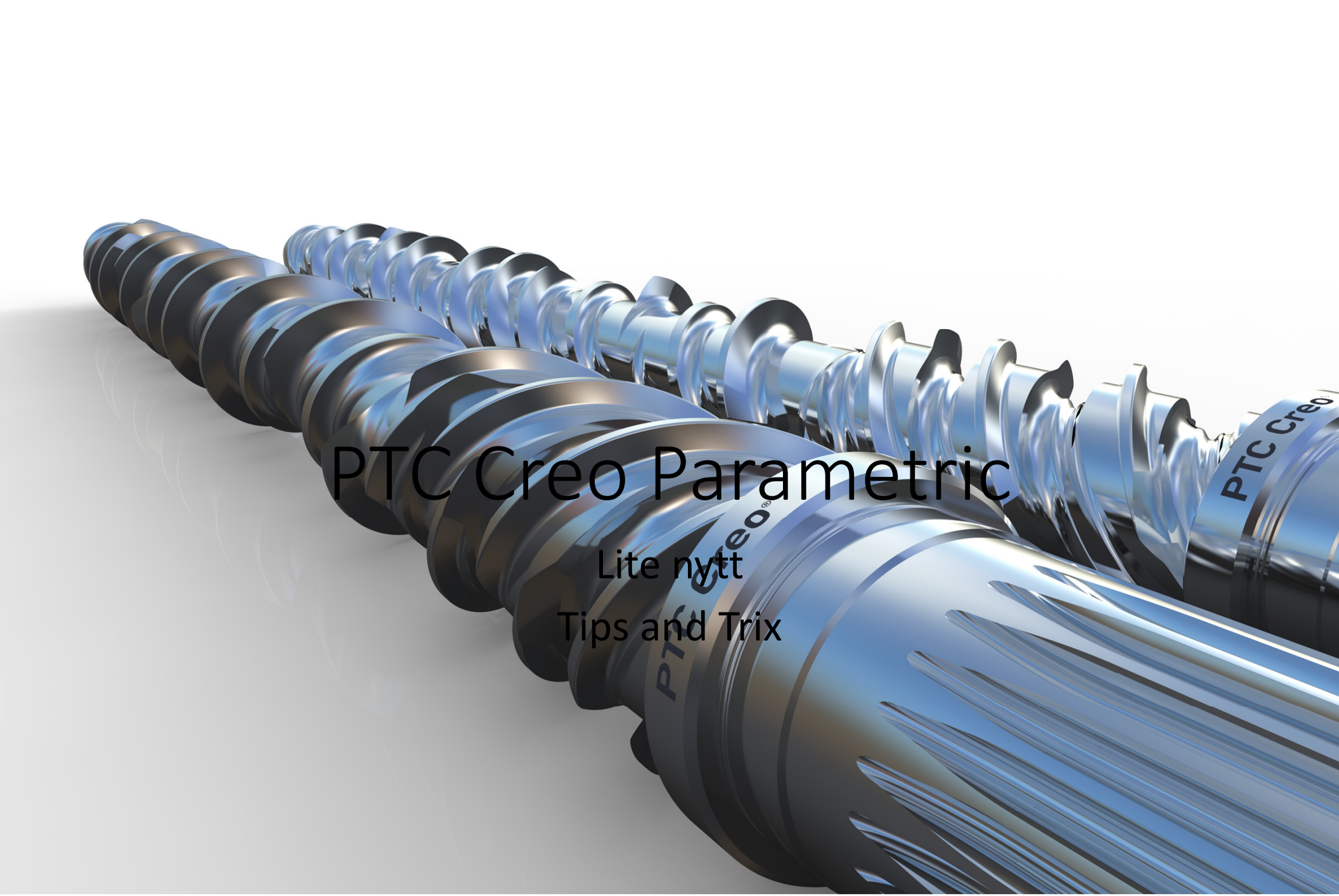


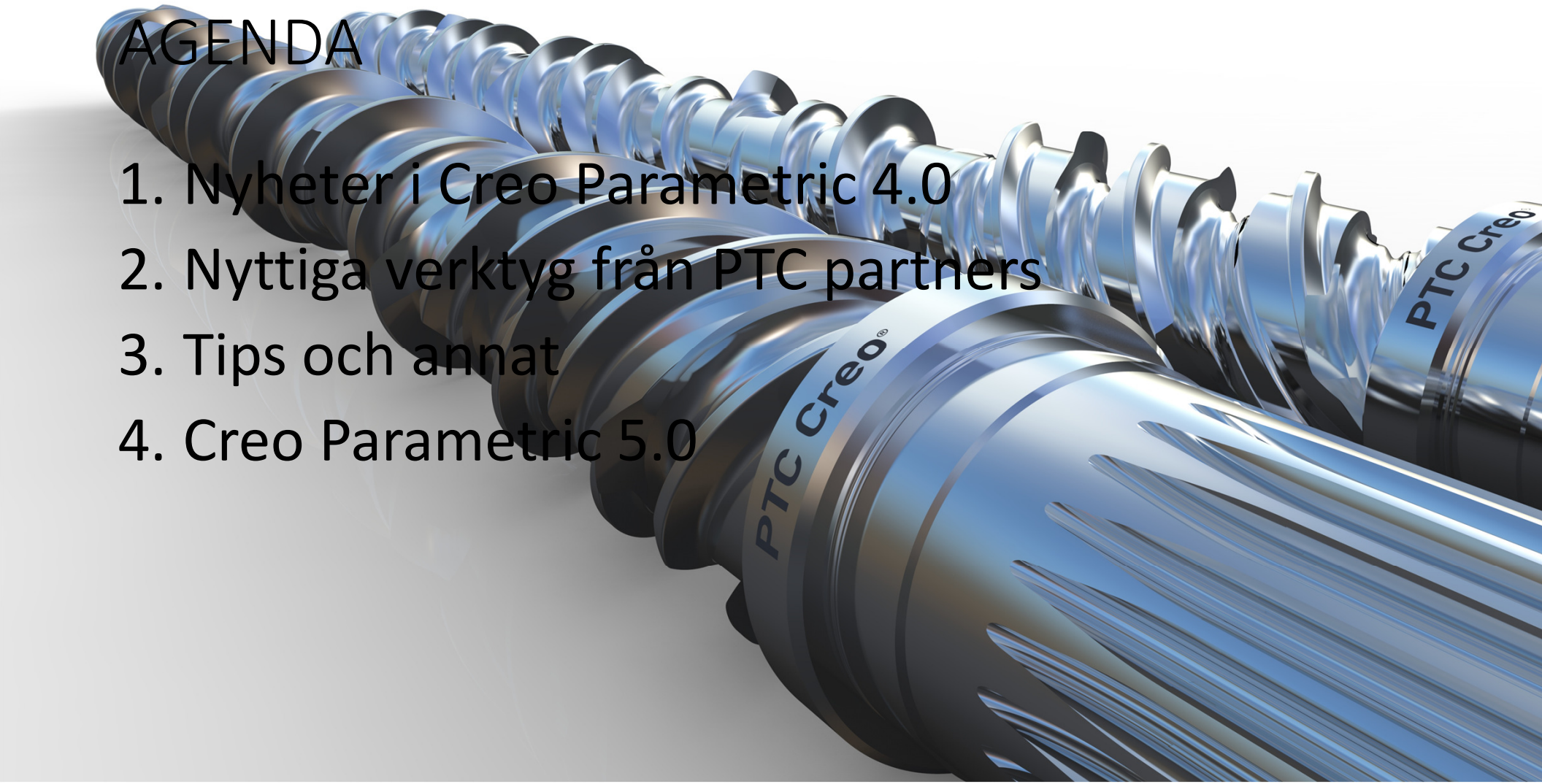
PTC Creo Parametric

Lite nytt

Tips and Trix

AGENDA

1. Nyheter i Creo Parametric 4.0
2. Nyttiga verktyg från PTC partners
3. Tips och annat
4. Creo Parametric 5.0



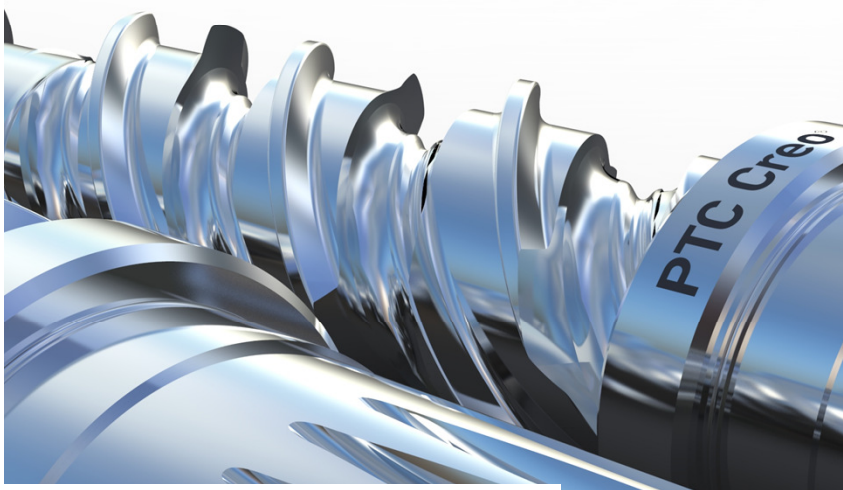
Creo Parametric 4.0 - Whats New

Creo Parametric 4.0 M010

- Run ModelCHECK for Creo Elements/Direct Modeling *.sdac and *.sdpc file using Open option
  - CS248781
- Publish Augmented Reality (AR) experience directly from Creo
  - CS257684
- New solution Clearance and Creepage Extension (CCX) added into Creo Parametric 4.0 M010
  - CS260097

