PDM system. Also if a Part Number is changed in CATIA the CATProduct/CATPart will become a new object in the PDM system.

CHAPTER 2

Supported Data Models

The PDM Workbench supports two different data models:

- BOM Part Structure Data Model
- CAD Document Structure Data Model

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: PDM structure in the BOM Part Structure Data Model*).



Picture 2: PDM structure in the BOM Part Structure Data Model

CAD Document Structure Data Model

In the *CAD Document Structure 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: PDM structure in the CAD Document Structure Data Model*).



Picture 3: PDM structure in the CAD Document Structure 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	? ×
User info	ormation
User	*
Password	
Database	InnovatorSolutions1112 💌 *
	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*).





For CATPart and CATDrawing as top-level object of the CATIA V5 window you have to use the actions "PDM Properties", "Unclaim", "Claim", 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 "Claim" to "Unclaim") when you newly activate the CATIA V5 window.



PDM Properties Unclaim Claim Duplicate Check CAD links Open related drawings

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 "PDM 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 (logged in as admin, database: Innovato	rSolutions1112)				? ×
PDM object type	Part Number	Major Rev.	Generation	Name	State
Part	-				
PDM object attributes					
Part Number					
Major Rev. 🗸					
Generation 👻					
Name					
State 🗸					
Unit					
Make / Buy					
Description					
Created on					
Modified on					
Created by					
Modified by					
Locked by					
		Load with Links	1	Download CATDrawings	related to BOM Part
				5	
					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 (see *Picture 9: "PDM Query" dialog – enter query criteria*).

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 "PDM 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 (log	ged in as admin, database: Innova	torSolutions1	112)				? ×
PDM object ty	pe		Part Number	Major Rev.	Generation	Name	State
Part		-					
PDM object at	tributes						
Part Number	Cr* 👻						
Major Rev.	A 🗸						
Generation	•						
Name	•						
State	-						
Unit	•						
Make / Buy	•						
Description							
Created on	-						
Modified on	•						
Created by	•						
Modified by	-						
Locked by	-						
			•	III			۱.
				Load with Links	a (Download CATDrawing	gs related to BOM Part
						[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 Dialog*).

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*).

PDM object type		Part Number 🖉	Maior Rev.	Generation	Name	State
Part	-	CrankPin	A	1		Preliminary
	-	CrankPinModel	A	1		Preliminary
PDM object attributes	+	Crankshaft_Assm	А	1		Preliminary
Part Number Cr*	-	🗘 CrankshaftLeft	А	1		Preliminary
Major Rev. A		🔅 CrankshaftLeftModel	A	1		Preliminary
Generation 🗸	1	🗘 CrankshaftRight	А	1		Preliminary
Name		🔅 CrankshaftRightModel	A	1		Preliminary
State						
Unit						
Males (Proc						
Wake / Buy						
Description						
Created on						
Modified on						
Created by						
Modified by						
Locked by						
				_		
	Ľ	< [
		6	Load with Links	a	Download CATDrawi	ngs related to BOM Pa
						,

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"*).



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 "PDM 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*).



Picture 12: Query result in PDM Structure window

"Select Date" Widget

It can be complicated to write a date in the correct format into the date fields of the query dialog. With this functionality you can select the dates from a calendar widget.

The date fields have a button on the right of the attribute value. Clicking on that button opens the "Date Selection" dialog:

		Date Selection		? ×
Created on 2018-05-15 << 2018-05-17	-	Salact guan made and	data(s) of the colocted at	tribute "Created on"
Modified on	-	○ equal ○ after > .	Defore < In the selected at the selecte	to
Created by	-	2018	5	- 15 -
Modified by	-	2010		
Locked by	•	2018	· > ·	r 1/ •
Is Template 🗌				OK Gancel
Standard Part 🔄				

Picture 13: Date Field and Date Selection Dialog

In this dialog you can choose the options

equal ..

Query for objects with a specific date. Only the values of the first date have to be selected.

- after > ..
 Query for objects newer than the selected date. Only the values of the first date have to be selected.
- before < ..
 Query for objects older than the selected date. Only the values of the first date have to be selected.
 - from .. to .. Query for objects between two dates. The values of both dates have to be selected.

The values of the dates can be selected from the drop down lists. The user can also type in the values manually.

After OK is pressed the date is filled into the query dialog:



Picture 14: Date field filled after the "Date Selection Dialog"

Download Drawing Option

In the BOM Part Structure Data Model the "Query" result dialog contains a check box with which you 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 (see *Picture 15: CATDrawing documents related to part item*):

BO	M BOM Structure	Alternate	s AML	Documents CAD Do	ocuments Go	als Changes Part	Submission Warrants	
Ac	tions 👻 🛛 Pick Related	•	+	X 🛛 🖬 🔒	💼 🖑 🖌	🖡 👧 🔊 🖡 Hide	Search Criteria 🔻	
	Document Number 🛦	Revision	Name	Туре	State	Native File []	Viewable File []	Authoring Tool
	NewPart1	Α		Mechanical/Part	Preliminary	NewPart1.CATPart	1 Select file	CATIA
₽	NewPart1-Drw1	Α		Mechanical/Drawing	Preliminary	NewPart1-Drw1.C	. 🟦 <u>Select file</u>	CATIA
₽	NewPart1-Drw2	А		Mechanical/Drawing	Preliminary	NewPart1-Drw2.C	1 Select file	CATIA

Picture 15: CATDrawing documents related to part item

When the check box is checked (see *Picture 16: "Query" result dialog with "Download CATDrawing related to BOM Part" check box*) then the related CATDrawings are downloaded and opened in the CATIA session when the part is loaded (see *Picture 17: CATDrawings opened in CATIA session*).

PDM object type		Part Number	Major Rev.	Generation	Name	State
Part		NewPart1	*	1		Preliminary
DM object attributes			👩 Open in new F	DM Window		
Part Number NewPart1	-		Unclaim			
Major Rev. A	•		-			
Generation	•		Claim			
Name	•		Promote			
State	-		1+2 Revise			
Unit	-					
Description	•		By Load *			
Description			→ ○ Relate active I	ile to Part		
			Seate new ge	neration		
Created on	-		🖾 Duplicate			
Modified on	-		🕵 Duplicate Stru	cture		
Created by	-					
Modified by	•		A Open in Aras			
Locked by	-		Expand	•		
	•		in .			
			🔎 Load with Links	12 I	Download CATDrawin	ngs related to BOM P

Picture 16: "Query" result dialog with "Download CATDrawing related to BOM Part" check box

New P	Part1.CATPart	
	NewPart1-Drw2.CATDrawing (NewPart1, A, 1, , Preliminary)	
1	NewPart1-Drw1.CATDrawing (NewPart1, A, 1, , Preliminary)	
j	Sheet.1	
	Rear v	
	Top vi	

Picture 17: CATDrawings opened in CATIA session

Automatically loading CATDrawings or linked CATParts

→

It is possible to automatically load linked CATDrawings or CATParts when a CATPart or a CATProduct structure is loaded. There are two check boxes in the "PDM Query" dialog where this behavior can be set (see *Picture 18: "Load with Links" and "Load with Drawings" check boxes*).

DM shistbar							
	-	Doc. Number	Major Rev.	Generation	Name	State	D
All CATIA Files	•						
DM object attributes							
Doc. Number	-						
Major Rev.	-						
Generation	-						
Name	-						
State	-						
Description							
Created on	-						
Modified on	-						
Created by	-						
Modified by	-						
Locked by	-						
Is Template							
Standard Part							
xternal Owner T-Systems.Mechanical.CAT	IA 👻 *	•					
	(
					🖬 Load	with Links 🔰 Load w	th Drawin
						C Ourse 1	Class

Picture 18: "Load with Links" and "Load with Drawings" check boxes

If the corresponding checkboxes are checked simply opening the CATPart document or loading the CATProduct will also load the related CATDrawing or the linked CATPart documents.

Additional Options for "Load with Links"

The "Load with Links" functionality is used to download documents referenced by CATParts (Aras Innovator relation (default): "/CAD Structure/Reference") (see *Picture 19: Activate "Load with Links"*).

PDM Query (logged in as admin, database: InnovatorSolutions	1112)					
PDM object type	Doc. Number	Major Rev.	Generation	Name	State	De
All CATIA Files						
PDM object attributes						
Doc. Number						
Major Rev.						
Generation						
Name						
State						
Description						
Granted an						
Medified on						
Modified by						
Locked by						
Is Template						
Standard Part						
External Owner T-Systems Mechanical CATIA	4					
- of sector methods of the						
				🔎 Load	with Links 🗌 Load w	ith Drawing
					Query	Close

Picture 19: Activate "Load with Links"

By default, the functionality only downloads the linked files into the PDM Workbench exchange directory to make sure that there is no broken link in the referencing CATPart.

In addition to the standard behaviour you can choose three options in the PDM Workbench options panel (see *Picture 20: Additional options for "Load with Links"*).

PDM Workbench Options
Version PDM Workbench CATIA V5 Module Version: 9.0.0 CATIA Level: V5R28 (c) T-Systems 2005 - 2018
Query Dialog Keep Query Dialog Values for other Types Customize List View
PDM Relations
Loading related Files Download CATDrawings related to BOM Part Load with Links
 Load with Drawing Loading PDM Structures Enable Activate/Deactivate Enable Hide/Show
 No Change Expand Neighborhood [mm] Clearance for Expand Neighborhood
Copy Position
Load With Links Load current generation of linked Document Load multiple levels of linked Documents Open linked Documents in own Window
Cancel

Picture 20: Additional options for "Load with Links"

- Load current generation of linked Document: Get the "Current" generation instead of "AsSaved" (this option cannot be used if UseCadStructureVariantExpand=true)
- Load multiple levels of linked Documents: Also load the documents referenced by a referenced document.
- Open linked Documents in own Window: This option is only available in CAD Document Structure Data Model. The linked documents are opened in their own CATIA window and can be used in the same way as if they were opened normally (Update, Claim, Unclaim, ...)

Filter Attribute Values are kept when changing the Type

If the value of an attribute is set, and if the new type has an attribute with the same name, then the value is kept for the new type.

This example shows how the document number attribute value for querying CAD Documents is kept when you switch to querying Part items.

The first query is for All CATIA Files with the criteria Document Number "Engine*" (see *Picture 21: Document number query value for CAD Documents*).

DM object type			Doc. Number	A	Major Rev.	Generation	Name	State	D
All CATIA Files		• 🔬	Engine		A	1		Preliminary	
DM object attributes			-						
Doc. Number Engine*	-								
Major Rev.	-								
Generation	•								
Name	-								
State	-								
Description									
Created on									
Modified on	-								
Created by									
Modified by	-								
Locked by	-								
Is Template									
Standard Part 🗌									
ternal Owner T-Systems.Mechanical.CA	TIA - XITI	•			m				
							🖬 Load	with Links 🔲 Load with	Drawin

Picture 21: Document number query value for CAD Documents

Then the type will be changed to "Part" (see *Picture 22: Switching to Part items*).

DM Query (logged in as admin, database: InnovatorSolution	s1112)						? ×
PDM object type		Doc. Number	×	Major Rev.	Generation	Name	State	De:
All CATIA Files	- E	Engine		Α	1		Preliminary	
All CATIA Files	1 -							
CATPart								
CATProduct								
CATDrawing								
catalog								
cgr								
Archive								
Part								
Assembly	1							
Component								
Std. Part								
Design Table								
Created on								
Modified on								
Created by								
Modified by								
Locked by								
LOCKed by								
Is Template								
Standard Part								
External Owner T-Systems.Mechanical.CATIA V	1			11				÷.
						🖾 Load	with Links 🔲 Load with	Drawing
							Query	Close

Picture 22: Switching to Part items

The query criteria remains "Engine*" for the Part Number attribute (see *Picture 23: Part number value is taken from CAD Document number value*).

DM object type		Part Number	Major Rev.	Generation	Name	State	De
Part	•						
DM object attributes							
Part Number Engine*	-						
Major Rev.	•						
Generation	•						
Name	•						
State	•						
Unit	•						
Make / Buy	•						
Description							
Created on	-						
Modified on	-						
Locked by	-						
Created by	-						
Modified by	•						
			m				
					_	_	
					🖬 Load	with Links 🔎 Load wi	ith Drawin

Picture 23: Part number value is taken from CAD Document number value

Load of multiple Assemblies in the Query Dialog

It is possible to select multiple CATProduct CAD Documents in the query dialog for loading. The functionality makes sure that the generations included in the selected structures are unique, that is, that only one generation of each CAD Document is loaded.

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 you select the direction "Uses Part BOM" for the selected "Assembly" object "Engine" (see *Picture 24: Action "Expand Single Level"*).

Engine, A, 1, , Preliminary	(logged in as admin, data	abase: InnovatorSolut	ions1112)	
Set Engine , A, 1 ,	Center graph Reframe On Hide/Show Properties Open Sub-Tree Cot Cot Cot Paste Paste Paste Special Delete Engine, A, 1, , Prelim	Alt+Enter Ctrl+X Ctrl+C Ctrl+V Del		
	Expand Exp <u>a</u> nd Multiple Leve De_Expand E Compare with Assen	els 🕨	Uses Part BOM Attaches Is Used in Part BOM	x y

Picture 24: Action "Expand Single Level"

The related objects for the selected "Assembly" object "Engine" are shown (see *Picture 25: Result of expand single level*).

In the PDM 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 25: 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 (see *Picture 26: Result of expand single level without relations*).



Picture 26: 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.

"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" (see *Picture 27: Three CAD structure expand modes*).

The existing default mode is the "AsSaved" mode.

🋐 Rack_Assm, A, 1, , Preliminary (logged in as admin, databa	ase: InnovatorSolution	s1112)	
S				
🔛 Rack_Assm, A, 1, 🖻	Center graph			
	Reframe On			
R	Hide/Show			
	Properties	Alt+Enter		
	Open Sub-Tree	ALC: LINC		
	Cut.	Ctrl+X		
66 	Copy	Ctdl+C		
4] <u>C</u> opy	Ctri+C		
	Paste	Ctrl+V		
	<u>D</u> elete	Del		
	Rack_Assm, A, 1, , Prelim	inary object 🔹 🕨		
	E <u>x</u> pand	•	<u>C</u> AD Sub-Structure	1
	Expand <u>M</u> ultiple Levels	•	CAD Sub-Structure (Current)	
	De <u>-</u> Expand		CAD Sub-Structure (Released)	
	Open in new PDM Winde	DW	Uses Values from Design Table	
1	<u>U</u> nclaim		Is attached by	7
TE			Used in Structure	l l.
1	Unclaim all		 D <u>e</u> rives Drawing	x \y

Picture 27: Three CAD structure expand modes

In an example, a CATPart in a structure exists in two generations, generation 1 and generation 2. Generation 1 of the model has no whole (see *Picture 28: Generation 1 of CATPart*).



Picture 28: Generation 1 of CATPart

The structure in PDM uses the generation 1 of the CAD Document (see *Picture 29: CAD structure containing generation 1 of CATPart*).



Picture 29: 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 you attempt that, you get a warning (see *Picture 30: Warning about different expand resolution*).



Picture 30: 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" (see *Picture 31: CAD structure expanded as "Current"*).



Picture 31: CAD structure expanded as "Current"

The "Current" structure, which contains the latest generations of all CAD Documents, can be loaded into the CATIA session (see *Picture 32: CATIA structure containing the latest generations of the CATIA documents*).



Picture 32: CATIA structure containing the latest generations of the CATIA documents

When you update a CATIA structure that was not expanded as saved a confirmation dialog appears (see *Picture 33: Confirm "Update" action*). If you continue the update, all the CAD structure relations in the loaded structure will be updated to the current generations of the documents.



Picture 33: Confirm "Update" action

Expand Multiple Levels

It is also possible to expand a relation direction in multiple levels from the selected object.

Select the object from which to be expanded 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 the direction "Uses Part BOM" for the selected "Assembly" object "Engine" is selected (see *Picture 34: Action "Expand Multiple Levels"*).

🛐 Engine, A, 1, , Preliminary	(logged in as admin, datab	ase: InnovatorSolu	tions1112)	- • •
Ö Engino A 1	Droliminant			
Tigine, A, I	C <u>e</u> nter graph			
	<u>R</u> eframe On			
	Hide/Show			
	Pr <u>o</u> perties	Alt+Enter		
	Ope <u>n</u> Sub-Tree			
A CONTRACTOR OF A CONTRACTOR A	Cu <u>t</u>	Ctrl+X		
l i		Ctrl+C		
l C	<u>P</u> aste	Ctrl+V		
	Paste <u>Special</u>			
	<u>D</u> elete	Del		
	Engine, A, 1, , Preliminar	ry object 🔹 🕨		
	E <u>x</u> pand	•		
	Expand Multiple Levels	•	<u>U</u> ses Part BOM	z
	De_Expand		Is Used in Part BOM	×
	Compare with Assembly	,		,

Picture 34: Action "Expand Multiple Levels"

The related objects in multiple levels are shown for the selected "Assembly" object "Engine" (see *Picture 35: Result of expand multiple levels*).

In the PDM 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 PDM structure tree.



Picture 35: Result of expand multiple levels

Explicit Display and Control of "AsSaved" and "Current" Expand Resolutions

If a CAD structure is expanded with the "Current" expand resolution the PDM structure window contains information about which generations of the CAD Documents are saved (which generation would be retrieved with the "AsSaved" expand resolution).

Also, in the "Update" process the user can define whether the CAD Structure relations should be moved to the current generation of the child CAD Document or not.

Expanding a CAD structure as "Current" where some of the nodes are saved in an older generation shows which generation of the CAD Document is saved:



Picture 36: CAD Structure expanded as "Current"


As a comparison, this is the same structure expanded "AsSaved":

Picture 37: CAD Structure expanded "AsSaved"

If a CAD structure has been expanded as "Current" then in the "Update to PDM" dialog the user has the choice whether to update the CAD Structure relations to the current CAD Document generation. The default is "Yes". If the structure is not changed then the user can change this to "No", thereby keeping the CAD Structure relations pointing to the old CAD Document generations:

File name CATIA Part Number PDM Display Locked By PLM State File Status Operation Result Rack. CATProduct Rack Rack, A, 1, Preliminary (not all child no pwbuser! Preliminary Modified Update : 1. Update to current "Cste: A.1 ExcedParts. CATProduct FixedParts FixedParts, A, 1, Preliminary (not all chi pwbuser! Preliminary Modified Update : 1. Update to current "CylinderB	pdate to PDM?							
	ile name ack.CATProduct ixedParts.CATProduct	CATIA Part Number Rack FixedParts	PDM Display Rack, A, J., Preliminary (not all child no FixedParts, A, J., Preliminary (not all chi	Locked By pwbuser1 pwbuser1	PLM State Preliminary Preliminary	File Status Modified Modified	Operation Update : 1. Update to current "Case: A.1 Update : 1. Update to current "CylinderB	Result
aste new CAD document consistion?			Undate adding to Concel?	1				

Picture 38: "Update to PDM" dialog with "Update to Current" information

De-Expand

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

In *Picture 39: PDM structure before the De-Expand* you can see the PDM structure before the de-expand.



Picture 39: PDM structure before the De-Expand

First you select the root element of the substructure 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 you select the object "FixedParts_Assm" as root of the substructure to be de-expanded (see *Picture 40: Action "De-Expand"*).

🛐 Engine, A, 1, , Preliminary (logged in as admin, o	atabase: InnovatorSolutions1112)	- • •
Engine, A, 1, , Preliminary		
ConnectionRod.1 /	C <u>e</u> nter graph	
	<u>R</u> eframe On	
	Hide/Show	I, Preliminary
Crankshaft Assm.1	Properties A	Alt+Enter V
	Ope <u>n</u> Sub-Tree	~
CrankPin.1 / Cra	Cuţ	Ctrl+X
	<u>С</u> ору	Ctrl+C
	Paste	Ctrl+V
CrankshaftLeft.1	Paste <u>Special</u>	
CrankshaftLe	<u>D</u> elete	Del . 1 Preliminary
	ConnectionRod, A, 1, , Preliminary of	ject
	Expand	
CrankshaftRiv	Expand Multiple Levels	l, A, 1, , Prelimin،
	De_Expand	í k

Picture 40: Action "De-Expand"

In *Picture 41: PDM structure after the De-Expand* you see that the selected substructure 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 substructure will not be deleted from the PDM structure.



Picture 41: 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 42: Action "Properties"*).



Picture 42: Action "Properties"

The "Properties" dialog will be opened (see *Picture 43: "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		? ×
Current selection :	Engine, A, 1, , Preliminary	
Properties Up	date Item	
Assembly	·	
Part Number	Engine	
Major Rev.	A	
Generation	1	
Name		
State	Preliminary	
Unit		
Make / Buy		
Description		
		0.0000000000000000000000000000000000000
New Version		
Туре	Assembly	
Assigned Creator		
Designated User		
Created by	Innovator Admin	
Created on	2018-03-19T10:47:35	
Modified by	Innovator Admin	
Modified on	2018-03-19T10:47:36	
Locked by	Innovator Admin	
Release Date		
Effective Date		
		More
	OK 1 Anthe	1 Class
1000 C		Close

Picture 43: "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 44: "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	and the second se	? X
Current selectio	n : Engine, A, 1, , Preliminary	-
Properties	Update Item	
Assembly -	-	
Part Number	Engine *	
Major Rev.	A	
Generation	1	
Name		
Unit	▼	
Make / Buy		
Description		
se Médisononno		
Туре	Assembly	
		More
	OK Apply	Close

Picture 44: "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 to PDM" 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.

Claim

Claim Part in PDM Workbench Window

You have to claim 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 "Claim" (see *Picture 45: Action "Claim"*). 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.

S Engine, A, 1, , Preliminary (logged in a:	8	Hide/Show		300)
Engine, A, 1, , Prelimina	:	Properties	Alt+Enter	
↓ Crankshaft_Assm, A		Ope <u>n</u> Sub-Tree		
CrankshaftLeft, /	y,	Cut	Ctrl+X	
CrankshaftLe		<u>P</u> aste	Ctrl+C	
CrankshaftRight	_	Paste <u>S</u> pecial		
CrankshaftRi		<u>D</u> elete	Del	
CrankPin, A, 1, ,		Engine, A, 1, , Prelimina	ry object 🕨	
		E <u>x</u> pand	•	
FixedParts_Assm, A,		Expand Multiple Levels	,	
Rack_Assm, A, 1		Compare with Assembl	у	
🖕 - 🏠 Screw, A, 1, ,	1	Open <u>i</u> n new PDM Wine	dow	
ScrewMo	T	<u>U</u> nclaim		z
Rack, A. 1, . 1	F	Claim		х⁄у

Picture 45: Action "Claim"

The selected object and the attached documents will be claimed by you (see *Picture 46: Object is claimed*).

2DM Workbench	?	\times
Information		
PDM Messages:		
Claimed PDM object 'Engine (CAD/Mechanical/Assembly)'. Claimed PDM object 'Engine (Part/Assembly)'.		
Action succeeded.		
	C	lose

Picture 46: Object is claimed

The background color of the claimed object changed to green in the PDM Workbench window (see *Picture 47: Claimed object*).



Picture 47: Claimed object

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

Claim Object in Query Dialog

It is also possible to claim the object from the query result list of the "PDM Query" dialog. In the example you select the object and click on "Claim" in the context menu (see *Picture 48: Action "Claim" in the query result list*). 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.

PDM Query (logged in as pwbuser1, database: InnovatorSolutions2300 PDM object type Part Numbe Major Rev Generation Name State ConnectionRod Preliminary Part 🔅 ConnectionRodMode Preliminary CrankPin Preliminary PDM object attributes CrankPinModel Crankshaft_Assm CrankshaftLeft CrankshaftLeft Preliminary Preliminary Part Number C Maior Rev. Preliminary Preliminar Generation CrankshaftRight Preliminary Name 🔅 CrankshaftRightl Preliminar State 00 CylinderBlock 📷 Open in new PDM Windo reliminar Unit Preliminary Make / Buy Descriptio Promote 🐻 Load * Created on Modified on 5 Create new Created by 🟒 Duplicate Modified by 😫 Duplicate Structure Download CATDrawings related to BOM Part Open in Aras Query Close Expand

A multi-select of objects is also supported.

Picture 48: Action "Claim" in the query result list

Claim Document in CATIA V5 Window

You have to claim 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*-*Claim* (see *Picture 49: Action "Claim" in the CATIA V5 window*).



Picture 49: Action "Claim" in the CATIA V5 window

For further details of the "Claim" dialogs please refer to the chapter *Claim Part in PDM Workbench Window*.

The claim 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 "Claim" button of the "PDM Workbench context commands" toolbar. The icons in this toolbar are only repainted (e.g. switch from "Claim" to "Unclaim") when you newly activate the CATIA V5 window.

A multi-select of objects is also supported.

Unclaim

Unclaim Part in PDM Workbench Window

When an object is claimed by you then you have to unclaim 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 "Unclaim" context action (see *Picture 50: Action "Unclaim"*). 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.



Picture 50: Action "Unclaim"

The selected object and the attached documents will be unclaimed by you (see *Picture 51: Object is unclaimed*).

PDM Workbench	?	×
[Information		
PDM Messages:		
Unclaimed PDM object 'Engine (CAD/Mechanical/Assembly)'. Unclaimed PDM object 'Engine (Part/Assembly)'		
one dimension of general signification of the second of th		
Action succeeded.		
	C	ose

Picture 51: Object is unclaimed

The background color of the unclaimed object changed to blank in the PDM Workbench window (see *Picture 52: Unclaimed object*).



Picture 52: Unclaimed object

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

Unclaim Object in Query Dialog

It is also possible to unclaim the object from the query result list of the "PDM Query" dialog.

In the example you select the object and click on "Unclaim" in the context menu (see *Picture 53: Action "Unclaim" in the query result list*). 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 (Ic	ogged in as pwbuser1, database: Inno	ovatorSo	lutions	s2300)						? X
PDM object ty	уре			Part Number		Major Rev.	Ge	neration	Name	State
Part			J 🗘	ConnectionRod		A	1			Preliminary
				ConnectionRodModel		A	1			Preliminary
PDM object a	ttributos		0	CrankPin		A	1			Preliminary
PDIVI ODJECI a	l'indutes	-	<u></u>	CrankPinModel		A	1			Preliminary
Part Number	C* ~	í í	¢	Crankshaft_Assm		A	1			Preliminary
Major Rev.	~		0	CrankshaftLeft		A	1			Preliminary
Generation		-	2	CrankshaftLeftModel		A	1			Preliminary
		-	0	CrankshaftRight		A	1			Preliminary
Name	~	100000		CrankshaftRightMode		A	1			Preliminary
State	~		0	Cylinder_Assm			1			Preliminary
Unit			Q	CylinderBlock	1	Open in new PDM Window	v			Preliminary
Males (Den		-	2	CylinderBlockModel	13	Unclaim				Preliminary
Wake / buy	~				1					
Description					1					
					7					
					1.0					
					1+2					
					1	Load *				
Created on	~	100000			1.14					
Modified on	~				7.60					
Created by					÷	Create new generation				
Manalesala		-			1	Duplicate				
modified by	~				\rightarrow	. Dupileate				
					1	Duplicate Structure				
					A	Onen in Aras		i Do	ownload CATDrawings	related to BOM Par
					-11	openning			Out	erv Close
Contract of Contract					* *	Expand				ciose

Picture 53: Action "Unclaim" in the query result list

Unclaim Document in CATIA V5 Window

When an object is claimed by you then you can unclaim 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 *Unclaim* (see *Picture 54: Action "Unclaim" in the CATIA V5 window*).



Picture 54: Action "Unclaim" in the CATIA V5 window

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

The unclaim 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 "Unclaim" button within the "PDM Workbench context commands" toolbar. The icons in this toolbar are only repainted (e.g. switch from "Claim" to "Unclaim") when you newly activate the CATIA V5 window.

A multi-select of objects is also supported.

Claim All

Claim All Parts in PDM Workbench Window

It is possible to claim the selected object and the objects in its substructure.

Select the object and right-click to open the context menu. Choose "Claim All" (see *Picture 55: Action "Claim All"*).



Picture 55: Action "Claim All"

You have to confirm the "Claim All" action of the objects (see *Picture 56: Confirm the "Claim All" action*).

Claim PD	M Objects?	\times
?	Are you sure you want to claim Crankshaft_Assm, A, 1, , Preliminary CrankshaftLeft, A, 1, , Preliminary CrankshaftLeftModel, A, 1, , Preliminary CrankshaftRightModel, A, 1, , Preliminary CrankPin, A, 1, , Preliminary CrankPinModel, A, 1, , Preliminary?	
	Ja Nein	

Picture 56: Confirm the "Claim All" action

The objects and the attached documents will be claimed by you (see *Picture 57: Objects are claimed*).

Information		
Claimed PDM object 'CrankshaftLeftModel (Part/Component)'.		
Claimed PDM object 'CrankshaftRight (CAD/Mechanical/Assembly)'.		
Claimed PDM object 'CrankshaftRight (Part/Assembly)'.		
Claimed PDM object 'CrankshaftRightModel (CAD/Mechanical/Part)'.		
Claimed PDM object 'CrankshaftRightModel (Part/Component)'.		
Claimed PDM object 'CrankPin (CAD/Mechanical/Assembly)'.		
Claimed PDM object 'CrankPin (Part/Assembly)'.		
Claimed PDM object 'CrankPinModel (CAD/Mechanical/Part)'.		
Claimed PDM object 'CrankPinModel (Part/Component)'.		
 Action succeeded. 		
	[mms	



The background color of the claimed objects changed to green in the PDM Workbench window (see *Picture 58: Claimed objects*).



Picture 58: Claimed objects

Claim All Documents in CATIA V5 Window

It is possible to claim the selected object and the objects in its substructure.

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*-Claim All (see Picture 59: Action "Claim All" in the CATIA V5 window).



Picture 59: Action "Claim All" in the CATIA V5 window

For further details of the "Claim" dialogs please refer to the chapter *Claim All Parts in PDM Workbench Window*.

A multi-select of objects is also supported.

Unclaim All

Unclaim All Parts in PDM Workbench Window

It is possible to unclaim the selected object and the objects in its substructure.

Select the object and right-click to open the context menu. Choose "Unclaim All" (see *Picture 60: Action "Unclaim All"*).

