

ASFALIS EX8.3



**ASFALIS EX8.3**

Begleitheft zur Version

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Wir freuen uns, Ihnen die neue Version EX 8.3 von ASFALIS zur Verfügung stellen zu können. Bei der Installation stehen wir Ihnen im Rahmen Ihres Wartungsvertrages gerne zur Seite.

⇒ Wir erwarten Mitte 2021 das nächste Hauptrelease EX9 von ASFALIS mit umfangreichen Änderungen. Das hier vorliegende EX8.3 ist das voraussichtlich letzte Release der bewährten EX8.x Serie mit hoher Stabilität und allen eingepflegten Requests, Bugfixes sowie aktuellsten Schnittstellen.

**Wir empfehlen allen Anwendern das Update auf diese Version zu erwägen.**

**Achtung, für Releases ab EX8.x muss der Lizenzdienst auf Sentinel RMS Version 9.2.1 geupdated werden, falls nicht bereits mit einer Vorgängerversion geschehen.**

Unter [LicenseServer] im Installationsmedium finden Sie den Ordner [1\_Install]. Verwenden Sie Sentinel RMS älter als 9.2.1, so aktualisieren Sie den Lizenzserver (-dienst) mittels setup.exe. Es ist nicht notwendig, die Lizenz erneut einzuspielen.

Wir bitten die Hinweise in diesem Dokument zu beachten, um die Umstellung reibungslos zu gestalten.

#### **Schnelleinstieg für erfahrene Benutzer:**

- Aktualisieren Sie den Lizenzserver (-dienst) auf Sentinel RMS Version 9.2.1. Installer im Ordner */LicenseServer/1\_Install*
- Nutzen Sie ASFALIS mit ELYBATCH, so könnte die Konfiguration eines bestehenden ASFALIS EX8.x weiterhin verwendet werden. Um den Ist-Stand zu behalten, müssen einige mitgelieferte C++ Runtimes installiert werden (vcredist), ansonsten reicht (für ein mit elybatch.exe betriebenes ASFALIS) das gewohnte Überkopieren des EX 8.3 */bin* Ordners aus. Wir empfehlen die Installation aller vcredist\* aus dem Ordner */Support/Runtime* des Installationsmediums.
- Nutzen Sie ASFALIS mit Slavenodes oder TransServer, so sollte Kontakt mit unserem Support aufgenommen werden, um die Alteinstellungen erhalten zu können.

# 1. Neu in diesem Release

## 1.1 CAD Support

Inklusive der neuen Formate unterstützt ASFALIS nun:

- CATIA V5 R19 – R30
  - CATIA V4.2.1 - V4.2.5
  - 3DEXPERIENCE(Plug-in) R2017x, R2018x, R2019x, R2020x
  - NX I-DEAS 6.3 , 6.4 , 6.5 , 6.6 , 6.7
  - Creo Parametric 2.0 - 7.0
  - Parasolid V7 – V32.0
  - NX9 - NX1926 Series
  - SolidWorks 2015 - 2020
  - Inventor 2017 - 2021
  - Creo Elements/Direct Modeling 18.1 - 20.2
  - IGES 5.2, 5.3
  - STEP AP203, 214, 242 ed1/ed2
  - STEP AP242 BOM
  - ACIS R6-R29
  - JT 6.4 – 10.5 (Import), JT 8.0 – 10.5 (Export)
  - PLM XML
  - 3DXML(SA) R201x – R2019x
- 
- CATIA V5 standalone R7 – R30 (Import), R14, R19, R20 (Export)
  - Creo Parametric standalone 2000i - WF5, CP1.0 – CP6.0 (Import)
  - NX standalone UG10 - NX1899 (Import)
  - STL (Export)
  - XVL v3 , v5 , v7 , v9 , v10 , v11 (Export)
  - 3D PDF

## 1.2 Bugfixes / Änderungen

Siehe Kapitel 3ff. Dieses Release beinhaltet keine neue Funktionalität, sondern beinhaltet Detailverbesserungen gegenüber EX8.2 (eingepflegte Requests und Bugfixes) sowie Schnittstellenupdates.

## 2. Installation

### 2.1 ASFALIS Updateinstallation

Der Wechsel auf die Komponenten von ASFALIS EX 8.3 geschieht beim Betrieb per XML-Szenarios und Aufruf durch `elybatch.exe` durch bloßes Überkopieren der neuen Komponenten in Ihr ASFALIS `/bin`-Verzeichnis.

Ihr derzeitiger Ist-Stand bleibt erhalten und funktioniert auch mit EX 8.3.  
Eine existierende EX8x Lizenz kann auch EX8.3 betreiben.

Wir empfehlen die Installation aller `vcredist*` aus dem Ordner `/Support/Runtime` des Installationsmediums.

Wir stehen Ihnen hier im Rahmen Ihres Wartungsvertrages mit Rat und Tat zur Seite.



Bitte kopieren Sie die neuen EX8.3 Module in Ihr existierendes ASFALIS `/bin`-Verzeichnis. Installieren Sie alle `vcredist*` aus dem Ordner `/Support/Runtime`

Der Ordner `/Doc` des Installationsmediums enthält die Datei `Requirement_ENG.pdf`.  
Bitte beachten Sie die dortigen Informationen.

### 2.2 ASFALIS Neuinstallation

ASFALIS-Installationen sind meist nutzerspezifisch und schwer zu verallgemeinern.  
Neuinstallationen werden i.d.R. durch unseren Support durchgeführt, deshalb erfolgt hier keine gesonderte Beschreibung.