Creo Parametric 4.0 M020

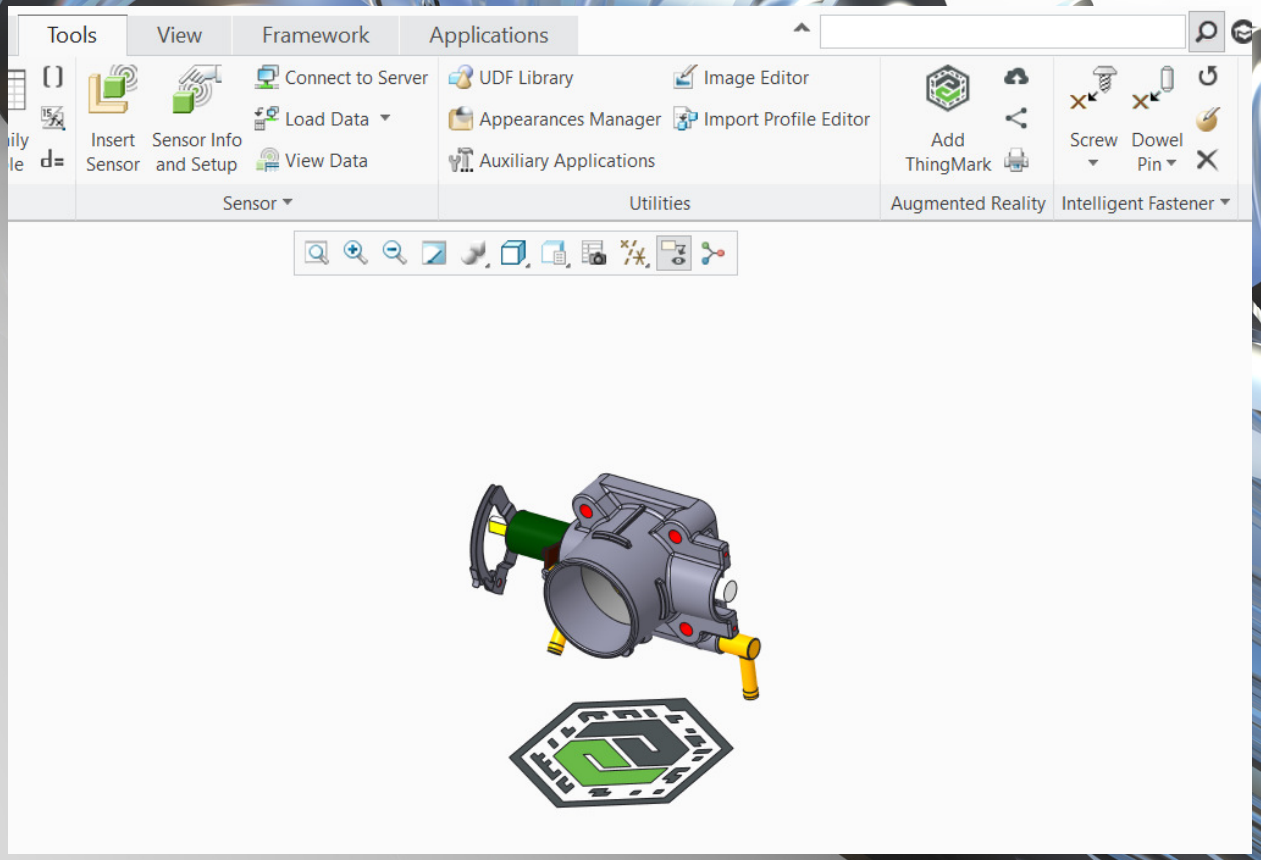
- STEP AP242 supports Semantic PMI (Product Manufacturing Information) on Import and Export
  - In previous releases and datecodes, only Graphical PMI has been supported, refer to CS193762
- Smart connected product design will be launched as **Creo Product Insight Extension**
  - The license option for **Creo Product Insight Extension** is **PRODUCT_INSIGHT** (**359**)
    - Product insight extension (Creo > Thingworx connection) with sensor features
      - Add **Sensor** by **File** > **Prepare** > **Model Properties**
    - Create an **Instrumented** assembly
  - Refer Creo Product Insight Extension Introduction video from http://learningexchange.ptc.com



Creo Parametric 4.0 M030

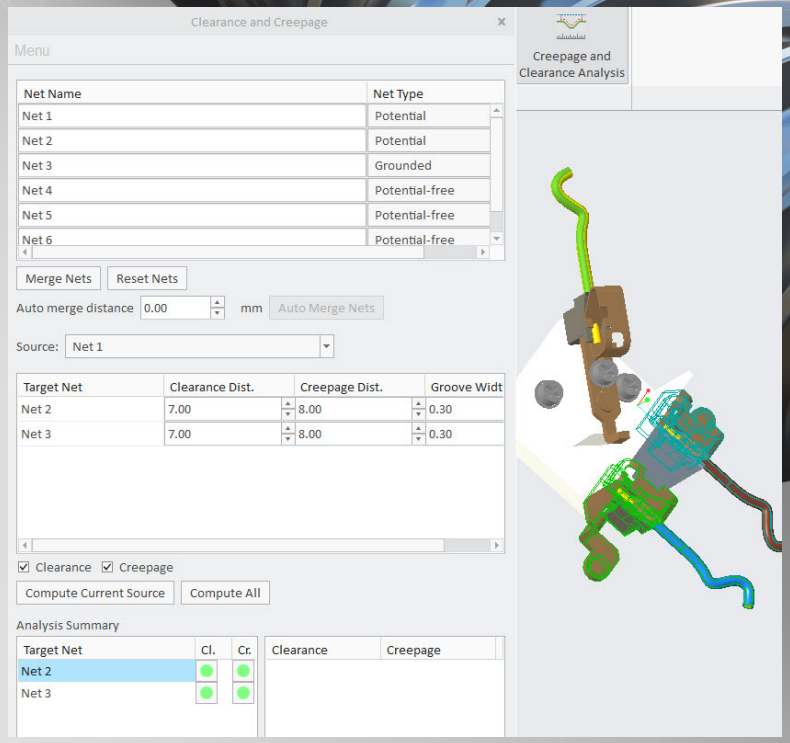
- **Creo Render Studio**
  - From Creo 4.0 M030, Creo Photo-Realistic Rendering is renamed to Creo Render Studio, also provide the related standalone App - **Creo Render Studio**
  - Requires CREOSHELL_Basic license to install and then Render Studio (feature 357) to run
- Creo 4.0 M030 import/open support **NX 11.0** and **Solidworks 2017**
  - CS134024
- **Creo Product Insight Extension**
  - Advanced sensor support
  - Virtual sensors
  - Mechanism analyses
  - Creo as a Service

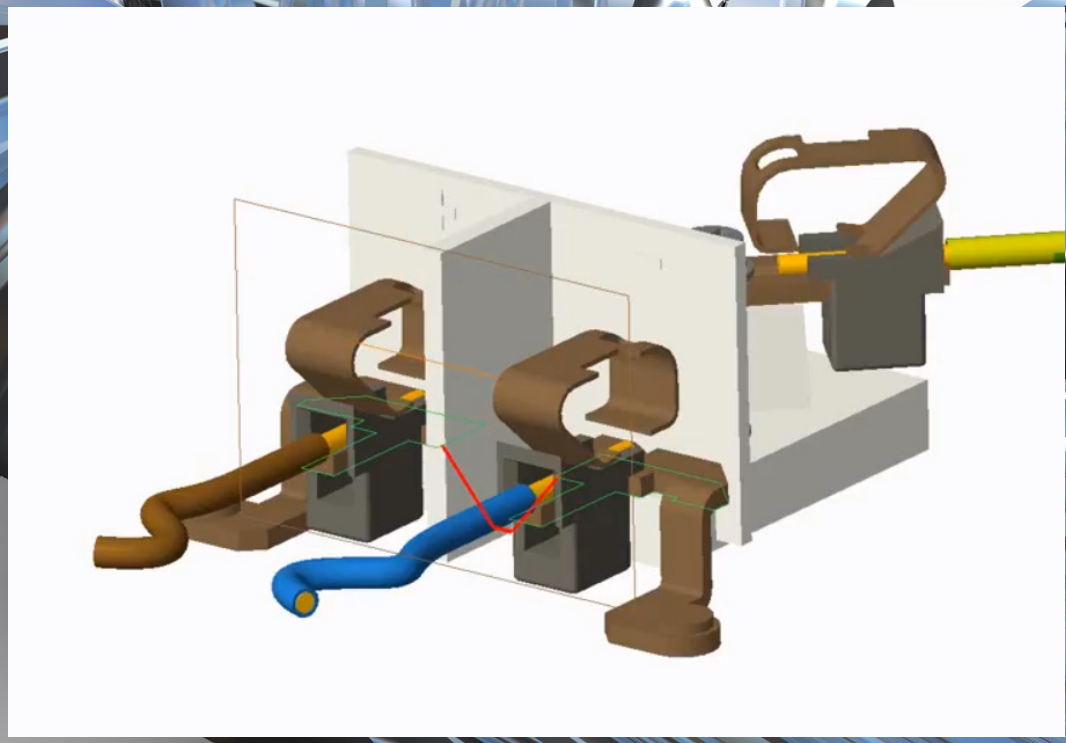
AUGMENTED REALITY (M010)



Creo Clearance and Creepage Extension (CCX)

