PRODUCT RELEASE ANNOUNCEMENT

Product Category	**Visualize 3D**
Product Group	CATIA V5 to Creo View – Windchill Independent
Release Version	19.3

Document Type	**Product Release Announcement**
Status	Released
Revision	2.0
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Contents

History

Revision	Update Information
1.0	Initial Release
2.0	Separate release document for Windchill support product created. Windchill specific topics removed from this document

Product Codes

Product Code	Product Description
CA5CVW-U1	CATIA V5 to Creo View
CA5CVW-S1	CATIA V5 to Creo View - PMI Add On
CA5CVW-S4	CATIA V5 to Creo View - JT Add On
CA5CVW-S5	CATIA V5 to Creo View - 3D PDF Add On

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 support is used

Application	Revision
JTOpen	Jtk8.0
Creo View DAPI	Creo 3.0 - M030 (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)
Window 8 64bit	Not formally qualified for CATIA V5 R18, R19, R20, R21 and CATIA V5-6R2012 Qualified and fully supported for CATIA V5-6R2013, CATIA V5-6R2014 and CATIA V5-6R2015 Only
Windows Server 2008 R2	Not formally qualified for CATIA V5 R18, R19 and R20 Qualified and fully supported for CATIA V5 R21, CATIA V5-6R2012, CATIA V5-6R2013, CATIA V5-6R2014 and CATIA V5-6R2015 Only
Windows Server 2012	Not formally qualified for CATIA V5 R18, R19, R20, CATIA V5 R21, CATIA V5-6R2012 and CATIA V5-6R2013 Qualified and fully supported for CATIA V5-6R2014, CATIA V5-6R2015 and CATIA V5-6R2016 Only

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
Support for ALL Versions	Product Download Information

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
12	V5 PMI now honours PMI – PMI association for GDT's, Simple Datum's & Datum Target's
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	Features allowing highlighting PMI-PMI have been introduced to complement the PMI-Geometry functionality. Please note • 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 M030 onwards
22	V5-6R2016 (R26) is now supported. NOTE: Separate media required at this release for Windows 10 support
23	The product is now supported in the Theorem UI. Customers are invited to download this UI from cdCAD_19.3_UI_WIN.01.zip
24	Updates have been made to the attribute filter (see documentation)
25	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-02391	Improvements to NOA Block Letters
CAS-02574	Fixed missing instance name and description if adapter/conversionMode=1
CAS-02589	Fixed problem where translations were slow & catia_pv continued to run after timeout.
CAS-02646	Tessellation issues have been fixed with work-arounds
CAS-02589	*Fixed tessellation issue rendering large cgr files. Was hours now takes minutes*
CAS-01695	*The setting for saving CATParts as CGRs can also be used to create cgrs from .model files*
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*