



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

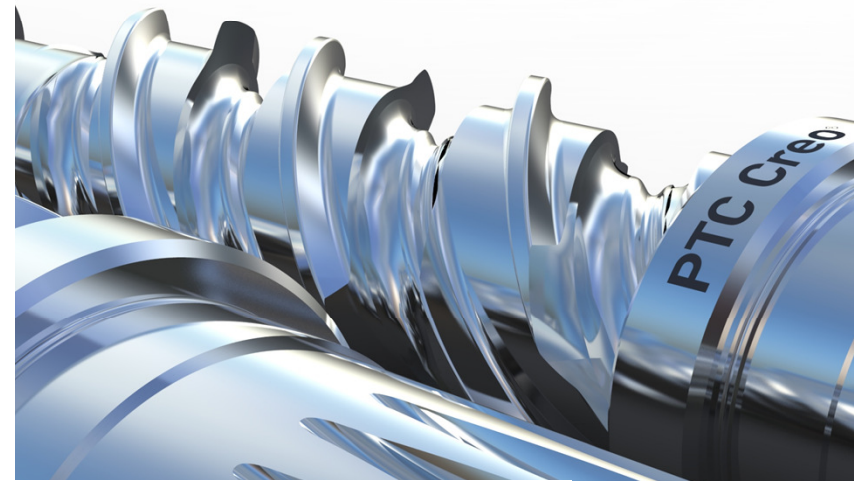
- Run ModelCHECK for Creo Elements/Direct Modeling *.sdac and *.sdpc file using Open option
 - CS248781
- Publish Augmented Reality (AR) experience directly from Creo
 - CS257684
- New solution Clearance and Creepage Extension (CCX) added into Creo Parametric 4.0 M010
 - CS260097

Creo Parametric 4.0 M020

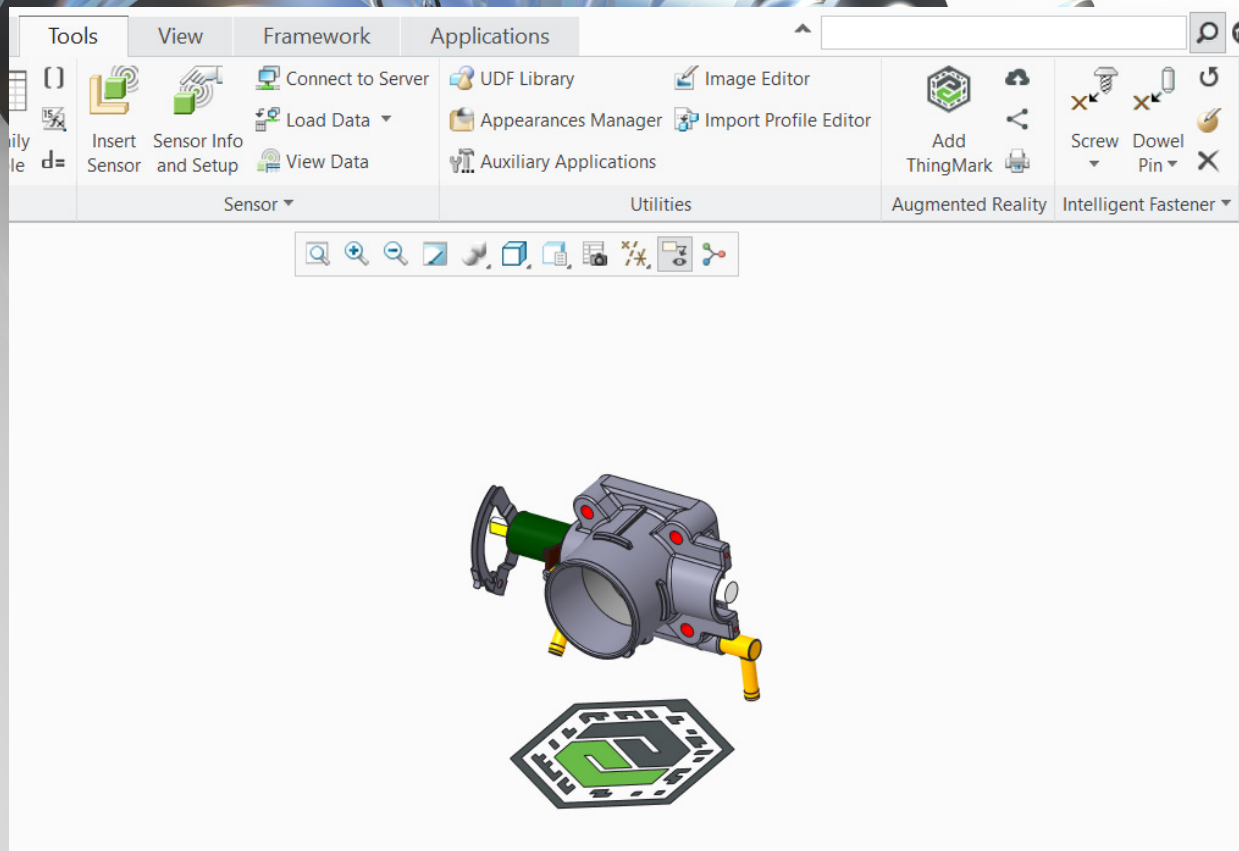
- STEP AP242 supports Semantic PMI (Product Manufacturing Information) on Import and Export
 - In previous releases and datecodes, only Graphical PMI has been supported, refer to CS193762
- Smart connected product design will be launched as **Creo Product Insight Extension**
 - The license option for **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](http://learningexchange.ptc.com) 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

Clearance and Creepage

Menu

Net Name	Net Type
Net 1	Potential
Net 2	Potential
Net 3	Grounded
Net 4	Potential-free
Net 5	Potential-free
Net 6	Potential-free

