1

What's New in Creo 11

In this section, find topics that describe the enhancements in this release, categorized by the functional area.

2

11.0.0.0

3

Creo Installation

Update Creo to support Java 17

Creo Parametric 11.0.0.0

Description

- Creo has updated its support of Java to version 17 to address several security vulnerabilities with the old version and to align with the Windchill Java support matrix.
- Customers can choose their preferred Java JRE distribution from either Oracle or Amazon Corretto.
- Creo will only require Java 17 JRE if users are accessing customization through J.Link or Java Object Toolkit, Creo Product Insight connected to Thingworx or Windchill PLM Connector.
- Regular communication with Windchill will not require users to have Java installed.

Benefits

- Maintain support of latest version of Java to address potential security vulnerabilities in older versions.
- Aligned Java support with Windchill.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

4

Assembly Design

Flip Tangent Constraint

Creo Parametric 11.0.0.0

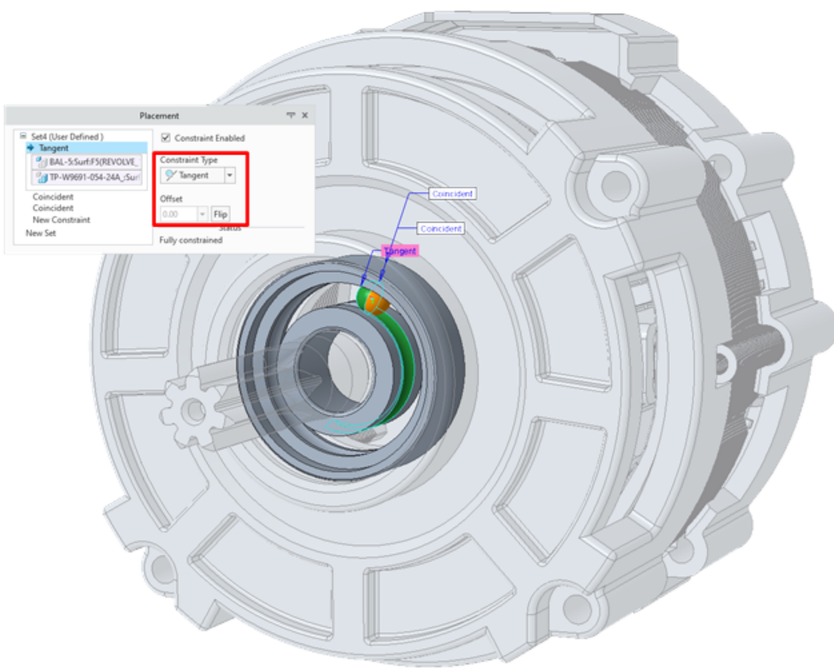
User Interface Location: In the **Component Placement** tab, click the arrow next to **Automatic** and select **Tangent**.

Videos

See the video on the Learning Connector.

Description

Easily place components with tangent constraints in the correct position. When applying a tangent placement constraint, you can flip the tangent direction while the placement point that was picked remains fixed. The tangent direction is kept with a pattern and during dragging and other operations.



Benefits

- Support of **Flip** for a tangent constraint
- Correctly and easily place component with tangent constraints
- Tangent direction is kept with patterns
- Pick point is understood

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Multibody Support in Shrinkwrap Feature

Creo Parametric 11.0.0.0

User Interface Location: Click **Model** ▸ **Shrinkwrap**.

Videos

See the video on the Learning Connector.

Description

When you create Shrinkwrap features in Creo 11, you can now collect bodies from the source assembly to add to the target part. The option **Autocollect all solid geometry** now enables you to collect solid bodies as the resulting geometry type. The following copying methods are available when the resulting geometry is bodies:

- Copy all bodies from the source part to the target part as separate objects.

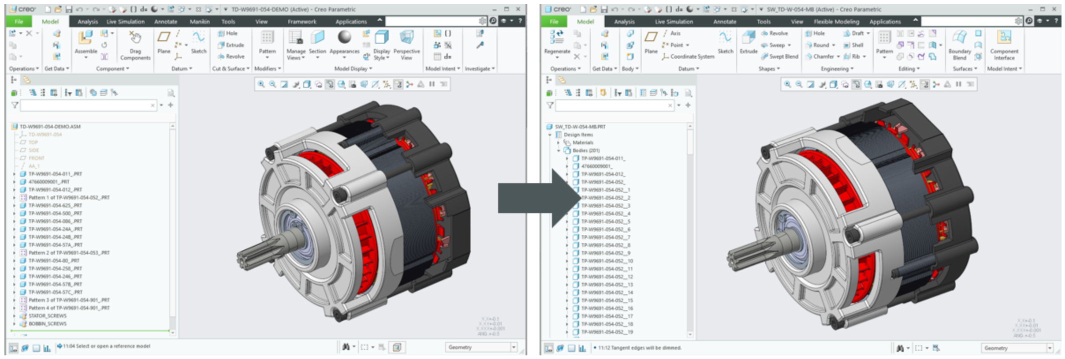
  The resulting body name, material, parameters, and construction attribute are transferred for each body.
- Merge all bodies from the same source part. Each resulting body represents the solid geometry of one source part.

  The resulting body name is the name of the source part. Material, parameters, and construction attributes are transferred from the first body in the source part.
- Merge all bodies from all source parts.

  All resulting geometry is added to one body. The **Body Options** tab becomes available to add the resulting geometry to an existing body or to create a new body.

In situations where bodies cannot be merged, you can choose to create a successful feature when you select the **Leave as separate objects if operation fails** checkbox.

Legacy Shrinkwrap features regenerate as before and are upgraded to the new version when you edit their definition.



Original Assembly | Resulting Multibody Part

Benefits

Faster and easier selection of desired geometry for creating simplified models, envelopes, or conceptual design.

Additional Information

Tips:	None.
Limitations:	Merge within the Shrinkwrap operations might fail, in particular, when parts are defined with different model accuracies.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Performance Reporting

Creo Parametric 11.0.0.0

User Interface Location: Click **Tools** ▸ **Investigate** ▸ **Performance Report**.

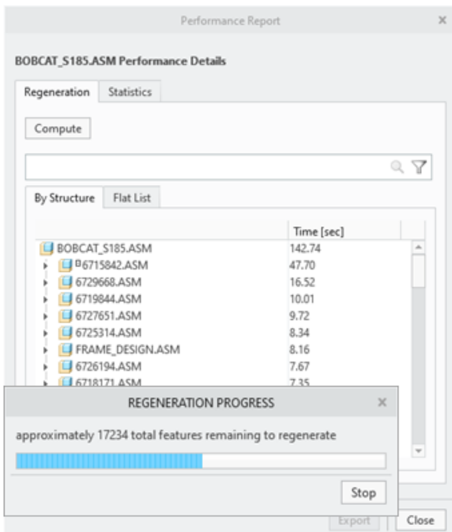
Videos

See the video on the Learning Connector.

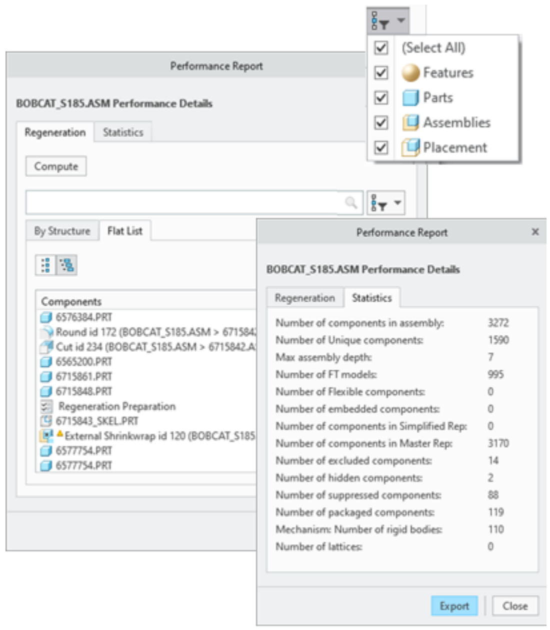
Description

Creo 11 introduces Performance Reports. Part and assembly regeneration performance can now be viewed with the **Performance Report** tool. After clicking **Compute** the part or assembly in the active window is fully regenerated and the resulting regeneration times are reported. By default, the list of components and features is sorted by regeneration time. This makes it easy to find the largest contributors, at the top of the list. In addition to the structured view, where sub-assemblies and parts can be expanded to show the sum of their content, the flat list view provides an overview of each feature, part, and component. Both the structured and the flat list can be searched. Additionally, the **Statistics** tab provides a summary of relevant performance content.

The figure below shows the **Performance Report**, **Regeneration** tab after clicking **Compute**.



The figure below shows the flat list of items in the report, the statistics of how many items of each type are in the model, and the filters you can use to display items in the lists.



Benefits

When working with a part and an assembly, you can understand the regeneration performance contribution of features and components in the selected part or assembly. This enhancement enables you to assess a model's regeneration performance to find potential bottle necks in a design, with the goal of changing and improving the bottle neck performance.

Additional Information

Tips:	None.
Limitations:	• Currently regeneration preparation and actual regeneration times per feature are reported and summed up to parts and assemblies. Other related efforts, mainly post regeneration propagation for various type of Family Table instances is not yet included. • Regeneration times are system dependent and will vary depending on other running processes. • Multiple occurrences of the same component are all added to the respective assembly total. This can give an inaccurate result for the regeneration time of that assembly. • Simplified representations are not supported. • Postprocessing of flexible components and assembly

	cuts is not yet reflected in the features' regeneration time.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

5

Cabling and Routed System

Remove Locations Overhaul

Creo Parametric 11.0.0.0

User Interface Location: Click **Cabling ▸ Locations ▸ Remove Locations**.

Videos

See the video on the Learning Connector.

Description

Remove Locations is one of the most frequently used tools in Cabling; therefore, it was overhauled as part of modernizing the Creo Cabling application.

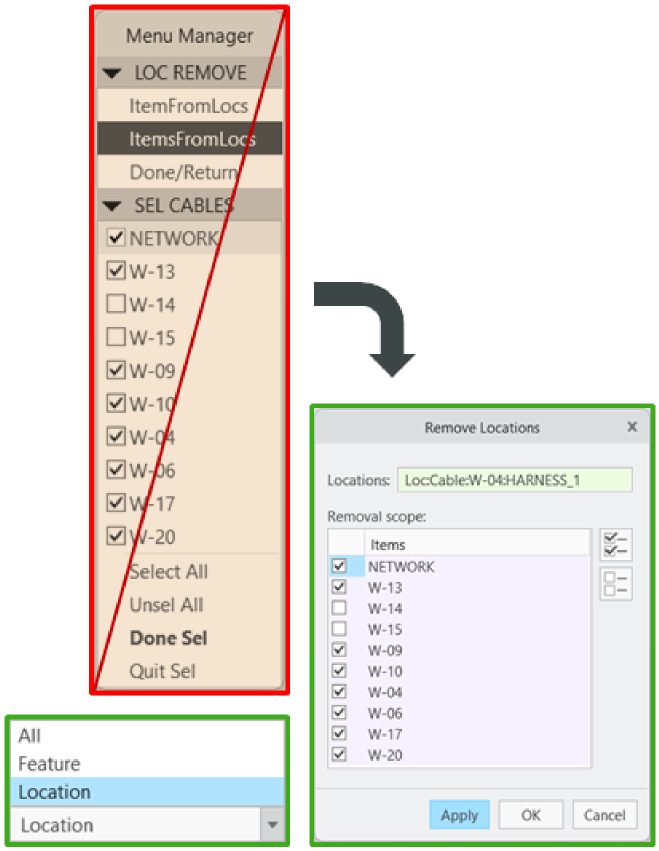
The user interface was upgraded from the legacy Menu Manager to a user-friendly dialog box.

The new user interface provides a dynamic preview in the graphics area as well as the cross-selection between the dialog box, model tree, and graphics area. The system feedback has been improved by adding warning glyphs for invalid operations. Now, the dialog box also supports the Undo and Redo operations.

The filtering capabilities have been improved to remember the previous state of checked or unchecked wires in subsequent selections.

Now, the Remove Locations tool also supports the **Apply** operation, which allows users to remove locations in different portions without restarting the tool.

The logic for the wires’ exclusion has been enhanced to be smarter and the overall usability of the tool has become more intuitive.



Benefits

With improved productivity and usability, it is now easier to use the Remove Locations tool.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Harness Settings

Creo Parametric 11.0.0.0

User Interface Location: Click **Cabling** ▸ **Harness** ▸ **Harness Settings**.

Videos

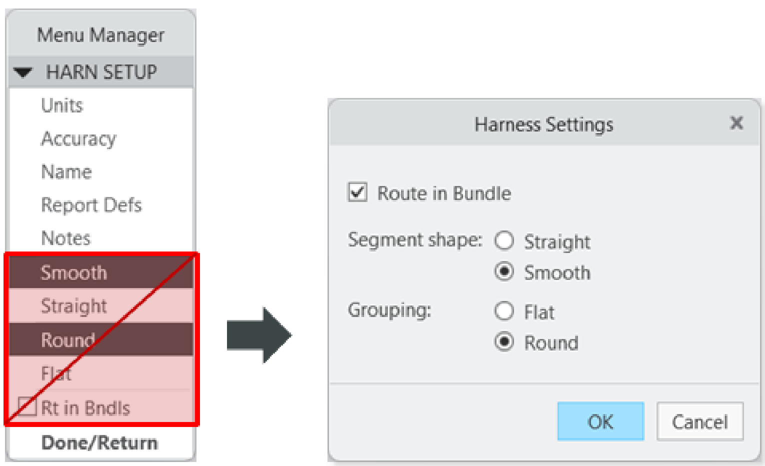
See the video on the Learning Connector.

Description

Previously, the Setup Menu Manager was used for routing settings like **Route in Bundles**, **Segment shape**, and **Grouping**. Now, these settings can be accessed from the **Harness Settings** dialog box.

This enhancement provides the following improvements:

- The harness settings have become asynchronous and can be accessed any time during routing.
- The harness settings are saved with the harness model and are preserved when the model is loaded again instead of being reset when a Creo session is ended.



Benefits

With this enhancement:

- Routing settings can be changed without having to exit the routing tools.
- Saving the routing settings with the harness eliminates the need to set these settings again in a new session.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Insert a Custom Component

Creo Parametric 11.0.0.0

User Interface Location: Click **Cabling** ▸ **Components** ▸ **Insert Component**.

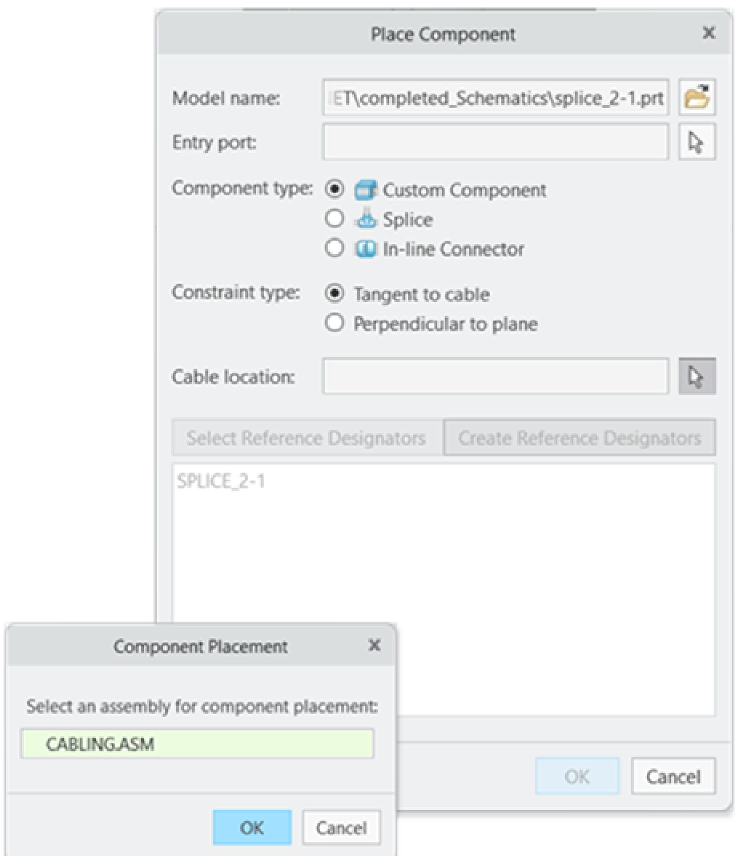
Videos

See the video on the Learning Connector.

Description

Previously, components could be assembled and created from within the Cabling application.

In this release, this functionality has been extended to the **Insert Component** operation. This enhancement gives you the flexibility to insert custom components, in-line connectors, and splices anywhere in the assembly.



Benefits

- The **Insert Component** operation is now aligned with the **Create** and **Assemble** operations.

- The improved dialog box layout and updated tooltips enhance the user experience.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Improved Selection Visibility

Creo Parametric 11.0.0.0

User Interface Location: N/A

Videos

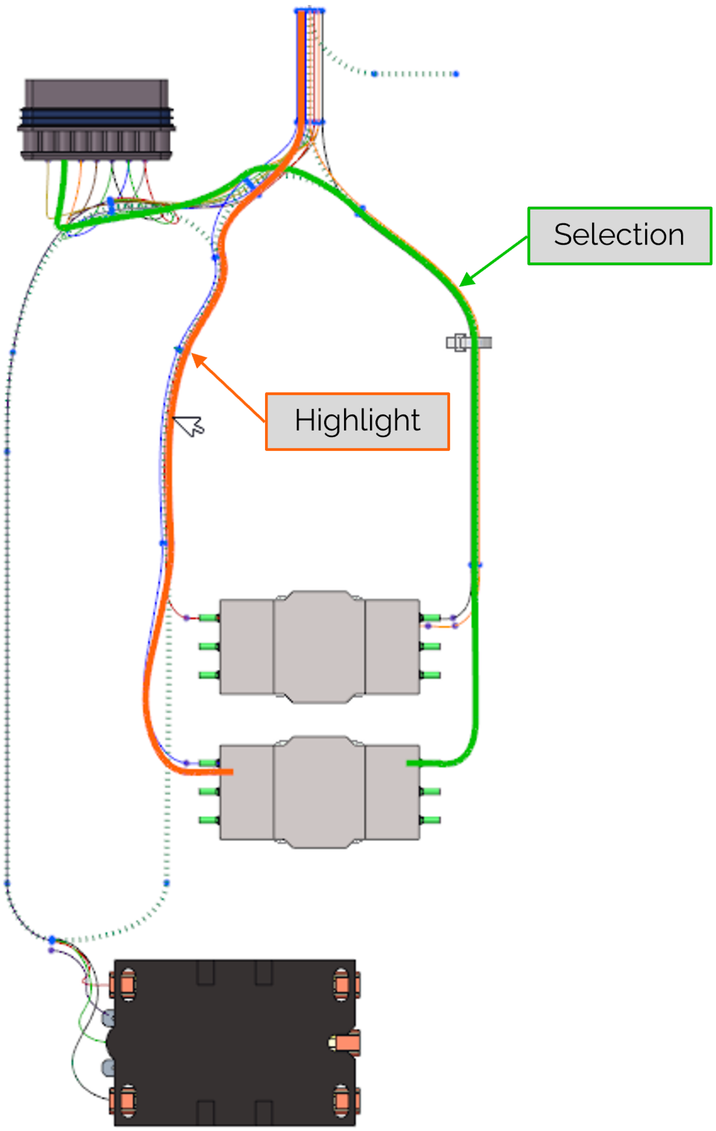
See the video on the Learning Connector.

Description

Previously, when you selected segments, wires, and location points, it affected only their color, making it difficult to identify the items selected in the graphics window, especially when cables had different spool colors.

The new visibility improvements include:

- Selection and prehighlight visibility in the graphics window for all cabling entities has been improved.
- Segments, wires, cables, and bundles have a bigger width when selected or highlighted.
- Location points scale up when selected or highlighted.
- Cosmetic features are more visible.



Benefits

This enhancement makes it easier to identify the selected cabling items in the graphics window.

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Tangency for Locations Placed on a Coordinate System

Creo Parametric 11.0.0.0

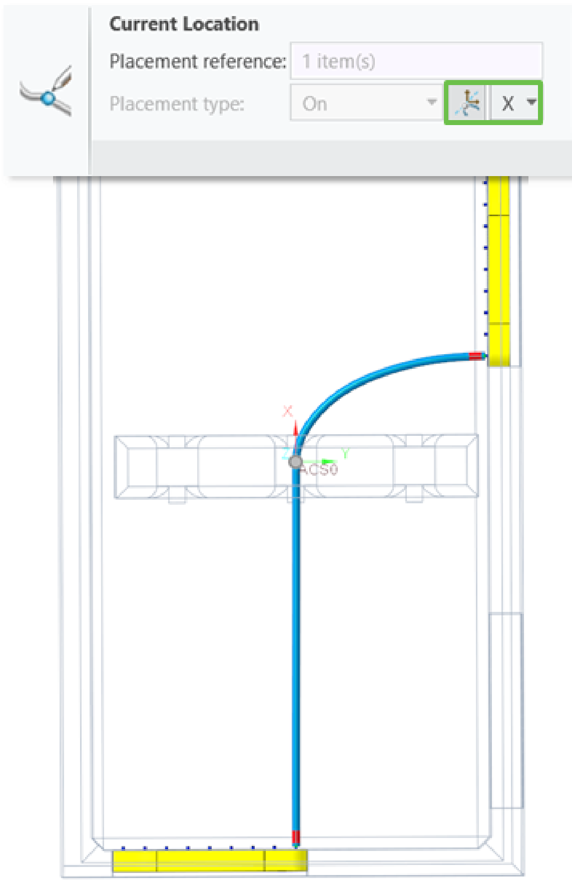
User Interface Location: Click **Cabling ▸ Locations ▸ Location**.

Videos

See the video on the Learning Connector.

Description

A new option  has been added in Creo to make segments tangent to the chosen axis when a location point is placed on a coordinate system. This is available for all routable entities such as wires, cables, bundles, networks, etc.



Benefits

This enhancement provides additional control over the segment shape when routing through a coordinate system.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Cabling Tree Enhancements

Creo Parametric 11.0.0.0

User Interface Location: Cabling Tree.

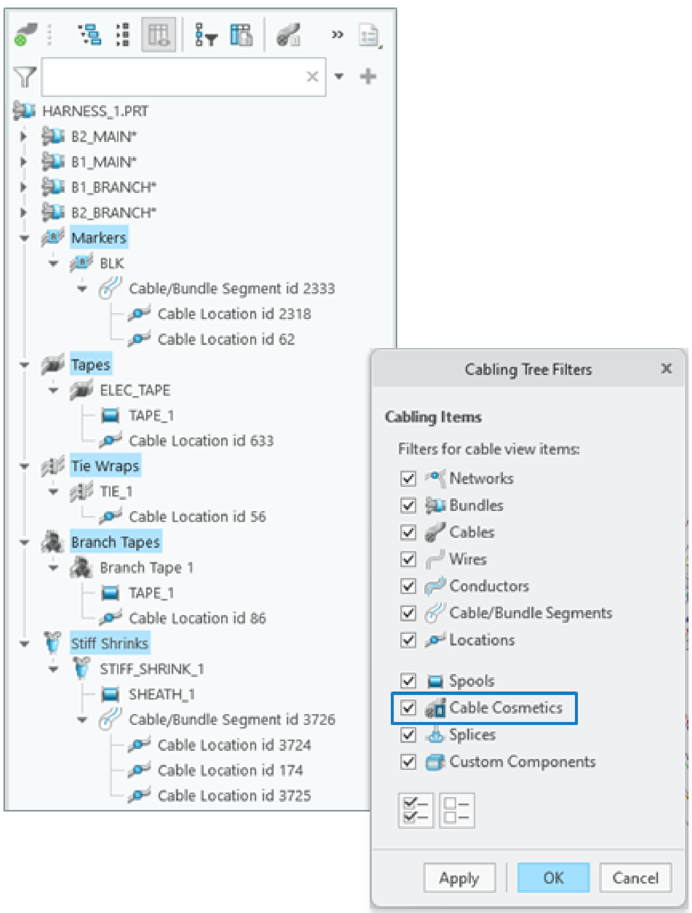
Videos

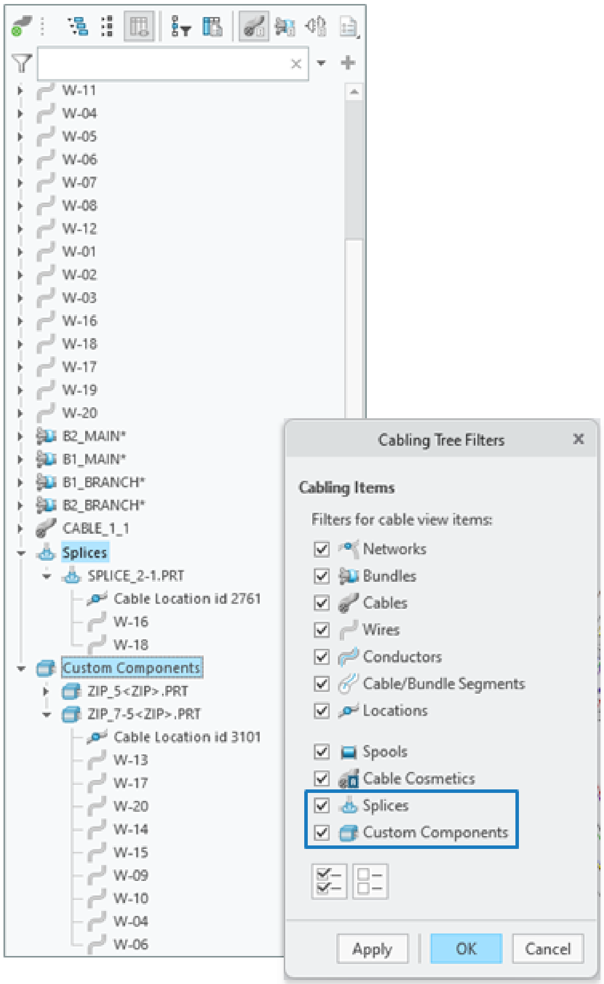
See the video on the Learning Connector.

Description

Previously, out of all cosmetic features only stiff shrinks were supported in the Cabling Tree.

In this release, all cosmetic features such as markers, tapes, tie wraps, branch tapes were added to the Cables and Bundles views of the Cabling tree. Additionally, Cabling Tree now displays Splices (including In-Line Connectors) and Custom Components in all views. New filters to control visibility in the Cabling Tree have been added for all new items.





Some additional enhancements also include:

- Displaying the active harness name on the first node in the Cabling Tree
- Elevating networks to the top of the tree structure
- Displaying terminators by their names instead of their IDs

Benefits

These Cabling Tree enhancements provide full visibility into the harness structure and its components, thereby increases productivity and efficiency in the cabling application.

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

6

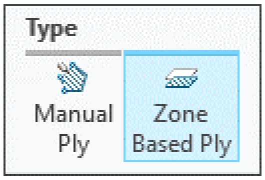
Composite Design

Zone Based Design

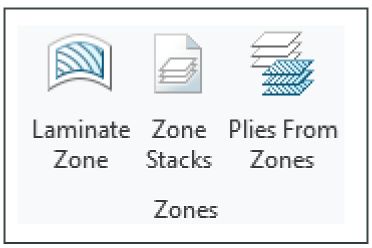
Creo Parametric 11.0.0.0

User Interface Location: In the Composite Design environment, do the following:

- Click **Ply**. Under **Type**, select **Zone Based Ply**.



- Use the following commands in the **Zones** group:
  - **Laminate Zone**
  - **Zone Stacks**
  - **Plies From Zones**

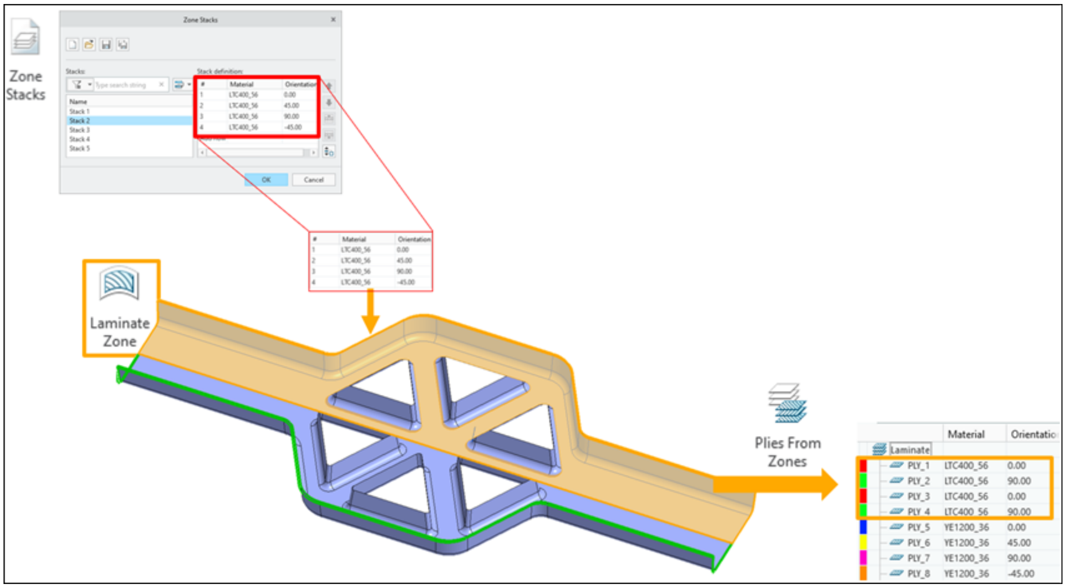


Videos

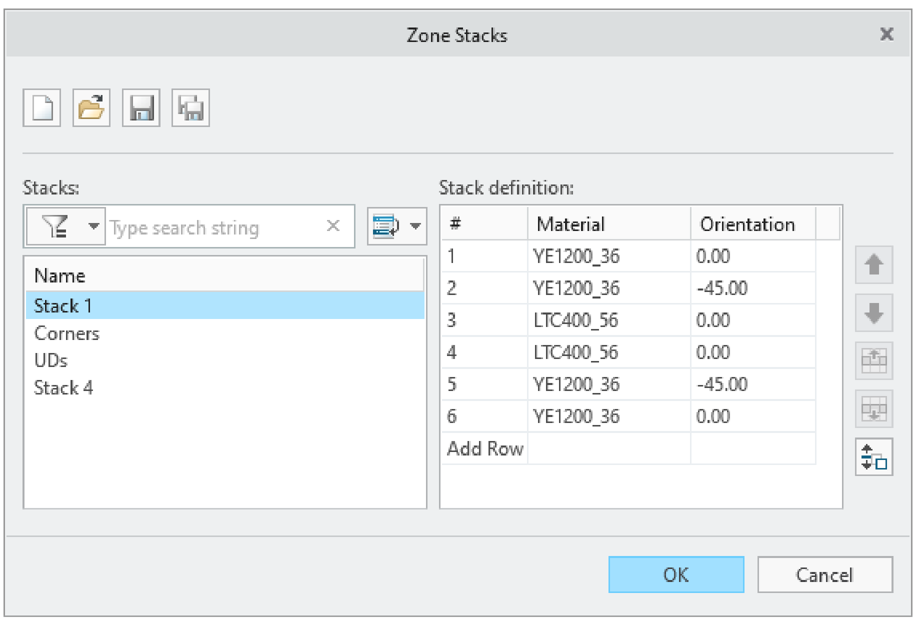
See the video on the Learning Connector.

Description

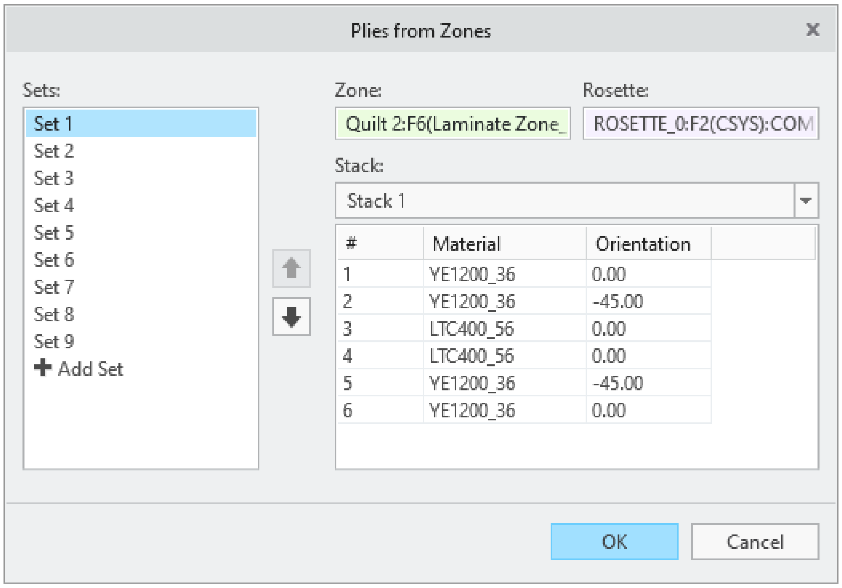
You can now use the new zone-based design to create zone-based plies and cores.



You can define the conceptual zone regions on the layup surface and create zone stack recipes that include the local laminate definitions.



You can combine the zones with zone stack recipes and rosettes using the **Plies From Zones** tool to automatically create zone-based plies and cores.



When you modify the boundary of a zone, the boundaries of the zone-based plies and cores that are created based on the zone get updated.

Benefits

Ability to define a conceptual top-down composite design using zones and zone stacks.

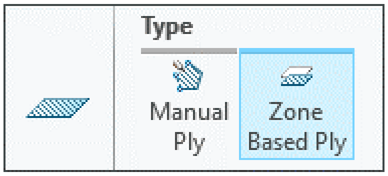
Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Zone Based Plies and Cores

Creo Parametric 11.0.0.0

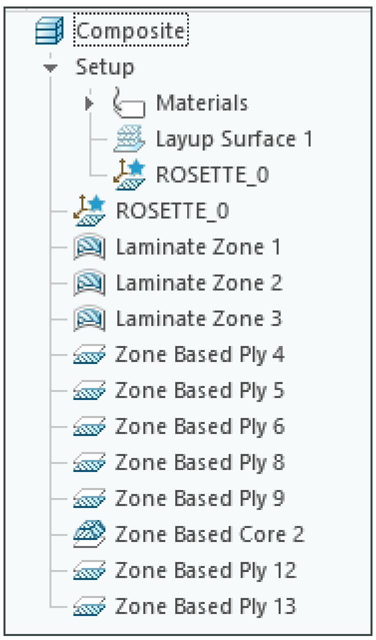
User Interface Location: In the Composite Design environment, click **Ply**. Under **Type**, select **Zone Based Ply**.



Description

You can now create zone-based plies and cores in the Composite Design environment.

You can individually create each zone-based ply or core. Or you can use the **Plies From Zones** tool to automatically create multiple zone-based plies and cores.



When you modify a zone contour, the child zone-based plies and cores follow the change.

Benefits

Ability to define a conceptual top-down composite design using zones and zone-based plies and cores.

Additional Information

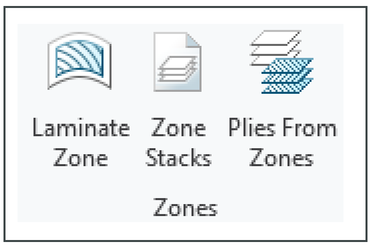
Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Zone Stacks

Creo Parametric 11.0.0.0

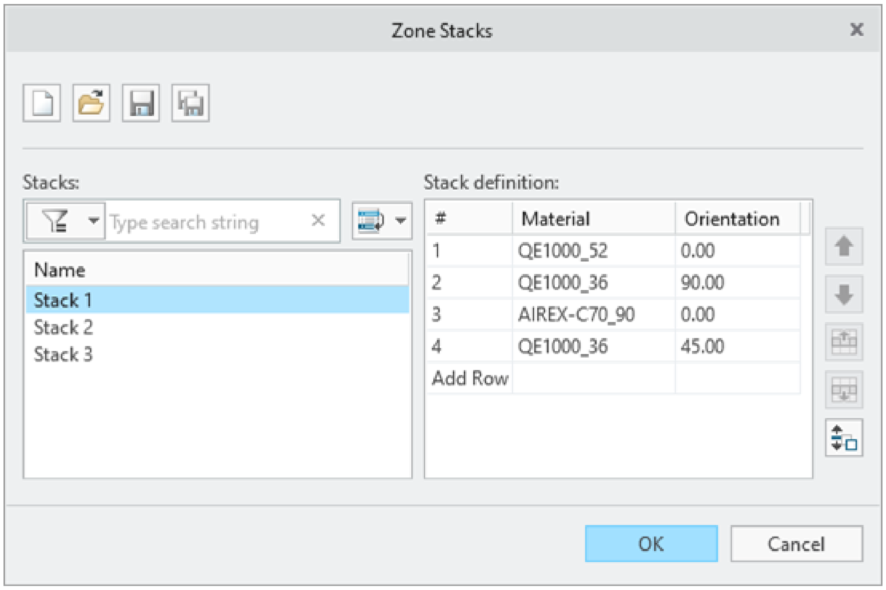
User Interface Location: In the Composite Design environment, click **Zones** ▸ **Zone Stacks**.



Description

You can now define the local laminate recipes using the **Zone Stacks** tool.

You can use a zone stack to define a stack of material and orientation angle combinations. After defining the stack content, you can duplicate it or make it symmetrical. You can then use the zone stack in the **Plies from Zones** tool to automatically create zone-based plies and cores in the region specified by a laminate zone. The zone stacks are saved with the composite model. Additionally, you can save them separately for reuse.



You can specify the full path to the default directory of your stack files library in one of the following ways:

- Click **File ▸ Options**. In the **Creo Parametric Options** dialog box, click **Applications ▸ Composite**. Specify the values under **Plies from Zones**.
- Set the `composite_stacks_dir` configuration option.

Benefits

Ability to define a conceptual top-down composite design using zones and zone stacks.

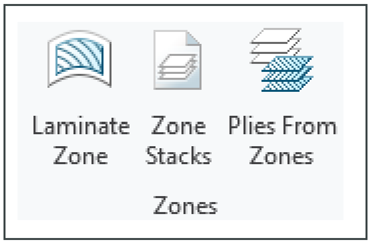
Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	New configuration option: • composite_stacks_dir <empty>*

Plies From Zones

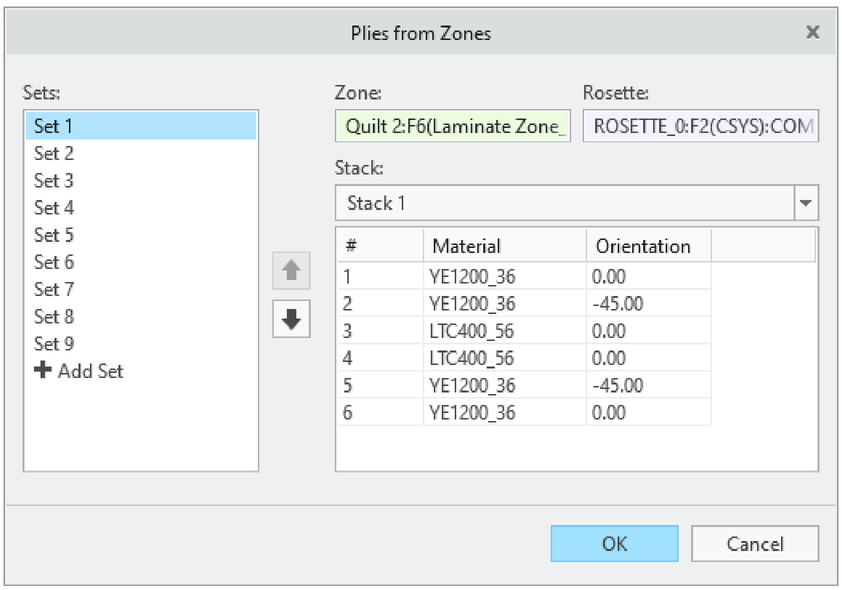
Creo Parametric 11.0.0.0

User Interface Location: In the Composite Design environment, click **Plies From Zones**.



Description

You can now create many zone-based plies and cores using the specified combination of a zone, a zone stack, and a rosette in the **Plies From Zones** tool.



The zone-based plies and cores that are created using the **Plies From Zones** tool are grouped to provide more clarity in the Composite Tree. You can ungroup and reorder the group. You can use the **Plies From Zones** tool multiple times.

Benefits

This enhancement supports a conceptual top-down Composite Design by combining zones and zone stacks, and automatically creating zone-based plies and cores.

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

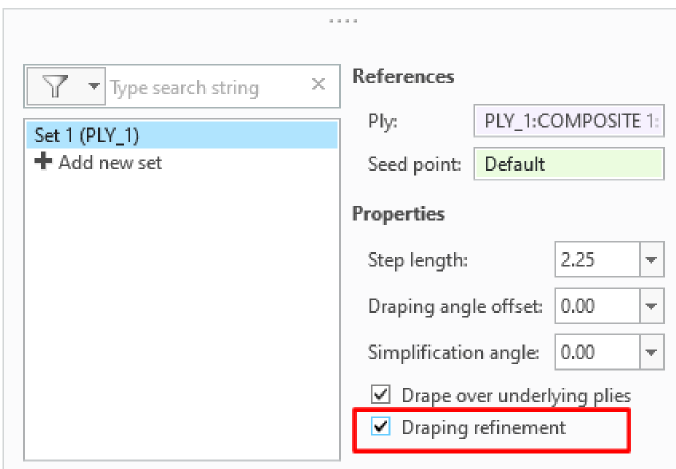
Draping Algorithm Enhancements

Creo Parametric 11.0.0.0

User Interface Location: In the Composite Design environment, Click **Draping Simulation**.

Description

You can now perform a draping simulation on the ply shapes that were unsupported in earlier versions. Also, you can now generate better flat patterns.



The draping simulation capabilities are enhanced to remove the initial limitations. For example, you can select a location for the seed point such that the fixed warp fiber or the fixed weft fiber enters and exits the ply boundary multiple times.

You can use the **Draping refinement** option to improve the draping results for small plies on the relatively large layup surface.