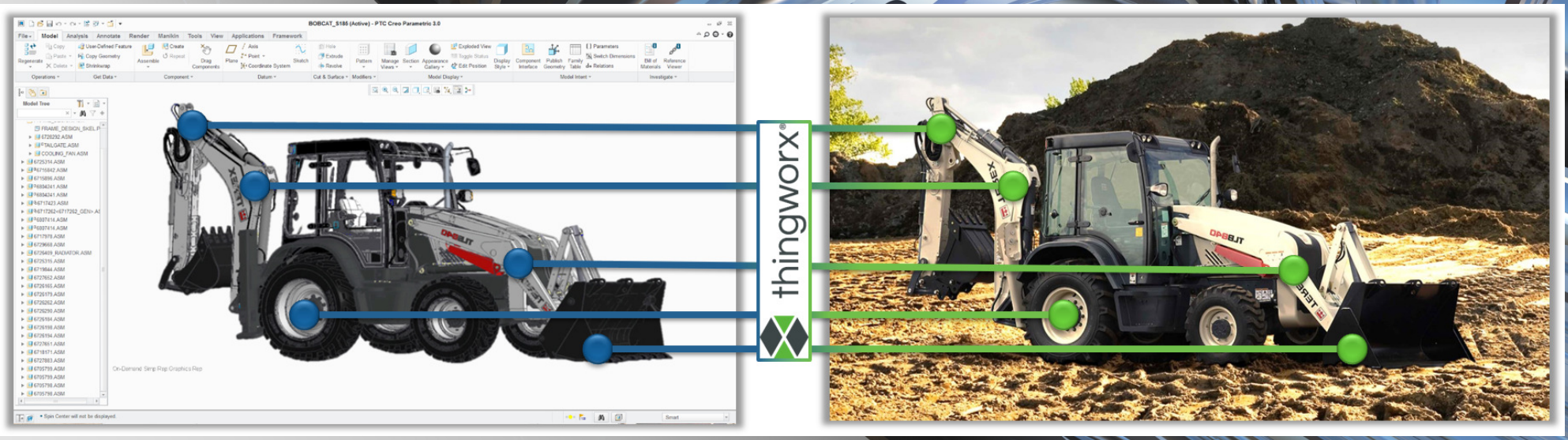
- Ersätter PTC® Creo® Spark Analysis Extension





Creo Product Insight Extension

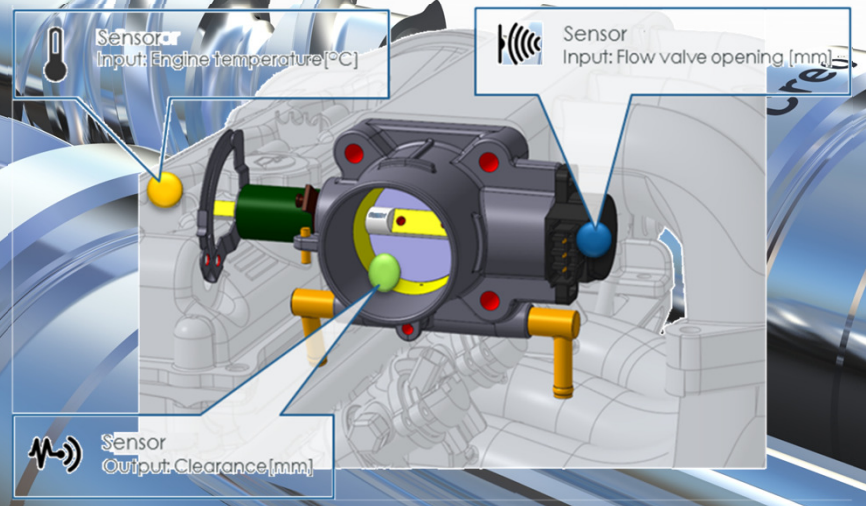
Creo Product Insight extension supports smart, connected product design allowing companies to take advantage of the IoT and **replace design assumptions with facts**



Creo Product Insight Extension

“Physical” Sensors (M020)

- Easily define and place “Measure” Sensors
  - Add **physical sensors** to Creo Assemblies
- New **Instrumented Assembly** sub-type
  - Protecting reused/released design data
- Associated parameter and input definitions
  - Define **parameters** and **associated calculations**
- Read real-world data from ThingWorx (or a data file) and use input variables to drive analyses
  - Run analyses and report results back to data table



CREO Render Studio

- Ersätter Creo Advanced Rendering (ARX) = Keyshot
- Fristående Applikation

Creo Parametric 4.0 M030

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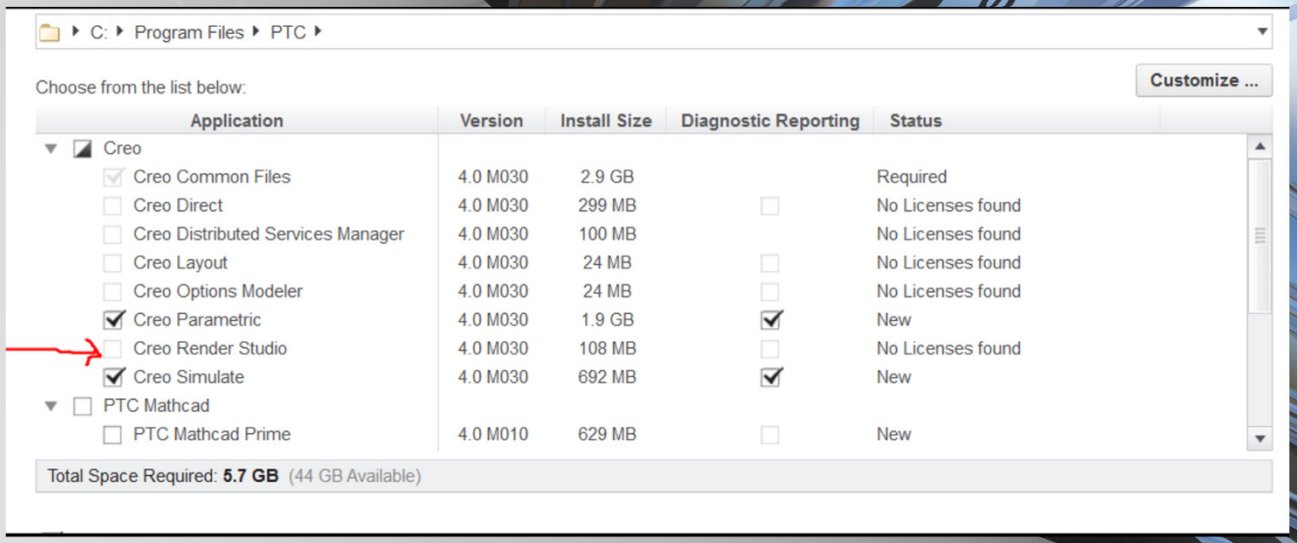
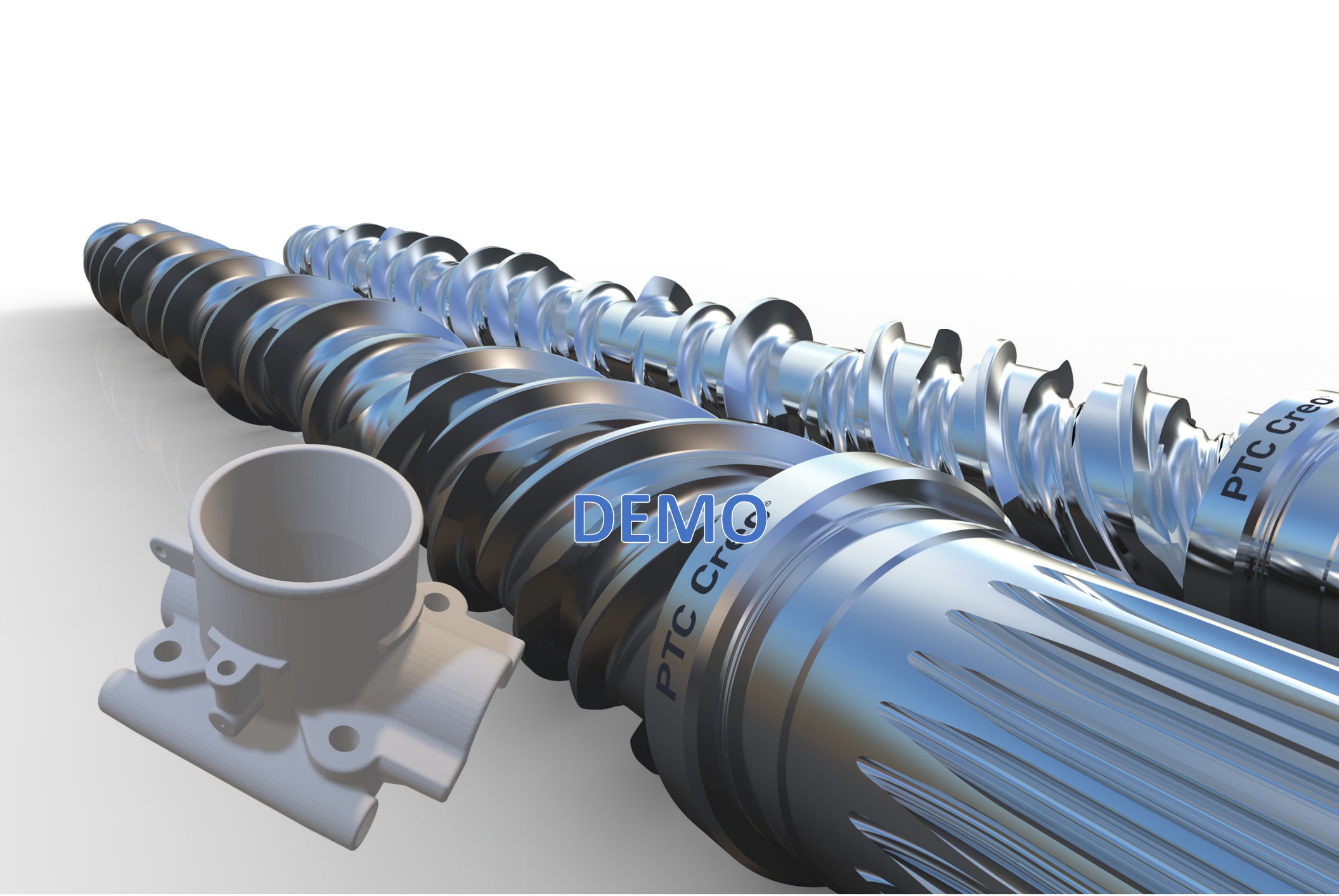



DEMO

Lite praktiska verktyg att känna till

CREO|SON

Welcome to OpenSource Automation for PTC's CREO Parametric!