Merge Nets Reset Nets

Auto merge distance: 0.00 mm Auto Merge Nets

Source: Net 1

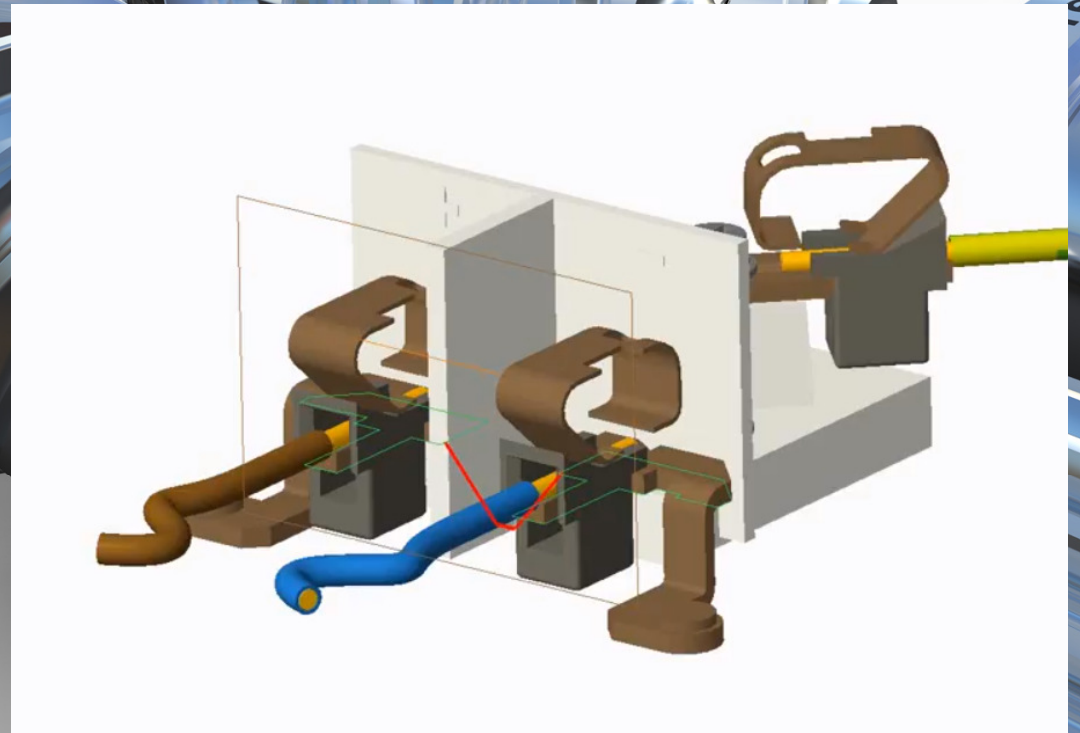
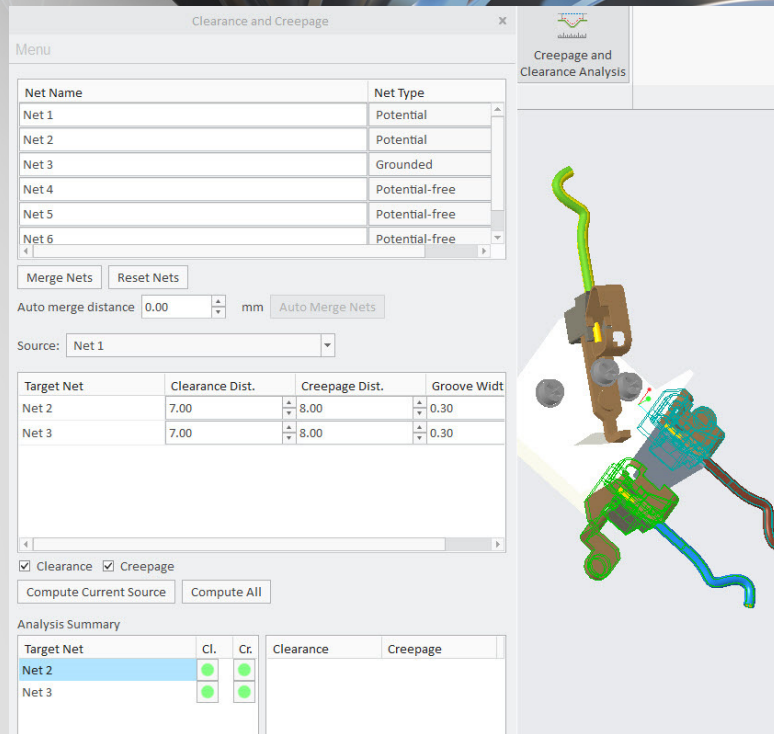
Target Net	Clearance Dist.	Creepage Dist.	Groove Width
Net 2	7.00	8.00	0.30
Net 3	7.00	8.00	0.30

Clearance Creepage

Compute Current Source Compute All

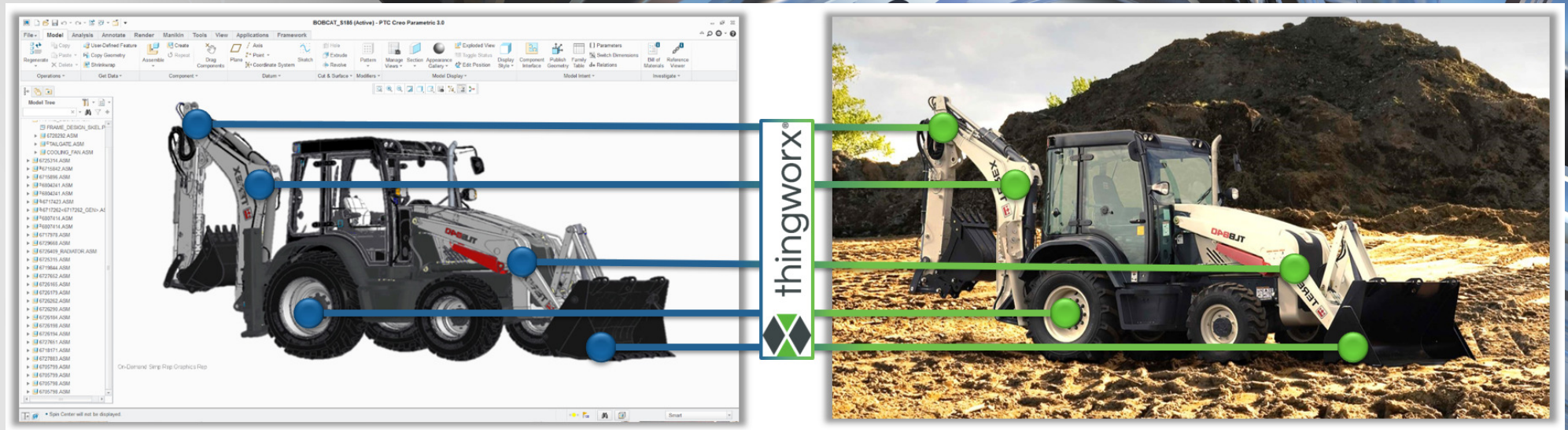
Analysis Summary

Target Net	Cl.	Cr.	Clearance	Creepage
Net 2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		
Net 3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		



