



# Creo Elements/Direct Modeling Whats New

## Content of Modeling Whats New

New in 20.8

New in 20.7

New in 20.6

New in 20.5

New in 20.4

New in 20.3

New in 20.2

New in 20.1 + earlier

## Creo Elements/Direct Modeling Express

PE9.0 ~~20.7

PE8.0 ~~20.3

PE6.0 ~~19.0

PE4.0 ~~18.0

PE3.0 ~~17.0

PE2.0 ~~1?.0

An attempt to compile "What's New" information on different versions of Creo Elements/Direct Modeling.

The PTC online information about 'What's New' is evaluated from time to time; e.g.: [Whats NEW 20.4 OSD\\_Main](https://support.ptc.com/help/creo/ced_modeling/r20.4.0.0/en/#page/ced_modeling%2FOSDM_Main%2Fwhatsnew.html%23)  
([https://support.ptc.com/help/creo/ced\\_modeling/r20.4.0.0/en/#page/ced\\_modeling%2FOSDM\\_Main%2Fwhatsnew.html%23](https://support.ptc.com/help/creo/ced_modeling/r20.4.0.0/en/#page/ced_modeling%2FOSDM_Main%2Fwhatsnew.html%23))

## New in 20.8

- Modeling ( 20.8.0.0 )
  - Seed and Boundary selection
  - Updated versions for data exchange support
  - Performance Advisor enhancements
- Annotation
  - Updated standard compliance for symbols – ISO 21920–2021
- Sheet Metal
  - Overlap check during unfolding operations
- Part Library
  - Standard Parts Browser enables access to frequently-used standard parts

## New in 20.7

- Modeling ( 20.7.0.0 )
  - Copy Color option in the Simplification module and the Gather Faces command

- Create Side Faces option in various Creo Elements/Direct Modeling commands
- Creo View MCAD Recipe Editor Included in Creo Elements/Direct Modeling
- Performance Advisor for Creo Elements/Direct Modeling
- Annotation
  - Bolt circle creation
  - Export to Creo View MCAD from Annotation
  - Create searchable PDF drawings

## New in 20.6

- Modeling ( 20.6.3.0 )
  - Update Bends in the Modify Material Command
  - Simplification Enhancements
  - Tree Icon Indicating the Context of a Selective Instance
  - Refined Structure Browser Window Behavior
  - Creo View MCAD Export and Printing Enhancements
- Annotation
  - Enhanced Multi-language Drawings Support
  - Broadened Standards Support – ISO 10135–2007 // ISO 13715–2017
  - Inspection Dimensions
  - Usability and Productivity Enhancements
- Part Library
  - New Commands to Check Standard Parts

## New in 20.5

- Modeling ( 20.5.0.0 )
  - Graphical Activation Indication for Configurations
  - Enhanced Simplification Capabilities
  - Multi-level Features in Sheet Metal
  - New Icons to Differentiate the CAD and Data Management User Interface Windows
- Annotation
  - Areal Surface Finish Support
  - Multi-language Drawings Support
  - Improved Import and Export of DXF/DWG Drawings

## New in 20.4

- Modeling ( 20.4.4.0 )
  - Capture and Re-use Selections in Modeling

- Zoom-To-Selected for Geometry
- Associate Simplified Models
- Improved Import of Creo Multibody Parts
- Performance Enhancements – a setting "Load Model at Center"
- Annotation
  - Dimension Between Reference Lines
  - Delete Multiple Sheets in Annotation
  - Adjacent Text in Annotation
  - Enhanced Standard-Compliant Surface Finish Symbols in Annotation – ISO 1302-2002 and ASME Y14.36-2018.
- Part Library
  - New Search Similar Parts Tool in Part Library

## New in 20.3

- Modeling ( 20.3.5.0 )
  - Enhanced Augmented Reality (AR) Design Share
  - Auxiliary View for General Positioning Workflows
  - Quick Component Preview – Middle-click parts or assemblies in the Structure Browser.
  - Zoom-To-Selected for Geometry
  - Enhanced PVZ Export
  - Enhanced Customization for PVZ Workflows
  - Importing Multibody Parts
- Annotation
  - All Over and All Around Modifiers
  - Indicators for Dimensional Tolerancing
  - Lock Views
- Part Library
  - Creating DXF or MI files of Sheet Metal Parts

## New in 20.2

- Modeling ( 20.2.4.0 )
  - Augmented Reality (AR) Design Share in Creo Elements/Direct Modeling
  - Enhanced Usability in Creo Elements/Direct Modeling – Position after Load.
  - New Web-based Help Centers On ptc.com
  - Enhanced Export to Creo View
- Annotation
  - Part-colored Shaded Sections

- Updated Standards Support in Annotation
  - ISO 1101:2017
  - ISO 5459
  - ISO 14405:2016
  - ASME Y14.5 – 2009
  - ISO 2553:2013
- Part Library
  - Creating Thread Shape – exact shape of outer and inner threads
  - Creating Knurls
  - Creating a Simplified Assembly

## New in 20.1 + earlier

- The Web-based Help Centers are not available for versions 20.1 and earlier releases. Therefore the list is ends here. It would require local installations to pick up more information of older versions.