S Engine, A, 1, , Preliminary (logged in as pwbuser1, dat	8	Hide/Show
Engine, A, 1, , Preliminary		Properties Alt+Enter
Crankshaft_Assm, A, 1, , Prelim		Ope <u>n</u> Sub-Tree
CrankshaftLeft, A, 1, , Prelir	y,	Cut Ctrl+X
CrankshaftLeftModel, A		Paste Ctrl+V
CrankshaftRight, A, 1, , Prel		Paste Special
CrankshaftRightModel,		Delete Del
CrankPin, A, 1, , Preliminary		Crankshaft_Assm, A, 1, , Preliminary (claimed by pwbuser1) object •
CrankPinModel, A, 1, , F		Expand •
FixedParts_Assm, A, 1, , Prelim	1	Expand Multiple Levels
Rack_Assm, A, 1, , Prelimin		Compare with Assembly
Screw, A, 1, , Preliminar	1	Open <u>i</u> n new PDM Window
ScrewModel, A, 1, , I	F	Unclaim
Rack, A, 1, , Preliminary		Claim
		Claim all

Picture 60: Action "Unclaim All"

You have to confirm the "Unclaim All" action of the objects (see *Picture 61: Confirm the "Unclaim All" action*).



Picture 61: Confirm the "Unclaim All" action

The objects and the attached documents will be unclaimed by you (see *Picture 62: Objects are unclaimed*).



Picture 62: Objects are unclaimed

The background color of the unclaimed objects changed to blank in the PDM Workbench window (see *Picture 63: Unclaimed objects*).

Tengine, A, 1, , Preliminary (logged in as pwbuser1, database: InnovatorSolutions2300)	- I X
Engine, A, 1, , Preliminary	
🖕 🖕 🏠 CrankshaftLeft, A, 1, , Preliminary	
CrankshaftLeftModel, A, 1, , Preliminary	
- CrankshaftRight, A, 1, , Preliminary	
CrankshaftRightModel, A, 1, , Preliminary	
- 🔆 CrankPin, A, 1, , Preliminary	
CrankPinModel, A, 1, , Preliminary	
- TixedParts_Assm, A, 1, , Preliminary	
- 🛱 Rack_Assm, A, 1, , Preliminary	
Screw, A, 1, , Preliminary	
ScrewModel, A, 1, , Preliminary	
Rack, A, 1, , Preliminary	z
DockModel A. 1. Droliningny	x∕y ►

Picture 63: Unclaimed objects

Unclaim All Documents in CATIA V5 Window

When an object is claimed by you then you can unclaim it in the PDM system to make it available for all other users. It is possible to unclaim the selected object and the objects in its substructure.

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 *Unclaim All* (see *Picture 64: Action "Unclaim All" in the CATIA V5 window*).



Picture 64: Action "Unclaim All" in the CATIA V5 window

For further details of the "Unclaim All" dialogs please refer to the chapter Unclaim All Parts in PDM Workbench Window.

A multi-select of objects is also supported.

Warning when the User wants to unclaim modified Files

There is a functionality which warns the user when he is about to unclaim CATIA files that are modified in the session:



Picture 65: Warning Dialog at Unclaim

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 66: Action "Promote"*).

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

🛐 Engine, A, 1, , Preliminary (logged i	n as admin, database: Innov	vatorSolutions11	12) 🗖 🗖 💌
A 244			
Engine, A, 1, , F	C <u>e</u> nter graph		
Connection	<u>R</u> eframe On		. Preliminary
	Hide/Show		
Connect 😭	Pr <u>o</u> perties	Alt+Enter	RodModel, A, 1, , Preliminary
-Ö Crankshaft	Ope <u>n</u> Sub-Tree		1 Preliminary
	Cu <u>t</u>	Ctrl +X	r,, rrenning
📕 🗧 🖓 CrankPil	<u>C</u> opy	Ctrl+C	ry
	<u>P</u> aste	Ctrl+V	A 1 Droliminan
	Paste <u>Special</u>		, A, 1, , Pleiminary
📕 🍦 📿 Cranksh	<u>D</u> elete	Del	, Preliminary
	Engine, A, 1, , Preliminary	object 🕨	tleftModel A 1 Preliminan
	Evnand		tertimodel, A, 1, , i reininary
📕 🗢 🖓 Cranksh	Expand Expand Multiple Levels		λ, 1, , Preliminary
	De-Expand		aftDightModel A 1 Drolimin
	Compare with Assembly		
🖓 Cylinder_A	Open in new PDM Window	v	reliminary
	Claim		
📃 🛛 🖵 🏹 Pisto 🔼	Promote		, , Preliminary
1/2	Re <u>v</u> ise		
PistonPi	Load		, 1, , Preliminary
	Load <u>w</u> ith Links		hínarv 🕨

Picture 66: Action "Promote"

You will be asked if you really want to promote the objects. Please confirm with "Yes" (see *Picture 67: Confirm the "Promote" action*).



Picture 67: 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 68: Object is promoted*).



Picture 68: 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 69: Action "Revise"*).

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

🛐 Engine, A, 1, , Released (logged in as admin,	database: InnovatorSolutions1112)	- • •
A 34e		
Engine, A, 1, , Release	C <u>e</u> nter graph	
-3 ConnectionRod.1	<u>R</u> eframe On	minary
	<u>H</u> ide/Show	
ConnectionRoc	Properties Alt+Enter	lodel, A, 1, , Preliminary
Crankshaft Assm 1	Ope <u>n</u> Sub-Tree	eliminary
	Cu <u>t</u> Ctrl+X	
CrankPin.1 / Cr.	<u>C</u> opy Ctrl+C	
	Paste Ctrl+V	Preliminary
	Paste Special	, i reiminary
CrankshaftLeft.:	<u>D</u> elete Del	iminary
CrankshaftL	Engine, A, 1, , Released object	Aodel, A, 1, , Preliminary
CrankshaftRigh	Expand	Preliminary
	De-Expand	uhtModol A 1 Prolimina
	Compare with Assembly	intmodel, A, I, , Freimina
📮 📮 Cylinder_Assm.1 / 🛱	 Open in new PDM Window	inary
	<u>U</u> nclaim	
Piston.1 / Pisto		
PistonMode	Promote	iminary
- O DictonDin Accor	>2 Re <u>v</u> ise	roliminan 7
PistonPin_Assm	Load	
	Load <u>w</u> ith Links	

Picture 69: Action "Revise"

You will be asked if you really want to revise the object. Please confirm with "Yes" (see *Picture 70: Confirm the "Revise" action*).

Engine, A, 1, , Released (logged in as admin, database: InnovatorSolutions1112)	
Engine, A, 1, , Released	
- ConnectionRod.1 / ConnectionRod, A, 1, , Preliminary	
ConnectionRodModel.1 / ConnectionRodModel, A, 1, , F	Preliminary
- Crankshaft_Assm.1 / Crankshaft_Assm, A, 1, , Preliminary	
↓ ↓ CrankPin.1 / CrankPin, A, 1, , Preliminary	
CrankPin Revise PDM Object?	
T - CrankshaftLe Are you sure you want to revise inary	
Engine, A, 1, , Released? Idel, A, 1, , I	Preliminary
eliminary	
Cranksha	1, , Prelimina
Cylinder_Assm.1 / Cylinder_Assm, A, 1, , Preliminary	
Piston.1 / Piston, A, 1, , Preliminary	
PistonModel.1 / PistonModel, A, 1, , Preliminary	
PistonPin_Assm.1 / PistonPin_Assm, A, 1, , Preliminary	F
	x∕~y ►

Picture 70: Confirm the "Revise" action

The selected object will be revised (see Picture 71: Object is revised).

CATProduct, Assembly (logged in as admin, database: InnovatorSolutions1112)	
 Engine, B, 2, , Preliminary Engine, B, 2, , Preliminary Engine, A, 1, , Released (not current) 	
PDM Workbench	? ×
Information	
Revised PDM object 'Engine (Mechanical/Assembly)'. Revised PDM object 'Engine (Assembly)'.	
Action succeeded.	
	Close
Cranksnaukignut / Cranksnaukignu, A, 1, , rienninary	
🚽 🖓 CrankshaftRightModel.1 / CrankshaftRightModel, A, 1, , Prelimina	
Cylinder_Assm.1 / Cylinder_Assm, A, 1, , Preliminary	
Piston. 1 / Piston, A, 1, , Preliminary	

Picture 71: 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 72: Action "Update structure relations"*).

S C	ATProduct (logged in as admin,	database: InnovatorSolutio	ons1112)		
	FixedParts Assm. R	2 Preliminary			
		2, , ricinfindity			
Ĩ	Engine, A, 1, , Prom	C <u>e</u> nter graph			
l T	Connection R	<u>R</u> eframe On			ConnectionRod, A, 1, , Preliminary
	🗆 🖵 🎦 Connecti 🦉	Hide/Show			ructure / ConnectionRodModel, A, 1, , Prelimir
	- 🗟 Crankshaft A 🚍	Properties	Alt+Enter		Crankshaft Assm A 1 Proliminary
		Open Sub-Tree	Ctrl+X		Crankshart_Assin, A, 1, , Freiminary
	CrankPin.	<u>С</u> ору	Ctrl+C		nkPin, A, 1, , Preliminary
	🔄 🖵 🛐 Crank 🛃	Paste	Ctrl+V		ure / CrankPinModel, A, 1, , Preliminary
	🗧 🚮 Cranksha				/ CrankshaftLeft, A, 1, , Preliminary
	Crank	<u>D</u> elete	Del		Structure / CrankshaftLeftModel, A, 1, , Prelim
		Engine, A, 1, , Preliminary	object	•	e / CrankshaftRight, A, 1, , Preliminary
	L Crank	Expand		٠	VStructure / CrapkshaftPightModel A 1 Pro
HI		Expand Multiple Levels		•	
Ī	Cylinder_Assi 📔	De_Expand			vlinder_Assm, A, 1, , Preliminary
	🗧 🗧 🔯 Piston.1, 📲	Unclaim	vv		A, 1, , Preliminary
	🛛 🖵 🌆 Piston 🚾	Claim			e / PistonModel, A, 1, , Preliminary
	🚽 🗿 Piston Pin 🚊	Unclaim all			/ PistonPin_Assm, A, 1, , Preliminary
	- Biston Z	Claim all Promote			ure / PistonPincomp. A. 1 Preliminary
		Update structure relations	;		Jre/Structure / PistonPincompModel, A, 1, , Pr
	The second se	Update parent relation			SnapRing, A, 1, , Preliminary
	🛛 🖵 🚳 Sn 陸	Load			ructure / SnapRingModel, A, 1, , Preliminary
V		Load with Links			SpapRing A 1 Preliminary
		Load in Context			

Picture 72: 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 73: Structure relations*).





Update Parent Relation

This functionality is only available for the CAD Document Structure Data Model.

Like "Update structure relations", this functionality updates the structure relations of a used document to the latest generation 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 PDM structure, not for the root node (see *Picture 74: Action "Update parent relation"*).

N Engine, A, 1, , Preliminary (logged in as ad	min, database: InnovatorSolutions	1112)	
Engine, A, 1, , Preliminary			
FixedParts_Assm.1, /CAF) Structure/Structure / F	ixedParts Ass	<mark>m, B, 2, , Preliminary</mark>
+- (a) Connection Rod 1 /C4	C <u>e</u> nter graph		A 1 Preliminary
	<u>R</u> eframe On		r, i, i remandry
👎 🔞 Crankshaft_Assm.1, /(🖉	<u>H</u> ide/Show		m, A, 1, , Preliminary
🛓 🖏 Cylinder Assm.1. /CAI 🚆	Pr <u>o</u> perties	Alt+Enter	. 1 Preliminary
	Ope <u>n</u> Sub-Tree		, _, ,
	, Cut	Ctrl+X	
	<u>⊡</u> opy	Ctrl+C	
4	Paste	Ctri+v	
_	Papte 2pecia		
	<u>D</u> elete	Del	
	<u>FixedParts_Assm</u> , B, 2, , Prelimin	ary object 🔹 🕨	
	E <u>x</u> pand	•	
	Expand Multiple Levels	•	
	De <u>-</u> Expand		
	Open in new PDM Window		
1	Unclaim		
1	Unclaim all		
	Claim all		
2	9 Pro <u>m</u> ote		
1	2 Revise		
	Update structure relations		
	Update parent relation		
	Load		7
	Load with Links		x
	Select Nodes		

Picture 74: Action "Update parent relation"

You have to select the required generation (see *Picture 75: Action "Update parent relation" – select version*).

Update parent relation
Available versions
OK Apply Cancel

Picture 75: Action "Update parent relation" – select version

The relation has been updated (see Picture 76: Parent relation is updated).



Picture 76: Parent relation is updated

The structure relations will have to be expanded again to show the current status (see *Picture 77: Updated structure relation*):

🛐 Engine, A, 1, , Preliminary (logged in as admin, database: InnovatorSolutions1112)
FixedParts_Assm.1, /CAD Structure/Structure / FixedParts_Assm, A, 1, , Released (not current)
ConnectionRod.1, /CAD Structure/Structure / ConnectionRod, A, 1, , Preliminary
🛨 📆 Crankshaft_Assm.1, /CAD Structure/Structure / Crankshaft_Assm, A, 1, , Preliminary
🛨 📆 Cylinder_Assm.1, /CAD Structure/Structure / Cylinder_Assm, A, 1, , Preliminary
Cylinder_Assm.1, /CAD Structure/Structure / Cylinder_Assm, A, 1, , Preliminary

Picture 77: 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 78: Action "Delete"*).

S Component, Assembly (logged in as add	min, database: InnovatorSo	olutions1112)	
C NewPart1, A, 1, , Prelin	C <u>e</u> nter graph		
C NewPart2, A, 1, , Prelin	Reframe On Hide/Show		
NewProduct1, A, 1, , P	Properties	Alt+Enter	
NewProduct2, A, 1, , P	E Ope <u>n</u> Sub-Tree	Ctrl+X	
RewPart3, A, 1, , Prelin	💼 Сору	Ctrl+C	
	Paste Paste Special	Ctrl+V	z
	<u>D</u> elete	Del	х∕у

Picture 78: Action "Delete"

A confirmation message box is displayed listing the objects which will be deleted (see *Picture 79: Confirm the "*Delete" action). When you confirm this dialog with "Yes" the objects will be deleted.

Component, Assembly (I	ogged in as admin, database: InnovatorSolutions1112)	
 NewPart1, A NewPart2, A NewProduct NewProduct NewProduct NewPart3, A 	Delete PDM Objects?	
	Ja Nein	x y

Picture 79: Confirm the "Delete" action

The operation result dialog is displayed containing error or success messages (see *Picture 80: Objects are deleted*).

PDM Workbench	? X
Information	
Deleted PDM object NewPartl (Mechanical/Part). Deleted PDM object NewPartl (Mechanical/Part). Deleted PDM object NewProduct1 (Assembly). Deleted PDM object NewProduct1 (Assembly). Deleted PDM object NewProduct2 (Assembly). Deleted PDM object NewProduct2 (Assembly). Deleted PDM object NewProduct2 (Component). Deleted PDM object NewPart2 (Component). Deleted PDM object NewPart2 (Mechanical/Part).	E
Action succeeded.	
	Close

Picture 80: Objects are deleted

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

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 81: Action "Create new version"*).



Picture 81: Action "Create new version"

Delete newest Version

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

You have to select the last generation of the document and click on the right mouse button. The context menu will be opened. There you select "Delete newest version" (see *Picture 82: Action "Delete newest version"*).

ScrewModel, A, 1, , Preliminary (logged in as admin, data	: InnovatorSoluti	ons1112)	
ScrewModel, A, 1, , Preliminary			
Attaches / Screwiviodel, A, Z, ,	C <u>e</u> nter graph		
	<u>R</u> eframe On		
	<u>H</u> ide/Show		
	Pr <u>o</u> perties	Alt+Enter	
	Open Sub-Tre	e	
	🖌 Cut	Ctrl+X	
		Ctrl+C	
	Paste	Ctrl+V	
	Paste <u>S</u> pecial.		
	<u>D</u> elete	Del	
	Scre <u>w</u> Model,	A, 2, , Preliminary object	•
	Expand		•
	Exp <u>a</u> nd Multi	ple Levels	•
	De <u>-</u> Expand		
	Dpen <u>i</u> n new	PDM Window	
	☑ <u>U</u> nclaim		
	🔊 Pro <u>m</u> ote		
	2 Revise		
	B Open <u>F</u> ile		
	Open File ten	nporaril <u>y</u>	
	Highlight CA	TIA Nodes	
	Delete relation	n	z
	🗙 Delete newes	t version	

Picture 82: Action "Delete newest version"

Then you are asked to confirm the delete of the newest generation. You have to click the "Yes" button (see *Picture 83: Confirm the "Delete* newest version).

ScrewModel, A, 1, , Preliminary (logged in as admin, database: InnovatorSolutions1112)	×
ScrewModel, A, 1, , Preliminary Attaches / ScrewModel, A, 2, , Preliminary	
Delete newest file of PDM Object?	
Ja Nein	

Picture 83: Confirm the "Delete newest version" action

The newest generation will be deleted. The document will be removed from the window (see *Picture 84: Newest generation object is deleted*).

	ScrewModel, A, 1, , Preliminary (logged in as admin, database: InnovatorSolutions1112) ScrewModel, A, 1, , Preliminary ScrewModel, A, 1, , Preliminary	
PDM Workb	nch	? ×
PDM Mes	ages: M object ScrewModel (Mechanical/Part).	
A 🕑	ion succeeded.	Close

Picture 84: Newest generation object is deleted

You have to re-expand the document in order to see the document that is attached to the component (see *Picture 85: Re-Expand of the document*).

ScrewModel, A, 1, , Preliminary (logged in as admin, database: InnovatorSolutions1112)	- • •
ScrewModel, A, 1, , Preliminary Attaches / ScrewModel, A, 1, , Preliminary	

Picture 85: Re-Expand of the document

Only one new Generation of a CAD Document per "Claim" Action

It can be configured what only one new generation of a CAD Document will be created for a claimed document. This new generation will be created at the first update after the claim. Further updates will overwrite the newly created generation. If a new generation of the CAD Document should be created explicitly then the user has to unclaim the CAD Document and claim it again before performing the next update.

CAD Document generations which are read-only, for example because they are released or frozen can be claimed if they are current. In this case the new generation will be created by the "Claim" process, and the first update will not create another new generation.

If an already claimed CAD Document becomes read-only later, then a new generation of the CAD Document will be created at update, since the claimed generation cannot be overwritten.

Unlink and Delete newest Version

This function is a combination of "delete Part CAD relation" and "delete newest CAD version".

You have to select the last generation of the document and click on the right mouse button. The context menu will be opened. There you select "Unlink and delete newest version" (see *Picture 86: Action "Unlink and delete newest version"*).

NewPart1, A, 1, , Preliminary (logged in as admin, database	: Inno	vatorSolutions1112)		- • ×
NewPart1, A, 1, , Preimmary				
Attaches / NewPart1, A, 2, , Pre	1	Center graph		
		Reframe On		
		Hide/Show		
		Properties	Alt+Enter	
		Open Sub-Tree	Alternation	
		Cut	Ctrl+X	
		Copy	Ctrl+C	
		Darte	Ctrl+V	
	•	Paste Special	Cui+v	
	_			-
		<u>D</u> elete	Del	
		Ne <u>w</u> Part1, A, 2, , Prelimin	nary object 🕨	
		Expand	,	-
		Expand Multiple Levels	•	
	1	De <u>-</u> Expand		
	1	Open in new PDM Wind	ow	
	7	<u>U</u> nclaim		
	"E			
	7	Pro <u>m</u> ote		
	1+2	Regise		
		Open <u>F</u> ile		
	N	Open File temporarily		
	5	Highlight CATIA Nodes		
	.	Delete relation		7
	-	Delete newest version		Ì
	2	Unlin <u>k</u> and delete newes	t version	* ~y

Picture 86: Action "Unlink and delete newest version"

Then you are asked to confirm the delete of the newest generation. You have to click the "Yes" button (see *Picture 87: Confirm the "Unlink and delete newest version" action*).



Picture 87: Confirm the "Unlink and delete newest version" action

The newest generation will be unlinked and deleted. The document will be removed from the window (see *Picture 88: Newest generation object is unlinked and deleted*).



Picture 88: Newest generation object is unlinked and deleted

No document is related to the part. You can continue creating new geometry for it.

Open File

This functionality is only available for the CAD Document Structure Data Model.

You can open a single CATIA V5 Drawing file existing in the PDM database 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 database 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 89: Action "Open File"*).



Picture 89: 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 90: Open File – progress bar*).

Open File	×
	Downloading selected file object from PDM system
	Downloading CAD files
Status :	
Estimated time remain	ning :
	Cancel

Picture 90: Open File – progress bar

The geometry opens in its corresponding CATIA V5 native window (see *Picture 91: 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 91: Split window after Open File – PDM Workbench node and CATIA drawing

Open File Temporary

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

In the first step you load the current generation of the file. In this example you open the generation "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 92: Action "Open File"*).



Picture 92: Action "Open File"



The current generation of the file is loaded in CATIA V5, now (see Picture 93: Current file).

Picture 93: Current file

Then you query for a different generation (in this case generation "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 94: Action "Open File Temporary"*).



Picture 94: Action "Open File Temporary"

The generation "1" of the file is opened temporarily in CATIA V5 (see *Picture 95: Temporarily 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 95: Temporarily opened file

Now you can compare the both generations of the file.

Load

It is possible to load geometry corresponding to an expanded PDM 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 96: Action "Load"*).

Engine, B, 2, , Preliminary (logg	jed in as admin, datab	ase: InnovatorSoluti	ons1112)
A 342			
Engine, B, 2,	C <u>e</u> nter graph		1
Connecti	<u>R</u> eframe On		1, Preliminary
	Hide/Show		
Conn E	Pr <u>o</u> perties	Alt+Enter	onRodModel, A, 1, , Preliminary
Cranksha	Ope <u>n</u> Sub-Tree		A, 1, , Preliminary
	Cut	Ctrl +X	
	Copy	Ctrl+C	nary
│ │ │ └-☆ <mark>त</mark> [@]	Paste	Ctrl+V	del. A. 1 Preliminary
- 242			- <u> </u>
Crank	<u>D</u> elete	Del	, 1, , Preliminary
📕 🖵 🏹 <mark>G</mark>	Engine, B, 2, , Prelimi	inary object	aftLeftModel, A, 1, , Preliminary
-Ö Cront	Expand	,	t A 1 Proliminand
	Expand Multiple Leve	els 🕨	, , , , , , Freinninary
📗 🔰 🦵 🔁 🚾 🔞	De_Expand		shaftRightModel, A, 1, , Prelimin
	Compare with Assen	nbly	Droliminan
	Open <u>i</u> n new PDM W	indow	, Freimmary
📕 🗧 🗭 Pistor 🏱	<u>U</u> nclaim		
			1 Droliminan
	Promote		, I, , Preliminary
📄 🕴 🗢 📿 Pistor 👼	z ne <u>v</u> oe		A, 1, , Preliminary
	Load with Links		up A 1 Proliminan
	Load in Context		
	Add Temp		anDincomoMpdel, A, 1, , Prelim

Picture 96: Action "Load"

The PDM Workbench downloads the corresponding CAD files (CATParts, CATProducts, and CATDrawings) to the client's PDM Workbench exchange directory. The download progress is indicated by the "Load" progress bar (see *Picture 97: Load - progress bar*).

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

Picture 97: Load - progress bar

The geometry downloaded opens in a CATIA V5 native window (see *Picture 98: Split window after Load – PDM Workbench and CATIA V5 nodes*).

In the left window (PDM Workbench window) you see the expanded PDM 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 98: Split window after Load – PDM Workbench and CATIA V5 nodes

Additional Options for "Loading PDM Structures"

If not the complete structure is loaded, that is, if parts of the structure are de-expanded or filtered out due to a specific configuration, the missing CATPart and CATProduct nodes can be displayed as deactivated, or as being in no-show. Conversely, nodes that are loaded can be set to active, or being in "show" mode. It is also possible to not change CATProduct or CATPart nodes with respect to their activate/deactivate and show/no-show status at all.

Depending on the way of working, different settings can make sense:

- Enable Activate/Deactivate: This option should be used if the designer works with Hide/Show. This way the Hide/Show status the designer explicitly sets will not be changed when the structure is loaded.
- Enable Hide/Show: This option should be used if the designer works with Activate/Deactivate.
- No Change: This option should be used if the designer works with both Activate/Deactivate and Hide/Show.
| PDM Workbench Options |
|---|
| Version
PDM Workbench CATIA V5 Module
Version: 9.0.0
CATIA Level: V5R28
(c) T-Systems 2005 - 2018 |
| Query Dialog |
| |
| PDM Relations |
| Loading related Files |
| Download CATDrawings related to BOM Part |
| Load with Links |
| Load with Drawing |
| Loading PDM Structures |
| Enable Activate/Deactivate |
| O Enable Hide/Show |
| No Change |
| Expand Neighborhood |
| 0 [mm] Clearance for Expand Neighborhood |
| Copy Position |
| Copy Position |
| Load With Links |
| Load current generation of linked Document |
| Load multiple levels of linked Documents |
| Open linked Documents in own Window |
| OK Gancel |

Picture 99: Additional options for "Loading PDM Structures"

Load Substructures in Context

BOM Part Structure Data Model

It is possible to load only selected nodes of a PDM structure to CATIA.

In the first step you expand a PDM structure in the PDM Structure window. Instead of loading the complete structure with "Load" you select any number of nodes in the tree and select the action "Load in Context" (see *Picture 100: Action "Load in Context" – with some structure nodes* selected).



Picture 100: Action "Load in Context" – with some structure nodes selected

You have to confirm the action (see Picture 101: Confirm the "Load in Context" action).



Picture 101: Confirm the "Load in Context" action

The selected objects are then loaded to CATIA (see *Picture 102: "Load in Context" – Selected objects loaded to CATIA*).



Picture 102: "Load in Context" - Selected objects loaded to CATIA

This amounts to the same as de-expanding all the unwanted parts of the structure and loading, but it is often faster and more convenient.

CAD Document Structure Data Model

It is possible to only load selected nodes of a PDM structure to CATIA.

In the first step you expand a PDM structure in the PDM Structure window. Instead of loading the complete structure with "Load" you select any number of nodes in the tree and select the action "Load in Context" (see *Picture 103: Action "Load in Context" – with some structure nodes selected*).



Picture 103: Action "Load in Context" – with some structure nodes selected

You have to confirm the action (see Picture 104: Confirm the "Load in Context" action).



Picture 104: Confirm the "Load in Context" action

The complete PDM structure gets reduced to only the sub-set of the structure which contains the selected nodes (see *Picture 105: "Load in Context" – Reduced PDM structure in PDM Structure window*).



Picture 105: "Load in Context" – Reduced PDM structure in PDM Structure window

This reduced structure is then loaded to CATIA (see *Picture 106: "Load in Context" – Reduced structure loaded to CATIA*).



Picture 106: "Load in Context" - Reduced structure loaded to CATIA

This amounts to the same as de-expanding all the unwanted parts of the structure and loading, but it is often faster and more convenient.

Add Temp

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

In the first step you load the current generation 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 107: Action "Load"*).

Engine, B, 2, , Preliminary (logged in as ac	lmin, database: Innov	vatorSolutions1112)	
A 244			
Engine, B, 2, , Prelir	C <u>e</u> nter graph		
- ConnectionRod	<u>R</u> eframe On		liminary
	Hide/Show		
Connection 🖆	Properties	Alt+Enter	Model, A, 1, , Preliminary
=-Ö Crankshaft Asst	Ope <u>n</u> Sub-Tree		Preliminan
	Cut	Ctrl+X	remininary
CrankPin.1 /] Сору	Ctrl+C	
	Paste	Ctrl+V	L. Dualing in an
C CrankPin			L, Preliminary
CrankshaftLe	<u>D</u> elete	Del	elim inary
Cranksha	Engine, B, 2, , Prelin	ninary object 🔹 🕨	Model, A, 1, , Preliminary
31 21 21 21 21 21 21 21 21 21 21 21 21 21	E <u>x</u> pand	•	
CranksnaπR	Expand Multiple Le	vels	, Preliminary
Cranksha	De_Expand		ightModel, A, 1, , Prelimin
A	Compare with Asse	mbly	
Cylinder_Assm.	Open in new PDM	Window	ninary
Piston,1 / Pi	Unclaim		
	- - -		
	- Revice		eliminary
PistonPin A	Load		Preliminary
	Load with Links		Z Z
	Load in Context		,, Preliminary x
	Add Temp		

Picture 107: Action "Load"

The current generation of the structure is loaded in CATIA V5, now (see *Picture 108: Loaded geometry for revision "B"*).



Picture 108: Loaded geometry for revision "B"

Then you query for a different generation (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 109: Action "Add Temp"*).

Engine, A, 1, , Released (not current) (logged in as	admin, database: InnovatorSolutio	ons1112)	- • •
A 244			
Engine, A, 1, , Released (C <u>e</u> nter graph		
-3 ConnectionRod 1 / C	Reframe On		
	Hide/Show		
ConnectionRodN	Properties	Alt+Enter	, Preliminary
Grankshaft Assm 1 /	Ope <u>n</u> Sub-Tree	Chilly	
		Ctrl+C	
CrankPin.1 / Cran	Daste	Ctrl+V	
	Paste Special	Curty	
			ary
$\left \begin{array}{c} \\ \\ \\ \\ \\ \end{array} \right \left \begin{array}{c} \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\$	<u>D</u> elete	Del	_
3te	Engine, A, 1, , Released (not curre	ent) object 🔹 🕨	
	Expand	•	<mark>, , Preliminary</mark>
CrankshaftRight.	Expand Multiple Levels	•	
342	De_Expand		·
	Compare with Assembly		A, 1, , Prelimin
- Cylinder Assm 1 / Cy	Open in new PDM Window		
Piston.1 / Piston,			
Picton Model 1	Promote		
Instolliviodel.	2 Re <u>v</u> ise		
PistonPin_Assm.1	Load		
	Load with Links		Z
	Load in Context		⊻ x∕∽y
	Add Temp		

Picture 109: Action "Add Temp"

The CATProducts will not be loaded. Only the CATParts are loaded and positioned correctly (see *Picture 110: 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 110: 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 111: Action "Duplicate"*).

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



Picture 111: 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 112: "PDM Create" dialog for duplicate*).

PDM Create		? ×
Create dialog Engine	for type Assembly:	
Part Number Name	Engine	*
Unit	EA	-
Make / Buy Description	Make	-
Create dialog Engine	for type CATProduct:	
Doc. Number	Engine	*
Name		
Description		
	OK Apply	Cancel

Picture 112: "PDM Create" dialog for duplicate

When you close the dialog with "OK" (see *Picture 113: 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 114: Duplicated CATProduct object*).

PDM Create		? X
Create dialog Engine	for type Assembly:	
Part Number	EngineNew	*
Name		
Unit	EA	
Make / Buy	Make	-
Description		
Create dialog Engine	for type CATProduct:	
Doc. Number	EngineNew	*
Name		
Description		
	OK Apply	Cancel

Picture 113: Filled "PDM Create" dialog for duplicate



Picture 114: Duplicated CATProduct object

Improved "Parts in Session" Functionality in CAD Document Structure Data Model

The "Select a part in session" list previously was only populated in the BOM Part Structure Data Model, by part items that have been loaded into the CATIA session. Now, in the "Duplicate" process in the CAD Document Structure Data Model, it contains a list of Part items that are related to the CAD Documents that are duplicated.