Creo Product Insight Extension

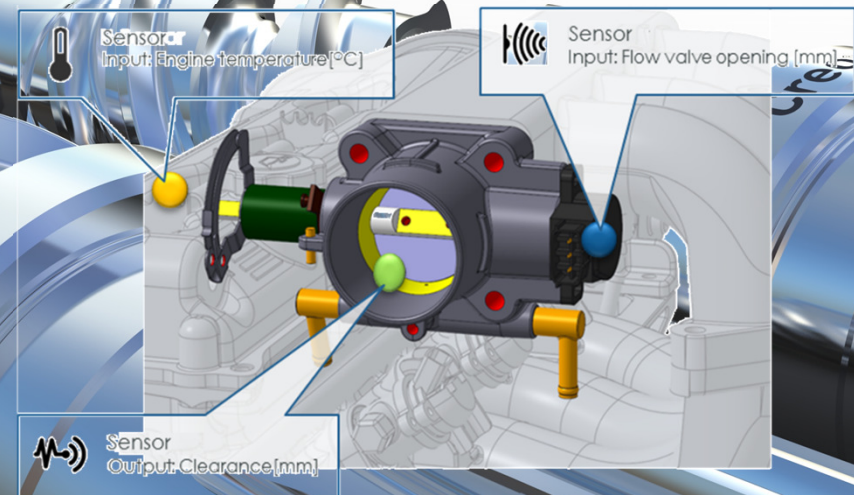
Creo Product Insight extension supports smart, connected product design allowing companies to take advantage of the IoT and **replace design assumptions with facts**



