



PRODUCT RELEASE ANNOUNCEMENT

Product Category	Publish 3D
Product Group	CATIA V5 for 3D PDF
Release Version	19.0

Document Type	Product Release Announcement
Status	Released
Revision	1.0
Author	Product Manager
Issued	12/01/2016



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History

Revision	Update Information
1.0	Initial Release

Product Codes

Product Code	Product Description
CA5PDF-U1	CATIA V5 Publish 3D On Demand Interactive & Batch processing
CA5PDF-I1	CATIA V5 Publish 3D Interactive

Supported Application Revisions

The following application revisions have been qualified with this release

Application	Revision
CATIA V5	CATIA V5 R19, CATIA V5 R20, CATIA V5 R21, CATIA V5-6R2012, CATIA V5-6R2013, CATIA V5-6R2014 & CATIA V5-6R2015
Adobe Reader	Adobe Reader XI

Supporting API's

The following supporting API support is used

Application	Revision
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 R19, CATIA V5 R20, CATIA V5 R21 and CATIA V5-6R2012 Qualified and fully supported for CATIA V5-6R2013 & CATIA V5-6R2014 Only
Windows Server 2008 R2	Not formally qualified for CATIA V5 R19 and CATIA V5 R20 Qualified and fully supported for CATIA V5 R21, CATIA V5-6R2012, CATIA V5-6R2013, CATIA V5-6R2014 & CATIA V5-6R2015 Only
Windows Server 2012	Not formally qualified for CATIA V5 R19, CATIA V5 R20, CATIA V5 R21, CATIA V5-6R2012 & CATIA V5-6R2013 Qualified and fully supported for CATIA V5-6R2014 & CATIA V5-6R2015 Only



Distribution Media

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

CATIA V5 Version	URL for CD image download
On Demand support for ALL Versions	cdCAD 19.0 CA5PDF WIN64.01.zip
CATIA V5R19 Interactive	cdCAD 19.0 CA5PDF WIN64 R19SAVEAS.01.zip
CATIA V5R20 Interactive	cdCAD 19.0 CA5PDF WIN64 R20SAVEAS.01.zip
CATIA V5R21 Interactive	cdCAD 19.0 CA5PDF WIN64 R21SAVEAS.01.zip
CATIA V5R22 Interactive	cdCAD 19.0 CA5PDF WIN64 R22SAVEAS.01.zip
CATIA V5R23 Interactive	cdCAD 19.0 CA5PDF WIN64 R23SAVEAS.01.zip
CATIA V5R24 Interactive	cdCAD 19.0 CA5PDF WIN64 R24SAVEAS.01.zip
CATIA V5R25 Interactive	cdCAD 19.0 CA5PDF WIN64 R25SAVEAS.01.zip
Unified Interface	cdCAD 19.0 UI WIN64.01.zip



New Features / Enhancements

The following new features or enhancements have been introduced with this release

Ref ID	New Feature / Enhancement Description
1	<i>Updated API Support:</i> Hoops v8.1 libraries have been adopted
2	<i>PDF File Size:</i> Files containing PMI are now smaller in size
3	<i>PDF Table Enhancements:</i> BOM and Instance tables now support top level Nodes (REQ00132) Text Size, RGB and justification controls have been added
4	<i>PMI Filled Fonts:</i> Filled fonts are now supported (REQ00140)
5	<i>Auto File Suffix:</i> PDF files are now auto-named with the .pdf suffix
6	<i>Semantic PMI:</i> Semantic PMI attributes are added into the PDF file definition
7	<i>Level of Detail:</i> level of detail now defaults to Medium which produces smaller files with no degradation in quality. 'ultra_low' also added as an option.
8	<i>Hidden Wireframe:</i> Hidden wireframe is now not processed by default unless using output_mbd
9	<i>Publishing Template Updates:</i> Templates now support: <ul style="list-style-type: none"> • Centre of Rotation button option • Enhanced user interaction via flexible javaScript utilities • Scroll / 3D / Assembly /Property Panel interaction • Scroll table scroll bar • Enhanced table row filtering • Digital Signature
10	<i>PDF Document Properties:</i> Title, Author and Subject can now be specified (REQ00147)
11	<i>View Optimization:</i> View zoom factors are optimized and can be controlled by user selection. In addition improvements have been made to the clipping of models within views, ensuring the near and far clipping planes allow for the model size.
12	<i>Mirrored Components:</i> Mirrored components with no scaling are now supported
13	<i>Small PMI:</i> Very small PMI objects (<3mm) previously appeared very course. Now resolved.



14	<i>PMI Edge Associations:</i> PMI edge highlighting is now supported. This can be disabled via the ‘no_pmi_edge_associations’ option
15	<i>Configuration Options:</i> The precedence of command line arguments and configuration file arguments can now be set
16	<i>View Grouping:</i> By default the geometry for each view is grouped into a single node. A secondary method has been made available (‘view_geom_exploded’) that creates a node for each referenced geometric item. In some cases this can produce a faster translation and smaller files size, dependent upon how many views there are and how much geometry is shared between views
17	<i>Robustness Improvements:</i> General robustness improvements have been made
18	<p><i>Section Properties:</i> Control of the following section properties have been added (REQ00153):</p> <ul style="list-style-type: none"> • Cutting Plane Visibility • Capping Visibility • Cutting Plane Intersections Visibility • Cutting Plane Opacity • Cutting Plane Colour • Cutting Plane Intersections Colour
19	<i>Instance Table Sequence Numbers:</i> Added support for Instance Table sequence numbers (REQ00131)
20	<i>Standard View Names:</i> Standard View prefixes are now controlled via the manifest file (REQ00128)
21	<i>Configurable Selection Box Part Names:</i> Users can now elect which property is to be used to display as a Part Name in the selection box (REQ00120)
22	<i>DLName Processing:</i> The CATIA V5 Interactive Save As to 3DPDF now operates correctly if DLNAME is active
23	<i>Parameter Attributes:</i> Support for parameter attributes added
24	<i>Model Based Definition Support:</i> The output_mbd option allows below part specification tree information to be read and processed as structural information in PDF.
25	<i>PMI Control:</i> The user can now control whether none, all Part only or assembly PMI is read



Known Limitations and Restrictions

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

Ref ID	Limitations and Restrictions
1	<i>Help Shortcut:</i> The Help short-cut point to <install_dir\help.html. This redirects to the latest help on the web site at http://www.theorem.com/Documentation There is, however, no short-cut to the help shipped on the CD, which can be accessed at: <install_dir\Guides\V5_3DPDF_index.htm
2	<i>PMI Flipping:</i> PDF does not support flipped PMI
3	<i>Unified Interface Export:</i> When exporting from the Unified Interface, the output name uses the top level assembly name by default, regardless of selection. This can be overridden via the <i>Translator Options->Confirmation</i> panel
4	<i>Unified Interface Viewer:</i> The Unified Interface Viewer does not display PMI at the assembly level
5	<i>Unified Interface Viewer:</i> There is no option to export the entire assembly from the Viewer, when nothing is selected
6	<i>R21 Interactive Save As:</i> When pdf is selected as the type to save in the Save As menu, the file extension is not automatically changed to pdf (stays as .CATPart/Product).



Resolved Support Calls

The following list of Support Calls are resolved with this release

Support Call ID	Brief Description
CAS-02338	<i>NOA Processing:</i> Part specific issues with NOA processing resolved
CAS-02414	<i>Sketch Curves Omitted from View:</i> Resolved