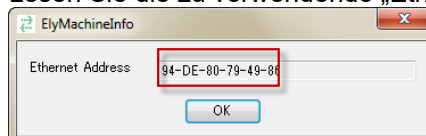
### 2.3 Lizenzserver

#### 2.3.1 Lizenzanforderung

Zum Betrieb von EX 8.3 ist eine Lizenz erforderlich.  
(Diese muss, wie nachfolgend beschrieben, bei CAMTEX angefordert werden.)

Es wird eine bestimmte MAC-Adresse Ihrer Netzwerkkarte verwendet. Ermitteln Sie bitte die Elysium HostID des Lizenzservers ausschließlich wie folgt:

- a. Unter [LicenseServer] im Installationsmedium finden Sie den Ordner [2\_HostID].  
Starten Sie die Datei **elymacinfo.exe** (Es erfolgt keine Installation)
- b. Lesen Sie die zu verwendende „Ethernet Address“ ab.



- c. Senden Sie uns Ihre Lizenzanforderung zusammen mit dieser „Ethernet Address“.

### 2.3.2 Lizenzverwaltung / Lizenzeingabe

Nach Erhalt Ihrer Lizenzanforderung wird Ihre Lizenz ausgestellt und Ihnen per email übersandt. Lizenzdateien haben die Endung **\*.elylic**.

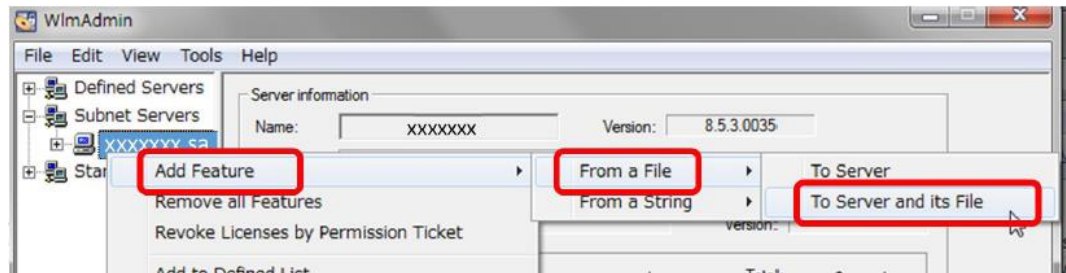
Zur Verwaltung des installierten Lizenzservers benötigen Sie die Daten aus dem Ordner [3\_Administration] Ihres Installationsmediums.

Am besten kopieren Sie sich den kompletten Ordner nach z.B.  
**c:\elysium\licenseserver\3\_Administration**

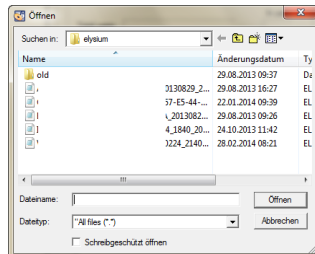
Starten Sie die Datei **WlmAdmin.exe** aus dem Ordner [3\_Administration].  
Sie brauchen hierzu Administratorrechte.

Klicken Sie auf der linken Seite den Punkt „Subnet Servers“ auf. Der Name Ihres Lizenzservers sollte zu sehen sein.

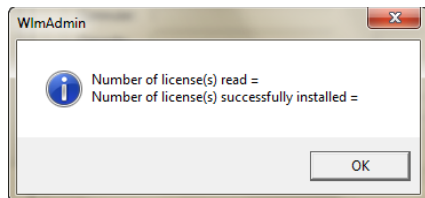
Markieren Sie Ihren Lizenzserver und klicken Sie mit der rechten Maustaste darauf.



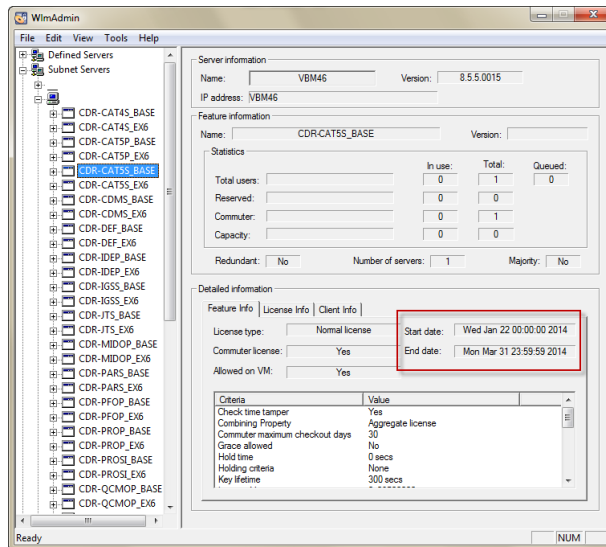
Wählen Sie [Add Feature] – [From a File] – [To Server and its File]



Verweisen Sie im [Öffnen] Dialog zur erhaltenen Lizenzdatei **\*.elylic**.  
-> Die Datei \*.elylic kann an einem beliebigen Ort liegen. Sie wird nur ausgelesen.



Ihre Lizenz wurde erfolgreich importiert.



Sie können unterhalb Ihres Rechnernamens nun die erteilten Lizenzen einsehen und z.B. das Ablaufdatum auslesen.

Die Lizenzierung ist hiermit abgeschlossen.



#### **Tipp für Admins:**

Die Lizenzverwaltung mit *[wlmadmin.exe]* nach 2.3.2 kann auch von einem entfernten Rechner über das Netzwerk ausgeführt werden.