This is an Open Source Initiative by Simplified Logic, Inc.

SF SIMPLIFY

MECHANICAL DESIGN AUTOMATION

usg engineering professionals | PLM & CAD Consultancy

uTools

sPurge
sBatch
sPlotdate
sFlush

Brought to you by PLM & CAD Consultancy http://www.usgengineering.nl version 5.0

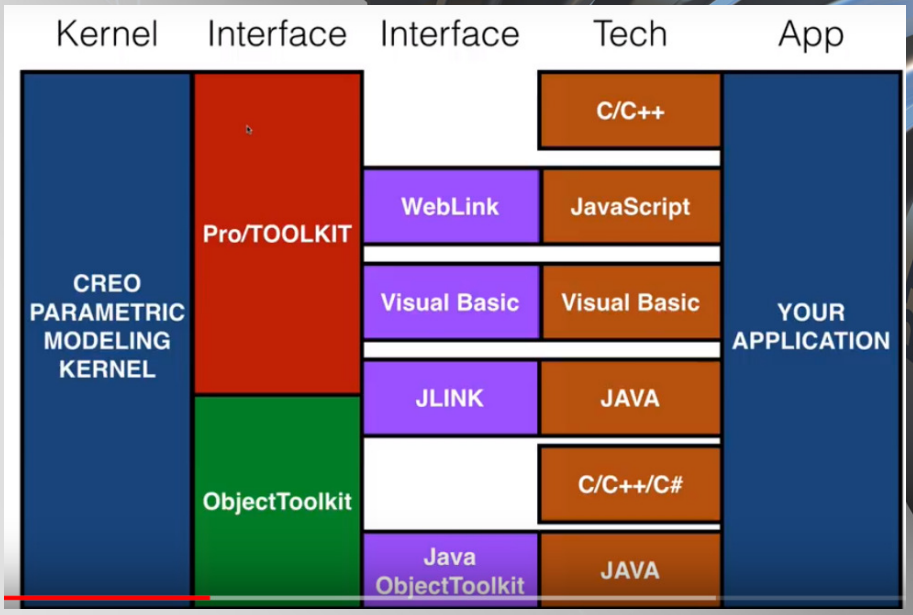
Nitro-PROGRAM

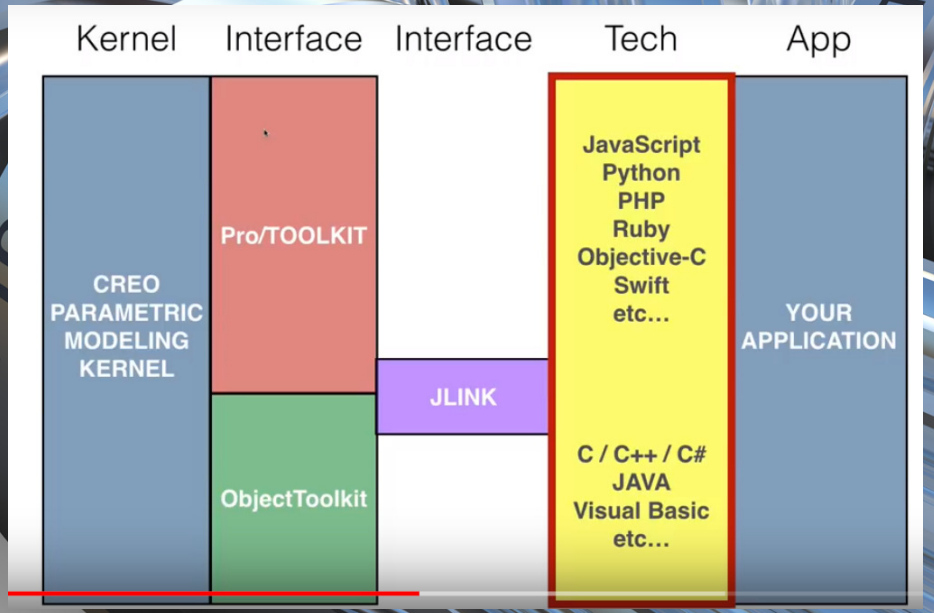
Graphical Pro/PROGRAM Editor!

Designed for unparalleled ease-of-use for drag-and-drop program editing within Parts and Assemblies.
THIS is a MUST-HAVE Tool for ANY serious CREO Automations!

CREO|SON

- Nytt gratis API utvecklat av Simplified Logic Inc.
- Fungerar med alla programmeringsspråk via JSON anrop

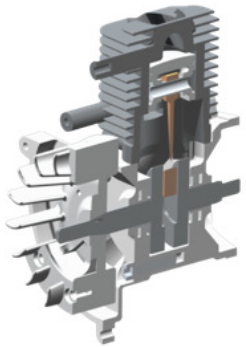




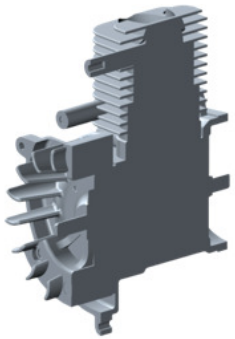
Software Factory

SF SIMPLIFY

MECHANICAL DESIGN AUTOMATION



unbearbeitetes Modell

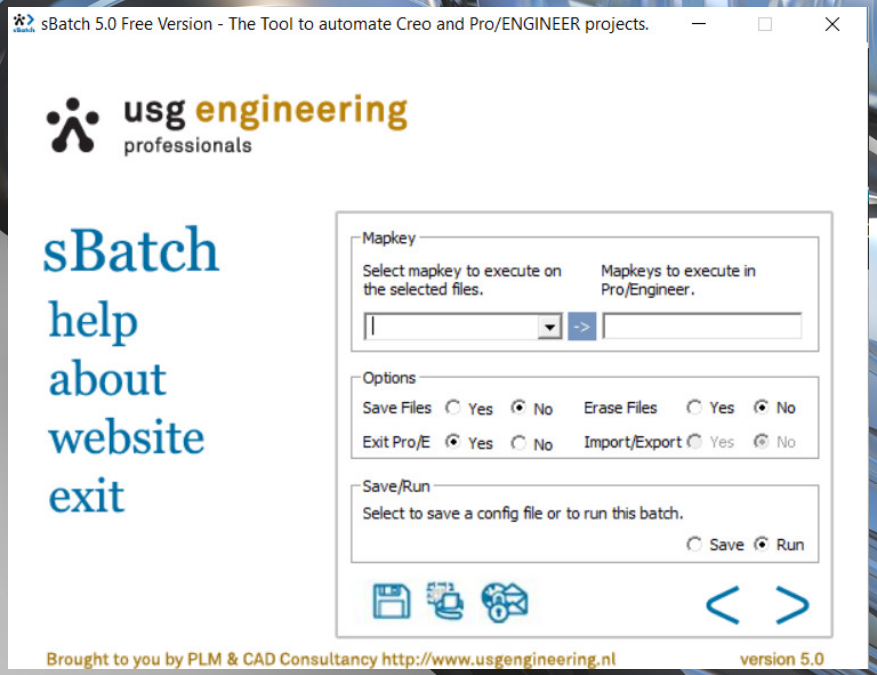


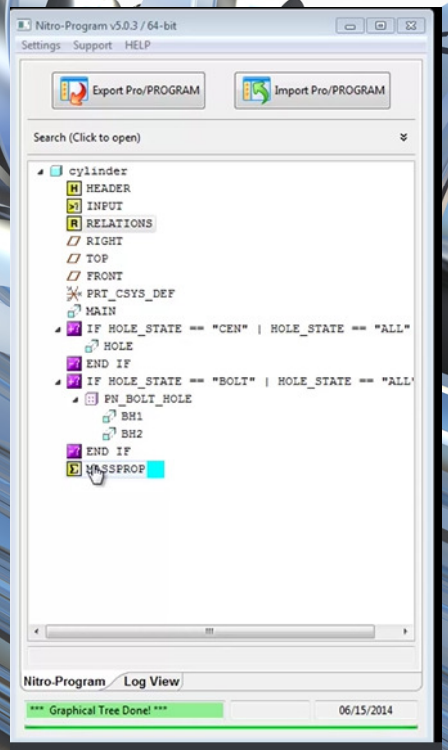
Modell nach **SF** SIMPLIFY

Nitro-PROGRAM

Graphical Pro/PROGRAM Editor!

