



Product Release Announcement

Product Category	CADPublish
Product Group	CATIA V5 > 3D PDF
Product Release Version	27.0

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History

Revision	Update Information
1.0	Initial document release for Version 27.0

Legacy Product Codes

Product Code	Product Description
CA5PDF-U1	CATIA V5 Publish 3D On Demand Interactive & Batch processing
CA5PDF-I1	CATIA V5 Publish 3D Interactive
CA5PDF-A1	Publish 3D Configuration Tool for CATIA users
CA5PDF-A2	Publish 3D Java Script Add-on Module for CATIA V5
CA5PDF-S1	Publish 3D consultancy services add-on module to support export of CATIA V5 Motion data to PDF

Current Product Codes

Product Code	Product Description
CA5PDF-U1	CADPublish Enhance for CATIA V5
CA5PDF-I2	CADPublish Create for CATIA V5
CA5PDF-S1	Publish 3D consultancy services add-on module to support export of CATIA V5 Motion data to PDF

Supported Application Revisions

The following application revisions have been qualified with this release

Application	Revision
CATIA V5	CATIA V5-6R2020, CATIA V5-6R2021, CATIA V5-6R2022, CATIA V5-6R2023, CATIA V5-6R2024
Adobe Reader	Adobe Reader 7.0 and above

Supported Operating Systems

The following operating systems have been qualified with this release

Operating System	Qualification Status
Windows 10 64bit	Qualified and fully supported

Installation Media

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

Product	URL for CD image download
All supported versions of CATIA V5	CAD_27.0_CA5PDF_WIN.01
Theorem Unified Interface	CAD_27.0_UI_WIN.01

Licensing System (FlexLM)

The licensing server has been upgraded to version 11.19. It is recommended that you upgrade your server to this version

Product	URL for CD Image download
Flex License Server	CAD_FLEXLM_11.19_WIN.01

New Features / Enhancements

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

Ref ID	New Feature / Enhancement Description
1	Support for CATIA V5-6R2024
2	add_design_meta_data <file> argument created which enables external attributes in a designated file to be displayed in the PDF.
3	inst_pmi argument created which allows PMI on multiple Instances to be displayed
4	design_naming <format> argument created and in conjunction with design_name will enable the root node of the object in the PDF to be named as specified by the user. For example design_naming #PN# will name the root node of the object in the PDF the part name specified in CATIA V5.

Known Limitations and Restrictions

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

Ref ID	Limitations and Restrictions
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Resolved Support Calls


The following list of Support Calls are resolved with this release

Support Call ID	Brief Description
TH-386	Weld feature annotation now displayed correctly in PDF, weld feature annotation now aligns with how it is displayed in CATIA V5.
TH-551	Animations now working on selected assemblies when using the Motion template, read_motion_rel option required in order for animations to be read.
TH-585	Axis systems and point sketches are now named in the tree structure in the PDF as per the naming convention in CATIA V5 when using the options output_mbd_leaf_nodes and output_mbd_axis_groups. Tree structure in the PDF also aligns with CATIA V5.
TH-709	PMI on both part instances now displayed in the views in the PDF with the option inst_pmi applied.




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