

Release Note: CADdoctor EX4.3

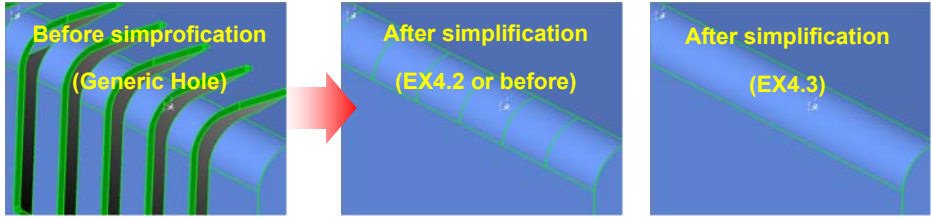
■CADdoctor EX4.3

CADdoctor EX4.3 is a minor upgrade for CADdoctor EX4.2.

The following are the highlights of new features, including enhancements and bug fixes.

■Highlights of New Features

No.	Features	Description																																																												
1.	OS version upgrade	<p>[Windows7- compliant] The modules support Windows7 as follows. (X: Available, -: Not available)</p> <table border="1"> <thead> <tr> <th>Product</th> <th>Support Version</th> <th>Windows7</th> <th>Note</th> </tr> </thead> <tbody> <tr> <td colspan="4">CADdoctor Main body</td> </tr> <tr> <td rowspan="7">Stand-alone</td> <td>CATIA V4</td> <td>V4.2.1 – V4.2.5</td> <td>X</td> </tr> <tr> <td>Parasolid</td> <td>V7 – V22</td> <td>-</td> </tr> <tr> <td>IGES</td> <td>- V5.3</td> <td>X</td> </tr> <tr> <td>ACIS</td> <td>R6 – R20</td> <td>-</td> </tr> <tr> <td>STEP</td> <td>AP203, AP214</td> <td>-</td> </tr> <tr> <td>CATIA V5</td> <td>R6 – R19</td> <td>X</td> </tr> <tr> <td>JT</td> <td>In: 6.4 – 9.4 Out: 8.0 – 9.4</td> <td>-</td> </tr> <tr> <td rowspan="4">Plug-in</td> <td>CATIA V5</td> <td>R18, R19, R20</td> <td>-</td> </tr> <tr> <td>Pro/E</td> <td>Wildfire3, Wildfire4</td> <td>X</td> <td>Wildfire4 M120 or later is required. Wildfire3 isn't supported.</td> </tr> <tr> <td>NX</td> <td>NX5, NX6, NX7</td> <td>-</td> </tr> <tr> <td>I-DEAS</td> <td>I-deas12, NX I-deas5, 6, 6.1</td> <td>-</td> </tr> <tr> <td rowspan="5">Stand-alone (Polygon)</td> <td>STL</td> <td></td> <td>X</td> </tr> <tr> <td>VRML</td> <td></td> <td>X</td> </tr> <tr> <td>OBJ</td> <td></td> <td>X</td> </tr> <tr> <td>Nastran bulk data</td> <td></td> <td>X</td> </tr> <tr> <td>Point Cloud</td> <td></td> <td>X</td> </tr> </tbody> </table> <p>*Compatible with Windows7 Professional</p>	Product	Support Version	Windows7	Note	CADdoctor Main body				Stand-alone	CATIA V4	V4.2.1 – V4.2.5	X	Parasolid	V7 – V22	-	IGES	- V5.3	X	ACIS	R6 – R20	-	STEP	AP203, AP214	-	CATIA V5	R6 – R19	X	JT	In: 6.4 – 9.4 Out: 8.0 – 9.4	-	Plug-in	CATIA V5	R18, R19, R20	-	Pro/E	Wildfire3, Wildfire4	X	Wildfire4 M120 or later is required. Wildfire3 isn't supported.	NX	NX5, NX6, NX7	-	I-DEAS	I-deas12, NX I-deas5, 6, 6.1	-	Stand-alone (Polygon)	STL		X	VRML		X	OBJ		X	Nastran bulk data		X	Point Cloud		X
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2.	CAD version upgrade *Input and output Op	<ul style="list-style-type: none"> ◆ CATIA V5 R20(Plug-in) ◆ NX7(Plug-in) ◆ NX I-deas6.1(Plug-in) ◆ JT V9.4 																																																												