Benefits

This enhancement enables you to perform the draping simulation and create better flat patterns for specific, previously difficult, or unsupported ply shapes.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Draping with Uncured Thickness

Creo Parametric 11.0.0.0

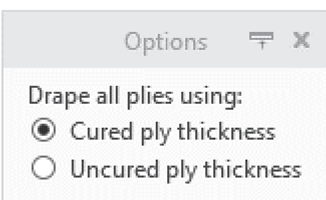
User Interface Location: In the Composite Design environment, do the following:

1. Click **Draping Simulation**.
2. Click the **Options** tab.

Description

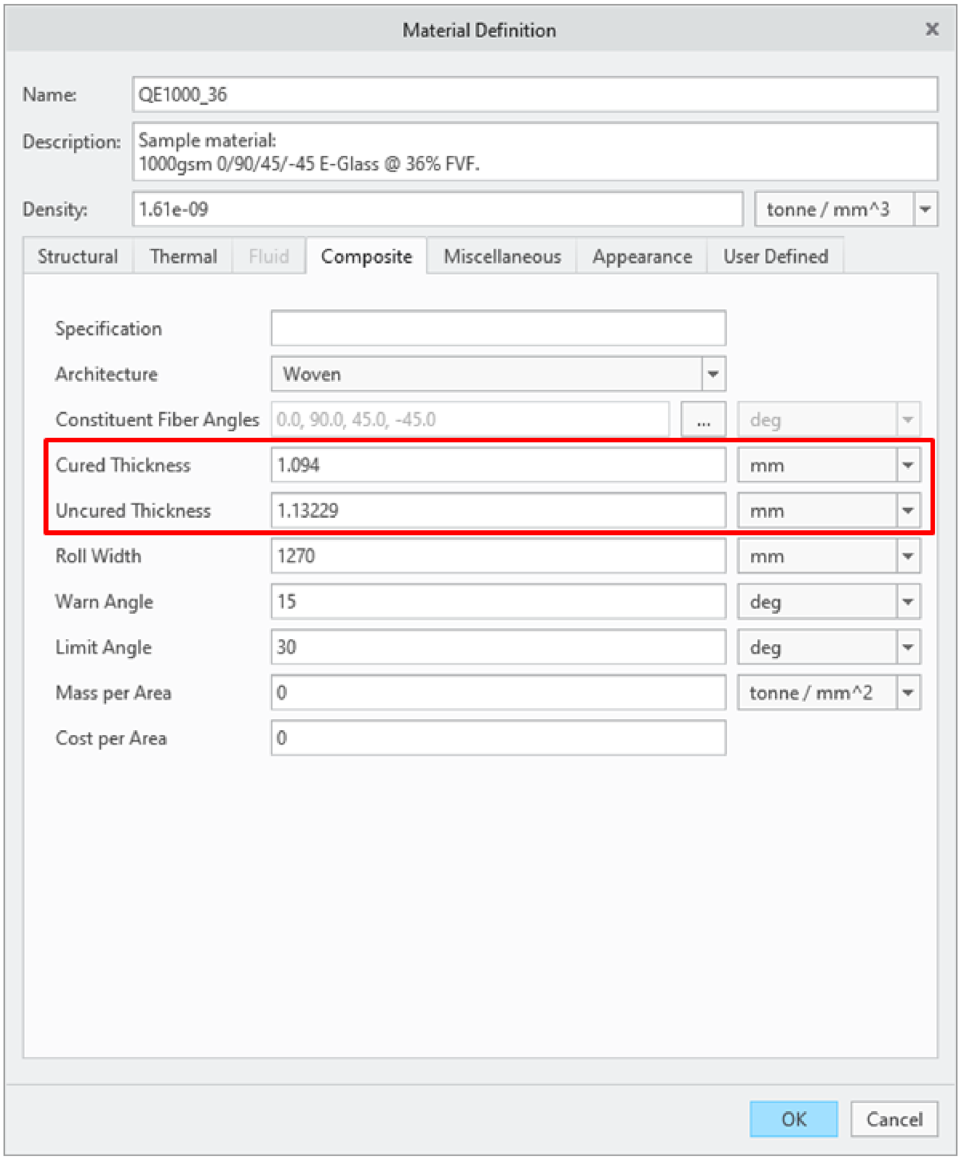
You can now perform a draping simulation based on an uncured ply thickness. This is helpful when generating files for a laser projection and creating ply flat patterns.

When you are performing the draping simulation, you can specify whether to use a cured ply thickness or an uncured ply thickness in the **Options** tab.



The selected type of ply thickness, cured or uncured, is indicated in the graphics window during the draping simulation.

You can specify the cured or the uncured ply thickness value using the **Composite** tab in the **Material Definition** dialog box.



Benefits

This enhancement enables you to perform a draping simulation on the plies based on the uncured thickness, and prepare the flat pattern and laser projection files based on the obtained draping results.

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Flat Ply Preview in a Separate Window

Creo Parametric 11.0.0.0

User Interface Location: In the Composite Design environment, do one of the following:

- Click **Draping Simulation** and then select **Auxiliary Display**.

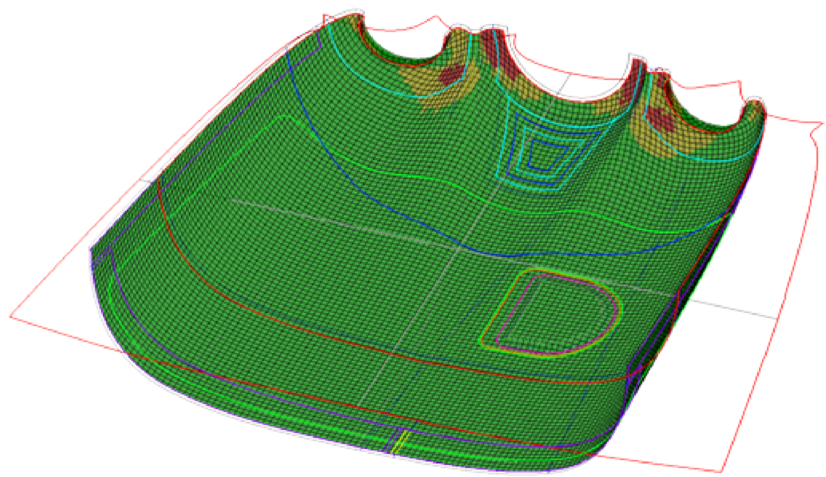
- On the graphics toolbar, click **Draping Results ▸** **Flat Pattern Preview**.

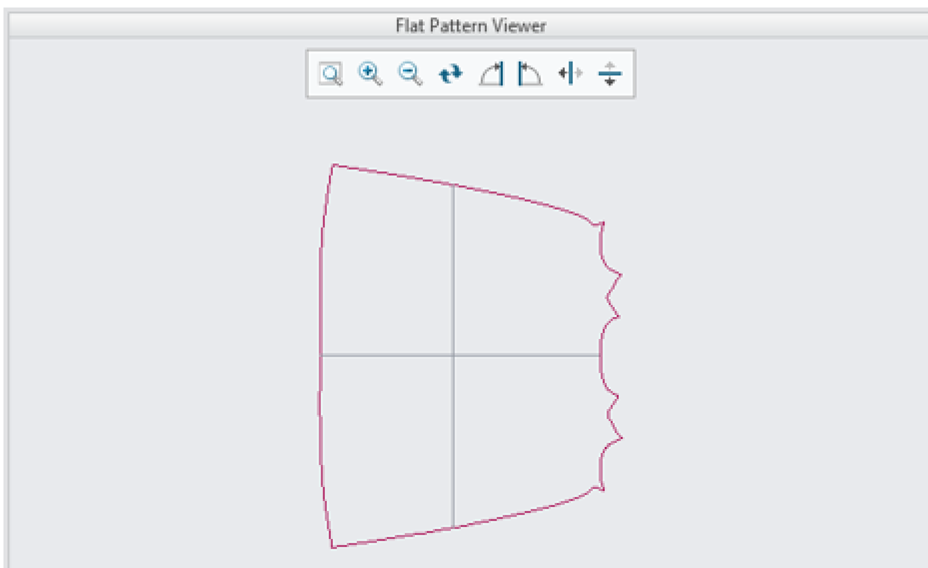
Description

Now, when you are performing a draping simulation on a ply, you can see the flat pattern preview of the ply much more clearly in a separate window. To control the display of the window, use **Auxiliary Display** in the **Draping Simulation** tab.

You can also see the flat pattern of plies, one at a time in a separate window, when viewing the results outside of the draping simulation feature directly from the Laminate Tree. To control the display of the window, use **Draping Results ▸** **Flat Pattern Preview** on the graphics toolbar.

With the separate window, the flat pattern can be observed much more clearly.





Benefits

This enhancement improves productivity by providing a better visibility of the ply and core flat pattern contours.

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Viewing Draping Simulation Results

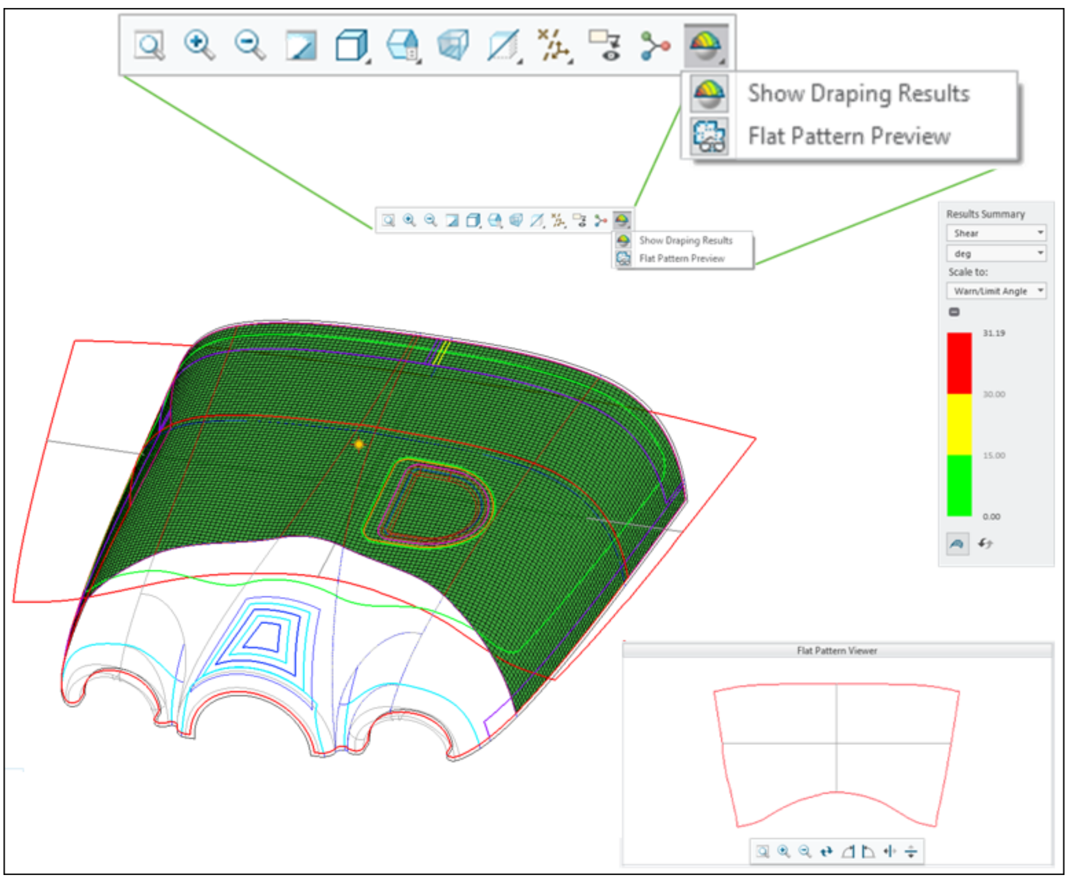
Creo Parametric 11.0.0.0

User Interface Location: In the Composite Design environment, do the following:

- On the graphics toolbar, click **Draping Results** ▸ **Show Draping Results**.

Description

When you are working outside the draping simulation feature, you can view the draping results of a successfully draped ply using the commands on the graphics toolbar. You can now view the draping results in any mode of the Laminate Tree.



Both, the display of draping results and the flat pattern preview, are independent of each other. You can view draping results, a flat pattern, or both at a time.

Benefits

This enhancement improves usability by giving you an easier access to and a better visibility of the draping simulation and the flat pattern results.

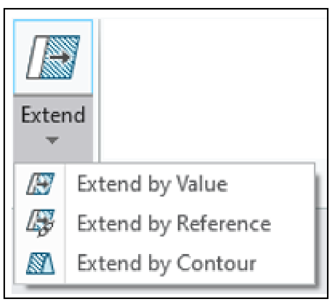
Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Extend Ply—By Value

Creo Parametric 11.0.0.0

User Interface Location: In the Composite Design environment, click **Extend** ▸  **Extend by Value**.

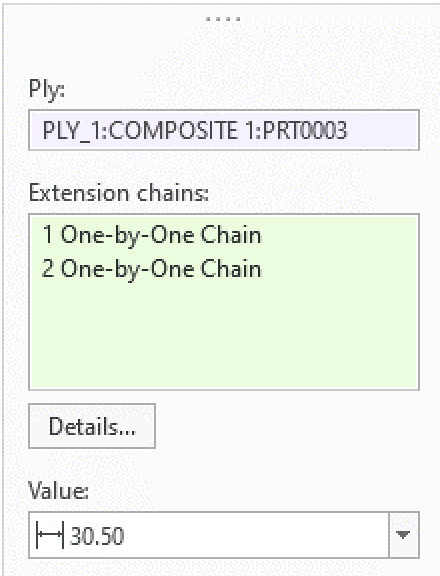


Videos

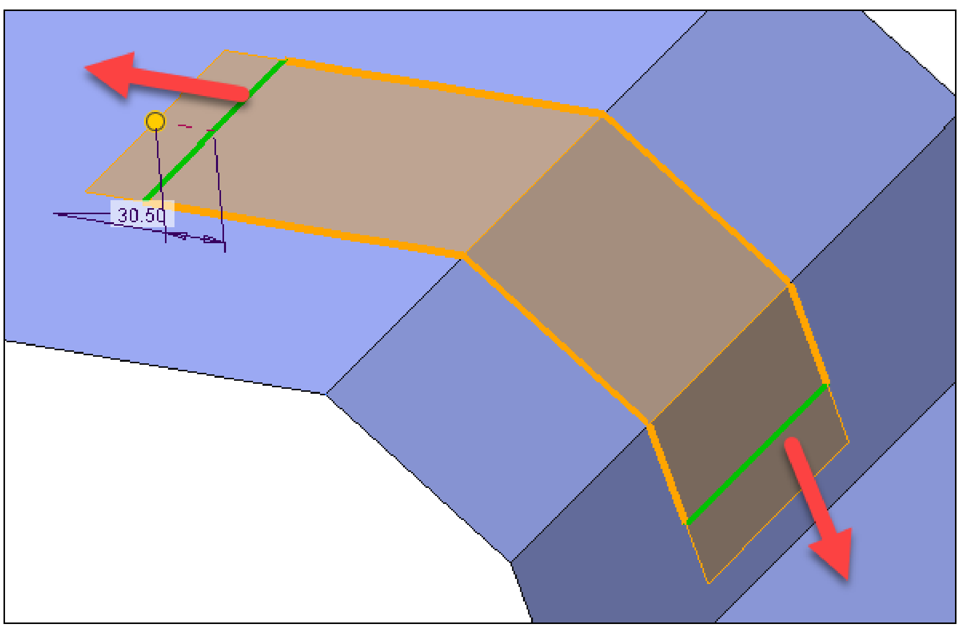
See the video on the Learning Connector.

Description

You can now extend the boundary of plies and cores by a specified value.



You can extend one or more edges or chains in a single extend by value operation. However, you can extend plies or cores only inside the layup quilt.



Benefits

This enhancement increases productivity and provides more flexibility during the composite design and preparation for manufacturing.

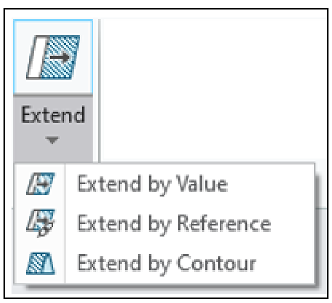
Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Extend Ply—By Reference

Creo Parametric 11.0.0.0

User Interface Location: In the Composite Design environment, click **Extend** ▸  **Extend by Reference**.



Videos

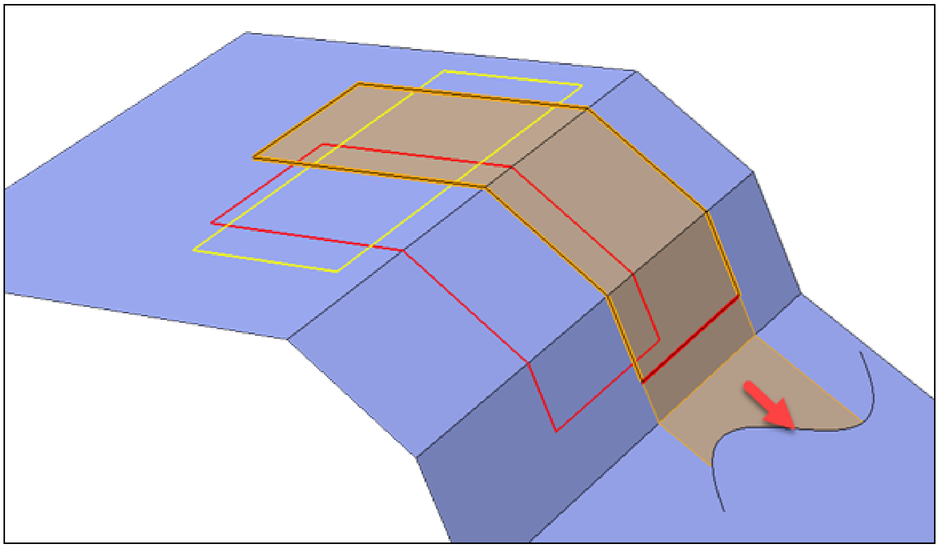
See the video on the Learning Connector.

Description

You can now extend the boundary of a ply up to a specified reference.

Multiple reference types, such as a curve, an edge, a ply boundary or a zone boundary loop, are supported.

You can use an outer loop or an inner loop as a reference, and the extend action is performed inside the layup quilt.



Benefits

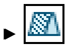
This enhancement provides more flexibility during the Composite Design and preparation for manufacturing.

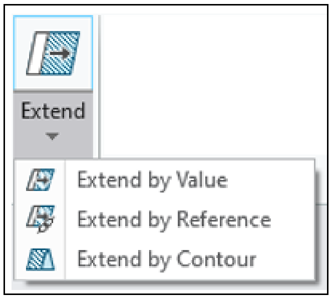
Additional Information

Tips:	None.
Limitations:	You can extend only plies, not cores.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Extend Ply—By Contour

Creo Parametric 11.0.0.0

User Interface Location: In the Composite Design environment click **Extend** ▸  **Extend by Contour**.

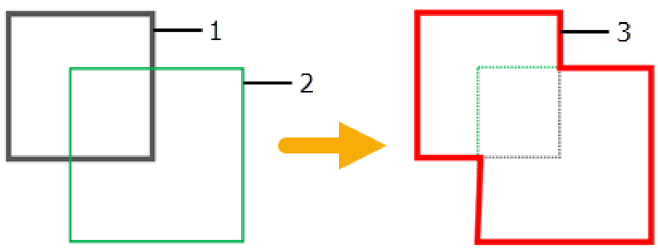


Videos

See the video on the Learning Connector.

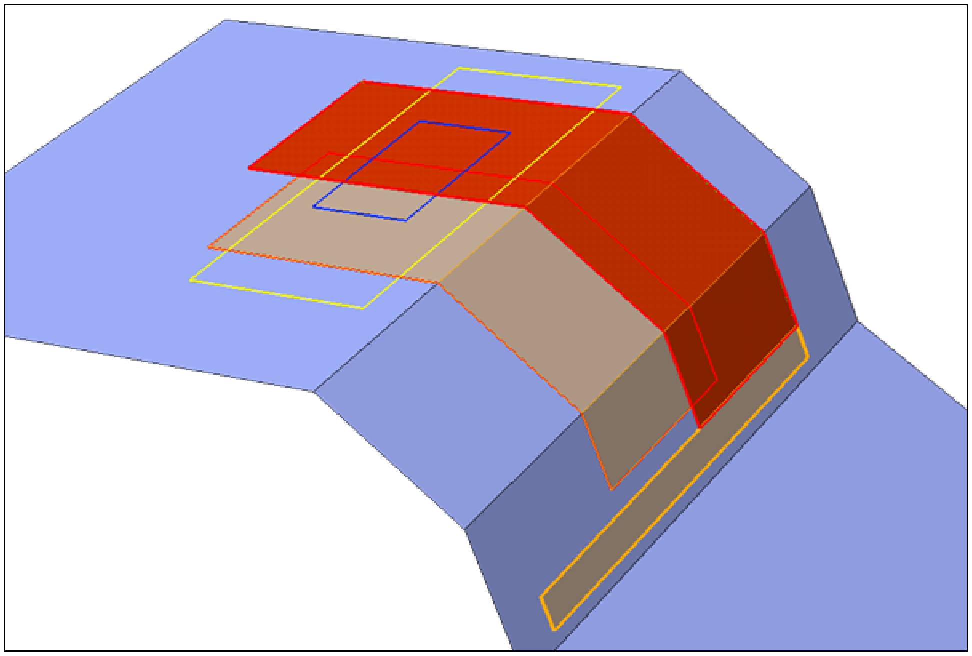
Description

You can now extend the ply boundary by joining it with one or more closed loop contours.



1 Original ply boundary
2 Extension contour
3 Extended ply boundary

You can select an existing closed loop contour. Alternatively, you can create a new closed loop contour on-the-fly using curves and edges.



The extend action is always performed inside the layup quilt.

Benefits

This enhancement provides more flexibility during the Composite Design and preparation for manufacturing.

Additional Information

Tips:	None.
Limitations:	You can extend only plies, not cores.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Core Sample

Creo Parametric 11.0.0.0

User Interface Location: In the Composite Design environment, click **Core Sample**.

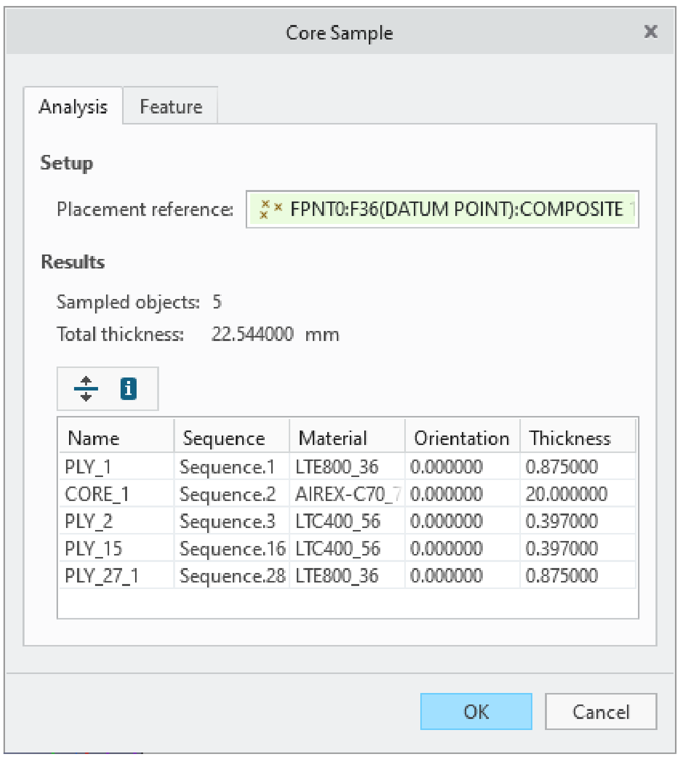
Videos

See the video on the Learning Connector.

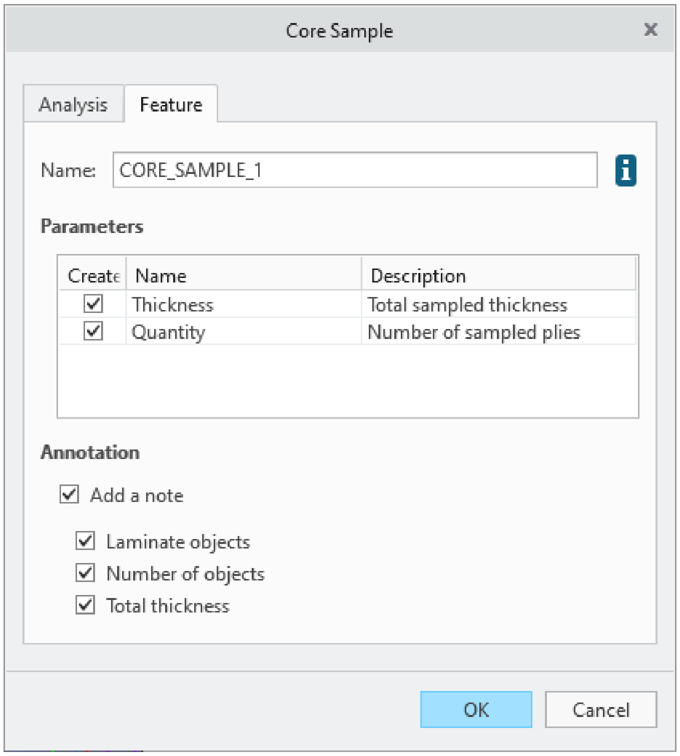
Description

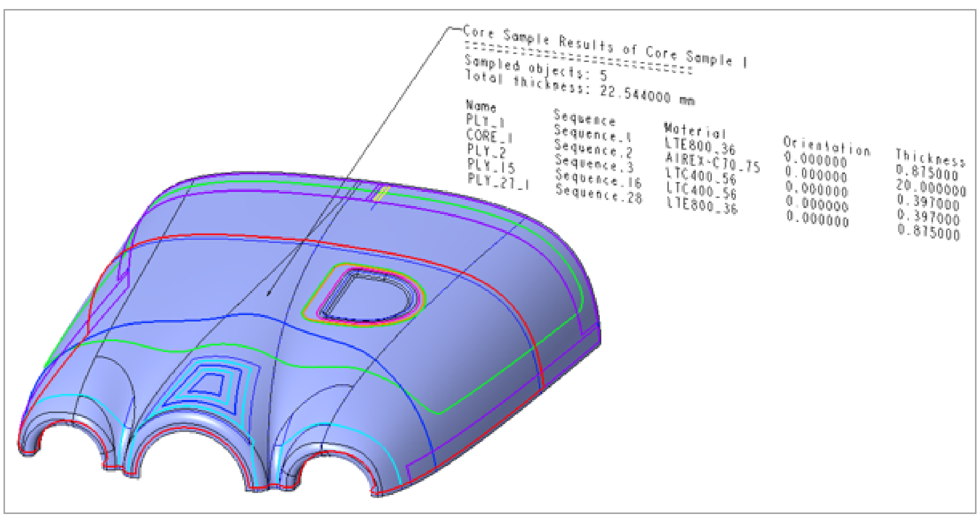
You can now analyze the laminate and get a list of local laminate objects at a selected datum point location using the **Core Sample** command.

When you create a core sample feature, two parameters are provided to report the total sampled thickness and the number of sampled laminate objects in addition to the list of local laminate objects.

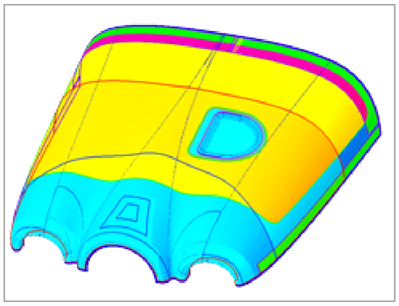


You can also create a note with the core sample feature to display the list of local laminate objects in the graphics window and in a ply book drawing.





You can use the `Thickness` parameter to run a user-defined analysis (UDA) and display a color plot of the laminate thickness on the layup surface.



Benefits

This enhancement provides a better insight into the composite design and adds more clarity to the ply book drawing.

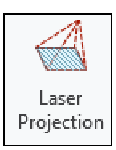
Additional Information

Tips:	None.
Limitations:	Only datum points that are created in the Composite Design environment can be selected.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Laser Projection File Support

Creo Parametric 11.0.0.0

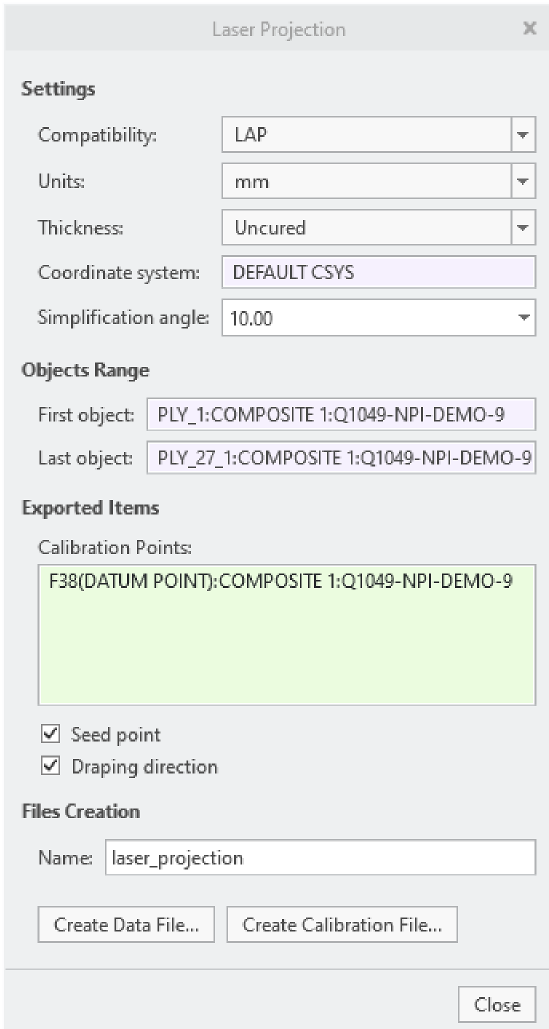
User Interface Location: In the Composite Design environment, click **Utilities** ▸  **Laser Projection**.



Description

You can now create laser projection data files from the composite laminate list to support the laser projection hardware and procedure. The laser projection files can be created for a single, multiple, or all plies and cores, using the `LAP` or `Virtek` file format.

You can create the projection calibration files and the projection data files. Optionally, you can also include the seed point and the draping direction information. To create a laser projection file, you can perform draping simulation based on uncured material thickness.



You can specify the full path to the default directory for storing the laser projection files in one of the following ways:

- Click **File ▸ Options**. In the **Creo Parametric Options** dialog box, click **Applications ▸ Composite**. Specify the values under **Laser Projection**.
- Set the `composite_laser_projection_dir` configuration option.

Benefits

This enhancement supports the laser guided manufacturing process and helps increase the composite product quality.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	New configuration option: • `composite_laser_projection_dir` `<empty>`*

Removing Initial Limitations of the Ply Definition and Solidification

Creo Parametric 11.0.0.0

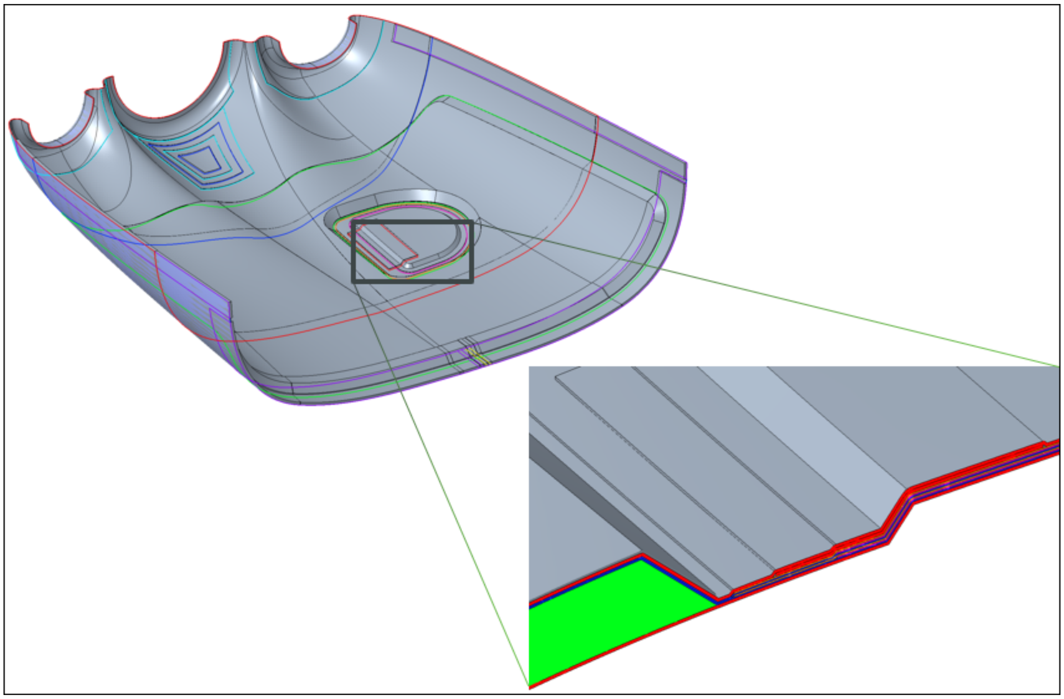
User Interface Location: In the Composite Design environment, do one of the following:

- Click **Ply**.
- Click **Core**.
- Click **Solidify Plies**.

Description

You can now use the outer boundary of the layup quilt to define the boundary of a ply or a core.

You can solidify a laminate even when the plies and cores are crossing sharp or non-tangent edges on the layup surface and when they are defined up to the outer boundary of the layup surface.



Benefits

This enhancement broadens the ways in which you can define the ply boundary and improves the solidification capability.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Highlight and Select from Laminate Section

Creo Parametric 11.0.0.0

User Interface Location: In the Composite Design environment, do the following:

1. Select **Ply** or **Composite Objects** in the selection filter located at the bottom-right corner of the graphics window.
2. Do one of the following:
   - Move the mouse pointer over a ply in a laminate section in the graphics window to highlight the respective ply.
   - Select a ply in a laminate section in the graphics window to select the respective ply.

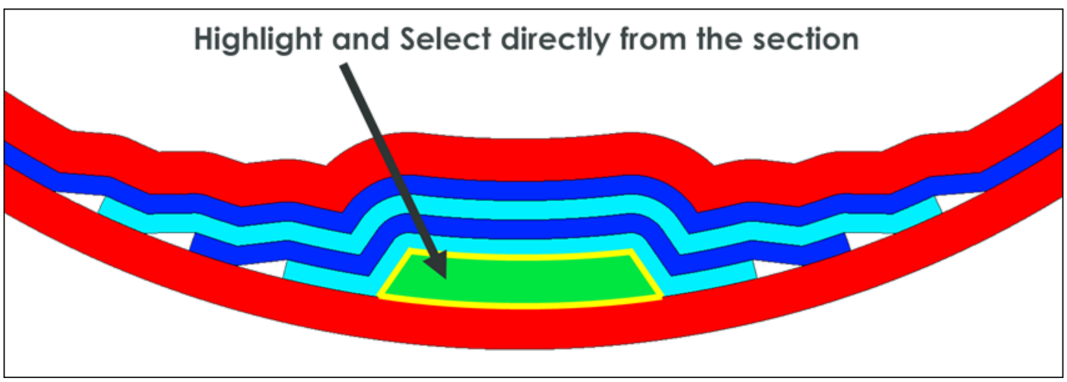
Videos

See the video on the Learning Connector.

Description

You can now highlight and select plies and cores directly from a laminate section.

You can move the mouse pointer over a ply in a laminate section to display its name. To use an object-action approach, you can select a ply directly in the laminate section. When you select the ply in the laminate section, the ply name is highlighted in the Laminate Tree.



Benefits

This enhancement provides an easy way to identify plies in the laminate section.

The direct highlighting and selection from a laminate section provides better insight in the Composite Design environment and helps improve productivity.

Additional Information

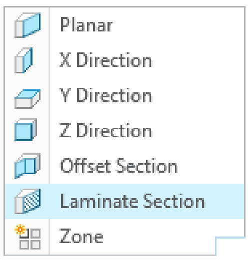
Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Laminate Section at Part Level

Creo Parametric 11.0.0.0

User Interface Location: In a part, select the **View** tab, and then click **Section** ▸ **Laminate Section**.

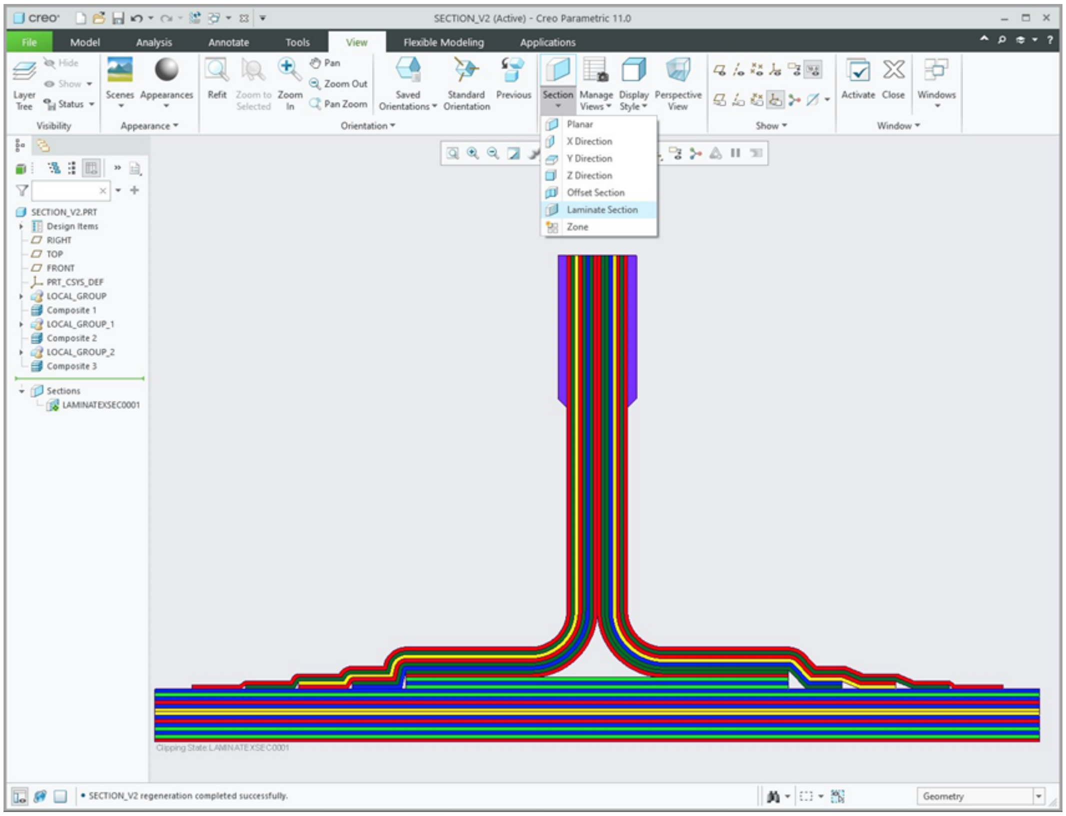


Videos

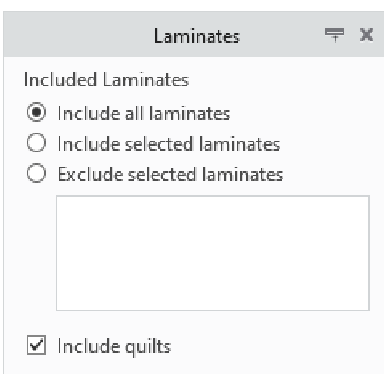
See the video on the Learning Connector.

Description

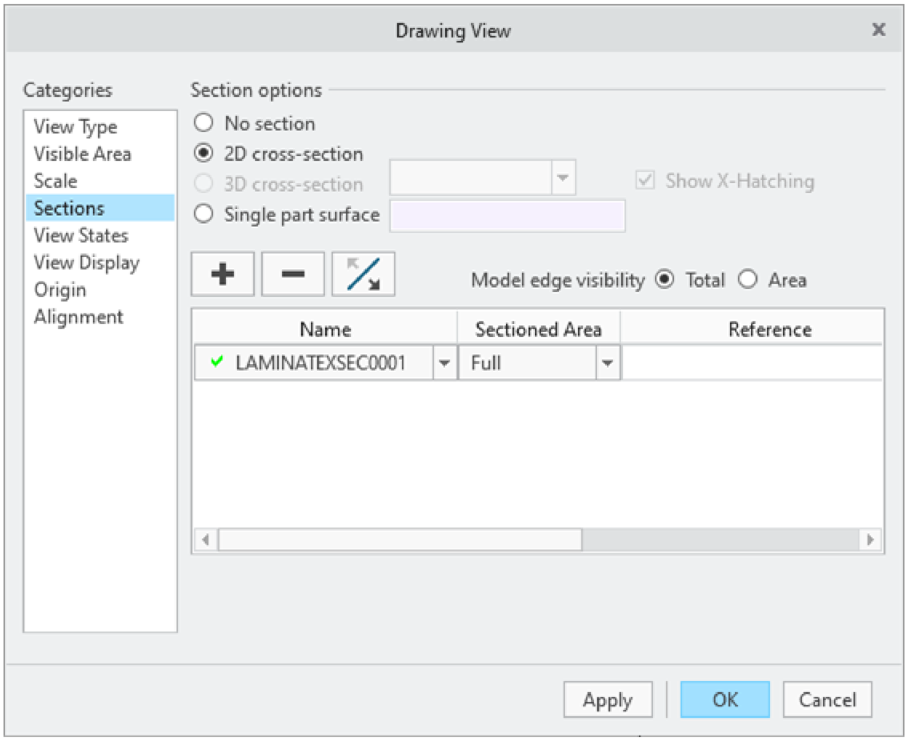
You can now create a laminate section in a part without opening the Composite Design environment.



You can create a laminate section for many composite features at a time and get a holistic view of their section.



You can show it in a normal drawing view like any other section created in a part.



Benefits

This enhancement provides wider access, more insight into the composite design, and an improved ply book and drawing documentation.