### 3. Produkterweiterungen / Änderungen der Spezifikation (engl.)

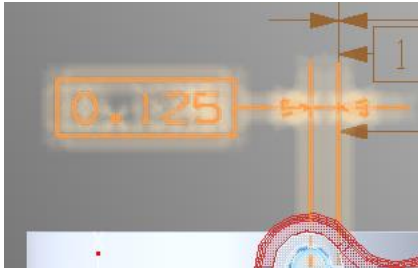
<p>(1) Common for all Components</p> <ul style="list-style-type: none"> <li>- Fixed an issue that an interruption of processes due to forced termination or timeout failed during repeating retrieval of acquiring Elysium license or checking free CAD license (FLEXlm). (00055235)</li> </ul>
<p>(2) CATIA V5(Plug-in) ENF Writer</p> <ul style="list-style-type: none"> <li>- Enabled to translate the user defined parameter right under a part. (00054198)</li> <li>- Fixed an issue that the text color of PMI could not be translated correctly. (00055329)</li> <li>- Fixed an issue that hidden elements could not be translated when the original model was created on R15 or earlier.</li> <li>- Fixed a model-dependent issue that the translated geometry was different from the geometry displayed on CATIA V5 right after opening the model.</li> </ul>
<p>(3) CATIA V5(Plug-in) ENF Reader</p> <ul style="list-style-type: none"> <li>- Fixed a model-dependent issue that related elements of length dimension were lost during PMI translation. (00055321)</li> </ul>
<p>(4) NX(Plug-in) ENF Writer</p> <ul style="list-style-type: none"> <li>- Enhanced to translate the standard (e.g., JIS, ISO, etc.) of surface finish as that of component when NX model includes surface finish. (00055322)</li> <li>- Fixed an issue that an invisible line was translated as graphic PMI when translating surface finish without leader line. (00055325)</li> </ul>
<p>(5) NX(Plug-in) ENF Reader</p> <ul style="list-style-type: none"> <li>- Fixed an issue that the name of edge was lost during seam line removal process. (00051882)</li> </ul>
<p>(6) Creo Parametric (Plug-in) ENF Writer</p> <ul style="list-style-type: none"> <li>- Added a new parameter "ConvertCombstate" to distinguish the translation of Combined View from PMI translation. (00054946) <ul style="list-style-type: none"> <li>- EX8.2 or earlier: It was impossible to translate Combined View only without translating PMI because Combined View was translated when PMI translation was activated by the parameter "XConvert***".</li> <li>- EX8.3: It is possible to specify PMI translation "XConvert***" and Combined View translation "ConvertCombstate" independently. Please set both parameters when translating both of PMI and Combined View.</li> </ul> </li> <li>- Fixed an issue that a part of related elements which are shared among multiple components were lost in PMI conversion. (00055530)</li> </ul>
<p>(7) Creo Parametric (Plug-in) ENF Reader</p> <ul style="list-style-type: none"> <li>- Fixed an issue that the value of Creo configuration "default_decimal_places" which specify the number of decimal places was changed to "12" after translation with ASFALIS SmartLauncher (Plug-in). (00054646)</li> </ul>
<p>(8) SOLIDWORKS ENF Writer</p> <ul style="list-style-type: none"> <li>- Fixed a model-dependent issue that it consumed a lot of resources when opening the model. (Fixed in SOLIDWORKS 2017 or later)</li> </ul>
<p>(9) Parasolid ENF Reader</p> <ul style="list-style-type: none"> <li>- Fixed an issue that the name of edge was lost during seam line removal process. (00051882)</li> </ul>
<p>(10) STEP ENF Writer</p> <ul style="list-style-type: none"> <li>- Enabled to translate name of isolated curves and points.</li> <li>- Enabled to translate transparency of volumes.</li> <li>- Enhanced to translate relations from PMI to other graphic PMI in STEP AP242 to ENF translation.</li> <li>- Enhanced Saved View translation in STEP AP242 to ENF translation. It automatically selects a suitable way to translate Saved View according to the CAD system written in the header section of the STEP file.</li> <li>- Extended the functionality of the parameter "PartIDMapping" to support the translation Part ID in STEP BOM to PartName in ENF in STEP AP242 BOM XML to ENF translation. Due to this enhancement, the default value of this parameter is changed as below. <ul style="list-style-type: none"> <li>- The default behavior of EX8.2 or earlier:</li> </ul> </li> </ul>