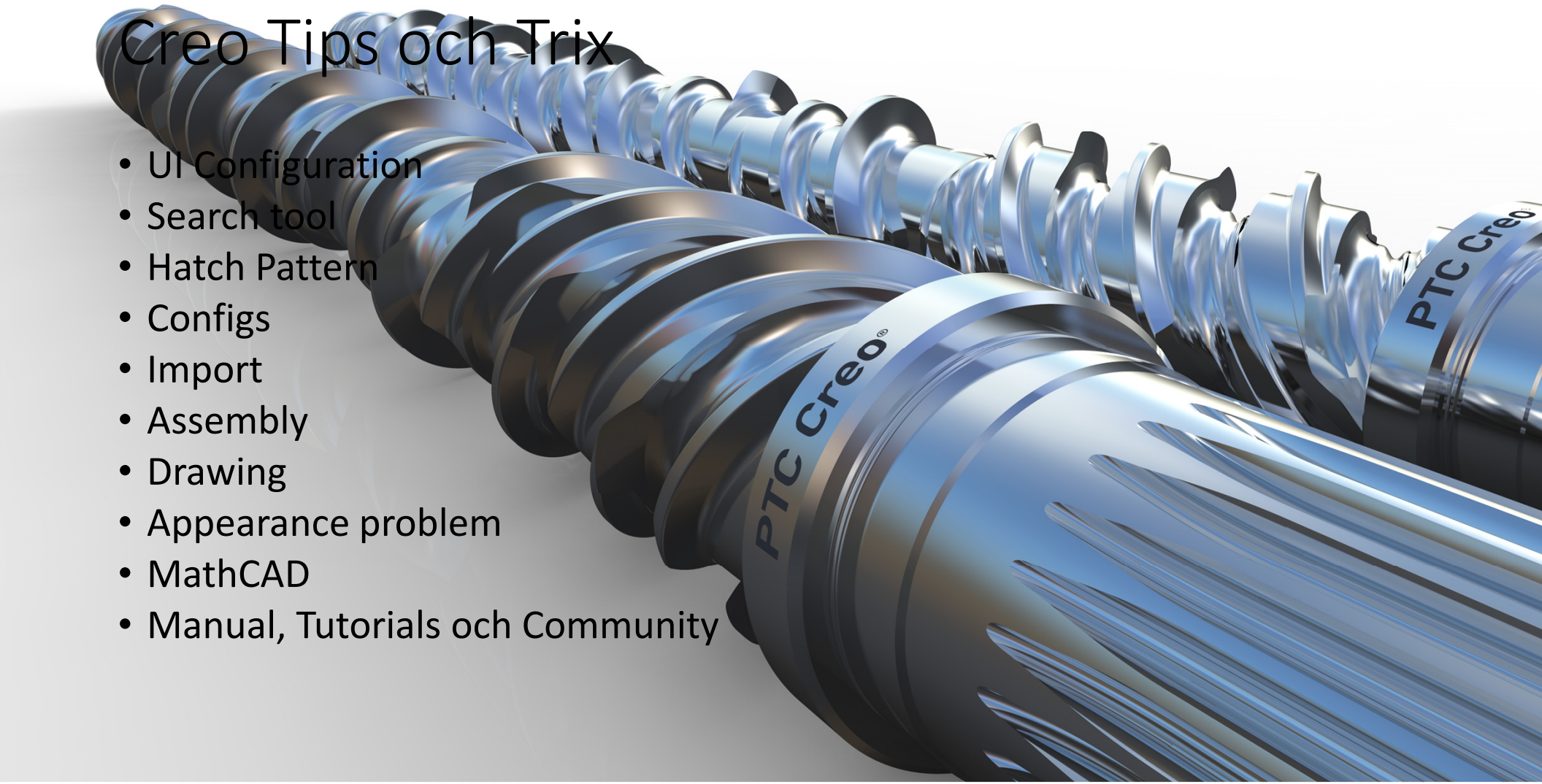
Designed for unparalleled ease-of-use for drag-and-drop program editing within Parts and Assemblies.
THIS is a MUST-HAVE Tool for ANY serious CREO Automations!





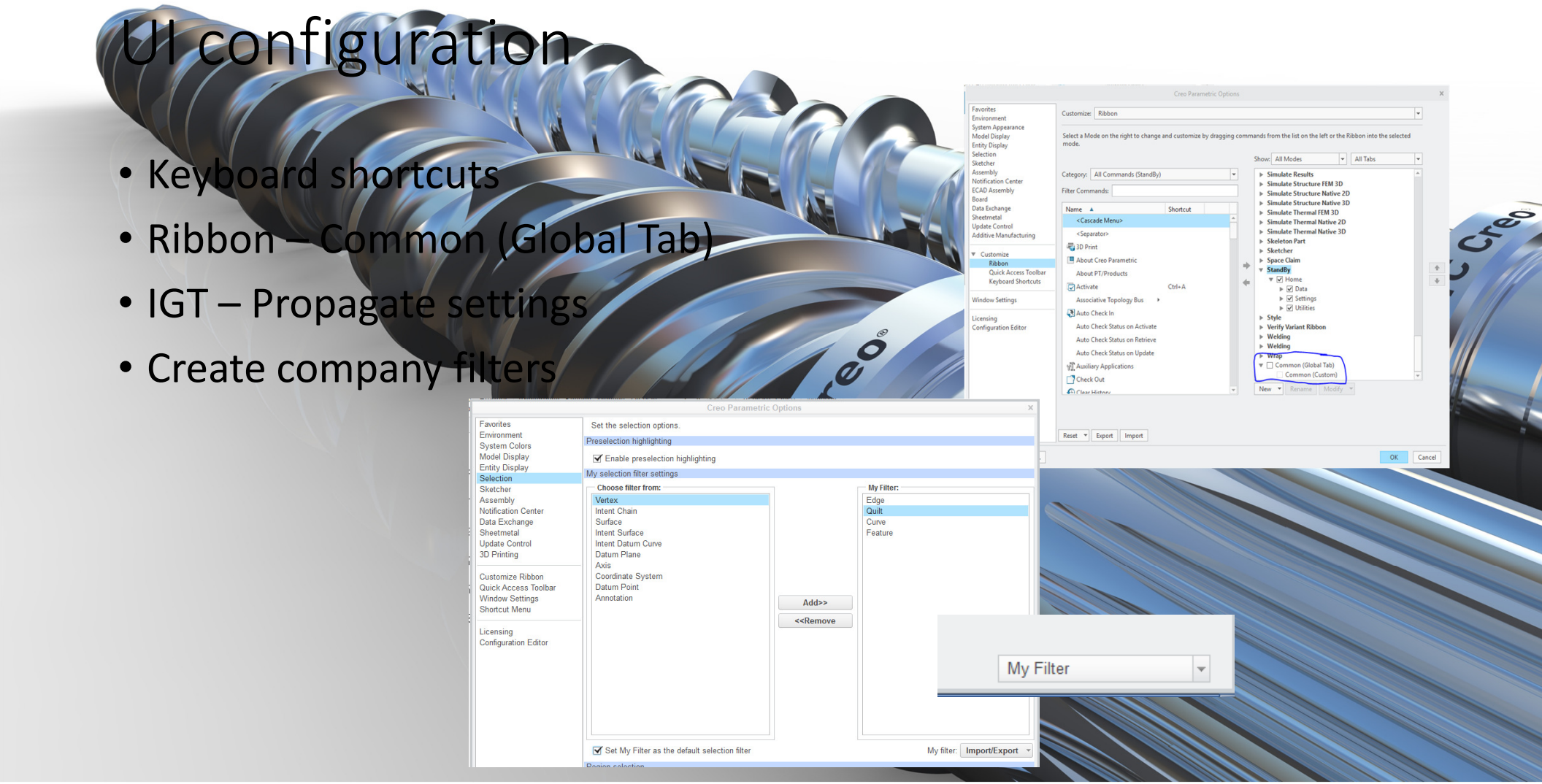
Creo Tips och Trix

- UI Configuration
- Search tool
- Hatch Pattern
- Configs
- Import
- Assembly
- Drawing
- Appearance problem
- MathCAD
- Manual, Tutorials och Community



UI configuration

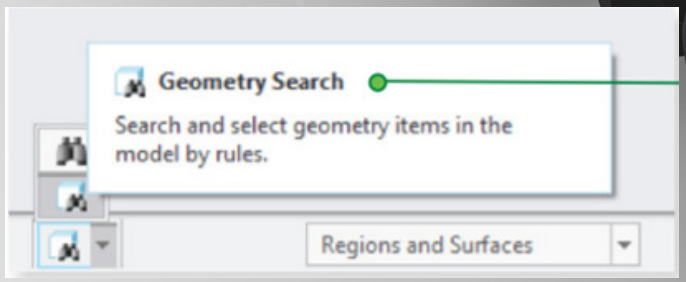
- Keyboard shortcuts
- Ribbon – Common (Global Tab)
- IGT – Propagate settings
- Create company filters

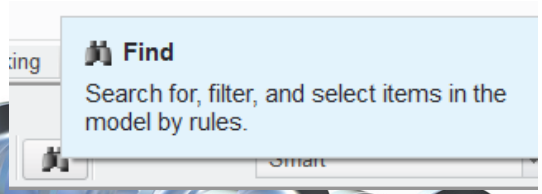


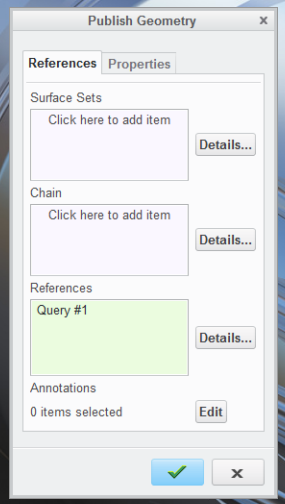
Search Tool

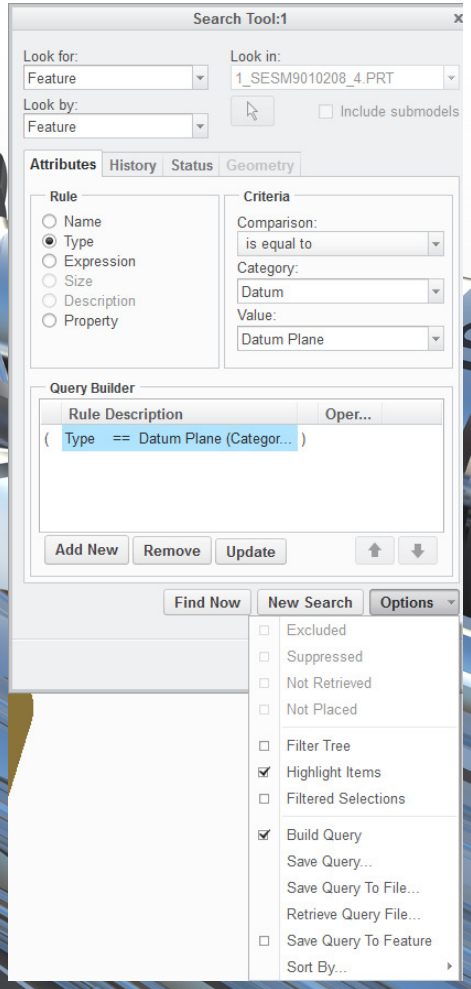
- Build Query – Build an advanced serch criteria
- Save Query – Save a search definition as a layer
- Save Query to File – Save a search definition as a qry file on disk
- Retreive Query File – Retreive a qry file from disk
- Save Query to Feature – Use a search definition in Publish Geometry or Copy Geometry