Additional Information

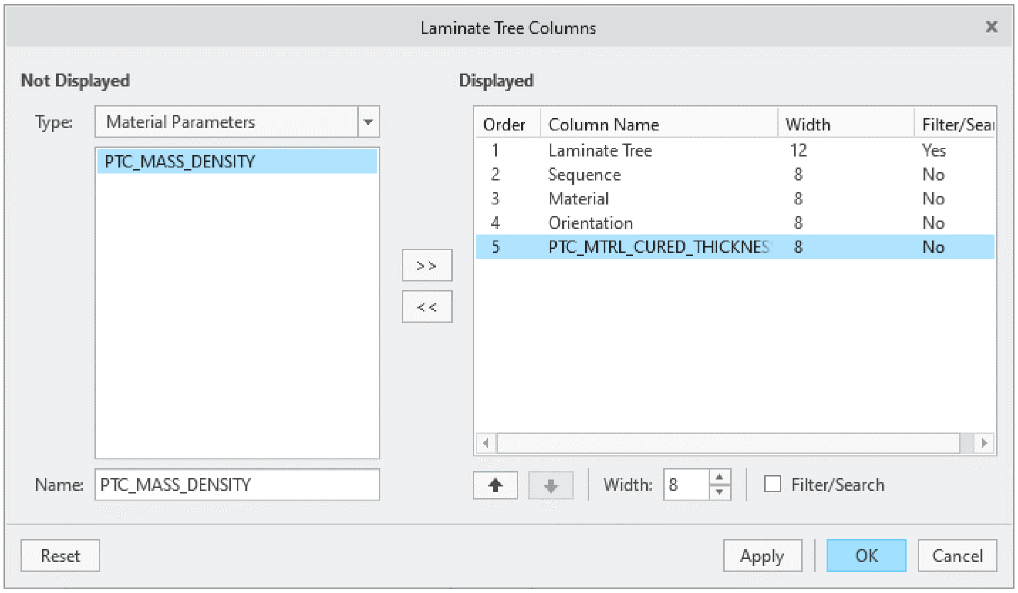
Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Material Parameters in a Laminate Tree Column

Creo Parametric 11.0.0.0

User Interface Location: In the Composite Design environment, do the following:

1. On the Laminate Manager, click **Tree Columns**. The **Laminate Tree Columns** dialog box opens.
2. In the **Type** box, select **Material Parameters**.



Description

When working in the Composite Design environment, you can now display the material parameters, such as the cured thickness of the material, in a column of the Laminate Manager and get better insight into the plies' properties.

Benefits

This enhancement enables you to see the material property next to the ply listed in the Laminate Manager.

Additional Information

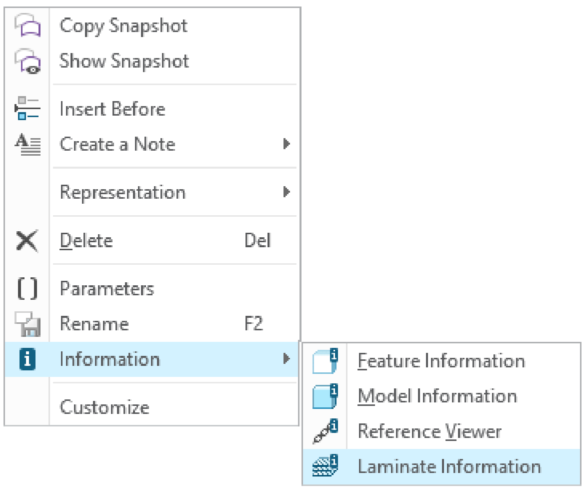
Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Laminate Information from a Composite Feature

Creo Parametric 11.0.0.0

User Interface Location: In a Part, do the following:

1. Right-click a composite feature.
2. Select **Information** ▸ **Laminate Information**.



Description

You can now obtain the laminate information of a composite feature directly from the part. You can perform this activity without entering the Composite Design environment and without a Composite license.

Laminate Information

PART NAME :	LAMINATE-INFO-PART-LEVEL
FEATURE NUMBER :	53
INTERNAL FEATURE ID :	14595
FEATURE NAME :	Composite 1

Laminate Information of Composite Feature Composite 1

Name	Sequence	Material	Orientation
PLY_1	Sequence.1	LTE800_36	0.000000
CORE_1	Sequence.2	AIREX-C70_75	0.000000
PLY_2	Sequence.3	LTC400_56	0.000000
PLY_3	Sequence.4	YE1200_36	0.000000
PLY_4	Sequence.5	YE1200_36	90.000000
PLY_5	Sequence.6	YE1200_36	0.000000
PLY_6	Sequence.7	YE1200_36	90.000000
PLY_7	Sequence.8	YE1200_36	0.000000
PLY_8	Sequence.9	YE1200_36	90.000000
PLY_9	Sequence.10	LTC400_56	45.000000
PLY_10	Sequence.11	LTC400_56	-45.000000
PLY_11	Sequence.12	LTC400_56	0.000000
PLY_12	Sequence.13	LTC400_56	90.000000
PLY_13	Sequence.14	LTC400_56	0.000000
PLY_14	Sequence.15	LTC400_56	90.000000
PLY_15	Sequence.16	LTC400_56	0.000000
PLY_16	Sequence.17	LTC400_56	90.000000
PLY_17	Sequence.18	LTC400_56	90.000000
PLY_18	Sequence.19	LTC400_56	90.000000
PLY_19_1	Sequence.20	YE1200_36	0.000000

Benefits

This enhancement provides you with an easier insight into the Composite Design, without the need to enter the Composite Design environment and without a Composite license.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

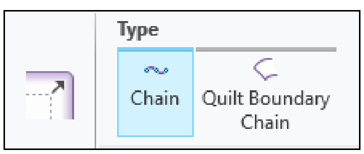
Support for Offset Feature

Creo Parametric 11.0.0.0

User Interface Location: In the Composite Design environment, click **Model** ▸ **Offset**.

Description

You can now use the **Offset** command in the Composite Design environment for creating offset curve features. You can use ply or core boundary as a reference.



Benefits

This enhancement improves productivity inside the Composite Design environment.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

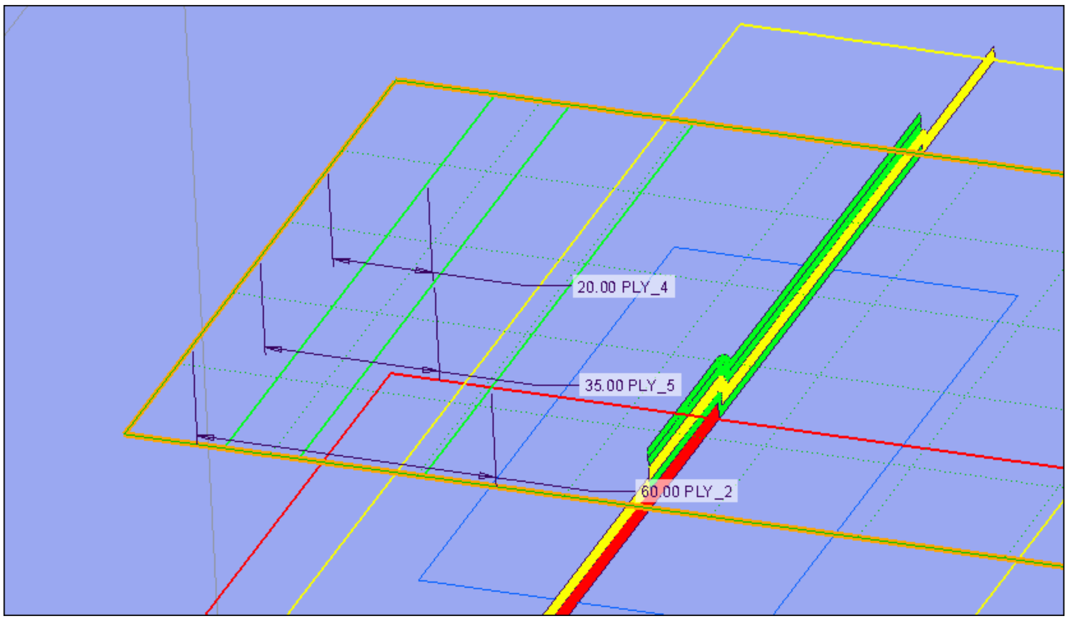
Transition Plies Enhancements

Creo Parametric 11.0.0.0

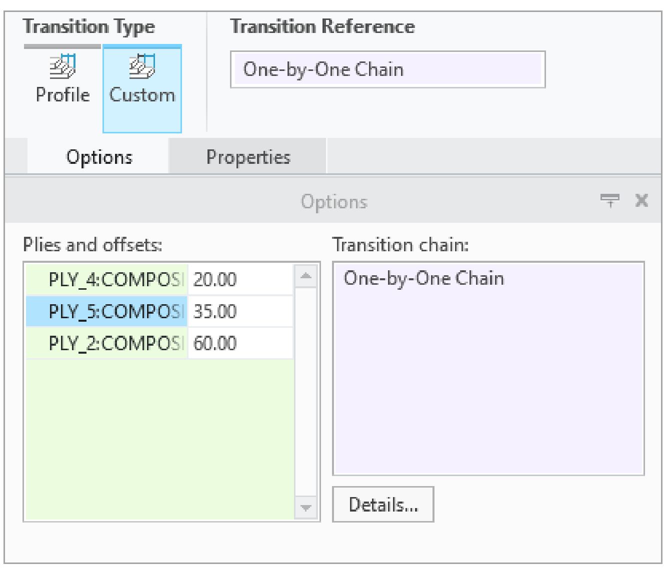
User Interface Location: In the Composite Design environment, click **Plies** ▸ **Transition Plies**.

Description

You can now modify the transition dimensions for the laminate objects directly in the graphics area.



The user interface for the **Custom** transition type is improved, enabling you to define the transition values directly next to the selected laminate objects.



Benefits

This enhancement improves usability and productivity when working with transitions in the Composite Design environment.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Select Related Support in Composites

Creo Parametric 11.0.0.0

User Interface Location: In the Composite Design environment, use one of the following operations:

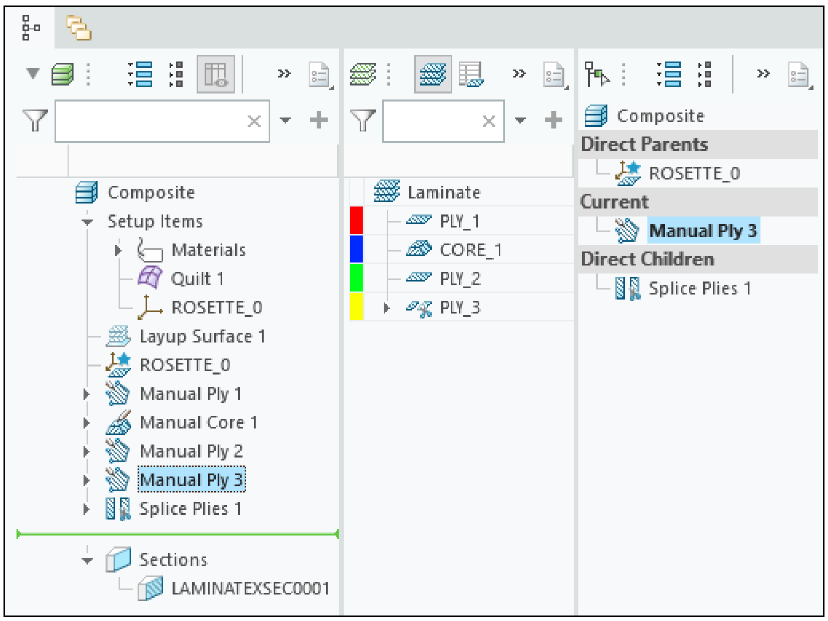
- On the Composite Tree, click **Select Related**, and then select a feature in the Composite Tree.
- Select a feature in the Composite Tree, and then select **Select Related** on the mini toolbar.
- Select a feature in the Composite Tree, and then move the mouse pointer over **Select Related** on the mini toolbar.

Description

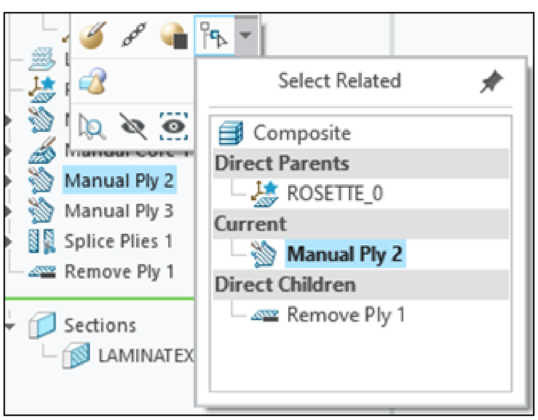
You can now select a feature to see its design relationships in the Composite Design environment.

Previously, when working in the Composite Design environment, there was no easy way to see the direct parents and children of a selected feature. Now, you can see the direct parents and children of the selected feature in one of the following ways:

- When you click the **Select Related** command on the Composite Tree or on the mini toolbar, the Select Related Tree opens. You can then select a feature to see its design relationships.



- When you select a feature and then move the mouse pointer over the **Select Related** command on the mini toolbar, the design relationships of the feature are displayed in a temporary window.



Benefits

This enhancement provides an easy way to investigate the design relationships of a feature.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

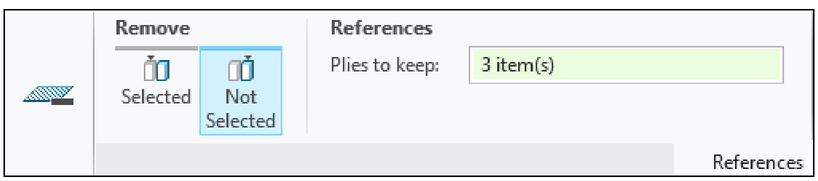
Remove Ply Enhancement

Creo Parametric 11.0.0.0

User Interface Location: In the Composite Design environment, click **Plies** ▸ **Remove Ply**.

Description

You can now use the new **Not Selected** option in the **Remove Ply** tab to remove the plies and cores that are not selected.



Depending on the situation, the new definition scheme can help define the desired outcome with fewer selections. This enhancement provides a more robust way to handle specific situations such as, when there are more plies to be removed than the plies to be included in the Composite Design.

Benefits

This enhancement provides expanded use cases for removing plies and cores when working in the Composite Design environment.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

7

Creo Harness Manufacturing

3D Edit Retention

Creo Parametric 11.0.0.0

Description

HMX now supports a fully iterative design process.

With 3D Edit Retention, you can edit the harness in the three-dimensional assembly by adding, deleting, or editing wires, cables, and connectors while retaining the previous edits made to the drawing.

Benefits

This enhancement significantly reduces the time spent on post drawing cleanup activities.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Productivity Enhancements in HMX

Creo Parametric 11.0.0.0

Description

The following capabilities have been improved in HMX:

- Splice handling for assemblies with increased splice complexities.
- Additional table customizations.
- Retention of user generated tables.
- Selection of starting connectors for flattening a harness.
- Retention of selected note types.
- Customization of drawing names.

Benefits

The enhancements made to these capabilities improve productivity and user experience.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

8

Creo Toolkit

Improvements to the Creo Toolkit Help

Creo Parametric 11.0.0.0

Description

The experience for using Creo toolkits has been enhanced as follows:

- Creo TOOLKIT deliverables for the supported releases are now available from 404-Redirect | PTC. Click **Creo Toolkit Help** page that lists the Creo TOOLKIT deliverables.
- A new online format is now available for the following Creo toolkits:
  - `otk_cpp`—Creo Object TOOLKIT C++
  - `otk_java`—Creo Object TOOLKIT Java

  This Help Center is provided in addition to the APIWizard and PDFs that are currently supported. You can start typing your search phrase in the Help Center and the results are immediately highlighted and available for selection from the search results list.

Benefits

The landing page for Creo TOOLKIT Help enables you to access the toolkit deliverables from a single location, improving the ease of access to the deliverables. The Help Center for the Creo Object TOOLKIT C++ and Creo Object TOOLKIT Java combines information from the APIs and User's Guide and provides improved search.

Additional Information

The following table provides a full list of deliverables supported for the Creo toolkits.

Toolkit	APIWizard	User's Guide (PDF)	Link to Help Center
protoolkit	<creo_toolkit_loadpoint>\protkdoc\index.html	Creo® Parametric TOOLKIT User's Guide	NA
otk_cpp	<creo_otk_loadpoint_doc>\objecttoolkit_Creo\index.html	Creo Object TOOLKIT C++ User's Guide	Creo Object TOOLKIT C++ Help Center
otk_java	<creo_otk_java_loadpoint_	Creo Object TOOLKIT JAVA User's Guide	Creo Object TOOLKIT JAVA

Toolkit	APIWizard	User's Guide (PDF)	Link to Help Center
	doc>\ objecttoolkit_ Creo\ index.html		Help Center
creojs	<creojs_ loadpoint>\ creojsdoc\ index.html	Creo.JS User's Guide	NA
pfcweblink	<creo_weblink_ loadpoint>\ weblinkdoc\ index.html	Creo® Parametric Web.Link™ User's Guide	NA
pfcvb	<creo_vbapi_ loadpoint>\ vbapidoc\ index.html	Creo® Parametric VB API User's Guide	NA

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Ability to Generate Check-In Comments Using the Toolkit API

Creo Parametric 11.0.0.0

Description

A new public API is added to enable the Creo TOOLKIT application to generate check-in comments when storing data back into Windchill.

Benefits

When storing Creo data back into Windchill, it is essential to document the history of every object. Typically, check-in comments are leveraged for this purpose. If you use Creo TOOLKIT to perform automated workflows to check data back into Windchill, it could be difficult to automatically capture comments for multiple objects.

To automate this workflow, a new public API is introduced in Creo TOOLKIT. This API enables the generation of check-in comments when checking Creo data back into Windchill. These comments are accessible under the **History** tab in Windchill.

```
extern ProError ProServercheckinoptsCommentSet
(ProServerCheckinOptions
opts, wchar_t* checkin_comment);
/*
ProServercheckinoptsCommentSet
Purpose: Sets the history comment for checkin
Input Arguments:
opts - The checkin/upload options.
checkin_comment - The history comment for checkin
Output Arguments:
none
Return Values:
PRO_TK_NO_ERROR - The function succeeded.
PRO_TK_BAD_INPUTS - One or more arguments was invalid.
*/
```

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

9

Data Exchange

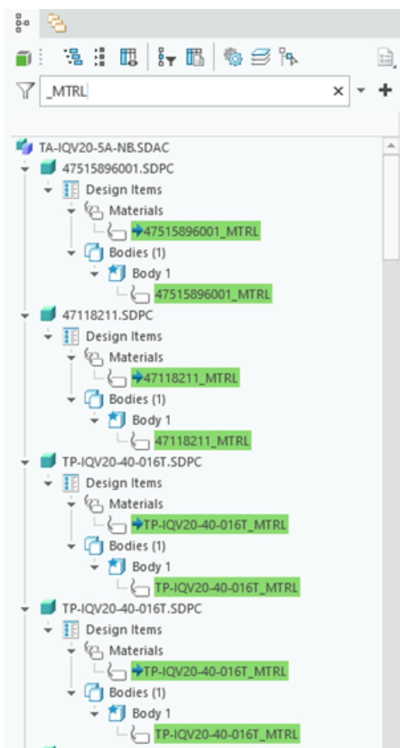
Density Transfer from Creo Elements/ Direct to Creo Parametric

Creo Parametric 11.0.0.0

User Interface Location: **File ▸ Import**, or **File ▸ Open**.

Description

In Creo Parametric 11.0.0.0, the transfer of density information from Creo Elements/Direct to Creo Parametric is optimized. The **Open** and **Import** workflows automatically create a dedicated material for each imported part. The material density is set to the *Base Density* of the respective source part. The material name is derived from the source part name, by adding the suffix *_MTRL* to each part name. In addition, the part parameter `PTC_MASTER_MATERIAL` is automatically set to the created material. With this enhancement, the original density information for imported models is leveraged automatically.



Benefits

Density information was previously transferred only as a parameter that had to be linked via a relation to the target model density. This was a cumbersome process, and it required a regeneration when opening the model.

Now, with the enhanced interoperability between Creo Elements/Direct and Creo Parametric , Creo Elements/Direct data can be leveraged easily in Creo Parametric. You no longer need to use relations in import templates to assign density.

Additional Information

Tips:	In addition to the previous parameter *CED_DENSITY*, there is now a new component parameter called *CED_COMPONENT_INSTANCE_DENSITY*. This parameter is set to the instance-level density value.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Treating out-of-date Creo Unite models as Missing Components

Creo Parametric 11.0.0.0

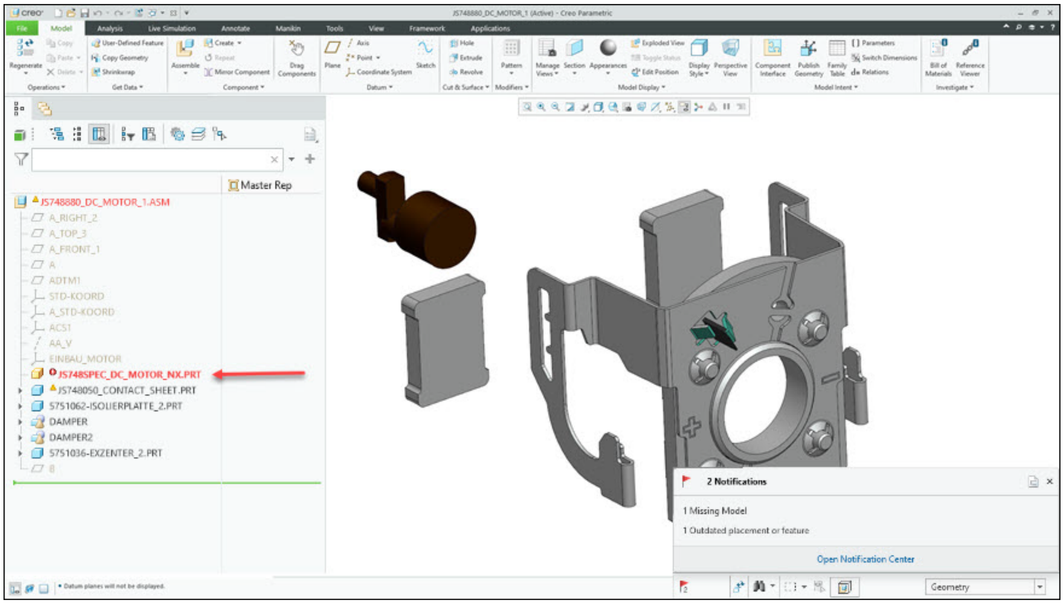
User Interface Location: Click **File** ▸ **Open**.

Videos

See the video on the Learning Connector.

Description

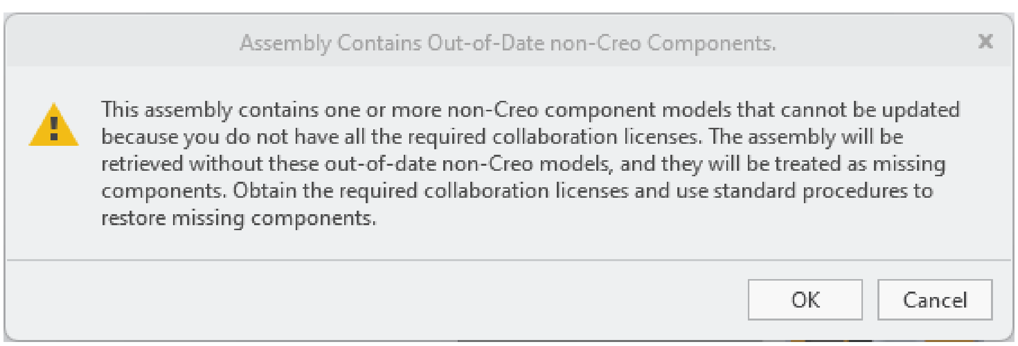
Creo Parametric 11.0.0.0 has improved the behavior with out-of-date non-Creo wrapper models opened with the Creo Unite technology. When a collaboration license is unavailable, out-of-date non-Creo wrappers will be represented as missing components. Previously, when the collaboration license was not available, the Creo Unite wrapper was not updated if the non-Creo model had been modified, and Creo Parametric retrieved the non-updated wrapper. As a result, users occasionally overlooked the warnings and continue using an outdated representation of the model.



To mitigate the risk of using an out-of-date wrapper when the collaboration license is unavailable, Creo Parametric 11.0.0.0 introduces a new configuration option to control this behavior:

`intf3d_open_outofdate_unite`—`yes`*, `no`, `no_with_warning`

- `yes`—Retrieves out-of-date wrapper files without a collaboration license (default behavior).
- `no`—Limits retrieving out-of-date wrapper files without a collaboration license and shows that component as missing.
- `no_with_warning`—Provides a notification message when retrieving out-of-date wrapper files without a collaboration license.



Benefits

- Provides a configuration option to control the behavior when opening a non-Creo model without a collaboration license. Enhances clarity for users that assume they are getting the latest representation of non-Creo model, when in fact they are retrieving a previous out-of-date representation.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	Yes. Previously, without the collaboration license, the Creo Unite wrapper was not updated if the non-Creo model had been modified and Creo retrieved the out-of-date wrapper.
Configuration option associated with this functionality:	`intf3d_open_outofdate_unite`

10

Data Management

Commonspace Folder Directly Links to the Primary Active Workspace Context

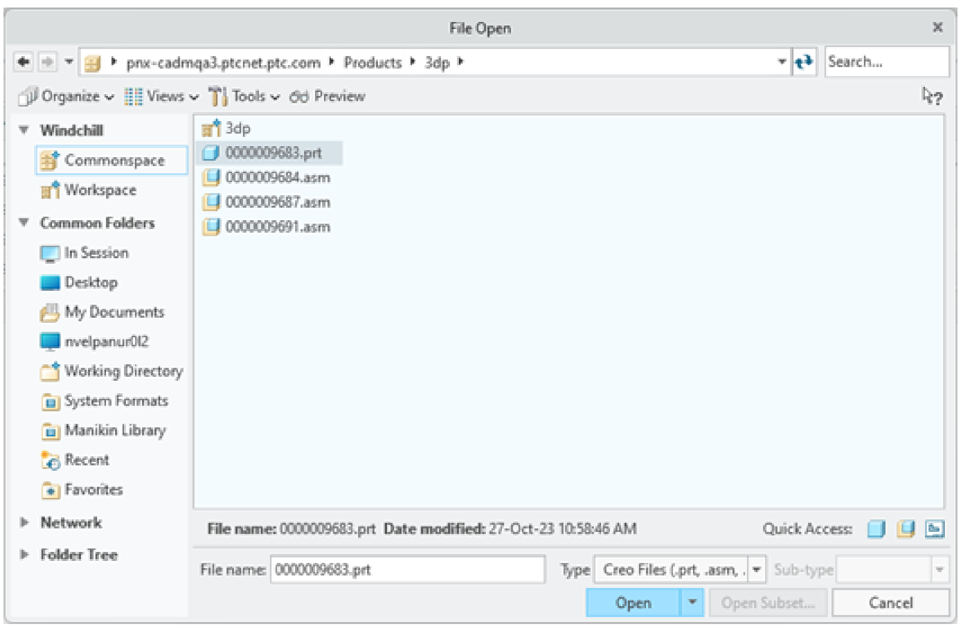
Creo Parametric 11.0.0.0

User Interface Location: Click **File** ▸ **Open** or Folder Navigator.

Description

Previously, when you wanted to load models in a Creo session and navigated to the Commonspace folder, it opened the top-level context instead of the active workspace context.

In this release, when connected to a Windchill session, navigating to the Commonspace folder using either **File** ▸ **Open** or the Folder Navigator, opens the active primary workspace context by default.



Benefits

This enhancement improves navigation and the user experience.

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

11

Detailed Drawings

Enhancement: Change the Default Value of the Configuration Option create_ drawing_dims_only to yes

Creo Parametric 11.0.0.0

Description

The default value for the existing config.pro option `create_drawing_dims_only` has been updated to `Yes`. This purpose of this enhancement is to set a more frequently used value and to better align with the recommended Model-Based Design practices.

Benefits

The default value for the `create_drawing_dims_only` option is now aligned with the common usage.

Additional Information

Tips:	If preferred, you can continue setting the configuration option value to `No`. Both values are acceptable.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	`create_drawing_dims_only`

12

ECAD

ECAD Context Data Explorer Enhancements

Creo Parametric 11.0.0.0

User Interface Location: In an ECAD assembly, click **Model** ▸ **ECAD Context Data Explorer**.

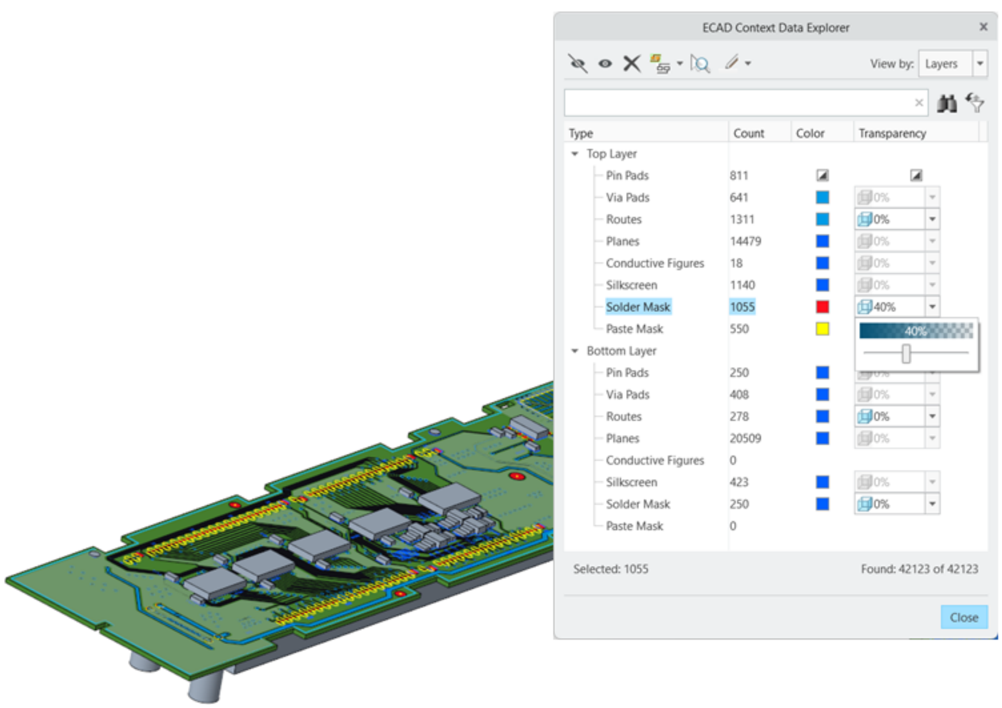
Videos

See the video on the Learning Connector.

Description

Two new columns were added to the ECAD Context Data Explorer, one for color and one to set transparency:

- Colors can be applied to ECAD layers and to entities inside the Nets. A mixed state is added to show that a specific layer can have more than one color applied to entities on it. This can be changed by selecting one color for the whole layer.
- Transparency control for surface and quilt representations. Transparency control lets you choose an opacity value from 0 to 100 percent for any ECAD layer that uses the supported representations. This helps when layers overlap with each other or cover the board geometry that you want to see.



Benefits

- Improved visualization of overlapping ECAD layers with the help of transparency.
- Better visibility of ECAD layer colors.
- Greater clarity of layer color coding.

Additional Information

Tips:	None.
Limitations:	Transparency is only available for surface and quilt representations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

13

Fundamentals

Changed Default Values of Configuration Options to Improve the Display Quality

Creo Parametric 11.0.0.0

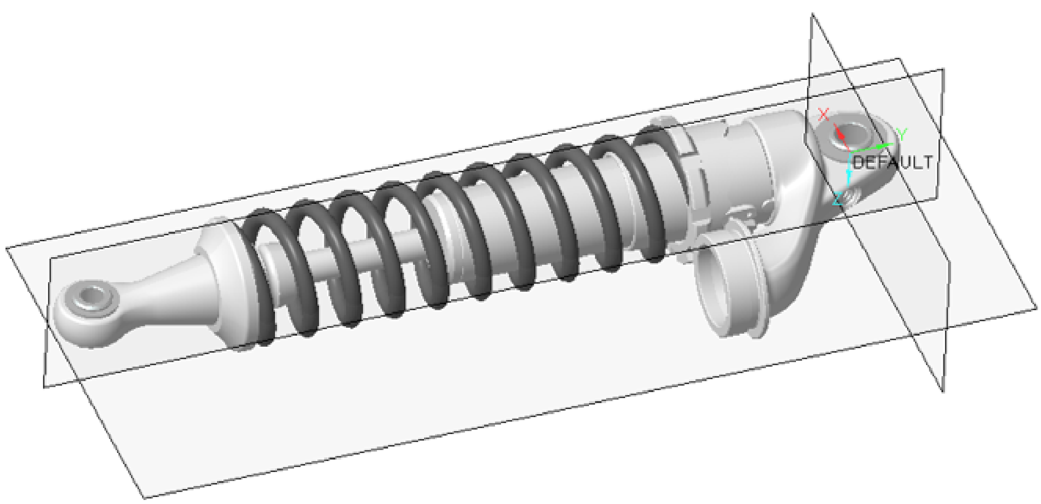
Description

The default values for a set of configuration options have been changed to improve the display quality and enhance the out-of-the-box experience.

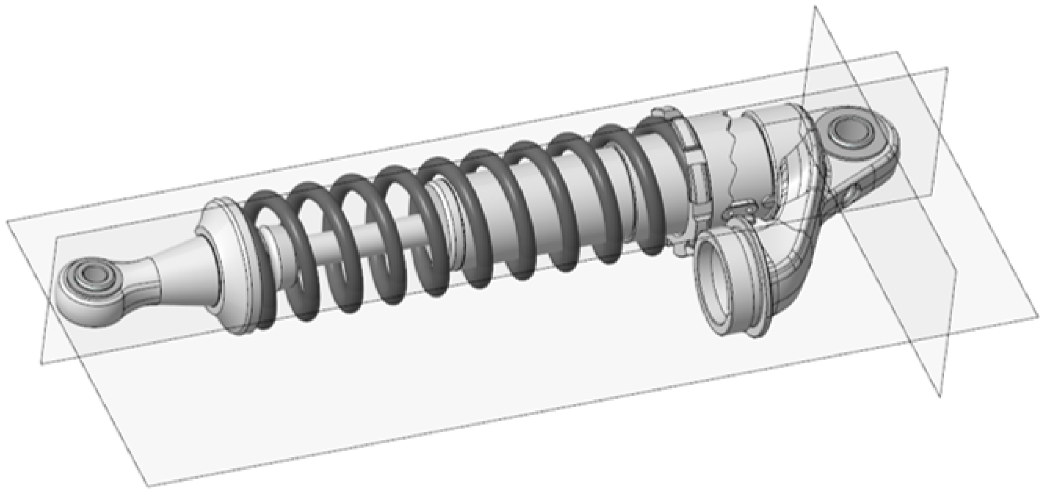
This enhancement makes the following changes:

- Increases the tessellation quality for edges
- Increases the shade and edge quality
- Turns off the display of axes, points, coordinate systems, and a spin center
- Dims the edges between tangent surfaces
- Enables the Full Screen Anti-Aliasing (FSAA) functionality
- Prehighlights the Model Tree, the Layer Tree, or the 3D Detail Tree beneath the pointer
- Displays datum features during dynamic spinning

The following graphic shows the default display of a model in Creo Parametric 10.0.0.0:



The following graphic shows the default display of the same model in Creo Parametric 11.0.0.0:



For more information on the changed options and the new values, see Support Article CS402384.

Benefits

This enhancement improves the display quality and the overall user experience.

Additional Information

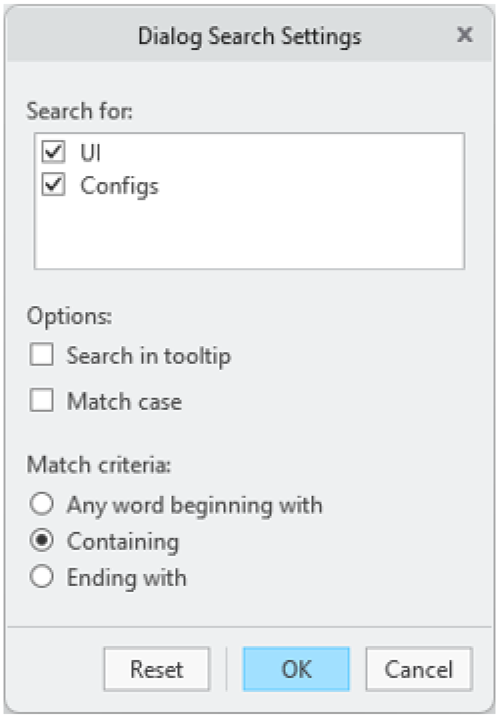
Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	Changed the default values of the following configuration options: • `display` `shade`, `wireframe`, `hiddenvis`, `hiddeninvis`, `shadewithedges`*, `shadewithreflect` • `display_axes` `yes`, `no`* • `display_coord_sys` `yes`, `no`* • `display_points` `yes`, `no`* • `edge_display_quality` `low`, `normal`, `high`*, `very_high` • `edge_tess_quality` `high`*, `medium`, `low` • `enable_fsaa` `off`, `2`, `4`, `8`*, `16` • `prehighlight_tree` `yes`*, `no` • `shade_quality` `8`* • `spin_center_display` `yes`, `no`* • `spin_with_part_entities` `yes`*, `no` • `visible_message_lines` `1`*, `<integer>` • `tangent_edge_display` `solid`, `no`, `centerline`, `phantom`, `dimmed`*

Search Functionality in Creo Options User Interface

Creo Parametric 11.0.0.0

User Interface Location: Click **File** ▸ **Options**.



Videos

See the video on the Learning Connector.

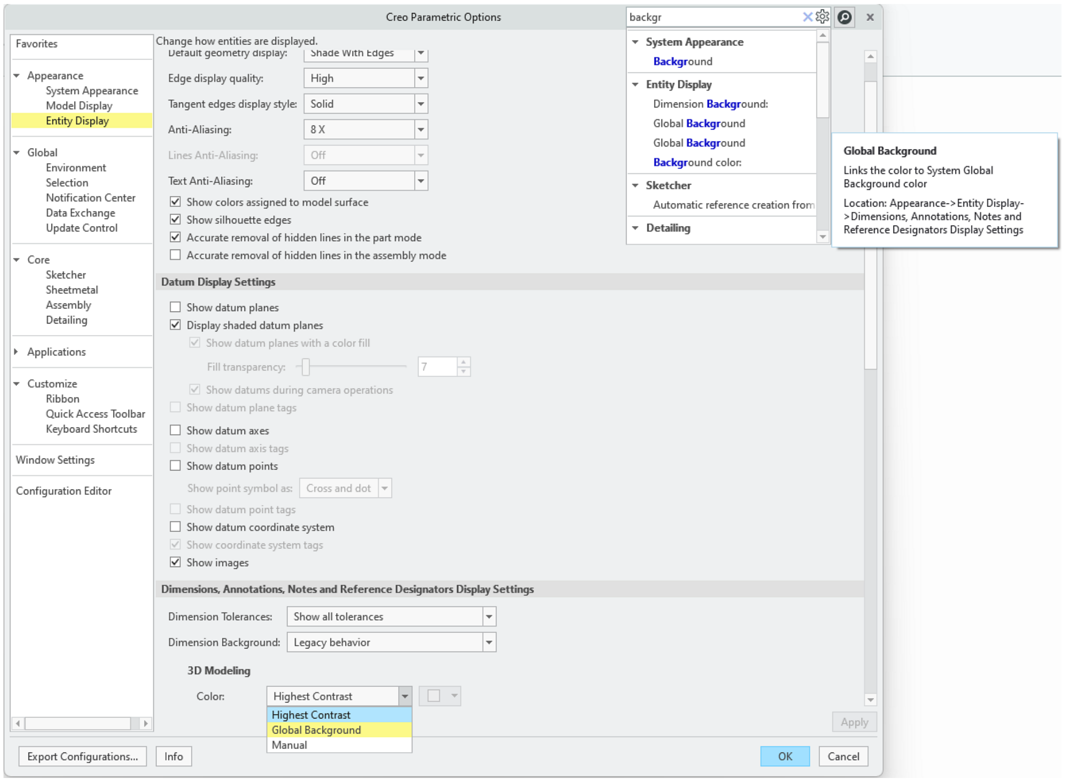
Description

A search tool has been added to the Creo Options dialog box. With this enhancement you can do a quick search for options and configuration options within the dialog box.

You can customize the search criteria and behavior from the search settings dialog box.

You can search by option name, description, tooltip, label, or value. The search results appear after you type at least two characters. When you hover a pointer over the items in the results list, it shows the matching options and highlights them on the pages where they are located.

Additionally, the search results also show the additional config.pro options that are not present in the user interface. You can change their settings directly from the results list.



Benefits

Improved user experience when working in Creo Options dialog box.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

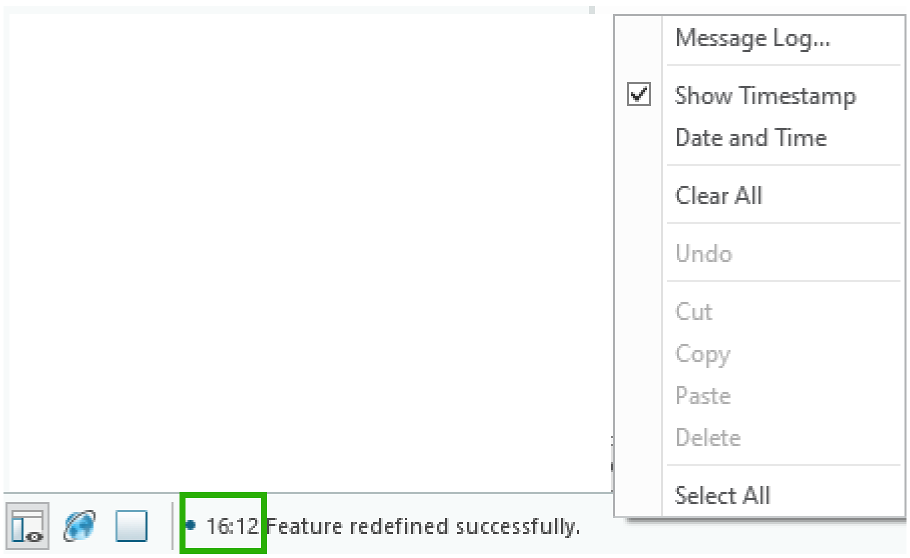
Enhancement: Message Time Stamp in the Status Bar

Creo Parametric 11.0.0.0

User Interface Location: Status bar in Creo.