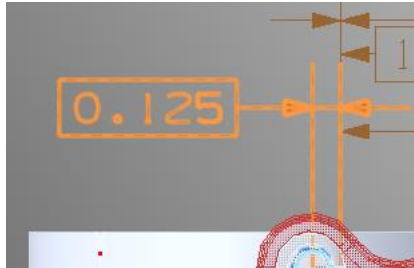
<p>The component name was translated to PartName in ENF.  Part ID(*) in STEP BOM was translated to PartNumber in ENF.  (*) Identifier.id was used as Part ID in preference to Id.id.</p> <ul style="list-style-type: none"> <li>- The default behavior of EX8.3:  Part ID(*) in STEP BOM is translated to PartName in ENF.  The component name is translated to PartNumber in ENF.  (*) Identifier.id is used as Part ID in preference to Id.id.</li> </ul>
<p>(11) STEP ENF Reader</p> <ul style="list-style-type: none"> <li>- Enabled to translate name of isolated curves and points.</li> <li>- Enabled to translate transparency of volumes.</li> <li>- Enhanced to translate relations from PMI to other graphic PMI in ENF to STEP AP242 translation. (00055770, 00055772, 00055774)</li> <li>- Enhanced to translate relations from PMI to volumes in ENF to STEP AP242 translation. (00055773)</li> <li>- Added a new parameter "SavedViewMethod" to specify how to translate Saved Views for a target CAD system in ENF to STEP AP242 translation.</li> <li>- Enabled to translate PMI Validation Properties (Affected Geometry) in ENF to STEP AP242 translation.</li> <li>- Enabled to preserve the show/hide status of PMIs for each 3D View when using STEP importing function of SOLIDWORKS to import translated STEP AP242.</li> <li>- Fixed an issue that a target point of Saved View was translated to a wrong position in ENF to STEP AP242 translation.</li> </ul>
<p>(12) JT ENF Writer</p> <ul style="list-style-type: none"> <li>- Added new parameters "ForceBodyNode" and "ForceBodyNodeFile" that enable nodes that are not Body Nodes in JT to be treated as Body Nodes in JT to ENF translation. (00054361)</li> <li>- Fixed a model-dependent issue that instance placement and polygon color were not translated correctly.</li> </ul>
<p>(13) JT ENF Reader</p> <ul style="list-style-type: none"> <li>- Fixed a model-dependent issue that redundant related elements were converted for assembly PMIs.</li> </ul>
<p>(14) 3D PDF ENF Reader</p> <ul style="list-style-type: none"> <li>- Enabled to translate parallel-on-screen annotations to 3D PDF.</li> <li>- Fixed an issue that the viewing performance was lowered for 3D PDF, Validation Report (3D PDF) and PDQ Check Result Report (3D PDF) when they were exported from CATIA V5.</li> <li>- Fixed an issue that Validation Report (3D PDF) and PDQ Check Result Report (3D PDF) exported from CATIA V5 displayed unnecessary wires. (00055644)</li> </ul>
<p>(15) CATIA V5(Plug-in) to JT</p> <ul style="list-style-type: none"> <li>- Updated Daimler translation setting files to support Daimler JT Supplier Package 20.06.</li> </ul>
<p>(16) CATIA V5(Plug-in) to 3D PDF</p> <ul style="list-style-type: none"> <li>- Fixed an issue that bold points are shown on the thumbnails in View Carousel. (00055582)</li> </ul>
<p>(17) Creo Parametric (Plug-in) to STEP</p> <ul style="list-style-type: none"> <li>- Fixed an issue that it failed to translate a model which includes quilts. (00054860)</li> </ul>
<p>(18) CAD Validator</p> <ul style="list-style-type: none"> <li>- Improved the accuracy of component mapping to match components which have different names but are located at the same position when comparing assembly models.</li> <li>- Improved the instance name validation logic to ignore the serial number added to the end of instance name and not to detect it as a difference when the second model of validation is CATIA V5. (00055326)</li> <li>- Enhanced the validation between source NX and target CATIA V5 which is translated from NX(Plug-in) to CATIA V5(Plug-in) as follows. <ul style="list-style-type: none"> <li>- Fixed an issue that differences in an identification line of line weld were detected excessively. (00055327)</li> <li>- Fixed a model-dependent issue that differences in a target point of a model view were detected excessively. (00055330)</li> <li>- Changed the tolerance of transparency validation not to detect unnecessary differences. (00055331)</li> </ul> </li> </ul>

- Fixed an issue that differences in a leader zone type of GDT were detected excessively. (00055775)
- Changed the PMI highlighting method in 3D HTML validation report to make it easier to distinguish PMI.

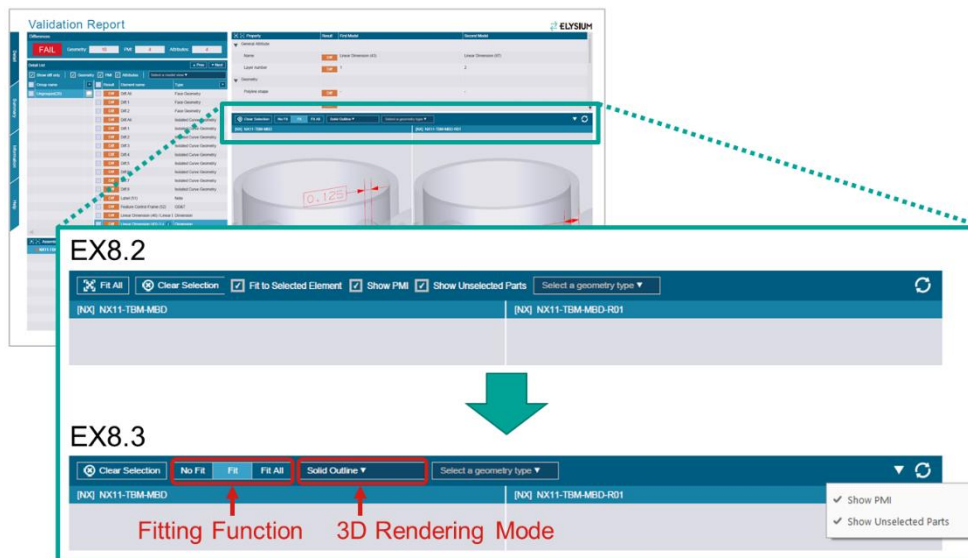
EX8.2



EX8.3



- Improved UI of 3D PDF validation report.
  - Added a UI component to switch the render mode of the two 3D views.
  - Changed UI of fitting functions.



- Enabled to create groups automatically by applying a customized script in a post process. Please contact Elysium or partners for details on automated grouping and CAD Validator customization.
- Fixed an issue that a Diff-Analyzer file could fail to be exported when an anti-virus software is installed.

(19) Data Package Studio

- Provided a public JavaScript API (TDP JS API) to enable simple access and control of a 3D View widget, such as show/hide PMI items. Please refer to the Data Package Studio documentation (PDF) for the details. The Data Package Studio online documentation is also available besides PDF.

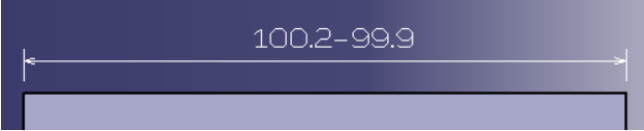
URL	<a href="https://doc.elysium-global.com">https://doc.elysium-global.com</a>
ID	cs_dps
Password	LAjXGNtP

\* The contents of the online documentation are the same as that of PDF.