During the "Duplication" process, in this example of a CATDrawing and two linked CATParts, it is not necessary to use the "Query for part" entry to get access to the Part items which are linked to the CAD Documents to be duplicated, since they can be selected in the list under "Select a part in session":

PDM Create	?	\times
Create dialog for typ NewPart3	e CATPart:	
Corresponding part	No part	~*
Doc. Number	No part	^ *
Name Description	Create new Component Create new Std. Part	*
	Select a part in session: NewDrawing1-Part, A, 1, NewD	
	NewPart3-Part, A, 1, NewPart3- NewPart4-Part, A, 1, NewPart4-	~
ls Template	< >	
Standard Part		
[OK Apply OC	ancel

Picture 115: Dropdown list containing related parts

Deat Niveshaw				*
Name	New	Part3-Part-new1		*
Unit	E A	Parts-Part-new I		
Make / Buy	LA		· · ·	
Description	Iviake		~	
Create dialog	for typ	pe CATPart:		
Create dialog NewPart3	for typ	pe CATPart:		
Create dialog NewPart3 Correspondin	for typ	Create new Compone	ent	~
Create dialog NewPart3 Correspondin Doc. Nu	for typ ng part umber	Create new Compone NewPart3-new1	ent	~
Create dialog NewPart3 Correspondin Doc. No	for typ ng part umber Name	Create new Compone NewPart3-new1 NewPart3-new1	ent	~
Create dialog NewPart3 Correspondin Doc. Nu Descr	for typ ng part umber Name ription	Create new Compone NewPart3-new1 NewPart3-new1	ent	~
Create dialog NewPart3 Correspondin Doc. Ni Descr	for typ ng part umber Name ription	Create new Compone NewPart3-new1 NewPart3-new1	ent	~
Create dialog NewPart3 Correspondin Doc. No Descr	for typ ng part umber Name ription	Create new Compone NewPart3-new1 NewPart3-new1	ent	~
Create dialog NewPart3 Correspondin Doc. Nr Descr	for typ ng part umber Name ription	Create new Compone NewPart3-new1 NewPart3-new1	ent	~
Create dialog NewPart3 Correspondin Doc. Nr Descr Is Ter	for typ ng part umber Name ription	Create new Compone NewPart3-new1 NewPart3-new1	ent	~

Even if a new part item is created during the process, it is accessible later in the "Duplication" process:

Picture 116: Dialog where a new part is created

If the "Duplicate" process continues this newly created part item is also available in the dropdown list:

Create dialog for type CATPart: NewPart4 Corresponding part Doc. Number Name Description Lescription Is Template Standard Part	PDM Create	?	×
Corresponding part Doc. Number Name Description Is Template Standard Part No part No part No part No part Create new Component Create new Std. Part Query for part Select a part in session: NewPart3-Part-new1, A, 1, New NewPart3-Part, A, 1, NewPart3- NewPart4-Part, A, 1, NewPart4- Standard Part	Create dialog for typ NewPart4	e CATPart:	
Doc. Number No part * Name Create new Component * Description Create new Std. Part * Query for part Select a part in session: * NewPart3-Part-new1, A, 1, NewD NewPart3-Part, A, 1, NewD * Is Template NewPart4-Part, A, 1, NewPart4- Y * Standard Part * *	Corresponding part	No part	~*
Name Create new Component Create new Std. Part Query for part Select a part in session: NewPart3-Part.new1, A, 1, New NewDrawing1-Part, A, 1, NewD NewPart3-Part, A, 1, NewPart3- NewPart4-Part, A, 1, NewPart4- Standard Part	Doc. Number	No part	^ *
Create new Std. Part Query for part Select a part in session: NewPart3-Part-new1, A, 1, New NewDrawing1-Part, A, 1, NewD NewPart3-Part, A, 1, NewPart3- NewPart4-Part, A, 1, NewPart4- Standard Part	Name	Create new Component	*
Select a part in session: NewPart3-Part-new1, A, 1, New NewDrawing1-Part, A, 1, NewD NewPart3-Part, A, 1, NewPart3- NewPart4-Part, A, 1, NewPart3- NewPart4-Part, A, 1, NewPart4- Standard Part	Description	Create new Std. Part Query for part	
Is Template		Select a part in session:	
Is Template Standard Part		NewPart3-Part-new1, A, 1, New	
Is Template Standard Part		NewDrawing1-Part, A, 1, NewD	- 1
Standard Part	Is Template	NewPart3-Part A 1 NewPart4-	~
	Standard Part	< >	
		OK Apply Ca	incel

Picture 117: Dropdown list containing newly created Part

Selecting this part from the list has the same effect as querying for it and selecting it from the query results:

PDM Create			?	×
Create dialog NewPart4	for typ	e Component:		
Part Number	NewP	art3-Part-new1	*	
Name	NewP	art3-Part-new1	*	
Unit	EA			
Make / Buy	Make		\sim	
Description				
Create dialog NewPart4	for typ	e CATPart:		
Correspondin	g part	NewPart3-Part-new1,	A, 1, Nev	* ~
Doc. Nu	mber	NewPart4-new1		*
	Name	NewPart4-new1		*
Descr	iption			
ls Ten	nplate			
Standar	d Part			
	Г	0.0%	1.00	
STATE OF STATE	L	OK Apply	-	ancel

Picture 118: Dialog with newly created Part selected

With this functionality it is simpler to relate all the duplicated CAD Documents to the same Part item:



Picture 119: Dialog with newly created Part selected

Duplicate Structure

It is possible to duplicate CATProduct structures, not only single CATPart or CATProduct documents.

The Part Numbers of duplicated CATParts or CATProducts must not be controlled by internal CATIA business logic like knowledge ware.

In the PDM Workbench Schema file one of the following two variants can be configured. It is not possible to use both variants in the same PDM Workbench environment.

Variant A (only available in CAD Document Structure Data Model)

Use Case:

Engine Engine FixedParts_Assm (FixedParts_Assm.1) FixedParts_Assm (FixedParts_Assm.1) CylinderBlock (CylinderBlock.1) CylinderBlock (CylinderBlock.1) Rack_Assm (Rack_Assm.1) NewRack_Assm (Rack_Assm.1) NewRack (Rack.1) Rack (Rack.1) NewScrew (Screw.1) NewScrew (Screw.4) → NewScrew (Screw.3) NewScrew (Screw.2) Constraints 🗓 Constraints Cylinder_Assm (Cylinder_Assm.1) Cylinder_Assm (Cylinder_Assm.1) ConnectionRod (ConnectionRod.1) ConnectionRod (ConnectionRod.1) Crankshaft_Assm (Crankshaft_Assm.1) Crankshaft_Assm (Crankshaft_Assm.1) Constraints Constraints Applications Applications

- Update in PDM required
- New sub-structure will be related to existing structure

Select a CATProduct and click the context action "Duplicate Structure" (see *Picture 120: Structure to be duplicated*).



Picture 120: Structure to be duplicated

After clicking on "Duplicate Structure" you will get a list of the CATProducts and CATParts which are contained in the selected structure (see *Picture 121: Pre-selected list of documents*). Initially all CATIA documents are checked.

List of	CATIA Documents	8	x
CA	TIA documents to duplicate:		
	Engine		-
	FixedParts_Assm		
	CylinderBlock		
	CylinderBlockModel		
	Rack_Assm		
	Rack		
	RackModel		=
	Screw		-
	ScrewModel		
	Cylinder_Assm		
	PistonPin_Assm		
	SnapRing		
	SnapRingModel		
	PistonPincomp		
1	PistonPincompModel		
1	Piston		
	PistonModel		
	ConnectionRod		
	ConnectionRodModel		-
		OK SCanc	el

Picture 121: Pre-selected list of documents

You can uncheck any of the documents in the list (see *Picture 122: Document list with unchecked documents*). Only the checked documents will be duplicated.

List of (CATIA Documents	? ×
	TA documents to duplicate:	
	Engine	<u>^</u>
	FixedParts_Assm	
1	CylinderBlock	
1	CylinderBlockModel	
	Rack_Assm	
	Rack	
	RackModel	
	Screw	
	ScrewModel	
	Cylinder_Assm	
1	PistonPin_Assm	
	SnapRing	
1	SnapRingModel	
	PistonPincomp	
	PistonPincompModel	
	Piston	
	PistonModel	
	ConnectionRod	
	ConnectionRodModel	*
		OK Second Cancel

Picture 122: Document list with unchecked documents

In this example the "Rack_Assm" substructure of the "Engine" structure will be duplicated.

You right-click on "Rack_Assm" \rightarrow "PDM Workbench" \rightarrow "Duplicate Structure" (see *Picture 123: Selecting a substructure to duplicate*).



Picture 123: Selecting a substructure to duplicate

The list of the CATIA documents which are contained in the selected substructure appears (see *Picture 124: Example with small substructure*).



Picture 124: Example with small substructure

If you click on "OK" the "PDM Create" dialogs for each checked CATIA document will appear (see *Picture 125: Changed key attribute*). As in the single-document "Duplicate" functionality they will contain the values of the document to be duplicated. You need to change the key attribute.

New Values (Ra	ack_Assm)	×
Doc. Number	Rack_Assm-New	*
Name		1993-19
Description		
Is Template		
	OK Apply OK Canc	el

Picture 125: Changed key attribute

This is the structure that is being duplicated (see Picture 126: Structure being duplicated).



Picture 126: Structure being duplicated

When all dialogs have been filled out the "Duplicate" process starts (see *Picture 127: Duplicate Structure – progress bar*).

U	pdate to PDM?								Ŀ	
	File name	CATIA Part Number	PDM Display		Locked By	PLM State	File Status	Operation		Result
	RackModel.CATPart	RackModel-New					Modified	Create		
	ScrewModel.CATPart	ScrewModel-New					Modified	Create		
	Rack.CATProduct	Rack-New					Modified	Create : 1. Creat	te PDM File Object of typ	SUCCESS
	Screw.CATProduct	Screw-New					Modified	Create : 1. Creat	te PDM File Object of typ	SUCCESS
I	Rack_Assm.CATProduct	Rack_Assm-New					Modified	Create : 1. Creat	te PDM File Object of typ	SUCCESS
	No CAD document PDM item es Created '/CAD/Mechanical/Asse Created relation of type 'CAD Str Added instance 'Rack1' to CAD Added instance 'Screw.1' to CAD Added instance 'Screw.1' to CAD Added instance 'Screw.2' to CAD Added instance 'Screw.2' to CAD	ists yet, creating it. mbly with name 'Rack, ucture'. Structure relation. Utture: Structure relation. Structure relation. Structure relation.	Assm-New'.	Update in PD	M time remainin	Updating c	hanges to PDM Uploading f 40	system iles to PDM % completed 0sec		
	Action succeeded.	Yes	import with reconnect	No	UNIOCK and	N Jave: N	0		Upd	late Close

Picture 127: Duplicate Structure - progress bar

If a substructure has been duplicated the existing complete PDM structure is not changed yet, it still contains the old substructure ... (see *Picture 128: Existing PDM structure containing old substructure*)



Picture 128: Existing PDM structure containing old substructure

... even though the CATIA structure already contains the new substructure (see *Picture 129: CATIA structure containing new substructure*).



Picture 129: CATIA structure containing new substructure

For actually linking the newly duplicated substructure to the existing structure a PDM update has to be performed. This is the same behavior as the single-document "Duplicate" functionality has.

Performing the PDM Update (see Picture 130: Update with new substructure).

Engine.CATProduct	👝 🖂 🖾 Triglete, A, I, , Peliminary Dogged in as admin, database. Innovato/Solution:1112)	
Fingine Findine Findine Solution	Provide A L. Training Provide A L. Tra	2
RackModel-New (RackModel 1)		
Soren-New (Soren 1) Soren-New (Soren 2) Contraints Contrai	Thream CRM Reference Reference <threference< th=""> <threference< th=""> <th< td=""><td></td></th<></threference<></threference<>	
Applications	Contrarse-CO document prevention Two Report with Teccoments" Table Schold After Table Speet and Teccoments Teccoments (Speet After Teccoments) (Sp	į,

Picture 130: Update with new substructure

The result of the update is described in the text area of the window (see *Picture 131: Update has changed the structure to the new substructure*).

Update to PDM?	-			- California	1.000	Barton Theory & Barton &	X
File name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Result
RackModel-New.CATPart	RackModel-New	RackModel-New, A, 1, , Preliminary	admin	Preliminary	Modified	Update	SUCCESS
ScrewModel-New.CATPart	ScrewModel-New	ScrewModel-New, A, 1, , Preliminary	admin	Preliminary	Modified	Update	SUCCESS
FixedParts_Assm.CATProduct	FixedParts_Assm	FixedParts_Assm, A, 1, , Preliminary	admin	Preliminary	Modified	Update : 1. Add sub-structure relation "	SUCCESS
<		II					
Part Number 'FixedParts, Assm': Created relation of type 'CAD Structure'. Added instance, from CAD Structure relation. Removing instance from CAD Structure relation. Deleted instance Rack, Assm.' I CAD Structure relation, updated quantity to '0'. Removed CAD Structure relation.							
opioaded screwwoder-wew.cA	irait						
Create new CAD document gener	ration? <u>No</u>	Import with Reconnect? No	Unlock aft	er Save?	•	Шр	date Close

Picture 131: Update has changed the structure to the new substructure

After the update the complete PDM structure contains the new substructure (see *Picture 132: Existing PDM structure containing new substructure*).



Picture 132: Existing PDM structure containing new substructure

Variant B

Use Case:



- Update in PDM required
- New sub-structure will not be related to existing structure

When using "Duplicate Structure" it is possible to add the original Item to a property in the new Item (CAD/Part).

You can select any node in a loaded CATProduct structure, including the root node. If the root node is selected then the whole structure will be duplicated, otherwise a part of the complete structure.

Select a CATProduct and click the context action "Duplicate Structure" (see *Picture 133: Structure to be duplicated*).



Picture 133: Structure to be duplicated

After clicking on "Duplicate Structure" you will get a list of the CATProducts and CATParts which are contained in the selected structure (see *Picture 134: Duplicate structure – Preselected list of documents*). Initially all CATIA documents are checked.

Duplie	ate structure					? ×
- Na Sou	ming rules to apply: ce rule:	Target rule:				
			Apply rule			
	Source name	Target na	ame		Selected node	
5555555555	Engine -FixedParts_Assm CylinderBlock CylinderBlockModel Rack_Assm Rack Rack Screw Screw Screw			II	Copy node Source name Target name	Аррју
						OK Gancel

Picture 134: Duplicate structure – Pre-selected list of documents

You can uncheck any of the documents in the list. Only the checked documents will be duplicated.

You have to select the sub-tree or object to be unchecked in the list of the left side. On the right side of the dialog the information about the selected node will be updated, e.g. check-box "Copy node" and field "Source name" will be filled. (see *Picture 135: Duplicate structure – Selected node*).

urce rule:	Target rule:	
	Apply ru	ule
Source name	Target name	Selected node
Engine		Copy node
-FixedParts_Assm		Source name
CylinderBlock		Rack
CylinderBlockModel		Target name
Rack_Assm		
Rack		Ap
RackModel		
Screw		
ScrewModel		
Screw		

Picture 135: Duplicate structure - Selected node

Naming rules to apply: ource rule:	Target rule:	
	Apply	y rule
Source name	Target name	Selected node
Engine		Copy node
-FixedParts Assm	-	Source name
CylinderBlock		Rack
 CylinderBlockModel 		Target name
Rack_Assm		
Rack		Арр
RackModel		
Screw		
ScrewModel		
Screw		-

You have to uncheck the check-box "Copy node" (see *Picture 136: Duplicate structure – Uncheck selected node*).

Picture 136: Duplicate structure – Uncheck selected node

In the next step please click the "Apply" button. The sub-tree or object will be unchecked in the list of the left side (*Picture 137: Duplicate structure – Document list with unchecked documents*).

urce rule:	Target rule:	
	Apply rule	le
Source name	Target name	Selected node
Engine		Copy node
-FixedParts_Assm		Source name
CylinderBlock		Rack
CylinderBlockModel		Target name
Rack_Assm		
Rack		
RackModel		
Screw		
ScrewModel		
Screw		

Picture 137: Duplicate structure – Document list with unchecked documents

You can define a naming rule for all nodes to be duplicated by filling the "Source rule" and "Target rule" using wildcard "*" (see *Picture 138: Duplicate structure – Fill naming rule*).

Duplic	ate structure		? ×
Na	ming rules to apply:		
Sour	ce rule:	Target rule:	
*		New*	
		Apply rule	
	Source name	Target name Selected node	
1	Engine	Copy node	
	-FixedParts Assm	_ Source name	5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5
	CylinderBlock	Rack	
 Image: A second s	CylinderBlockModel	Target name	
1	Rack_Assm		
X	Rack		Apply
X	RackModel		
 ✓ 	Screw		
	ScrewModel		
	Screw	•	
		<u>● ок</u>	Cancel

Picture 138: Duplicate structure - Fill naming rule

When you click the "Apply rule" button the Target name will be filled for the objects to be duplicated (see *Picture 139: Duplicate structure – New Target names*).

urce rule:	Target rule:			
	New*			
		Apply rule		
Source name		Target name	*	Selected node
Engine		NewEngine		Copy node
-FixedParts Assm		NewFixedParts Assm	E	Source name
CylinderBlock		NewCylinderBlock		Rack
CylinderBlockModel		NewCylinderBlockModel		Target name
Rack_Assm		NewRack_Assm		
Rack				Арг
RackModel				
Screw		NewScrew		
ScrewModel		NewScrewModel		
Screw		NewScrew		

Picture 139: Duplicate structure – New Target names

Alternatively, you can define the Target name on the right side of the dialog by selecting the different nodes of the list of the left (see *Picture 140: Duplicate structure – Fill single Target name*).

urce rule:	Target rule: New*				
		Apply rule			
Source name		Target name	^	Selected node	
Engine		NewEngine		Copy node	
-FixedParts_Assm	-	NewFixedParts_Assm	E	Source name	
CylinderBlock		NewCylinderBlock		Rack_Assm	
CylinderBlockModel		NewCylinderBlockModel		larget name	
Rack_Assm		NewRack_Assm		NewKack_Assmilest	-
Rack					Ар
 RackModel 					
Screw		NewScrew			
ScrewModel		NewScrewModel			
Screw		NewScrew	-		

Picture 140: Duplicate structure – Fill single Target name

Click the "Apply" button to set the Target name (see *Picture 141: Duplicate structure – Filled single Target name*).

Naming rules to apply:	Target rules				
*	New*				
		Apply rule			
Source name		Target name	*	Selected node	
Engine		NewEngine		Copy node	
 -FixedParts Assm 	-	NewFixedParts Assm	=	Source name	
 CvlinderBlock 		NewCylinderBlock		Rack_Assm	
CylinderBlockModel		NewCylinderBlockModel		Target name	
Rack Assm		NewRack AssmTest		NewRack_AssmTest	
×Rack			_		Apply
RackModel					
Screw		NewScrew			
ScrewModel		NewScrewModel			
Screw		NewScrew	-		

Picture 141: Duplicate structure – Filled single Target name

When you click on "OK" the structure will be duplicated (see *Picture 142: Duplicated structure in CATIA*).

NexEngine.CATProduct	
NewFinder NewFindedParts_Assm (FixedParts_Assm 1)	\triangle
🔶 💀 NewCylinder_Assm (Cylinder_Assm1)	
NewConnectionRod (ConnectionRod.1)	
See New Contests (Laser (Contests))	
70M Worklands	
Information	
Copy Structure finished.	
_	
Action succeeded.	
	ł
	У— <u>1</u> .х

Picture 142: Duplicated structure in CATIA

The new structure has to be synchronized in Aras Innovator (see *Picture 143: Synchronize the duplicated structure*).

Cylinder_Assr	File name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Result A	-
Conservices	NewCylinderBlockModel CAT	NeuCulinderBlock				Modified	Create		
ConnectionRe	NewScrewModel.CATPart	NewScrewModel				Modified	Create		
Crankshaft An	NewScrew CATProduct	NewScrew				Medified	Create 1. Create PDM File Object of typ-		
cramonarore	NewCylinderBlock.CATProduct	NewCylinderBlock				Modified	Create : 1. Create PDM File Object of typ		
traints .	NewReck Assem CATProduct	NewRack Assm				Modified	Create : 1. Create PDM File Object of typ	-	
	NewSnapRingModel.CATPart	NewSnapRingModel				Modified	Create		
ons	NewPistonPincompModel.CA	NewPistonPincom				Modified	Create		
2340	NewSnapRing.CATProduct	NewSnapRing				Modified	Create : 1. Create PDM File Object of typ		
	NewPistonPincomp.CATProd	NewPistonPincomp				Medified	Create : 1. Create PDM File Object of typ		
	NewPistonModel.CATPart	NewPistonModel				Modified	Create		
	NewPistonPin_Assm.CATProd	NewPistonPin_Assm				Modified	Create : 1. Create PDM File Object of typ	+	
	Create new CAD document gener	ation? No	Unlock after Save? No	Create correspo	nding BOM par	rts? <u>No</u>		Update Close	

Picture 143: Synchronize the duplicated structure

In this example the "Rack_Assm" substructure of the "Engine" structure will be duplicated.

You right-click on "Rack_Assm" \rightarrow "PDM Workbench" \rightarrow "Duplicate Structure" (see *Picture 144: Selecting a substructure to duplicate*).

Engine.CATProduct		
FixedParts_Assm (Fix	xedParts_Assm.1) ylinderBlock 1)	4
+-* Cylinder_Assm ()	Access 13 Cgnter graph Beframe On Hide/Show Prgperties Alt+Enter	
Crankshaft_Assr	Cut Ctrl+X Copy Ctrl+C Paste Ctrl+V Paste Special	
2	Delete Del Rack Assm.1 object · PDM Workbench · DDM Properties	
C F	Components Representations Selection Mode	,
	Claim all	
	Duplicate Structure 생각 Ingert PDM Node 영광 Beplace Node	×Ĺx

Picture 144: Selecting a substructure to duplicate

The list of the CATIA documents which are contained in the selected substructure appears (see *Picture 145: Example with small substructure*).

Duplicate structure	and the second se	? <mark>×</mark>
Naming rules to app Source rule:	oly: Target rule: Apply rule	
Source name	Target name	Selected node
 Rack_Assm -Rack -RackModel -Screw -ScrewModel -Screw -ScrewWodel -Screw -Screw -Screw -Screw 	- - - - - - - - -	Copy node Source name Target name Apply
		OK Gancel

Picture 145: Example with small substructure

You can define a naming rule for all nodes to be duplicated by filling the "Source rule" and "Target rule" using wildcard "*". When you click the "Apply rule" button the Target name will be filled for the objects to be duplicated.

Duplicate structure				? 💌
Naming rules to apply: Source rule: *	Target rule: New*	Apply rule		
Source name		Target name		- Selected node
Rack_Assm -Rack -RackModel Screw -ScrewModel -ScrewModel -ScrewModel -Screw -Screw	- - - - - - - - -	NewRack_Assm NewRack NewRackModel NewScrew NewScrewModel NewScrew NewScrew NewScrew NewScrew NewScrewModel NewScrew	11	Copy node Source name Target name Apply
				OK Cancel

Picture 146: Example with small substructure – Filled Target names

When you click on "OK" the structure will be duplicated (see *Picture 147: Duplicated substructure in new CATIA window*).



Picture 147: Duplicated substructure in new CATIA window

The duplicated substructure will be opened in a new window.

The new structure has to be synchronized in Aras Innovator.

Variant B - Duplicate Structure enhancements

There are some new functionalities in the Duplicate structure dialog:

Duplicate structure					?	×
- Naming rules to apply:						
Source rule:	Target rule:					
Duplicate Standard Parts		Ap	ply rule			
Duplicate all / Select pode	s to duplicate	Clear target	names			
Bupicate all / Select hode		cical target				
Source name	CATIA display	Target name	^	Selected node		
20210421_001	20210421_001			Copy node		
-20210421_001	DRW			Source name		
-20210421_002	20210421_002 (20210421_002.1)					
20210421_003	20210421_003 (20210421_003.1)			Target name		
20210421_007	20210421_007 (20210421_007.1)					
20210421_007	DRW					Apply
20210421_008	20210421_008 (20210421_008.1)					
×20210421_009_std	20210421_009_std (20210421_009_std.1)					
20210421_008	20210421_008 (20210421_008.2)					
20210421_003	20210421_003 (20210421_003.2)					
20210421_007	20210421_007 (20210421_007.1)					
20210421_007	DRW					
20210421_008	20210421_008 (20210421_008.1)					
×20210421_009_std	20210421_009_std (20210421_009_std.1)					
20210421_008	20210421_008 (20210421_008.2)					
-20210421 004	20210421_004 (20210421_004.1)		~			
				G OK	- I 😜	Cancel

Picture 148: New function in the "Duplicate structure" dialog

1. Apply rule

This function was changed. Now you can use the rule to create new target names multiple times. The target name is only modified if the rule hits the source name. If the source name is not hit by the rule, the target name will stay unchanged.

2. Clear target names

This function clears all target names.

- 3. CATIA display This column shows the same text, like the nodes in the CATIA structure tree.
- 4. Duplicate all / Select nodes to duplicate By default all nodes except Standard Parts are selected to be duplicated. If you only want to duplicate some dedicated nodes and their path to root, you can uncheck the box. In this case, all nodes are excluded from the duplicate action. You can select the specific nodes and enable the "Copy node" manually.

It is possible to disable the "CATIA display" column. In this case the original column of the "Duplicate Structure" dialog is activated.

Dupl	icate structure				? ×
Na Sou	ming rules to apply: irce rule: Duplicate Standard Parts Duplicate all / Select nodes to duplicate	Target rule:	Apply rule Clear target names		
11	Source name 20210421_001 -20210421_001	DRW	Target name	^	Selected node Copy node Source name
555	-20210421_002 20210421_003 20210421_007	- 			Target name
× ××	20210421_007 20210421_008 20210421_009_std	DRW 			Apply
555	20210421_008 20210421_003 20210421_007	 		~	
					OK Gancel

Picture 149: "Duplicate Structure" dialog, hide "CATIA display" column

Create Relation between Windows

You might modify the PDM structure by adding existing objects from several PDM Workbench windows to the PDM 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 150: Action "Copy" between windows*). Of course, you also can use the short cut "CTRL+C".



Picture 150: 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 151: Action "Paste" between windows*). Of course, you also can use the short cut "CTRL+V".

gine, B. 2., Pielminary (logged in as admin, database: InnovatorSolutions1112)	Con I-ED Con I NewProduct2, A.1., Preliminary (logged in as admin, database: InnovatorGolutions1112)	0 6 2
<pre>vet 1. Therease house hou</pre>	Theorem 1, A. Ference program is a sine defauit inclusion (12)	
ExpineerBlockModel 1 / CylinderBlockModel, A, 1, Preliminary Rack_Assm.1 / Rack_Assm, A, 1, Preliminary Rack_1 / Rack_A, 1, Preliminary	L,	L,

Picture 151: Action "Paste" between windows

You specify the relation you want to create in the structure between the two objects (see *Picture 152: 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.

Close

Picture 152: Select the new relation

The instance object gets inserted into the existing PDM structure tree and the new relation gets created in the PDM system (see *Picture 153: PDM structure with inserted object*).



Picture 153: PDM structure with inserted object

Create CAD in Parent

It is possible to create a new CATPart and CATProduct CAD Document (or Part item, in the BOM Part Structure Data Model) directly in the CATProduct structure.

The "PDM Workbench" context menu in the CATIA window has the action "PDM Create in Context" (see *Picture 154: Action "PDM Create in Context"*).

NewProduct1.CATProduct	ct		
NewProduct1			<u>ل</u> تر ا
Applications	Center graph Reframe On → Hide/Show → Properties Alt+Enter → Open Sub-Tree → Cut Ctrl+X → Copy Ctrl+C → Paste Ctrl+V Paste Special → Delete Del NewProduct1 object →		A
	PDM Workbench ▶ Components ▶ Representations ▶ Selection Mode ▶	EDM Properties PDM Create in Context Guighlight PDM Nodes Claim Cla	
		✓ U₂e BOM Position ☐ Open related drawings	x x

Picture 154: Action "PDM Create in Context"

Starting this action has the same effect as the "Create" toolbar action (see *Picture 156: Action "Create"*), except that only "CATPart" and "CATProduct" are selectable in the list (see *Picture 155: Create in Context – Select object type*).

Sele	ct a PDM type for per	for	?	x
P	DM Object Types —			
C	ATPart			
C	ATProduct			2.5
-		DK]	🥥 Ca	ncel

Picture 155: Create in Context – Select object type



Picture 156: Action "Create"

After the action has completed the PDM items which correspond to the newly created CATPart or CATProduct node in the CATProduct structure have been created (see *Picture 157: Created object* and *Picture 158: Update result window for create object*).

NewProduct1.CATProduct	
NewProduct1 NewProduct2 (NewProduct2.1) Applications	y

Picture 157: Created object

Jpdate to PDM?								
File name	CATIA Part Number	PDM Display	Locked By	PLM State	BOM Type	File Status	Operation	Result
New.CATProduct	New				BOM	Modified	Create : 1. Create PDM Object of type "/.	SUCCESS
•			III					
No Part PDM item exists yet, cree Created '/Part/Assembly' with ni No main file PDM item exists yet No CAD document PDM item ex Created '/CAD/Mechanical/Asse No relation exists between part a Created relation of type 'Part CA	sting it. ame 'NewProduct2'. ; creating it. ists yet, creating it. mbly' with name 'New nd main file yet, creatir D'.	Product2'. ng it.						
Uploaded NewProduct2.CATPro	duct							
Create new CAD document gener	nation? <u>No</u>						<u>10</u>	odate Clo

Picture 158: Update result window for create object

Starting the Update command will link the newly created PDM item to the loaded structure (see *Picture 159: Update window for create relation* and *Picture 160: Update result window for create relation*).

Jpdate to PDM?								
File name	CATIA Part Number	PDM Display	Locked By	PLM State	BOM Type	File Status	Operation	Result
NewProduct1.CATProduct	NewProduct1	NewProduct1, A, J, , Preliminary	admin	Preliminary	вом	Modified	Update : 1. Add sub-part relation "NewP	
Create new CAD document gen	eration? No						<u>.</u> U;	odate Clo

Picture 159: Update window for create relation

File name	CATIA Part Number	PDM Display	Locked By	PLM State	BOM Type	File Status	Operation	Result
NewProduct1.CATProduct	NewProduct1	NewProduct1, A, 1, , Preliminary	admin	Preliminary	BOM	Modified	Update : 1. Add sub-part relation "NewP	SUCCESS
Part Number 'NewProduct1': Added instance 'NewProduct2 Uploaded NewProduct1.CATPr	1' to Part BOM relation. roduct		m					
Create new CAD document gen	eration?						100	

Picture 160: Update result window for create relation

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.