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



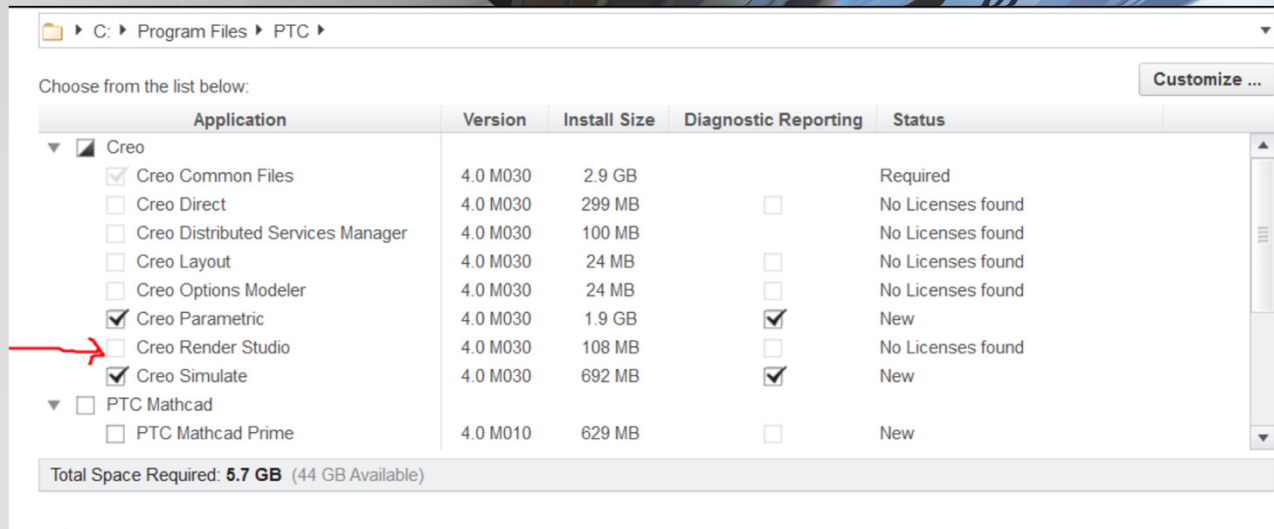
CREO Render Studio

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

Creo Parametric 4.0 M030

• Creo Render Studio

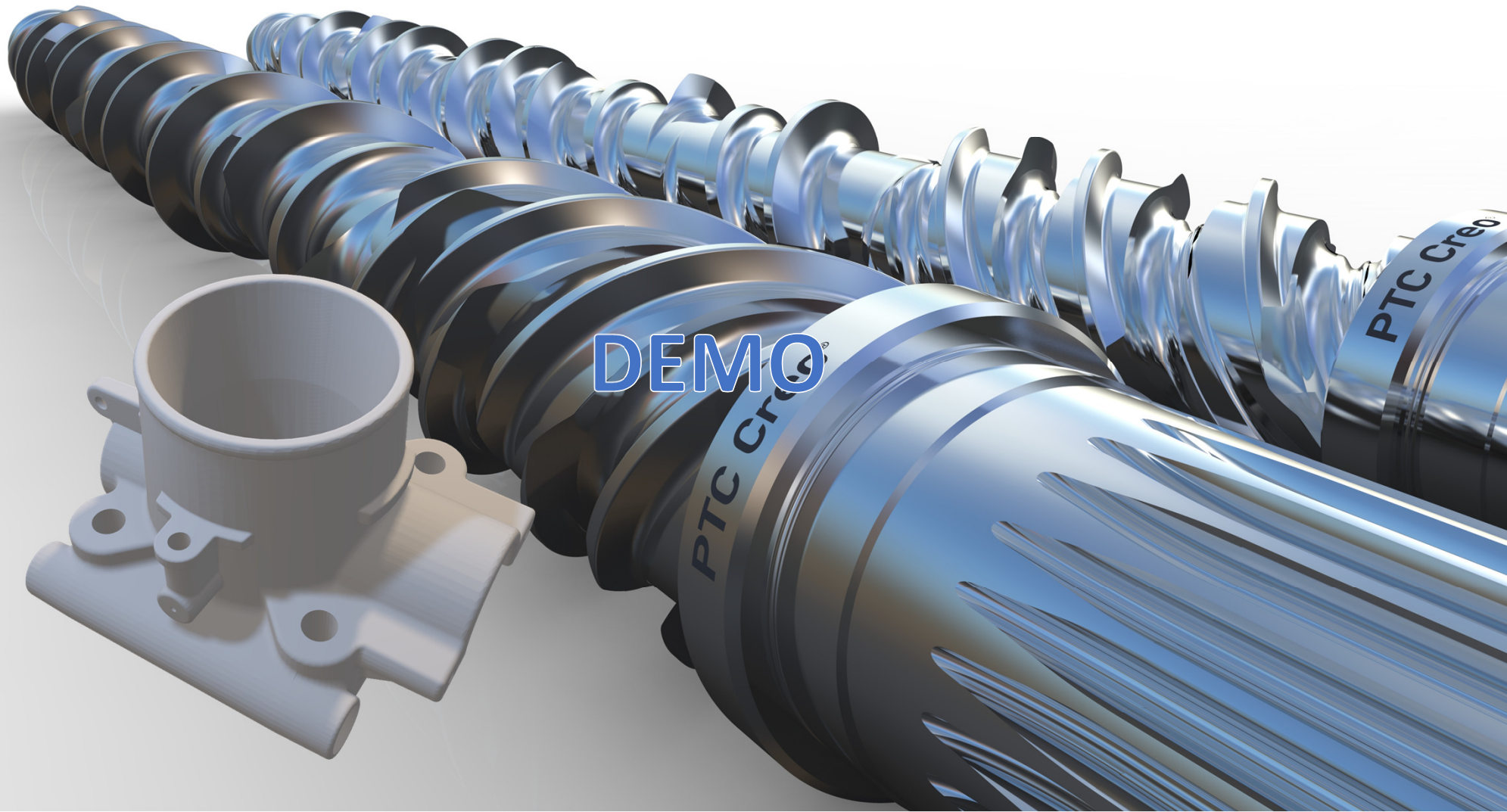
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Choose from the list below:

Application	Version	Install Size	Diagnostic Reporting	Status
Creo				
<input checked="" type="checkbox"/> Creo Common Files	4.0 M030	2.9 GB		Required
<input type="checkbox"/> Creo Direct	4.0 M030	299 MB	<input type="checkbox"/>	No Licenses found
<input type="checkbox"/> Creo Distributed Services Manager	4.0 M030	100 MB	<input type="checkbox"/>	No Licenses found
<input type="checkbox"/> Creo Layout	4.0 M030	24 MB	<input type="checkbox"/>	No Licenses found
<input type="checkbox"/> Creo Options Modeler	4.0 M030	24 MB	<input type="checkbox"/>	No Licenses found
<input checked="" type="checkbox"/> Creo Parametric	4.0 M030	1.9 GB	<input checked="" type="checkbox"/>	New
<input type="checkbox"/> Creo Render Studio	4.0 M030	108 MB	<input type="checkbox"/>	No Licenses found
<input checked="" type="checkbox"/> Creo Simulate	4.0 M030	692 MB	<input checked="" type="checkbox"/>	New
PTC Mathcad				
<input type="checkbox"/> PTC Mathcad Prime	4.0 M010	629 MB	<input type="checkbox"/>	New

Total Space Required: **5.7 GB** (44 GB Available)



DEMO

PTC Creo

PTC Creo

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

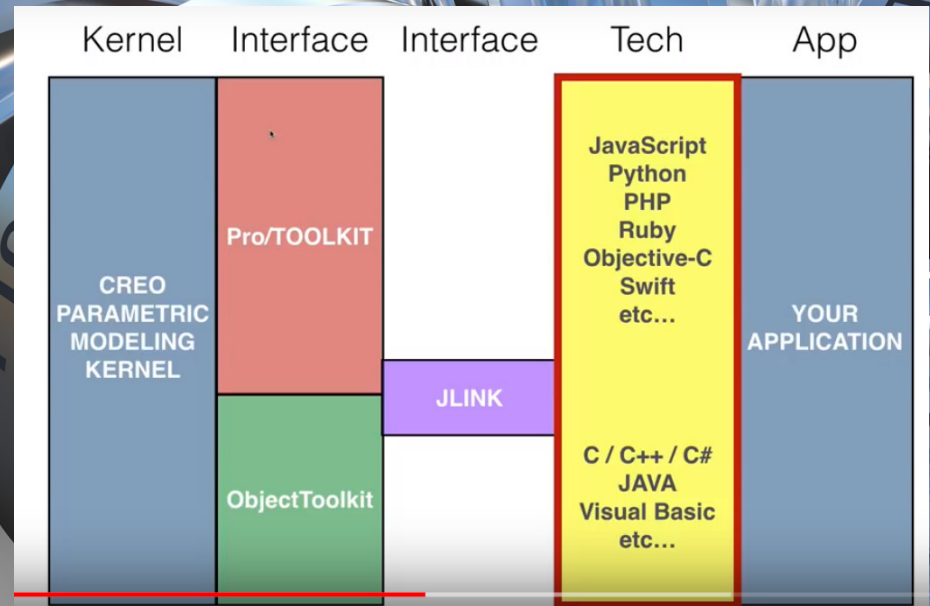
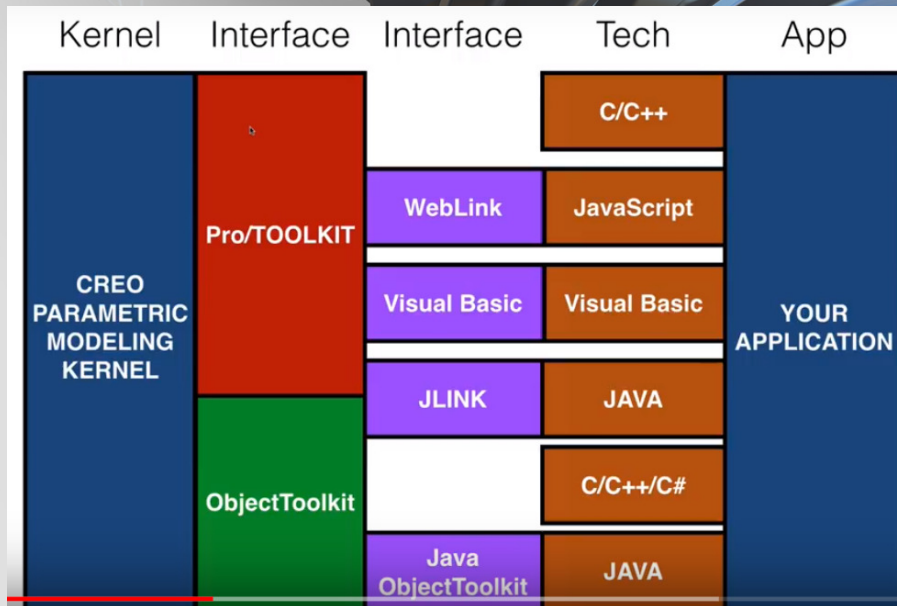
Graphical Pro/PROGRAM Editor!