Description

The status bar in Creo 11 has been enhanced to show a time stamp for the system messages. With this enhancement you can determine the precise time when a message was generated. To select the display of the time stamp, right-click in the message area and then select the **Show Timestamp** checkbox. Your preference will be saved in the `.ui` customization file under the category `General>Others`



Benefits

Easier review and better understanding of time context in message log.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Improved Rename Workflow in the Model Trees

Creo Parametric 11.0.0.0

User Interface Location: Model Tree, Layer Tree, Design Tree, Quilt/Body Evolution Tree.

Videos

See the video on the Learning Connector.

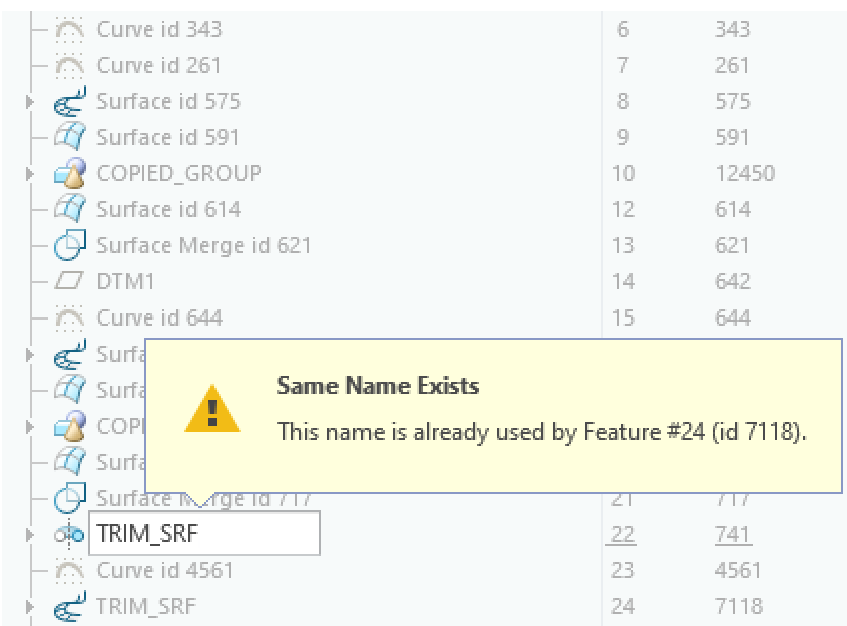
Description

The rename workflow has been enhanced for better handling of naming conflicts.

The rename workflow has been enhanced to verify for any naming conflicts and identify special characters that are incompatible with the model tree names. Previously, a conflicting name was rejected, and the system reverted to the original name, requiring you to run the rename workflow again.

The enhancements to handling of the naming conflicts include:

- Warning message— When a newly entered name results in a conflict, a warning message is displayed, providing information about the conflict. This includes details about:
  - Duplicate name
  - Unsupported name characters
- Persistent dialog box—The rename dialog box remains open and retains the entered name, allowing you to modify.
- Automatic character conversion—If a newly entered name contains spaces, they are automatically converted to underscores ("_").



Benefits

Improved usability when renaming model tree items.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Improved Expand and Collapse Workflows for all Navigation Trees

Creo Parametric 11.0.0.0

User Interface Location: All Navigation Trees in Creo on page such as Model Tree, Layer Tree, Design Tree, Quilt/Body Evolution Tree, Drawing Tree, Design Objects Tree and so on.

Videos

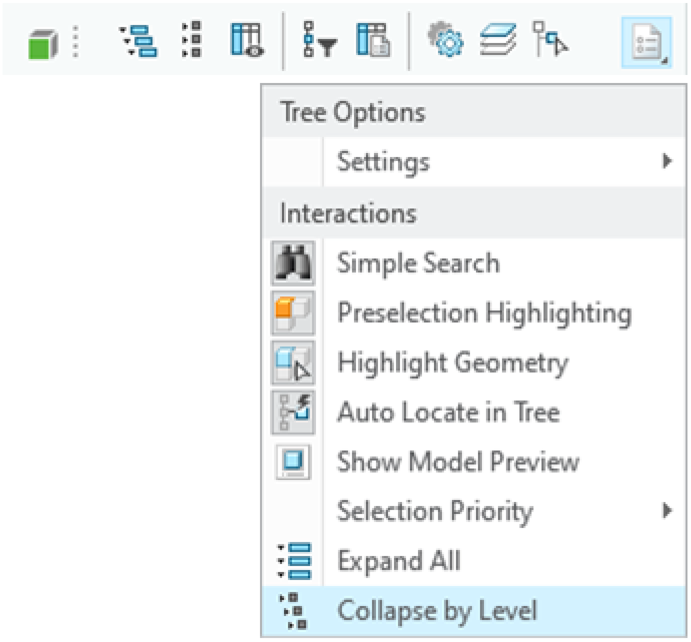
See the video on the Learning Connector.

Description

The expand and collapse workflows have been enhanced to improve usability. The enhancements include:

- Addition of two new commands **Expand by Level** and **Collapse by level** to all the navigation trees:
  - **Expand by Level**—Incrementally expands the selected branch by one level with each click.
  - **Collapse by level**—Incrementally collapses the selected branch by one level with each click.
- Change in **Collapse All** actions:
  - Clicking **Collapse All**, keeps the top node and first level node visible and collapses the rest.
  - The shortcut SHIFT+ tree expander arrow now invokes the expand branch and collapse branch, respectively.
- Change the location of the following commands:
  - **Expand by Level** and **Collapse by level** are available in the tree toolbar, as more frequently used commands.

- **Expand by Level** and **Collapse by level** are located under the tree options.



Benefits

Improved usability when working with the navigation trees.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Improved Sorting in the File Open Dialog Box

Creo Parametric 11.0.0.0

User Interface Location: Click **File** ▸ **Open**.

Videos

See the video on the Learning Connector.

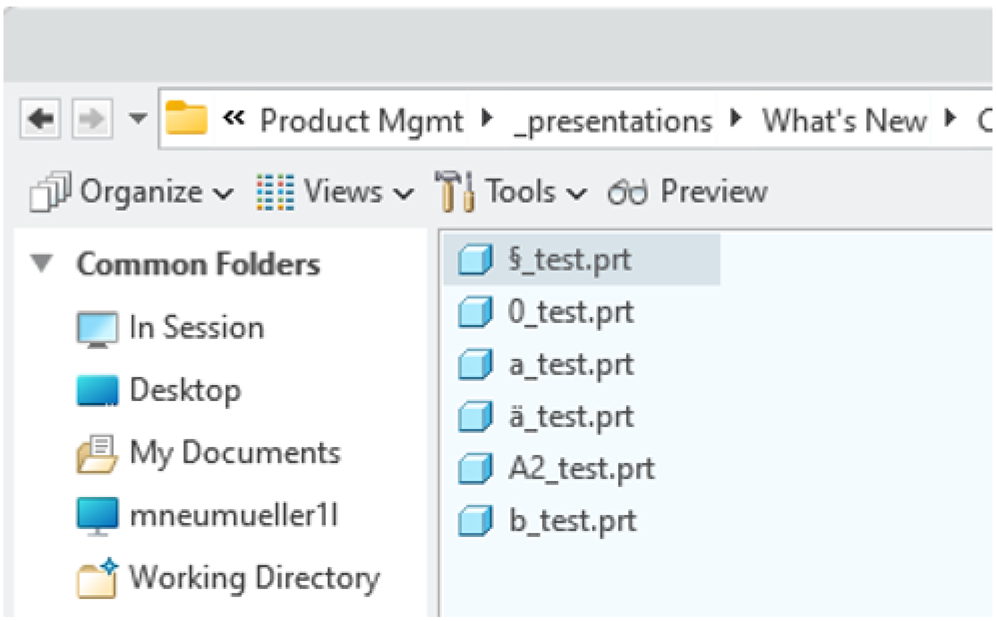
Description

In Creo 11, sorting in **File Open** dialog box has been improved to align with the sort order used in the Microsoft Windows environments.

Previously, the file sorting in the **File Open** dialog box differed from the commonly used sorting method used in the Microsoft Windows environments.

When displaying file names in an alphabetical order:

- The list will begin with file names starting with special characters and numbers.
- The list will continue with file names in the alphabetical order, without grouping them into uppercase and lowercase characters.



Benefits

Improved user experience when navigating files within applications.

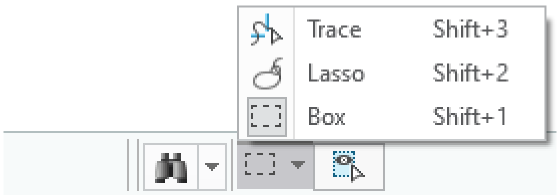
Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Enhanced Surface Selection Capability

Creo Parametric 11.0.0.0

User Interface Location: Status bar in Creo.



Videos

See the video on the Learning Connector.

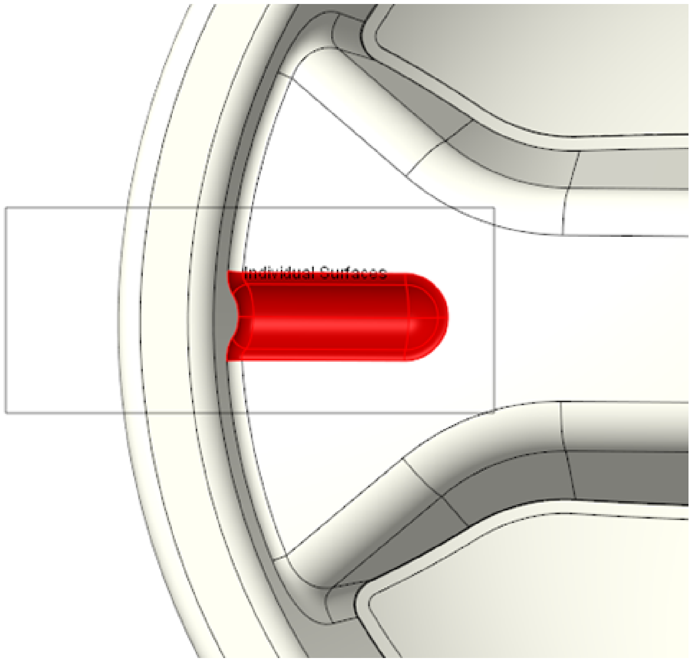
Description

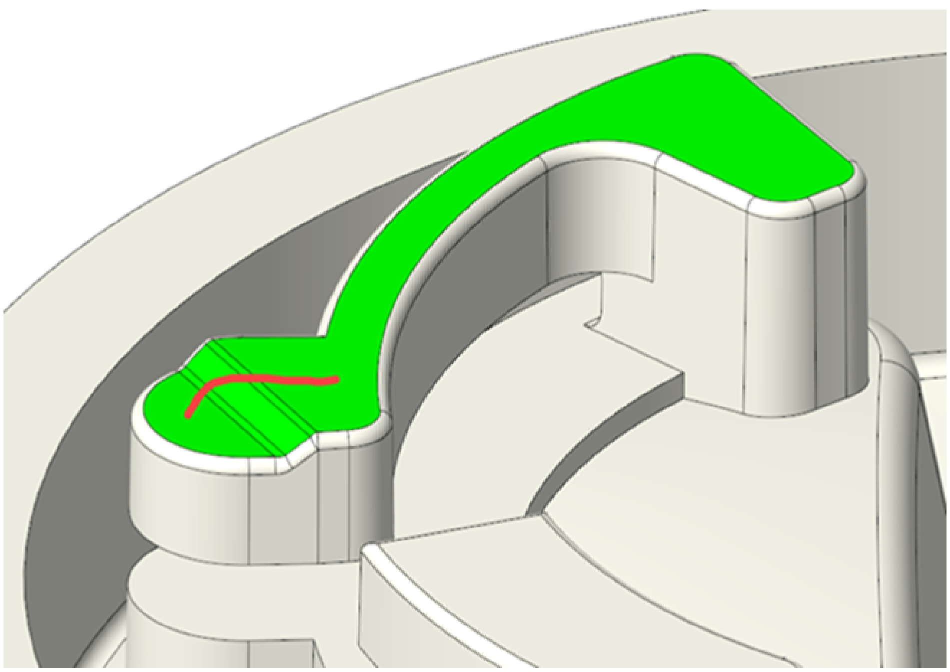
In Creo 11, the new region selection capabilities, Lasso Selection and Trace Selection, have been added, and the existing Box Selection has been enhanced for surfaces in Part and Assembly mode.

The enhancements in the selection method include:

- Box Selection
  - Selects surfaces enclosed within the selection box.
  - Flexibility to control the selection of only visible surfaces or surfaces that are hidden by other surfaces within the box.
- Lasso Selection
  - Selects surfaces enclosed within the defined loop.
  - Selects only visible surfaces.
- Trace Selection
  - Selects surfaces that are traced over.
  - Selects only visible surfaces.

Additionally, the selection methods support object/action workflows, making them available for all surface sets collectors and for the surface collector of the Color Tool. For example, you can use the Trace selection to select Boundary Surfaces in the Seed & Boundary selection rule.





Benefits

- Faster selection of multiple individual references for coloring.
- Easier selection of semantic references for 3D annotation in the Model-Based Definition.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Enhancement: Separate Mapkeys Configuration

Creo Parametric 11.0.0.0

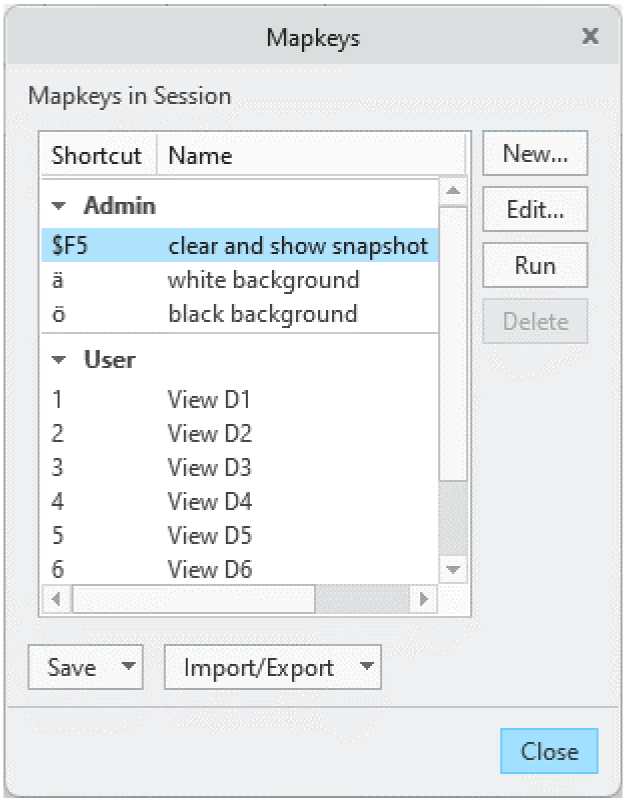
User Interface Location: Click **File** ▸ **Options** ▸ **Mapkeys Settings**. The Mapkeys dialog box opens.

Description

In Creo 11, you can now manage your mapkeys in a separate new mapkeys.pro file. Previously, your mapkeys were stored in the config.pro file. To maintain compatibility, the new file contains all mapkey definitions with the same syntax as in the previous versions.

The enhancements to the mapkeys include:

- Two levels of mapkeys that can be saved in two locations:
  - Admin-level mapkeys: within the installation directory structure.
  - User-level mapkeys: in the user settings directory.
- Both the user and admin mapkeys are loaded and shown in the mapkeys dialog box, grouped by their level.
- If the mapkeys.pro file does not exist, the mapkeys are loaded from the config.pro file, and the mapkey definitions are saved in a new mapkeys.pro file.
- Save, import, and export your mapkeys using new commands in the dialog box. The mapkeys dialog now includes additional commands:
  - **Save Changed**—Saves changes made in the user mapkeys.
  - **Save Selected**—Saves the selected user mapkeys.
  - **Import**—Imports a previously saved mapkey file.
  - **Export Selected**—Exports the selected mapkeys.
  - **Export All**—Exports all user mapkeys.



Benefits

- Increased flexibility for managing Creo Parametric configuration and mapkey settings.
- Avoids clutter in the highly customized config.pro files that include mapkeys.
- Separation of Creo Parametric settings and mapkey settings.

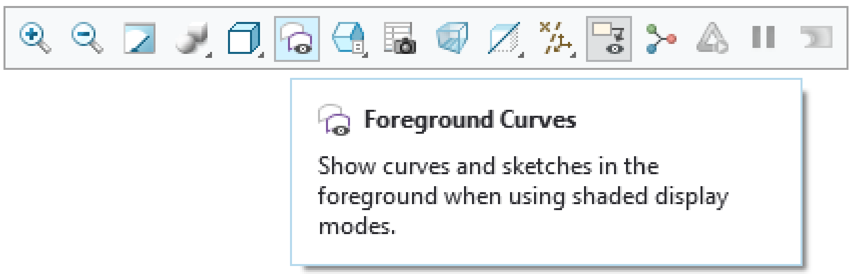
Additional Information

Tips:	Administrator mapkeys can be changed in the Creo session but not saved.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

New Display Setting in the In-Graphics-Toolbar

Creo Parametric 11.0.0.0

User Interface Location: In-graphics toolbar



Videos

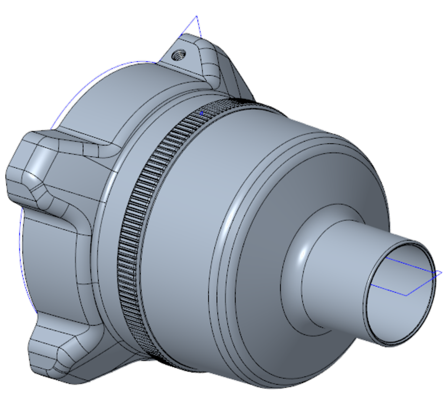
See the video on the Learning Connector.

Description

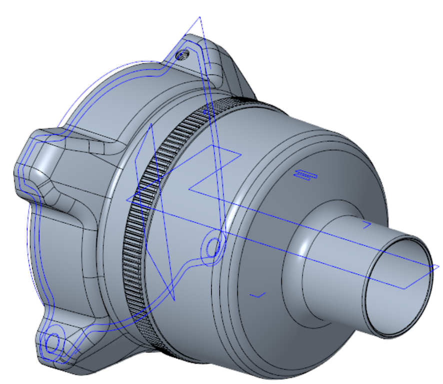
A new display setting **Foreground Curves** has been added to the in-graphics toolbar.

When selected, all curves are fully displayed in the foreground, making it easier to see and select curves during design workflows.

You can customize the appearance of the display setting from the context menu. To configure the default display behavior, navigate to the Options dialog box (**File ▸ Options**). In the Options dialog box, click **System Appearance ▸ Model Display ▸ Shaded Model Display Settings ▸ Show curves or sketches in foreground when using shaded display modes**.



Standard curves display



Curves displayed in foreground

Benefits

Easier selection of curves and sketches in design workflows.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	`foreground_curves.*no`, `yes` Shows visible curves in the foreground when in shaded display modes.

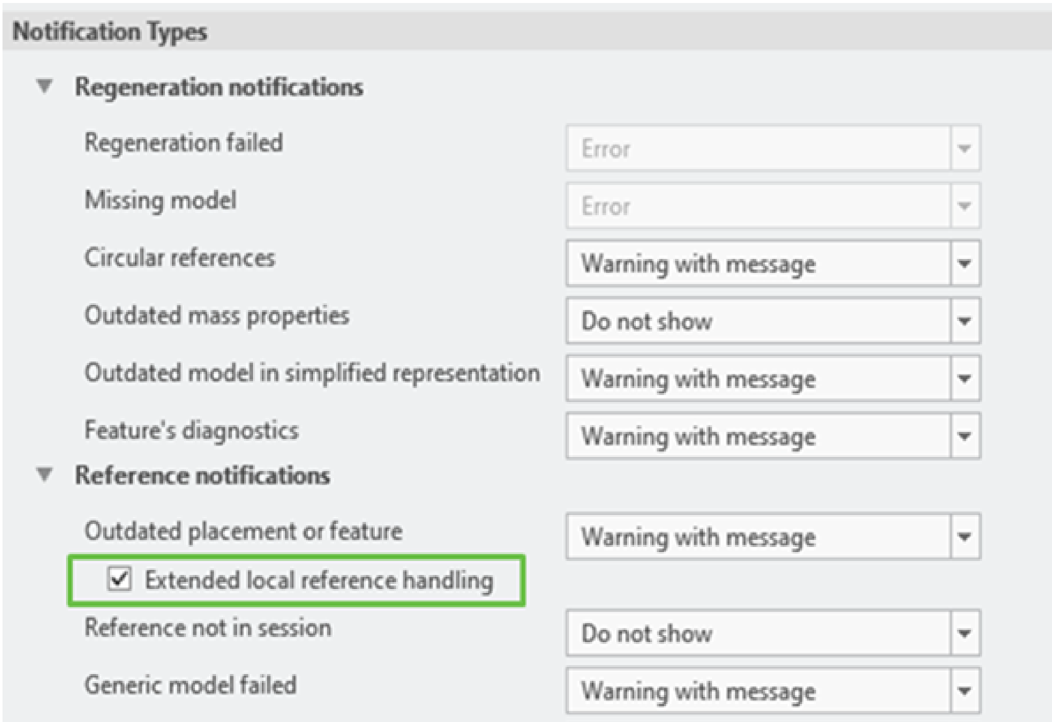
Enhanced Reporting of Missing References in Creo

Creo Parametric 11.0.0.0

User Interface Location: On the status bar, click **Notification Center**.

Description

Creo 11, enhances the reporting of missing references within the notification center. You can turn on the notifications for the category **Extended local reference handling** in the **Notification Center** tab of the Creo Options dialog box.



This option enables Creo to report missing references within the current part or assembly context, even for features that do not fail. For example, you can see the notifications for missing weak references or missing references for which an alternate reference was found. The reference status is shown in the model tree column **Parent Details**. It indicates whether local references could not be found and are missing. Previously, this was only reported for failed features and their child features.

Additionally, with this enhancement new queries are added to the Model Tree search queries:

- Features with Missing References
- Features with Missing Local References
- Features with Missing External References
- Components with Missing Placement References

After searching for the listed references, you can further investigate the missing references in the Global reference viewer.

Benefits

- Personalize and optimize the notification center reporting to better understand the missing or alternate local references in your model.
- Locate and resolve the model tree queries using the model tree queries and the Global Reference Viewer.

Additional Information

Tips:	None.
Limitations:	Reporting does not include geometrically consumed references, not-in-geometry references, or missing relation references.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	`nmgr_ref_changed_include.yes`, `*no` Checks local references in local features.

Enhancement: Real Number to String in Relations

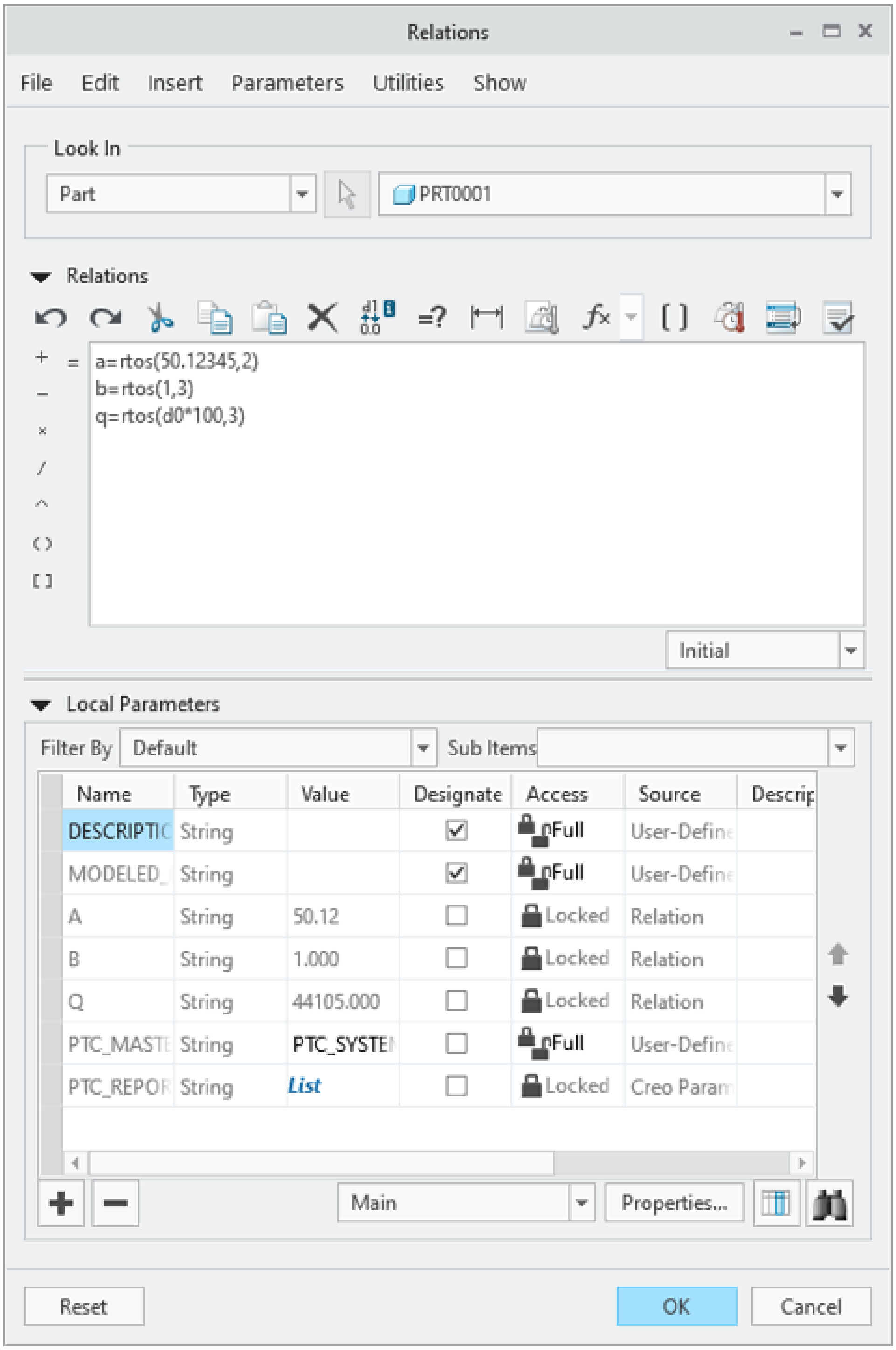
Creo Parametric 11.0.0.0

User Interface Location: rtos() is available when defining relations.

Description

In Creo, relations can now support the conversion of a real number to a string. This enhancement allows you to control the number of decimal places in the resulting string by specifying a number after the input value.

For example `rtos(50.12345, 2)=50.12`



Benefits

- Define a relation to convert a real number to a string.
- Control the number of decimal places in the string, and add it to a note.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Enhancement: Model Units as Parameters

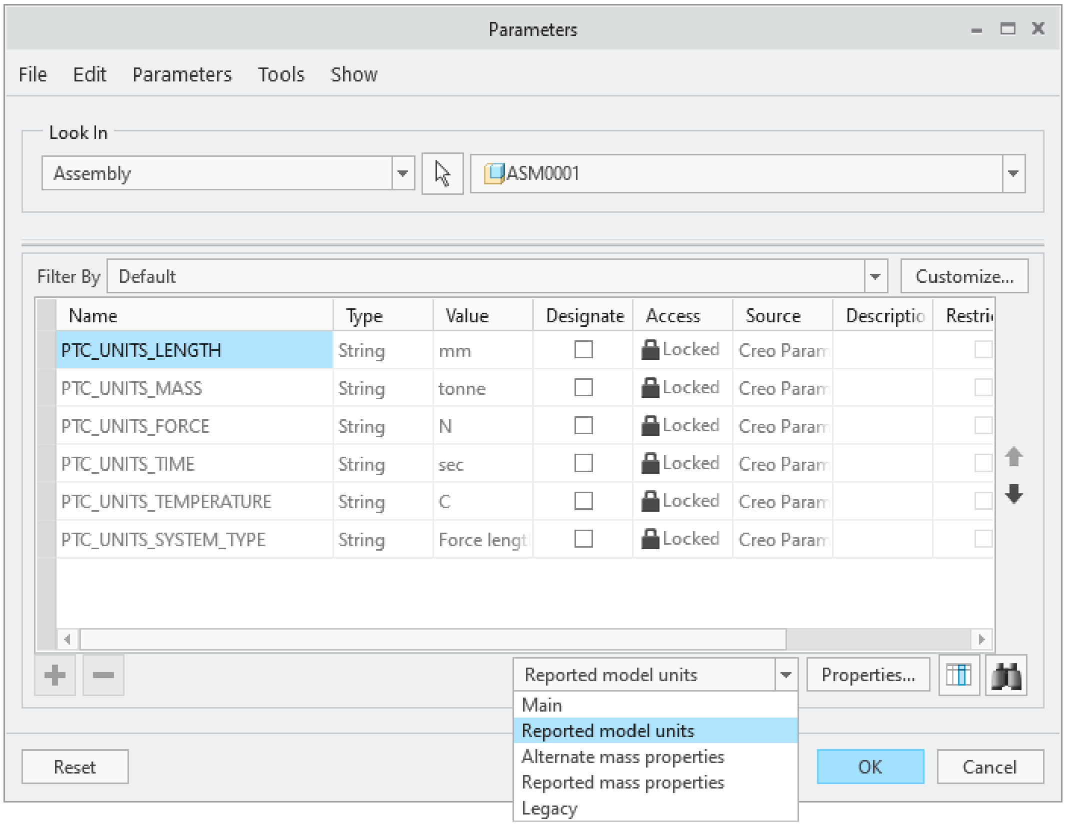
Creo Parametric 11.0.0.0

User Interface Location:

1. On the **Tools** tab, click [ ] **Parameters**. The **Parameters** dialog box opens.
2. Switch parameter set to **Reported model units**.

Description

The model units parameters are now available as system parameters for parts and assemblies. The values of the parameters are updated when the model units are changed to a different type. The model units parameters can be used in annotations and can be designated for viewing in Windchill.



Benefits

Use the new system parameters to indicate the current model unit system. These parameters will automatically update when model unit types are changed.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Enhancements to the Family Table

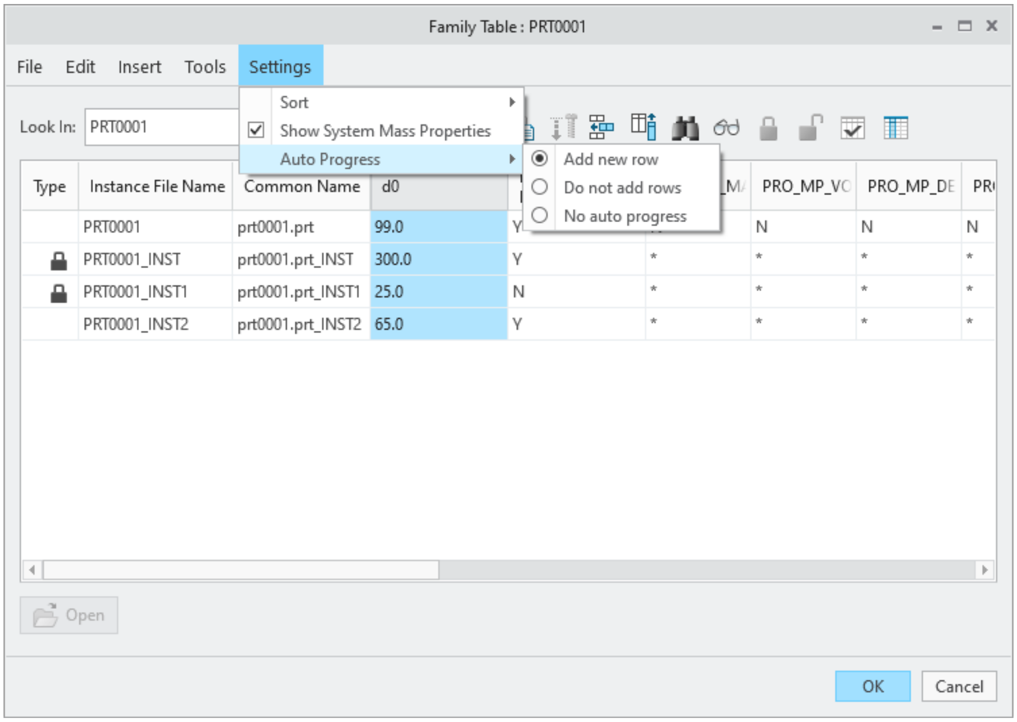
Creo Parametric 11.0.0.0

User Interface Location: **Tools** ▸ **Family Table**

Description

The following enhancements have been added to the Family Table dialog box:

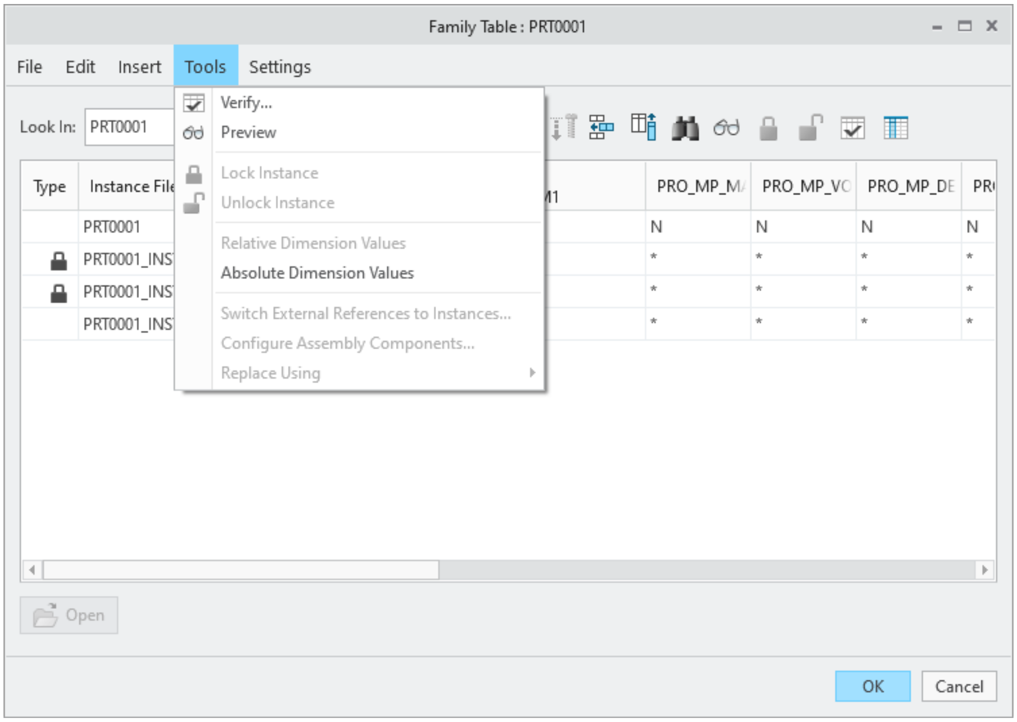
- New **Settings** menu added to the Family Table dialog box. This menu provides improved access to sorting, system mass properties, and controlling the auto progression behavior on pressing Enter.



- Family Table columns support sorting for verified cells. To sort the columns, click the column header or choose the sort type from the sort menu in **Settings ▸ Sort**.
- When selected, the display settings of the system mass properties are retained for that Creo session.
- Auto Progress behavior—You can now control the progression behavior when Enter is pressed. You can select from the following:
  - **Add new row** (Default)
  - **Do not add rows**
  - **No auto progress**

  The selected option is retained and stored as an user interface customization. To reset the setting, click **File ▸ Options ▸ Manage UI Customization**.
- New user interface customization in the **Tools** menu:
  - Separate commands for locking and unlocking the selected instance.

- Separate commands for Absolute Dimension Values and Relative Dimension Values.



Benefits

Better usability and productivity when working with the Family Table.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

14

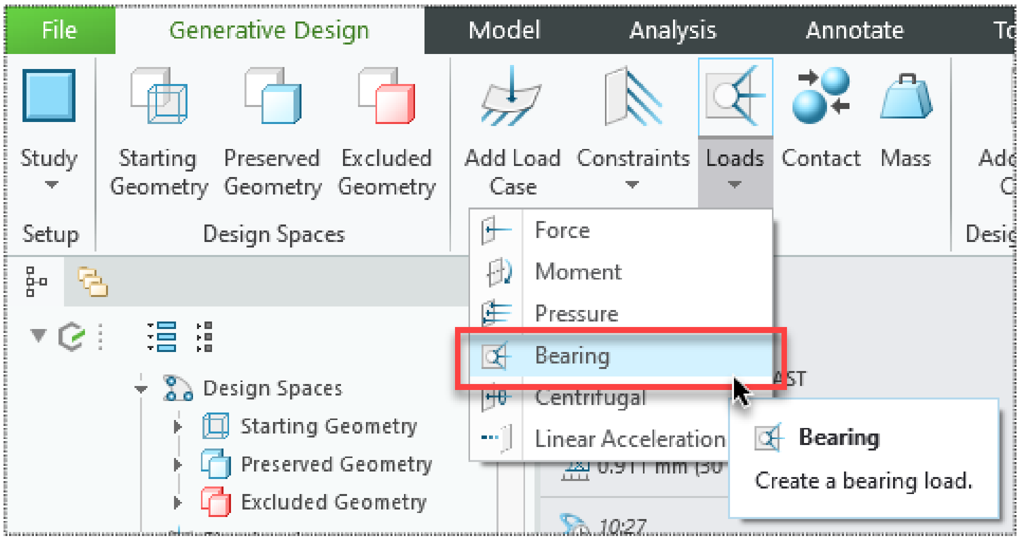
Generative Design

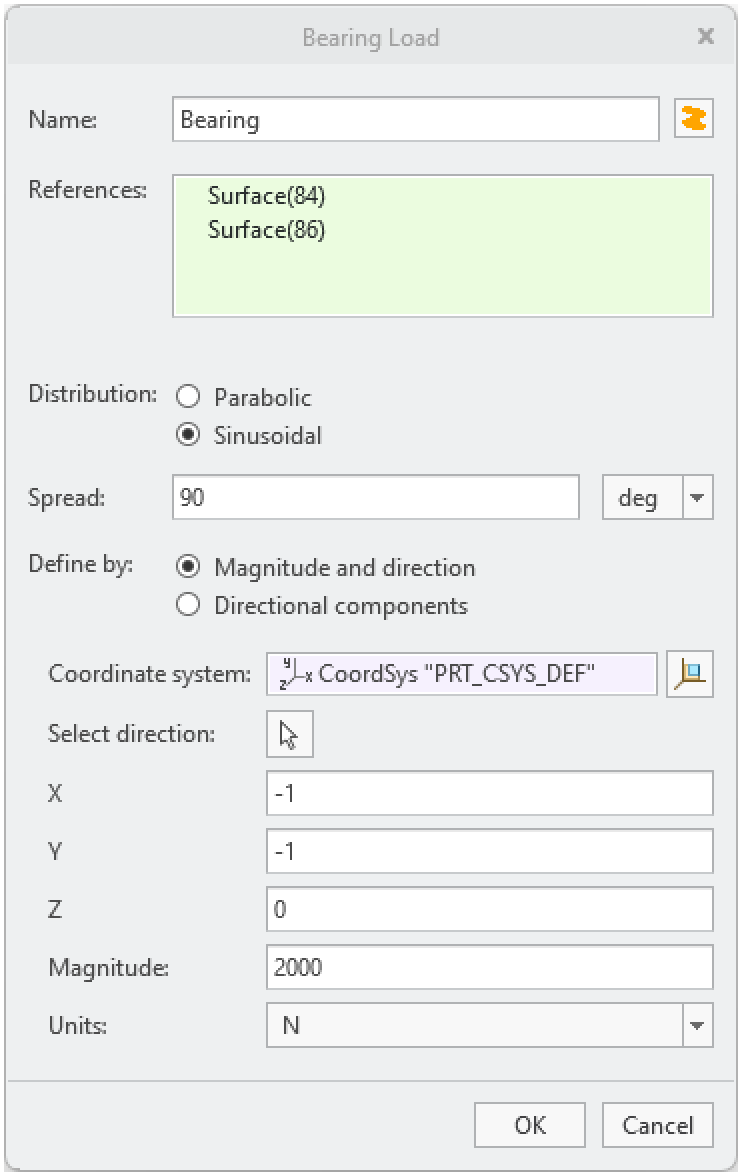
Support for Bearing Loads in Generative Design

Creo Parametric 11.0.0.0

User Interface Location:

- In the Generative Design application, click **Loads** ▸ **Bearing**. The **Bearing Load** dialog box opens.





Videos

See the video on the Learning Connector.

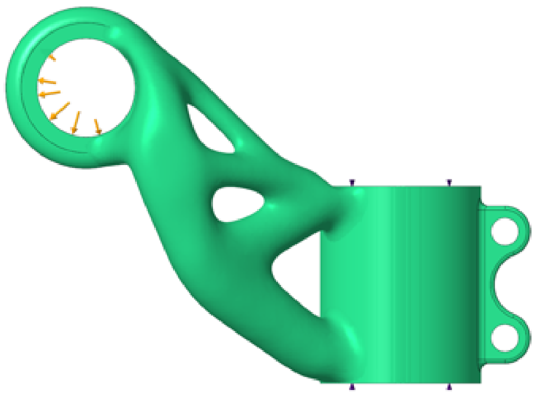
Description

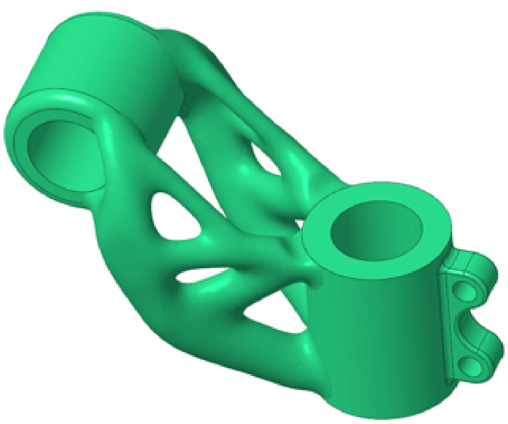
In Generative Design, you can now define a total bearing force load on cylindrical surfaces.