(20) Slave Node

- Fixed an issue that it failed to run "ASFALIS Component Service" when a service registered user could not access the shared folder.

## 4. Bekannte Probleme (engl.)

(1)	The character codes of the text log may be mixed in ASFALIS EX6.* or later.
(2)	The total value of mass properties may not be translated in CAD to ENF translation when Memory Saving Mode is enabled.
(3)	CATIA V5(Plug-in) ENF Writer Due to CATIA V5 API limitation, the face color may not be translated correctly. (on CATIA V5-6R2018 Sp1 or later)
(4)	CATIA V5(Plug-in) ENF Writer The left and right values may flip during CATIA V5 to ENF translation when the source model is created by V5-6R2019 and has a dimension which includes "-"(dash) between the two values.  
(5)	CATIA V5(Plug-in) ENF Reader 3D Annotation Option A CAD API defect causes the following problems: <ul style="list-style-type: none"> <li>- The leader line for diameters can be translated as left and right reversed.</li> <li>- Angular dimensions will not be translated. (on CATIA V5-6R2013)</li> </ul>
(6)	3DEXPERIENCE ENF Writer Due to 3DEXPERIENCE API specification changes, the specification of hidden elements translation differs depending on the 3DEXPERIENCE version. The elements whose display flag is "Show" but whose layer filter status is "Hidden" on 3DEXPERIENCE will be translated as below when the value of the parameter "ConvertNoShowElement" is set to "0" (0: Don't translate hidden elements). <ul style="list-style-type: none"> <li>- R2019x FD05/FD06, R2020x Golden/FD01: The elements are NOT translated to ENF</li> <li>- R2019x FD04 or earlier/FD07 or later, R2020x FD02 or later: The elements are translated to ENF</li> </ul>
(7)	3DEXPERIENCE ENF Reader (3DEXPERIENCE R2017x) Due to 3DEXPERIENCE API limitation, the clipping plane of capture is not translated.
(8)	Creo Parametric (Plug-in) ENF Writer (Creo Parametric 2.0) A CAD API defect causes the following problem: <ul style="list-style-type: none"> <li>- Hidden entities of quilt may be translated as visible entities.</li> </ul> This issue has been fixed in M030.
(9)	Creo Parametric (Plug-in) ENF Writer When using Creo Parametric 7.0, the mass properties of assembly will not be translated to ENF if the density is not set in parts under Creo assembly model.
(10)	Creo Parametric (Plug-in) ENF Reader The translation time of a model with multi volumes might be longer than with a single volume. Please refer to the following information available on PTC's Support Site for further details.  <a href="https://www.ptc.com/en/support/article?n=CS317562">https://www.ptc.com/en/support/article?n=CS317562</a>
(11)	Creo Parametric (Plug-in) ENF Reader When using Creo Parametric 7.0, the mass properties of assembly will not be translated to Creo if the density is not set in the template of part which is used in translation.
(12)	Creo Parametric (Plug-in) ENF Writer/Reader When the following option is written in the configuration files of Creo Parametric (config.sup or config.pro), ENF Writer/Reader may crash depending on model data. In the case of crash, specify the value "no" for this option: <pre>show_geom_checks_on_creation yes</pre>

<p>(13) Creo Parametric (Plug-in) ENF Writer/Reader The following issues have been found in Creo Parametric ENF Writer/Reader.</p> <ul style="list-style-type: none"> <li>- When exporting neutral files with Creo Parametric ENF Writer. <ul style="list-style-type: none"> <li>- There is a model-dependent issue that the instance color and the transparency may not be translated correctly. (Creo Parametric 3.0 or later)</li> </ul> </li> <li>- When importing neutral files with Creo Parametric ENF Reader. <ul style="list-style-type: none"> <li>- There is an issue that the color and the transparency of the back side of faces are not translated correctly. (Creo Parametric 3.0 / 4.0)</li> <li>- There is an issue that the processing time for translating isolated curves will increase. (It takes extremely long to translate a model with more than 10 thousand isolated curves) (Creo Parametric 3.0 or later)</li> </ul> </li> </ul>
<p>(14) Creo Parametric (Plug-in) ENF Writer/Reader It may fail to translate in Creo Parametric ENF Writer/Reader because of Creo Parametric crashing. We confirmed that the error has occurred between 0.1% to 4% (1 to 40 times for every one thousand times) depending on the environment, according to our test. (Now we are investigating the cause of the error.)</p>
<p>(15) Creo Parametric (Plug-in) ENF Writer/Reader When executing translation on Windows 8.1 using Creo Parametric 3.0 or later version, translation may fail depending on the application status of Windows Update. Please refer to the following information available on PTC's Support Site for further details.</p> <p><a href="https://www.ptc.com/en/support/article?n=CS269539">https://www.ptc.com/en/support/article?n=CS269539</a></p>
<p>(16) NX (Plug-in) ENF Writer 3D Annotation Option There is a restriction on the data translation of 3D annotation with NX ENF Writer</p> <ul style="list-style-type: none"> <li>- The texts of "Hole Dimension" will not be translated when translating NX data from NX9.0 or later.</li> </ul>
<p>(17) NX (Plug-in) ENF Writer When translating from NX by using ASFALIS SmartLauncher (Plug-in) with on-memory mode, properties (e.g., show/hide status etc.) and translation target elements could not be translated correctly according to the configuration of NX assembly load options. Related menu: NX Menu &gt; [File] &gt; [Options] &gt; [Assembly Load Options]</p> <p>To export by using the on-memory mode, select "Fully Load" as the load option to open the NX model. It has been reported that the following selections in load option may cause incorrect translation.</p> <ul style="list-style-type: none"> <li>- "Partially Load - Lightweight Display"</li> <li>- "Minimally Load - Lightweight Display"</li> </ul> <p>Note that the default setting of NX assembly load options is changed to "Partially Load - Lightweight Display" from NX 1899 Series.</p>
<p>(18) NX (Plug-in) ENF Reader (NX9 or earlier) When the model that has the part and sub-assembly including Drafting PMI is translated using NX9 Legacy adapters, there is an issue that the model may not be displayed when switching the Model View in NX.</p>
<p>(19) NX (Plug-in) ENF Writer/Reader (NX11) There have been reports that NX11 to/from ENF translation failed with some versions of graphic board driver. The following is a combination of graphic boards and driver versions that have been confirmed to fail translation:</p> <ul style="list-style-type: none"> <li>- NVIDIA Quadro P600, v23.21.13.8816 or later</li> <li>- NVIDIA Quadro P620, v24.21.13.9793 / v24.21.14.1195</li> <li>- NVIDIA Quadro K420, v23.21.13.9077</li> </ul> <p>Set the following environment variable to avoid translation errors. JT_OGL45=1</p>
<p>(20) NX (Plug-in) ENF Writer/Reader The translation of multi-plane section (e.g., two parallel-plane type and box type) is available only in NX11 or later.</p>
<p>(21) NX I-deas ENF Writer</p>