If a Part Number can be changed during Update (autoname or manual input), the Part Number of the CATPart or CATProduct must not be controlled by internal CATIA business logic like knowledge ware.



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

An "Update to PDM" dialog opens and asks to confirm the described actions. In this example the CATIA documents will be created (see Picture 161: Confirm the "Update" (with Create) action).

File name	CATIA Part Number	PDM Display	Locked By	PLM State	BOM Type	File Status	Operation	Result
NewCylinderBlockModel.CAT	NewCylinderBlock				BOM	Modified	Create : 1. Create PDM Object of type */	
NewRackModel.CATPart	RackModel				BOM	Modified	Create : 1. Create PDM Object of type */	
NewScrewModel.CATPart	ScrewModel				BOM	Modified	Create : 1. Create PDM Object of type "/	
NewRack.CATProduct	NewRack				BOM	Modified	Create : 1. Create PDM Object of type */	
NewScrew.CATProduct	NewScrew				BOM	Modified	Create : 1. Create PDM Object of type */	
NewCylinderBlock.CATProduct	NewCylinderBlock				BOM	Modified	Create : 1. Create PDM Object of type "/	
NewRack_Assm.CATProduct	NewRack_Assm				BOM	Modified	Create : 1. Create PDM Object of type */	
NewSnapRingModel.CATPart	SnapRingModel				BOM	Modified	Create : 1. Create PDM Object of type "/	
NewPistonPincompModel.CA	PistonPincompMo				BOM	Modified	Create : 1. Create PDM Object of type "/	
NewSnapRing.CATProduct	NewSnapRing				BOM	Modified	Create : 1. Create PDM Object of type */	
NewPistonPincomp.CATProd	NewPistonPincomp				BOM	Modified	Create : 1. Create PDM Object of type "/	
Create new CAD document gener	ation? No							

Picture 161: Confirm the "Update" (with Create) action

The progress of the Update will be shown with the progress bars (see Picture 162: Update - progress bars).

12	
"Ne	wCylinderBlockModel"
Status :	0% completed
Estimated time remaining :	0sec
Cancel	
Update in PDM	×
---	-----------------------------------
8	Updating changes to PDM system
	Uploading files to PDM
, Status : Estimated time remaining	40% completed : 4sec Cancel

Picture 162: Update – progress bars

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 163: Objects are updated (with Create)*).

ate to PDM?		<u> </u>						
ile name	CATIA Part Number	PDM Display	Locked By	PLM State	BOM Type	File Status	Operation	Result
lewCylinderBlockModel.CAT	NewCylinderBlock				BOM	Modified	Create : 1. Create PDM Object of type "/	SUCCESS
NewRackModel.CATPart	RackModel				BOM	Modified	Create : 1. Create PDM Object of type "/	SUCCESS
NewScrewModel.CATPart	ScrewModel				BOM	Modified	Create : 1. Create PDM Object of type "/	SUCCESS
NewRack.CATProduct	NewRack				BOM	Modified	Create : 1. Create PDM Object of type "/	SUCCESS
NewScrew.CATProduct	NewScrew				BOM	Modified	Create : 1. Create PDM Object of type "/	SUCCESS
NewCylinderBlock.CATProduct	NewCylinderBlock				BOM	Modified	Create : 1. Create PDM Object of type "/	SUCCESS
NewRack_Assm.CATProduct	NewRack_Assm				BOM	Modified	Create : 1. Create PDM Object of type "/	SUCCESS
NewSnapRingModel.CATPart	SnapRingModel				BOM	Modified	Create : 1. Create PDM Object of type "/	SUCCESS
NewPistonPincompModel.CA	PistonPincompMo				BOM	Modified	Create : 1. Create PDM Object of type "/	SUCCESS
NewSnapRing.CATProduct	NewSnapRing				BOM	Modified	Create : 1. Create PDM Object of type "/	SUCCESS
< [III					
Jploaded NewFixedParts_Assm.0	ATProduct							
Uploaded NewCylinder_Assm.CA	TProduct							
Uploaded NewConnectionRod.C	ATProduct							
Uploaded NewCrankshaft_Assm.	CATProduct							
Uploaded NewEngine.CATProdu	ct							
reate new CAD document gener	ation? No							

Picture 163: Objects are updated (with Create)

If there are no objects to be created then it will only be asked if you want to update (see *Picture 164: Confirm the "Update" action*).

le name	CATIA Part Number	PDM Display	Locked By	PLM State	BOM Type	File Status	Operation	Result
ewFixedParts_Assm.CATPro	NewFixedParts_Ass	NewFixedParts_Assm, A, 1, , Preliminary	admin	Preliminary	BOM	Modified	Update	
ewCylinder_Assm.CATProdu	NewCylinder_Assm	NewCylinder_Assm, A, 1, , Preliminary	admin	Preliminary	BOM	Modified	Update	
ewConnectionRod.CATProd	NewConnectionRod	NewConnectionRod, A, 1, , Preliminary	admin	Preliminary	BOM	Modified	Update	
ewCrankshaft_Assm.CATPro	NewCrankshaft_As	NewCrankshaft_Assm, A, 1, , Preliminary	admin	Preliminary	BOM	Modified	Update	
ewEngine.CATProduct	NewEngine	NewEngine, A, 1, , Preliminary	admin	Preliminary	BOM	Modified	Update	
ate new CAD document gener	ration?							
ate new cap document gene	No I							

Picture 164: Confirm the "Update" action

When the "Update" process has finished you are informed about the actions that have been performed (see *Picture 165: Objects are updated*).

Update to PDM?								
File name	CATIA Part Number	PDM Display	Locked By	PLM State	BOM Type	File Status	Operation	Result
NewFixedParts_Assm.CATPro	NewFixedParts_Ass	NewFixedParts_Assm, A, 1, , Preliminary	admin	Preliminary	BOM	Modified	Update	SUCCESS
NewCylinder_Assm.CATProdu	NewCylinder_Assm	NewCylinder_Assm, A, 1, , Preliminary	admin	Preliminary	BOM	Modified	Update	SUCCESS
NewConnectionRod.CATProd	NewConnectionRod	NewConnectionRod, A, 1, , Preliminary	admin	Preliminary	BOM	Modified	Update	SUCCESS
NewCrankshaft_Assm.CATPro	NewCrankshaft_As	NewCrankshaft_Assm, A, 1, , Preliminary	admin	Preliminary	BOM	Modified	Update	SUCCESS
NewEngine.CATProduct	NewEngine	NewEngine, A, 1, , Preliminary	admin	Preliminary	BOM	Modified	Update	SUCCESS
Uploaded NewFixedParts, Assm.C Uploaded NewCylinder, Assm.CA Uploaded NewConnectionRod.Cl Uploaded NewCrankshaft, Assm.1	ATProduct TProduct ATProduct CATProduct	n						
Uploaded NewEngine.CATProdu	ct							
Create new CAD document generation	ation? No							Update

Picture 165: Objects are updated

Option to block the Update if linked File is not saved

When the user tries to update a CATDrawing which has links to CATParts that are loaded in the CATIA session, or a CATPart which has reference links to other CATParts, then the information to update those CATParts first is presented:



Picture 166: Information about CATParts to be updated

Option: "Update to PDM" Dialog only shown if new Documents to be created exist

It can be configured that the "Update to PDM" dialog is only shown if there are new items to create, or if there are warnings or errors during the execution.

Add newly created and updated Part or CAD Items to existing Items

It is possible to link the Part and CAD items during the "Update" process to an existing item. This can be done by using a custom method which is called at the end of the "Update" process.

Examples:

- 1. The user wants to link a new top-level item to a selected Folder/Project item.
- 2. The user has several Change Items he has to work on. During the "Update" process the user selects the current Change Item he is working on from a list of his Change Items. This Change Item can be related to all updated Part or CAD items. This gives the possibility to understand later why a certain change was made.

The following use case shows a sample implementation to link a new top-level item to a selected Folder/Project item.

With the correct configuration the user can select an entry from a list of folder names in the "Update to PDM" dialog. The list is the one returned by the custom method defined by the setting "CustomMethod_PostProcUpdate":

Create new CAD document generation?	No	PWB Folder	Folder Five 🗸 🗸	
			Folder Five	
			Folder Four	
			Folder One	
And I REAL PROPERTY AND		 A second s	Folder Six	
			Folder Three	
			Folder Two	
			Sector Sector Sector	

Picture 167: Folder list

The item that the user has selected will be the one that the newly created root Part item will be related to.

If the folder item and relationship is defined in the Schema file the user can expand the relations in the PDM structure window and find out which folder, if any, a Part item is related to, and which Part items are related to a folder:

NewProduct3, A, 1, , Preliminary	(logged in as admin, da	tabase: InnovatorSolutions1	206)
NewProduct3. A. 1, , Prelim - Part BOM, 1, / NewP - Attaches / NewProdu	Center graph Reframe On Hide/Show Properties Open Sub-Tree Cop Sopy Paste Paste Special	Alt+Enter Ctrl+X Ctrl+C Ctrl+V	
	<u>D</u> elete Ne <u>w</u> Product3, A, 1,	Del , Preliminary object	-
	E <u>x</u> pand	•	<u>U</u> ses Part BOM
	Expand Multiple Lev	vels 🕨	<u>A</u> ttaches
	De <u>-</u> Expand		Is Used in Part BOM
	Compare with Asse	mbly	ls in Folders
1	Open in new PDM V	Vindow	

Picture 168: Expanding "Is in Folders" in the PDM structure window



Picture 169: Expanding "Folder Items" in the PDM structure window



Picture 170: Expanded folder items in the PDM structure window

Deny Create of CAD at top-level Structure in BOM Part Structure Data Model

It is possible to create a top-level Part structure in Aras Innovator without attached CADs. The structure can be loaded to CATIA, the missing CATIA files are created on the fly in CATIA.

This functionality allows to prevent the update of the top-level Part structure, including the save of the "On the fly" created CATIA files.



Picture 171: Non-CAD top-level structure with on the fly created CATProducts

During update, the on the fly created CATProducts are skipped.

Jpdate to PDM?							-	
File name	CATIA Part Number	PDM Display	Locked By	PLM State	BOM Type	File Status	Operation	Result
NoCad_20210415_002.CATProduct	NoCad_20210415_002				BOM	Modified	Create : 1. Create file	. SKIPPED
NoCad_20210415_003.CATProduct	NoCad_20210415_003				BOM	Modified	Create : 1. Create file	. SKIPPED
NoCad_20210415_001.CATProduct	NoCad_20210415_001				BOM	Modified	Create : 1. Create file	. SKIPPED
20210415_006.CATPart	20210415_006	20210415_006, A, 1, , Preliminary	admin	Preliminary	BOM	Modified	Update	SUCCESS
20210415_004.CATProduct	20210415_004	20210415_004, A, 1, , Preliminary	admin	Preliminary	BOM	Modified	Update	SUCCESS
Uploaded 20210415_004,CATProduct								
Create new CAD document generation	? No							Jpdate

Picture 172: Update non-CAD top-level structure -> Result SKIPPED

Select Type of additional Parts in CAD Document Structure Data Model

This functionality is only available in the CAD Document Structure Data Model.

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.

You can select a specific part type to be created (see *Picture 173: Action "Set PDM Type to"*).

20180509_Product001.CATProduct 20180509_Product001 20180509_Product001	_		-	
Applications	Center graph Beframe On Hide/Show Properties Open Sub-Tree Cut Cut Sopy Paste Paste Special Delete 20180509_Part001.	Alt+Enter Ctrl+X Ctrl+C Ctrl+V Del 1 object		
	PD <u>M</u> Workbench Components Represent <u>a</u> tions Selection Mode			CaTPart (Non-BOM) *

Picture 173: Action "Set PDM Type to"

When updating a CATProduct structure the type of the additional part will be shown in the "Operation" column of the "Update to PDM" dialog (see *Picture 174: "Update to PDM" dialog for CATProduct structure*). The "Create corresponding BOM parts" button will be hidden.

pdate to PDM?		A STREET OF			THE REAL		
File name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Result
20180509_Part001.CATPart 20180509_Product001.CATPro	20180509_Part001 20180509_Product0	20180509_Product001, A, 1, , Preliminary	admin	Preliminary	Modified Modified	Create 1. ^{1,7} Part/Component [*] PDM Object will be created additionally Update: 1. Add sub-structure relation ² 20180509, Product001/20180509, Part001.1 ⁺ 2. C.,	
Create new CAD document gener	ation? <u>No</u>	Unlock after Save? No				1	Ipdate Clos

Picture 174: "Update to PDM" dialog for CATProduct structure

When updating a single CATPart the "Create corresponding BOM parts" button will be shown (see *Picture 175: "Update to PDM" dialog* for CATPart document). If a BOM type was selected for the CATPart, the button will be deactivated, otherwise the "Create corresponding BOM parts" button will be active. In this case the default part type will be created for the CATPart when setting the button to "YES"

pdate to PDM?							
File name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Result
20180509_Part002.CATPart	20180509_Part002				Modified	Create : 1. Create PDM File Object of typ	
Create new CAD document ger	eration? No	Unlock after Save? No	Create correspo	nding BOM pa	rts? No	Upd	ate Clo

Picture 175: "Update to PDM" 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 item 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.

Non-BOM CATParts and CATProducts

In the BOM Part Structure Data Model it is 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 (see *Picture 176: Setting a CATProduct to the non-BOM type*).

Engine.CATProduct		- • •
• Engine • • • FixedParts	_Assm (FixedParts_Assm.1)	\triangle
Cylinder_A +-% Piston +-% Piston +-% Constra +-% Connectio +-% Crankshaft	Center graph Reframe On m.1)	
🕈 🛄 Constraint	Delete Del	
Applications	Cylinder_Assm.1 object	
	PDM Workbench	
	Components Representations Selection Mode	Assembly (BOM) *
		U-U. ×_Ix

Picture 176: 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 (see *Picture 177: Setting a CATPart to the non-BOM type*).



Picture 177: Setting a CATPart to the non-BOM type

In the "Update to PDM" dialog the BOM types can be verified (see *Picture 178: "Update to PDM" dialog with non-BOM part items*).

odate to PDM?								
File name	CATIA Part Number	PDM Display	Locked By	PLM State	BOM Type	File Status	Operation	Result
CylinderBlockModel.CATPart	CylinderBlockModel				BOM	Modified	Create : 1. Create PDM Object of type "/	
RackModel.CATPart	RackModel				BOM	Modified	Create : 1. Create PDM Object of type */	
crewModel.CATPart	ScrewModel				BOM	Modified	Create : 1. Create PDM Object of type "/	
Rack.CATProduct	Rack				BOM	Modified	Create : 1. Create PDM Object of type "/	
crew.CATProduct	Screw				BOM	Modified	Create : 1. Create PDM Object of type */	
ylinderBlock.CATProduct	CylinderBlock				BOM	Modified	Create : 1. Create PDM Object of type "/	
Rack_Assm.CATProduct	Rack_Assm				BOM	Modified	Create : 1. Create PDM Object of type "/	
inapRingModel.CATPart	SnapRingModel				Non-BOM	Modified	Create	
istonPincompModel.CATPart	PistonPincompMo				Non-BOM	Modified	Create	
napRing.CATProduct	SnapRing				Non-BOM	Modified	Create : 1. Create PDM File Object of typ	
'istonPincomp.CATProduct	PistonPincomp				Non-BOM	Modified	Create : 1. Create PDM File Object of typ	
istonModel.CATPart	PistonModel				Non-BOM	Modified	Create	
'istonPin_Assm.CATProduct	PistonPin_Assm				Non-BOM	Modified	Create : 1. Create PDM File Object of typ	
Piston.CATProduct	Piston				Non-BOM	Modified	Create : 1. Create PDM File Object of typ	
ConnectionRodModel.CATPart	ConnectionRodMo				Non-BOM	Modified	Create	
rankshaftRightModel.CATPart	CrankshaftRightM				BOM	Modified	Create : 1. Create PDM Object of type "/	_
.rankPinModel.CATPart	CrankPinModel				BOM	Modified	Create : 1. Create PDM Object of type "/	
CrankshaftLeftModel.CATPart	CrankshaftLeftMo				BOM	Modified	Create : 1. Create PDM Object of type "/	
CrankshaftRight.CATProduct	CrankshaftRight				BOM	Modified	Create : 1. Create PDM Object of type */	
CrankPin.CATProduct	CrankPin				BOM	Modified	Create : 1. Create PDM Object of type "/	
CrankshaftLeft.CATProduct	CrankshaftLeft				BOM	Modified	Create : 1. Create PDM Object of type */	
FixedParts_Assm.CATProduct	FixedParts_Assm				BOM	Modified	Create : 1. Create PDM Object of type "/	
Cylinder_Assm.CATProduct	Cylinder_Assm				Non-BOM	Modified	Create : 1. Create PDM File Object of typ	
ConnectionRod.CATProduct	ConnectionRod				BOM	Modified	Create : 1. Create PDM Object of type */	_
Crankshaft_Assm.CATProduct	Crankshaft_Assm				BOM	Modified	Create : 1. Create PDM Object of type "/	
ngine.CATProduct	Engine				BOM	Modified	Create : 1. Create PDM Object of type "/	
reate new CAD document gener	ation? No						<u>.</u>	lpdate Clo

Picture 178: "Update to PDM" dialog with non-BOM part items

The result is a structure in PDM which contains both Part structures and CATIA Document structures (see *Picture 179: Resulting PDM structure*).



Picture 179: 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.

Attach additional non-BOM CATParts to Part

When using the BOM Part Structure Data Model only one CATPart can be attached to an Aras Innovator Part (Component). It can be configured to allow additional non-BOM CATParts attached to an Aras Innovator Part (Component).

When loading the Component, the additional CATParts are only loaded into the CATIA session if they are referenced by a CATIA file which is part of the normal BOM structure. It is also possible to load a non-BOM CATPart directly.

To create a non-BOM CATPart it is possible to set the PLM type of a CATPart. There is also a functionality to relate the active non-BOM CATPart to an Aras Innovator Part.

To use the functionality the usage of non-BOM CATIA files in BOM mode must be enabled.

To create a non-BOM CATPart and attach it to an Aras Innovator Component:

Set PLM type of active CATPart to "Non-BOM".



Picture 180: Select PDM type of CATPart

Then use the normal "Update" functionality to save the CATPart in Aras Innovator:

File name 0210323_011.CATPart	CATIA Part Number 20210323_011	PDM Display	Locked By	PLM State	BOM Type Non-BOM	File Status Modified	Operation Create : 1. Create PDM File Object o	of type	Result
						PDM Create Create dialog fo 20210323_011	? or type CATPart:	×	
<						Doc. Number Name Description	AUX2_3D ~	•	
Create new CAD document	generation? No					<u> </u>	le OK	cel	

Picture 181: Update non-BOM file in BOM Part Structure Data Model

Later you can attach the already registered CATPart to an Aras Innovator Component by using the new function "Relate active non-BOM CATPart to Part":

PDM object ty	pe		Teilenummer		Hauptrevision	Generation	Name	Status
Part		`	20210323_020	_	٨	h		Preliminary
PDM object at	tributes			👔 Open	in new PDM Window			
Part Number	20210323_020 ~	-		Claim				
Major Rev.	~			7 Brown	ata			
Generation	~			Tup Paulice				
Name	~	1		The Lord				
State	~	1		Es Luau	anti u Fileta Bast			
Unit	~			→1,2° Kelate	active File to Part			
Make / Buy	~			Relate	active non Bom CATP	art to Part		
Description				Show	Neighborhood			
				>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>	newest generation			
				Duplic	ate			
Created on				🗳 Duplic	ate Structure			
Modified on				🔏 Open	in Aras			
Created by	`			++ Expan	d	•		
Modified by	~		<					
				-				

Picture 182: Relate active non-BOM File to Part

It is also possible to relate a not yet registered CATPart as non-BOM CATPart to the selected Aras Innovator Part (this function has to be enabled separately). In this case the CATPart is registered using its current Part Number and the first NonBom3DRepType. There is no additional user action.

If you use the non-BOM CATParts as linked reference geometries in the BOM structure, it may be useful to open these files when opening the product structure. Therefore, the new PDM Workbench Options setting "Open linked Documents in own Window" is available for the BOM Part Structure Data Model:

Load With Links
Load current generation of linked Document
Load multiple levels of linked Documents
Open linked Documents in own Window
Open linked Documents of CATProcess in own Window



Reconnect at Update

Up to PDM Workbench version 9.0 the "Reconnect" functionality was combined with the "Auto Name" functionality. Now the "Reconnect" functionality is no longer connected with the "Auto Name" functionality.

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

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

Import Product Structure

Open a structure from disk and use "Update" to save the structure into Aras Innovator. If some of the files may be already stored in Aras Innovator, select "Import with Reconnect" before performing the update (see *Picture 184: "Update to PDM" dialog with "Import with Reconnect" button*).

Update to PDM?							-	- 0	×
File name	CATIA Part Number	PDM Display		Locked By	PLM State	File Status	Operation	Result	^
CylinderBlockModel.CATPart	CylinderBlockModel					Modified	Create		
RackModel.CATPart	RackModel					Modified	Create		
ScrewModel.CATPart	ScrewModel					Modified	Create		
Rack.CATProduct	Rack					Modified	Create : 1. Create PDM File Object of typ		
Screw.CATProduct	Screw					Modified	Create : 1. Create PDM File Object of typ		
CylinderBlock.CATProduct	CylinderBlock					Modified	Create : 1. Create PDM File Object of typ		
Rack_Assm.CATProduct	Rack_Assm					Modified	Create : 1. Create PDM File Object of typ		
SnapRingModel.CATPart	SnapRingModel					Modified	Create		
PistonPincompModel.CATPart	PistonPincompMo					Modified	Create		
SnapRing.CATProduct	SnapRing					Modified	Create : 1. Create PDM File Object of typ		
PistonPincomp.CATProduct	PistonPincomp					Modified	Create : 1. Create PDM File Object of typ		~
Create new CAD document gener	ration? No	Import with Reconnect?	Yes	Create corr	esponding BOI	M parts?	10	Update	Close

Picture 184: "Update to PDM" dialog with "Import with Reconnect" button

Update to PDM?							-	- 0	×
File name	CATIA Part Number	PDM Display	Lo	cked By	PLM State	File Status	Operation	Result	^
CylinderBlockModel.CATPart	CylinderBlockModel					Modified	Create		
RackModel.CATPart	RackModel					Modified	Create		
ScrewModel.CATPart	ScrewModel					Modified	Create		
Rack.CATProduct	Rack					Modified	Create : 1. Create PDM File Object of typ	SUCCESS	s
Screw.CATProduct	Screw					Modified	Create : 1. Create PDM File Object of typ	SUCCESS	s
CylinderBlock.CATProduct	CylinderBlock					Modified	Create : 1. Create PDM File Object of typ	SUCCESS	s
Rack_Assm.CATProduct	Rack_Assm					Modified	Create : 1. Create PDM File Object of typ	SUCCESS	s
SnapRingModel.CATPart	SnapRingModel					Modified	Create		
PistonPincompModel.CATPart	PistonPincompMo					Modified	Create		
SnapRing.CATProduct	SnapRing					Modified	Create : 1. Create PDM File Object of typ	SUCCESS	s v
<									>
Part Number 'ConnectionRod':									^
Reconnected ConnectionRod to Reconnected ConnectionRodMa	existing item								
Reconnected ConnectionRoawie	oder to existing item								
Part Number 'Crankshaft_Assm'									
Reconnected Crankshaft_Assm t	to existing item								
Part Number 'Engine':									
Reconnected Engine to existing	item								
									~
Create new CAD document gene	ration? No	Import with Reconnect?	Yes	Create corre	sponding BON	/i parts?	lo		
Action succeeded.								Update	Close

Picture 185: Messages about reconnected items

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

Import CATDrawing

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

Open the CATDrawing and use File \rightarrow Desk to open the directly referenced CATPart(s)/CATProduct(s) (see *Picture 186: Opening referenced 3D geometry files*).

▶ ♦/ヾ≓№ 餐[」]● 宮園盤協同	°e ∰ S 45 % % % %	
reDrawing CATDrawing		
Paraller Parameters Par		
. ,]		
	Parefersioner Commentationer Comment	_

Picture 186: Opening referenced 3D geometry files

Update the opened structure like described in "Import Product Structure". Do not close the renamed product structure after update (see *Picture 187: Reconnect referenced Product Structure*).



Picture 187: Reconnect referenced Product Structure

After save/reconnect the structure in Aras Innovator the CATDrawing activates the "Update current Sheet" button that indicates that the CATDrawing needs an update (see *Picture 188: Updating the current sheet*).

	B CATTA VS Sant Elle Edit View Jouent Tools Window Help	2		
	◆◆冬/く井比冬 ●白豆類時間 ●田田時間間第二	B / S S X E A	V Contraction	
	Contracting CATDrawing			
	Construction C			
	1	Originations Sectorem Originations	SATZI ————————————————————————————————————	
□=====×:◎≤>:(#:):60:==●:(3:==):2:=====:0:(3:======0):(3:======0):(3:======0):(3:======0):(3:=====0):(3:======0):(3:======0):(3:======0):(3:=======0):(3:========):(3:======):(3:======):(3:======):(3:======):(3:======):(3:=======):(3:========):(3:========):(3:========):(3:=========):(3:==========	L 🗃 🖬 🎝 🕈 🔊 🕫 👘 🖗 👘 🖷 🌒 🖗 👘 🛛 🗮 🖉 🔍 🔍 🖾 💆 🧎 🛄	は 🖇 🖻 🚺 🖩 無四牧 🔒 🖴 🖣 🥝		PSATU Dearny

Picture 188: Updating the current sheet

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

Use the PDM Workbench function "Update" to save the CATDrawing in Aras Innovator. 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 structure.

For this you have to click on the "PDM Structure" icon ¹. Then a PDM Workbench window with the PDM structure will be opened (see *Picture 189: PDM structure for geometry*).



Picture 189: PDM structure for geometry

Please note that with BOM part structures, the expanded PDM structure (window in the middle) usually does not contain the related CATIA files, but the PDM 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 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 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 190: PDM structure and geometry in CATIA V5*).



Picture 190: PDM structure and geometry in CATIA V5

In the CATIA V5 you can make some changes in the geometry (see *Picture 191: Making changes in the geometry*) that make the objects in the PDM structure dirty. This state change will not be displayed automatically.



Picture 191: 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 structure will be updated.

Now e.g. the dirty object owned by the session user will be displayed in yellow in the PDM 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 192: Refreshed PDM structure*).



Picture 192: Refreshed PDM structure

"Refresh" is active in the Main Toolbar

This also applies to CATPart and CATDrawing windows.

That makes it possible to refresh the Claimed/Unclaimed status of a CATPart or CATDrawing CAD Document in the CATIA session.

PDM Status Information in the CATIA Tree

A PDM status information can be displayed on product structure nodes within the CATIA tree. These nodes show an additional icon mask and additional text and tooltip text corresponding to some information from the PDM system.

If configured by the administrator, a PDM status information is displayed on product structure nodes within the CATIA tree:



Picture 193: PDM status information in the CATIA tree

These nodes show an additional icon mask in the upper right corner of the icon and additional text and tooltip text corresponding to some information from the PDM system.

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 claim the PDM object. Or you have selected an object in the PDM 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 194: Action "Highlight PDM Nodes"*).



Picture 194: 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 195: 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 195: 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 PDM structure, e.g. in order to claim the PDM object. Or you have selected an object in the geometry and want to see the corresponding object in the PDM structure.

For this you can select an object in the PDM structure and click on the right mouse button. The context menu opens and you have to choose *PDM Workbench*→*Highlight CATIA Nodes* (see *Picture 196: Action "Highlight CATIA Nodes"*).



Picture 196: 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 structure will be highlighted (marked) (see *Picture 197: 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 197: 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 198: Action "Open in New Window"*).

📓 Engine, A, 1, , Preliminary (logged in as admin, database: InnovatorSolutions1112)								
Findine A 1 Preliminary	Fraince A.1. Proliminant							
ConnectionRod.1 / ConnectionRod.1	ConnectionRod.1 / ConnectionRod, A, 1, , Preliminary							
ConnectionRodModel	.1 / ConnectionRodMod	lel, <mark>A, 1</mark> , , Prel	iminary					
Crankshaft Assm.1 / C								
	C <u>e</u> nter graph							
CrankPin.1 / Crank	Reframe On							
CrankPin Mode	Hide/Show	Alt. Enter						
	Open Sub-Tree	Alt+Enter						
	Cut	Ctrl+X						
CrankshaftLeft	Сору	Ctrl+C	liminary					
CrankshaftRight.1	<u>P</u> aste	Ctrl+V						
	Paste <u>S</u> pecial		Preliminary					
	Delete	Del						
Cylinder_Assm.1 / Cyli	Crankshaft Assm A 1 Declimina	ny object						
Piston.1 / Piston, /	Clankshart_Assin, A, 1, , Freiming	ily object						
Piston Model 1	Expand	•						
	Expand Multiple Levels	•						
PistonPin_Assm.1 , 📙	De_Expand		Z					
	Compare with Assembly		x y					
E	Open in new PDIM Window							

Picture 198: Action "Open in New Window"

The selected objects will be opened in a new window (see *Picture 199: Selected objects in the new window*).



Picture 199: Selected objects in the new window

Comparing PDM Structure Trees

It is possible to compare two structures, or two generations of the same structure, displaying the differences between these two structures.

You start with two expanded structures in one or two PDM structure windows. First, you select and copy (CTRL-C) the root node of one of the structures. Then you right-click on the root node of the other structure and select the context action "Compare with Assembly". This opens the window which displays the differences between the two structures.



Picture 200: Two CAD Document structures to be compared



Picture 201: Window containing the differences between the two structures

Selecting Nodes in the PDM Structure Window

It is possible to select specific child nodes of a structure by defining logical AND or OR combinations of values of different attributes.

You right-click on the root node of an expanded structure and select the context action "Select Nodes". The dialog where you can enter the attribute values appears:



Picture 202: "Select Nodes" dialog

You can choose whether the Boolean condition with which the attribute values are combined is AND (the default) or OR.

After clicking the "OK" button the child nodes whose attribute values match the defined criteria are selected:



Picture 203: Selected Nodes

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 204: Action "Force Load CATPart").