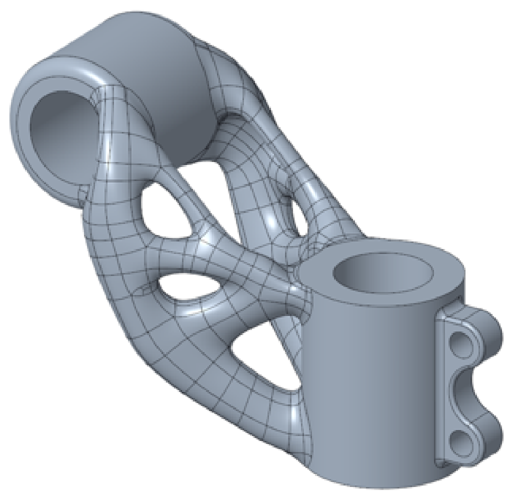
- You can define the force distribution as a parabolic function or sinusoidal function. The default force distribution is parabolic.

- You can set the spread angle that defines the angle from the load direction vector on which the load needs to be distributed on each side. The default spread angle is 90 degrees.
- You can select **Magnitude and direction** or **Directional components** to define the magnitude and direction of the load.

Once defined, the bearing load is listed in the Model Tree under the loads section in the given load set.







Previously, you could not capture the proper loading conditions needed for optimization when bearing loads were required. You had to either define standard loading conditions that would not capture the design intent or over-constrain the model to account for bearing conditions.

Benefits

With this enhancement, you can now define the loading conditions for the optimization more accurately, capturing the design intent.

Additional Information

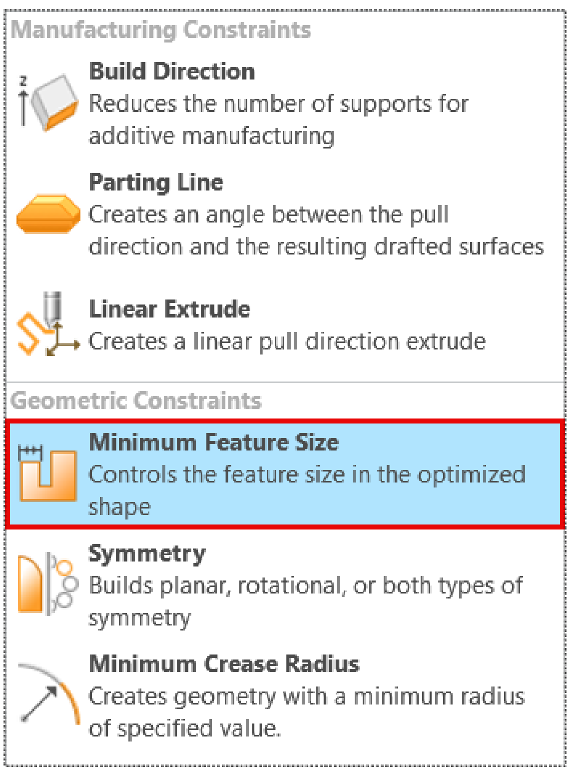
Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

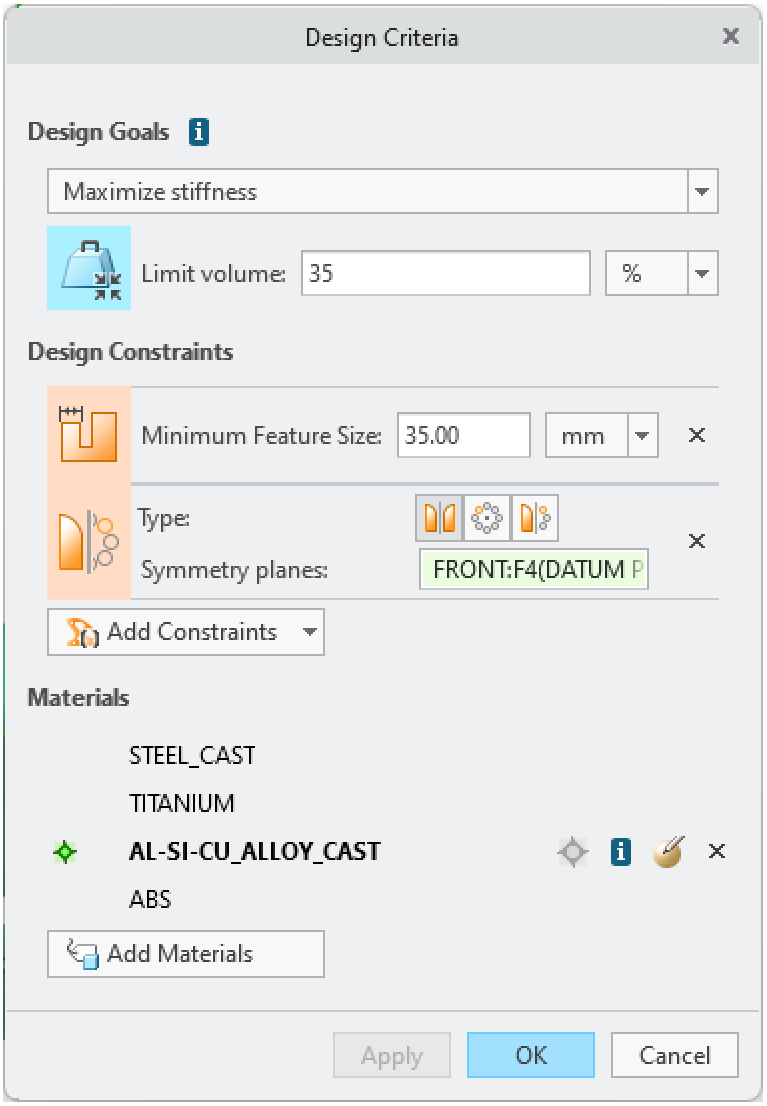
Support for Minimum Feature Size in Generative Design

Creo Parametric 11.0.0.0

User Interface Location:

1. In the Generative Design application, click **Add Design Criteria**. The **Design Criteria** dialog box opens.
2. Under **Design Constraints**, click **Add Constraints**
3. Under **Geometric Constraints**, select **Minimum Feature Size** from the list.





Videos

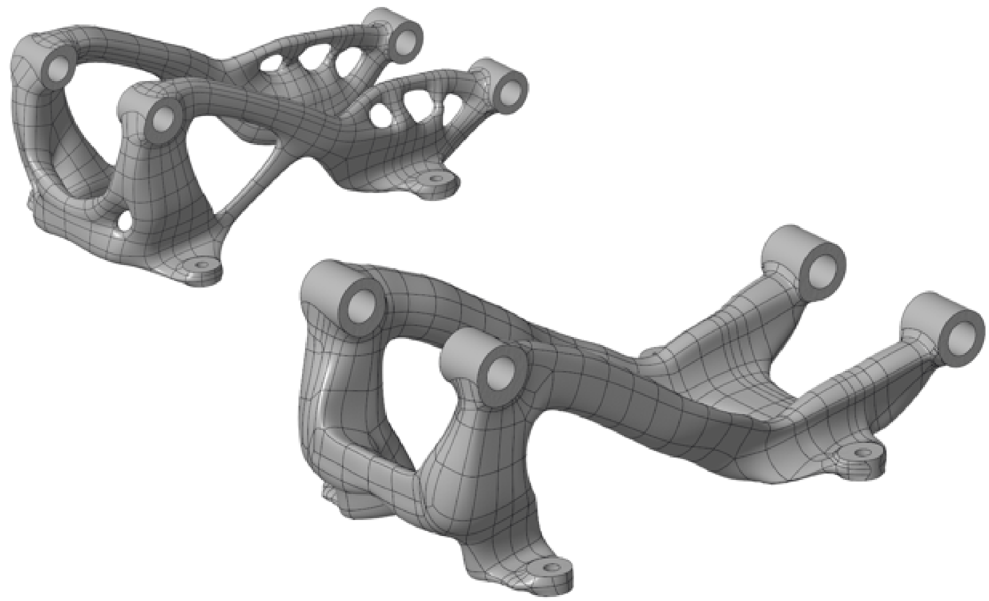
See the video on the Learning Connector.

Description

In Generative Design, you can now define the desired minimum feature size of the resulting geometry that controls the optimization thickness.

The new **Minimum Feature Size** geometry constraint allows you to set the minimum feature size for each design criteria in the units of your choice. The preferred value for the minimum feature size is 3 times the element size, with no upper limit.

The minimum feature size is listed under the **Additional Information** for the design criteria node in the Generative Tree.



Previously, you could not control the minimum thickness of the optimized shape. This resulted in situations where very thin features were created in the resulting model, which could be difficult to manufacture or result in the areas of weakness.

Benefits

This enhancement allows you to define the minimum feature size for optimization to capture your design intent. It controls the thickness of the resulting geometry and avoids thin-walled structures in the optimization.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Support for Planar Symmetry During the Reconstruction

Creo Parametric 11.0.0.0

User Interface Location: Click **Applications** ▸ **Generative Design**.

Videos

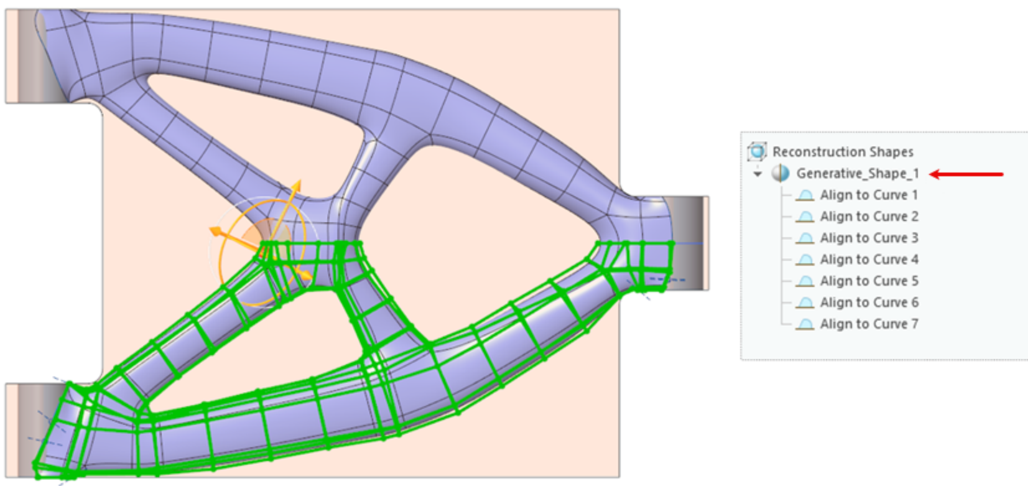
See the video on the Learning Connector.

Description

Generative Design now preserves planar symmetry during the reconstruction of optimized results.

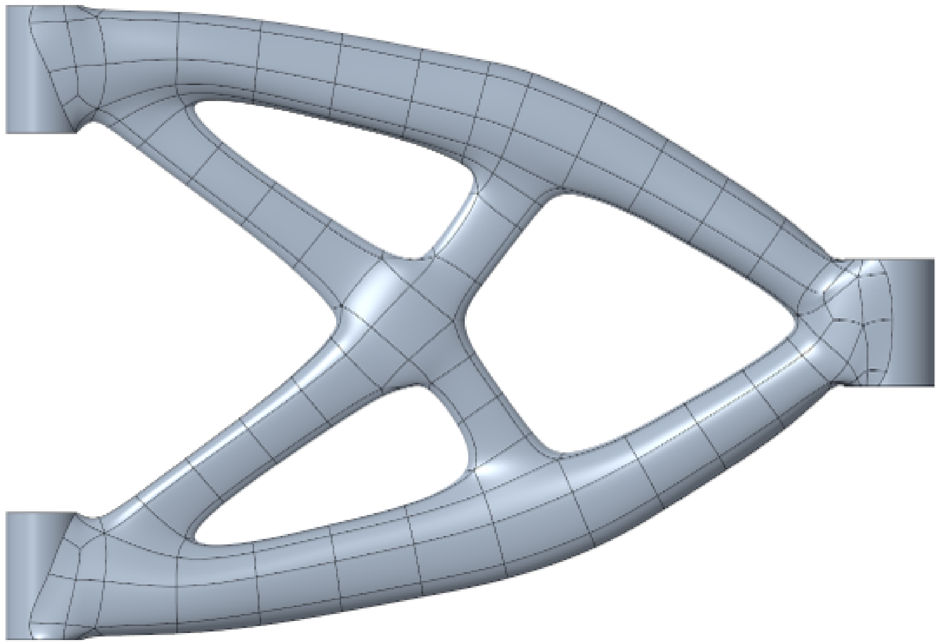
When a planar geometry constraint is defined for optimization, a plane is used during the reconstruction to mirror the Freestyle geometry. The preserved geometry used in the optimization operation is also symmetrical across the planar reference.

The Freestyle feature is shown as patterned in the Model Tree. When edited, it shows the shape as mirrored in the Freestyle tree. Modifying the control mesh automatically updates the mirrored reference.

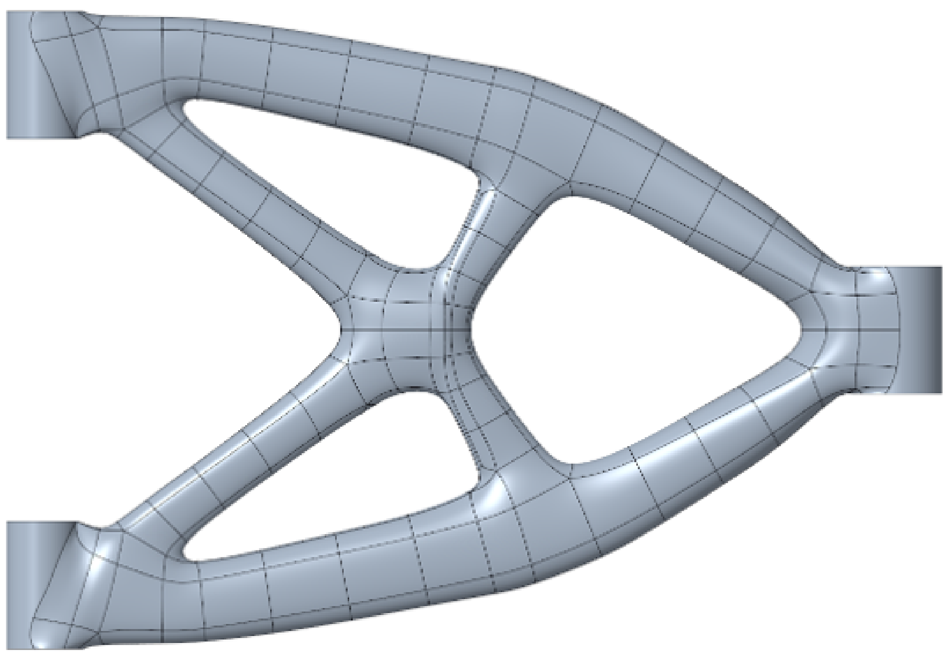


Previously, the reconstruction model did not preserve the symmetrical geometry constraint used in the optimization setup. This resulted in the reconstruction geometry, defined with the subdivisional surfaces created in Freestyle, that was not exactly symmetrical across the symmetrical plane. In addition, you needed to edit the Freestyle entities on both sides of the symmetrical plane to maintain symmetry.

The following graphic shows the reconstruction model in Creo 10.0:



The following graphic shows the reconstruction model in Creo 11.0:



Benefits

This enhancement helps you perform a successful reconstruction operation, achieving the proper quality of the reconstruction shape. It improves the editing outcomes of the Freestyle shape by preserving symmetry and maintaining the design intent during reconstruction.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

15

Manufacturing

Lattice Export

Creo Parametric 11.0.0.0

User Interface Location: Click **File** ▸ **Save As** ▸ **Type** ▸ **3MF (*.3mf)** or **Stereolithography (*.stl)** ▸ **Options** ▸ **Lattice Settings** ▸ **Penetration into Shell** for beam-based lattice, or **Merge and Blend with Shell** for formula-based lattice.

Videos

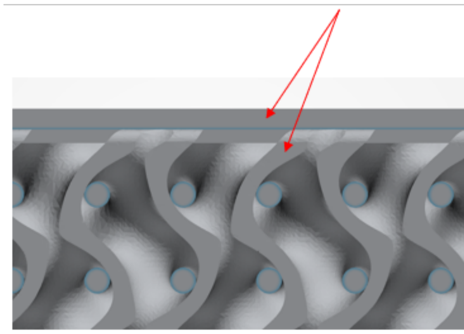
See the video on the Learning Connector.

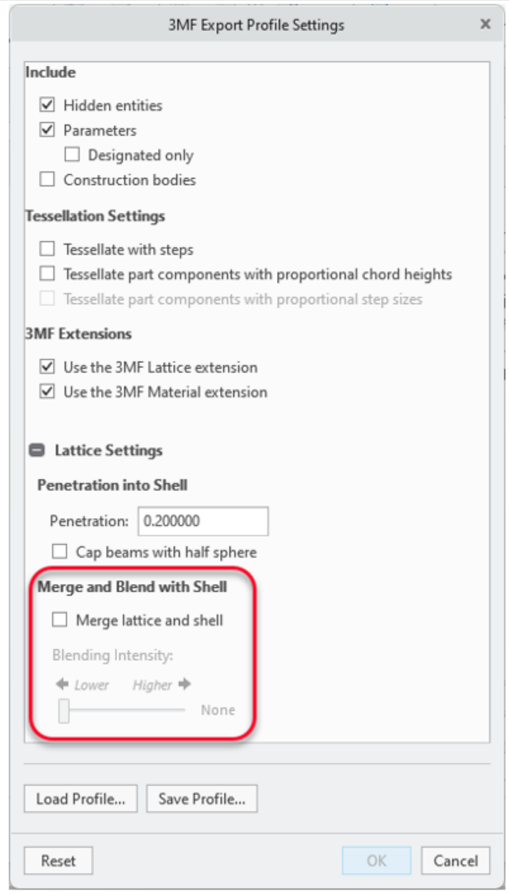
Description

Lattice export options are available in Part mode. You can now export tessellated 3D models from Creo Part mode. In previous releases the only option was available in the Tray Assembly

- Now it is possible to export a lattice feature in Part mode, similar as in the Tray Assembly.
- For several releases Creo Additive Manufacturing users can export lattices using two options:
  - Blend the formula-based lattices with the shell, and add a blending option which smooths the transition between the formula-based cells and the shell.
  - Add a penetration value between simplified beams that touch the shell and the shell itself. There is also an option of adding a semi-sphere on the tip of such penetrating beams.

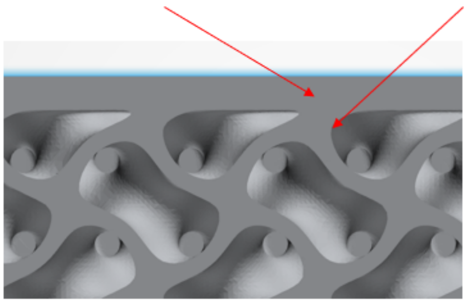
  Now these options are available in Part mode.
- Shell and lattice as separate objects

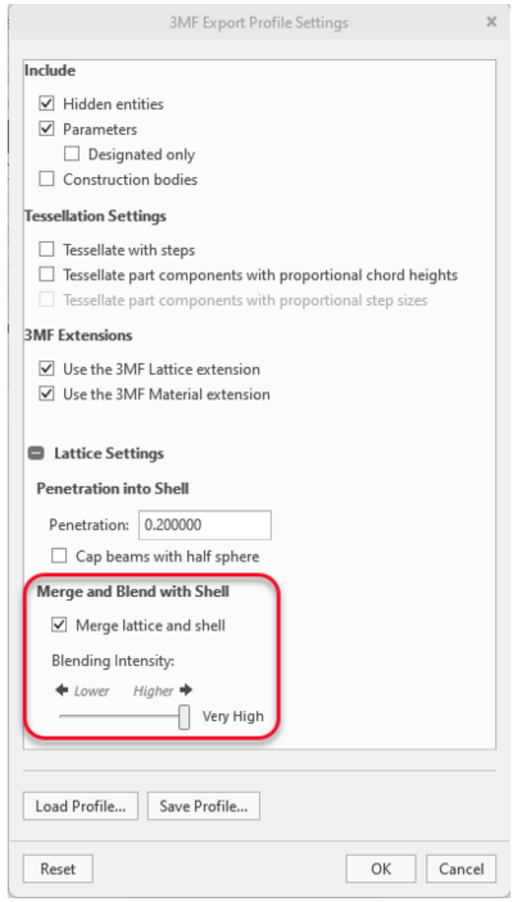




- Shell and lattice as one object

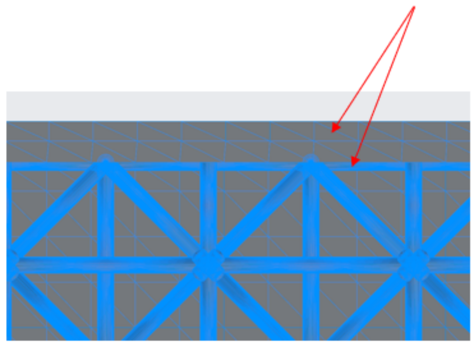
  Blending option to smooth the corners

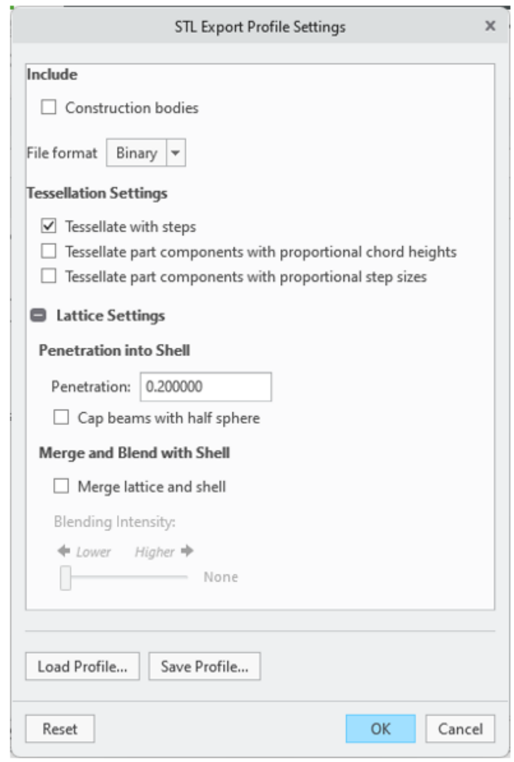




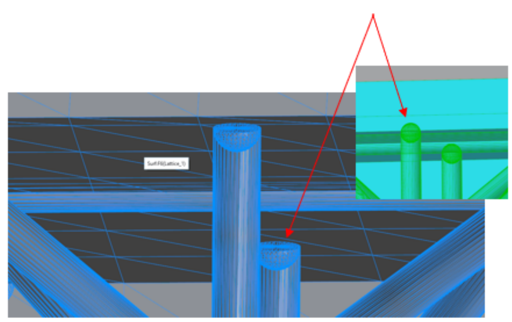
- Add a penetration value between the beams touching the shell and the shell itself.

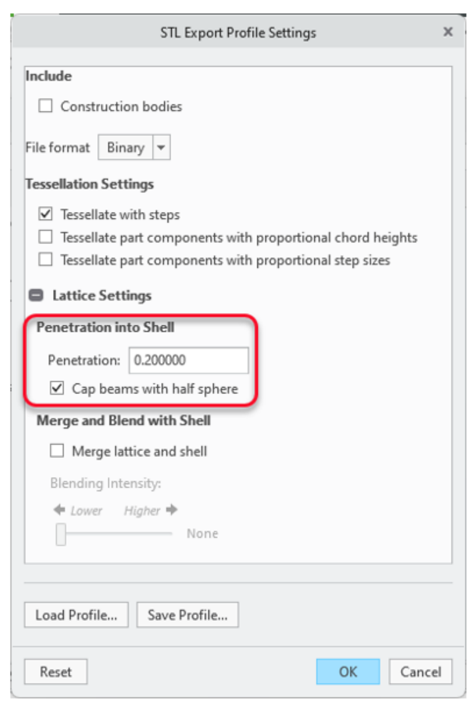
  0.2 penetration and no cap beams



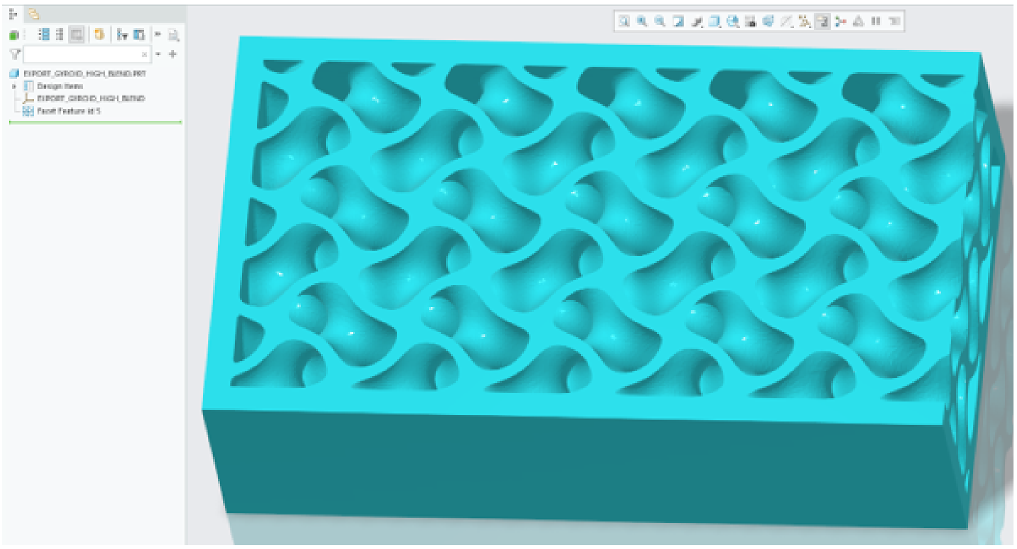


- Option of adding a semi-sphere on the tip of such penetrating beams





- Re-imported STL export from Creo with both options: Merging the lattice with the shell and blending option active to smooth the corners



Benefits

- Export option available for Creo users which creates lattice features
- More flexibility to send 3D models with lattices to 3D Printer applications
- Available for 3MF and STL

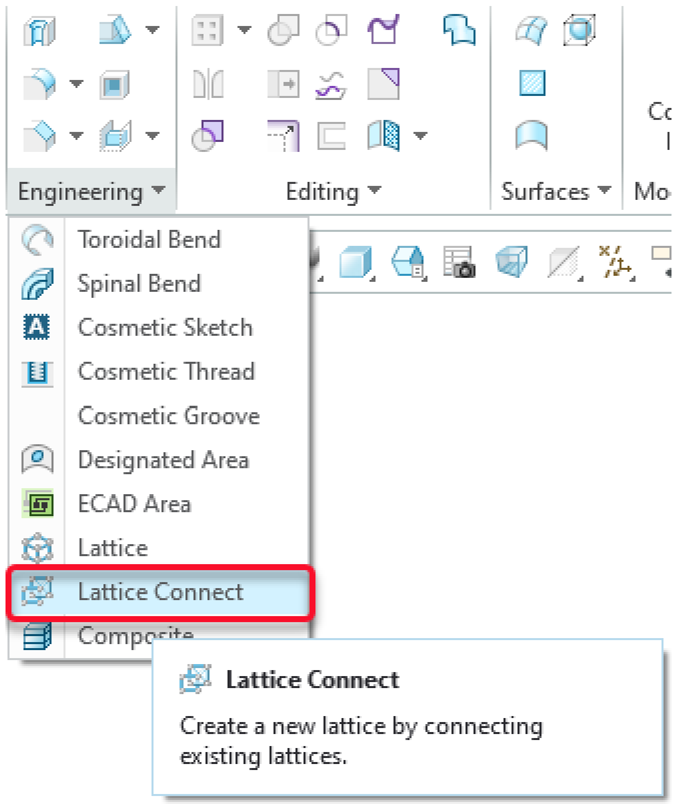
Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Lattice Connect Feature

Creo Parametric 11.0.0.0

User Interface Location: Click **Model** ▸ **Engineering** ▸ **Lattice Connect**.



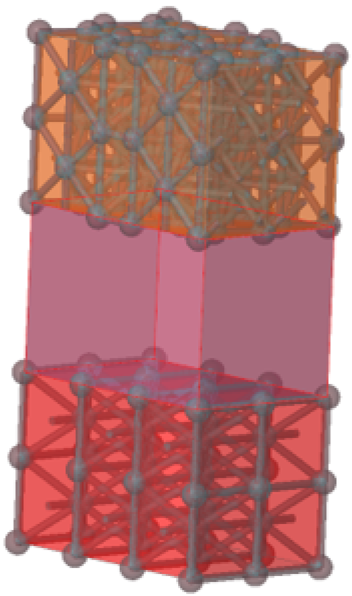
Videos

See the video on the Learning Connector.

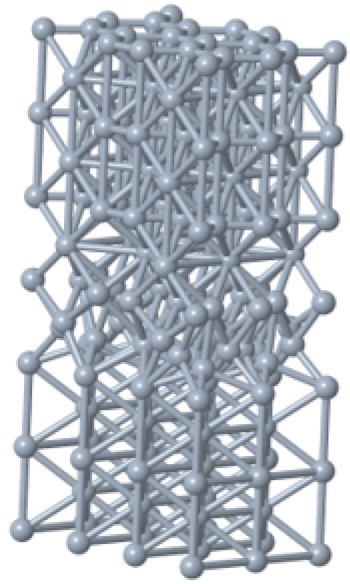
Description

A new lattice command to connect two or more separate lattices has been added to Creo Parametric 11. The result is a continuous lattice structure

Before



After

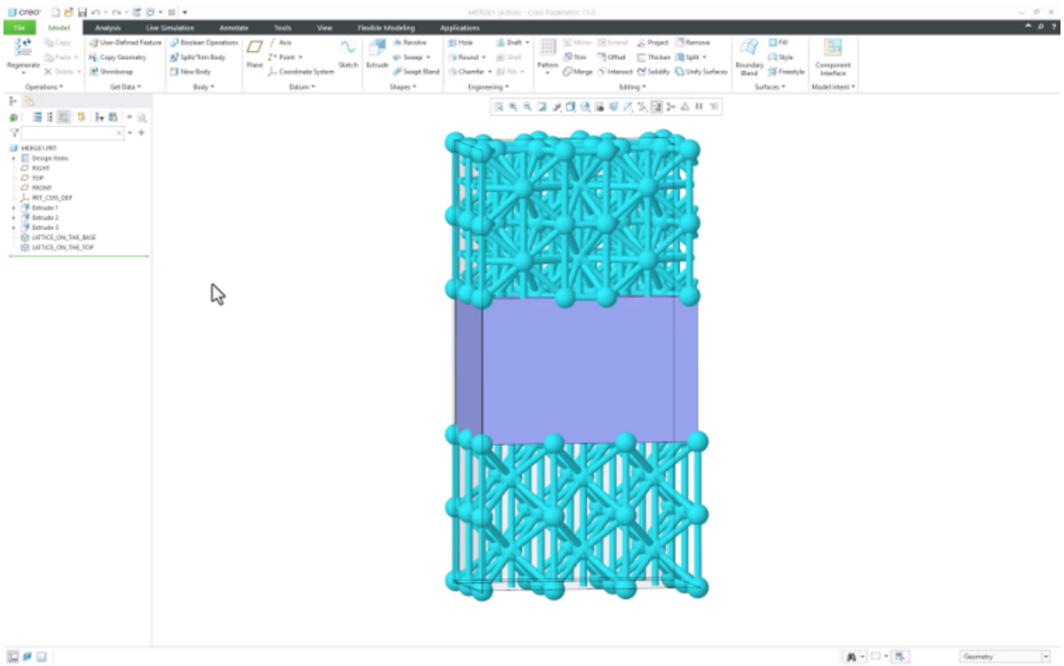


Consider the following for the reference lattices:

- Input lattices should be simplified beam lattices.
- Input lattices can be of the same or different cell types, different cell sizes, or different cell orientations.
- In this first implementation most regular beam lattices and stochastic Voronoi are supported. Extended support for more lattice types and use cases will be added in next releases.

Benefits

- More flexibility to create complex lattices
- Straightforward workflow for creating continuous lattice structure
- Supported inside the same familiar Lattice user interface



Additional Information

Tips:	None.
Limitations:	• Input should be lattices with Simplified representation • Intended to work with Voronoi - Inside Volume lattice types These lattice types cannot be connected: • Stochastic lattices with beams on bounding surfaces • Stochastic lattices with Delaunay triangulation • Stochastic lattices with any density variation • Auxetic lattices • Lattices with full geometry representation • Lattices with quasi-radial or herringbone cell propagation type
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Lattice: Simplified Beam Lattices Adjust along the Warp Feature

Creo Parametric 11.0.0.0

User Interface Location: Click **Engineering** ▸ **Lattice** ▸ **Lattice Type** ▸ **Beams** ▸ **Representation** ▸ **Simplified**.

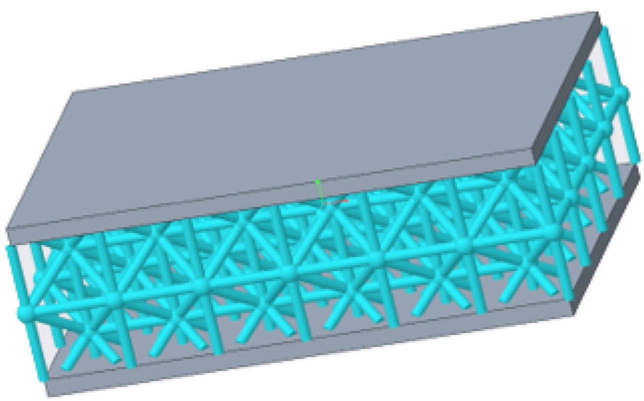
Videos

See the video on the Learning Connector.

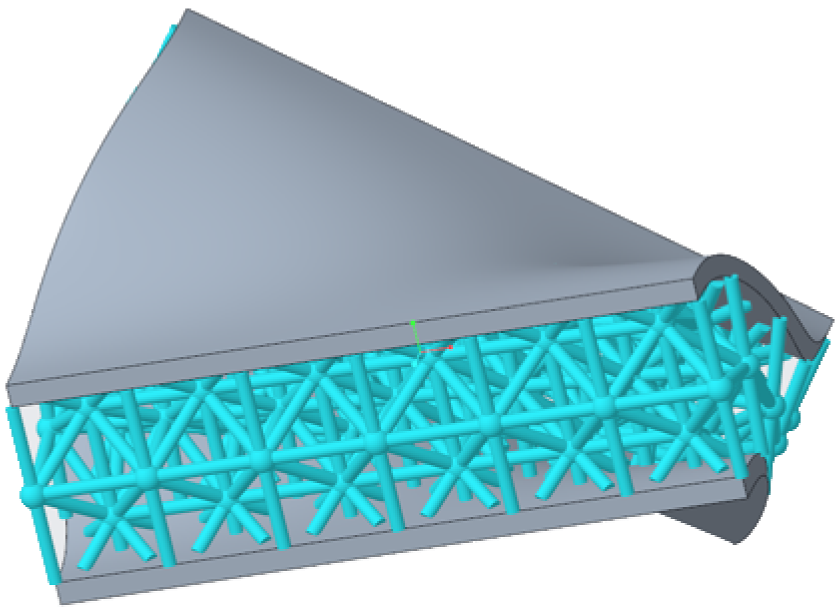
Description

You can now modify a part that contains simplified beam lattices using the Warp feature. The internal lattice beams will adjust accordingly.

Original part that contains lattice:



Part modified by Warp feature:



Benefits

- More flexibility to create lattices that follow complex shapes
- Simplified lattices can be used in more use cases
- Faster simulation by using simplified lattices

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Lattice: Pore Size as a Metric that Drives Lattice Construction

Creo Parametric 11.0.0.0

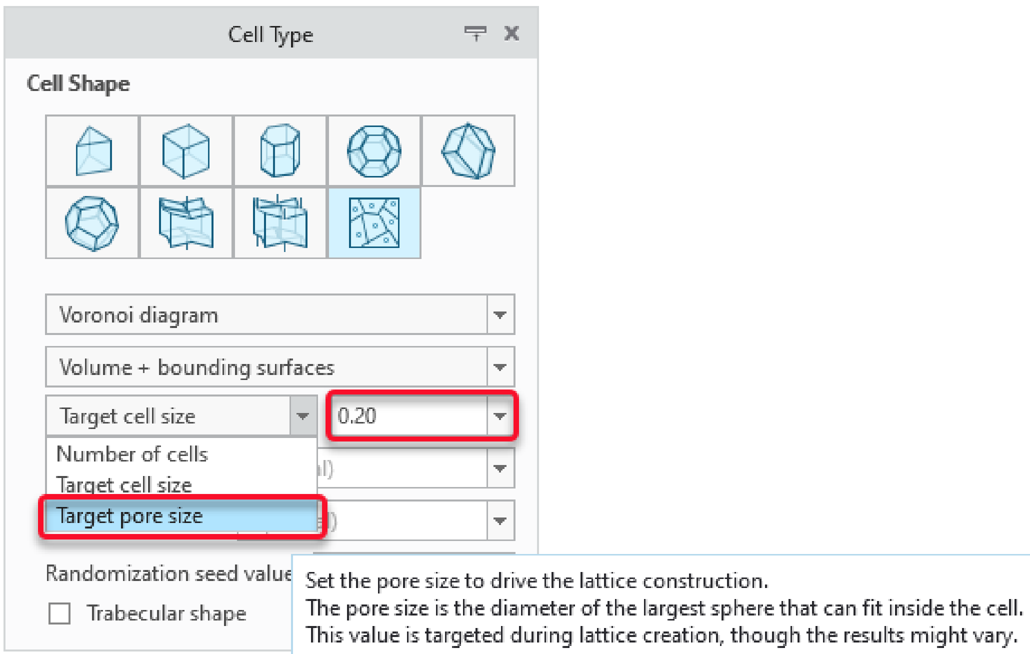
User Interface Location: Click **Engineering** ▸ **Lattice** ▸ **Lattice Type** ▸ **Beams** ▸ **Cell Type** ▸ Stochastic.

Videos

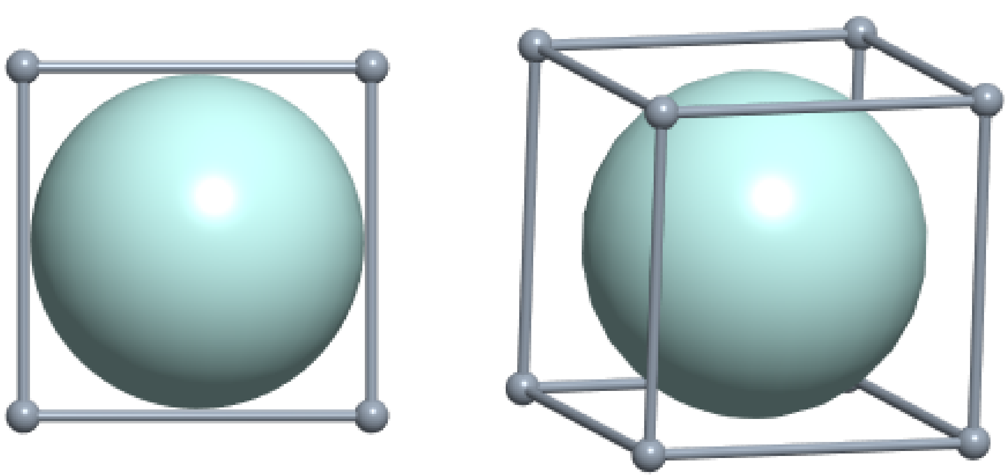
See the video on the Learning Connector.

Description

You can now create a stochastic lattice based on a pore size requirement. The **Target pore size** option uses pore size as a metric that drives stochastic lattice construction.

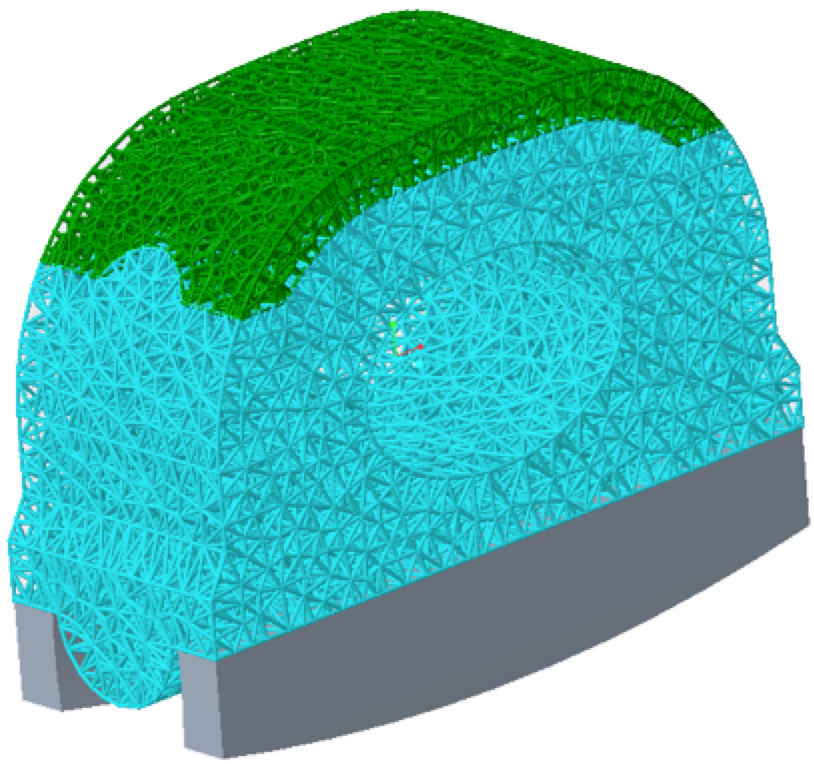


Pore size is defined as the diameter of the largest sphere that can be fit through the faces of the cell.



Benefits

- Streamlined workflow specific for the creation of stochastic lattices with target porosity
- Straightforward workflow for medical implants
- Supported inside the same familiar Lattice user interface

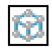
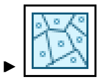


Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Lattice: Randomization Setting for Stochastic Lattice

Creo Parametric 11.0.0.0

User Interface Location: Click **Engineering** ▸  **Lattice** ▸ **Cell Type** ▸  Stochastic.

