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Document - CS251133

Document Information

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Document Details

Title	Which new capabilities or enhancements are planned for Creo Parametric 4.0 M010, M020, M030 and M040
Description	• Which new capabilities or enhancements are planned for Creo Parametric 4.0 M010, M020 and M030 • Mirror feature enhancement (part mode) is not available in Creo Parametric 4.0 F000 • New **Instrumented** Assembly sub-type creation is not available in Creo 4.0 M020
Applies To	• Creo Parametric 4.0 F000 to M030
Cause	

Resolution	*Below is forward looking information subject to change without notice*
	
	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
	
	• Mirror feature enhancement (part mode) is planned for M020
	
	   • The initial delivery of the planned Mirror feature in Creo Parametric has changed from Creo 4.0 M020 to Creo 5.0
	   • Ability to edit the mirror to add or remove features
	
	• 3D Sweep feature
	
	   • The initial delivery of the planned 3D Sweep capabilities in Creo Parametric has changed from Creo 4.0 M020 to **Creo 5.0**
	
	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

Creo Parametric 4.0 M040

- **Creo Flow Analysis** (*Planned* for Creo 4.0 M040)

  - This application will enable Computational Fluid Dynamics (CFD)
  - Refer to CS173869 for more information

- **Creo Augmented Reality (AR) Design Share**

  - CS277017

- Enhancements to **Creo Product Insight Extension** in Creo Parametric 4.0 M040

  - **Sensor Data View and Emulation** - Status indicator —When you load data into the table in the **Sensor Data View and Emulation** dialog box, a yellow status flag may appear. This flag indicates there is a need for regeneration after running the selected data slice. The yellow flag is in addition to the check mark and red flag.
  - **Sensor Data View and Emulation - Export to server**—Analysis results that are reported into the sensor data table through a Calculated or Virtual sensor can be uploaded to the ThingWorx server. They can then be added to the respective properties of the Thing as Virtual sensor results, assuming that a matching ThingWorx analysis model is linked to the Creo Parametric assembly model. To export the analysis results to the server, click the arrow next to **Export to csv** and click **Export to server**. A confirmation dialog box appears as you are adding data directly to the ThingWorx server.

- Enhancements to **Clearance and Creepage Extension (CCX)** in Creo Parametric 4.0 M040

  - The usability of the Clearance and Creepage Analysis is improved with new workflows and an updated user interface. The workflow is divided into four operations and the tasks for each operation are contained within a tab.

- Configuration Option for Annotation Visibility Reintroduced in Creo 4.0 M040

  - The configuration option, **visible_annotations_scope** is reintroduced. Use this configuration option to specify whether submodel annotations are visible in assemblies.

    - When set to **active mode only**, only annotations in the active model are visible.
    - When set to **all**, all annotations in submodels appear. The default is active models only.

- New Configuration Option for the Model Tree Added in Creo Parametric 4.0 M040

  - The configuration option, **enable_dragdrop_on_components** is added to Creo Parametric 4.0 M040. Use this configuration option to prevent the restructuring of the Model Tree.

    - If you set **enable_dragdrop_on_components** to **Reorder** you can reorder only. Restructuring using the drag and drop operation is not possible, but the Restructure command (Model ▸ Component ▸ Restructure) is still available.
    - If you set to **All**, you can use the drag and drop operation to reorder components and features and the Restructure command to restructure the Model Tree.
    - The default is **All**.

- Visibility of Combination State Is Improved in Creo 4.0 M040

  - Axis assignment to and removal from Model–Based Definition (MBD) combination states is improved. The previous inconsistency for semi-MBD states relating to axis visibility control is removed. Historically, semi-MBD states controlled axis visibility. This is not consistent with the new notion of semi-MBD states introduced in Creo Parametric 4.0. This inconsistency is corrected and semi-MBD states no longer control axis visibility.

- Compatibility for Printer Configuration Files Is Extended in Creo 4.0 M040

  - The Plot functionality in Creo Parametric 4.0 M040 provides extended compatibility for printer configuration files (.pcf) that specify a specific rotation and that were created in Creo Parametric 3.0 M090 and earlier.

    - When you set **rotate_plotting** in the **.pcf** file to **yes**, automatic rotation is disabled and rotation logic corresponding to Creo Parametric 3.0 M090 and earlier is enabled.
    - When you set **rotate_plotting** to **no**, standard out-of-the-box plot configuration occurs.

- A Check Is Added for Duplicate Draft Datum Feature Symbols in Creo Parametric 4.0 M040

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