3.	<p>Geometry Simplification *Geometry Simplification Op)</p>	<p>[Automatic Face Merging after the Removal of Geometry Features] Multiply-divided faces created after the automatic removal of round holes, generic holes, bosses/ribs and steps are now automatically merged. Related check item: [Round Hole] [Generic Hole] [Bosses/Rib] [Step]</p>  <p>[Improvement of the removal of Fillet] Improvement of the removal of Fillet to resolve the issue that several fillets have been failing in the simplification. *Related check item: [Fillet]</p>
4.	<p>Other Enhancements</p>	<p>[Other Enhancements]</p> <ul style="list-style-type: none"> ◆ In the Auto-Heal Pre/Post Command on the Reverse Engineering mode/Polygon PDQ mode in the Wizard menu, new menu items are added: Wrapping Polygon, Polygon Offset and Filling all holes. *Related menu: [File]-[Wizard] ◆ Plane surfaces on CATIA V5(Plug-in) are now convertible to DatumPlanes on NX(Plug-in).
5.	<p>Specification Changes</p>	<p>[Specification Changes]</p> <ul style="list-style-type: none"> ◆ On exporting a file with assemblies on CATIA V5 Stand-alone, CATProduct are created and allocated for every each assembly; on CADdoctor EX4.2 or before, only a CATProduct file has been exported regardless of the type of assembly structure an original file has. ◆ The menu name has been changed from “Convert hidden instance” to “Convert elements under hidden instance” in the Output option of CATIA V5 Plug-in. ◆ The default export target system displayed after the startup of CADdoctor has been changed from CATIA_V4 to CATIA_V5.
6.	<p>Bug Fixes</p>	<p>[Major Bugs Fixed]</p> <ul style="list-style-type: none"> ◆ On geometry verification, the issue that nothing has been displayed in the window, when selecting a model with nothing to verify and clicking the Zoom Current icon and then the Toggle Display Mode icon, is fixed. *Related menu: [Geometry Verification] ◆ RGB color shift observed on the data translation from CATIA V5(Plug-in) to Pro/E(Plug-in) through CADdoctor is fixed. ◆ RGB color shift observed on the data translation from Pro/E(Plug-in) to CATIA V5(Plug-in) through CADdoctor is fixed. ◆ The error that has occurred at the startup of CATIA V5 Plug-in is fixed. This error has arisen under the circumstance where CATIA V5 had been installed and the location of CATEnv folder had been changed from the default setting by your option. ◆ On geometry simplification, the issue that the Remove All the Steps function has not worked is fixed. The issue that unmodified edges have remained after manually modifying a step so as to fit to the lower step is also fixed. *Related check item: [Step]

		<ul style="list-style-type: none"> ◆ On geometry simplification, the issue that steps has been unable to be removed on a specific geometry on the option of the Prohibit Invalid Geometry Creation is resolved. *Related check item: [Step] ◆ The issue that a specific model has been failing in conversion to a Elysium neutral file using CATIA V5 Plug-in is resolved. ◆ The issue that Fitting Surface has resulted in an error on a specific model is resolved. *Related menu: [CAD-Polygon Fit]-[Fitting Surface] ◆ The issue that a specific model of VRML has been failing in import is resolved. ◆ The issue that a specific model of STEP has been failing in import is resolved. ◆ The issue that JT has been unable to be exported on a specific model is resolved. ◆ The issue that a specific model has been unable to turn solid when activating the solid option on the Polygon Offset is resolved. *Related menu: [Polygon PDQ]-[Offset] ◆ The issue that STL has been unable to be exported on the Wizard is resolved. *Related menu: [File]-[Wizard] ◆ The issue that some free edges have remained after the Polygon Stitching is resolved. * Related menu: [Polygon PDQ]-[Stitch] ◆ The issue that the Segmentation function has not worked on a specific model is resolved. * Related menu: [Reverse Engineering]-[Segmentation]
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■Supported CAD version