	Due to the IDI API limitation, hidden entities of Coordinate System (CS) will be translated as visible entities.
(22)	NX I-deas ENF Writer/Reader The NX I-deas API defect may cause an inconvenience that NX I-deas window is displayed during the translation with NX I-deas6.7.
(23)	SOLIDWORKS ENF Writer The SOLIDWORKS API defect may cause a model-dependent issue that face colors are not translated correctly.
(24)	SOLIDWORKS ENF Reader The SOLIDWORKS API defect may cause a translation error when the original SW model has many parts. The following environment variable settings may help to avoid the translation error. <pre> ELY_SW_PREOPEN_DOCUMENT=1 ELY_CONFIG_ELCFB_ENABLE_SERVER=1 ELY_FBT_FORCE_PARALLEL_PROCESSING=1 ELY_CONFIG_ELCFB_SERVER_RESTART_COUNT=1000 ELY_CONFIG_ELCFB_NUM_OF_PROCESSES=1 </pre>
(25)	SOLIDWORKS ENF Writer/Reader Please note that an error may occur when using SOLIDWORKS ENF Writer/Reader on the same computer at the same time. Please ensure that SOLIDWORKS adapters will not be used on the same computer at the same time when setting Slave Nodes.
(26)	Autodesk Inventor ENF Reader It may encounter an error when trying to save a data on a mounted folder on network.
(27)	Autodesk Inventor ENF Reader The part names such as Part1, Part2 and so on which are often used by default may get renamed to Part1U0, Part2U1 and so on during translation depending on the environment.
(28)	Autodesk Inventor ENF Reader The memory consumption is higher when translating assembly models using Inventor 2017 compared to Inventor 2016 or earlier. This is because a thumbnail is created at data import of e.g., ACIS files due to a restriction from CAD system side.
(29)	Parasolid ENF Reader Translating a model containing multibyte character in its element name to a Parasolid text format (.x_t, .xmt_txt) file may cause problems when importing to another system. You can avoid the error by using a binary format (*.x_b, *.xmt_bin).
(30)	STEP ENF Reader When the STEP file of the assembly model translated by ASFALIS is opened by SOLIDWORKS 2009 or before, a defect in SOLIDWORKS causes the following problems. <ul style="list-style-type: none"> <li>- Step files will be read as empty assembly models.</li> <li>- The assembly structure and matrix information will be lost and read as parts.</li> </ul>
(31)	JT ENF Writer 3D Annotation Option There are restrictions on the data translation of 3D annotation with JT ENF Writer. <ul style="list-style-type: none"> <li>- Following properties of GDT are not supported: <ul style="list-style-type: none"> <li>- Common Zone</li> <li>- Maximum</li> <li>- Unit Basis Value</li> <li>- All over</li> <li>- Third or later datum feature of Compound Datum</li> </ul> </li> <li>- Following property of surface finish is not supported: <ul style="list-style-type: none"> <li>- Tolerance of Machining Allowance</li> </ul> </li> <li>- Special characters (such as super-/subscript) in PMI in NX data will be translated to standard characters in the translation of JT data exported from NX.</li> </ul>
(32)	JT ENF Writer The JT library defect may cause an incorrect translation of Polygon geometry when translating ULP (Ultra Lightweight Precise) elements which are created in JT v10.3 or earlier.
(33)	3D PDF ENF Reader

<ul style="list-style-type: none"> <li>- It has been reported that parallel-on-screen PMIs are not shown on the thumbnails in View Carousel.</li> <li>- It has been reported that parallel-on-screen PMIs are not translated to 3D PDF as parallel-on-screen if they have a leader line.</li> </ul>
<p>(34) 3D PDF Report The display (highlight / exclusive show of selected elements) may not function correctly due to the restriction on the 3D PDF rendering.</p>
<p>(35) 3D PDF Report Please note that there is an issue that Left/Right 3D Views are not synchronized to each other when manipulating a model with 3D mouse/Spaceball.</p>
<p>(36) 3D PDF Report Please note that you may fail to open 3D PDF reports when the model contains many PMIs. (As a reference, we failed when a model had more than 3,000 PMIs in our test.)</p>
<p>(37) 3D PDF Report When opening a 3D PDF attached to another PDF, save the 3D PDF report before opening. It is reported that directly opening the attached 3D PDF before saving does not work correctly.</p>
<p>(38) 3D PDF Report It has been reported that the viewing performance of 3D PDF report is lower than EX8.1 when the report includes an attribute difference on B-rep (e.g., color and transparency) with many related elements.</p>
<p>(39) HTML Report Processing time as well as memory consumption will increase compared to EX7.1 when opening HTML validation report that contains many detected differences. (For your reference, increased by 50 percent in our test.)</p>
<p>(40) HTML Report The validation results of hatching patterns and caps of cross sections are displayed in 3D view only under the following condition.</p> <ul style="list-style-type: none"> <li>- The cross section is defined only by a single plane</li> <li>- There is no interference between each solid</li> </ul>
<p>(41) "Supplier Name" will not be applied to the file names of top files (PLM or JT files) when performing CATIA V5 to PLM XML or CATIA V5 to JT translation on ASFALIS Desktop Controller or ASFALIS TransServer with Daimler settings on.</p>