## Creo Elements/Direct Modeling Express

The connection to the full/professional version can be made based on the :release-name and the :date . See the list above. Not all features are available in the free Express version.

- use (display (oli::sd-inq-version)) for detailed version information of Modeling.
- use (display (oli::sd-inq-version "ANNOTATION")) for detailed version information of Annotation.

## PE9.0 ~~20.7

```
:FULL-STRING      : Creo Elements/Direct Modeling Express 90.0P1017  17-Oct-2024
:PRODUCT-NAME     : Creo Elements/Direct Modeling
:RELEASE-NAME     : 20.7
:VERSION          : 90.0P1017
:VERSION-NUMBER   : 90.0.0.0
:MAJOR            : 90
:MINOR           : 0
:DATE            : 17-Oct-2024
:BASE-DIRECTORY   : C:/Program Files/PTC/Creo Elements/Direct Modeling Express 9.0
:FILING-RELEASE-NAME: 20.7
:FILING-MAJOR     : 20
:FILING-MINOR    : 7
```

```
:FULL-STRING      : CoCreate Annotation Release 20.7  10-Oct-2024 build 15 (1307.15)
:VERSION          : 20.7
:MAJOR            : 20
:MINOR           : 7
:DATE            : 10-Oct-2024
:BUILD           : 1307.15
```

## PE8.0 ~~20.3

:FULL-STRING : Creo Elements/Direct Modeling Express 20.3BP0810 10-Aug-2020  
:PRODUCT-NAME : Creo Elements/Direct Modeling  
:RELEASE-NAME : 20.3  
:VERSION : 20.3BP0810  
:VERSION-NUMBER : 80.0.0.24  
:MAJOR : 80  
:MINOR : 0  
:DATE : 10-Aug-2020  
:BASE-DIRECTORY : C:/Program Files/PTC/Creo Elements/Direct Modeling Express 8.0  
:FILING-RELEASE-NAME: 20.3  
:FILING-MAJOR : 20  
:FILING-MINOR : 3

## PE6.0 ~~19.0

:FULL-STRING : PTC Creo Elements/Direct Modeling Express 60.0 21-May-2014  
:PRODUCT-NAME : PTC Creo Elements/Direct Modeling  
:RELEASE-NAME : 19.0  
:VERSION : 60.0  
:VERSION-NUMBER : 60.0.0.392  
:MAJOR : 60  
:MINOR : 0  
:DATE : 21-May-2014  
:BASE-DIRECTORY : C:/Program Files/PTC/Creo Elements/Direct Modeling Express 6.0  
:FILING-RELEASE-NAME: 19.0  
:FILING-MAJOR : 19  
:FILING-MINOR : 0

:FULL-STRING : CoCreate Annotation Release 19.0 8-May-2014 build 07 (1071.07)  
:VERSION : 19.0  
:MAJOR : 19  
:MINOR : 0  
:DATE : 8-May-2014  
:BUILD : 1071.07

## PE4.0 ~~18.0

:FULL-STRING : Creo Elements/Direct Modeling Express 40.0AP0727 27-Jul-2011  
:PRODUCT-NAME : Creo Elements/Direct Modeling  
:RELEASE-NAME : 18.0  
:VERSION : 40.0AP0727  
:VERSION-NUMBER : 40.0.1.3  
:MAJOR : 40  
:MINOR : 0  
:DATE : 27-Jul-2011  
:BASE-DIRECTORY : C:/Program Files (x86)/PTC/Creo Elements/Direct Modeling Express 4.0  
:FILING-RELEASE-NAME: 18.0  
:FILING-MAJOR : 18  
:FILING-MINOR : 0

:FULL-STRING : CoCreate Annotation Release 18.0 22-Jul-2011 build 14 (879.14)  
:VERSION : 18.0  
:MAJOR : 18  
:MINOR : 0  
:DATE : 22-Jul-2011  
:BUILD : 879.14

## PE3.0 ~~17.0

:FULL-STRING : CoCreate Modeling PE 30.00 17-Jun-2010  
:PRODUCT-NAME : CoCreate Modeling  
:RELEASE-NAME : 17.0  
:VERSION : 30.00  
:VERSION-NUMBER : 30.0.0.15  
:MAJOR : 30  
:MINOR : 0  
:DATE : 17-Jun-2010  
:BASE-DIRECTORY : C:/Program Files (x86)/CoCreate/CoCreate Modeling Personal Edition 3.0  
:FILING-RELEASE-NAME: 17.0  
:FILING-MAJOR : 17  
:FILING-MINOR : 0

:FULL-STRING : CoCreate Annotation Release 17.0 17-Jun-2010 build 14 (789.14)  
:VERSION : 17.0  
:MAJOR : 17  
:MINOR : 0  
:DATE : 17-Jun-2010  
:BUILD : 789.14

PE2.0 ~~1?.0

to be done

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