No.	Product	Support Version	Note
1.	CATIA V4	V4.2.1-V4.2.5	
2.	CATIA V5(plug-in)	R18,R19,R20	
3.	CATIA V5(InterOp)	R6-R19	
4.	Pro/E(plug-in)	Wildfire3,4	
5.	I-DEAS(plug-in)	32bit: I-deas12, NX I-deas5, 6, 6.1	32bit only
6.	NX(plug-in)	NX5,NX6,NX7	
7.	CADCEUS	32bit : 5R1-4 or later	Via ENF (*.enf), 32bit only
8.	Parasolid	V7-V22	
9.	ACIS	R6-R20	
10.	JT	Import: 6.4-9.4 Export: 8.0-9.4	
11	STEP	AP203, AP214	
12.	IGES	Import: -V5.3 Export: V5.2, V5.3	

■Recommended System Environment

[32bit]

Architecture	Microsoft Windows 2000 / Windows XP Professional / Windows Vista / Windows7 Professional (*1) running on IBM PC/AT. (*2)
CPU	Microsoft Windows 2000 / Windows XP Professional / Windows Vista / Windows7 Professional (*1)
Memory	1GB minimum
OS	Windows 2000 / Windows XP Professional / Windows Vista / Windows7 Professional (*1)
HDD	Hard Disk(200MB minimum free area, +data area, +swap area, additional 500MB required for CATIAV5 Stand - Alone user), CD-ROM Drive
Display	1024 x 768 resolution, 65536 colors or higher Graphics Board : Open GL Board that supports double buffering.(*3)

[64bit]

Architecture	Windows XP Professional x64bit Edition,/ Windows Vista x64bit Edition / Windows7 Professional x64bit Edition (*1) running on IBM PC/AT. (*2)
CPU	CPU supports Windows XP Professional x64bit Edition / Windows Vista x64bit Edition / Windows7 Professional x64bit Edition (*1)
Memory	1GB min.
OS	Windows XP Professional x64bit Edition / Windows Vista x64bit Edition / Windows7 Professional x64bit Edition (*1)
HDD	Hard Disk(500MB minimum free area, +data area, +swap area, additional 1GB required for CATIAV5 Stand - Alone user), CD-ROM Drive
Display	1024-768 resolution, 65536 colors min. Graphics Board : Open GL Board that supports double buffering.(*3)

(*1) Microsoft is a registered trademark of Microsoft Corporation.

(*2) IBM is a registered trademark of International Business Machines Corporation.

(*3) OpenGL is a registered trademark of Silicon Graphics, Inc.

■Remarks

The file import of CATIA V5 Stand-alone

The following two problems about the file import of CATIA V5 Stand-alone are known issues. We are investigating about the issues to resolve them in the next version or later.

1: During the import of a model, the dialog that demands for CATIA V5 license may appear. This dialog doesn't affect the result of the translation result and it goes away after the translation. (The dialog may have a slightly difference depending on the environment.)

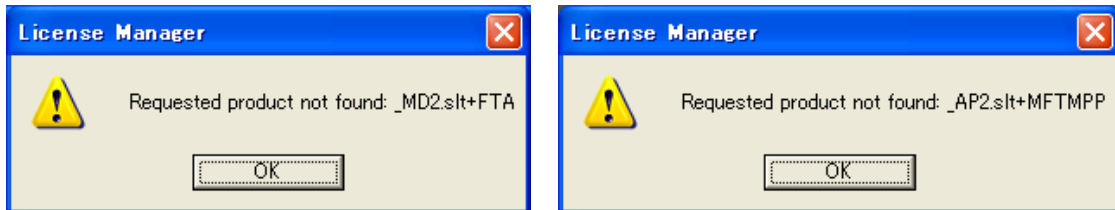


Fig. The sample of dialogs.

2: After importing CATProduct included some shared assemblies by using CATIA V5 Stand-alone, the shared assemblies will not be shared and a number will be added after to the name of the divided file.