Videos

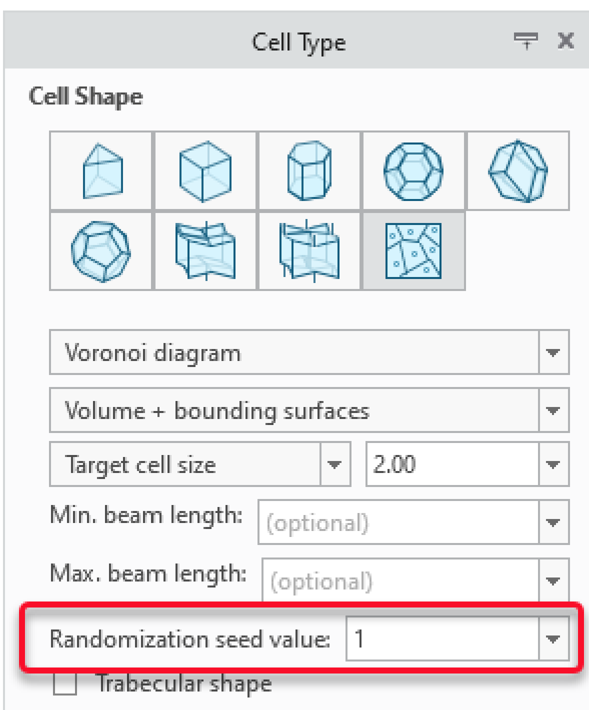
See the video on the Learning Connector.

Description

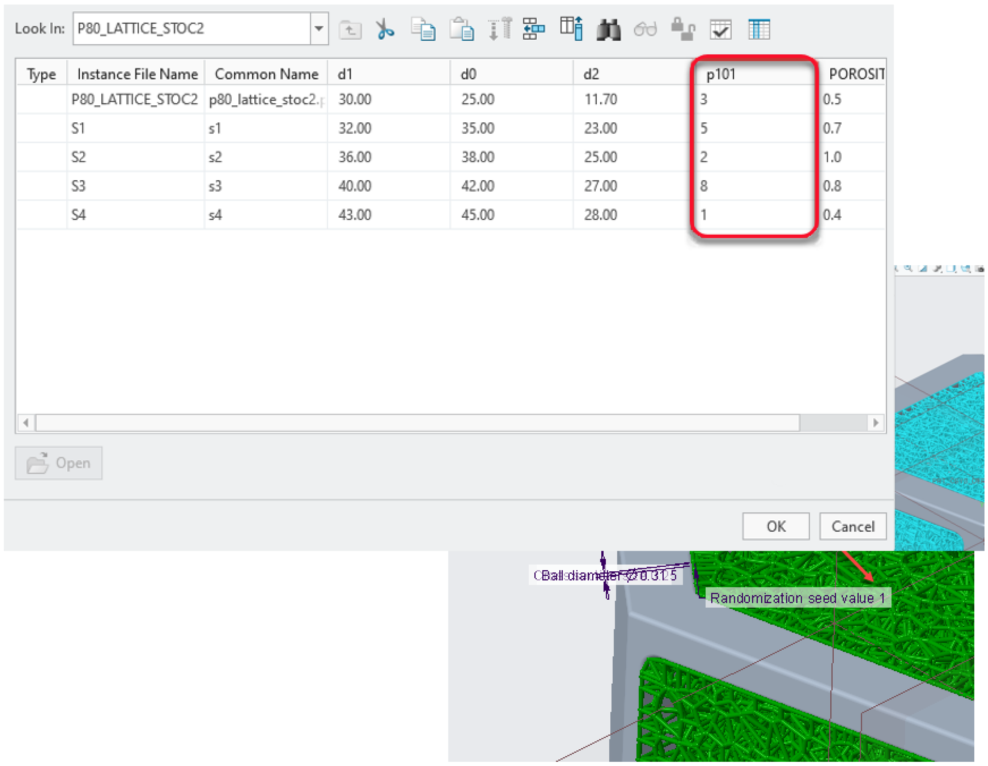
Creo Parametric 11 contains a new **Randomization seed value** setting for stochastic lattice. This functionality now allows you to create a family of parts containing stochastic lattices using family tables. Each stochastic lattice instance can be randomized independently.

Previously, when a part in a family table containing a stochastic lattice was randomized, all other instances got randomized.

- Display the randomization seed value of a stochastic lattice while editing or defining the lattice



- Dimension available for family tables to display instances of stochastic lattice that vary by their randomization setting



- Randomize each stochastic lattice instance independently

Benefits

- Each stochastic lattice in the family table can be randomized separately
- More clarity defining stochastic lattices
- Simple to use

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

HSM 4-Axis Rotary Machining

Creo Parametric 11.0.0.0

User Interface Location:

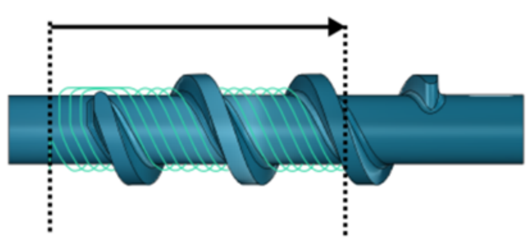
- In Manufacturing, click **Mill** ▸ **High Speed Milling** ▸ **HSM 4 Axis Rotary Finish**.
- In Manufacturing, click **Mill** ▸ **High Speed Milling** ▸ **HSM 4 Axis Rotary Rough**.

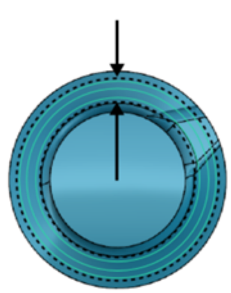
Videos

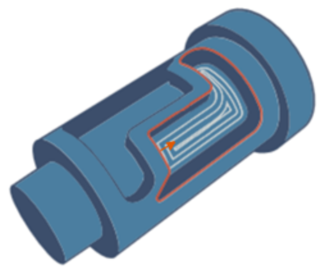
See the video on the Learning Connector.

Description

The new 4-axis rotary roughing and rotary finishing toolpaths are now available in this release. These toolpaths can pass 360 degrees and can be used on screw-type parts. They are useful in automotive and oilfield crankshafts, camshafts, and drill heads. The new toolpaths provide different controls to define the machining area, such as Axial containment, Radial containment, and Containment Loops. The supported tools for these toolpaths are End Mill, Ball Mill, and Bull Nose Mill.







Benefits

This enhancement provides the following benefits:

- Generates a machining toolpath that overcomes the 360-degree limitation
- Provides automated roughing and finishing sequences
- Creates efficient toolpaths for complex rotary parts that require 4-axis rotary machining

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Multiple Mill Volume Support for HSM Rough and HSM Rest Rough Toolpaths

Creo Parametric 11.0.0.0

User Interface Location:

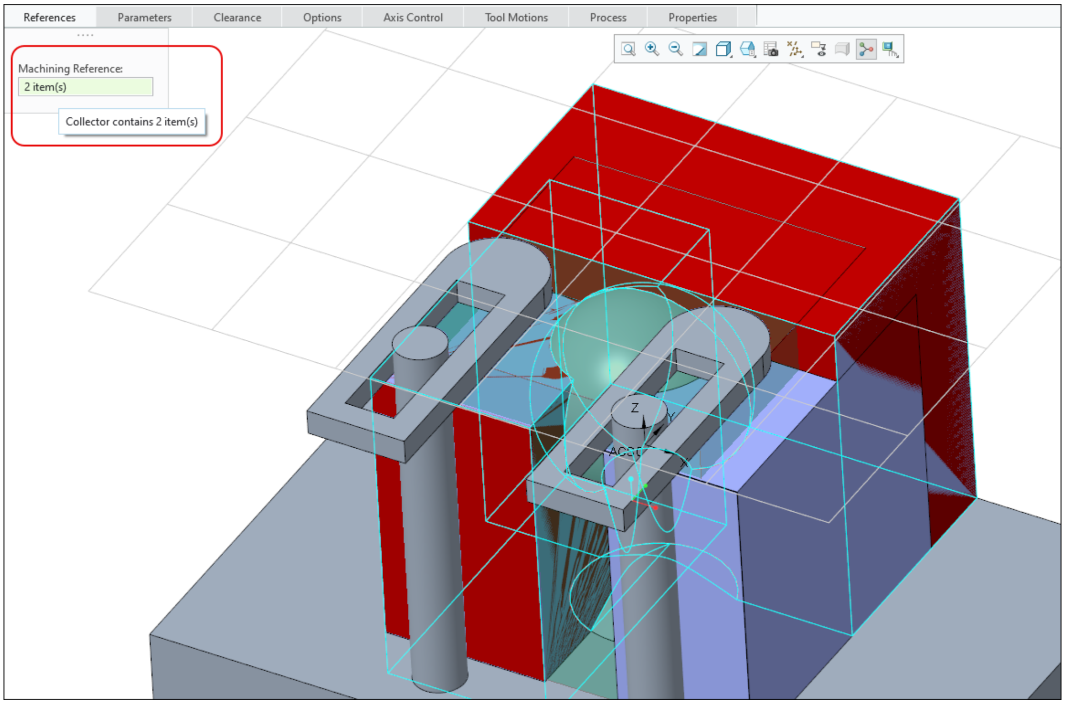
- In Manufacturing, click **Mill** ▸ **High Speed Milling** ▸ **HSM Rough**.
- In Manufacturing, click **Mill** ▸ **High Speed Milling** ▸ **HSM Rest Rough**.

Videos

See the video on the Learning Connector.

Description

Selection of multiple mill volumes is now supported for HSM rough and rest rough toolpaths. You can select multiple mill volumes in the **Machining Reference** collector on the **References** tab for the HSM rough and rest rough toolpaths. The selection of multiple mill volumes is supported for 3+2 axis machining.



Benefits

This easy-to-use enhancement provides the following benefits:

- Reduces programming time
- Enables you to define localized machining volume

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Tool Holder Degouge for HSM Toolpaths and Solid Tools

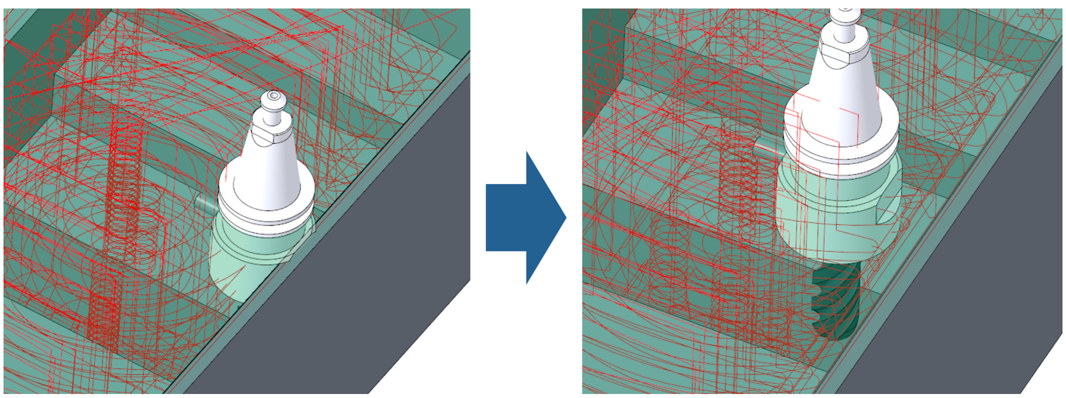
Creo Parametric 11.0.0.0

Videos

See the video on the Learning Connector.

Description

Previously, the profile or shape of the solid tool holder was not considered in toolpath calculations. Now the HSM toolpaths can degouge profile of the solid tool holders also. The toolpath considers the holder shape corresponding to the part design of the solid tool holder. A solid tool holder is degouged against a reference part and in-process stock.



Benefits

This enhancement provides the following benefits:

- Enhances accuracy of toolpaths
- Considers actual solid tool holder shape in toolpath calculations for HSM steps

Additional Information

Tips:	Ensure that you assemble the solid tool and solid tool holder in the correct position for generating an accurate toolpath and simulation. Placing the tool inside the holder generates an incorrect toolpath and simulation.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	`enable_hsmseq_holder_degouge` `yes, no*`

Box Selection Support for Auto Deburring Sequences

Creo Parametric 11.0.0.0

User Interface Location:

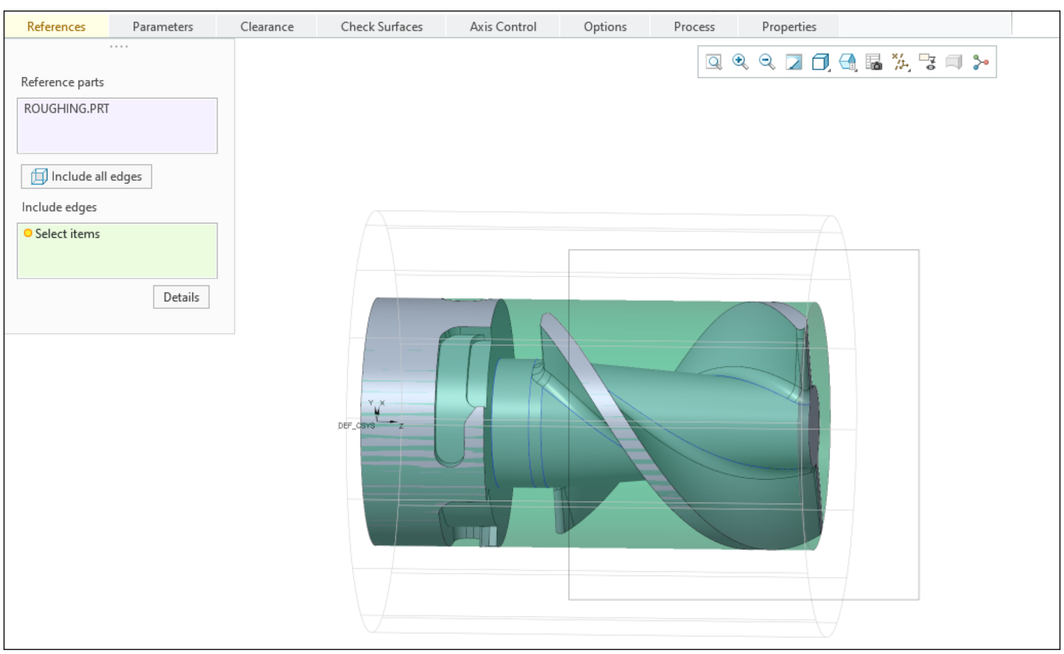
1. In Manufacturing, click **Mill ▸ High Speed Milling ▸ Auto Deburring**.
2. Select the **References** tab.

Videos

See the video on the Learning Connector.

Description

The Auto Deburring functionality is now expanded to include support for the box selection of edges. You can use the box selection method to select edges for including or excluding them from machining. You can select multiple edges in a single action instead of selecting a single edge every time. The box selection method gives you more control over which edges to consider for machining.



Benefits

This easy-to-use enhancement provides the following benefits:

- Reduces programming time
- Provides more control on selection of edges for machining

Additional Information

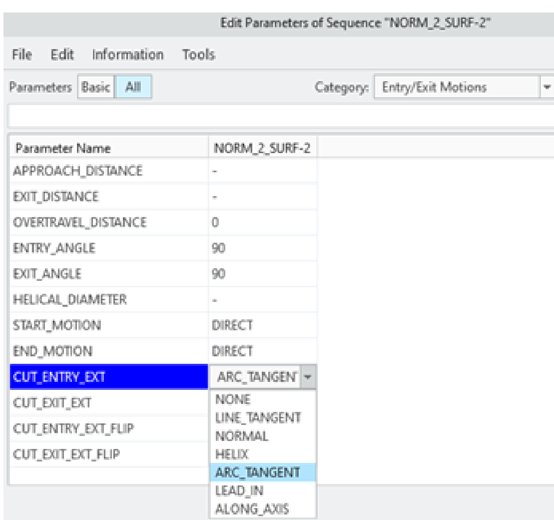
Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Tangential Arc Support for Entry and Exit Motions in Trajectory Milling

Creo Parametric 11.0.0.0

User Interface Location:

1. In Manufacturing, click **Mill** ▸ **Trajectory**. The **Trajectory** tab opens.
2. Click the **Parameters** tab.
3. In the parameters list, set the *CUT_ENTRY_EXT* or *CUT_EXIT_EXT* parameter to ARC_TANGENT under the **Entry/Exit Motions** category.

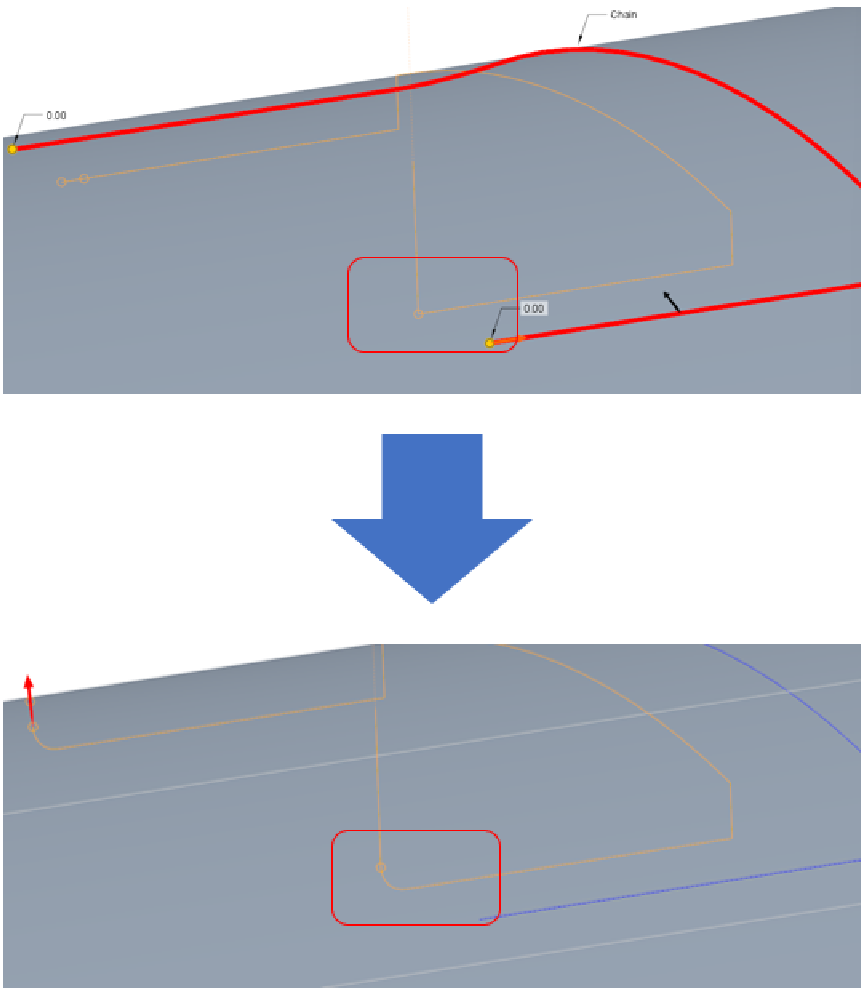


Videos

See the video on the Learning Connector.

Description

This enhancement supports the creation of lead-in or lead-out arcs along the tool axis using ARC_TANGENT. These arcs are tangential to the cut. You can use ARC_TANGENT as the value of the *CUT_ENTRY_EXT* parameter for the entry motion or *CUT_EXIT_EXT* parameter for the exit motion.



Benefits

This enhancement provides the following benefits:

- Defines the entry and exit movements along the direction of the cut
- Reduces the possibility of breaking small tools

• Supports 3, 4, and 5-axis trajectory steps

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

New Option for Trajectory Curves That Are Not Coincident with Normal Surfaces

Creo Parametric 11.0.0.0

User Interface Location:

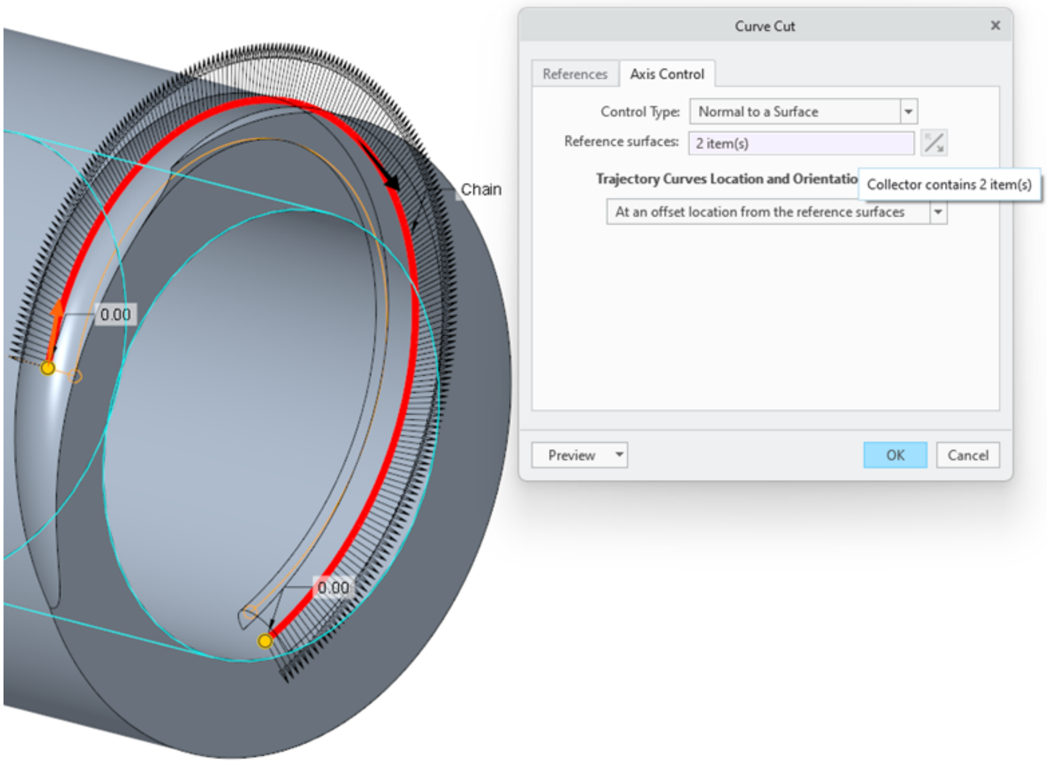
1. In Manufacturing, click **Mill** ▸ **Trajectory**. The **Trajectory** tab opens.
2. Click **Tool Motions** ▸ **Curve Cut**. The **Curve Cut** dialog box opens.
3. In the **Trajectory Curve** collector, specify the curve cut reference along which you want to create the cut motion.
4. Select the **Axis Control** tab.

Videos

See the video on the Learning Connector.

Description

This enhancement provides an option to define a 4 or 5-axis toolpath, where the curve and the surface that defines the normal for the resulting toolpath are not coincident. The trajectory curves are distant from the normal reference surfaces. This option is supported for the Milling, End Mill, and Ball Mill tools.



Benefits

This enhancement provides the following benefits:

- Provides flexibility to create 4 or 5-axis trajectory toolpaths
- Requires fewer clicks because there is no need to create additional geometry
- Works with vertical lead-in and lead-out motions

Additional Information

Tips:	The trajectory curve must lie within the extreme edges of the normal surface reference for generating the toolpath successfully.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Support for Trimming or Extending the Retract Movements to a User-Defined Plane

Creo Parametric 11.0.0.0

User Interface Location:

1. In Manufacturing, click one of the following:
   - **Mill** ▶ **Volume Rough**
   - **Mill** ▶ **Profile Milling**
   - **Mill** ▶ **Face**
   - **Mill** ▶ **Trajectory**

   For holemaking, click **Mill** and then select a holemaking type command in the **Holemaking Cycles** group.
2. Select the **Clearance** tab.

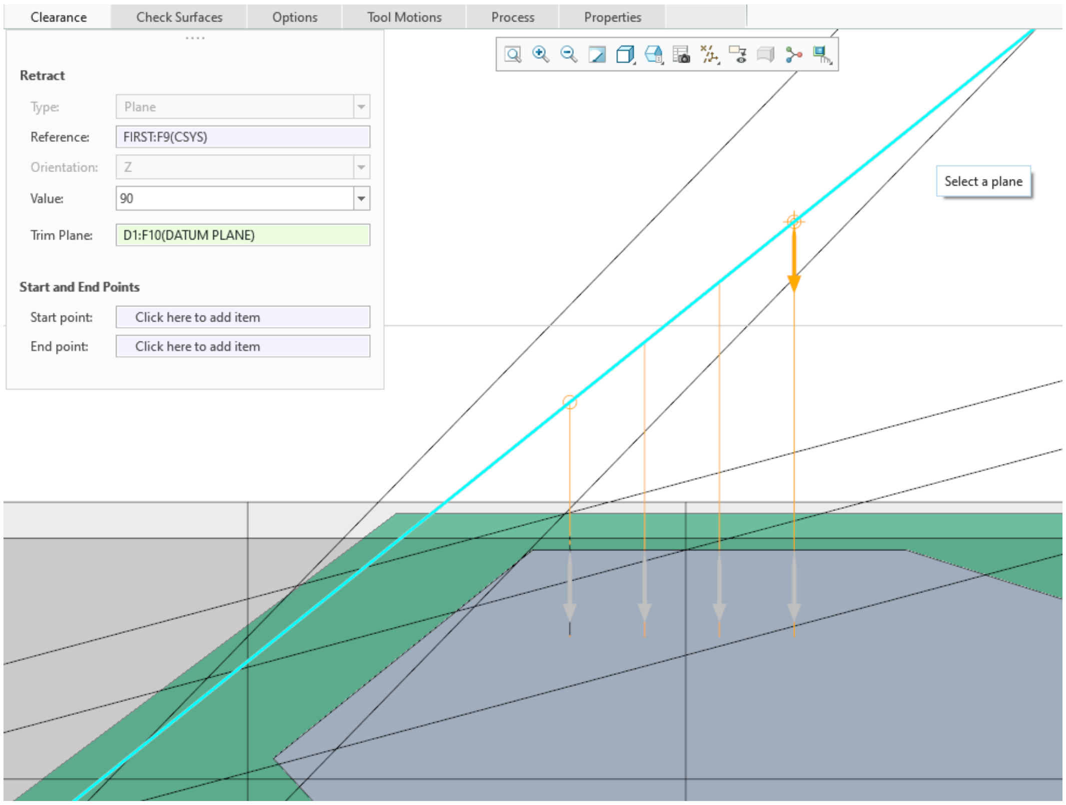
Videos

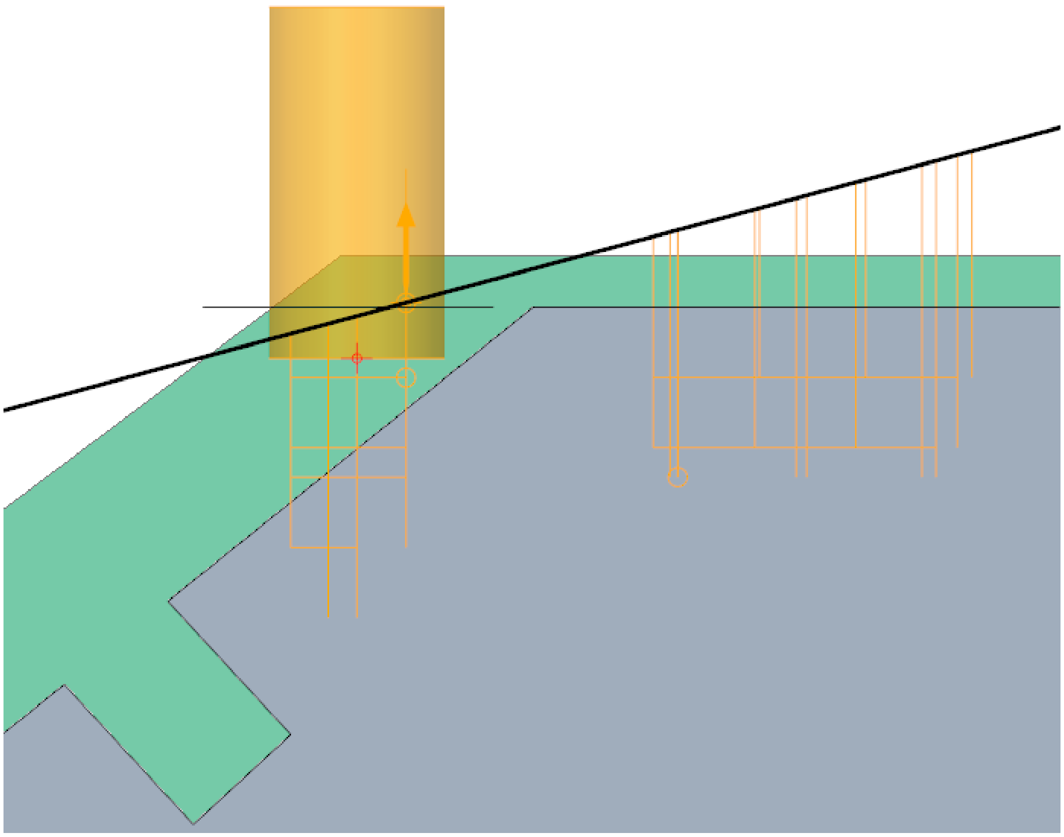
See the video on the Learning Connector.

Description

Use the new **Trim Plane** option for trimming or extending the retract move to a user-defined datum plane. This option is applicable for the following 3-axis Milling NC sequences:

- Volume milling
- Profile milling
- Face milling
- Trajectory milling
- Holemaking in Mill mode


Clearance
Check Surfaces
Options
Tool Motions
Process
Properties
Retract
Type:
Plane
Reference:
FIRST:F9(CSYS)
Orientation:
Z
Value:
90
Trim Plane:
D1:F10(DATUM PLANE)
Start and End Points
Start point:
Click here to add item
End point:
Click here to add item
Select a plane



Benefits

This enhancement provides the following benefits:

- More flexible retract options for milling
- Higher productivity with less time spent on retracts

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Engraving Toolpath Enhancements

Creo Parametric 11.0.0.0

User Interface Location:

1. In Manufacturing, click **Mill** ▸ **Milling** ▸ **Engraving**.
2. Select the **Parameters** tab.

Videos

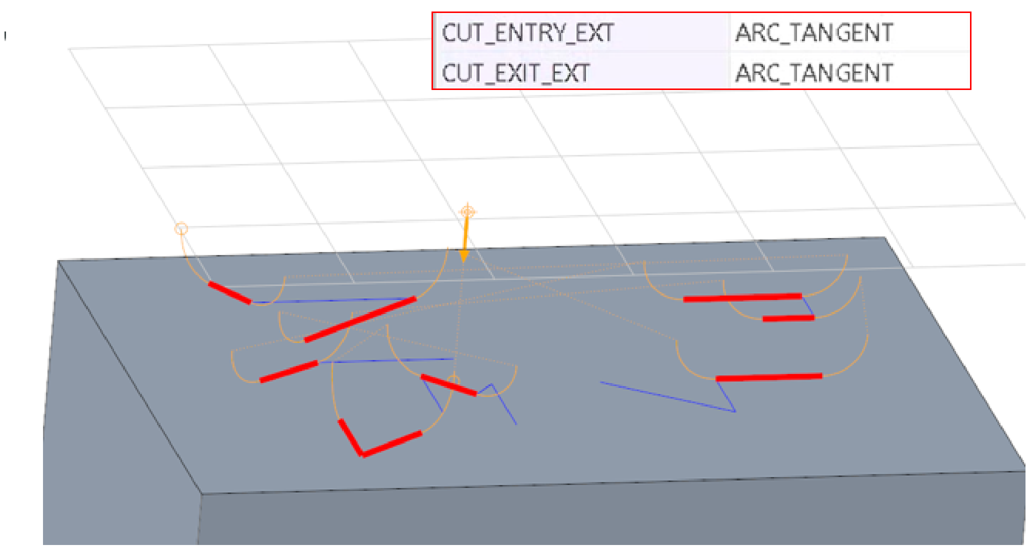
See the video on the Learning Connector.

Description

Previously, it was not possible to select individual curve segments from a single chain or multiple chains when creating an engraving NC sequence. It is now possible to select multiple curve segments or an individual curve segment from a single chain or multiple chains in the **References** collector and the **Chain** dialog box.

The new OPTIMIZE_LINKS parameter considers the shortest distance for connecting all curve segments in the specified order. The CUT_ENTRY_EXT and CUT_EXIT_EXT parameters with the ARC_TANGENT option is also now supported for the vertical entry and exit motions in engraving sequences.

These enhancements are supported for the 3, 4, and 5-axis engraving toolpaths.



Benefits

These enhancements provide the following benefits:

- Provide more flexibility and consistency
- Require fewer clicks to obtain the desired results

Additional Information

Tips:	None.
Limitations:	You can select only individual curve segments at a time. Combination of individual curve segments and whole chains is not supported.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Modernized 4-Axis Area Turning User Interface

Creo Parametric 11.0.0.0

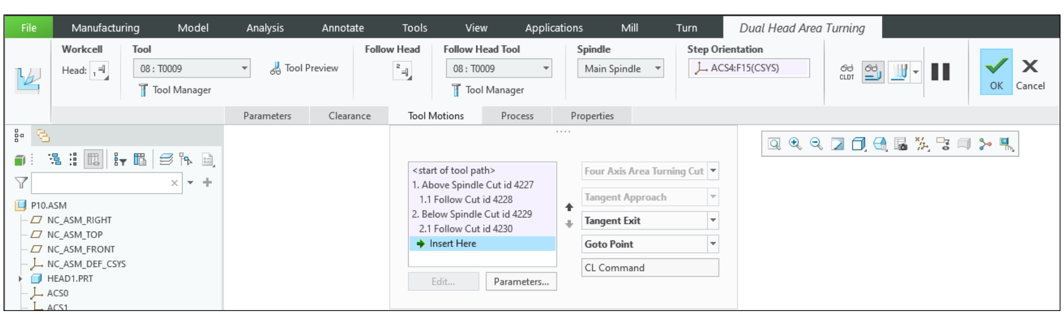
User Interface Location: In Manufacturing, click **Turn** ▸ **Turning** ▸ **Four Axis Area Turning**.

Videos

See the video on the Learning Connector.

Description

Previously, the **Four Axis Area Turning** command generated a toolpath based on the old menu manager. The UI for the **Four Axis Area Turning** command is now modernized and the old menu manager-based toolpath is no longer applicable for this command.



The CLEAR_DIST, TURN_FOLLOW_TPROF_DIR, and USER_OUTPOINT_POINT parameters are now supported for the 4-axis Area Turning sequences also. The CUTCOM statement at each slice in the sequence is also supported for 4-axis Area Turning.

Benefits

This enhancement provides the following benefits:

- Reduces menu manager-based toolpaths
- Leverages enhancements made for other toolpaths
- Provides a UI consistent with other toolpaths in Creo NC

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.

Configuration option associated with this functionality:	For CUTCOM support:
	`mfg_areaturn_cutcom_each_slice`
	`yes, no*`

Show or Hide Manufacturing Geometry

Creo Parametric 11.0.0.0

User Interface Location: Graphics Toolbar

Videos

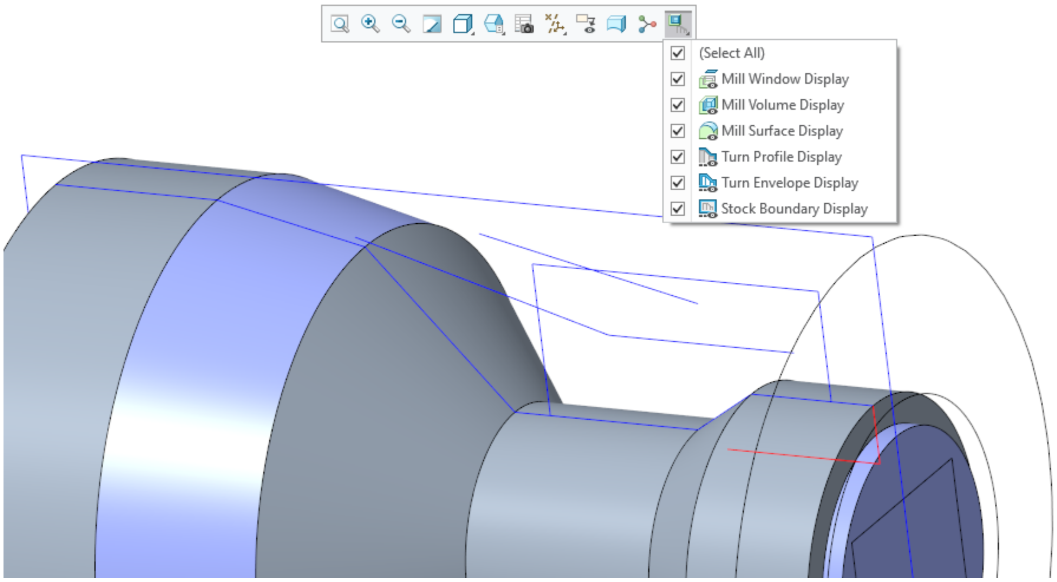
See the video on the Learning Connector.

Description

It is now possible to show or hide manufacturing-related geometry directly using the Graphics toolbar. Use **Mfg Geom Display Filters** in the Graphics toolbar for selecting the display filters and applying them to the following manufacturing features:

- Mill window
- Mill volume
- Mill surface
- Turn profile
- Turn envelope

- Stock boundary



Benefits

This enhancement provides the following benefits:

- More flexibility when selecting the manufacturing geometries
- Fewer clicks resulting in higher productivity

Additional Information

Tips:	Right-click the Graphics toolbar and select the **Mfg Geom Display Filters** checkbox. Activate the command to be included in the Graphics toolbar.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Separate CUTCOM Strategies at the Work Center Level

Creo Parametric 11.0.0.0

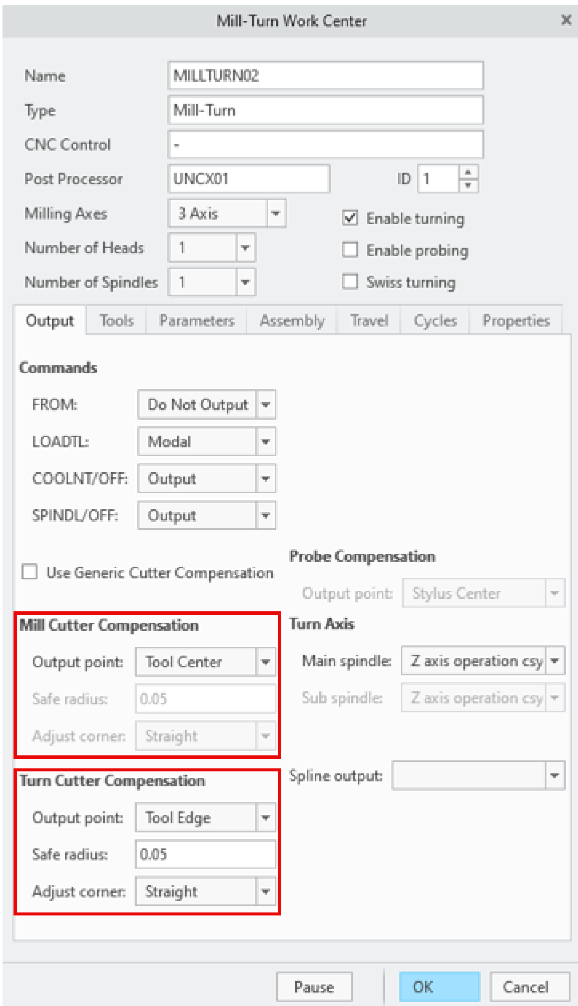
User Interface Location: Click **Manufacturing** ▸ **Work Center** ▸ **Mill-Turn**.

Videos

See the video on the Learning Connector.

Description

Previously, it was not possible to define cutter compensation separately for the milling and turning sequences at the work center level. With this new enhancement, the work center definition is improved to allow separate CUTCOM strategies for the milling and turning sequences. You can now define separate cutter compensation options for the milling and turning sequences, if required.



Benefits

This enhancement provides the following benefits:

- Provides flexibility to define different CUTCOM strategy for the milling and turning sequences
- Enables definition at the work center level

- Eliminates workarounds

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

New Option for Skipping CL Lines Unrelated to the Toolpath Motion

Creo Parametric 11.0.0.0

User Interface Location:

1. In Manufacturing, click in the tab of the sequence you are creating.

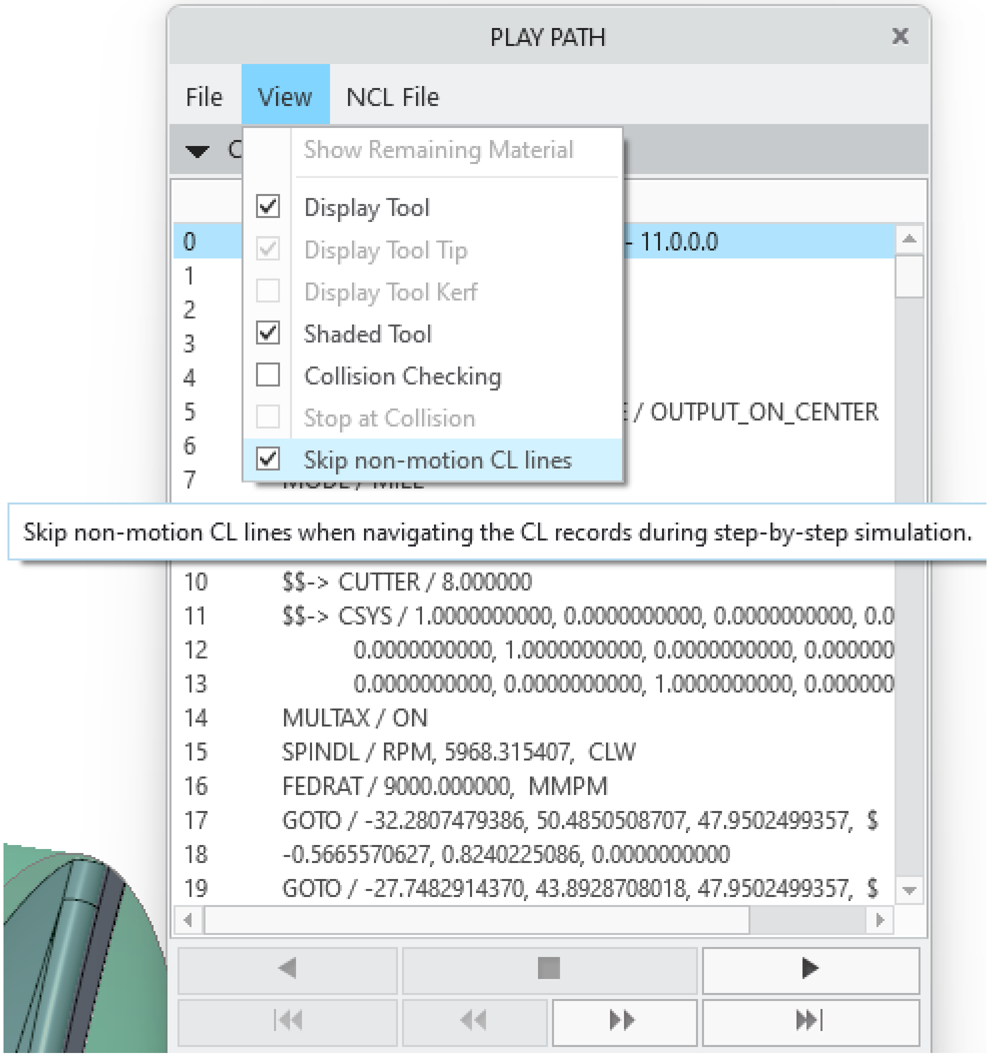
   The **PLAY PATH** dialog box opens.
2. Click **View**.