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!

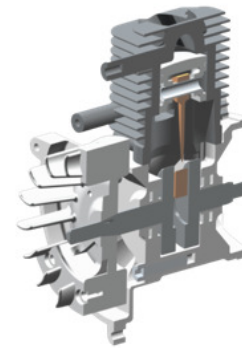
CREO | SON

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



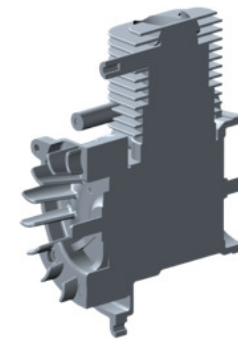
Software Factory

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!

 | PLM & CAD Consultancy


uTools

- sPurge
- sBatch
- sPlotdate
- sFlush



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

sBatch 5.0 Free Version - The Tool to automate Creo and Pro/ENGINEER projects.



sBatch
help
about
website
exit

Mapkey

Select mapkey to execute on the selected files. Mapkeys to execute in Pro/Engineer.

Options






Save Files Yes No Erase Files Yes No

Exit Pro/E Yes No Import/Export Yes No

Save/Run

Select to save a config file or to run this batch.

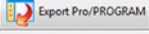
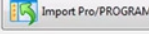
Save Run

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

Nitro-Program v5.0.3 / 64-bit

Settings Support HELP

Search (Click to open)

- cylinder
 - HEADER
 - INPUT
 - RELATIONS
 - RIGHT
 - TOP
 - FRONT
 - PRT_CSYS_DEF
 - MAIN
 - IF HOLE_STATE == "CEN" | HOLE_STATE == "ALL"
 - HOLE
 - END IF
 - IF HOLE_STATE == "BOLT" | HOLE_STATE == "ALL"
 - FN_BOLT_HOLE
 - BH1
 - BH2
 - END IF
 - MASSPROP

