



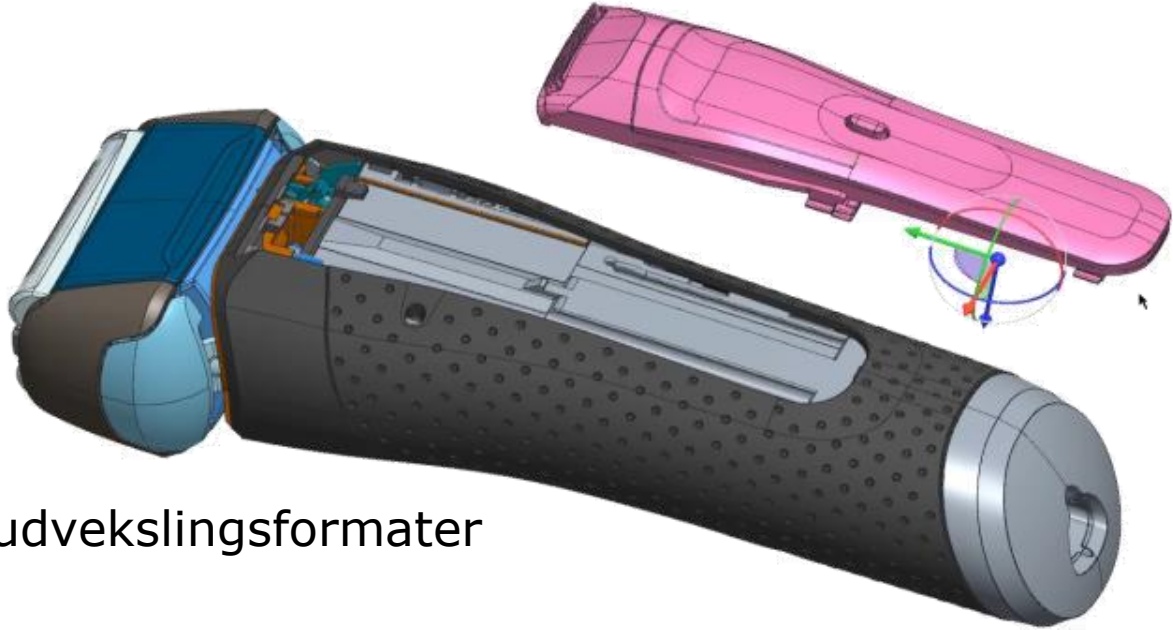
FREMMEDE DATA – PTC/USER 2017

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Creo kan håndtere fremmede filformater

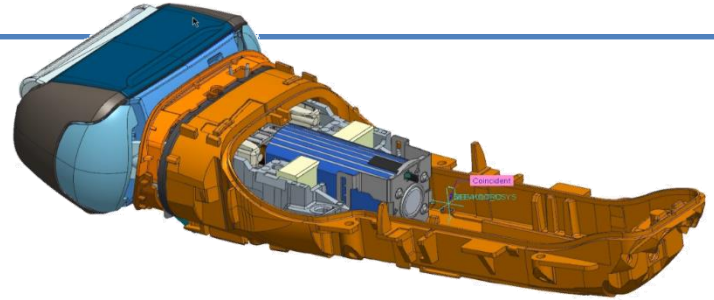
- Catia version 4
- Catia version 5
- Siemens NX
- SolidWorks
- SolidEdge
- Inventor