## 5. Beschränkungen bestimmter Module (engl.)

<p>(1) New adapters are supported for the following adapters.</p> <ul style="list-style-type: none"> <li>- CATIA V5 (Plug-in) ENF Writer/Reader</li> <li>- 3DEXPERIENCE (Plug-in) ENF Writer/Reader</li> <li>- NX (Plug-in) ENF Writer/Reader</li> <li>- Creo Parametric (Plug-in) ENF Writer/Reader</li> <li>- JT ENF Writer/Reader</li> <li>- PLM XML ENF Writer/Reader</li> <li>- STEP AP242 BOM XML ENF Writer/Reader</li> <li>- SOLIDWORKS ENF Writer/Reader</li> </ul> <p>New adapters other than the above are available but not supported.</p>
<p>(2) ENF Reader Because of the overhead for parallelization, performance of the new adapter in terms of speed and memory usage may sometimes be worse than the legacy adapter, especially for small models.</p>
<p>(3) ENF Reader Interval of progress update is rough compared to the legacy adapter.</p>
<p>(4) CATIA V5 (Plug-in) ENF Writer Parameters "NoShow" and "NoShowInstance" accept only values "0"/"1", not "Yes"/"No".</p>
<p>(5) CATIA V5 (Plug-in) ENF Writer Parameters "PartNumber", "BodyName", "PartName" and "WriteProperty" are not available.</p>
<p>(6) CATIA V5 (Plug-in) ENF Writer User attributes are not translated when the value of the parameter "ProductMode" is set to "2".</p>
<p>(7) CATIA V5 (Plug-in) ENF Reader Parameters "InstancePathDisp" and "KeepNativeFilename" accept only values "0"/"1", not "Yes"/"No".</p>
<p>(8) CATIA V5 (Plug-in) ENF Reader When an assembly is imported as internal component, the order of instances may not be preserved.</p>
<p>(9) Creo Parametric (Plug-in) ENF Writer The translation time may be longer than the legacy adapter.</p>
<p>(10) NX (Plug-in) ENF Writer Parameter "ProductMode" does not support "2" as its value.</p>
<p>(11) JT ENF Reader When a relative tolerance is specified during tessellation, the translation speed of parallelization mode may sometimes be much slower than non-parallelization mode.</p>

## 6. Anmerkungen (engl.)

(1)	CATIA V5 (Plug-in) ENF Reader Please note that it may take long to translate using V5-6R2014 or later when the number of referenced objects of PMI is large.
(2)	"Remove Invisible Part" and "Extract Visible Face" of Geometry Simplifier If you wish to run the sub process "RemoveInvisiblePart" or "ExtractVisibleFaceRemove", it is recommended to use a machine with a video board which supports OpenGL Ver3.0 or later. The power of these functionalities will be equivalent as EX7.2 if you run these sub processes on a machine without the GPU.
(3)	For PDQ Checker, check tolerance could be referred by another check item.
(4)	It has been reported that you may experience unsteadiness in the performance of SOLIDWORKS in the environment where Microsoft Update "KB3072630" is applied. Please check whether your environment meets these conditions when SOLIDWORKS ENF Writer/Reader does not behave properly. This issue is fixed in SOLIDWORKS 2016 PR1 and SOLIDWORKS 2015 SP5.0. Please refer to the following page on SOLIDWORKS website for the details of the behavior and workaround. <a href="https://www.solidworks.com/sw/support/TechKB3072630.htm">https://www.solidworks.com/sw/support/TechKB3072630.htm</a>
(5)	Regarding Visual C++ Runtime libraries Visual C++ Libraries are required to run the ASFALIS components. Please install the libraries from the included installer (vcredist_x86.exe/vcredist_x64.exe) if your environment does not have the Runtime libraries. Included installer: <ul style="list-style-type: none"><li>- Microsoft Visual C++ 2005 Service Pack 1 Redistributable Package MFC Security Update (KB2538242)</li><li>- Microsoft Visual C++ 2008 Service Pack 1 Redistributable Package MFC Security Update (KB2538243)</li><li>- Microsoft Visual C++ 2010 Service Pack 1 Redistributable Package MFC Security Update (KB2565063)</li><li>- Visual C++ Redistributable for Visual Studio 2012</li><li>- Visual C++ Redistributable for Visual Studio 2013</li><li>- Visual C++ Redistributable for Visual Studio 2015 - 2019</li></ul>
(6)	SOLIDWORKS ENF Writer: This does not support the translation of CAD models that are created in SOLIDWORKS 2017sp2 or later using 3D Interconnect functionality and contain CAD data in third-party native format.



## 7. Adapter Status

Enhancement, Specification Changes, and Issue Fixing are implemented according to the policy below.