Picture 204: Action "Force Load CATPart"

Insert from PDM

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

You right-click on an existing CATProduct node in a CATIA Structure window and select "Insert PDM Node" (see *Picture 205: Action "Insert PDM Node"*).

Sensitive CATProduct	
FixedParts_Assm (FixedParts_Assm.1)	<i>ф</i>
CylinderBlock (CylinderBlock.1)	
Rack_Assm (C <u>e</u> nter graph Rack_Rack (Rac Beframe On Hide/Show Screw (Sc	
Screw (Sc	
Paste Ctrl+V Constrair Paste Special	
Cylinder_Assm (
+- PDM Workbench +- Connection Rod Components → Crankshaft_Assr Representations	PDM Properties PDM Create in Context Elighlight PDM Nodes
CrankshaftRi Selection Mode	Claim
CrankshaftLeft (CrankshaftLeft.1)	Id* Insert PDM Node XX Id* Replace Node X

Picture 205: Action "Insert PDM Node"

A restricted "PDM Query" dialog opens. In the CAD Document Structure Data Model CATPart and CATProduct items can be queried; in the BOM Part Structure Data Model Part items can be selected (see *Picture 206: Insert PDM Node – "PDM Query" dialog type selection*).

PDM Query for Insert/Replace						? ×
PDM object type	Part Number	Major Rev.	Generation	Name	State	De
Part						
Part						
Assembly						
Std Dart						
Major Kev.	-					
Generation						
Name 👻						
State						
Unit						
Make / Buy						
Description						
Created on						
Modified on						
Created by						
Modified by						
Locked by						
	•	III				F
					Query	Close

Picture 206: Insert PDM Node – "PDM Query" dialog type selection

After the query is performed all resulting items are displayed in the result list, like in the regular "PDM Query" dialog (see *Picture 207: Insert PDM Node – query result*).

PDM Query for Insert/Replace						? ×
PDM object type	Part Number	Major Rev.	Generation	Name	State	De
Part 🔻	Screw	A	1		Preliminary	
PDM object attributes	稔 ScrewModel	А	1		Preliminary	
Part Number Screw*						
Major Rev. 👻						
Generation 👻						
Name						
State 👻						
Unit						
Make / Buy						
Description						
Created on						
Modified on						
Created by						
Modified by						
Locked by						
	•					•
					Query	Close

Picture 207: 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 (see *Picture 208: Item inserted in existing structure*).



Picture 208: 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.

Insert from Aras Innovator keep "PDM Query" Dialog

The "PDM Query" dialog stays open. The user can insert multiple items from the same "PDM Query" dialog.

Replace from PDM

The node selected in the CATIA structure can be replaced by the CATIA file which corresponds to an existing Aras 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 PDM 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 209: "Impacts on Replace" standard CATIA dialog*).

CATIA V5 - [Engine.CATProduct]	
Start File Edit View Insert Tools CMI Analyze Window Help	- 6 x
' <mark>~4X</mark> ASA68A'~6 6 4'®@\$\$\$\$	
Fingine Engine	
+	· · · · · · · · · · · · · · · · · · ·
Crankshaft (Crankshaft.1)	
- Constraints	
Coincidence.1 (ConnectionRod.1,Crankshaft.1)	
Coincidence.2 (ConnectionRod.1,Cylinder.1)	
Offset.3 (ConnectionRod.1,Crankshaft.1)	sacts On Replace
Offset 4 (ConnectionRod 1 Cylinder 1)	hese elements are going to be impacted by the replace command. Do you want to conti Affected
	Type Name Source links
Coincidence.5 (Crankshaft.1,FixedParts.1)	CONNECTION Coincidence.2 Engine CONNECTION Offset.6 Engine
Offset.6 (ConnectionRod.1,FixedParts.1)	CONNECTION Offset.4 Engine CONNECTION Offset.3 Engine
- Offset.8 (Cylinder.1,FixedParts.1)	
Fix.9 (FixedParts.1)	o you want to replace all the instances of the selected element ?
Coincidence.10 (Crankshaft.1,FixedParts.1)	Yes O No
Offset.11 (Crankshaft.1,FixedParts.1)	Cancel Cancel
+ Applications	, z
Do you want to continue the replacement?	

Picture 209: "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 210: Constraints destroyed by "Replace" operation*).



Picture 210: 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:

You right-click on an existing CATProduct or CATPart node in a CATIA structure window and select "Replace Node" (see *Picture 211: Action "Replace Node"*).



Picture 211: Action "Replace Node"

As in the "Insert PDM Node" case, a restricted query window opens. You 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 (see *Picture 212: Select replacing Node*).

M object type		Part	lumber	Major Rev.	Generation	Name	State	D
ırt		🗘 🗘 Screw		A	1		Preliminary	
M object attributes		🔅 Screw	Model	А	1		Preliminary	
rt Number Screw*	-	Screv	ModelRed	А	1		Preliminary	-
Major Rev.	-							
Generation	-							
Name	-							
State	<u> </u>							
Unit	-							
Inter (Burg	_							
hake / buy	•							
rescription								
Created on	-							
odified on	-							
Created by	-							
lodified by	_							
Locked by	-							
counce by								
		•						

Picture 212: Select replacing Node

Then you get to decide whether only the selected node or all instances of the document should be replaced (see *Picture 213: "Replace all instances" prompt*).



Picture 213: "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 File Name as the file to be replaced, because both files would be located in the same directory (PDM Workbench exchange directory).

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.

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 Document Structure Data Model*).

You click on the "Synchronize to BOM" context menu on the CATProduct document (see *Picture 214: Action "Synchronize to BOM"*).

🛐 Crankshaft_Assm, A, 1, , Preliminar	(logged in as admin, database: InnovatorSolutions1112)	
Crankshaft_Assm, A, 1, Preliminar Crankshaft_Assm, A, 	(logged in as admin, database: InnovatorSolutions1112) Cgnter graph Beframe On Hide/Show Properties Alt+Enter Open Sub-Tree Gut Ctrl+X Sopy Ctrl+C Paste Special Delete Del Crankshaft, Assm, A, 1, , Preliminary object Egpand •	eliminary Model, A, 1, , Preliminary t, A, 1, , Preliminary hkshaftLeftModel, A, 1, , Preliminary ght, A, 1, , Preliminary ankshaftRightModel, A, 1, , Preliminary
	Expand Expand Multiple Levels De_Expand Open in new PDM Window Unclaim Claim Unclaim all Gaim all Promote Regise Update structure relations	
	Load Load with Links Load with Links Select Nodes Load remp Highlight CATIA Nodes Synchronize to BOM Create new version Duplicate Duplicate Structure	×Čy

Picture 214: Action "Synchronize to BOM"

You have to confirm the synchronization to BOM (see *Picture 215: Action "Synchronize to BOM" – Confirmation*).



Picture 215: Action "Synchronize to BOM" – Confirmation

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

The resulting structure can be expanded in the PDM Structure window (see *Picture 216: Created or updated PDM structure*).



Picture 216: Created or updated PDM structure

This functionality is also available in the Aras Innovator web client, too:

	NewProduct3	Δ	_			Mechanical/Asse
1			Z	Open		incontantour, robotin
			☆	Favorite		
			 	Unclaim		
			C	Promote		
			*	Navigate	>	
			Ê	Reports	>	
			<	Share	>	
			•••	More	>	Properties
			4,	Admin	>	Create New Version
						Delete >
						Add Item(s) To Change
						Manual Release
						Create New Revision
						Change Discussion Definition
						Add Item to Bookmarks
						Open In CATIA
						Open In Creo
						Open In NX
						Synchronize in BOM

Picture 217: "Synchronize in BOM" in Aras Innovator web client

Attribute Mapping

CATPart and CATProduct

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 218: Part mapping – Standard attributes in the "Properties" dialog* and *Picture 219: Part mapping – Configuration of standard attributes in Aras Innovator*).

Properties					?	x
Current selection	n: NewProduct1					-
Graphic Pr	roduct Mechanical	Drafting				
Product						
Part Number	NewProduct1				_ 🎽	
Revision						
Definition						
Nomenclature	NewProduct1 - Nomenc	lature				
Source	Unknown 🔻					
Description	NewProduct1 - Description	on				
Define other p	oroperties					
					More	
			OK OK	Apply	Clos	e

Picture 218: Part mapping - Standard attributes in the "Properties" dialog

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File Edit Views Search Actions Reports Tools Help
Name
Configuration
Beschreibung
.4
Cattions Chandrad Attributes User Attributes Device Attributes Deviced Files
Securitys Scandard Actibuces Osel Actibuces Drawing Actibuces Derived Files
Actions 🔻 🛛 Pick Related 🔤 📮 🎊 🗶 🖾 🖾 🖬 🖉 🔮 💼 🔗 🗛 👧 🕅 Hide Search Criteria 🔽
Cad Name Innovator Name Mapping Direction ItemType
CadNomenclature name Cad < - > Innovator Part
CadReferenceDescription description Cad < - > Innovator Part

Picture 219: Part mapping – Configuration of standard attributes in Aras Innovator

In the "PDM Create" dialog the attributes are pre-filled (see *Picture 220: Part mapping – Pre-filled "PDM Create" dialog*).

PDM Create	? 🛁	x
Create dialog NewProduct1	for type Assembly:	
Part Number	NewProduct1	*
Name	NewProduct1 - Nomenclature	
Unit		
Make / Buy		
Description	NewProduct1 - Description	
Create dialog NewProduct1	for type CATProduct:	
Doc. Number	NewProduct1	*
Name		
Description		
	OK Apply Gance	el

Picture 220: Part mapping – Pre-filled "PDM Create" dialog

After creating the part with Update the defined CATIA attribute values have been written to the PDM part object (see *Picture 221: Part mapping – Standard attributes in the "Properties" dialog of the PDM node* and *Picture 222: Part mapping – Standard attributes in Aras Innovator*).

Properties Up	date Item	
Assembly		-
Part Number	NewProduct1	
Major Rev.	A	
Generation	1	
Name	NewProduct1 - Nomenclature	
State	Preliminary	
Unit		
Make / Buy		
Description	NewProduct1 - Description	1
		=
New Version	7	
Туре	Assembly	
Assigned Creator		
Designated User		
Created by	Innovator Admin	
Created on	2018-03-22T16:17:15	
Modified by	Innovator Admin	
Modified on	2018-03-22T16:17:15	
Locked by	Innovator Admin	
Release Date		
Effective Date		

Picture 221: Part mapping – Standard attributes in the "Properties" dialog of the PDM node

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(j 192.168.168.168/InnovatorServer1112/Client/X-salt=std_11.0.0.6920-X/scripts/Innovator.aspx					
aras NewProduct1 × File Edit Views Search ↓ □ ↓ ↓ ↓ ↓ ↓	CR® Actions Reports Tools Help I IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII				
Part	Part Number Revision State	Assigned Creator			
	NewProduct1 A Preliminary				
1 1 1	NewProduct1 - Nomenclature	····			
Created By: Innovator Admin Created On: 3/22/2018 Modified By: Innovator Admin Modified On: 3/22/2018 Locked By: Innovator Admin Major Rev: A Release Date: Effective Date: Generation: 1 State: Preliminary	Type Unit Make / Buy Cost Assembly Long Description NewProduct1 - Description 	Effective Date			
Changes Pending BOM BOM Structure Altern	ates AML Documents CAD Documents Goals Chang	es Part Submission Warran			
Actions - Pick Related -		Hide Search Criteria			
Sequence A Part Number	Revision Name Ty	pe Quantity			

Picture 222: Part mapping – Standard attributes in Aras Innovator

In the next example the standard CATIA attributes "Nomenclature" and "Description" are mapped to the attributes "name" and "description" of the Aras Innovator document object (see *Picture 223: CAD Document mapping – Standard attributes in the "Properties" dialog* and *Picture 224: CAD Document mapping – Configuration of standard attributes in Aras Innovator*)
Properties		? <mark>x</mark>
Current selection :	NewProduct2	Ţ
Creatia Dead	Aust Mashaninel Durking	
Product		
Part Number	lewProduct2	_₿
Revision		_
Definition		
Nomenclature N	JewProduct2 - Nomenclature	
Source U	Inknown 🔻	_
Description	lewProduct2 - Description	
Define other pro	operties	
		More
	OK Apply	Close

Picture 223: CAD Document mapping – Standard attributes in the "Properties" dialog

🥹 Aras Innovator - Mozilla Firefox
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File Edit Views Search Actions Reports Tools Help
[] 🖹 🔽 🔗 🖶 💵 🚇 🔒 🔐 ສ 🖓 🖑 🖆 🥐
Name
Configuration
beschreibung
Settings Standard Attributes User Attributes Drawing Attributes Derived Files
Actions 🗸 🛛 Pick Related 🔽 📮 🌮 🗙 🖾 😻 🖉 🔒 🔐 🎸 🔗 National Hide Search Criteria 🔽
Cad Name Innovator Name Mapping Direction ItemType
CadNomenclature name Cad < - > Innovator CAD Document
CadReferenceDescription description Cad < - > Innovator CAD Document

Picture 224: CAD Document mapping – Configuration of standard attributes in Aras Innovator

In the "PDM Create" dialog the attributes are pre-filled (see *Picture 225: CAD Document mapping – Pre-filled "PDM Create" dialog*).

PDM Create	?
Create dialog f NewProduct2	or type Assembly:
Part Number Name	NewProduct2 *
Unit	•
Make / Buy	•
Description	
Create dialog f NewProduct2	or type CATProduct:
Doc. Number	NewProduct2 *
Name	NewProduct2 - Nomenclature
Description	NewProduct2 - Description
	OK Apply Cancel

Picture 225: CAD Document mapping – Pre-filled "PDM Create" dialog

After creating the part with Update the defined CATIA attribute values have been written to the CAD Document item (see *Picture 226: CAD Document mapping – Standard attributes in the "Properties" dialog of the PDM node* and *Picture 227: CAD Document mapping – Standard attributes in Aras Innovator*)

Part Number	NewProduct2	*	Doc. Number	NewProduct2
Major Rev.	A		Major Rev.	A
Generation	1		Generation	1
Name			Name	NewProduct2 - Nomenclature
Unit		-	Description	NewProduct2 - Description
Make / Buy				
Description				
			Туре	Mechanical/Assembly
			Is Template	
Туре	Assembly		Standard Part	

Picture 226: CAD Document mapping – Standard attributes in the "Properties" dialog of the PDM node

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(192.168.168.168/InnovatorServ	ver1112/Client/X-salt=std_11.0.	0.6920-X/scripts/Innovator.as	spx
aras ■ NewProduct2 × File Edit Views Search □ □ □ ↓ □ ↓ ○ □ ■	CR® Actions Reports Took	s Help ° [↑] ? ✓	
CAD Document Description: Created By: Innovator Admin Created On: 3/22/2018 Modified By: Innovator Admin Modified On: 3/22/2018 Locked By: Innovator Admin Major Rev: A Release Date: Effective Date: Generation: 1 State: Preliminary	Document Number Retr NewProduct2 A Name NewProduct2 - Nomenclature Aur Type Aur Assembly Description CA NewProduct2 - Description Assigned Creator Part NewProduct2	rision State Preliminary thoring Tool Version TIA VISR27 Designated User From Template	Select an image Native File Native File Select file Select file
Changes Pending Standard Template			
Structure Parents Files C Actions ▼ Pick Related ▼ Sequence Document Numb	changes CAD Design Table	Name	Hide Search Criteria Type

Picture 227: CAD Document mapping – Standard attributes in Aras Innovator

User-defined CATIA properties can also be mapped (see *Picture 228: Configuration of user-defined attributes in Aras Innovator*).

The standard attribute mapping is defined additionally (see *Picture 224: CAD Document mapping – Configuration of standard attributes in Aras Innovator*).

🥹 Aras Innovator - Mozilla Firefox
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INNOVATOR®
File Edit Views Search Actions Reports Tools Help
Name
Configuration
Beschreibung
Settings Standard Attributes User Attributes Drawing Attributes Derived Files
Actions 🗸 Create Related 🔽 📮 🎋 🗙 🖾 🚳 🔒 😭 % 📯 🔗 Hide Search Criteria 🔻
Cad Label Innovator Name Mapping Directi ItemType
CAD Name name Cad < - > Innov CAD Document
CAD Description description Cad < - > Innov CAD Document

Picture 228: 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 item (see *Picture 231: User-defined attributes mapping – Standard Attributes in the "Properties" dialog of the PDM node and Picture 232: User-defined attributes mapping – Standard attributes in Aras Innovator*).

Pr	roperties		?	x
	Current selectior	: NewProduct3		-
	Product G	raphic Mechanical Drafting		
	1			
	Desiduat			
	Product Part Number	NewProduct3		9
	Revision	Rewritedeeb		•
	Definition		-	
	Nomenclature	NewProduct3		
	Source	Unknown 👻		
	Description	own description		
	Define other r	properties		
			M	ore
		OK Apply		lose
	1000		_	

Picture 229: User-defined attributes mapping – Standard attributes in the "Properties" dialog

In the "PDM Create" dialog the attributes for the CAD Document are pre-filled (see *Picture 230: User-defined attributes mapping – Pre-filled "PDM Create" dialog*).

PDM Create		?	×
Create dialog f NewProduct3	or type Assembly:		
Part Number Name Unit	NewProduct3		_*
Make / Buy			-
Description			
Create dialog f NewProduct3	for type CATProduct:		
Doc. Number	NewProduct3		*
Name	NewProduct3		
Description	own description		
	OK Apply	۵ د	ancel

Picture 230: User-defined attributes mapping – Pre-filled "PDM Create" dialog

After creating the Part with Update the defined CATIA attribute values have been written to the CAD Document item (see *Picture 231: User-defined attributes mapping – Standard Attributes in the "Properties" dialog of the PDM node and Picture 232: User-defined attributes mapping – Standard attributes in Aras Innovator*).

Assembly (BOM)					? <mark>×</mark>
No PDM Relation	Assembly -			CATProduct	
Pa	art Number	NewProduct3	*	Doc. Number	NewProduct3 *
	Major Rev.	A		Major Rev.	A
	Generation	1		Generation	1
	Name			Name	NewProduct3
	Unit	•		Description	own description
1	Make / Buy	•			
C	Description				
				Туре	Mechanical/Assembly
				Is Template	
	Туре	Assembly		Standard Part	
					OK Scancel

Picture 231: User-defined attributes mapping – Standard Attributes in the "Properties" dialog of the PDM node

Aras Innovator - Mozilla Firefox			
(1) 192.168.168.168/InnovatorServe	r1112/Client/X-salt=std 11.0.	0.6920-X/scripts/Innovator.a	spx
INNOVATO			
	🚺 NewProduct3 ×		
File Edit Views Search	Actions Reports Tool	s Help	
		° 📩 ? 🗸	
CAD Document	Document Number Rev	vision State	Select an image
	NewProduct3 A	Preliminary	
	Name		
	NewProduct3		1
Created By: Innovator Admin	Type Au	thoring Tool Versi	on
Created On: 3/23/2018	Assembly CA	TIA V5R27	7
Modified By: Innovator Admin	Description		
Locked By: Innovator Admin	own description		1
Major Rev: A		4	
Release Date:	Assigned Creator	Designated User	Native File
Effective Date:	•••	•	• <u>NewProduct3.C</u>
State: Preliminary	Part	From Template	Viewable File
,	NewProduct3 👫		.t. Select file
Changes Pending			
Standard			
Template			
Structure Parents Files Ch	anges CAD Design Table		
Actions 👻 Pick Related 💌	+ 🖗 🗙 🖾	A ^2 🗎 🗎 🕯	Aide Search Criteria 🔻
Sequence Document Numbe	r 🛦 Revision	Name	Type State

Picture 232: User-defined attributes mapping – Standard attributes in Aras Innovator

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 233: User-defined attributes in the "Properties" dialog*).

Properties			? X
Current selection	n : NewProduct3		-
Product G	raphic Mechanical	Drafting	
	ispine [incension]		
Product	-		
Part Number	NewProduct3		
Revision			
Definition			
Nomenclature	NewProduct3		
Source	Unknown		
Description	own description		
- Product: Add	led Properties		
CAD News	led Properties	New Decident 2	
CAD Name		NewProduct3	1.0.00
CAD Descripti	on	own description	
Define other p	properties		
s set strand.			annan an a
			More
		OK Annly	Close
COMP.		Ск	Close

Picture 233: 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.

CATDrawing

PDM attribute values of the drawing CAD Document item can be mapped to CATDrawing attributes (see *Picture 234: Drawing attributes mapping – Configuration of Drawing attributes in Aras Innovator*).

🕹 Aras Innovator - Mozilla Firefox
() 192.168.168.168/InnovatorServer1112/Client/X-salt=std_11.0.0.6920-X/scripts/Innovator.aspx
aras INNOVATOR®
[■ 77F7BC4E822E43E4 × [■ Engine-Drw1 ■ CAD 至 4941DA9BEF474C47
File Edit Views Search Actions Reports Tools Help
Name
Configuration
Settings Standard Attributes User Attributes Drawing Attributes Derived Files
Actions 🔻 Create Related 🔄 📮 🎊 🗙 🖾 🚳 🔒 💼 🕈 🍄 😣 🔗 Hide Search Criteria 🔽
CAD Name Innovator Name Mapping Directi ItemType
AttrDrw1 item_number Cad <- Innovator CAD Document
AttrDrw2 name Cad < - > Innov CAD Document
AttrDrw3 major_rev Cad <- Innovator CAD Document

Picture 234: Drawing attributes mapping – Configuration of Drawing attributes in Aras Innovator

After the drawing is created in PDM the CATDrawing file contains the attributes defined in the Drawing Attribute definition in the Aras Innovator configuration (see *Picture 235: Drawing attributes mapping – CATDrawing attribute mapping*).



Picture 235: Drawing attributes mapping – CATDrawing attribute mapping

You can modify one of the drawing attributes (see *Picture 236: Drawing attributes mapping – Modify drawing attribute value*).

AttrDrw2	Engine-Drw1 New Name
----------	----------------------

Picture 236: Drawing attributes mapping – Modify drawing attribute value

The modified attribute value is stored in CATIA V5 (see *Picture 237: Drawing attributes mapping – Modified drawing attribute value*).

Sengine-Drw1.CATDrawing (Engine,	A, 1, , Preliminary)
Engine-Drw1.CATDrawing (Engine, Engine-Drw1 Parameters AttrDrw1 B AttrDrw1	Sheet.1 Engine-Drw1 Engine-Drw1 New Name
AttrDrw3 Sheet.1 Front view Rear view	
Tengine-Drw1, A, 1, Engine-Drw1 Na	ne, Preliminary (logged in as admin, database: InnovatorSolutions1112) ine-Drw1 Name Preliminary

Picture 237: Drawing attributes mapping – Modified drawing attribute value

After a PDM update, the new attribute value is written into its corresponding PDM attribute (see *Picture 238: Drawing attributes mapping – Modified PDM attribute value*).

🛐 Engine-Drw1, A, 1	I, Engine-Drw1 New Name, Prelimina	ry (logged ir	n as admin, database: Inn
🗟 Engine-Dr	w1, A, 1, Engine-Drw1 Ne	w Name	Preliminary
Properties			? ×
Current selection :	Engine-Drw1, A, 1, Engine-Drw1 New	Name, Preli	minary 👻
Properties Up	date Item		
CATDrawing		1	
Doc. Number	Engine-Drw1		
Major Rev.	A		
Generation	1		
Name	Engine-Drw1 New Name		
State	Preliminary		
Description			

Picture 238: Drawing attributes mapping – Modified PDM attribute value

If, on the other hand, you change 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 239: Drawing attributes mapping – PDM attribute value modified from Aras Innovator*).

	Engine-Drw1,	A, 1, Engine-Drw1 New Name, Preliminary (logged in as admin, database: Inno
	🖹 Engine-	Drw1, A, 1, Engine-Drw1 New Name, Preliminary
Pr	operties	? <mark>×</mark>
(Current selection	n : Engine-Drw1, A, 1, Engine-Drw1 New Name, Preliminary
	Properties	Update Item
	CATDrawing	
	Doc. Number	Engine-Drw1 *
	Major Rev.	A
	Generation	1
	Name	Engine-Drw1 Name from PDM
	Description	
	Ture	
	Is Template	Mechanical/Urawing
	is remplate	

Picture 239: Drawing attributes mapping – PDM attribute value modified from Aras Innovator

You can see the PDM attributes in the parameters of the CATDrawing (see *Picture 240: Drawing attributes mapping – Drawing attribute value changed to PDM attribute value*).



Picture 240: Drawing attributes mapping – Drawing attribute value changed to PDM attribute value

Inertia attributes mapping

You can configure the server to let the CAD client calculate and provide some inertia attribute values (like the center of gravity) when the CAD file is uploaded to the PDM system. These values can be mapped to properties of your CAD or Part items in Aras Innovator.

If an inertia attribute mapping is configured on the server, the calculation of inertia values is performed automatically during the "Update" process whenever a CATPart was created or modified. Inertia values are not calculated for other document types.

Take care that the calculation of the mass can only be accurate if you provide the correct material in your part definition (see *Picture 241: Sample material definition in CATIA*).



Picture 241: Sample material definition in CATIA

Here are sample pictures of the results of an inertia calculation in CATIA and a mapping to CAD Document item properties in Aras Innovator (see *Picture 242: CATIA tree and inertia properties and Picture 243: Inertia properties mapped to Aras Innovator*):

CATIA V5		_
<u>Start</u> <u>File</u> <u>E</u> dit <u>V</u> iew <u>I</u> nsert <u>T</u> ool	<u>W</u> indow <u>H</u> elp	
S S & S S S S S S	<u> </u>	
C_895_Product01.CATProduct (AsSaved)		
🐝 C_895_Product01		
- C 895 Part01 (C 895 Part01.1)	Properties	? ×
C 895 Part01	Current selection : C_895_Part01/C_895	_Part01.1/C_895_Product01
	Product Graphic Mechanical I	Mass Colors
	General	ter of Gravity
yz piane	Density: 8900kg_m3 x= 30	Omm
zx plane	Volume: 2,16e-005m3 Mass: 0.192kg Z= 4	965mm
🗣 🐺 PartBody	Surface: 0,007m2	
Copper	Inertia Matrix	
+-• C_895_Part01 (C_895_Part01.2)	IXX= 1,572e-005kgxm2 IXY= 0kgx	m2 Ixz= 0kgxm2
Applications	Izx= 0kgxm2 Izy= 0kgx	m2 Izz= 1,169e-004kgxm2
	Only main body	
		More
		OK Apply Close

Picture 242: CATIA tree and inertia properties

<u>9</u>					Ara	s Innovator - Inter	net Explorer								L	
INNOVAT	ORº											LOCAL	L TIME lay, March 8, 201	3:19)" 2	nnovator Adr <u>eqcut</u>
ile Edit View Search	Actions Rep	orts Tools H	telp													
7 G 🖓 🗎 🛛 🖈	9 🛛 🗯	i 🔒 💕 5	C 🖸 ?										1	Search Items &	Files	P
Contents (Properties		AAA	Simple Search	 Image: S0 	tā 👪 🗔	Current	Today						р	review: Properti	ies - 🖡
	CAD Down		Type	State	Native File []	Viewable File []	Authoring Tool	Changes	Standard	Template	Mass	COGX	COGY	COGZ	Area	Volume
Administration	CAD Docun	nenc					-									
Change Management			Mechanical/Part	Preliminary	C 895 Part01.CAT		CATIA				0.19224	0.03	0.000965065	0.0045	0.00678	0.0100216
Dashboards			Mechanical/Assembly	Preliminary	C 895 Product01.C.		CATIA									
Design	Created By:	2/7/2018														
 Documents 	Created Un: Modified Ibc	Innevator Admin														
CAD DesignTa	Modified On:	3/8/2018														
GAD Documents	Locked By:	Innevator Admin														
Documents Y	Major Rev: Release Date:	A	<													>
eatv	localh	ost		InnovatorSolutio	rø1112	Items 1-2.Page 1. 0	f ()									Messages:

Picture 243: Inertia properties mapped to Aras Innovator

PDM to CAD Attribute Mapping only for CATIA Files claimed by the User

Changed PDM attribute values should only be written into CATIA files that are claimed by the user. This behaviour should be optional.

Allow mapping of Part and CAD Property to the same CATIA Standard Attribute

By default, you can either map a Part or a CAD Document property to a CATIA standard attribute like Nomenclature, Definition, Revision, and Reference Description. This causes a problem if you work in BOM Part Structure Data Model with additional non-BOM CATIA files. For instance, if you map the value of the Part property "name" to the CATIA attribute "Nomenclature" you cannot map any CAD Document property to the same CATIA attribute. This means the CATIA attribute "Nomenclature" cannot be controlled by any Aras Innovator property for a non-BOM CATIA file.

This functionality allows to define a property mapping for both types:

~	Settings	Standard Att	ributes	Inertia At	tributes	User Attributes	Drawing Attribu	ıtes		
🔵 Standard Attribute 🗸 🏠										
6	🕫 🍈 🕞 🖳 🛛 🕰 Hidden 🗸 💽 - 🕎 - 🗣 -									
E	Cad Na	me	Innovat	or Name	Mapping	Direction	ItemType			
	CadNor	menclature	name		Cad < - >	Innovator	CAD Document			
	CadNor	menclature	name		Cad < - >	Innovator	Part			

Picture 244: Mapping of CATIA attribute Nomenclature from CAD and Part

The mapping of the Part property is handled with a higher priority. This means only if there is no Part, the mapping of the CAD will be used.

The mapping direction from CATIA to Aras Innovator is not affected.

Internal CATIA Information can be written to user-defined CATIA Properties

The CATIA internal attributes have to be defined in the PDM Workbench Schema file to be copied to the user defined attributes in CATIA. They are marked in red and can be checked in the "Properties" dialog of the CATIA object (see *Picture 245: User-defined attributes with internal CATIA information in "Properties" dialog*).

Properties		? 7	x
Current selection : NewPart1			Ŧ
Product Graphic Mechanical	Mass Colors Visualization		
Deaduret			
Product Part Number NewPart1		A	
Revision		- •	
Definition			
Nomenclature NewPart1 - Nomenclat	ture		
Source Unknown			
Description NewPart1 - Description	ו		
Product: Added Properties			
CAD Document Name			
CATIA Environment	V5R27		
The Mass	0.028		
The Volume	2.80.005		
The Denit	1000		
The Density	1000		
The Area	0.0064	_	
Center of Gravity X	0.035		
Center of Gravity Y	0.01		
Center of Gravity Z	0.01		
Define other properties			
		More.	
	OK Amelal	Clara	
		Close	

Picture 245: User-defined attributes with internal CATIA information in "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 246: Configuration of user-defined attributes in Aras Innovator*). The "CAD Document Name" is also mapped to an attribute in Aras Innovator; but it is no internal CATIA attribute.