- Foruden almindelige 3D-udvekslingsformater
 - Step
 - Iges



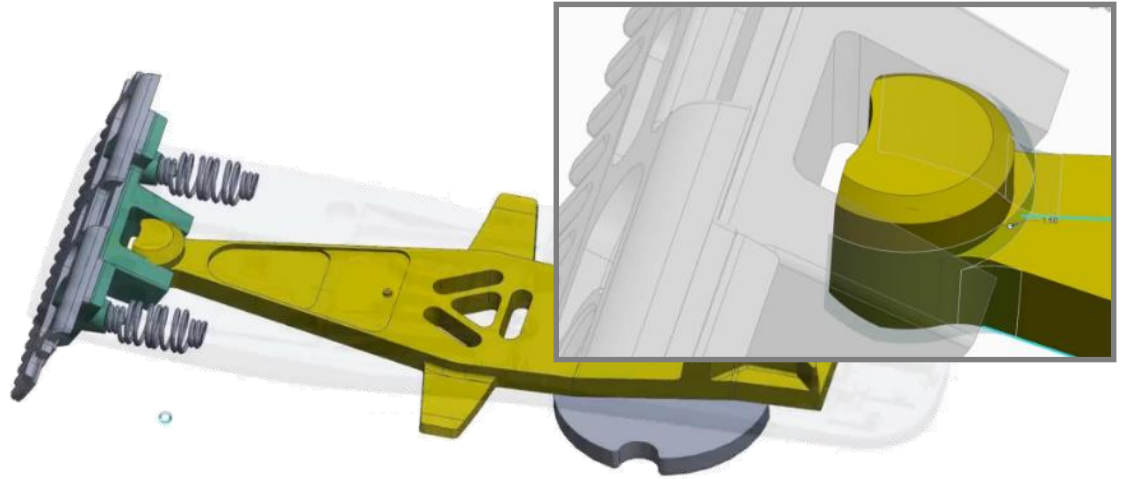
Fremmede filformater

- 2 Grundscenarier
 - Consolidation
 - Vi ønsker at vedligeholde vores dokumentation i Creo
 - Ændringer foretages med Flexible Modeling
 - Nymodellering sker med traditionelle Creo-features
 - Collaboration
 - Vi ønsker at samarbejde omkring data fra fremmede systemer
 - Ændringer foretages hvor det giver mening

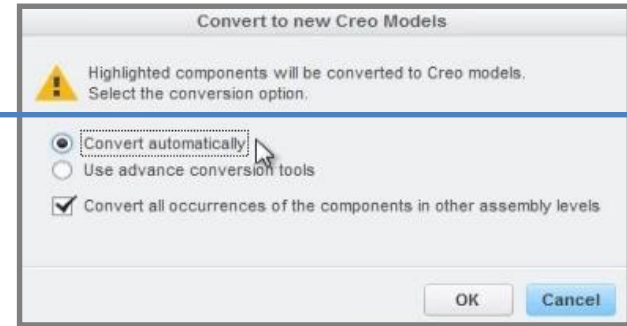


Fremmede filformater

- Consolidation
- **File Import** - SOLIDWORKS, CATIA, NX, Solid Edge and Inventor.
 - File Import converts non-PTC data to Creo objects
 - The native file and the PTC Creo object are both managed within Windchill or user hard drive



- Collaboration
- **File Open** - SOLIDWORKS, CATIA, NX.
 - Using File Open does not create a new primary business object to be managed in Windchill
 - The Creo object is created on-demand when the model is selected for editing
 - The native file and B-Rep content is managed in Windchill as a hidden object and visible on user hard drive if not using Windchill.
- **Save As**
 - Collaboration Extensions enable engineers to save 3D data as native SOLIDWORKS, CATIA and Siemens NX formats
- **Automatic Update**
 - Collaboration Extensions automatically detect when non-PTC data is out-of-date and gives the user the option to replace the content in session



Fremmede filformater

- Consolidation
 - Standard funktionalitet
- Collaboration
 - Collaboration Extensions til Creo for hvert af de fremmede CAD-systemer
 - Under Windchill-kontrol
 - Versioner er vigtige →

File Format	Import/Open	Update/Save As
CATIA V4	Up to 4.2.5	4.1.9
CATIA V5	R10 – R21 V5 6R2012 – 2013	R16 – R21 V5 6R2012 – 2013
Siemens NX	UG 13 to NX 8.5	NX 8.5 (Update) NX 3.0 (Export)
SolidWorks	2004 to 2013	2004 to 2013 (Update) 2005 (Export)