Nitro-Program / Log View

*** Graphical Tree Done! *** 06/15/2014

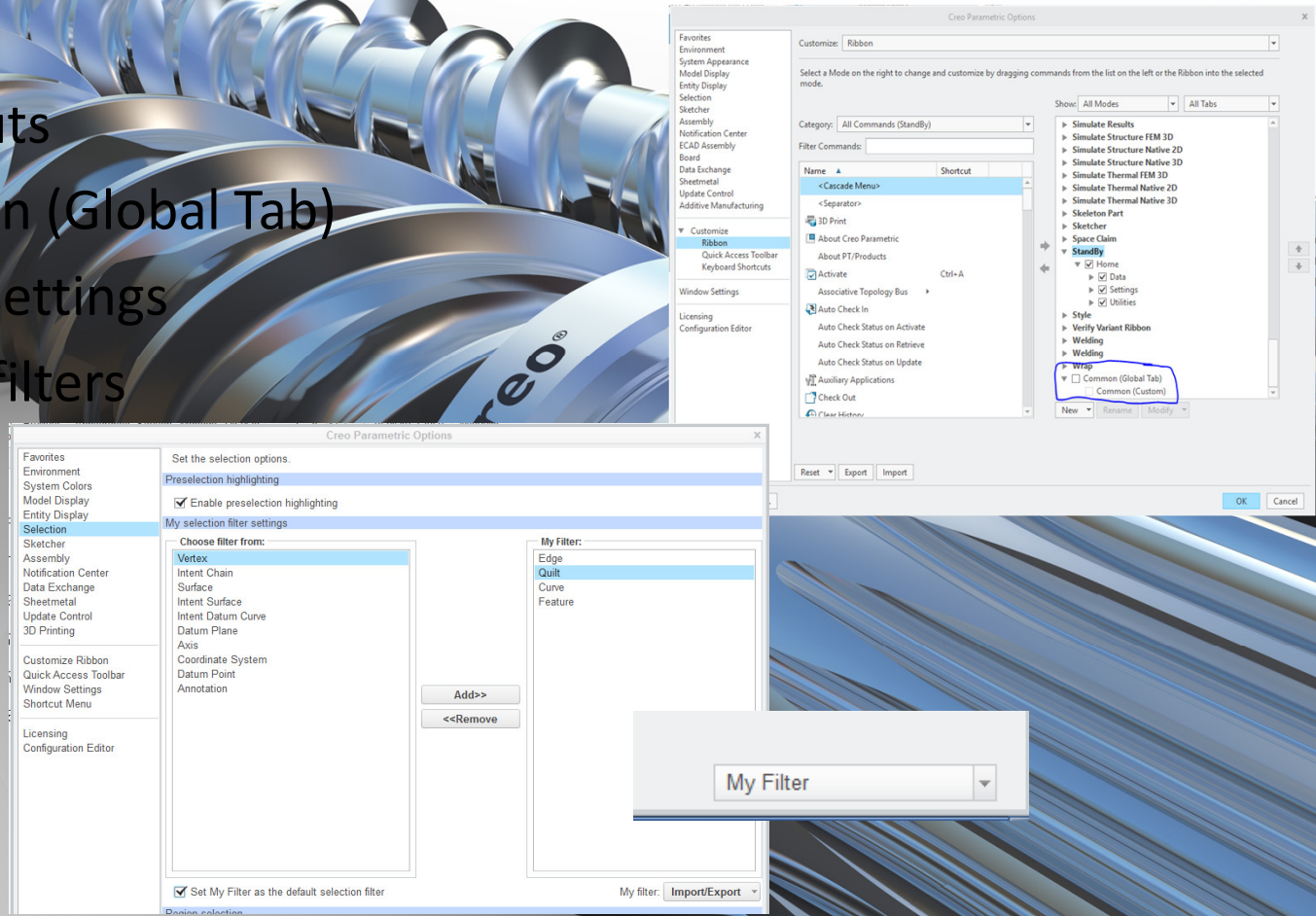
Creo Tips och Trix



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

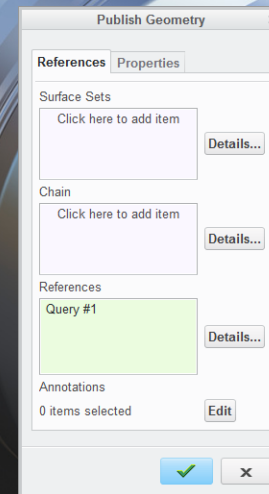
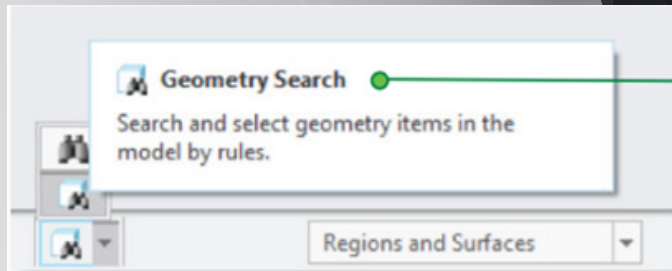
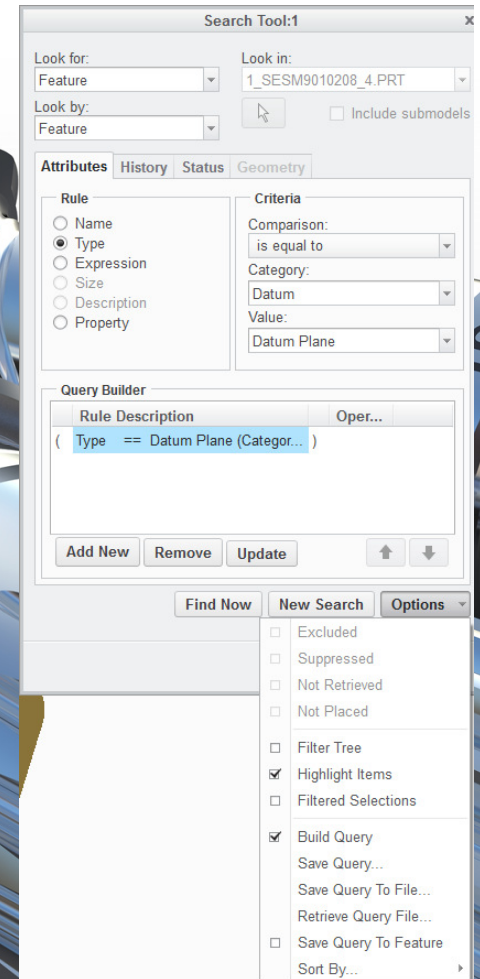
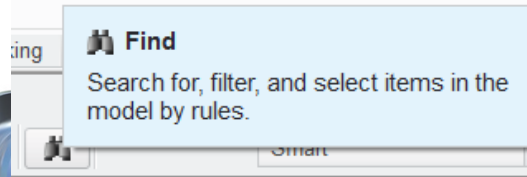
UI configuration

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



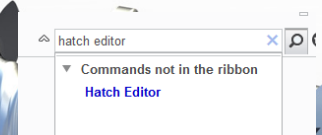
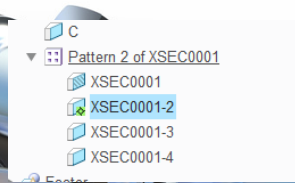
Search Tool

- Build Query – Build an advanced search criteria
- Save Query – Save a search definition as a layer
- Save Query to File – Save a search definition as a qry file on disk
- Retrieve Query File – Retrieve 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



```
HBRICK.pat
1 *HBENGLET, Brickwork english bond Free patterns from www.AUTOCADhatch.com
2 0,0,10, 0,150,215,-10
3 0,0,75, 0,150,215,-10
4 90,0,10, 0,225,65,-85
5 90,-10,10, 0,225,65,-85
6 0,46.25,85, 0,150,102.5,-10
7 0,46.25,150, 0,150,102.5,-10
8 90,36.25,85, 0,112.5,65,-85
9 90,46.25,85, 0,112.5,65,-85
10
```