Videos

See the video on the Learning Connector.

Description

Previously, it was not possible to skip the CL lines unrelated to the toolpath motion when navigating the CL records. It is now possible to skip the CL lines that are not related to the toolpath motion when using the CL Player. You can skip such CL lines when navigating to the previous CL record or to the next CL record during the step-by-step simulation of the tool motion.



Benefits

This easy-to-use enhancement provides added flexibility when navigating CL records.

Additional Information

Tips:	None.
Limitations:	The CL records that are created when you insert CL commands using **CL Command** on the **Tool Motions** tab are not skipped.

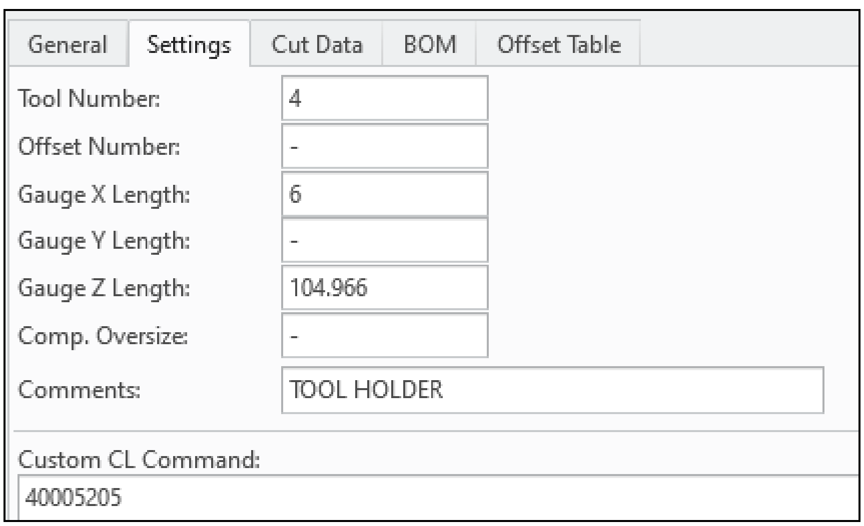
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

GAUGE_Y_LENGTH Parameter Support for the Tool Definition

Creo Parametric 11.0.0.0

User Interface Location:

1. In Manufacturing, click one of the following:
   - Click **Tool Manager** on the tab of the sequence that you want to create.
   - Click **Edit Tools** in the **Tool** list on the tab of the sequence that you want to edit.
2. Select the **Settings** tab.



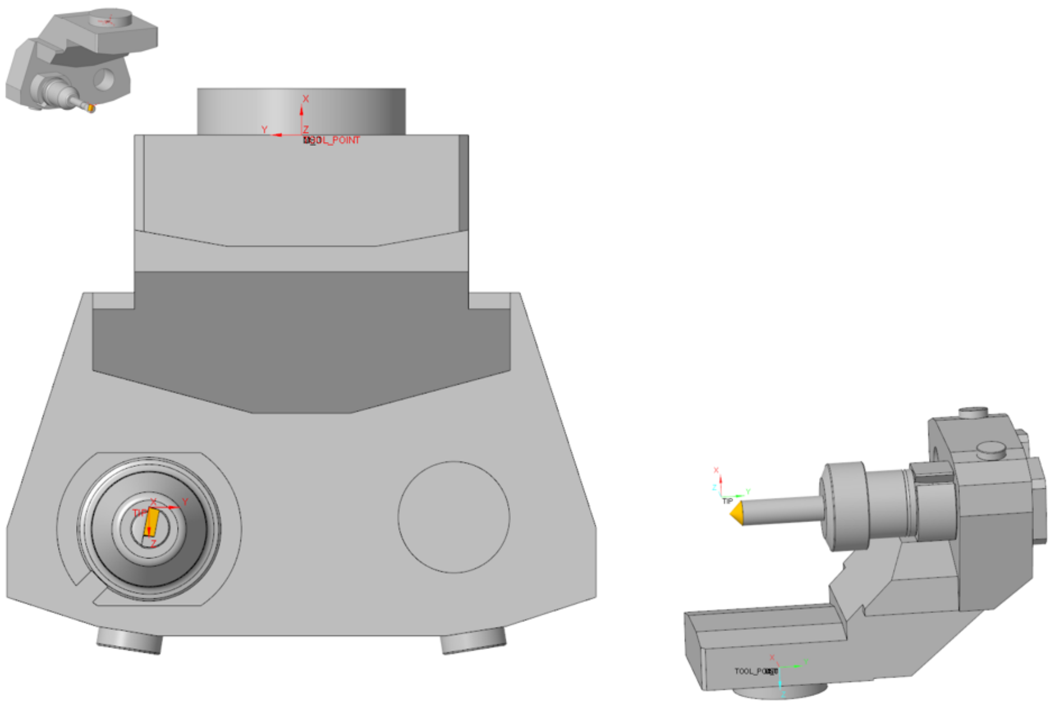
Videos

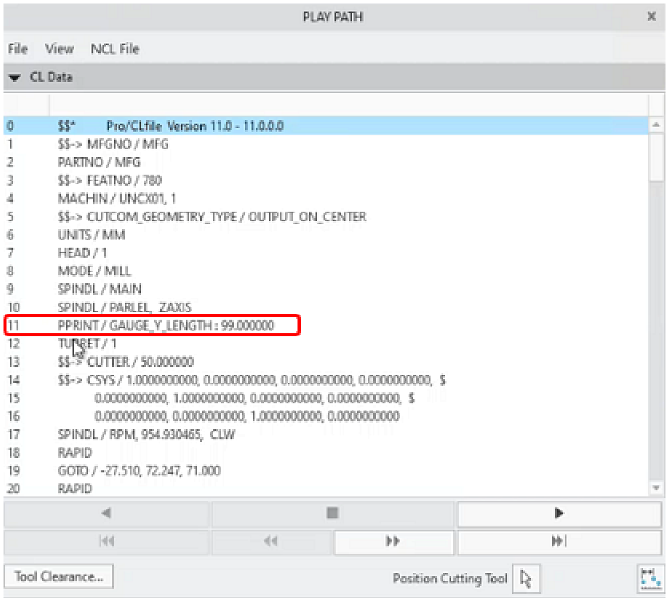
See the video on the Learning Connector.

Description

It is now possible to specify the Gauge_Y_Length parameter in the tool definition. You can include the parameter value in the CL data output using the PPRINT statement. The parameter value is available for use by the post-processor and simulation systems. The **Gauge Y Length** option is available for the following tools:

- Mill tools
- Turning tools
- Drilling tools





Benefits

This enhancement provides the following benefits:

- Unlocks an additional tool positioning option
- Provides more flexibility

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

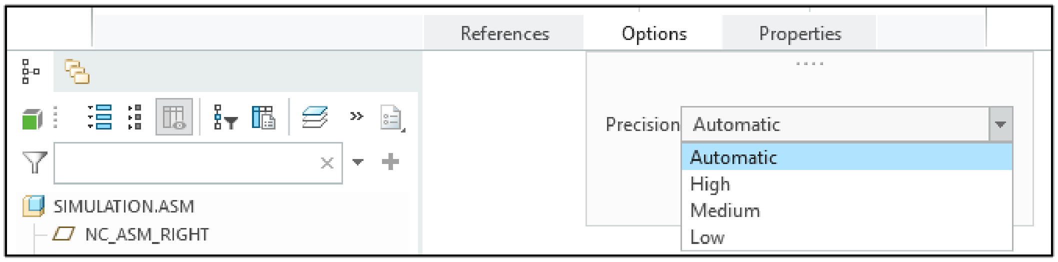
New Precision Option for the Stock Model

Creo Parametric 11.0.0.0

User Interface Location:

1. In Manufacturing, click **Manufacturing** ▸ **Stock Model** on the **Mill** tab, **Turn** tab, or **Wire EDM** tab.

2. Select the **Options** tab.



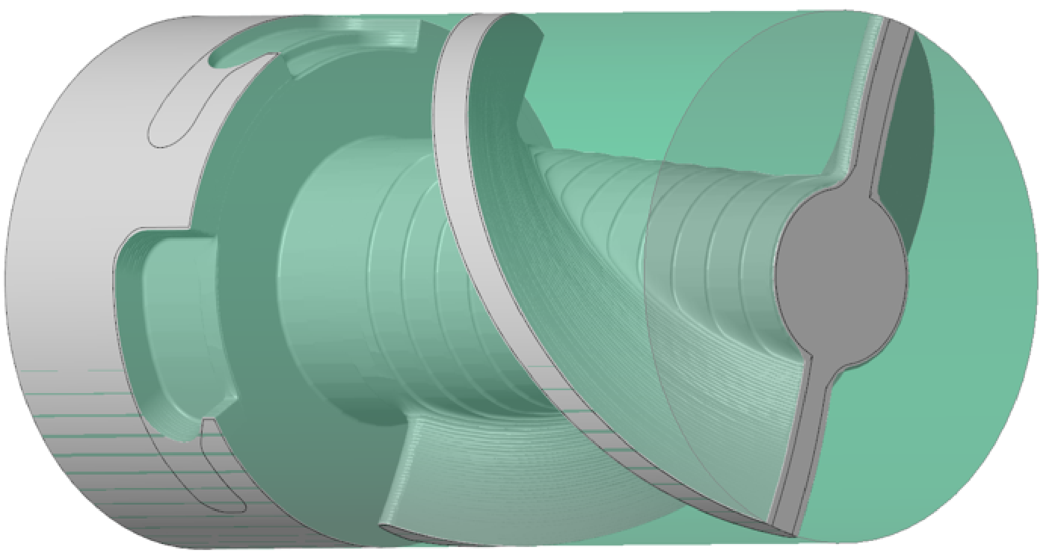
Videos

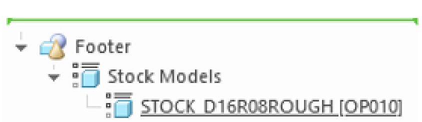
See the video on the Learning Connector.

Description

The new **Precision** option is now provided for stock models. The **Precision** option offers the following four precision settings:

- **Automatic**
- **High**
- **Medium**
- **Low**





Benefits

This enhancement provides the following benefits:

- Allows more control on stock model creation
- Provides more flexibility

Additional Information

Tips:	If you select the **Automatic** precision option, the precision settings from the default `mw_settings.xml` file are loaded for the stock model. Any updates to the precision settings in the `mw_settings.xml` file for material removal simulation is also applied to the stock model when using **Automatic**.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Enhanced Process Documentation

Creo Parametric 11.0.0.0

User Interface Location: In Manufacturing, click **Applications** ▸ **Process Documentation**.

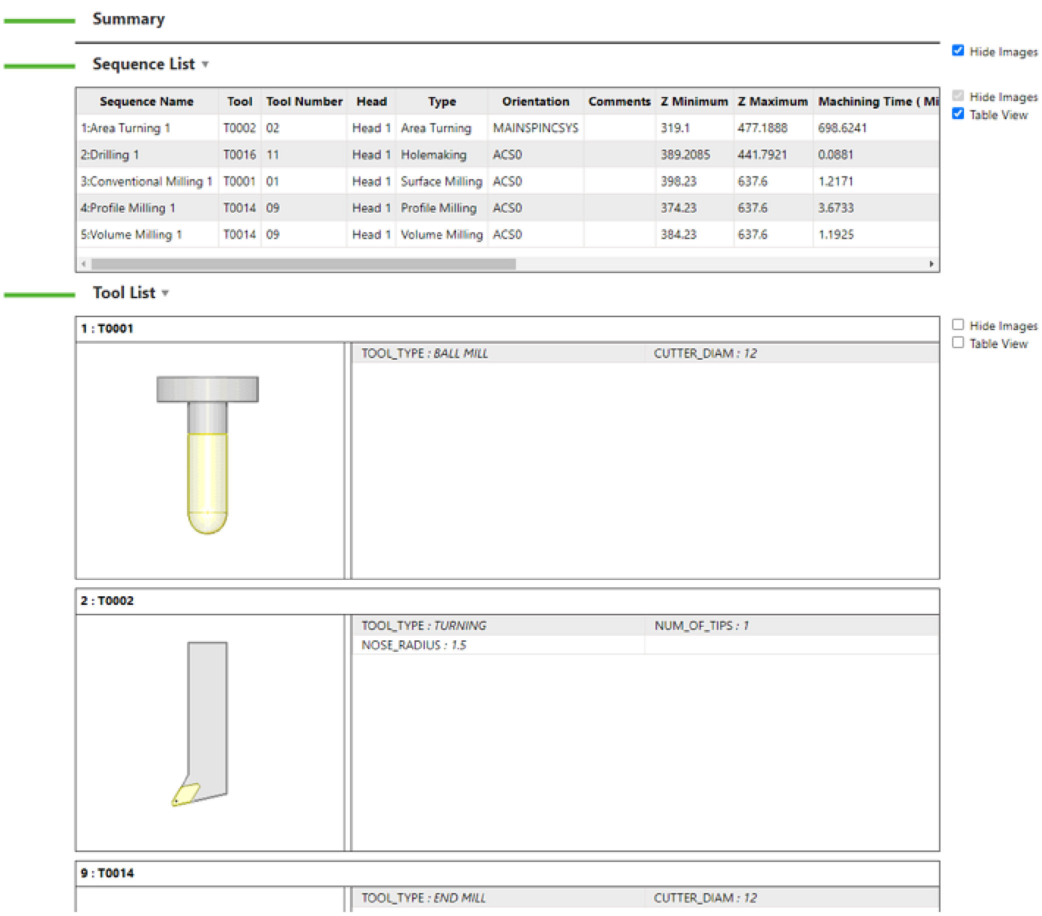
Videos

See the video on the Learning Connector.

Description

Process Documentation is now enhanced to provide an improved automatic shop floor report.

The image related to an operation in the **Summary** section of the customizable shop floor report includes the corresponding Program Zero coordinate system and the fixture defined for the operation. The manufacturing model and sequence toolpath images captured and included in the report are based on the current model view displayed in the graphics window. The automatic shop floor report opens in the Creo browser after it is generated. The report is saved in a folder.



Sequence Name	Tool	Tool Number	Head	Type	Orientation	Comments	Z Minimum	Z Maximum	Machining Time (Mi
1:Area Turning 1	T0002	02	Head 1	Area Turning	MAINSPINCSYS		319.1	477.1888	698.6241
2:Drilling 1	T0016	11	Head 1	Holemaking	ACS0		389.2085	441.7921	0.0881
3:Conventional Milling 1	T0001	01	Head 1	Surface Milling	ACS0		398.23	637.6	1.2171
4:Profile Milling 1	T0014	09	Head 1	Profile Milling	ACS0		374.23	637.6	3.6733
5:Volume Milling 1	T0014	09	Head 1	Volume Milling	ACS0		384.23	637.6	1.1925

Benefits

This enhancement provides the following benefits:

- Provides automatic shop floor documentation
- Requires fewer clicks to obtain the desired results

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

16

Model Analysis

Modelcheck Support for Multibody in Sheetmetal

Creo Parametric 11.0.0.0

User Interface Location: Click **File** ▸ **Prepare** ▸ **Model Check**.

Videos

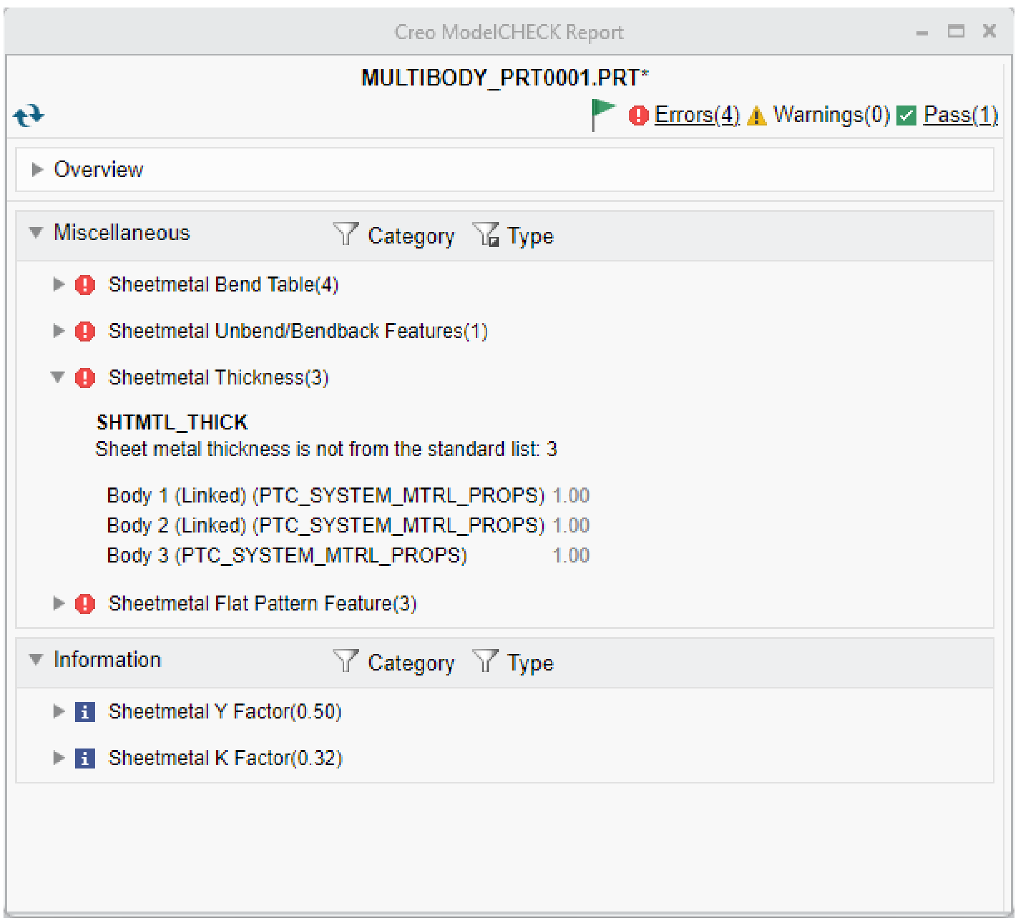
See the video on the Learning Connector.

Description

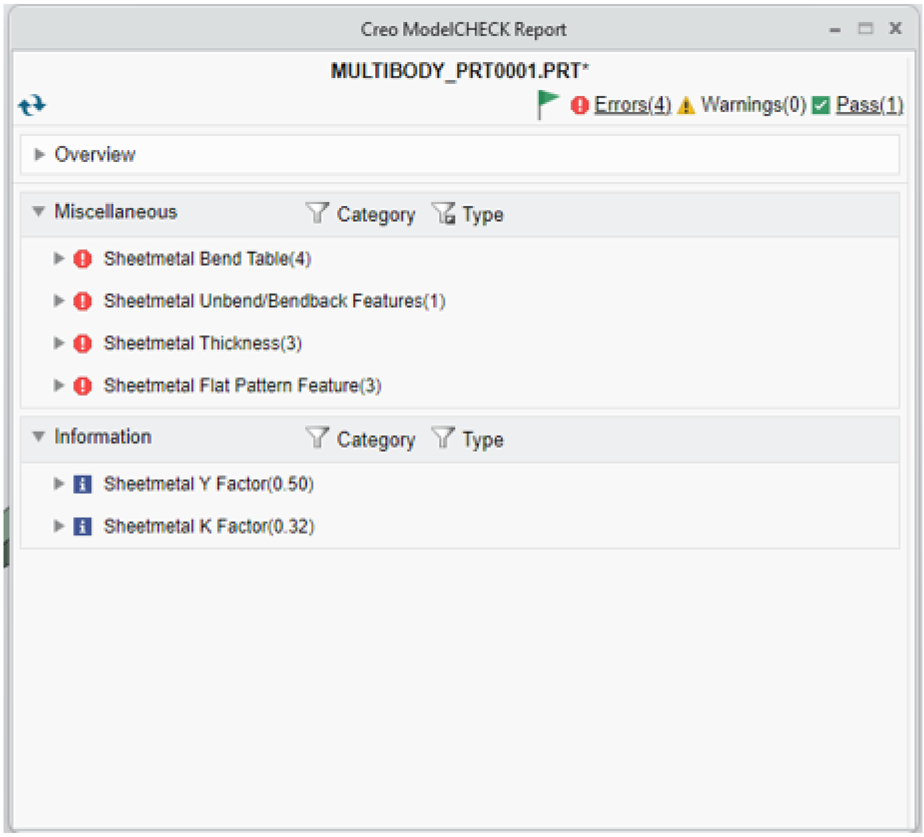
In Creo Parametric 11.0.0.0 model check capabilities are enhanced to accommodate full multibody support for sheet metal bodies.

The enhancements in Modelcheck include:

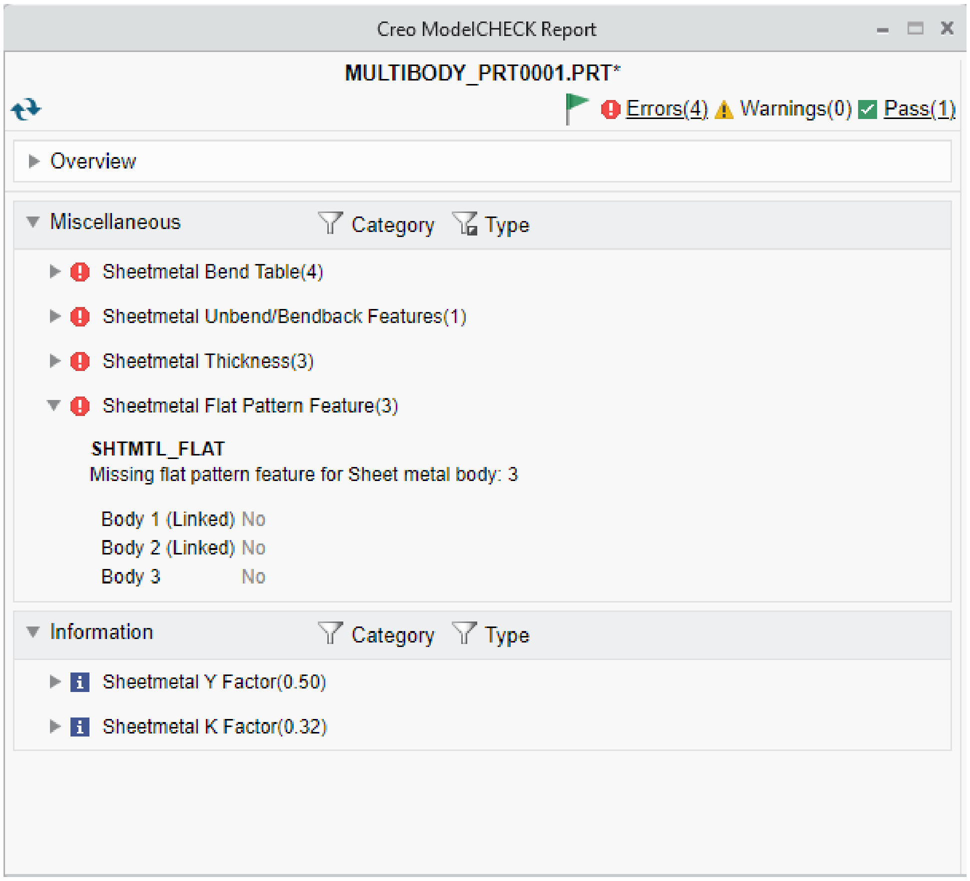
- Includes thickness checks for multiple materials.



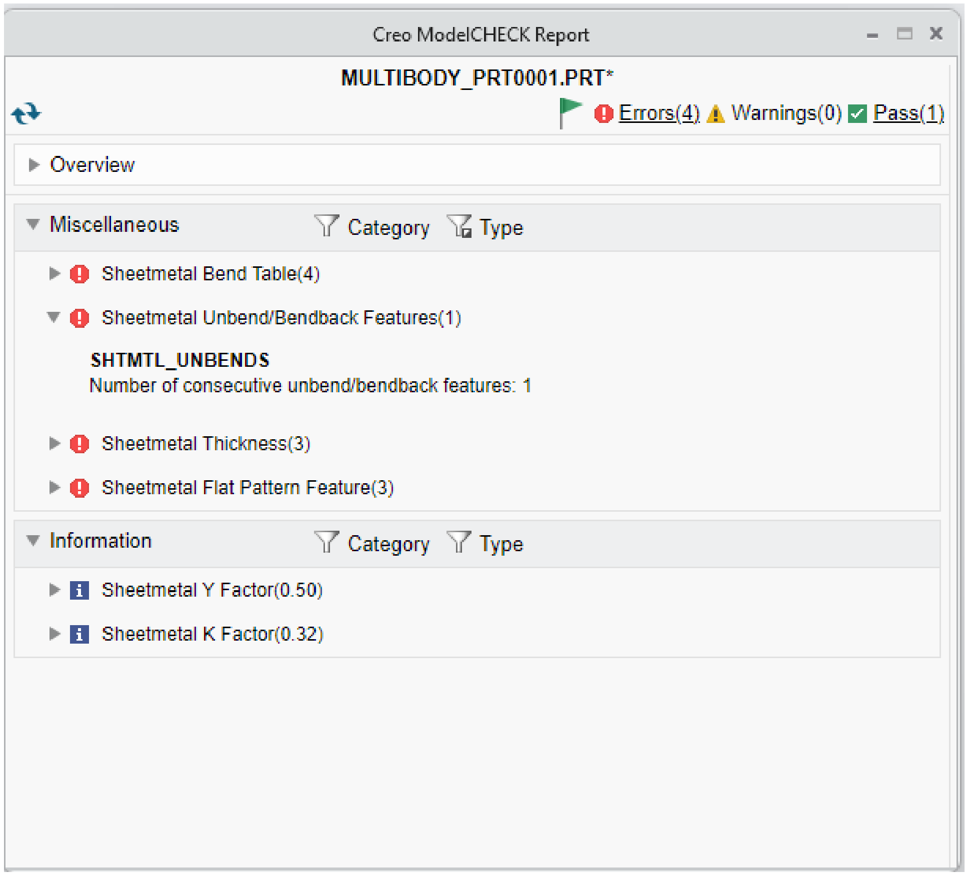
- Reports the Y and K factor for each sheet metal body.



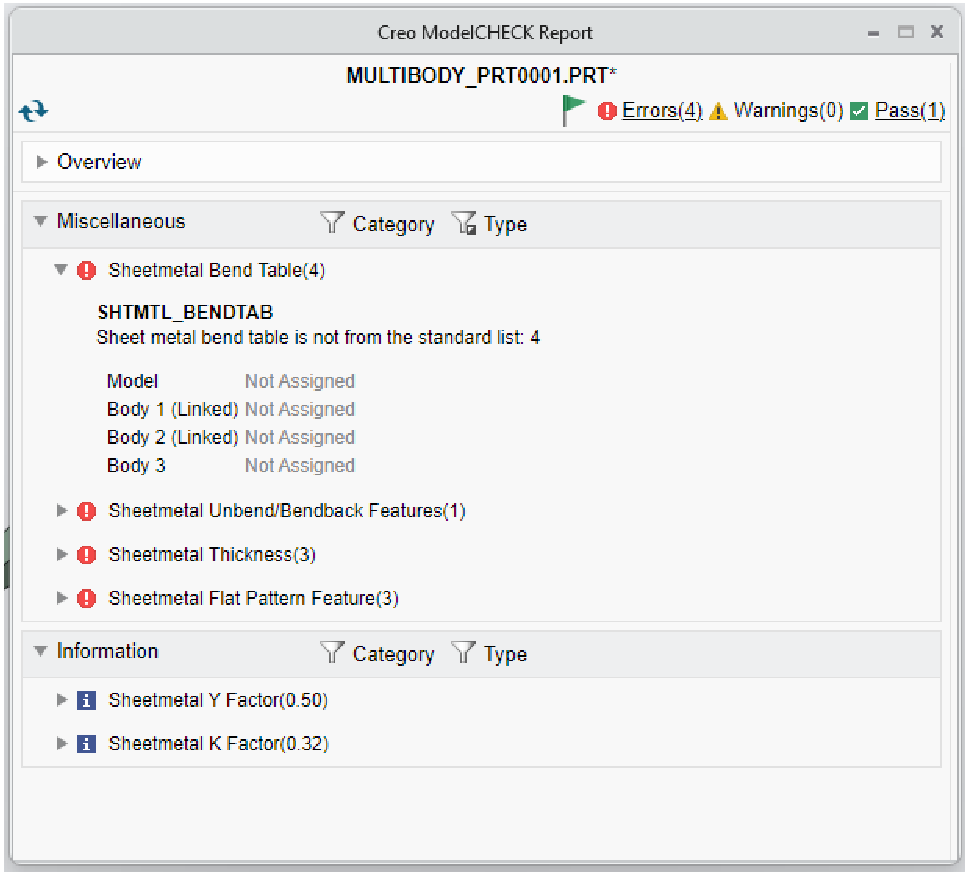
- New checks created for working with sheet metal multibody:
  - Check for existence of flat patterns in single and multibody parts.



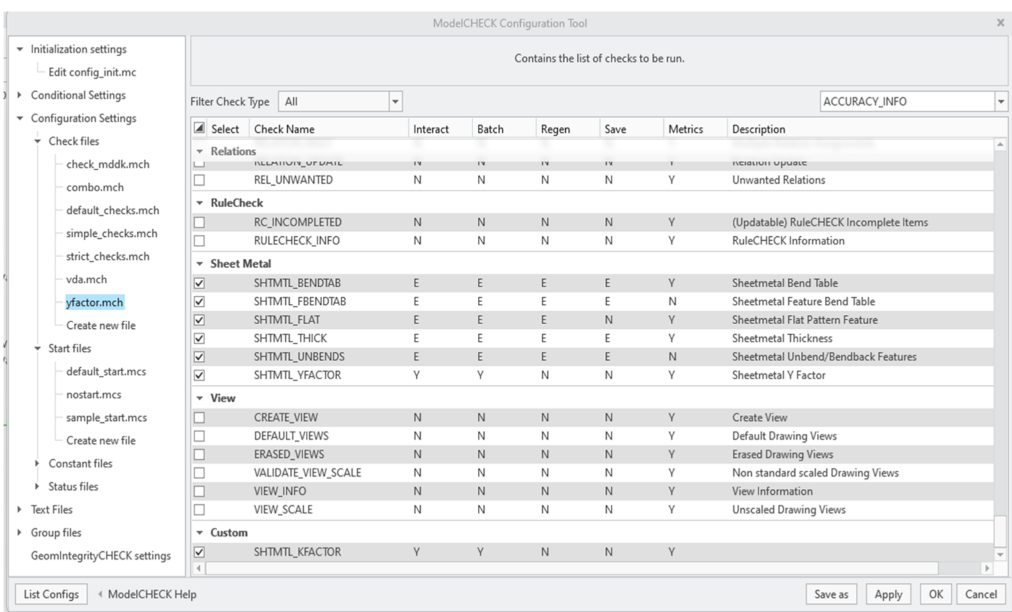
- Check for consecutive unbend and bend-back features in sheet metal parts.



- Check to specify the number of features in the model using the sheet metal bend table.



○ Check to specify which bodies are associated with their Bend Tables.



Benefits

Modelcheck can now be configured to fully cover sheet metal multibody designs.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Enhancement: Highlight Errors in the Model Tree

Creo Parametric 11.0.0.0

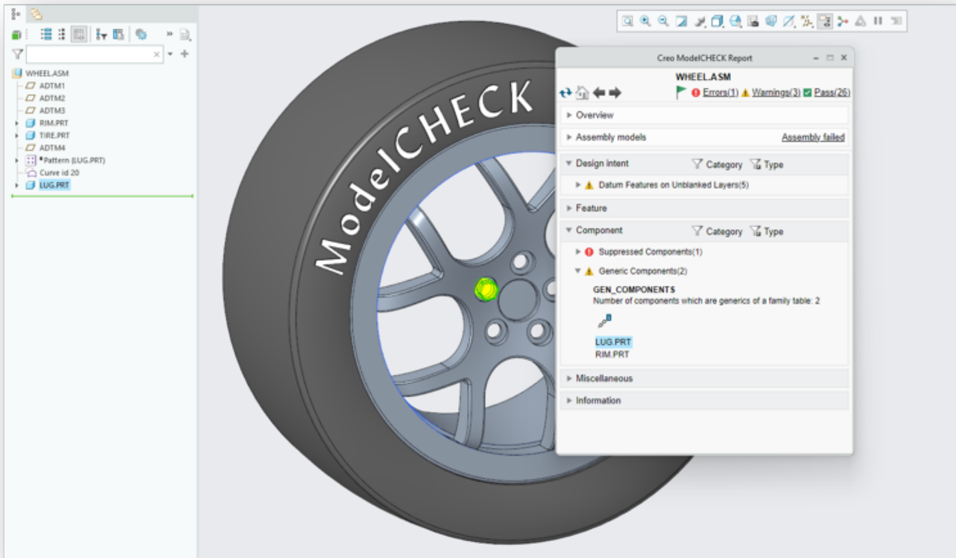
User Interface Location: In the ModelCHECK report, select a part. The corresponding part is highlighted in the Model Tree.

Videos

See the video on the Learning Connector.

Description

This enhancement highlights model errors in two places: within the model (as part of the existing functionality) and in the Model Tree. Highlighting the errors in the Model Tree helps in identifying the problematic area swiftly and speeding up the troubleshooting process, particularly when working with large models.



Benefits

- Improved usability when working in ModelCHECK.
- Improved workflow when troubleshooting errors detected by ModelCHECK.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Enhancement: Introducing Visual Indicators to the Summary Table

Creo Parametric 11.0.0.0

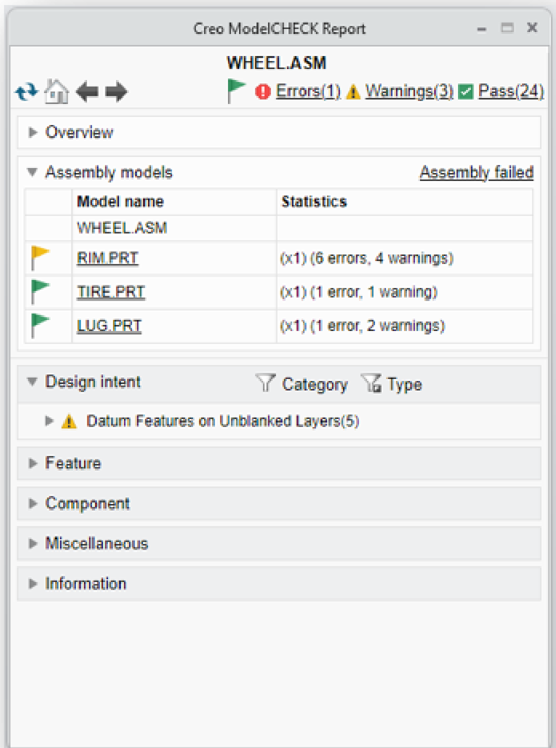
User Interface Location: In the ModelCHECK Report, under **Assembly models**.

Videos

See the video on the Learning Connector.

Description

Flag indicators have been added to the summary table to better indicate which objects in the model have passed or failed. Previously, Modelcheck showed only the status for the top-level model for an assembly or a drawing that was included in a Modelcheck report. With this enhancement, the report will provide a clear summary of each component within the assembly or drawing.



Benefits

Improved usability when working in ModelCHECK.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

New Flag for Indicating Passed Checks

Creo Parametric 11.0.0.0

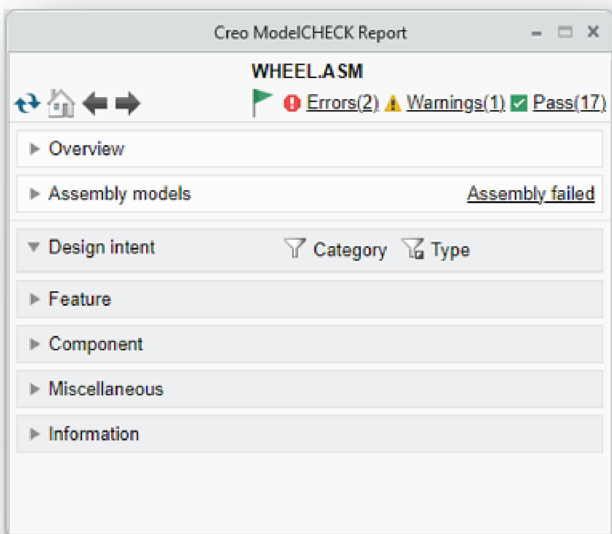
User Interface Location: In the ModelCHECK Report.

Videos

See the video on the Learning Connector.

Description

A new flag has been added in the ModelCheck report to indicate the number of checks that have passed. When you click **Pass** in the Modelcheck Report, the sections in the ModelCHECK Report are expanded to show a list of checks that are successful. You can further expand the individual checks to view more information for that check.



Benefits

- Improved usability when working in ModelCHECK.
- Provides a better insight and control when using ModelCHECK.
- Filters all the checks that were passed, providing a better understanding of the model.

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

New Condition to Check Since Last Saved Date

Creo Parametric 11.0.0.0

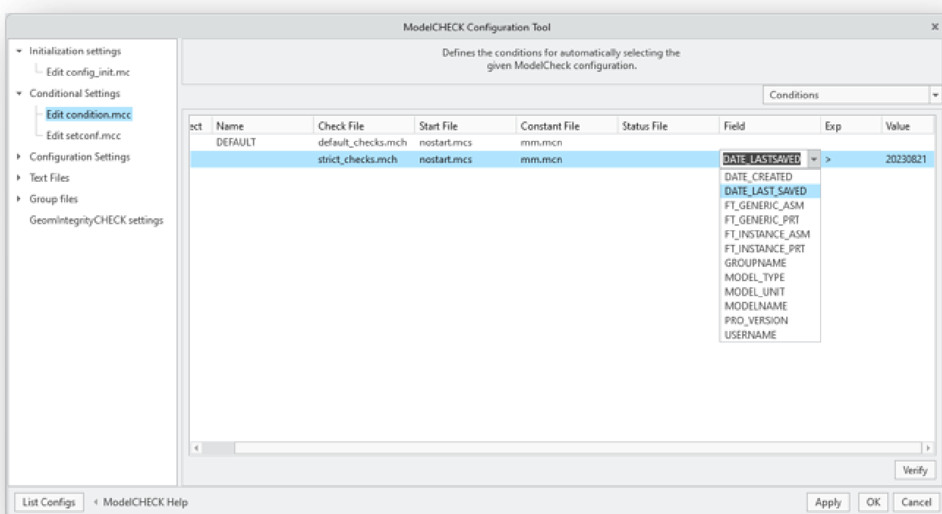
User Interface Location: **File ▸ Options ▸ Environment ▸ ModelCHECK settings ▸ Conditional Settings ▸ Edit condition.mcc ▸ Field**.

Videos

See the video on the Learning Connector.

Description

A new option has been added in the condition file to specify configuration based on the last saved date You can specify conditions for dates before, after, or within a specific range of the last saved date.



Benefits

Better insight and control when using ModelCHECK.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

New Check for Validating View Scale

Creo Parametric 11.0.0.0

User Interface Location: **File ▸ Options ▸ Environment ▸ ModelCHECK settings ▸ Conditional Settings ▸ Text Files ▸ view_scale.txt**.

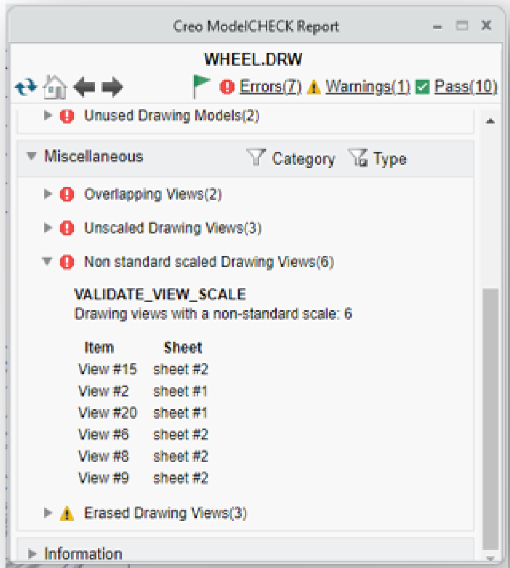
Videos

See the video on the Learning Connector.

Description

A new check has been added to validate the view scale in the models. This check is applicable only to drawings.

This check verifies whether the view scale specified on a drawing aligns with the recommended or previously determined values in the view scale list. With this enhancement, you have an option to highlight the reported view in the graphics window.



Benefits

- Improved usability when working in ModelCHECK.
- Improved workflow when troubleshooting errors detected by ModelCHECK.