🥹 Aras Innovator - Mozilla Firefox
() 192.168.168.168/InnovatorServer1112/Client/X-salt=std_11.0.0.6920-X/scripts/Innovator.aspx
aras INNOVATOR®
File Edit Views Search Actions Reports Tools Help
Name
Configuration
Settings Standard Attributes User Attributes Drawing Attributes Derived Files
Actions 🔻 🛛 Create Related 🔽 📮 👘 💥 🔯 👰 🔒 🔐 💞 🗛 🚕 🕟 Hide Search Criteria 💌
Cad Label Innovator Name Mapping Direction ItemType
CATIA Environment pwb_catia_environment Cad -> Innovator CAD Document
CAD Document Name pwb_name Cad < - > Innovator CAD Document

Picture 246: Configuration of user-defined attributes in Aras Innovator

Create CAD Document and Part with Templates

In the dialog which appears after you click on the toolbar action "Create" the type of the new object can be selected (see *Picture 247: Select a PDM type for the "PDM Create" dialog*).



Picture 247: Select a PDM type for the "PDM 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 248: "PDM Create" dialog for CATPart – Select Template*).



Picture 248: "PDM 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 249: "PDM Create" dialog for CATPart in BOM Part Structure Data Model*) or the document object to be created (see *Picture 250: "PDM Create" dialog for CATPart in CAD Document Structure Data Model*).



Picture 249: "PDM Create" dialog for CATPart in BOM Part Structure Data Model

Update to VUNIC					No.	
rachane New CATPart	C.A.B.Arger Fundeer	PCM Create Create stilling fort New Dec Number Tie Norme Decorption Standard Part Standard Part	соска у Р. н. с уре САТРан: м ОК Э. Арру . Э. Селос	Modified	veze Creater II. Oreate PDM File Object of typ.	,
Create new CAD docume	ent generation?	Create corresponding BOM pa	tan 100		<u>Issued</u>	Close

Picture 250: "PDM Create" dialog for CATPart in CAD Document Structure Data Model

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 251: Created Part*).



Picture 251: Created Part

Manage CATIA Templates in Aras 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 252: "Template File" functionality – Creating a CATPart).



Picture 252: "Template File" functionality – Creating a CATPart

If templates are configured, then you get to choose a Template File Name (see *Picture 253: "Template File" functionality – Selecting a Template File*).



Picture 253: "Template File" functionality – Selecting a Template File

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

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

This use case is applicable when using the BOM Part Structure Data Model ("UseBomPartStructure" is set to "true").

Example:

Create an Assembly (see *Picture 254: "Template File" functionality – Creating an Assembly*).

Se	lect a PDM type for perfor
Ē	PDM Object Types
	Assembly Component Std. Part CATPart CATProduct CATDrawing catalog
	OK SCancel

Picture 254: "Template File" functionality – Creating an Assembly

Fill all the necessary Assembly information on the "PDM 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.

Template File Support for "Create Part" with Templates depending on the Part Type

Originally the "Create" functionality only creates the "Part" business item in the database when a part type is selected, without a corresponding CAD Document item.

An extension of the "Create" functionality allows to create Part items with their corresponding CAD Document, and the native file of this CAD Document can be based on a list of Template Files which is defined specifically for this part type.

These Part items can also be created in the context of a parent Assembly.

Usage

If a CATIA file type, for example "CATProduct", is selected in the "Create" action, a list which contains all the Template Files of this file type is shown. The user can select one of the Template Files from the list.

Select a PDM type for performing a Create ?	×
PDM Object Types	
Assembly Harness Component Std. Part CATPart CATProduct CATDrawing catalog	
	ancel

Picture 255: Creating a new CATProduct CAD item



Picture 256: List containing all CATProduct Template Files

The "Create" functionality will create both the CAD Document and the corresponding Part item, with the Part having the "default" classification.

With the new functionality it is possible to create one of the specific part types, and to be able to choose the Template File from a list specific for that part type:

→



Picture 257: Creating a new Part item

→



Picture 258: List containing Template Files corresponding to the selected Part

Here, for the part type "/Part/Design/Harness" two specific Template Files for CATProduct are defined.

It is also possible to create a Part item with a corresponding CATIA Template File, as described above, in the context of a parent CATProduct. For this, a CATProduct in a CATIA structure loaded from PDM has to be selected, and the PDM Workbench context action "PDM Create in Context" has to be clicked:

NewProduct3.0	CATProduct
🕵 NewProduc	<mark>t3</mark>
🔶 🌄 NewPar	C <u>e</u> nter graph
💁 🌄 New Par	Reframe On
💶 🙀 NewPar	Hide/Show Alt+Enter
💶 🙀 NewPar	Dopen Sub-Tree
Applications	K Cut Ctrl+X
	Copy Ctrl+C
	Paste Ctrl+V
	Paste Special
	Delete Del
	NewProduct3 object
	PDM Workbench
	Components
	Represent <u>a</u> tions

Picture 259: "PDM Create in Context" context menu entry



This will display the same dialogs as the regular "Create" process:

Picture 260: Creating a new Part item in context

•	
,	

PDM Create		?	×
Create dialog for type (CATProduct:		
CATProduct Template			<u> </u>
	Harness-1.CATProduct Harness-2.CATProduct	t t	
Concession of the Concession o			

Picture 261: List containing Template Files corresponding to the selected Part in context

→ The following "Update" process includes adding the instance of the new Part to the parent Assembly:

NewProduct3 NewPart3.1 [NewPart3.CATPart] NewPart4.1 [NewPart4.CATPart] NewPart3.2 [NewPart3.CATPart] NewPart4.2 [NewPart4.CATPart] NewPart4.2 [NewPart4.CATPart] NewPart4.2 [NewPart4.CATPart] NewPart4.2 [NewPart4.CATPart] NewPart4.2 [NewPart4.CATPart]							× ×	
	Update to PDM?							- 🗆 X
	File name New.CATProduct NewProduct3.CATProduct	CATIA Part Number New NewProduct3	PDM Display NewProduct3, A, 1, , Preliminary	Locked By admin	PLM State Preliminary	File Status Modified Modified	Operation Create : 1. Create PDM Object o Update : 1. Add sub-part relatio	of type "/Part/Design/Harness" n "NewProduct3/New.1"

Picture 262: "Update to PDM" dialog after "PDM Create in Context" action

Standard Part Support

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 items in PDM. Instead, the Standard Part item which has the same Part Number as the CATPart's CATIA Part Number will be queried and added to the PDM structure instead. Using this functionality Standard Part geometry can be added to a PDM structure without having to first load the Standard Parts into the CATIA session.

"Standard Part" Functionality for BOM Part Structure Data Model

In the BOM Part Structure Data Model, 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 PDM structures that the designer works on.

You can query for a Standard Part explicitly (see *Picture 263: Querying for a Standard Part*). Please note that regular users cannot claim and modify Standard Parts, they can only use them in their structures:

🛐 20180511_StandardPart001, A, 1, , Preliminary (logged in as t	estuser, database: InnovatorSolutions1	112)						
C 20190511 Standard Part001 A 1 Brokin	oinan							
sy zotovsti_standardraitovi, A, I, , Preiminiary								
- 😒 Attaches (Part CAD) / 20180511_StandardPart001, A, 1, , Preliminary								
PDM Query (logged in as testuser, database: Innovator/Solutions1112)								
PDM object type								
Std. Part	Part Number ≥	Major Rev.	Generation	Name	State			
	20180511_StandardPart001	A	1		Preliminary			
PDM object attributes Part Number *								
Maior Rev.								
Generation								
Name								
State 🚽								
Unit								
Make / Buy								
Description								
Created on 🗸								
Modified on 🚽								
Created by								
Modified by								
Locked by								
					•			
				Load with Links	E Load with Drawing			
Query					Close			
					Ok			

Picture 263: Querying for a Standard Part

Standard Parts can be used like regular parts. The exceptions are that regular users cannot 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.

"Standard Part" Functionality for CAD Document Structure Data Model

The "Standard Part" functionality is also available in the CAD Document Structure Data Model.

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 "PDM Query" dialog.

Regular users cannot claim or otherwise modify Standard Part CAD Documents (see *Picture 264: Using Standard Parts as a regular user*).

CATPart (logged in as testuser, database: Innova	torSo	olutions1112)		
 20180511_StdPart003, A, 1, , Pre 20180511_StdPart001, A, 1, , Pre 20180511_StdPart002, A, 1, , Pre 		Center graph Reframe On Hide/Show Properties Open Sub-Tree Cut Copy Paste Paste Special	Alt+Enter Ctrl+X Ctrl+C Ctrl+V	
		Delete 20180511_StdPart003, A, 1, , Prelim Expand Expand Multiple Levels De_Expand Open in new PDM Window Unclaim Claim Unlock all	Del	, , , , , , , , , , , , , , , , , , ,

Picture 264: Using Standard Parts as a regular user

They can use Standard Parts in the CATIA structures that they work on (see *Picture 265: Using Standard Parts in CATIA structures*).



Picture 265: Using Standard Parts in CATIA structures

Adding Standard Parts to an existing structure at first does not seem different from adding other CATPart nodes ... (see *Picture 266: "Update to PDM" dialog with Standard Parts*)

/pdate to PDM?	20180511 2018 	Product010.CATProdu 0511_Product01 0180511_Part01(0180511_StdPart 0180511_StdPart cations	ct 0 (20180511_Part010.1) 001 (20180511_StdPart001.1 002 (20180511_StdPart002.1)					z ×
File name 20180511_Pan01(20180511_Produc	J.CATPart t010.CATPro	CATIA Part Number 20180511 Part010 20180511_Product0	PDM Display	Locked By	PLM State	File Status Modified Modified	Operation Create Create : 1. Create PDM File Ol	Rex	Jit
Create new CAD d	ocument gener	ation? No						Update	Close

Picture 266: "Update to PDM" 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 (see *Picture 267: "Update to PDM" dialog with Standard Parts – Result* and *Picture 268: Existing Standard Parts being used in a new structure*).

Update to PDM?							×
File name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Result
20180511_Part010.CATPart 20180511_Product010.CATPro	20180511_Part010 20180511_Product0				Modified Modified	Create Create : 1. Create PDM File Object of typ	SUCCESS SUCCESS
 Created relation of type 'CAD Str Added instance '20180511_Part01 Created relation of type 'CAD Str Added instance '20180511_StdPa Created relation of type 'CAD Str 	ucture'. 0.1' to CAD Structure re ucture'. 1001.1' to CAD Structur ucture'.	lation. e relation.					•
Uploaded 20180511_Part010.CAT Uploaded 20180511_Product010.	Part CATProduct	e relation.					H
Create new CAD document gener	ation? <u>No</u>					<u>. Upr</u>	ate Close

Picture 267: "Update to PDM" dialog with Standard Parts – Result



Picture 268: 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 (see *Picture 269: CATIA Catalog containing Standard Part CATParts*).

🚱 💭 🗢 📕 🕨 Bibliotheken 🕨 Dokumente 🕨 Öffentliche Doku	ment	e 🕨 StandardParts 🗸 🗘 StandardParts durchsuchen 🔎
Organisieren 🔻 Freigeben für 🔻 Neuer Ordner		#= * 🚺 🔞
E Desktop	* 11	Bibliothek "Dokumente" Anordnen nach: Ordner •
E Bilder		Name
Dokumente Öffentliche Dokumente		20180511_StdPart00.CATPart 20180511_StdPart02_CATPart
Dassault Systemes StandardParts		CatalogDocumentL.catalog
4 Elemente		

Picture 269: CATIA Catalog containing Standard Part CATParts

Then the Standard Parts can be inserted to a CATProduct structure (see *Picture 270: Standard Part CATParts created from a Catalog*).



Picture 270: Standard Part CATParts created from a Catalog



Picture 271: Inserted Standard Parts

In the "Update" process the Standard Part item from the database is taken (see *Picture 272: "Update to PDM" dialog with Standard Parts*).

pdate to PDM:								
File name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Rest	ult
20180511_Part011.CATPart	20180511_Part011				Modified	Create		
20180511_StdPart001.CATPart	20180511_StdPart001				Modified	Create		
20180511_StdPart002.CATPart	20180511_StdPart002				Modified	Create		
20180511_StdPart003.CATPart	20180511_StdPart003				Modified	Create		
20180511_Product011.CATPro	20180511_Product0				Modified	Create : 1. Create PDM File Object of ty	p	
								_
Create new CAD document genera	tion? No							
						1	Jpdate	CI
								-



→

The update results are displayed in the text area of the window (see *Picture 273: Update result*).

ine marine	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Result
0180511_Part011.CATPart	20180511_Part011				Modified	Create	SUCCESS
0180511_StdPart001.CATPart	20180511_StdPart001				Modified	Create	
180511_StdPart002.CATPart	20180511_StdPart002				Modified	Create	
0180511_StdPart003.CATPart	20180511_StdPart003				Modified	Create	
0180511_Product011.CATPro	20180511_Product0				Modified	Create : 1. Create PDM File Object of typ	SUCCESS
			m				
eated relation of type 'CAD St	ucture'.						
Ided instance '20180511 StdPa	rt001.1' to CAD Structur	e relation.					
indard part file PDM item exists, t	s. expanding for it (CAD stru	structure mode).					
	ucture'.						
eated relation of type 'CAD St		e relation					
eated relation of type 'CAD St ded instance '20180511_StdPa	rt002.1' to CAD Structur	c reactorn					
reated relation of type 'CAD St dded instance '20180511_StdPa andard part PDM item exists, o andard part file PDM item exists.	rt002.1' to CAD Structur uerying for it (CAD stru-	tture mode).					
reated relation of type 'CAD St dded instance '20180511_StdPa andard part PDM item exists, c andard part file PDM item exists reated relation of type 'CAD St	rt002.1' to CAD Structur uerying for it (CAD stru- is, expanding for it (CAD ucture'.	structure mode).					
reated relation of type 'CAD St 3ded instance '20180511_StdPa andard part PDM item exists, c andard part file PDM item exist reated relation of type 'CAD St 1ded instance '20180511_StdPa	rt002.1' to CAD Structur uerying for it (CAD stru- ts, expanding for it (CAD ucture'. rt003.1' to CAD Structur	e relation.					
and part the 'CAD St ded instance '20180511_StdP. andard part PDM item exists, c andard part file PDM item exists, eated relation of type 'CAD St ded instance '20180511_StdPa	rt002.1' to CAD Structur uerying for it (CAD stru- is, expanding for it (CAD ucture'. rt003.1' to CAD Structur	ture mode). structure mode). e relation.					

Picture 273: Update result

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

As with regular CATParts, the new Standard Part CATPart node is added to the CAD Document structure.

You can verify this with the "PDM Structure" button of the toolbar (see *Picture 274: "Show PDM Structure" icon*).



Picture 274: "Show PDM Structure" icon



Picture 275: CAD Document structure containing Standard Parts

CATIA Design Table Support

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

You can create a Design Table for a CATPart or a CATProduct (see *Picture 276: CATPart with Design Table*).

NewPart1.CATPart	
🔯 NewPart1	Z
— 🛹 xy plane	
🖅 yz plane	
zx plane	
Relations	
- III DesignTable.1	
Configuration	
Sheet.1	
PartBody	
+ L AbsoluteAxis	
🕂 nd Geometry	
- 🖬 Constraints	
- 😤 Parallelism.1	
- 🧭 Parallelism.2	
- 🤣 Parallelism.3	
- S Parallelism.4	
Ength.5	
Ength.6	

Picture 276: CATPart with Design Table

Updating to PDM will create a Document item for the Design Table and upload the file (see *Picture 277: "Update to PDM" dialog containing a Design Table*).

Update to PDM?			_	1.0				l	
File name CA	TIA Part Number	PDM Display		Locked By	PLM State	BOM Type	File Status	Operation	Result
NewPart1.CATPart Ne	wPart1	NewPart1, A, 1, , Preliminary		admin	Preliminary	BOM	Modified	Update : 1. Created "1" new design table.	
DesignTable1.xlsx De	signTable1						New	Create	
Create new CAD document generation	2 1	Import with Reconnect?	-						
erente nen er a usedment generatio	INO	No	2						
								Up	date Close

Picture 277: "Update to PDM" dialog containing a Design Table

After the update the Design Table is related to the CAD Document (see *Picture 278: Design Table document related to CAD Document*).



Picture 278: Design Table document related to CAD Document

The Design Table file can be modified and uploaded to PDM again (see *Picture 279: Editing a Design Table, Picture 280: Adding a line to the Design Table Excel sheet, and Picture 281: The Design Table is updated in the CATIA session*).

DesignTable.1 active, configuration row : 1	? ×
Design Table Properties	
Name : DesignTable.1	Activity
Comment : DesignTable created by umehring 03.04.2018	
Configurations Associations	
Filter:	Edit
Line PartBody\Sketch.1\Length.5\Length PartBody\Sketch.1\Length.6\Length	
<1> 1000mm 90mm	
Edit table	Duplicate data in CATIA model
	OK Apply Cancel

Picture 279: Editing a Design Table

×	DesignTable1.xlsx								
	А	В							
1	PartBody\Sketch.1\Length.5\Length (mm)	PartBody\Sketch.1\Length.6\Length (mm)							
2	1000	90							
3	2000	270							
4									
5									

Picture 280: Adding a line to the Design Table Excel sheet

DesignTable.1 active, configuration row : 1	8	<u> </u>
Design Table Properties		
Name : DesignTable.1	I	Activity
Comment : DesignTable created by umehring	3 03.04.2018	
Configurations Associations		
Filter :		Edit
Line PartBody\Sketch.1\Length.5\Length	PartBody\Sketch.1\Length.6\Length	
<1> 1000mm	90mm	
2 2000mm	270mm	
	Knowledge Report	
]
	From Summary	1
	DesignTable.1 Design Table Synchronization	I
T-Statute 1		
Edit table	Message :	
	The file of the design table DesignTable.1 has been modified	l
	This design table has been synchronized with this file	
		Close

Picture 281: 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 (see *Picture 282: Refreshed PDM Structure window containing the Design Table*).



Picture 282: Refreshed PDM Structure window containing the Design Table

The "Update" process 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.

Archive Support

It is possible to compress a complete CATProduct substructure 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 substructure which has not been created in PDM can be defined as an archive (see *Picture 283: Defining a CATProduct structure as an archive*).



Picture 283: Defining a CATProduct structure 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 (see *Picture 284: Resulting archive CAD Document in PDM*).



Picture 284: Resulting archive CAD Document in PDM

CATIA Catalog Support

CATIA Catalogs for CATParts are supported. The Catalogs can be created and updated by a "Standard Part Administrator". The "CATIA Catalog" functionality supports CATParts used as "Standard Parts", as "Templates" or CATParts holding a "Power Copy".

The "CATIA Catalog" functionality adds the Catalog keywords "PWB_CAD", and in case of BOM-CATParts "PWB_PART". The values of these keywords must not be changed by a user.

During open a Catalog in the CATIA Catalog Editor or Catalog Browser using the PDM Workbench, placeholder files for the referenced CATParts in Aras Innovator are automatically created in the PDM Workbench exchange directory. The actual geometry of a referenced CATPart will be fetched from Aras Innovator as soon as the geometry is needed by the native "CATIA Catalog" functionality.

Create Catalog

All CATParts added to a Catalog must be loaded from Aras Innovator in the current CATIA session in design mode.

Use native CATIA functionality Add Component →Select document in Session to add a CATPart to the Catalog (see Picture 285: Add Component to Catalog). You can also select an external feature of a CATPart loaded from Aras Innovator (PowerCopy).

CatalogDocument2.catalog	A Search Filter: Result Referenc Generativ Name	e Keywords Preview Generative Data 9. Definitions Date(D-M-Y) Definition	
71 11	Session document Collisers/Public/PWB_VMAP1112 Collisers/Public/PWB_VMAP1112 Collisers/Public/PWB_VMAP1112 r mm File name: Collisers/Public/PW Files of type: CATPant/:CATPant	X0180511_5tdPart001_CATPa X0180511_5tdPart002_CATPa X0180511_5tdPart003_CATPa X0180511_5tdPart003_CATPa X0180511_5tdPart03_CATPa X0180511_5tdPart03_CATPa X0180511_5tdPart03_CATPa X0180511_5tdPart03_CATPa X0180511_5tdPart03_CATPa X0180511_5tdPart03_CATPa X0180511_5tdPart03_CATPa X0180511_5tdPart03_CATPa	Select document in session

Picture 285: Add Component to Catalog

Use "Local preview" to avoid the download of the CATParts to show the preview (see *Picture 286: Select "Local preview"*).

D	escription Definition	
	Name: 20180511_StdPart001	
	Reference Keyword values Preview	
	 Local preview (stored in catalog) O Referenced document preview O External file preview 	5 D D D D D D D D D D D D D D D D D D D
	Select an external preview file	
	< III >	
	OK Cancel	

Picture 286: Select "Local preview"

When the needed CATParts are added you can use the normal "Update" functionality to store the Catalog document in Aras Innovator (see *Picture 287: Store Catalog document in Aras Innovator*).

Update to PDM?								
Update to PDM?	CATIA Part Number CatalogDocument2	PDM Display	Create a new CATIA docum Creation of a new object Doc. Number CatalogDo Name Description	Locked By ment of type 'catalo	PLM State	File Status Modified	Operation Create : 1. Create PDM File Object of typ.	Result
Create new CAD document gene	ration? No	Create corre	sponding BOM parts?	ок	Cancel	J	<u>ur</u>	idate Close

Picture 287: Store Catalog document in Aras Innovator

During update the Catalog will be renamed like according to the Document Number. The keywords "PWB_CAD", and in case of BOM-CATParts "PWB_PART" are automatically added during "Update" process (see *Picture 288: Catalog editor after update*).

CatalogDocument2.catalog		
CatalogDocument2.catalog CatalogDocument2.cata	Search Filter: Result Reference Keywords: Preview Generative Data Name PW8_CAD* 1 20180511_5tdPart001 72A60876D8184A8833F64FA86F6F4CD4	- <u>A</u> 2
a	Supporting Document - Not Available	Select Select Preview Laurch Berney Seguering Document Permove Preview

Picture 288: Catalog editor after update

Update Catalog

To Update a Catalog you have to be a "Standard Part Administrator". Use the "PDM Query" functionality for "catalog", then use "Open Catalog for Edit" (see *Picture 289: Open Catalog for Edit*).

PDM Query (logg	ed in as admin, database: Innovator	Solutions1	1112)						×
PDM object type			Doc. Number	<u>N</u>	Major Rev.	Generation	Name	State	De
catalog		•	📓 CatalogDocun	1en+2	٨	1	CatalogDocument2	Preliminary	
PDM object attri Doc. Number Major Rev. Generation Name State Description	* • •				Open in new PDM W Unlock Lock Open Catalog Brows Open Catalog For Ed	indow er it			
Created on	-								
Modified on	+								
Created by	-								
Modified by	•								
Locked by	.								
External Owner	T-Systems.Mechanical.CATIA 👻		1						
Query							Load with	Links 🗌 Load with	Drawing
									Ok

Picture 289: Open Catalog for Edit

Now you can add or remove CATParts from the Catalog. All CATParts added to a Catalog must be loaded from Aras Innovator in the current CATIA session in design mode.

Use the "Update" process to store the changes to Aras Innovator.

Open Catalog Browser

To use the Catalog Browser there has to be an active CATProduct or CATPart window.

Use the "PDM Query" functionality for "catalog" then use "Open Catalog Browser" (see *Picture 290: Open Catalog Browser*).

DM object type		Doc. Number	X	Maior Rev.	Generation	Name	State	C
catalog	•	CatalogDo				CatalogDocumer	nt2 Preliminary	
DM object attributes Doc. Number =			Open i Unlock Lock Open C	n new PDM Window Satalog Browser Satalog For Edit				
Created on Modified on Created by								
Modified by								
xternal Owner T-Systems.Mechanical.CATIA 👻	*							
		•						
						Load w	ith Links 🔲 Load with	n Drawir

Picture 290: Open Catalog Browser

The Catalog Browser will be opened (see Picture 291: Catalog Browser).

Catalog Browser:C:\Users\Public\PWB_XMAP1112\CatalogDocument2.catalog						
Current: PowerCopy		- E	= =			
C 20180511_StdPart001	•		E			
•		•	F			
Filter:			Launch			
Name						
1 20180511_StdPart001						
			Close			

Picture 291: Catalog Browser

You can use the Catalog Browser like a native CATIA Catalog Browser.

The function "Open As New Document" opens a new file with the same Part Number like the original CATPart, so you have to change the Part Number of the new document before using "Update" (see *Picture 292: Open As New Document*).



Picture 292: Open As New Document

Open Catalog for special Usage

There are some special CATIA functions like "Circuit Board Design" which need a Catalog as input.

To use this functionality, you have to open the Catalog from Aras Innovator into the CATIA session using "Open Catalog for Edit" before you start your special functionality. When you are asked for a Catalog during a function like "Circuit Board Design" you have to select the loaded Catalog document (see *Picture 293: Select Loaded document for Catalog*).

	Circuit Board Design		- • ×					
	Status 0%	100%						
	Selected files							
	🔒 Idf	C:\Users\Public\PWB_XMAP1112\CATIA_	LIB 🥳					
	🖁 Library							
Brow ?	💡 Catalog	No File						
	Catalog in read/write mode							
F e	Associate a csv file to catalog							
File	Holes options							
	O Draw only							
Loaded document	🕑 Drill all							
Cancel	O Custom							
	Drill only mounting (MTG) and tooling (TOOL) holes (IDF 3.0 only)							
· · · · · · · · · · · · · · · · · · ·	Drill holes for diameter superior or e	equal to 0,1 mm	É					
	User rule							
		🕚 ОК	Cancel					

Picture 293: Select Loaded document for Catalog

Support floating Content in Catalog

By default, the Catalog points to a fixed Aras Innovator generation of the contained Aras Innovator CAD (Part). If this functionality is enabled, you will get the latest generation of the corresponding CAD (Part).

Configurable Catalog Keywords

By default, the identifying attributes (id or item_number when using floating Catalog content) are stored in the values of "PWB_CAD", and in case of BOM-CATParts "PWB_PART".

PWB - FloatingCatalog1.catalog			
Beloating Cat	Search		
- 🔁 Kapitel.1	pitel.1 Filter: Kompt Result		
L 🛛 Komp			
	Reference Keywords	Preview Generative	Data
	Name	PWB_CAD*	PWB_PART*
	1 DOC_20190510_001	DOC_20190510_001	20190510_001

Picture 294: Configurable Catalog Keywords

It is possible to configure different Catalog keywords to store these values. In case of useBomPartStructure = true, if only BOM CATParts are used in the Catalog, it is also possible to remove the keyword for the CAD.
CATProcess File Support

CATProcess can be updated and loaded from Aras Innovator. If the CATProcess uses external references in the CATProcess Product List or in the CATProcess Resources List, the PDM Workbench creates a relation to the referenced items during update and downloads the referenced items during load.

Referenced CATProducts/CATParts must already exist in Aras Innovator. Update of the CATProcess will not update the referenced items. During update the user will get a warning if there are modifications in the referenced external structures.



Picture 295: CATProcess with external referenced CATPart in the ProductList and internal Component in the ResourcesList

The related Aras Innovator structure would use the "/CAD Structure/CATProcessProduct (ProductList Sub-Structure)" relation between the CATProcess and the referenced CATPart. The internal Component in the ResourcesList is not exposed to Aras Innovator.



Picture 296: Aras Innovator structure of CATProcess with external referenced CATPart in the ProductList and internal Component in the ResourceList

It is also possible to have a CATProcess that references multiple external items in the ProductList and in the ResourcesList.



Picture 297: CATProcess with external referenced structures in the ProductList and in the ResourceList

The related Aras Innovator structure would use the "/CAD Structure/CATProcessProduct (ProductList Sub-Structure)" relation and the "/CAD Structure/CATProcessResource (ResourcesList Sub-Structure)" relation to the directly referenced top-level CATProduct of the structures.



Picture 298: Aras structure of CATProcess with external referenced strucures in the ProductList and in the ResourceList

If the user wants to modify the referenced structures alongside with the CATProcess it is possible to open the referenced CATProducts/CATParts in a new window. This can also be done by the PDM Workbench Options setting "Open linked Documents of CATProcess in own Window":



Picture 299: "PDM Workbench Options" dialog – "Open linked Document of CATProcess in own Window"

Configurable CATIA Component 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 300: Embedded CATIA component nodes*).

NewProduct3.CATProduct	- • •
 NewProduct3 NewPart3 (NewPart3.2) SKIP_1 (SKIP_1.1) SKIP_1 (SKIP_1.1) NewPart3 (NewPart3.1) IGN_1 (IGN_1.1) IGN_1 (IGN_1.1) NewPart4 (NewPart4.1) NewPart4 (NewPart4.2) 	The second secon

Picture 300: Embedded CATIA component nodes

Electrical/Tubing Support

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" (see *Picture 301: Example document containing electrical components*).

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

CAA_EwrFilterSample.CATProduct
Product1
🕂 🐨 Geometrical Bundle1 (Geometrical Bundle1.1)
Electrical Bundle1 (Electrical Bundle1.1)
+- > wire-2.5mm-blue (Wire1)
+- > PN-AMP-3403B (Wire2)
+ > wire-2.5mm-blue.1 (Wire3)
💠 🕨 wire-2mm-yellow (Wire6)
+- > wire-2.5mm-black (Wire7)
Publications
Applications

Picture 301: Example document containing electrical components

To get the type of a component just use the "Update" functionality. If there is an unsupported component, a message box shows the type of the first unsupported component (see *Picture 302: Warning about unsupported CATIA component node*).



Picture 302: Warning about unsupported CATIA component node

Additional Rep Types

It is possible to load additional 3D rep types in addition to e.g. "CAD_3D" in the CATProduct structure at the same time (see *Picture 303: Two CATParts with different rep types related to the same part loaded at the same time*).

This is only possible when the different 3D rep type CATParts are defined as non-BOM CAD Documents. BOM-relevant CAD Documents are the CAD items that are related directly to the Part item with the "Part CAD" relation. Non-BOM CAD Documents are the "CAD Structure/Structure" child nodes of the BOM-relevant CAD Documents:



Picture 303: Two CATParts with different rep types related to the same part loaded at the same time

If the rep type filter functionality is configured in the Schema file a new command is available in the PDM Workbench toolbar (see *Picture 304: Filter Representation Type command*).



Filter Representation Type

Picture 304: Filter Representation Type command

This command opens the selection of the representation types to be loaded. Selection changes are persistent until CATIA is closed (see *Picture 305: Select non-BOM Representation types to be loaded*).