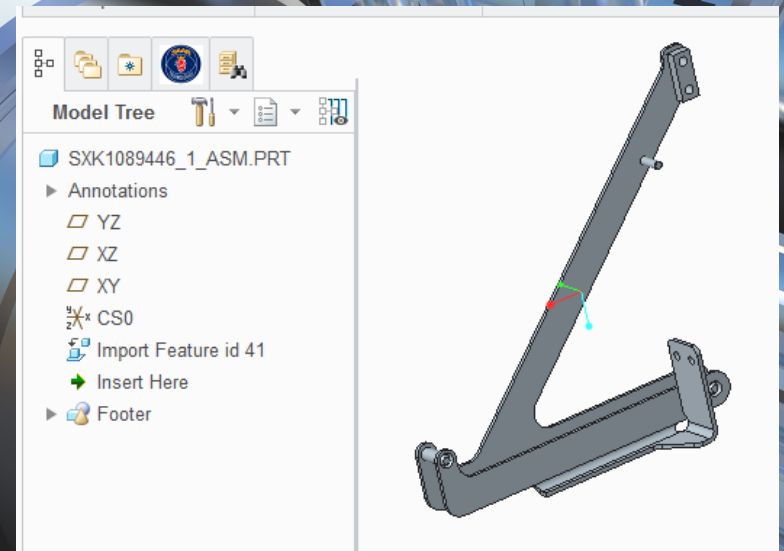
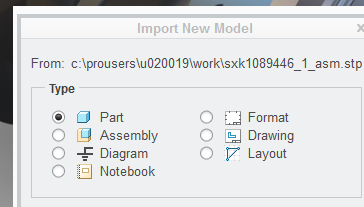
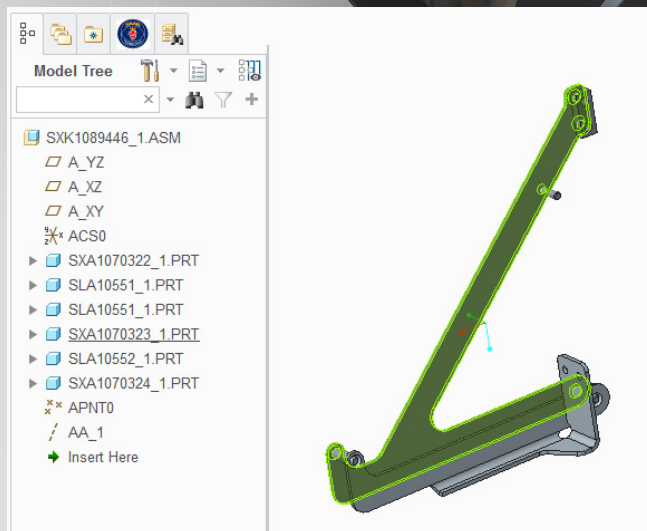


Configs to use

- `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

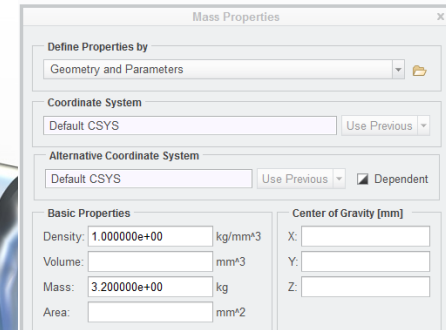
Import config

- 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



Mass Properties

Define Properties by
Geometry and Parameters

Coordinate System
Default CSYS Use Previous

Alternative Coordinate System
Default CSYS Use Previous Dependent

Basic Properties		Center of Gravity [mm]	
Density:	1.000000e+00 kg/mm ³	X:	
Volume:	mm ³	Y:	
Mass:	3.200000e+00 kg	Z:	
Area:	mm ²		

Component placement controls

Allows reorientation of components during placement

- Component Drag 
- Spin 
- Move 

Assembly constraints

How to change default constraint type

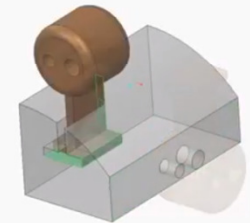
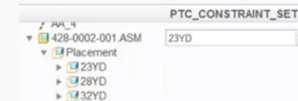
- Set config option `auto_constr_always_use_offset` - Never
 - Coincident will be the default constraint type when reference pairs are planes, linear edges or datum axes or planes to edge pairing
- Set config option `auto_constr_always_use_offset` - Yes
 - Offset (Angle or Normal) will be the default 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

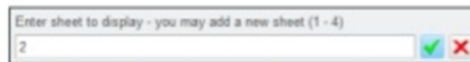
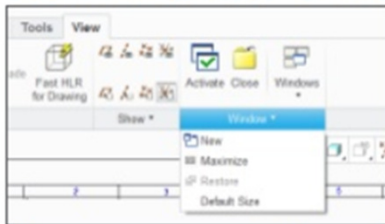
Constraint Sets in Creo

- 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.

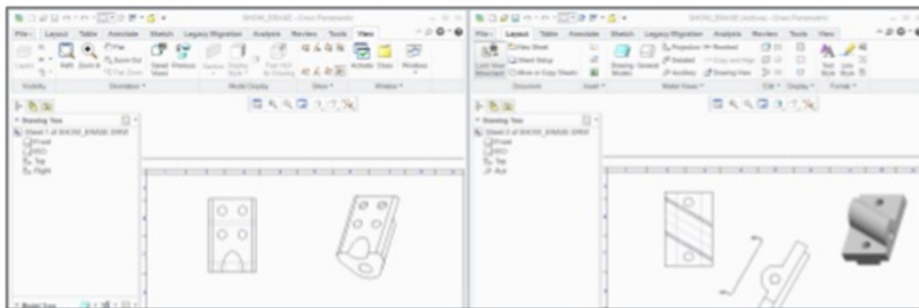


Drawing

- 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.

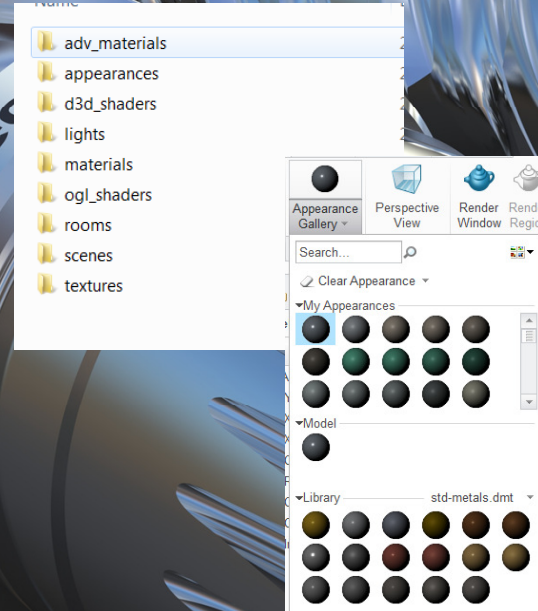
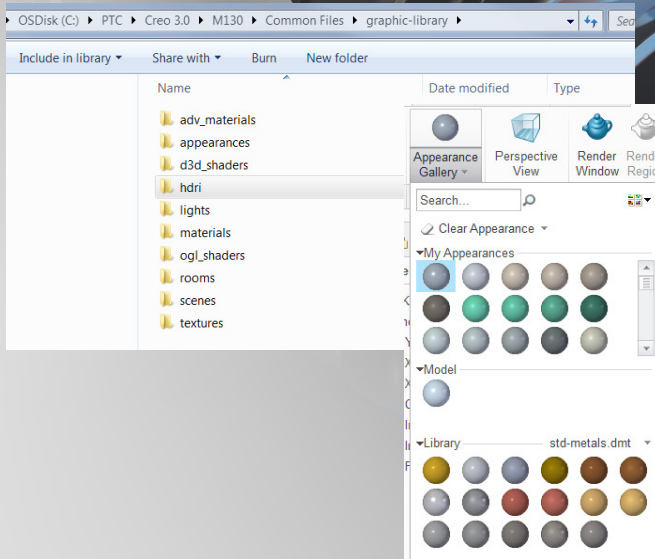


- You can now work in multiple drawing sheets simultaneously. (Also works well for large models where regular reorientation is time consuming).