- Geometry Search – in Flexible Modeling



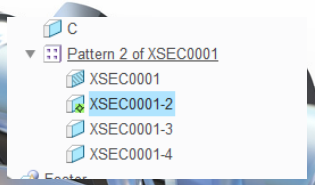


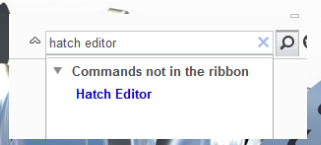


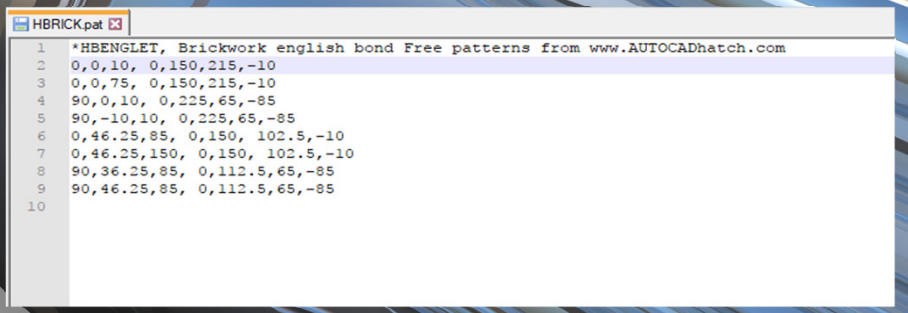


Hatch pattern

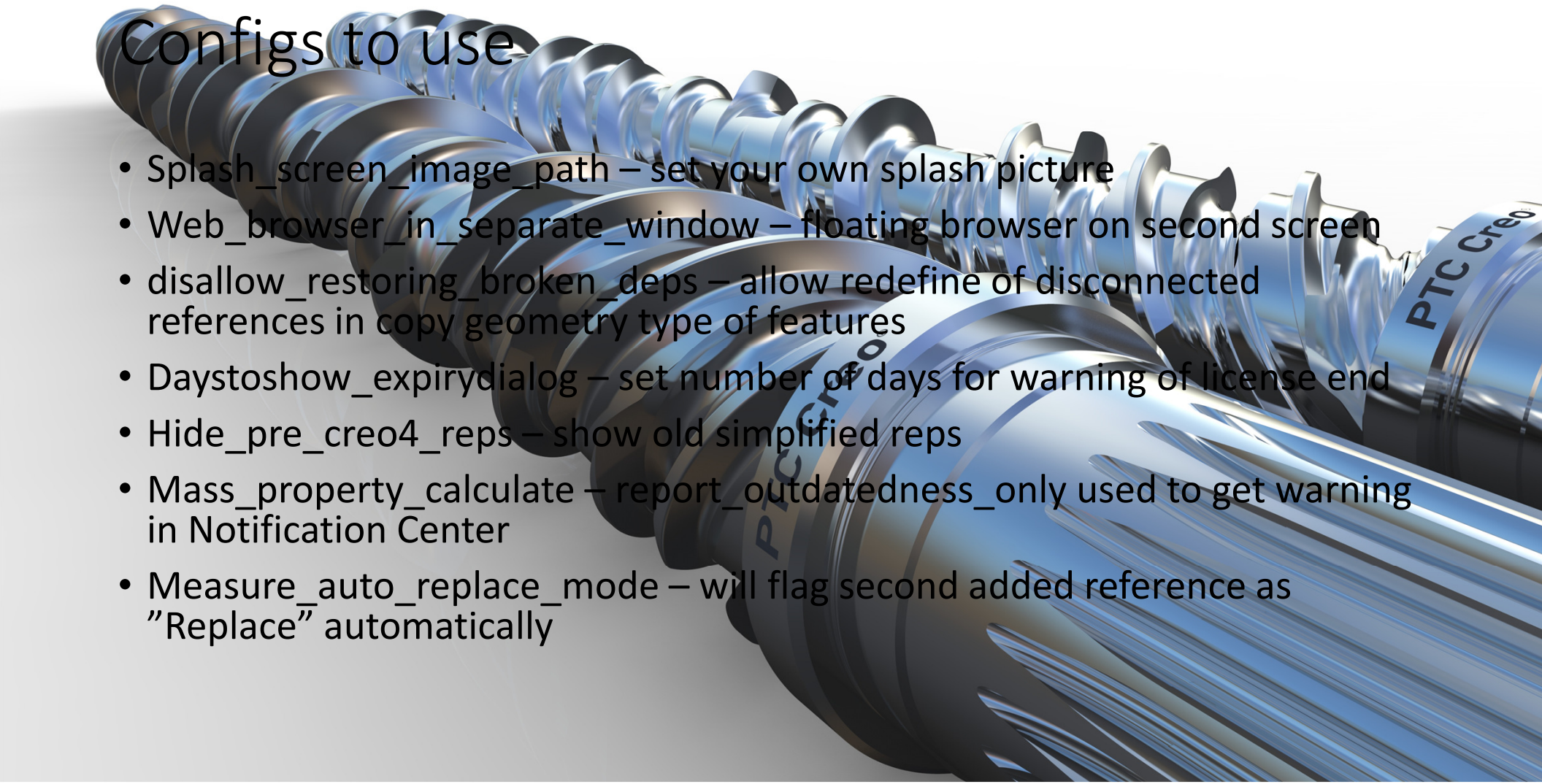
- Pattern of Cross sections
- Hatch Editor – Used to create and manage hatch patterns
- Linear and non-linear hatch patterns (.pat) – text hatch files can be found on net. Easy to create your own.
- Saving hatch pattern scale – hatch_pattern_cfg_file used to save scale for next use
- Config Default_hatch_type







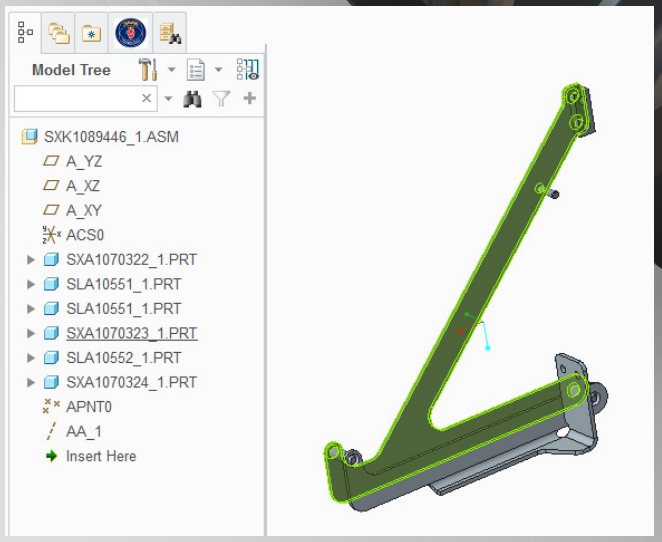
Configs to use

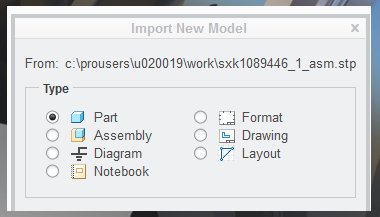


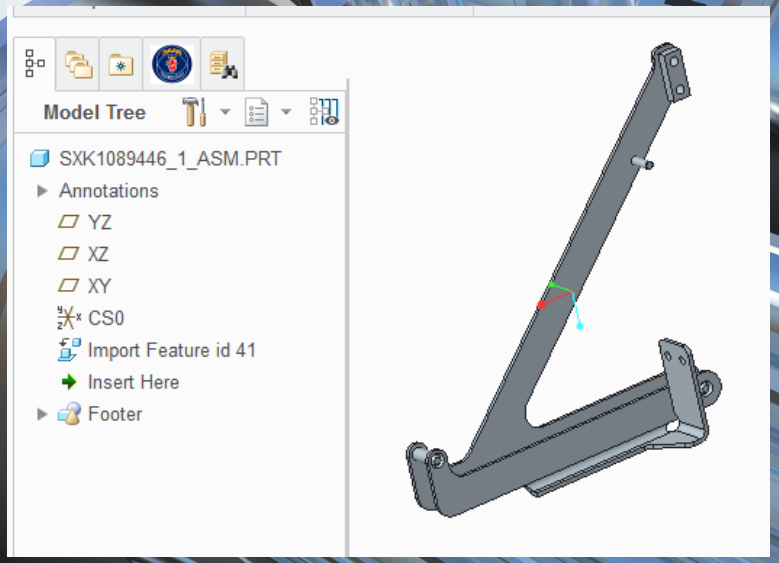
- Splash_screen_image_path – set your own splash picture
- Web_browser_in_separate_window – floating browser on second screen
- disallow_restoring_broken_deps – allow redefine of disconnected references in copy geometry type of features
- Daystoshow_expirydialog – set number of days for warning of license end
- Hide_pre_creo4_reps – show old simplified reps
- Mass_property_calculate – report_outdatedness_only used to get warning in Notification Center
- Measure_auto_replace_mode – will flag second added reference as ”Replace” automatically

Import config

- Config.pro option = intf3d_in_as_part yes
- Used to enable import of neutral assembly data as part with geometry in right position