Filter Repres ?
CAD_3D
CAD_MAF
🔎 LITE
CAD_KIN
CAD_WFR
OK OK Cancel

Picture 305: Select non-BOM Representation types to be loaded

Support generic Shape Representations

Depending on the CATIA installation it is possible to add a geometry file of a type like CATShape, jt, pkg, wrl, STL or others as a representation to a Component. The present functionality supports to store these additional geometry files in Aras Innovator. The extra geometry files are treated as read-only files, just like CATIA V4 model files.

It is possible to configure up to five additional types to be supported.

Product Stress Applicat	t1 ior	ct2 (Prod	uct2.1)				? ×
		Name	Source	Type	Default	Activated	Associate
		Shape 1	C:\xmap3\shapes\Wrl1.wrl	wrl	yes	yes	Associate From Session Remove Replace Rename
	Interview Interview						

Picture 306: Manage Representations

CATIA Documents are set to read-only if corresponding PDM Node is not modifiable

State	Name	Location	Action	Access	Save
Open	2.PWBDoc	C:\Users\Public\PWB_XMAP1112		Read Write	Save As
Open Read Only	Engine.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	Propagate direct
Open Read Only	Crankshaft_Assm.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	Reset
Open Read Only	ConnectionRod.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	
Open Read Only	Cylinder_Assm.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	
Open Read Only	FixedParts_Assm.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	
Open Read Only	CrankshaftLeft.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	
Open Read Only	CrankPin.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	200
Open Read Only	CrankshaftRight.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	4122
Open Read Only	Piston.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	
Open Read Only	PistonPin_Assm.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	
Open Read Only	Rack_Assm.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	
Open Read Only	CylinderBlock.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	
Open Read Only	SnapRing.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	
Open Read Only	PistonPincomp.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	
Open Read Only	Screw.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	
Open Read Only	Rack.CATProduct	C:\Users\Public\PWB_XMAP1112		Read Only	
Pattern Name: *	Apply Pattern				
attern Name: ^ Unsaved File(s) Left	Apply Pattern	Enable independent saves			

Loading a structure sets the corresponding CATIA files to read-only (see *Picture 307: Save Management*).

Picture 307: Save Management

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 308: Check if CATIA structure is valid*).

Product1.CATProduct	PDM Workbench
Froducti Gomponenti (Componenti.1) A Gomponenti (Parti.1) Z4 Gomponenti (Componenti.2) Parti (Parti.1)	Information Warnings This structure is not valid: The instance name "Part1.1" occurs more than once on the part "Product1". Attention: The following errors occurred: This structure is not valid: The instance name "Part1.1" occurs more than once on the part "Product1".
Applications	X Action failed.
	Close

Picture 308: Check if CATIA structure is valid

Thumbnails

In the "Properties" dialog for part and drawing documents a thumbnail will be shown (see *Picture 309: CAD Document properties in Aras Innovator*).

🕘 Aras Innovator - Mozilla Firefox	- 0 X
192.168.168.168/InnovatorServer1112/Client/X-salt=std_11.0.0.6920-X/scripts/Innovator.aspx	
	innovator Admin <u>.oqout</u>
File Edit Views Search Actions Reports Tools Help	
CAD Document Document Number Revision State	*
ScrewModel A Preliminary Name	E
PWB Name From Template Viewable File	
	-
Structure Parents Parts Files Changes CAD DesignTable	_ 🗆
Actions 🔻 🛛 Pick Related 🔽 📮 🌮 🗙 🔯 🚳 🔒 😭 🖑 🗛 욨 🖉 Hide Search Criteria	-
Sequence Document Number Revision Name Type State	Native File
Ready 0 Items found.	Aras Innovator

Picture 309: CAD Document properties in Aras Innovator

Link Management

When designing, the user can create many types of links between CATIA features or between a CATIA feature and a foreign object. Some of these links are presented to the user in the "Edit Links" command, others are not shown in the CATIA UI.

The information about the CATIA links, which are pointing from an object in one document to an object in another document ("external link"), can be retrieved and transferred to Aras Innovator.

Previous PDM Workbench releases supported the transfer of CATIA links between a CATDrawing and a CATProduct or CATPart ("Drawing" link in Aras Innovator) and between a CATPart and a CATPart ("Reference" link in Aras Innovator). Furthermore, regular product structure links and Design Table links were supported. This is still the default behaviour with this PDM Workbench release.

The PDM Workbench provides the capability to detect and transfer all CATIA external link types as they are provided and grouped by CATIA.

The regular product structure links and the Design Table links are managed, too.

Eight additional link types are supported now (with the PWB Configuration item setting UseAllCatiaLinkTypes = true) and the corresponding Aras Innovator CAD Structure subclasses (classification property, default label "Dependency") are named like the internal CATIA links:

- Contextual
- Reference (Original CATIA link type "Design")
- Drawing (Original CATIA link type "Downstream")
- IsComposedOf
- Product
- Reference
- Result
- RuleBase

Depending on the context, CATIA product links are created as two different classifications:

- Structure (regular product structure links from a CATProduct to another document)
- Product (other CATIA Product links not pointing from a CATProduct, but e.g. from a CATPart to a CATShape or cgr)

The links are created in Aras Innovator, if both the pointed and the pointing document are known and the pointed document is stored first.

The link information can be displayed in the regular Aras Innovator relationship grid of an Aras Innovator CAD Document and in the CATIA PWB PDM Structure window, for one level, or for multiple levels:



Picture 310: Expand → All pointed documents

Basic Drawing Link Support

When a CATDrawing CAD Document is created, you 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 311: Creating a CATDrawing document with a link to 3D geometry*).



Picture 311: Creating a CATDrawing document with a link to 3D geometry

When you create 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 312: PDM message about created drawing link*).

Jpdate to PDM?			1.0	-	_		
File name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Result
Drawing1.CATDrawing	Drawing1				Modified	Create : 1. Create PDM File Object of typ	SUCCESS
Part Number 'Engine-Drw': No file PDM item exists yet, crei No CAD document PDM item e Created '/CAD/Mechanical/Drø Created relation '/CAD Structum The following files have been u Engine-Drw. CATDrøwing	ating it. xists yet, creating it. wing' with name 'Engin e/Drawing' between 'En ploaded to PDM:	n e-Drw'. gine-Drw' and 'Engine'					•
Create new CAD document gene	eration? <u>No</u>	Create corresponding BOM parts?	No			<u>.Up</u>	late Close

Picture 312: PDM message about created drawing link

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 313: Expanding newly created drawing link*).

No. 10 Engine-Drw, A, 1, , Preliminary (Ic	ogged in as admin, database: l	InnovatorSolutions111	.2)	- • ×
Engine-Drw, A, 1, ,	C <u>e</u> nter graph			
	<u>R</u> eframe On			
	Hide/Show			
	Pr <u>o</u> perties	Alt+Enter		
	Ope <u>n</u> Sub-Tree			
X	Cu <u>t</u>	Ctrl+X		
	<u>С</u> ору	Ctrl+C		
l i i i i i i i i i i i i i i i i i i i	<u>P</u> aste	Ctrl+V		
	<u>D</u> elete	Del		
	Engine-Drw, A, 1, , Prelimir	nary object 🔹 🕨		
	E <u>x</u> pand	•	Is derived from Model	Ž
	De_Expand		Is derived from Model (Current)	х∕∽у
·	Open in new PDM Window		User Values from Design Table	

Picture 313: Expanding newly created drawing link

The document will be displayed in the window (see *Picture 314: Displaying newly created drawing link*).



Picture 314: Displaying newly created drawing link

If you create a drawing with links to more than one 3D geometry file then both linked documents will be displayed in the window (see *Picture 315: Displaying all created drawing links*).

20180511_Product030Drw, A, 1, , Preliminary (logged in as admin, database: InnovatorSolutions1112)	
20180511_Product030Drw, A, 1, , Preliminary	
s derived from Model / 20180511_Product031, A, 1, , Preliminary	7
Is derived from Model / 20180511_Product030, A, 1, , Preliminary	×⁄y

Picture 315: 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.

CATDrawing: Loading referenced Data as "Current"

It is possible to load the 3D data (CATParts or CATProduct structures) which are referenced by a CATDrawing as "Current" instead of "As Saved", which is the default.

In this example, the CATDrawing "Engine-Drw" was generated by the generation 1 of the CATProduct structure "Engine", but a second generation of the CATProduct structure has been created (see *Picture 316: "As Saved" drawing link*).



Picture 316: "As Saved" drawing link

The drawing relation can also be expanded as "Current" (see *Picture 317: "Current" drawing relation*).

CATProduct, CATDrawing (I CATProduct, CATDrawing (I Engine, A, 2, , Pre Engine, A, 1, , Pre	logged in as admin, database: I liminary liminary (newest versi	innovatorSolutions1112):2	
Engine-Drw, A, 1,	Center graph Beframe On Elide/Show Properties Open Sub-Tree Cot: Cot: Paste Paste Paste Delete Engine-Drw, A, 1, , Prelin	Alt+Enter Ctrl+X Ctrl+C Ctrl+V Del		
	Expand B De_Expand Open in new PDM Wind Unclaim	ow	Is derived from Model Is derived from Model (Current) Uses Values from Design Table Is gttached by	, z x – y

Picture 317: "Current" drawing relation

"Open with related 3D files" loads the related 3D documents as they are stored in the database (see *Picture 318: Loading "As Saved"*).

ine-Drw, A, 1, , Pr	eliminary (logged in as admi	n, database: Innova	torSolutions1112):3	
Ingine-Dr	Center graph			
	Reframe On			
8	Hide/Show			
	Pr <u>o</u> perties	Alt+Enter		
	Ope <u>n</u> Sub-Tree			
X		Ctrl +X		
P	<u>)</u> ⊆ору	Ctrl+C		
	<u>P</u> aste	Ctrl+V		
	Paste <u>Special</u>			
	Delete	Del	-	
			-	
_	Engine-Drw, A, 1, , Prelimir	nary object	_	
	Expand		•	
	De_Expand			
	Open in new PDM Window	V		
1	<u>U</u> nlock			
6	Lock			
1	Unloc <u>k</u> all			
2	Lock <u>a</u> ll			
7	l Pro <u>m</u> ote			
12	2 Re <u>v</u> ise			
61	Open <u>F</u> ile			
<u>-</u>	Open <u>w</u> ith related 3D files			
<u>e</u> t	Open with current related	<u>3</u> D files		
	Open File temporarily			
÷	Create new version			
÷>	Delete newest version			:
2	Duplicate			

Picture 318: Loading "As Saved"

This is the generation 1 of the CATProduct structure. The CATDrawing does not have to be refreshed because it was saved with this 3D geometry (see *Picture 319: Generation 1 of the CATProduct structure*).



Picture 319: Generation 1 of the CATProduct structure

"Open with current related 3D files" loads the newest generation of the related 3D documents (see *Picture 320: Loading "Current"*).

Engine-Drw, A, 1, , Preliminary	(logged in as admin, database: InnovatorSolutions1112):2	
	See the second	
Engine-Drw, A, 1, ,	Center graph	
	Reframe On	
	Bide/Show	
	Properties Alt+Enter	
	Dpen Sub-Tree	
	🔏 Cut	
	⊆opy Ctrl+C	
	Paste Ctrl+V	
	Paste Special	
	Delete Del	
	Engine-Drw, A, 1, , Preliminary object	
	Expand	
	De_Expand	
	Open in new PDM Window	
	[™] Unclaim	
	T Claim	
	T Unclaim all	
	Taim all	
	Promote	
	1>2 Revise	
	Open <u>F</u> ile	
	Open with related 3D files	
	Open with current related <u>3</u> D files	
	Open File temporarily	
	Create new version	
	Elete newest version	Z
	🟒 Duplicate	×

Picture 320: Loading "Current"

This is the generation 2 of the CATProduct structure. The CATDrawing has to be refreshed because it was saved with a previous generation of the 3D geometry (see *Picture 321: Generation 2 of the CATProduct structure*).



Picture 321: Generation 2 of the CATProduct structure

This functionality can also be used from the "PDM Query" dialog (see *Picture 322: Actions* "Open with related 3D files / Open with current related 3D files").

M object type			Doc Number	X	Major Rev	Generation	Name	State	D
II CATIA Files		- 🔊	Engine		Δ	2		Preliminan	
		-	Engine		A	1		Preliminary	
M object attributes		5	Engine-Drw	_	٨	1		Preliminary	-
Doc. Number E*				8	Open in new PDI	4 Window			
Major Rev.	-			1	<u>U</u> nclaim				
Generation *	•			18					
Name	-			28.	Open File				
State	•				Open with relate	12D files			
Description					Open with relate	1 5D files			
				<u>6</u> 6	Open with currer	t related 3D files			
				-23	Duplicate				
				* *	Expand	•			
Created on	-			_			_		
Modified on	•								
Created by	-								
Modified by	-								
Locked by									
Is Template									
Standard Part									
tornal Owner	*								
1-Systems.Mech	ianical.CATIA	•		"					
							Load	with Links 🔲 Load with	Drawin
									Cla

Picture 322: Actions "Open with related 3D files / Open with current related 3D files" in "PDM Query" dialog

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 323: Information when reference links are created*).

Jpdate to PDM?				- /1			X
File name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Result
LinkTargetPart1.CATPart	LinkTargetPart1				Modified	Create : 1. Create PDM File Object of typ.	SUCCESS
•							۱. ۲
Uploaded LinkTargetPart1.CAT	Part						
PDM Messages:							=
Created relation '/CAD Structur Created relation '/CAD Structur	e/Reference' between 'Li e/Reference' between 'Li	inkTargetPart1' and 'LinkSourcePart1' inkTargetPart1' and 'LinkSourcePart2'					
							-
Create new CAD document gene	eration? No	Create corresponding BOM parts?	No			<u></u>	date Close

Picture 323: Information when reference links are created

If the links are removed from the CATIA file then the corresponding PDM relations are deleted (see *Picture 324: Information when reference links are deleted*).

Jpdate to PDM?							X			
File name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Result			
LinkTargetPart1.CATPart	LinkTargetPart1	LinkTargetPart1, A, 1, , Preliminary	admin	Preliminary	Modified	Update : 1. Create new version of file ob	SUCCESS			
Uploaded LinkTargetPart1.CA	[Part	m					4			
PDM Messages: Deleted relation '/CAD Structure/Reference' between 'LinkTargetPart1' and 'LinkSourcePart2'										
Create new CAD document ger	neration? No	Create corresponding BOM parts?	No			Upd	ate Close			

Picture 324: 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 325: Expanding geometry links*).

🛐 LinkTargetPart1, A, 1, , Prelimina	ry (logged in as admin, databa	ase: InnovatorSolutions1112	
🔊 LinkTargetPart1, A, 1	Center graph Reframe On Hide/Show Properties Open Sub-Tree Open Sub-Tree Copy Paste Paste Paste Special	Alt+Enter Ctrl+X Ctrl+C Ctrl+V Del Preliminary object	
	Expand Expand Multiple Levels De_Expand Solution of the provided of the provid	b Iow	Uses Values from Design Table Uses Geometry from Uses Geometry from (Current) [s attached by Used in Structure Derives Drawing Derives Drawing (Current) Geometry is used in

Picture 325: Expanding geometry links

The document will be displayed in the window (see *Picture 326: Geometry link expansion result*).

LinkTargetPart1, A, 1, , Preliminary (logged in as admin, database: InnovatorSolutions1112)	- • •
LinkTargetPart1, A, 1, , Preliminary Uses Geometry from / LinkSourcePart1, A, 1, , Preliminary	x y

Picture 326: Geometry link expansion result

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 the context menu action "Relate active File to Part" (see *Picture 327: Action "Relate active File to Part"*).

PDM Query (logged in as admin, database: InnovatorSolutions	1112)				? ×
PDM object type		Part Number	Major Rev.	Generation	Name	State
Part 🔻	\$	NewPart1	A	1		Preliminary
DDM - List strikutes	4	NewPart2	А	1		Preliminary
Point Object attributes	C.	NewPart3	·			Preliminary
Main Day	0	NewProduct1	Open in new PDM Window			Preliminary
Major Rev.	0	NewProduct2	¹ /□ <u>U</u> nclaim			Preliminary
Generation	Þ	NewProduct3	Caim			Preliminary
Name			Promote			
State 👻			1+2 Revise			
Unit			Lord			
Make / Buy						
Description			A CF Relate active File to Part			
			5 Show Neighborhood			
			>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>			
			🟒 Duplicate			
Created on 🚽			Expand	•		
Modified on						
Created by						
Modified by						
Locked by						
	-					· ·
			Load with Links		Download CATDrawings r	elated to BOM Part
						Luery Close

Picture 327: Action "Relate active File to Part"

If there is already a CAD Document related to the part you are asked whether you want to overwrite the corresponding file (see *Picture 328: Confirm the "Relate active File to Part" action*).

DM object type			Part Number	Major Rev.	Generation	Name	State
Part		- 0	NewPart1	A	1		Preliminary
		\$	NewPart2	A	1		Preliminary
DM object attributes		1	NewPart3	А	1		Preliminary
art Number N*	-	0	NewProduct1	А	1		Preliminary
Major Rev.	-	0	NewProduct2	Α	1		Preliminary
Generation	-	0	NewProduct3	А	1		Preliminary
Name	-						
State	-						
Unit					-	×	
Make (Puer							
Make / Duy	-						
Description				Do you want to overwrite th	e related CATPart file?		
				Yes - Overwrite			
				No - Add as additional CAD	Document		
				Cancel - Cancel the action			
Created on	-						
Modified on	•						
Created by	•			Ja	lein Abbreche	n	
Modified by							
Locked by			_				
Local by	•				_		
		•					
				Load with Links		Download CATDrawi	nos related to BOM P
				- coud with thirds		Connous CATDiawi	ings related to boild Pi

Picture 328: Confirm the "Relate active File to Part" action - Overwrite

The file can only be overwritten if the BOM part item is not used in a BOM Part Structure Data Model (see *Picture 329: Information prompt for "Relate active File to Part" action*).

PDM object type		Part Number	Maior Rev.	Generation	Name	State
Part	-	Cylinder Assm	A	1		Preliminary
PDM object attributer		CylinderBlock	A	1		Preliminary
Part Number Cut	_	🗱 CylinderBlockModel	А	1		Preliminary
Major Rev.						
Generation						
Name	-					
State						
Unit		Information 1167			×	
Make / Bus	-	Information 1107				
Description		(i) This a Cylin	action is not possible bec derBlock, A, 1, , Prelimin	ause the part is used by ary	,	
Created on	-					
Modified on	-			UK		
Created by	-					
Modified by	-					
Locked by	-					
		•				
	l					
			Load with Links		Download CATDrawi	ngs related to BOM Pa

Picture 329: Information prompt for "Relate active File to Part" action

Delete Relation

PDM relations can be deleted in the PDM Structure window with a single context menu action, 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 330: Action "Delete relation"*).



Picture 330: Action "Delete relation"

Delete Relations of non-loaded Instances

Normally you can only delete relations during "Update" if the relation was loaded by the PDM Workbench. With this functionality it is possible to delete relations which are not loaded.

It is possible to use this functionality in the BOM Part Structure Data Model and in the CAD Document Structure Data Model.

Use Case

When your complete Assembly in Aras Innovator looks like this ...



Picture 331: Assembly in Aras Innovator

... but in CATIA you don't expand all relations before load (The missing structure could also be filtered out by a configuration) ...

20190425_001, A, 2, , Preliminary (logged in as admin, dat 20190425_001, A, 2, , Preliminary 20190425_001, A, 2, , Preliminary 20190425_002, A, 2, , Preliminary 20190425_002, A, 2, , Preliminary
20190425_001.CATProduct (Current)
20190425_001 20190425_002 (20190425_002.1) 20190425_003.1 [20190425_003.CATProduct]

Picture 332: Structure in PDM Workbench and CATIA window

... then you delete the broken link in CATIA.

20190425_001.CATProduct (Current)	
 20190425_001 20190425_002 (20190425_002.1) Applications 	y z

Picture 333: Delete broken link

During "Update" the relation of the broken link in Aras Innovator is deleted if existing and possible.



Picture 334: Delete instance relation

Bounding Box Management / "Show Neighbor" Functionality

The 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, you want to find out which bounding boxes of other CATParts in the structure "Engine.CATProduct" overlap with the bounding box of the CATPart "CrankShaftLeft.CATPart" (see *Picture 335: CATPart geometry in the context of a CATProduct structure*).



Picture 335: CATPart geometry in the context of a CATProduct structure

This is the corresponding CAD structure in PDM (see *Picture 336: CATPart document in CAD structure*).



Picture 336: CATPart document in CAD structure

In the first step you search for the CATPart document which you want the neighborhood of. Then you click on the context menu "Show Neighborhood" (see *Picture 337: Action "Show Neighborhood"*).

PDM Query (logged	in as admin, database: Innovator	Solutions1	.112)							X
PDM object type				Doc. Number		Maior Rev.	Generati	on	Name	State
All CATIA Files		-	a	CrankPin		A	1			Prelimina
			5	CrankPinModel		A	1			Prelimina
PDM object attribut	tes		-	Crankshaft_Assm		А	1			Prelimina
Doc. Number Cr	* 👻		(CrankshaftLeft		А	1			Prelimina
Major Rev.	•		(ک	CrankshaftLeftModel		A	1			Prelimina
Generation	•		٠	CrankshaftRight		A	1			Prelimina
Name	•		2)	CrankshaftRightModel		0	4147 1			Prelimina
State					5	Open in new PDIV	/ window			
Description	· · · · · · · · · · · · · · · · · · ·				14	Unclaim				
					2					
					1.5					
Created on						0 51				
Mad Cad an	•					Open File				
Modified on	•				1	Open File tempor	arily			
Created by	•				_	Show Neighborh				
Modified by	-				50	Create new versio	n			
Locked by	•				1	Delete newest ver	sion			
Is Template						D I I	31011			
Standard Part					\rightarrow	Duplicate				
External Owner -		*	_		* *	Expand	•			
External Owner []-	Systems.Mechanical.CATIA 👻		•	III	_					•
								Load	with Links 🔲	Load with Drawing
Query										Close
										Ok

Picture 337: Action "Show Neighborhood"

A query window appears where you can search for a CATProduct document (or an Assembly part in the BOM Part Structure Data Model) (see *Picture 338: "PDM Query" dialog for context Assembly node*).

M object type			Doc. Number 🖉	Maior Rev.	Generation	Name	State	D
ATProduct		- 🔊	Engine	A	1		Preliminary	
M object attributes								
Doc. Number Engine	•							
Major Rev.	-							
Generation	-							
Name	-							
State	-							
Description								
Created on								
Modified on	-							
Created by	-							
Modified by	-							
Locked by	-							
Is Template								
ernal Owner T-Systems.Mechan	ical.CATIA 👻 *							
		•		III				

Picture 338: "PDM 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 you double-click 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 (see *Picture 339: Reduced PDM structure containing only neighbor models*).



Picture 339: Reduced PDM structure containing only neighbor models

When this structure is loaded to CATIA you can see the geometry where the bounding boxes overlap with the originally selected CATPart's bounding box (see *Picture 340: Reduced structure loaded to CATIA*).



Picture 340: Reduced structure loaded to CATIA

If the selected structure does not contain the selected CATPart you will receive a warning message (see *Picture 341: The selected structure does not contain the selected CATPart*).

PDM Workbench	? X
- Information	
PDM Messages:	
Selected CATPart for neighborhood search was not found under the selected context or has no bounding box. (Warning)	
Warning.	
i 💦	Close

Picture 341: The selected structure does not contain the selected CATPart

Automatic Part Creation in CAD Document Structure Data Model

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

You 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 (see *Picture 342: "Create corresponding BOM parts" check box*).

Update to PDM?							- 🗆 🗙
File name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Result
CrankshaftRightModel.CATPart	CrankshaftRightM				Modified	Create	
CrankPinModel.CATPart	CrankPinModel				Modified	Create	
CrankshaftLeftModel.CATPart	CrankshaftLeftMo				Modified	Create	
CrankshaftRight.CATProduct	CrankshaftRight				Modified	Create : 1. Create PDM File Object of typ	
CrankPin.CATProduct	CrankPin				Modified	Create : 1. Create PDM File Object of typ	
CrankshaftLeft.CATProduct	CrankshaftLeft				Modified	Create : 1. Create PDM File Object of typ	
Crankshaft_Assm.CATProduct	Crankshaft_Assm				Modified	Create : 1. Create PDM File Object of typ	
Create new CAD document gener	ation? <u>No</u>	Create corresponding BOM parts?	Yes			Upo	late Close

Picture 342: "Create corresponding BOM parts" 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 (see *Picture 343: CAD structure with related Part items*).



Picture 343: CAD structure with related Part items

Support for the new CAD Structure Instance Handling introduced in Aras 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 (see *Picture 344: Structure with four instances*).



Picture 344: Structure with four instances

The four instances are stored in one CAD Structure relation which contains four CAD Instance relations (see *Picture 345: One CAD structure relation for each used CAD Document*).

6		Aras Innova	tor - Mozilla Firefox				x
(i) localhost/InnovatorServer11	2/Client/X-salt=std_11.0.0.6920-X/scrip	ts/Innovator.aspx			120	6 … 🛛 🏠] ≡
aras INNOVAT	'OR°					8 Innovator Ad Logout	dmin
77F7BC4E822E43E4#	a 20180511_StdPart001	💶 CAD 🛛 🔽 Part 🛛	08EE38283F1F43B79	🕺 All Employees 🦷 🗟	Rack_Assm ×		
File Edit Views Searc	h Actions Reports Tools	Help					
🕂 🗎 🗙 😏 🖶 I	💵 🚳 🔒 💼 🏷 🖉 🖞	? 🗸					
CAD Document	Document Number Revi Radc_Assm A Name A Type Authoring Te Assembly CATIA Description CATIA	sion State Preliminary	Seekt on mage				**
	Rep Туре		Native File				
	Default 3D		Rack Assm.CAT	-			
Structure Parents Parts	Files Changes CAD DesignTable		-			-	
Actions * Pick Related		i 🖬 (;)4)4	Hide Search Criteria				
Sequence Document Nur	mber Revision Nam	le	Type State	Native File []	Viewable File []	authoring_too	bl
iiii 128 Rack	A	Mechan	ical/Assembly Preliminary	Rack.CATProduct	1 Select file C	ATTA	
1256 Screw	A	Mechan	ical/Assembly Preliminary	Screw.CATProduct	1 Select file C	ATTA	
<	ш						3
Ready		Items 1-2.Page 1.	OF Items 2.Pages 1.			Aras Innov	/ator

Picture 345: One CAD structure relation for each used CAD Document

The CATIA instance information is stored in the CAD Instance relations (see *Picture 346: CAD Instance information*).

						ras Innovat	tor - Mozili	a Firefox								
() localhos	t/InnovatorServer1112/	/Client/X-salt=sl	td_11.0.0.69	20-X/scripts	/Innovator.asj	XX								120%	🛛	1 🗘
a r a INI	νονοι	0R°												2	Innova Logoul	ator Adn <u>t</u>
≡ 4 77F7	BC4E822E43E4A	20180511	_StdPart001	I CA	AD 💶 Pa	rt 🍐 O	8EE38283F	1F43B79	🕺 Al E	mployees	💿 Ri	ick_Assm	D42	9C022ABE	C4D76	× -
File Edit	Views Search	Actions R	Reports	Tools H	elp											
루 🗎	x 🖸 🖶 🚳	I 🚺 🔒	∎ う	¢° 🖞	? 🗸											
CAD SU	ucture	Document N	umber													
Instances	ucture	Screw	umber •	•												_
Instances Actions *	No Related •	Screw	umber		€ °° A	۰. <u>م</u>	∂ Hide	Search Criteria) tī	13				_
Instances Actions * Sequence	No Related No Related	Screw	umber • • • • •		R (* 1	۲. <u>مر</u> ۱	الله المعالم ال Transforma	Search Criteria ation Matrix) †Ξ	13 Et		à IS		_
Instances Actions * Sequence 128	No Related •	Screw	umber scription		₽ ° A	9991749e-	Hide Transforma 093 2.1347	Search Criteria ation Matrix 906292448396	• 046 0 -	7.22353401) †Ξ 0	JE E		a 16		_
Instances Actions * Sequence 128 256	Na Related	Document N Screw	umber scription		-6.86235730 I -6.87524974	19991749e-1	Hide Transform 093 2.1347 092 3.0106	Search Criteria ation Matrix 906292448394 021694478496	e-046 0 -: e-046 0 -:	7.22353401) † Ξ	JE E		à 15		_ 1
Instances Actions * Sequence 128 256 384	No Related No Rel	Screw	escription		€ ¢° ↓ -6.86235730 -1.37524974	19991749e-1 14310079e-1 1457253e-0	Hide Transform 093 2.1347 092 3.0106 062 2.79164	Search Criteria ation Matrix 906292448390 021694478490 19284397097e	e-046 0 -: e-046 0 -: e-046 0 -1	7.22353401 1.42618492 .215432671) 1 =	13	, 642 (1		_

Picture 346: CAD Instance information

"CAD is Master for Instances" Functionality

The PDM Workbench always controls instances by PDM. It reads the instance information from PDM (position, instance name, number of instances). It stores all instance information in PDM, by creating instances.

With this functionality, when a CAD structure is loaded from PDM, the instance information from the CATProduct file is taken, the instance information from PDM is ignored.

At "Update" the instance information in PDM is updated from the current values of the CATProduct, as before. The difference is that the "Load" process is not dependent of the correct, or even existing, instance information in the CAD structure to be loaded.

Check for CAD Document CATIA Release at "Update" Process

This functionality optionally asks before overwriting a file which has been created with a lower release of CATIA V5.

If you are about to overwrite a file which has been created with a lower release of CATIA V5, you are asked whether you want to continue (see *Picture 347: Asking the user whether to continue the "Update" process*).



Picture 347: 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 you click on the "Local Workspace" icon (see *Picture 348: "Local Workspace" icon*), a window containing a list of CATIA files appears (see *Picture 349: "Local Workspace" window*).



