PRODUCT RELEASE ANNOUNCEMENT

Product Category	Visualize 3D
Product Group	CATIA V5 for JT
Release Version	20.2

Document Type	Product Release Announcement
Status	Released
Revision	1.0
Author	Product Manager
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Contents

History

Revision	Update Information
1.0	Initial Release

Product Codes

Product Code	Product Description
CA5JT-U1	CATIA V5 (CAA Based) to JT (uni-directional)
JTCA5-U1	JT to CATIA V5 (CAA Based) (uni-directional)
CA5JT-A5	CATIA V5 (CAA Based) to JT (uni-directional) with PMI Add On
CA5JT-A6	CATIA V5 (CAA Based) to JT (uni-directional) with Electrical Add On
CA5JT-A7	CATIA V5 (CAA Based) to JT (uni-directional) with 3D PDF Add On
JTCA5-A1	JT to CATIA V5 (CAA Based) (uni-directional) with 3DPDF Add On
JTCA5-A2	JT to CATIA V5 (CAA Based) (uni-directional) with 3DXML Export Module
CATJT-U1	CATIA V4 to JT (uni-directional)
CAGJT-U1	Combined CATIA V4 & CATIA V5 to JT (uni-directional)
CAGJT-U2	Combined CATIA V4 & CATIA V5 to JT (uni-directional) with PMI Add On

Supported Application Revisions

The following application revisions have been qualified with this release

Application	Revision
CATIA V4	All revisions
CATIA V5	CATIA V5 R19, R21, CATIA V5-6R2012, CATIA V5-6R2013, CATIA V5-6R2014, CATIA V5-6R2015, CATIA V5-6R2016 & CATIA V5-6R2017
JTOpen	JTK 9.0.0.0
Adobe Reader	Adobe Reader XI

Supporting API's

The following supporting API support is used

Application	Revision
3D PDF API	HOOPS Exchange Publish 8.0

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 Revisions)
Window 8 64bit	Qualified and fully supported for CATIA V4, CATIA V5-6R2013, CATIA V5-6R2014 , CATIA V5-6R2015, CATIA V5-6R2016 & CATIA V5-6R2017
Windows Server 2008 R2	Qualified and fully supported for CATIA V4, CATIA V5 R21, CATIA V5-6R2012, CATIA V5-6R2013, CATIA V5-6R2014, CATIA V5-6R2015, CATIA V5-6R2016 & CATIA V5-6R2017
Windows Server 2012	Qualified and fully supported for CATIA V4, CATIA V5-6R2014 , CATIA V5-6R2015, CATIA V5-6R2016 & CATIA V5-6R2017
Window 10 64bit	Qualified and fully supported for CATIA V4, V5-6R2016 & V5-62017.

Installation Process Considerations

Starting with version 20.0 of Theorem Solutions applications changes have been made to the installation process. These changes have transitioned from a JAVA based installer package to a standard Microsoft Installer (.msi) package. A current limitation of the Microsoft msi method is that the software cannot be installed directly on to either externally mapped or network drives when installed from a client machine:

Therefore, the installation must be performed to a drive that is local to the machine itself.

If you are attempting to install the software onto a network drive, then please refer to the configuration guide available from the following link http://www.theorem.com/documentation/Documentation/Client_Configuration.pdf.

Distribution Media

CD images of the latest release are available from the following download site. The installation media for the bi-directional product is also the same media when installing a uni-directional variant.

Product Codes	Select link to download CD images
All Product Codes for all supported version of CATIA V5	https://s3-eu-west-1.amazonaws.com/theorem.software/CADverter_20.2/CAD_20.2_CA5JT_WIN.01.msi

Related Distribution Media

In addition to the translator software itself there are also two other related media distributions available. One is for the installation and configuration of the Theorem User Interface. This is designed as a free-standing user interface for the translator allowing the user to navigate the file system and initiate translations either interactively or in batch using standard drag and drop techniques. The second media distribution contains the Theorem License Manager which is installed on the nominated license server(s) and manages the user access to the purchased volume of concurrent Theorem application licenses.

Related Package	URL for CD image download
Theorem User Interface	https://s3-eu-west-1.amazonaws.com/theorem.software/CADverter_20.2/CAD_20.2_UI_WIN.03.msi
Theorem License Manager	https://s3-eu-west-1.amazonaws.com/theorem.software/LicenseManager/CAD_FLEXLM_11.14_WIN.01.msi

New Features / Enhancements

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

CATIA V5 to JT (CA5JT)