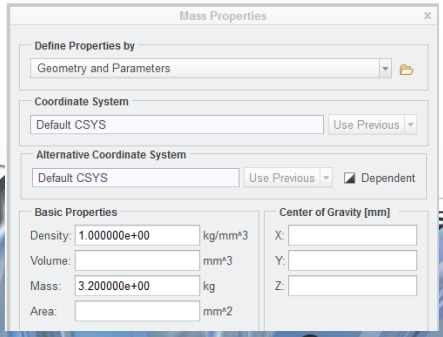


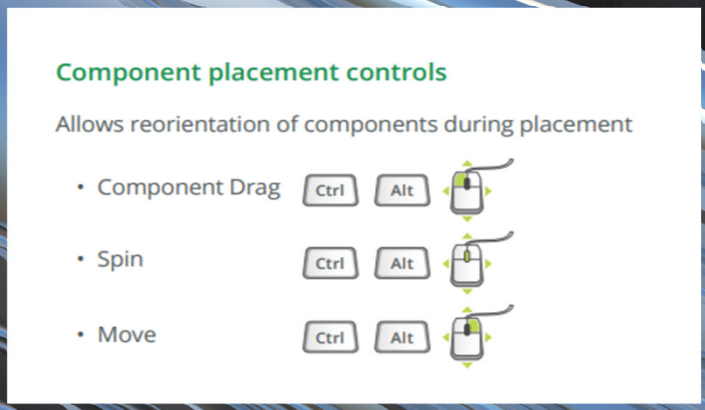




Assembly

- Assign Mass – Add mass data if known but not calculated
- Assembly cut – disable ”Automatic Update”. This will decrease memory usage and regeneration time.
- Set config comp_assemble_start move_then_place
- Set config package_constraints disallow
- Show ”Placement Folder” in model tree
- Use CNTR+ALT to avoid unwanted constraints





Assembly constraints

How to change default constraint type

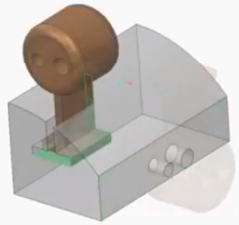
- Set config option auto_constr_always_use_offset - Never
  - Coincident will be the default constraint type when reference pairs are planes, linear edges or datum axes or planes to edge pairing
- Set config option auto_constr_always_use_offset - Yes
  - Offset (Angle or Normal) will be the default constraint type (never coincident) when reference pairs are planes, linear edges or datum axes, planes to edge pairing
- Set config option auto_constr_always_use_offset - No
  - System will pick up constraint type (coincident / distance / angle / normal) based on position and orientation of component

How to change default constraint type

- When auto_constr_always_use_offset is No, the following config options determines angle or normal constraint type using the angle epsilon
  - Comp_angle_offset_eps (-1)
  - Comp_normal_offset_eps (-91)
- If the component position does not fit angle or normal, then it will be either coincident or distance. System decides based on this config option
  - Auto_constr_offset_tolerance (0.5 times model size). If current distance between references is bigger than this value, then it will set the constraint type to distance. If less, then it will set to coincident

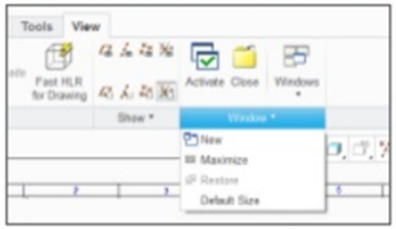
Constraint Sets in Creo

- Multiple constraint sets for components.
- Enable/Disable for alternate positions.
- New parameter – PTC_CONSTRAINT_SET can be used in Family table or in model tree column to switch positions.

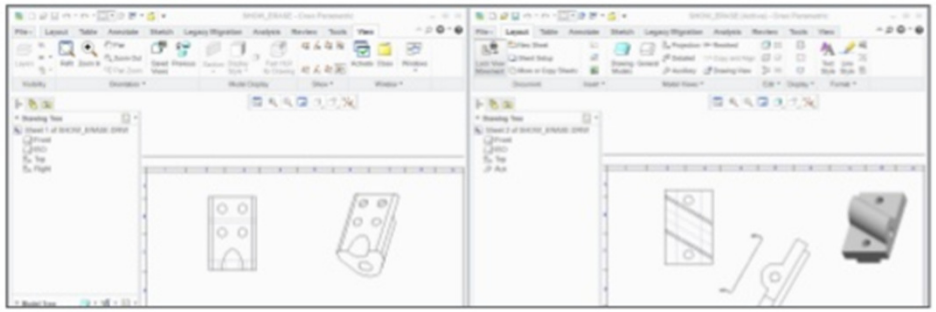


Drawing

- In a Drawing, select the Window overflow from the View tab and pick **New**. You will be prompted for a sheet number to navigate to.

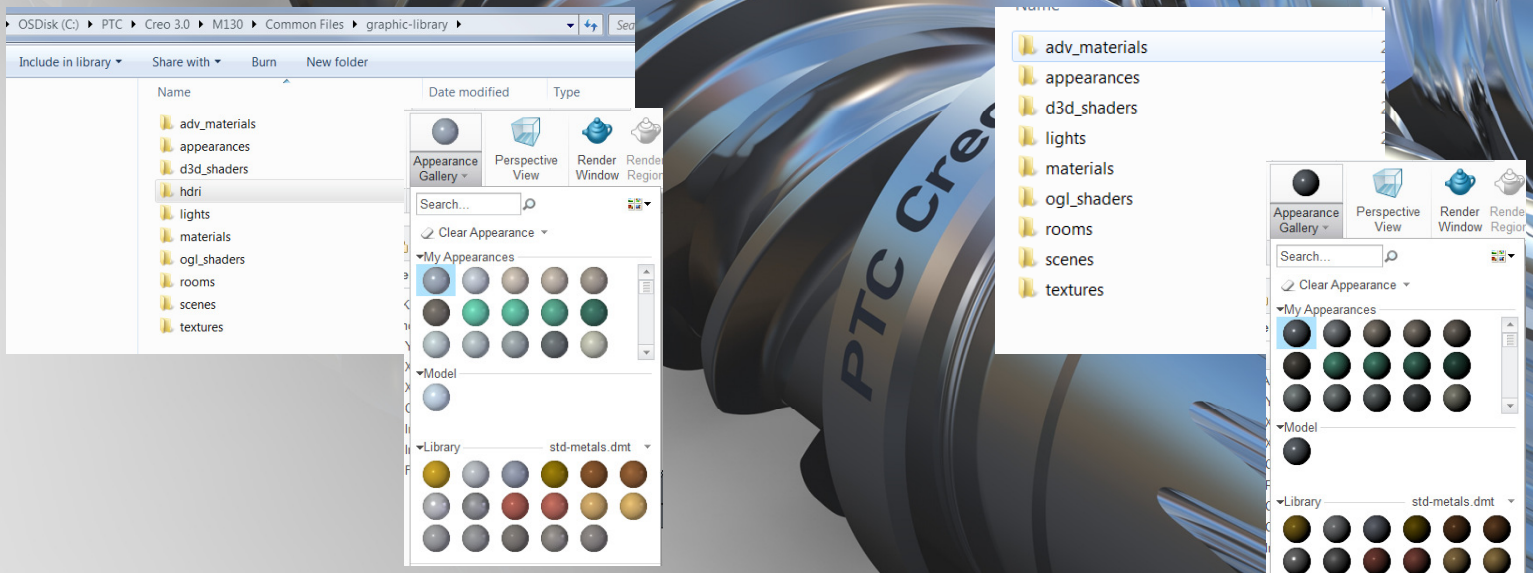


- You can now work in multiple drawing sheets simultaneously. (Also works well for large models where regular reorientation is time consuming).



Appearance gallery

- If you reuse the graphic library from earlier release make sure that you also include the HDRI folder



Print MBD to PDF

To Print a Model

1. Click **File** > **Print** > **Print**. The **Printer Configuration** dialog box opens.
2. On the **General** tab, Click to select a printer or to add a new printer. The selected printer type is saved for the current session only.
3. Specify the general printing options.

Command	Action
Destination	Select one of the following check boxes to choose the destination of the print: • **To File** – saves the prints of the model to the specified file • **To Printer** – prints the model using the selected printer
Combination States	Select one of the following: • **All** – Prints all combination states available in the model. • **Current** – Prints the active combination state. • **Range** – Prints a range of combination states. When you select **Range**, the **Define** option becomes available. Click **Define**, and select the combination states that you want to print from the **Define Range** dialog box
Format	Specify the format (*.frm) to be used as a background for printing. The **Format** drop down list lists the previously selected formats. Select a format from the drop down list or click **Browse** to select a different format.
Options	Specify the number of copies to be printed and set the plotter command to be used when sending a plot to the printer.

Simulate

To remove a feature within Creo Simulate:

1. Using Creo FMX, highlight the feature you'll remove.
2. On the **Refine Model** ribbon bar, click **Remove**.
3. **Preview** and/or **Accept** the change.

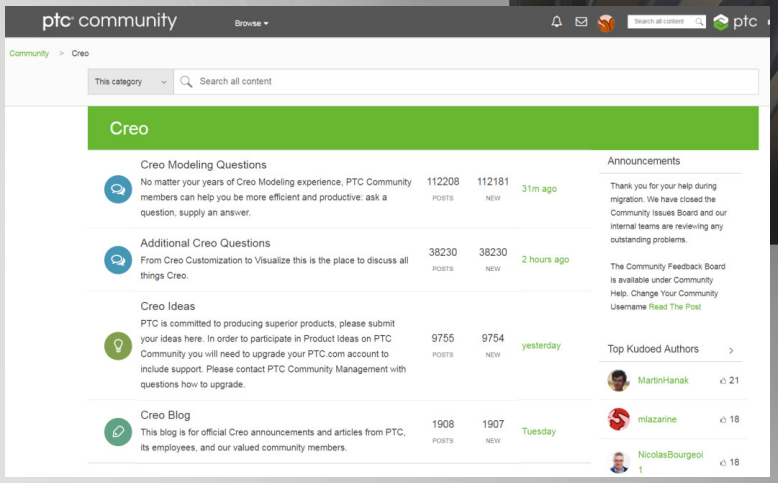
It's important to note that currently the change is only applied in Creo Simulate. You must promote changes made in Creo Simulate to apply them to the parametric model. To promote a change:

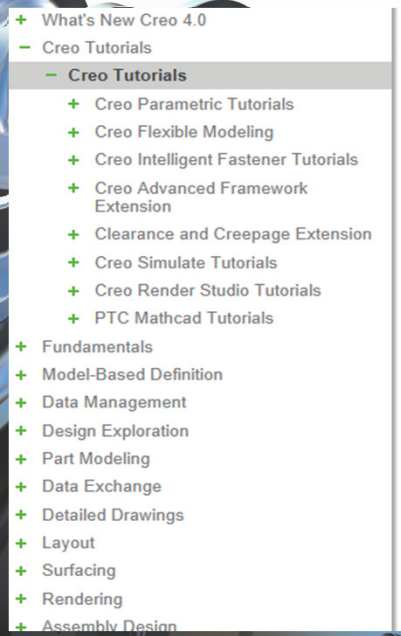
1. In the **Model Tree**, expand **Simulation Features**.
2. Right-click the removed feature and select **Promote**.