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: <ul style="list-style-type: none">• 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: <ul style="list-style-type: none">• 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.

PTC Creo

Simulate



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
- Search the PTC Knowledge Base

The screenshot shows the PTC Community website interface. At the top, there's a navigation bar with 'ptc community' and a search bar. Below that, the 'Creo' category is selected. The main content area is divided into several sections: 'Creo Modeling Questions' with a description and statistics (112208 posts, 112181 new, 31m ago); 'Additional Creo Questions' (38230 posts, 38230 new, 2 hours ago); 'Creo Ideas' (9755 posts, 9754 new, yesterday); and 'Creo Blog' (1908 posts, 1907 new, Tuesday). There are also 'Announcements' and 'Top Kudoed Authors' sections.

The screenshot shows a navigation menu for the PTC Knowledge Base. It includes a search bar at the top. The menu is organized into several categories, each with a plus sign icon: 'What's New Creo 4.0', 'Creo Tutorials', 'Creo Parametric Tutorials', 'Creo Flexible Modeling', 'Creo Intelligent Fastener Tutorials', 'Creo Advanced Framework Extension', 'Clearance and Creepage Extension', 'Creo Simulate Tutorials', 'Creo Render Studio Tutorials', 'PTC Mathcad Tutorials', 'Fundamentals', 'Model-Based Definition', 'Data Management', 'Design Exploration', 'Part Modeling', 'Data Exchange', 'Detailed Drawings', 'Layout', 'Surfacing', 'Rendering', and 'Assembly Design'.

Creo Tutorials

PTC provides tutorials to help you get started with Creo. You will need a valid account on PTC University Learning Exchange to download models.

- Creo Parametric
 - [Beginners Tutorials](#)
- Creo Flexible Modeling
 - [Beginners Tutorials](#)
- Creo Intelligent Fastener
 - [Beginners Tutorials](#)
- Creo Advanced Framework Extension
 - [Beginners Tutorials](#)
- Clearance and Creepage Extension
 - [Beginners Tutorials](#)
- Creo Simulate
 - [Beginners Tutorials](#)
- PTC Mathcad
 - [Beginners Tutorials](#)

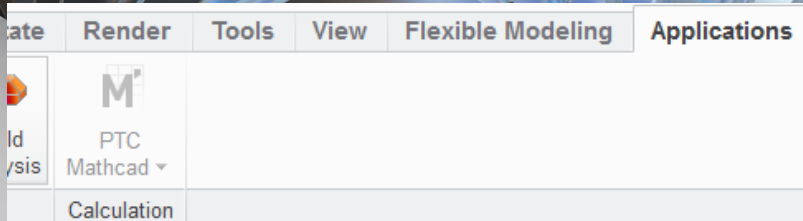
PTC Creo eSupport

Welcome Lars Bjors from PDSVISION AB
[Not Lars ?](#)

- Home
- My Company
- Creo & Pro/ENGINEER
- Windchill
- Mathcad
- Creo View & ProductView
- ↓
- +

Search knowledge articles, best practices, and more ...

Mathcad integrated in Creo



A screenshot of the PTC website's 'PTC Mathcad Worksheets' product page. The page features a navigation menu on the left with links for 'Creo Parametric', 'PTC Mathcad', 'PTC Mathcad Worksheets', 'Creo View', 'Student Editions', and 'PTC University'. The main content area displays four product listings, each with a 'PTC Mathcad' logo, a description, and an 'ORDER NOW' button. The products are: 'Roark's Worksheet Library - 6th Edition for PTC Mathcad Prime 4.0' (4,730.00 SEK), 'PTC Mathcad Worksheet Library - Volume 1' (5,780.00 SEK), 'PTC Mathcad Worksheet Library - Volume 2' (5,780.00 SEK), and 'PTC Mathcad Worksheet Library - Applied Math' (2,190.00 SEK). A 'Global Support now available with PTC eLearning!' banner is also visible.

A screenshot of the PTC Community website. The page title is 'PTC Mathcad - All Worksheets'. The breadcrumb trail is 'Community > PTC Mathcad > PTC Mathcad Blog > PTC Mathcad - All Worksheets'. A search bar is present with the text 'Search all content'. Below the title, there is a user profile for 'TudorMarin Level 1' with a date '04-25-2018'. A promotional message reads: 'Want a free copy of PTC Mathcad Express? [Download your copy today.](#) New Users to PTC Mathcad:'. Below this, there are two bullet points: '• [Getting Started](#)' and '• [Natural Math and Units](#)'. At the bottom, it says 'Explore worksheet categories:' followed by two bullet points: '• [Algebra & Geometry](#)' and '• [Calculus, Statistics, and Data Analysis](#)'.

Creo 5.0 is son on your desktop

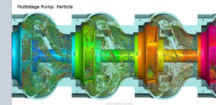
4.0	F000	December 15, 2016
	M010	March 29, 2017
	M020	June 27, 2017
	M030	<i>September 27, 2017</i>
	M040	<i>December 20, 2017</i>
	M050	<i>March 19, 2018</i>
5.0	F000	<i>March 13, 2018</i>
6.0	F000	<i>March, 2019</i>
7.0	F000	<i>March, 2020</i>
	F000	March 27, 2012

CREO 5.0: NEW PRODUCT INTRODUCTION



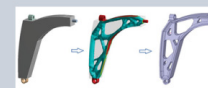
CFD

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



Technical Committees : CAD



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