CAD	Adapter Type		Shipment Status		Update Method	
			New Adapter	Legacy Adapter	New Adapter	Legacy Adapter
CATIA V5	Plug-in	Writer	△	✓	A	A
		Reader	✓	△	A	A
CATIA V5	Standalone	Writer	-	✓	-	A
		Reader	-	✓	-	A
3DEXPERIENCE	Plug-in	Writer	✓	-	A	-
		Reader	✓	-	A	-
CATIA V4	Standalone	Writer	-	✓	-	A
		Reader	-	✓	-	A
NX	Plug-in	Writer	△	✓	A	A
		Reader	✓	△	A	B
NX	Standalone	Writer	-	✓	-	A
Creo Parametric	Plug-in	Writer	△	✓	B	A
		Reader	✓	△	A	B
Creo Parametric	Standalone	Writer	-	✓	-	A
NX I-deas	Plug-in	Writer	-	✓	-	A
		Reader	-	✓	-	A
SOLIDWORKS	Plug-in	Writer	✓	△	A	B
		Reader	✓	△	A	B
Creo Elements/Direct	Plug-in	Writer	-	✓	-	A
		Reader	-	✓	-	A
Autodesk Inventor	Plug-in	Writer	△	✓	A	A
		Reader	△	✓	A	A
CADmeister	Standalone	Writer	-	✓	-	A
		Reader	-	✓	-	A
iCAD	Standalone	Writer	-	✓	-	A
		Reader	-	✓	-	A
Parasolid	Standalone	Writer	-	✓	-	A
		Reader	-	✓	-	A
ACIS	Standalone	Writer	-	✓	-	A
		Reader	-	✓	-	A
STEP	Standalone	Writer	-	✓	-	A
		Reader	-	✓	-	A
STEP BOM XML	Standalone	Writer	✓	-	A	-
		Reader	✓	-	A	-
JT	Standalone	Writer	✓	△	A	A
		Reader	✓	△	A	A
PLM XML	Standalone	Writer	✓	-	A	-
		Reader	✓	-	A	-
3DXML	Standalone	Writer	-	✓	-	A
IGES	Standalone	Writer	-	✓	-	A
		Reader	-	✓	-	A
STL	Standalone	Reader	-	✓	-	A
XVL	Standalone	Reader	-	✓	-	A

3D PDF	Standalone	Reader	-	✓	-	A
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**[Legend]**

Shipment Status	✓	Available and used by default
	△	Available but not used by default
	-	Not available
Update Method	A	All the updates are implemented to all the supported CAD versions
	B	Some of updates are implemented to all the supported CAD versions
	C	All the updates are implemented to latest 4 supported CAD versions Other CAD versions are guaranteed to operate but no update is implemented
	D	Some of updates are implemented to latest 4 supported CAD versions Other CAD versions are guaranteed to operate but no update is implemented
	-	No update is implemented

## 8. Unterstützte CAD Versionen

Bitte vergleichen Sie auch das gesonderte Dokument Requirement\_ENG.pdf.

CAD	Supported CAD Versions
CATIA V5 (Plug-in)	R19, R20, R21, R22(V5-6R2012), R23(V5-6R2013), R24(V5-6R2014), R25(V5-6R2015), R26(V5-6R2016), R27(V5-6R2017), <u>R28(V5-6R2018)</u> , <u>R29(V5-6R2019)</u> , <b>R30(V5-6R2020)</b>
CATIA V5 (SA)	Import: R7 - <b>R30(V5-6R2020)</b> Export: R14, R19, R20
3DEXPERIENCE (Plug-in)	<u>R2017x</u> , <u>R2018x</u> , <u>R2019x</u> , <u>R2020x</u>
CATIA V4 (SA)	V4.2.1 - V4.2.5
NX (Plug-in)	NX9, NX10, NX11, NX12, NX 1847 Series, <u>NX 1872 Series</u> , <b><u>NX 1899 Series</u></b> , <b><u>NX 1926 Series</u></b>
NX (SA)	UG10 - <b>NX 1899 Series</b>
Creo Parametric (Plug-in)	Creo Parametric 2.0, Creo Parametric 3.0, Creo Parametric 4.0, <u>Creo Parametric 5.0</u> , <u>Creo Parametric 6.0</u> , <b><u>Creo Parametric 7.0</u></b>
Creo Parametric (SA)	2000i - Creo Parametric 6.0
NX I-deas (Plug-in)	NX I-deas6.3, 6.4, 6.5, 6.6, 6.7
SOLIDWORKS (Plug-in)	SOLIDWORKS 2015, 2016, 2017, <u>2018</u> , <u>2019</u> , <u>2020</u>
Creo Elements/Direct (Plug-in)	Creo Elements/Direct 18.1, 19.0, 20.0, 20.1, 20.2, <b>20.3</b>
Autodesk Inventor (Plug-in)	Inventor 2017, 2018, 2019, 2020, <b>2021</b>
Parasolid (SA)	V7 - V32.0
ACIS (SA)	R6 - R29
STEP (SA)	AP203, AP214, AP242(ed1/ <b>ed2</b> )
JT (SA)	Import: v6.4 - v10.5 Export: v8.0 - v10.5
PLM XML (SA)	-
3DXML (SA)	R2010x - R2019x
IGES (SA)	5.2, 5.3
STL (SA)	-
XVL (SA)	P-XVL(V3), V-XVL(V5, V7, V9), U-XVL(V10, V11)

\*Rot: Neu mit EX8.3

\*CAD Plug-ins für ASFALIS SmartLauncher sind nur für unterstrichene Systeme verfügbar

**HILFE**

Im Rahmen Ihres Wartungsvertrages stehen wir Ihnen gerne bei der Implementierung der neuen Version zur Verfügung.

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Wir wünschen Ihnen viel Erfolg bei der Arbeit mit ASFALIS.

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