Ref ID	New Feature / Enhancement Description
1	Support for CATIA V5-6R2017 has been implemented with this release
2	Added support for writing JT MONIKER attributes to JT output
3	1. Redesign of config files to include environment variables. 2. The old "cmd" versions are now the main versions that get used by default and the original versions now have orig in their names. 3. The main versions now contain comment sections with all available settings shown.
	Added AdvancedOptions = "" default setting to the *cmd config files for ease of modification.
4	Improvements to Large Assembly Processing(LAP) for V5 to JT. LAP now allows the output structure type to be PER_PART, PLMXML or STEPBOM, in addition to the existing type (FULL SHATTER (with no intermediate assembly files).
5	1. Revamp of config option processing to prevent default options being added to the command line args. 2. Remove support of config option ProcessHiddenGeom for V5 3. Add capability to auto-generate default config files containing a list of config options with allowable values
6	In support of the JT "Moniker" attributes (MONIKER/BODY_ID_ATTRIB, MONIKER/GUID_TABLE_ATTRIB and MONIKER/MONIKER_DATA_ATTRIB)added via Parasolid, diagnostic code has been added to access and print them.
7	Added support for environment variables in config files. 2. Correct the JT file version number in debug messages. 3. Always output the config file name in progress files
8	Added a new command line arg "print_args" that prints the original and post config file command line args
9	Correct error that suppressed entity list for xml progress files
10	Allow for non-unique detail names when creating instances.
11	Fix for PMI edge association in JT Brep data. Previously when the modelviews option was used, incorrect IDs were being added to edges which caused any PMI association to fail.
12	1. Allow for spaces in config file supplied advanced options by supporting \"A B\" 2. Support instance_naming in config files
13	The CATIA V5 to JT export process now supports the creation of the assembly structure as using a PLMXML format file for the assembly with

	subordinate JT files for the components. This is selected using the JT configuration file option

JT to CATIA V5 (JTCA5)

Ref ID	New Feature / Enhancement Description
1	Support for CATIA V5-6R2017 has been implemented with this release
2	Implementation of the "plmxml_topnode" option in JT->VWR. This option reads the name of the top level node in a PLMXML file and outputs it to the specified output file. It is intended to support translations of files exported from Teamcenter where the file names are meaningless
3	Allow for the combination of both child instances and a representation at the top level in PLMXML data.
4	Correction to the accessing of PMI Model View properties. The wrong class was being used which could lead to a crash or null strings.
5	Prevent a crash when creating occurrences from PLMXML data.
6	Account for JT files with different units when reading a PLMXML assembly.
7	Changes to PMI glyph and PMI Polygon reading. 1. Fix the positioning of 3D glyphs. 2. Store the graphics type for glyphs as "rendered text", and the graphics type for polygons as "symbol".
8	The Theorem User Interface is supported for both interactive and batch processing of translations external to the CATIA V5 application

Known Limitations and Restrictions

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

CATIA V5 to JT (CA5JT)

Ref ID	Limitations and Restrictions
1	The Theorem FLEXlm license manager for this release should be updated to the latest release available. See section on “Related Distribution Media”

JT to CATIA V5 (JTCA5)

Ref ID	Limitations and Restrictions
1	The Theorem FLEXlm license manager for this release should be updated to the latest release available. See section on “Related Distribution Media”

Resolved Support Calls

The following list of Support Calls are resolved with this release

Support Call ID	Brief Description
CAS-01828	Change the defaults for model view geometry blanking, extended PMI and Parasolid entity ids to "ON".Check the "pointstate" attribute for "Capture" views and if set to 0, reset the view extents based on the PMI or geometry (if no PMI).
CAS-02503	V5 to JT text orientation appears to be flipped
CAS-02992	The creation of PMI user attributes has been changed to support Unicode strings as input. This allows special characters such as degree symbol to be supported. If a semantic PMI definition is available, this is now used.
CAS-03047	1. Major changes to the method of selective display of model geometry in Views. For assemblies, the visible leaf node part instances are associated to the view. Previously a design group was used to display all the visible faces.2. The POINTSTATE (V5) attribute is now ignored for views and the supplied extents are used instead of the extents being recalculated.3. The now obsolete (PMI*.cpp) have been removed.4. Pre-version JT 7.1 code has been removed from the PutPMI*.cpp functions.
CAS-03115	Processing V4 model files with inch units results in mm JT output
CAS-03125	Fix crash encountered whilst setting overriding materials after _geom nodes have been added into the JT structure Add support of PMI glyphs
CAS-03140	1. PMI User Attributes: process Unicode characters 0x2264 (<=) and 0x2265 (>=) as two separate characters because TcVis does not display them correctly.2. Extended Datum Target : Prevent crash if the string searched for the label and index is zero length.
CAS-03149	Reinstatement of Weld Symbols that were removed when JTTK 7.1 was introduced. The new code uses the new JTTK PMI classes.
CAS-03151	Remove override colour processing from the assembly read phase and add as a post-processing call. This improves efficiency as the override colours are only processed once instead of being processed for each node.
CAS-03191	Crash occurring when export PDF output inconjunction with JT
CAS-03204	Check that face 2D parametric edge curves are set to prevent a crash. If they are not set, make 3 attempts at increasing tolerances to regenerate the curves. If this fails, the face is rejected.
CAS-03207	V5>JT: License blocked when converting some parts
CAS-03210	V5>JT: License blocked when converting some parts
CAS-03211	V5>JT: License blocked when converting some parts
CAS-03216	Make sure that Model View names are unique for cases where the "modelviews" option is not being used.
CAS-03258	V5>JT: problem exporting to JT

CAS-03276	Fixes for "weld" parts.1. Fix to prevent the "number of shells" for a JT Brep from being used as a GCO id.2. Fix to obtain the weld parameters from the parent node rather than the root node for cases where weld parts are used in assemblies.
CAS-03287	Data related translation crash - V5 R27 -> JT
CAS-03305	V5>JT, V5>PDF: error during conversion
CAS-03310	CATIA V5 to JT creates incorrect output for assembly structure and missing geometry data
CAS-03326	V5>JT: Scaling / Positioning problem in TcVis with JTs from JT plugin