



PRODUCT RELEASE ANNOUNCEMENT

Product Category	Visualize 3D
Product Group	CATIA V5 to Creo View for Windchill
Release Version	19.3

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Contents

History	2
Product Codes	3
QA Configuration	3
PTC Windchill Support Matrices.....	3
Supported Application Revisions.....	4
Supporting API's	4
Supported Operating Systems.....	4
Distribution Media.....	5
New Features / Enhancements	6
Known Limitations and Restrictions.....	8
Resolved Support Calls	9



History

Revision	Update Information
1.0	Initial Release. Windchill only



Product Codes

Product Code	Product Description
CA5CVW-U1	CATIA V5 to Creo View
CA5CVW-U2	CATIA V5 to Creo View for Windchill
CA5CVW-S1	CATIA V5 to Creo View - PMI Add On
CA5CVW-S2	CATIA V5 to Creo View for Windchill Extended CGR
CA5CVW-S3	CATIA V5 to Creo View for Windchill Post Processing Module
CA5CVW-S4	CATIA V5 to Creo View - JT Add On
CA5CVW-S5	CATIA V5 to Creo View - 3D PDF Add On

QA Configuration

The following software configuration was used by PTC QA for certification against Windchill and Creo View 4.0. For additional supported configurations, see section PTC Windchill Support Matrices

CATIA version	Theorem CADverter	OS platform	Windchill Version
CATIA V5-6R2014	19.3.005	Windows 7	Windchill 11.0 M020
CATIA V5-6R2015		*Windows 10	Windchill 10.2 M030
CATIA V5-6R2016			CPS21

*Only CATIA V5-6R2016 was tested against Windows 10

PTC Windchill Support Matrices

The following is a link to the PTC support pages, (only accessible with a valid PTC customer login). This contains a matrix of supported software configurations. The information found by this link is subject to change and remains the property of PTC. The relevant sections are:

Windchill - Theorem CATIA V5 CADverter Compatibility (page 17)

Theorem CATIA V5 CADverter Feature Dependencies (page 18 & 19)

[Creo View 4.0 Adapters Software Matrices](#)



Supported Application Revisions

The following application revisions have been qualified with this release

Application	Revision
CATIA V5	R19, R20, R21, V5-6R2012, V5-6R2013, V5-6R2014, V5-6R2015 & V5-6R2016

Supporting API's

The following supporting API versions are used

Application	Revision
JTOpen	Jtk8.0
Creo View DAPI	Creo 3.1 - M010 (13.3.10.27)
3D PDF	API HOOPS Exchange Publish 2015 (v8.1)

Supported Operating Systems

The following operating systems have been qualified with this release

Operating System	Qualification Status
Windows 7 64bit	Qualified and fully supported (All CATIA V5 Revisions)
Windows 10 64bit	Qualified and fully supported for CATIA V5-R62016



Distribution Media

CD images of the latest release are available from the following download sites.

Product	URL for CD image download
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Support for ALL Versions	Product Download Information
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New Features / Enhancements

The following new features or enhancements have been introduced with this release. This document also includes fixes

Ref ID	New Feature / Enhancement Description
1	Instance naming can now be displayed honouring the setting in the CATIA V5 Tools>Options->Infrastructure->Product Structure->Nodes Customization panel More information regarding this new feature is included in the documentation
2	Representations for ALL node/ Named node can now be read from CATIA
3	Expanded levels (below part level) can now be read from the specification tree structure
4	Options to read part & assembly level PMI have improved control and granularity, providing clarity of output
5	Edge curves for a planar FTA construction geometry can be created as standalone curves allowing the plane to be viewed 'edge on'.
6	The ability to read solid colours as option as well as face colours.
7	Colour and opacity can be specified to be washed down from the root node of the specification tree.
8	It is now possible to create the wireframe geometry associated to a FTA Thread
9	Allows planar datum geometry in V5 to be read as a face and created to a user defined size.
10	CATParts can now inherit transparency from their parent product node in (CAS-02337)
11	V5 PMI now honours positional and orientation links with other PMI items. Requires Creo View Client from Creo View 3.1 M010 onwards
12	V5 PMI now honours PMI – PMI association for GDT's, Simple Datum's & Datum Target's. Requires Creo View Client from Creo View 3.1 M010 onwards
13	Added magnitude and units to attributes (e.g. mm, m) via the attribute mapping file
14	Axis systems are now added correctly to groups when OUTPUT_MBD is on
15	FTA is correctly scaled and the start and end angle honoured if rotated



16	New OUTPUT_MBD state in options list
17	Improvements in reading multiline text in NOA in regard to positioning and justification
18	Duplicate geometry association when also associated to Simple Datum/Datum Targets has been removed
19	Improvements in reading FTA Construction geometry when referenced from a CAPTURE
20	Improvements in honouring geometry hide / show state from a CAPTURE
21	<p>Features allowing highlighting PMI-PMI have been introduced to complement the PMI-Geometry functionality. Please note</p> <ul style="list-style-type: none"> • this is mutually exclusive to the pmi-pmi link, which associates flat to screen PMI to move together - this is a current limitation • Requires Creo View Client from Creo View 3.1 M010 onwards
22	V5-6R2016 (R26) is now supported. NOTE: Separate media required at this release for Windows 10 support
23	Multi-Fidelity is now supported. The ALTFILE processing has been changed so that if a multi-fidelity JOB file (.paj) is processed, the Creo View DAPI will create the requested additional fidelity .ol files, zip them up and add the zip file to the .pvoa file for storage in Windchill
24	The product is now supported in the Theorem UI. Customers are invited to download this UI from cdCAD_19.3_UI_WIN.01.zip
25	Updates have been made to the attribute filter (see documentation)
26	Extra debug code has been added to aid error tracking. (see documentation)



Known Limitations and Restrictions

The following limitations and restrictions have been identified during final testing prior to release

Ref ID Limitations and Restrictions

1	Although improvements to text positioning within an NOA have been made, there may be some issues if the NOA has texts with differing justifications. CAS-02579
CAS-02654	Layer mask – if a layer mask is reset then the recipe file needs to be edited manually to remove a blank line



Resolved Support Calls

The following list of Support Calls are resolved with this release

Support Call ID	Brief Description
CAS-02153	Data specific issue fixed for Windchill Positioning Assembly Publishing. Also CAS-02154, CAS-02193, CAS-2636
CAS-02391	Improvements to NOA (Note Object Attribute) Block Letters
CAS-02574	Fixed missing instance name and description if adapter/conversionMode=1. This relates to publishing in a Windchill context to facilitate Part and CAD cross-structure relationships.
CAS-02589	Fixed problem where translations were slow & catia_pv continued to run after timeout.
CAS-02646	Tessellation issues for bad faces have been fixed with a undocumented command available to support desk
CAS-02574	Fixed an issue where minimum memory mode was missing instance attributes
CAS-02636	Missing attributes are noew availablewhen using Windchill Positioning Assembly Publishing. See also CAS-02153
CAS-02813	Source_file_name is now set for all assembly translation modes See also CAS-02574
CAS-02589	Fixed tessellation issue rendering large cgr files. Was hours now takes minutes
CAS-01695	<i>The setting for saving CATParts as CGRs can also be used to create cgrs from .model files</i>
CAS-02909	No pvs created on Windows 10 if alternate format is specified in the job file
CAS-02880	Omit bad faces (new command) cures problem when some specific parts fail.
CAS-02579	PMI Positioning and note improvements
CAS-02844	Notes added to recipe file to clarify the use of Additional File Types