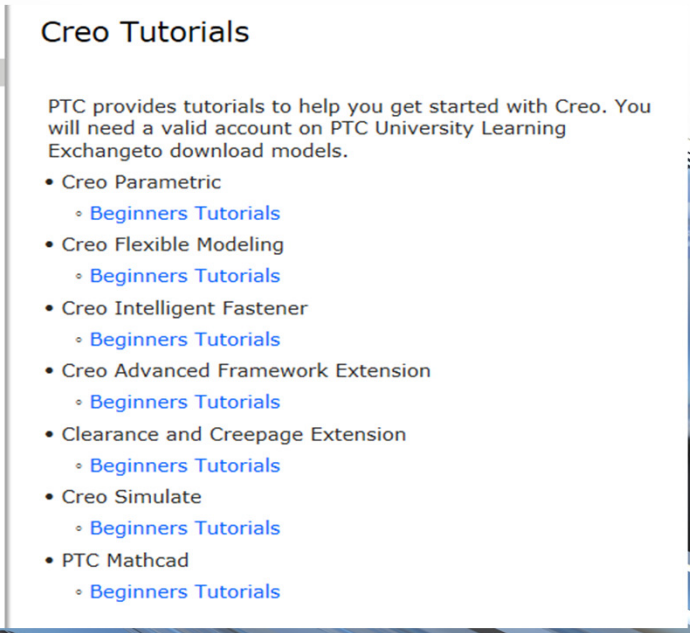
The system applies the change to the parametric model.

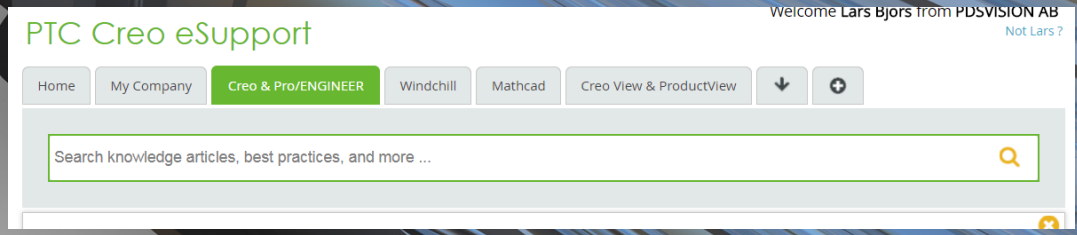
Learn more of Creo

- Read the manual
- Check Tutorials in manual
- Join PTC Community
- Seach the PTC Knowledge Base

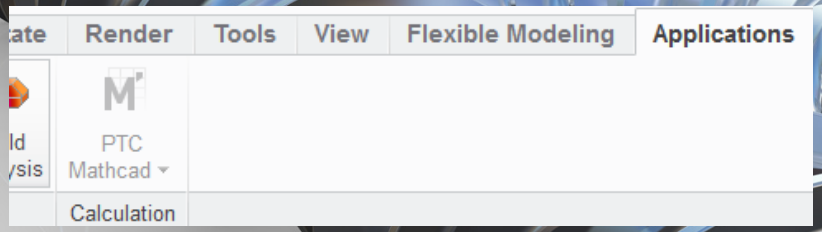


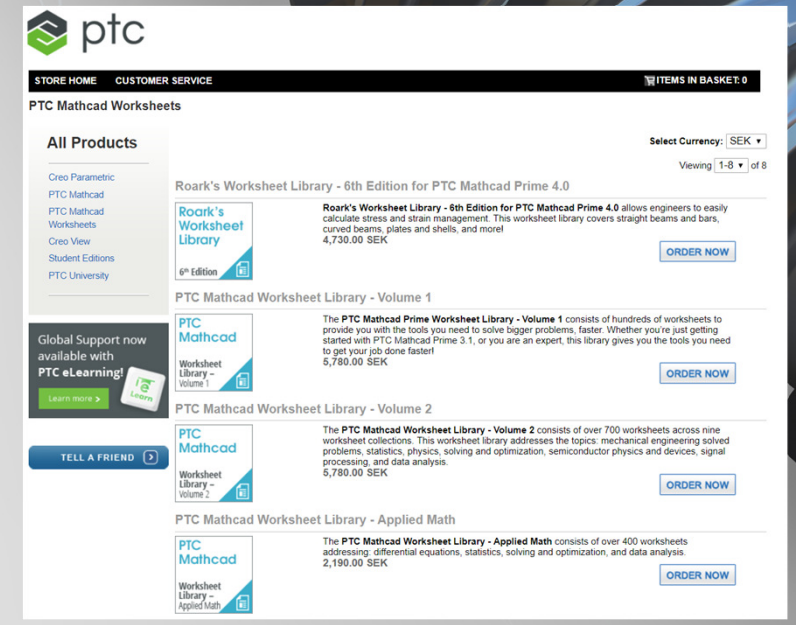


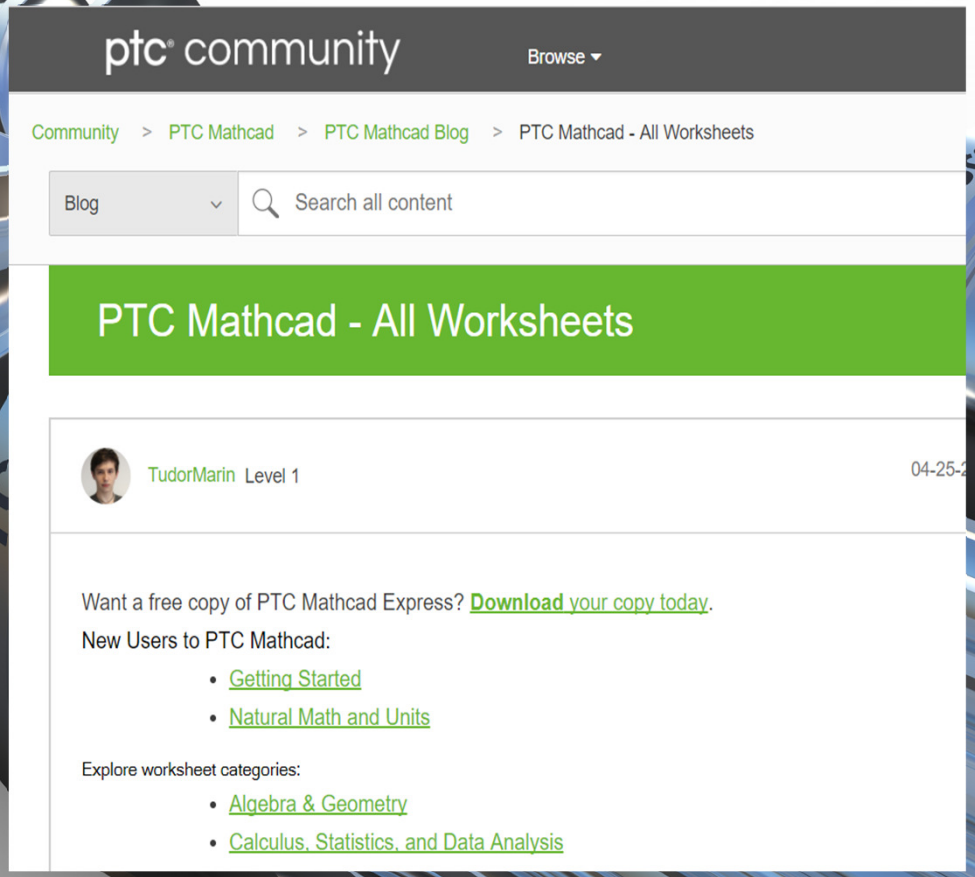




Mathcad integrated in Creo







Creo 5.0 is son on your desktop

4.0	F000	**December 15, 2016**
	M010	**March 29, 2017**
	M020	**June 27, 2017**
	M030	*September 27, 2017*
	M040	*December 20, 2017*
	M050	*March 19, 2018*
5.0	F000	*March 13, 2018*
6.0	F000	*March, 2019*
7.0	F000	*March, 2020*
	F000	**March 27, 2012**

Technical Committees : CAD

No Picture Available

CFD simulation for Creo

Created By: Jose Coronado

Thursday Aug 31, 2017 05:55 PM

Overview

This is a session to provide training on CFD solution fully embedded in creo – powered by simerics. And to receive feedback

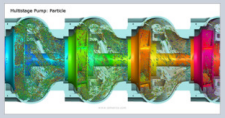
File Options

CREO 5.0: NEW PRODUCT INTRODUCTION

ptc

CFD

Computational fluid dynamics is the use of applied mathematics, physics and computational software to visualize how a gas and liquid flows



Topology Optimization

A mathematical way of analysing a model to improve its overall performance, reducing costs associated with weight/mass while maintaining the structural needs of the design.

3D Printing: Metal

Enable Creo to produce the geometry needed for metal printing (support structures) and to connect it with the metal 3D printer itself

Mold Machining

Functionality to create specialised NC toolpaths, specific for removing material to create molds