Picture 348: "Local Workspace" icon

→

Modified	File Name 📧	Part Number	Major Rev.	Generation	Name	State	Description	Created on	Modified on
No	ConnectionRod.C	ConnectionRod	A	1		Preliminary		2018-04-06T11:16:50	2018-04-06T11:17:
No	ConnectionRodMo	ConnectionRodMo	A	1		Preliminary		2018-04-06T11:16:48	2018-04-06T11:17
Yes	CrankPin.CATProd	CrankPin	Α	1		Preliminary		2018-04-06T11:16:58	2018-04-06T11:17
Yes	CrankPinModel.CA	CrankPinModel	A	1		Preliminary		2018-04-06T11:16:56	2018-04-06T11:17
Yes	Crankshaft_Assm	Crankshaft_Assm	Α	1		Preliminary		2018-04-06T11:17:04	2018-04-06T11:17
Yes	CrankshaftLeft.CA	CrankshaftLeft	A	1		Preliminary		2018-04-06T11:17:02	2018-04-06T11:17
Yes	CrankshaftLeftMo	CrankshaftLeftMo	Α	1		Preliminary		2018-04-06T11:17:00	2018-04-06T11:17:
Yes	CrankshaftRight.C	CrankshaftRight	A	1		Preliminary		2018-04-06T11:16:54	2018-04-06T11:17
Yes	CrankshaftRightM	CrankshaftRightM	A	1		Preliminary		2018-04-06T11:16:53	2018-04-06T11:17
No	Cylinder_Assm.CA	Cylinder_Assm	A	1		Preliminary		2018-04-06T11:16:46	2018-04-06T11:17
No	CylinderBlock.CAT	CylinderBlock	A	1		Preliminary		2018-04-06T11:16:18	2018-04-06T11:17
No	CylinderBlockMod	CylinderBlockModel	A	1		Preliminary		2018-04-06T11:16:16	2018-04-06T11:17
No	Engine.CATProduct	Engine	A	1		Preliminary		2018-04-06T11:17:06	2018-04-06T11:17
No	FixedParts_Assm.C	FixedParts_Assm	A	1		Preliminary		2018-04-06T11:16:30	2018-04-06T11:17
-	NewPart10.CATPart								2018-04-09T13:32
-	NewPart11.CATPart								2018-04-09T13:32
-	NewPart12.CATPart								2018-04-09T13:32
-	NewProduct10.CA								2018-04-09T13:32
-	NewProduct11.CA								2018-04-09T13:32
-	NewProduct12.CA								2018-04-09T13:32
No	Piston.CATProduct	Piston	A	1		Preliminary		2018-04-06T11:16:44	2018-04-06T11:17
No	PistonModel.CATP	PistonModel	Α	1		Preliminary		2018-04-06T11:16:42	2018-04-06T11:17
No	PistonPin_Assm.C	PistonPin_Assm	A	1		Preliminary		2018-04-06T11:16:40	2018-04-06T11:17
No	PistonPincomp.CA	PistonPincomp	A	1		Preliminary		2018-04-06T11:16:38	2018-04-06T11:17
No	PistonPincompMo	PistonPincompMo	A	1		Preliminary		2018-04-06T11:16:36	2018-04-06T11:17
IND	PistonPincompivio	PistonPincompinio	A	1		Preliminary		2010-04-00111:10:50	2018-04-00111:17

Picture 349: "Local Workspace" window

Configuration of BOM Part Structure

In the BOM Part Structure Data Model it is possible to create product configurations where, depending on 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 (see *Picture 350: Creating category "Color"*).

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> Configuration		
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> Localization	● Color ☆ 戸	
> Mass Operations		
> Notification		
V PDM Workbench	∧ Category	
> BOM Config Management		
BOMConfigurations	Name Mandatory Mutually Exclusive	
Categories	Q	
ConfigurationExpressions		

Picture 350: Creating category "Color"

Then option items which refer to the category "Color" are created, in this case named "Blue", "Green", and "Yellow" (see *Picture 351: Creating options "Blue", "Green", and "Yellow"*).

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Categories	Category	
ConfigurationExpressions	Cotor	
Lots		

Picture 351: Creating options "Blue", "Green", and "Yellow"

Then BOMConfiguration items are created which refer to these color options. The names are "BlueConfig", "GreenConfig", and "YellowConfig" (see *Picture 352: Creating BOMConfiguration items*).

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PWB Configurations			
> Secure Social		Lot Option	
> System Events	*		

Picture 352: 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 (see *Picture 353: Creating Configuration Expression items*).

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H PWB Configurations	Name Lot [] Range from Range to Date from [] Date to [] Milestone fro Milestone to [Rank from	
> Secure Social	BlueConfig	_
> System Events		_

Picture 353: Creating Configuration Expression items

Then a sample CATIA structure is imported, creating a PDM structure in Aras Innovator (see *Picture 354: Sample CATIA structure*).



Picture 354: Sample CATIA structure

The previously created ConfigurationExpression items can be related to either "Part BOM" or to "BOM Instance" relation items (see *Picture 355: Relating configuration expressions to PLM relations*).

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Part Number Northing Nation Assigned Creator Name Partimenary Displanted User Page User Marker Stary Electric table Campes Preding © BOM BOM Structure Alternates AML Documents CAD Documents Goals Changes © Parts ~ \hat{x} © Grade Preding Image Structure Alternates AML Documents CAD Documents Goals Changes © Parts ~ \hat{x} Image Structure Alternates AML Documents CAD Documents Goals Changes Image Structure Alternates AML Documents CAD Documents Coals Changes Image Structure Alternates AML Documents CAD Documents Coals Changes Image Structure Alternates AML Documents CAD Documents Coals Changes Image Structure Alternates AML Documents CAD Documents Coals Changes Image Structure Alternates AML Documents CAD Documents Coals Changes Image Structure Alternates AML Documents CAD Documents Coals Changes Image Structure Alternates AML Component 2 Image Structure Alternates AML C	/ State Preliminary Q 28 Configurat ×	Unit EA EA	Reference Designator	Configuration_	Configuration Dapress	Chan_	
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Picture 355: 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") (see *Picture 356: Creating Product Variant items*).



Picture 356: Creating Product Variant items

Now the previously created PDM structure can be expanded and loaded in different configurations (see *Picture 357: Setting a product variant for the BOM part expansion*).



Picture 357: Setting a product variant for the BOM part expansion

First, if no configuration is set, the complete structure is expanded and loaded (see *Picture 358: Expanding and loading the complete structure*).



Picture 358: Expanding and loading the complete structure

Then, if a particular product variant is set (see *Picture 359: Setting different product variant expand filters*), expanded, and loaded, then only the configured parts are expanded and loaded (see *Picture 360: Loaded the "Blue" variant (one BOM Instance), Picture 361: Loaded the "Green" variant (one Part BOM with all instances),* and *Picture 362: Loaded the "Yellow" variant (one BOM Instance)*).



Picture 359: Setting different product variant expand filters



Picture 360: Loaded the "Blue" variant (one BOM Instance)



Picture 361: Loaded the "Green" variant (one Part BOM with all instances)



Picture 362: Loaded the "Yellow" variant (one BOM Instance)

Check CAD Links

When CATIA documents with 3D links need to be imported this functionality helps you 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 you 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 (see *Picture 363: "Check CAD Links" icon*).



Picture 363: "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 (see *Picture 364: Result of "Check CAD Links" action*).

Resul											
CAT	IA Documents in PI	ом									
	Doc. Number	Major Rev.	Generation	Name	State	Description	Created on	Modified on			
	LinkSourcePart1	Α	1		Preliminary		2014-10-30T16:23:08	2014-10-30T16			
	LinkSourcePart2	Α	1		Preliminary		2014-10-30T16:23:17	2014-10-30T16	i:		
	PDM Workbench										?
	CATIA Link Cher Closed "D:\Data' Please first load LinkSourcePart LinkSourcePart Then open "D:\C Please create the LinkSourcePart LinkSourcePart Make sure "D:\D	:k Result: \CatiaFiles\Lin (or close and .CATPart, .CATPart Data\CatiaFile en the followi .CATPart, .CATPart ata\CatiaFiles	nks\LinkTargı re-load) the f s\Links\LinkT ng CATPart d s\Links\LinkT	etPart2.C following argetPar ocumen argetPar	ATPart" so lin g CATPart doc rt2.CATPart" a ts in PDM: t2.CATPart" is	iked documer iuments form igain. i opened while	nts can be removed fro PDM: : you do that, and that	m the CATIA ses the CATIA links	point	to the correct documents.	
	V Action su	cceeded.									
											Close

Picture 364: Result of "Check CAD Links" action

Displaying PDM Structure Instances as separate Nodes

The display of the PDM 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 PDM structure containing several instances of the same part is expanded then all the instances are shown as separate nodes (see *Picture 365: PDM structure showing every instance as a separate node*).



Picture 365: PDM 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.



The content of any PDM Structure window can be saved to a PWBDoc file (see *Picture 366: Example content of a PDM Structure window*).

Picture 366: Example content of a PDM Structure window

The content of this window can be saved by selecting "File \rightarrow Save As" from the menu (see *Picture 367: PWBDoc save dialog* and *Picture 368: Saving the window content under a specific name*).

Save As	and the second					×
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🛛 🕒 Öffer	ntliche TV-Aufzeichnungen	=				
🛛 📗 Öffer	ntliche Videos					
퉬 Öffer	ntlicher Desktop					
Þ 퉲 PDM	Workbench					
Þ 퉲 PWB	CACHE					
🍌 PWB	_XMAP					
PWB	_XMAP1109					
PWB	_XMAP1112					
Dateiname	: 11.PWBDoc					•
Dateityp	PWBDoc (*.PWBDoc)					
Ordner ausblend	len	Sav	re as new document	Speichern	Abbrech	nen

Picture 367: PWBDoc save dialog

😋 🔵 🗢 📙 « System (C:) 🕨 Benutzer 🕨 Öffentlig	ch ▶ PWB_XN	MAP 👻 🍫 PWB_XMAP durch	suchei
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PDM Workbench			
PWB_CACHE			
🐌 PWB_XMAP			
PWB_XMAP1109			
PWB_XMAP1112			
Dateiname: Session1.PWBDoc			

Picture 368: Saving the window content under a specific name

After saving, the new PWBDoc file can be seen in the Windows Explorer (see *Picture 369: Newly created PWBDoc file*).



Picture 369: Newly created PWBDoc file

In the same session, or in a later session, this file can be opened again (see *Picture 370: Opening a PWBDoc file (1/2)*, and *Picture 371: Opening a PWBDoc file (2/2)*).



Picture 370: Opening a PWBDoc file (1/2)
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Öffentliche Videos			
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Dateiname:			All Files (*.*)
		Open as read-only	Öffnen 🔫 Abbrechen

Picture 371: Opening a PWBDoc file (2/2)

The file can also directly be opened from the most recently file list (see *Picture 372: Opening a PWBDoc file from the most recently used file list*).

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\Leftrightarrow	. New	Ctrl+N	1 🗟 🗐		
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	Save Save As	Ctrl +S			
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	<u>2</u> CatalogDocumentL.cat	alog			

Picture 372: Opening a PWBDoc file from the most recently used file list

You have 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 (see *Picture 373: PDM Structure window opened from PWBDoc file*).



Picture 373: PDM Structure window opened from PWBDoc file

Allow deactivated CATProduct and CATPart Instances

It is possible to import and update a structure which contains deactivated nodes ("Case.1" in the picture, as opposed to "Screw.1", where only the representation is deactivated) (see *Picture 374: CATProduct structure with a deactivated node*).



Picture 374: CATProduct structure with a deactivated node

Previously, deactivated nodes were treated as not existing. With this new functionality the nodes can be treated like regular activated nodes.

Also, the activation state of a CATPart or CATProduct instance can be passed to a custom method when the corresponding PDM relation is created, making it possible to different parameter values based on the activation state.

Setting Configuration Information on Structure Relations

It is possible to set configuration information (configuration expression items) to Part BOM and BOM Instance relations in the PDM Structure window.

You can set or remove configuration expression items on the instance (BOM Instance) or reference (Part BOM) relation with the sub-menu actions of the context menu "Update Relation Configuration" (see *Picture 375: Action "Update Relation Configuration" sub-menu*).

Update Relation Con <u>f</u> iguration	Set Instance Relation Configuration
Custom Action	Set Reference Relation Configuration
	Remove Instance Relation Configuration
	Remove Reference Relation Configuration

Picture 375: Action "Update Relation Configuration" sub-menu

The relation has to be claimed by you for this.

A configuration expression on a Part BOM relation looks like this ... (see *Picture 376: Configuration expression on Part BOM relation*)

в	OM BOM St	ructure Alte	mates	AML C	ocuments	CAD Docur	nents Goals	Change	s Part Submission W	arrants			
Ac	tions + Pick F	Related 🔹	- P	\times	I 🚺 🔒	💼 🖉 🖌	A A 6	Hide Search	Criteria 🔹	19 49 🖟 🔛	10 Ib		
	Sequence 🛦	Part Number	Revision	Name	Туре	Quantity	State	Unit	Reference Designator	Configuration flag	Configura	tion Expression []	Changes
	128	NewPart3	А		Component	2	Preliminary						
Ê	256	NewPart4	Α		Component	2	Preliminary				r6-10		

Picture 376: Configuration expression on Part BOM relation

... and a configuration expression on a BOM Instance relation looks like this (see *Picture 377: Configuration expression on BOM Instance relation*).



Picture 377: Configuration expression on BOM Instance relation

Released Cache Mode

The PDM Workbench can efficiently use CGR cache files, which are stored in the Aras Innovator database, for the geometrical visualization of product structures. This is useful when loading large product structures in visualization mode to CATIA.

The PWB cache mode uses CATIA tools to create CGR cache files based on native files like CATIA V5 CATParts or CATIA V4 models, uses Aras Innovator to store and manage these files linked to a CAD Document item in Aras Innovator and uses the CATIA released cache capability to provide the data to you when loading a product structure.

If you use the PWB cache and have loaded a product structure in your current CATIA session in visualization mode, you can switch one or multiple visualized objects to design mode by using the regular CATIA commands or mouse actions (e.g. double-click on a visualized part) for this task.

This works fine even if you have decided to set your "PDM Workbench Options" setting to "Download cache file only", because the PDM Workbench will download any missing native file whenever CATIA requests it. However, this is not very efficient for switching many objects or larger portions of the product structure to design mode, because the PDM Workbench will then download one requested file after the other.

If the CATProduct contains links that force the load of related CATIA documents, the corresponding native files will be downloaded with each file separately. In such cases it is recommended to use the "PDM Workbench Options" setting "Download native file and cache file" – see next chapter for details.

Set the Cache File Download Mode in User Settings

Each user can define the personal download mode related to cache files in the PDM Workbench settings.

PDM Workbench Options
Version PDM Workbench CATIA V5 Module Version: 6.0.0 CATIA Level: V5R24 (c) T-Systems 2005 - 2016
Query Dialog
Keep Query Dialog Values for other Types Customize List View
PDM Relations
Loading related Files
Load with Links
Enable Hide/Show
Cache file download mode Do not use Download native file and cache file If claimed, download native file, otherwise cache file Download cache file only
Expand Neighborhood 0 [mm] Clearance for Expand Neighborhood
Copy Position
OK Scancel

Picture 378: "PDM Workbench Options" dialog - Setting the Released Cache options

You should set it according to your regular or current task. Any change will have immediate effect - from the next download on - for files, which do not exist in the local PDM Workbench exchange directory or are out of date there.

• Do not use.

This is the mode of previous PDM Workbench releases. Only native files will be downloaded from the server, even if cache files are provided there.

This mode can still make sense, if you regularly perform a detailed design based on a small number of loaded files.

• Download native file and cache file.

Both files are downloaded for each CAD item, the native file, and the cache file.

This mode makes sense, if you want a fast initial visualization of your product, but usually switch most or all of it to design mode later.

• If claimed, download native file, otherwise cache file.

The download will distinguish between items, which are claimed by the current user, and other files.

It only makes sense for small product structures and if your work methodology is in a way, that you generally claim all items in PDM that you expect to work on, before you load a product structure to CATIA.

• Download cache file only.

The native file would not be locally available for immediate access but can be downloaded on demand later.

This mode makes sense for design reviews, where you need the visualization data only, and for loading large product structures, where you might switch only a small portion of it to design mode later.

Command "Get original Geometry"

The command "Get original Geometry" provides an efficient way to collect your requests and download multiple missing files in one shot. Select your CATIA objects like you would do it for the "Switch to design mode" command and call the contextual PDM Workbench command "Get original Geometry". This would download any missing native file of your selected scope to your PDM Workbench exchange directory (see *Picture 379: Action "Get original Geometry"*).

Your CATIA selection remains active at the end of this command, so it is up to you to immediately switch these objects to design mode after the download or to continue working in visualization mode for some time and to switch to design mode any time later.



Picture 379: Action "Get original Geometry"

"Open in CATIA" from the Aras Innovator Client

Single CAD or Part items, or structures, can be loaded in CATIA from the Aras Innovator web client.

The precondition for this functionality is that CATIA V5 is started with PDM Workbench, and the user is logged in.

The "Open in CATIA" action in the Aras Innovator web client can be used to load the selected item in CATIA, if it is a component, or to expand the Assembly structure and load it in CATIA.



Picture 380: "Open in CATIA" context action

Depending on the selected data model a dialog is opened when the user selected "Open in CATIA" for an item in the Aras Innovator client.

In the CAD Document Structure Data Model are some options

- Add Temp The selected item and its structure is loaded temporarily in CATIA V5.
- Load with Links The selected CAD Document is loaded with the related items.
- Load with Drawing The selected CAD Document is loaded with the related drawings.

and the expand mode available:

As Saved

The structure is expanded "As Saved".

Latest Released

The structure is expanded "Latest Released".

• **As Current** The structure is expanded "As Current".

Open in CATIA Engine,Engine,A,1 $ imes$
Options Add Temp Load With Links Load With Drawing
Expand Mode As Saved C Latest Released As Current
Open in CATIA Cancel

Picture 381: "Open in CATIA" dialog – CAD Document Structure Data Model

In the BOM Part Structure Data Model are some options

- Add Temp
 - The selected item and its structure is loaded temporarily in CATIA V5.
- Load with Links The selected CAD Document is loaded with the related items.
- Load with Drawing

The selected CAD Document is loaded with the related drawings.

Open in CATIA Eng	ine,Engine,A,1	\times
Options Add Temp Load With Links Load With Drawing		
Open in CATIA	Cancel	

Picture 382: "Open in CATIA" dialog – BOM Part Structure Data Model

		_
Loa	ad <u>23</u>	
	Loading selected object from PDM system	
	Downloading files	
	Status : 50% completed	
	Cancel	
2		DS

Picture 383: Loading the structure in CATIA



Picture 384: The loaded structure

"Open in CATIA" from the Aras Innovator Client with Construction Space

The Construction Space feature enables the user to set a construction space (assembly/installation space) to filter CAD structures.

The construction space itself represents an axis-aligned bounding-box which is defined by an min point and a max point. This axis-aligned bounding box represents a virtual threedimensional cuboid, whose sides are the boundaries for the construction space.

Only structures which are located inside the construction space or intersect the construction space are loaded into the CAD application. Because of that the Construction Space feature is integrated into the "Open in CATIA" functionality. The feature is compatible with both CAD and BOM mode for "Open in CATIA".

→

Configuration

When installing the "Open in CATIA" data model the Construction Space feature is already included. Configurations might need to be done at the "PwbConstructionSpace" life cycle. *Please be aware that names of the life cycle states should not be modified to ensure the functionality of the Construction Space feature.* The identities to promote life cycle states can be modified to your needs.

Access Rights

In the TOC, the Construction Space can be found at Administration → PDM Workbench → Construction Space.

The identity "All Employees" has access to the Construction Space ItemType. Restrictions can be made via the TOC editor.

The usage of the Construction Space ItemType is recommended for the usage of the Construction Space feature.

Construction Space items which are in the "Released" state can be accessed by all users. In the "Draft" state only the creator can use the Construction Space for testing. If only creators should be allowed to access the Construction Space, then it should be left in the "Draft" state. All users of the "Aras PLM" identity can promote the life cycle state of the construction space. In the "Obsolete" state the construction space is no longer shown. The identities to promote life cycle states can be modified in the life cycle map.

Usage

In *Picture 385: Create a "Construction Space"* item, the fields marked yellow (Name and Bounding Box) are mandatory. Optionally a description and thumbnails of three different views can be set.

In the Relationship Tab the user can link CAD Documents and Parts to the construction space item. The construction space can then be found when using the "Open In CATIA" functionality with the linked CAD Documents or Parts.



Picture 385: Create a "Construction Space" item

In *Picture 386: "Open In CATIA" option selection* you can see the "Set Construction Space" option. To use the Construction Space feature you have to select "Set Construction Space".



Picture 386: "Open In CATIA" option selection

The previously created construction space linked to the CAD file from which the "Open in CATIA" function was opened is shown when the construction space is selected in the dropdown list. Please be aware that the bounding box values in *Picture 387: "Open in CATIA" with predefined Construction Space* are editable but not saved in the Construction Space item. Permanent changes should be made in the construction space item.

Options	Select Construction Space			
Add Temp	Piston + Cylinderblock	\checkmark		
Load With Links	XMin	XMax	Side View	
Load With Drawing	80	180		
Set Construction Space	YMin	YMax		
	140	240		
Expand Mode	ZMin	ZMax		
As Saved	245	345		
Latest Released				
O As Current	Description			
	Engine Test Construction Spa	ace for CylinderBlock and Piston		

Picture 387: "Open in CATIA" with predefined Construction Space

The message "No related Construction Space items found. Please set own Bounding Box values." appears if there is no Construction Space item linked to the current file. In this case the user can manually set a bounding box by filling the bounding box values into the input mask (see *Picture 388: "Open In CATIA" without predefined Construction Space*).

Options Add Temp	No related Construc Please set own Bou	tion Space Items found. nding Box values.	
Load With Links	XMin	XMax	
☐ Load With Drawing ✓ Set Construction Space	YMin	YMax	
Expand Mode Saved	ZMin	ZMax	
O Latest Released O As Current			
	_		
Open in CATIA Cancel			

Picture 388: "Open In CATIA" without predefined Construction Space

With a click on "Open in CATIA" the structures within or cutting the construction space are loaded into the CAD application.

"Open in Aras" from CATIA V5 Client

Single CAD or Part items can be loaded in the Aras Innovator web client from CATIA.

The existing Aras Innovator web client session of the default browser will be used to open the object. If there is no session running, the action will open the "Login" dialog first.

The "Open in Aras" action in CATIA V5 can be used to load the selected item in the Aras Innovator web client.



Picture 389: "Open in Aras" context action



Picture 390: "Open in Aras" toolbar action

Create Drawing CAD Document: Automatically select loaded Part in Session if a single Link exists

If a CATDrawing contains a single link to a 3D CATIA file then that file will be pre-selected in the "PDM Create" dialog for the CATDrawing.

In this example a CATPart has already been created in PDM (BOM Part Structure Data Model) (see *Picture 391: Single drawing link to a CATPart*).



Picture 391: Single drawing link to a CATPart

A CATDrawing with views to the CATPart's geometry is created. The "PDM Create" dialog for the CATDrawing has the CATPart's PDM items already pre-selected (see *Picture 392: CATPart's PDM items pre-selected in "PDM Create" dialog*).

PDM Create	<u>ि</u> २ – ×	
Create dialog Drawing1	for type Component:	
Part Number	PistonModel *	٦
Name		
Unit		
Make / Buy		
Description		
Create dialog Drawing1	for type CATDrawing:	
Correspondin	g part PistonModel, A, 1, , Preliminary 👻	
Doc. Nu	Imber PistonModel	
	Name	
Descr	iption	
	OK Apply Cancel	

Picture 392: CATPart's PDM items pre-selected in "PDM Create" dialog for CATDrawing

Manage Context Products

A user can only work in one Context Product at one time. This Context Product is used to store newly created files to the correct vault.

If the functionality is enabled, there must be at least one valid Context Product for every user.

The Context Product is set during the "Login" process in CATIA. If there are multiple valid Context Products, the user must select one. If there is no Context Product available for the user, the user cannot login.

Available (Context Prod	ucts	
Product X			
Product 0			
Product 1			

Picture 393: Select Context Product

The currently used Context Product can be seen in the "PDM Workbench Options" dialog.

Copy Position	
Copy Position	
Load With Links	
Load current generatio	n of linked Document
Load multiple levels of	linked Documents
Context Product	
Context Product of Session: Product 0	
	OK Gancel

Picture 394: Currently used Context Product

A designer must select a Context Product during selection of the CATIA start script. To avoid a second select it is possible to use an environment variable to set the Context Product.

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 395: PDM Workbench options*).



Picture 395: PDM Workbench options

Query Dialog

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 396: "Customize List View" dialog*).

PE	OM Session				l	? ×
	Attributes PDM Types		– Attribute Settings –			[
	Part Assembly Component Std. Part All CATIA Files CATPart CATProduct CATDrawing catalog cgr model	4 III >		Add -> <- Remove		Up Down
	Reset All				Preview	
						Close

Picture 396: "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 397: "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.

1	PDM Session					? ×
	Attributes		Attribute Settings —			1
	Part	*	Exclude-List of : /Par		/Part/Assembly	
	Assembly Component Std. Part All CATIA Files CATPart CATProduct CATDrawing catalog cgr model	4 III		Add -> <- Remove	Part Number Major Rev. Generation Name State Description Created on Modified on Created by Modified by	Up Down
	Reset All				Preview	
						Close

Picture 397: "Customize List View" dialog for "Assembly"

By clicking on the "Preview" button you can see a preview of the list view (see *Picture 398: Preview of the "List View" dialog*).



Picture 398: Preview of the "List View" dialog

PDM Relations

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

CATDrawings

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

Loading related Files

Defines whether only the selected structure, or also files which are related by drawing or reference links should be downloaded.

Loading PDM Structures

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

Cache File Download Mode

Defines the options of the "Released Cache" functionality. Please refer to the "Released Cache Mode" chapter for more information.

Expand Neighborhood

Defines the clearance in millimeters for the "Expand Neighborhood" functionality.

Copy Position

Defines whether the position information of copied relations should also be copied to the new relations.

PDM Session Configuration

No configuration has to be set currently from CATIA for the Aras Innovator 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 399: PDM Workbench toolbar after login*) in CATIA V5 ...

PDM Workbench		
I 🔍 🔀 🗿	1 2 @ 4 2	3

Picture 399: PDM Workbench toolbar after 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 be added 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 "PDM 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 *Picture 400: Action "Copy element attributes"*).

🋐 Engine, A, 1, , Preliminary (logged in as	admin, database: Innovator	Solutions1112)	- • •
244			
🚟 Engine, A, 1, , Prelimin	Center graph		
	Reframe On		
	- Hide/Show		
	Properties	Alt+Enter	
	Open Sub-Tree		
		Ctrl+X	
	<u>С</u> ору	Ctrl+C	
	<u>P</u> aste	Ctrl+V	
	Delete	Del	
	Engine, A, 1, , Preliminary	object 🕨	
	Expand	•	
	Expand Multiple Levels	•	
1	De <u>-</u> Expand		
	Compare with Assembly		
	Open in new PDM Window	N	
코	<u>U</u> nclaim		
TE .			
7	Promote		
1-2	Re <u>v</u> ise		
	Load		
	Load <u>w</u> ith Links		
	Load in Context		
B.	Add Temp		
<u>⊳</u>	Highlight CATIA Nodes		
	Copy element attributes		z
→	Create new version		ж
4	Duplicate		

Picture 400: 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 "PDM Create" dialog will be opened. It has the "Insert attribute values" button (see *Picture 401: "PDM Create" dialog for Assembly*).

	PDM Create
	Create dialog for type Assembly:
ļ	Part Number *
ļ	Name
	Unit
	Make / Buy
	Description
	Insert attribute values
	OK Apply OCancel

Picture 401: "PDM Create" dialog for Assembly

Create dialog	for type	Asse	mbly:		
Part Number	Engine				*
Name	2				
Unit	EA				•
Make / Buy	Make	_			-
Description					-
			Insert a	ttribute va	alues
		_		1.000	

When you click on the "Insert attribute values" button the attributes of the dialog will be filled (see *Picture 402: "PDM Create" dialog for Assembly – Inserted attribute values*).

Picture 402: "PDM 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 Aras Innovator Sequence Items

It is possible to optionally use Aras Innovator sequence items to rename CATIA structures or single CATIA documents when they are created (see *Picture 403: CATIA structure before and after import to PDM*).



Picture 403: CATIA structure before and after import to PDM

In the "Login" dialog you can select one of the autoname rule (Aras Innovator sequence item) names (see *Picture 404: "Login" dialog with autoname rule*).

PDM Login				
User informati	on			
User	admin 💌 *			
Password	*******			
Database	InnovatorSolutions 💌 *			
Autoname Rule	•			
	PwbTestSequence1 el PwbTestSequence2 CAD Document			

Picture 404: "Login" dialog with autoname rule

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

Later in the session you 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 *Picture 405: "Set PDM Configuration" dialog* and *Picture 406: Autoname rule combo box in "Set PDM Configuration" dialog*).



Picture 405: "Set PDM Configuration" dialog

Se	Set PDM Configuration				
9	Set PDM Configuration :				
	Autoname Rule			⊡	
		PwbTestSequence1 PwbTestSequence2 CAD Document		1	

Picture 406: Autoname rule combo box in "Set PDM Configuration" dialog

If an autoname rule is selected the "Update to PDM" dialog will contain the information which autoname rule is selected (see *Picture 407: Selected autoname rule displayed in "Update to PDM" dialog*).

U	Jsed autoname rule: PwbTestSe	quence1
	Update	Close

Picture 407: Selected autoname rule displayed in "Update to PDM" 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 *Picture 408: PDM structure named by sequence item*).

PwbTst100000119Suff1, A, 1, , Preliminary, Mechanical/Assembly, (Logged in as Usr1)
WbTst100000119Suff1, A, 1, , Preliminary, Mechanical/Assembly,
AD 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/Part
CAD Sub-Structure (/CAD Structure/Structure, CrankBolt.1) / PwbTst100000117Suff1, A, 1, , Preliminary, Mechanical/Part
CAD Sub-Structure (/CAD Structure/Structure, CrankShaftRight.1) / PwbTst100000116Suff1, A, 1, , Preliminary, Mechanical/Part
CAD Sub-Structure (/CAD Structure/Structure, ConnectionRod.1) / PwbTst100000120Suff1, A, 1, , Preliminary, Mechanical/Part
Ŧ 🖗 CAD Sub-Structure (/CAD Structure/Structure, PistonAssm.1) / PwbTst100000113Suff1, A, 1, , Preliminary, Mechanical/Assembly,

Picture 408: 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.

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



Picture 409: 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	h Attributes Dialog
Part Number	ConnectionRod 🗸
Major Rev.	A 🗸
Generation	1 -
Name	•
State	Preliminary 👻
Unit	EA 🔹
Make / Buy	Make 🔻
Description	
Created on	2014 10 21712/21/22
Modified on	2014-10-21712-21-22
Controlle	2014-10-31113:21:33 ▼
Created by	Innovator Admin 👻
Modified by	Innovator Admin 👻
Locked by	Innovator Admin 👻
	OK S Apply Cancel

Picture 410: Dialog with pre-filled attributes

Glossary

Unclaim	
	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.
Claim	
	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.
Gll	
	Graphics Interactive Interface. The GII is a powerful programming tool, which completes the Open System Access to the CATIA environment.
loon	
	Graphical representation of an object.
Object	
	An item or a relationship.
Query	
	To search the database for objects that match specific criteria.