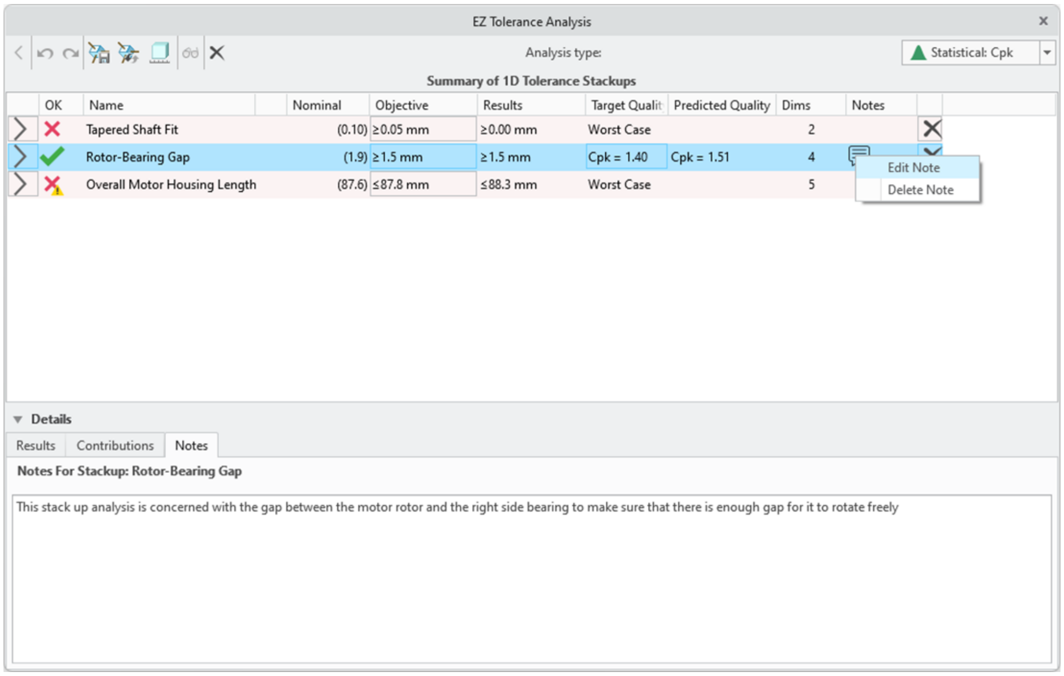
Additional Information

Tips:	To use this check, you must set up a check file in Modelcheck text files.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

EZ Tolerance Analysis Enhancement: Add Notes to Stackup

Creo Parametric 11.0.0.0

User Interface Location:



Videos

See the video on the Learning Connector.

Description

You can now add custom notes to stackups to provide additional information that is relevant to the stackup definition shown in the stackup report.

Custom notes can be added to the notes tab at the bottom of the stackup view, or by right-clicking the notes column of the desired stackup in the stackup summary view.

Benefits

With this enhancement, notes can be added to stackups, making it easier to provide necessary stackup information for stackup reports.

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

EZ Tolerance Analysis Enhancement: Improvements to the Stackup Report Generator

Creo Parametric 11.0.0.0

User Interface Location: Click **EZ Tolerance Analysis** ▸ **Generate Report**.

Videos

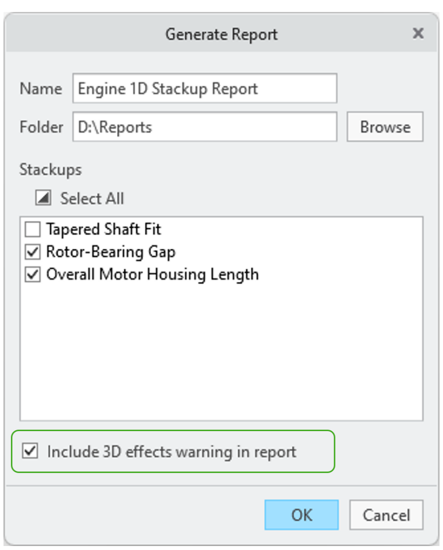
See the video on the Learning Connector.

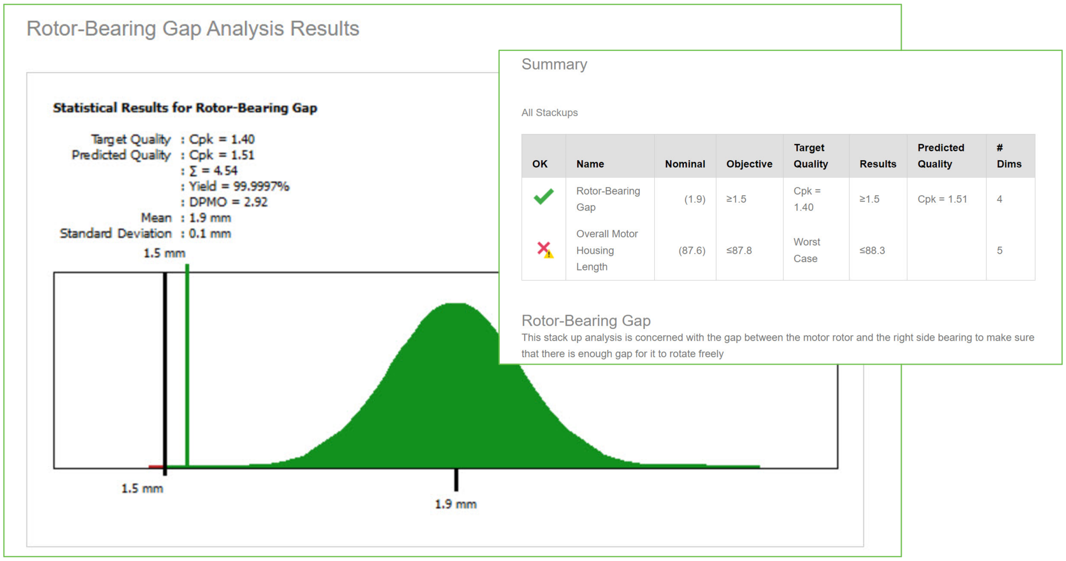
Description

With this enhancement you can,

- Select which tolerance stackups should be included while generating a stackup report.
- Choose to include 3D effects warning in the stackup report, which provides a warning that a 3D stackup analysis might be required for the selected stackup.

Additionally, now assembly shift bias like minimize, maximize, float, and center are also represented in the stackup reports.





Benefits

This enhancement provides additional control over content that is included in the generated stackup report.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

EZ Tolerance Analysis Enhancement: Nominal Value Defaults to the Measured Gap Between the Selected Components

Creo Parametric 11.0.0.0

User Interface Location: Click **EZ Tolerance Analysis ▸ Add Offset**.

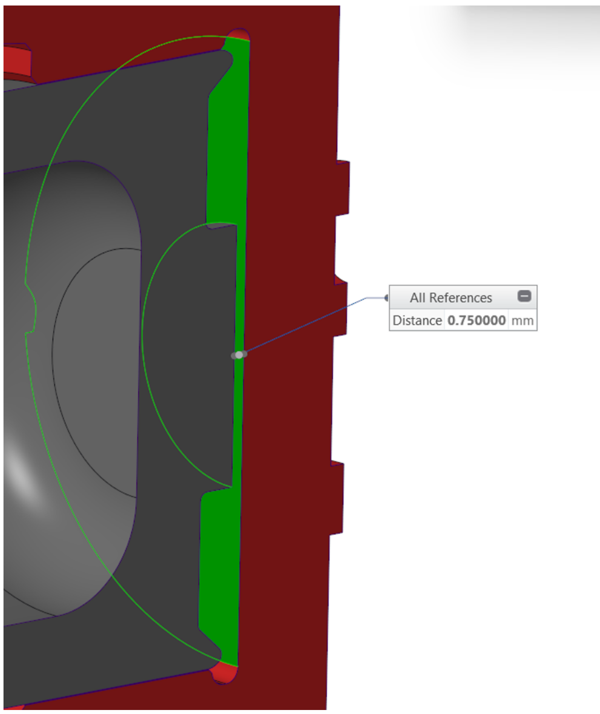
Videos

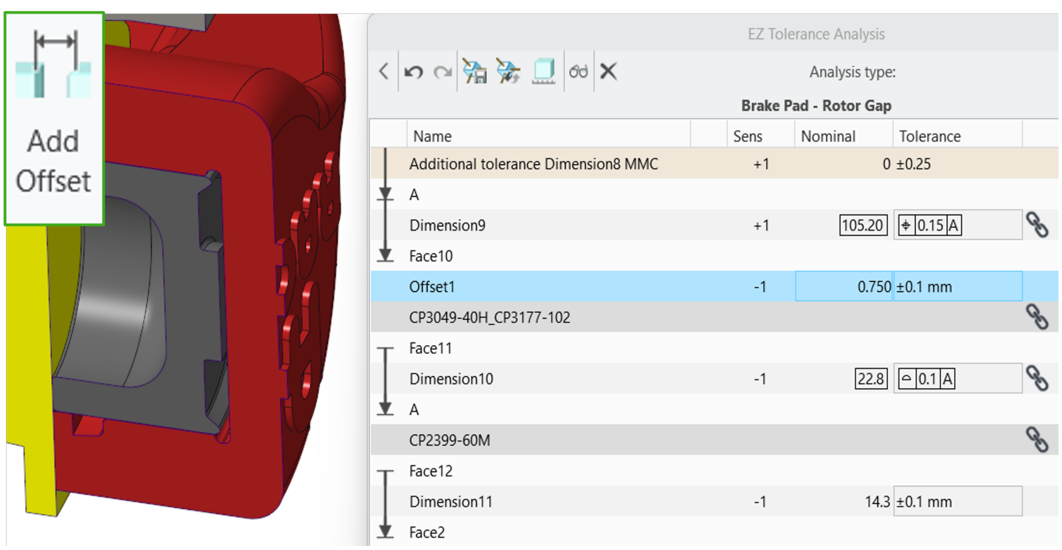
See the video on the Learning Connector.

Description

Previously, the value of offset was set to 0; the gap was measured manually, and then the value was updated accordingly by the user.

In the current release, when offset is added to a stackup between two components, the default nominal value is set as the measured distance between the two components.





Benefits

With this enhancement, less time is spent on manual measurements and updates to the stackup table.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

EZ Tolerance Analysis Enhancement: Support for Drafted Features of Size

Creo Parametric 11.0.0.0

User Interface Location: N/A

Videos

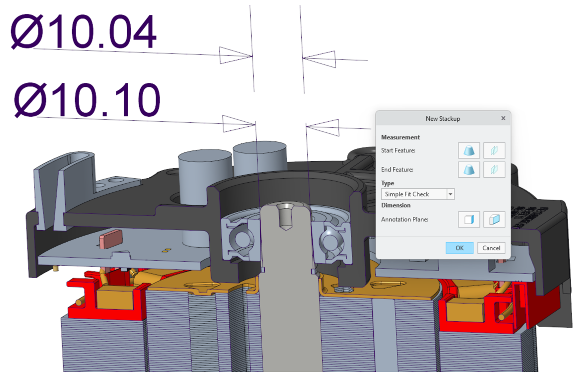
See the video on the Learning Connector.

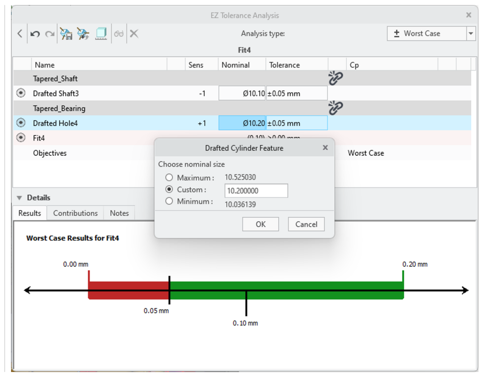
Description

Drafted features such as drafted slabs, slots, holes, and shafts can now be defined as part of a tolerance stackup in EZ Tolerance Analysis.

Only drafted features whose draft angle is below the threshold are included in the stackup definition. The default threshold is 5 degrees; this value can be adjusted in the `EZTAAppOptions.xml` options file.

The next images show drafted features that are added during the stackup creation, illustrating how they appear in the stackup table.





Benefits

This enhancement has improved flexibility when defining 1D tolerance stackups.

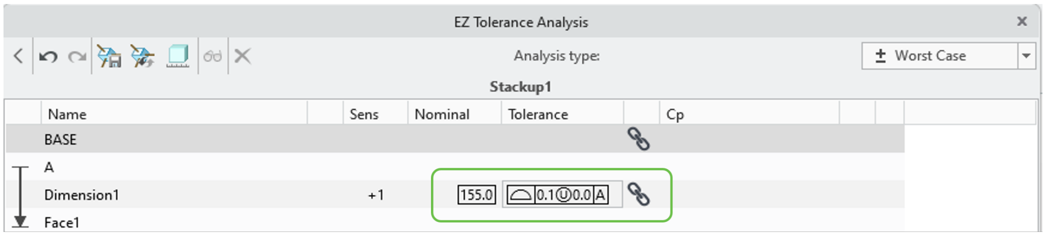
Additional Information

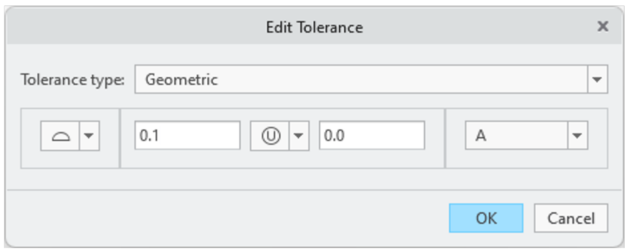
Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

EZ Tolerance Analysis Enhancement: Support for Unequally Disposed Profile Tolerances

Creo Parametric 11.0.0.0

User Interface Location:





Videos

See the video on the Learning Connector.

Description

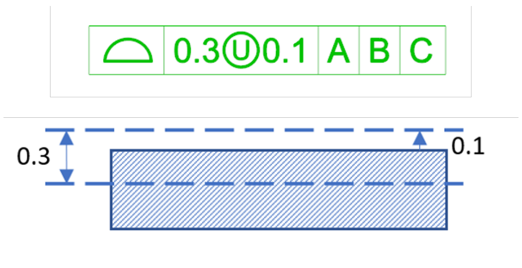
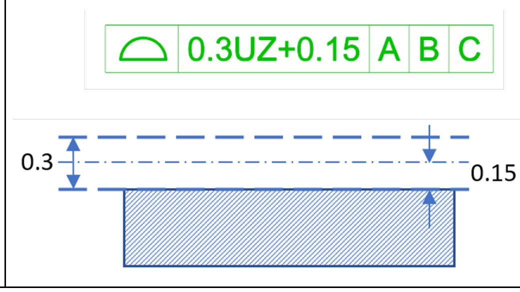
EZ Tolerance Analysis now supports unequally disposed profile tolerances for both ASME/ISO GPS models. The following types of geometrical tolerances support these specifications:

- Profile of a surface
- Profile of a line

Unequally disposed profiles can be defined for features directly from the EZ Tolerance Analysis Stackup user interface, or they can be validated as linked annotations from an existing geometrical tolerance with semantic references defined. The values that are used by the unequally disposed profile are used in stackup analysis calculations and are displayed in the results.

Unequally disposed profile tolerances are indicated differently in ASME/ISO, and the proper syntax is also different. In ASME, this specification is indicated with the Ⓤ symbol, while in ISO this specification is called Specified tolerance zone offset and is indicated with the letters `UZ`.

The table below explains how this specification is used in ASME/ISO cases with examples.

<table>
<tr>
<td>In this ASME example, the profile zone has been shifted, so that <code>0.1</code> is outside the material and <code>0.2</code> is inside the surface.

</td>
<td>In this ISO example, the center of the profile zone has been shifted to be <code>0.15</code> outside the material of the surface.

</td>
</tr>
</table>

Benefits

With this enhancement, models with unequally disposed profiles can now be validated for a 1D stackup analysis, and compliance with the ASME/ISO detailing standards has been improved.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

EZ Tolerance Analysis Enhancement: New XML Options File for Managing Application Settings

Creo Parametric 11.0.0.0

User Interface Location: N/A

Videos

See the video on the Learning Connector.

Description

The EZ Tolerance Analysis application options can now be managed by using the `EZTAAppOptions.xml` file that is installed with Creo. The application options are controlled with a dedicated XML tag, and CAD administrators can now preconfigure this XML file to set specific settings consistently for all users in the organization.

```
<EZtolSettings majorVersion="2" minorVersion="7">
    <Audio>
        <Enabled>true</Enabled>
    </Audio>
    <DefaultValues>
        <AnalysisType>WorstCaseAnalysisType</AnalysisType>
        <Document>
            <CpValue>1</CpValue>
        </Document>
        <DraftAngle>5</DraftAngle>
        <Tolerance>
            <Metric>
                <Linear>0.1</Linear>
                <Geometric>0.2</Geometric>
                <FeatureOfSize>0.05</FeatureOfSize>
            </Metric>
            <USCustomary>
                <Linear>0.010</Linear>
                <Geometric>0.020</Geometric>
                <FeatureOfSize>0.005</FeatureOfSize>
            </USCustomary>
        </Tolerance>
        <QualityMetric>
            <Type>CpkUnits</Type>
            <CpkValue>1</CpkValue>
            <DPMOValue>1349.8125</DPMOValue>
            <SigmaValue>3</SigmaValue>
            <YieldValue>99.73</YieldValue>
        </QualityMetric>
    </DefaultValues>
```

Benefits

With this enhancement, application settings can now be managed and updated in a centralized way.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	CAD administators can define the folder location of the `EZTAAppOptions.xml file` by using the `ezta_app_options_file` option. The default location of the `EZTAAppOptions.xml` is <Creo_Install_Home>\Common Files\applications\EZTOL

17

Model-Based Definition

Enhancement: Layer States Availability for Default All Combination State

Creo Parametric 11.0.0.0

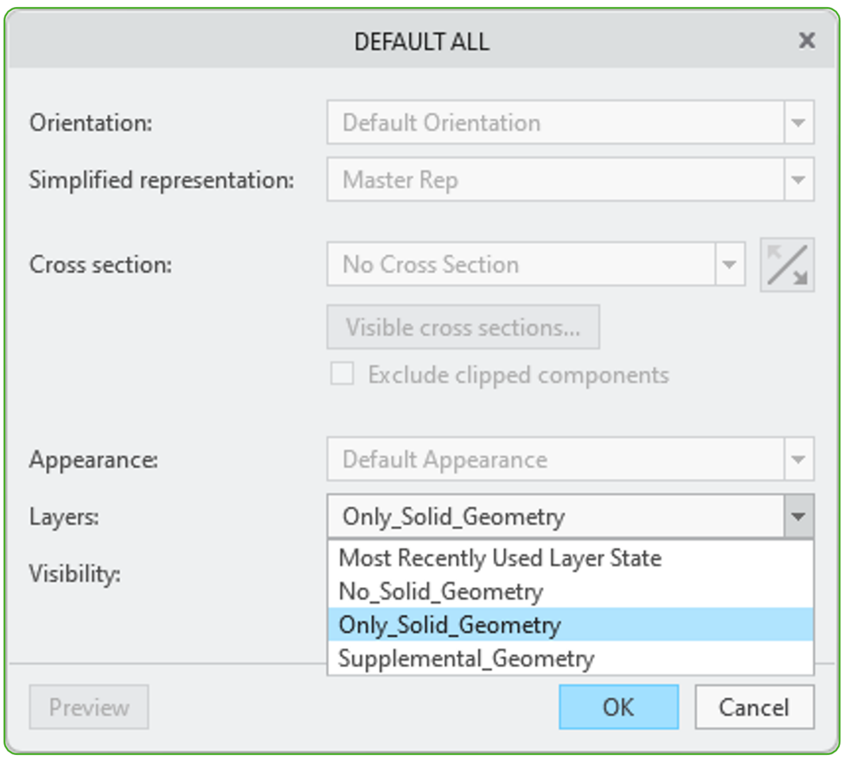
User Interface Location:

1. Click **View Manager**, on the **ALL** tab select **Default All**.
2. Click **Edit** and then select **Edit Definition**. The **DEFAUT ALL** dialog box opens.

Description

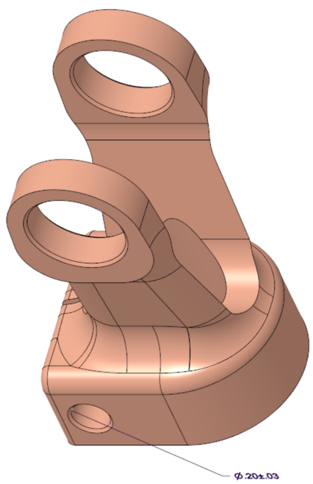
You can now define any layer state for the **Default All** combination state.

With this enhancement, any predefined layer state can now be set as the active layer state for the **Default All** combination state. This provides better control when defining the visibility of different items for the **Default All** combination state using layer states

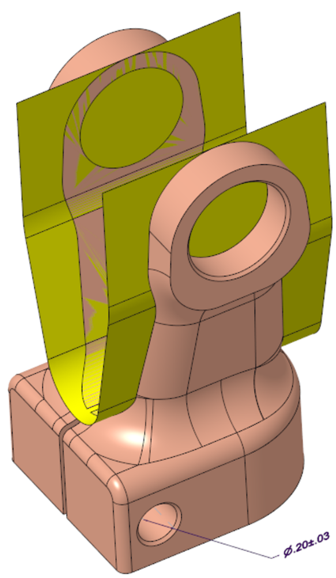


Examples of layer states:

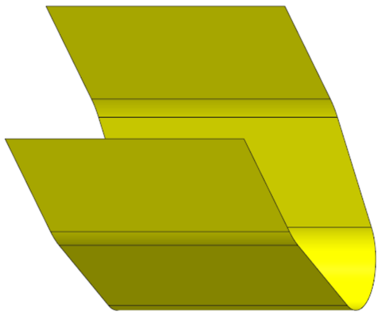
- Only_Solid_Geometry



- Supplemental_Geometry



- No_Solid_Geometry



Benefits

Improved control over the visibility of Creo entities in the **Default All**combination state.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Improved Selection of Cylindrical Surfaces for MBD Annotations

Creo Parametric 11.0.0.0

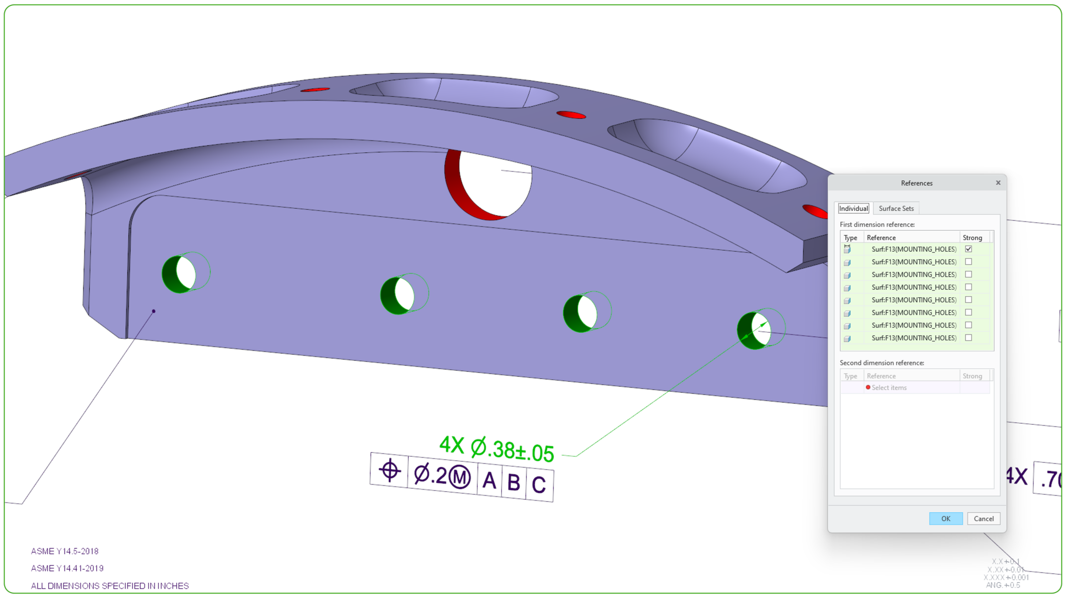
Videos

See the video on the Learning Connector.

Description

When an annotation is placed on a cylindrical feature such as a hole or shaft, both cylinder halves are now selected as a semantic reference for this annotation. This enhancement eliminates the need to select the second half manually.

The automatic collection of surfaces is supported during the first-time placement of the annotations, and when new cylindrical instances are being added. This behavior is supported for all types of annotations.



Benefits

- Simplified selection of multiple cylindrical features requiring fewer clicks.
- Improved usability when selecting holes or shafts as semantic references of annotations.

Additional Information

Tips:	None.
Limitations:	The automatic selection of the second half of the cylinder is not supported for the advanced collection methods like seed-and-bound. If a cylinder is divided using the divide surface feature, Creo will not collect the second half of the cylinder.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Semantic Query Tools Now Supports Inheritance Models

Creo Parametric 11.0.0.0

User Interface Location: Click any one of the following:

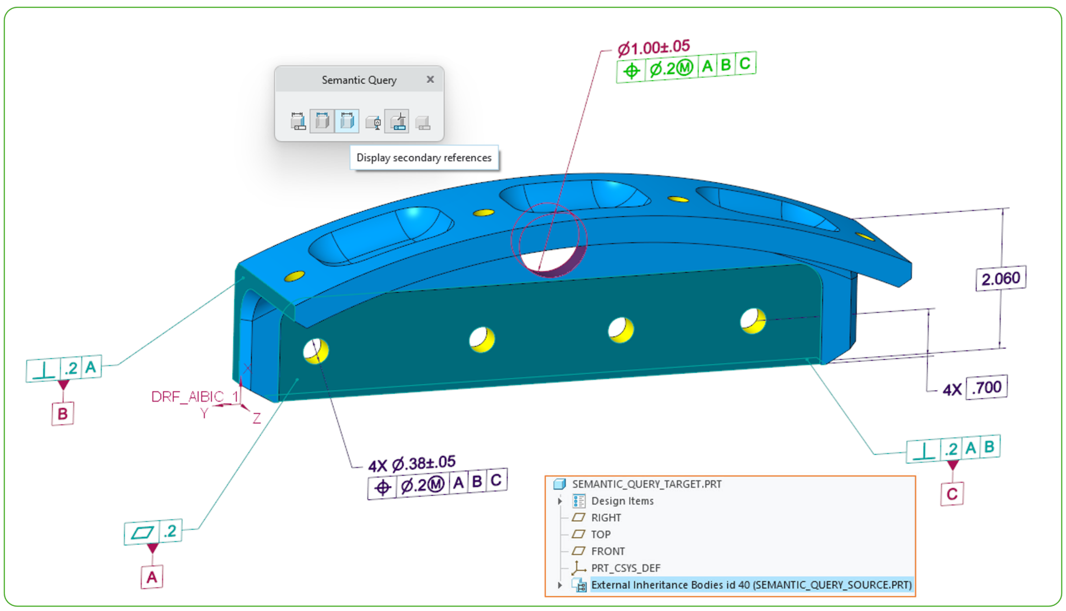
- **Annotate ▸**  **Semantic query**
- Select an annotation and then click  from the context menu.

Videos

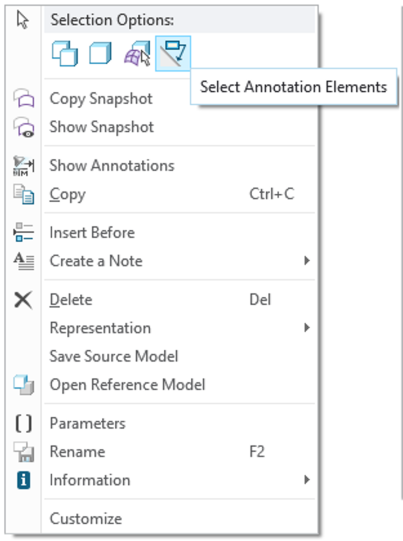
See the video on the Learning Connector.

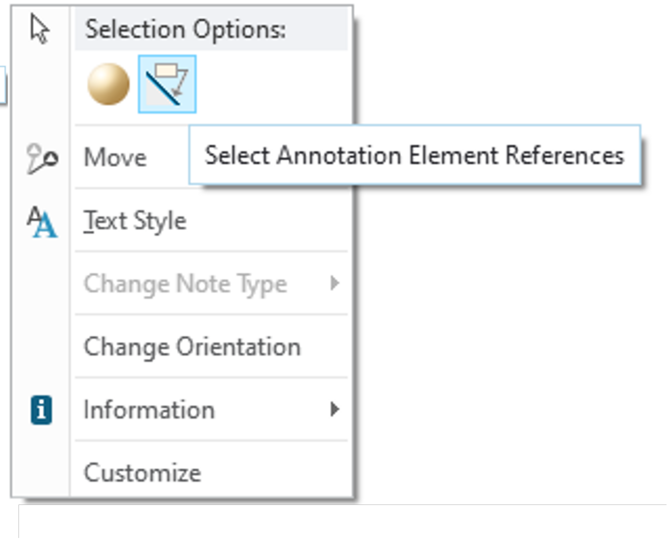
Description

With this enhancement, the semantic query tool now supports inheritance models. It can query annotations from the source and the inheritance model, enabling you to understand their associativity with the model surfaces.



Additionally, for inheritance models, you can now use the annotation element query commands for selecting the associated annotation elements or their references. These commands are available in the contextual menu.





Benefits

- Better understanding of annotation semantics for inheritance models.
- Quick and easy way to understand the associativity between annotations and the model surfaces, without having switch to the original model.

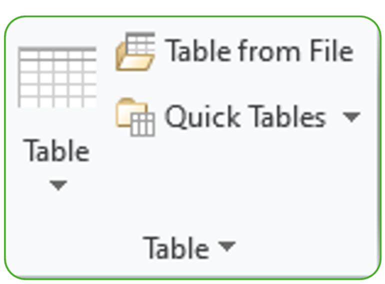
Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Create Tables in Model-Based Definition

Creo Parametric 11.0.0.0

User Interface Location: Click **Annotate** ▸ **Table** group.



Videos

See the video on the Learning Connector.

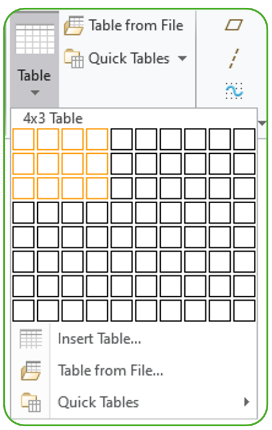
Description

This enhancement supports the creation of simple tables in Model-Based Definition (MBD).

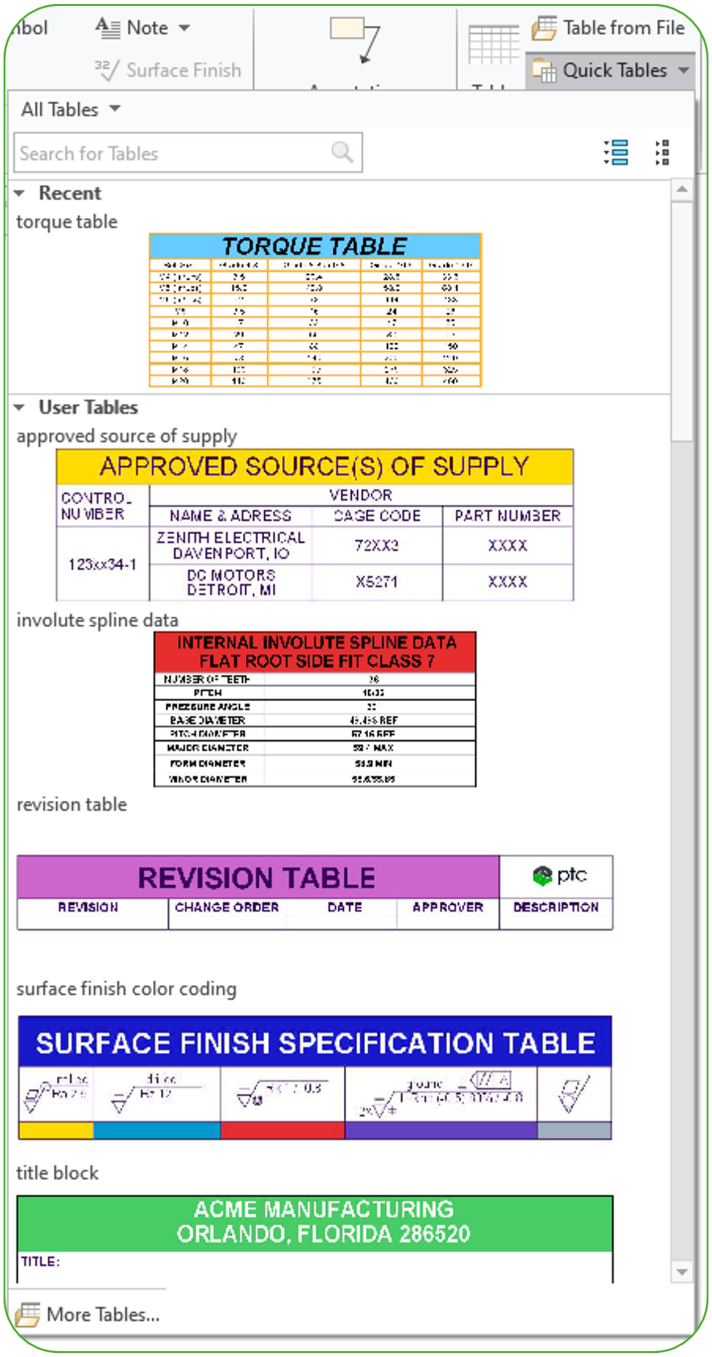
You can now create tables in MBD that can be placed on any annotation plane or set as flat-to-screen. The tables can be assigned to combination states like other supported annotations.

You can create a table in MBD using these methods:

- Use the table grid or the **Insert Table** command—Select the desired number of rows and columns when creating an empty table.
- Retrieve a table from an existing file—Retrieve predefined tables from supported file formats such as `*.CSV`, `*.XLSX` (Excel spreadsheet) or `*.tbl` files.



- Insert a table from the **Quick Tables** gallery—In the gallery, you can preview system-defined and user-defined tables. Search and pin the relevant table to be placed in the model.



After a table is selected using any of the above options, it becomes attached to the cursor. You can then choose the desired placement location for the table in the MBD model.

Benefits

The tables in MBD provide a quick and easy way to add and organize the engineering data in a tabular form.

Additional Information

Tips:	The default path from which the user-defined tables are retrieved can be defined using the `pro_table_dir` config.pro option.
Limitations:	Currently, Creo does not support tables with repeat regions. When you place a table with repeat regions, a message appears in MBD, and the table is placed as a simple table.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Tables in Model-Based Definition as Security Markings

Creo Parametric 11.0.0.0

User Interface Location:

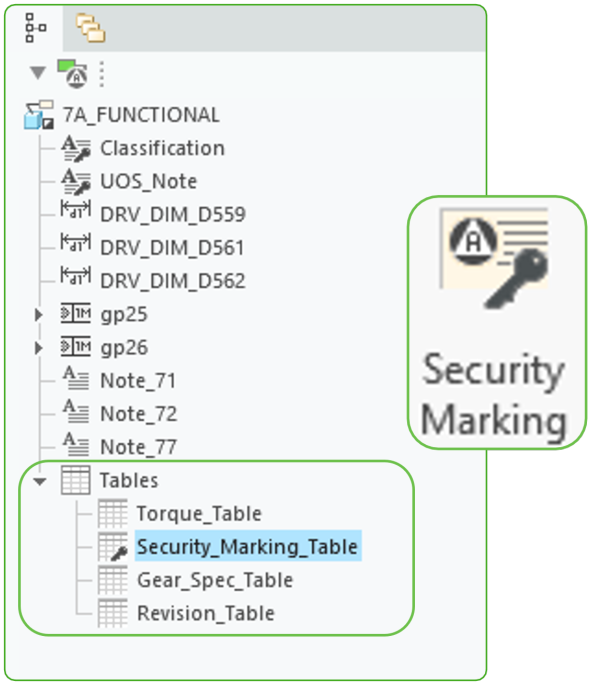
1. Select a table that was placed as flat-to-screen.
2. On the **Format** tab, click **Security Marking** in the **Format** group.

Videos

See the video on the Learning Connector.

Description

The flat-to-screen tables in MBD can be marked as a security marking table and added to all the existing combination states and also to the combination states that will be created later. The security marking tables are denoted in the Model Tree and Detail Tree with a special icon.



Benefits

This enhancement provides an easy and convenient way for creating security markings in a tabular form. It also offers enhanced formatting capabilities beyond what was previously possible for notes or symbols.

Additional Information

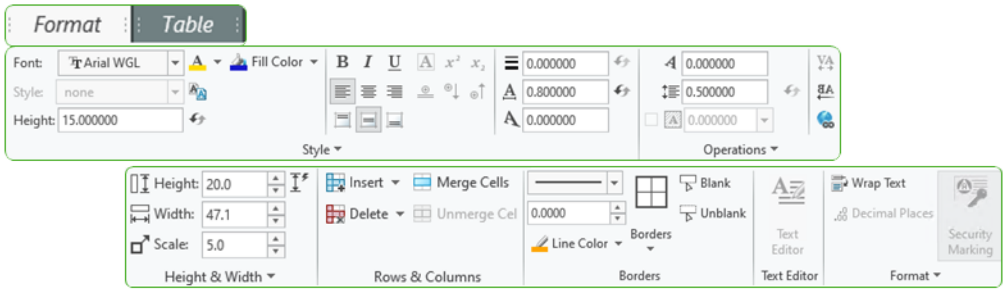
Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

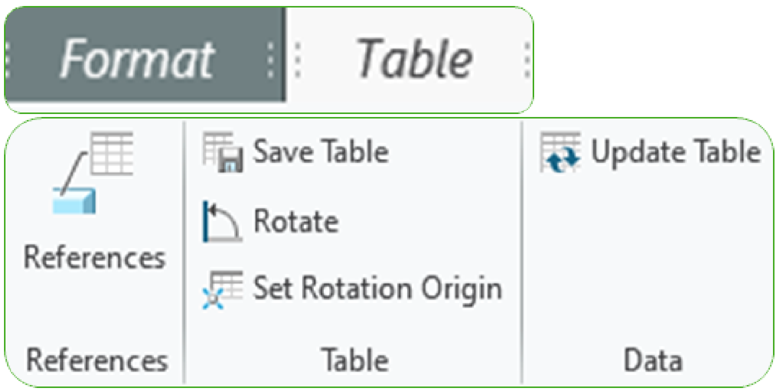
Contextual Formatting Options for Tables

Creo Parametric 11.0.0.0

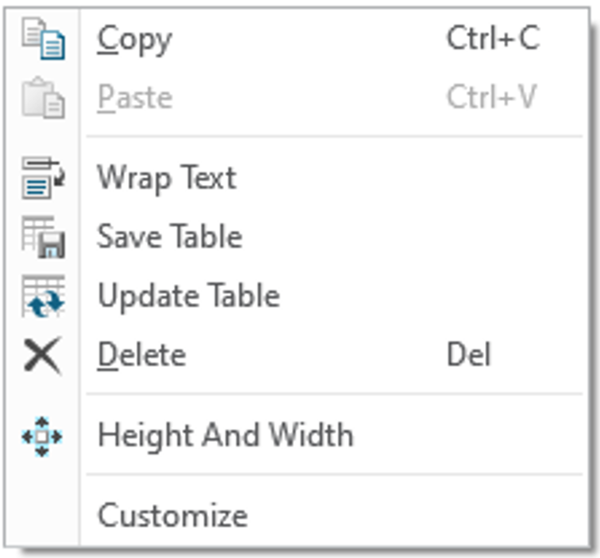
User Interface Location:

- Select the table, row, column, or cell. The **Format** and **Table** ribbon tab opens.





- Select the table, row, column, or cell. The contextual menu appears.



Videos

See the video on the Learning Connector.

Description

You can format tables and table objects quickly and easily.

When you select any table object, the contextual format, table ribbons, and the mini toolbar are shown with various formatting and table related commands. Use these commands to customize and make changes to the table or its contents as per your requirements.

Some of the formatting options available for tables in MBD include:

- Text formatting
- Cell formatting
- Fill color
- Add hyperlink
- Adjust the height and width of rows or columns.
- Add or delete rows or columns

- Scale table
- Merge or unmerge cells.
- Blank or unblank cell borders.
- Change font and thickness of table lines.
- Semantic query and definition

Benefits

Ability to quickly and easily update tables while maintaining full and intuitive control over text, cell, and table formatting.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

User Interface Elements for Table Interaction

Creo Parametric 11.0.0.0

User Interface Location: Select the table in graphics window. The table draggers are shown at the corners of the table.

Videos

See the video on the Learning Connector.

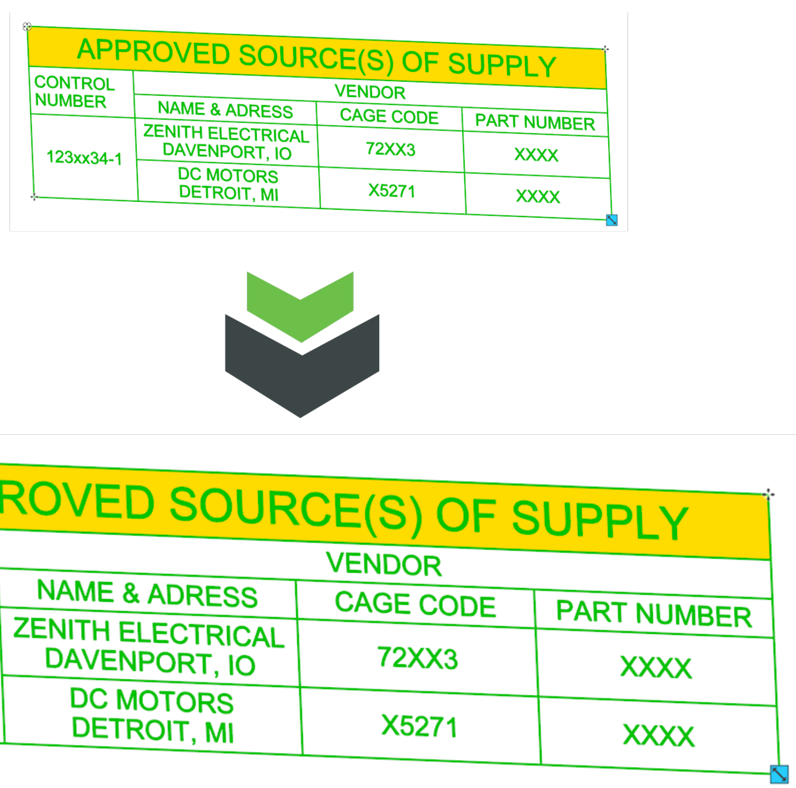
Description

Using the draggers on the table, adjust the table proportions and the table position according to your requirements.

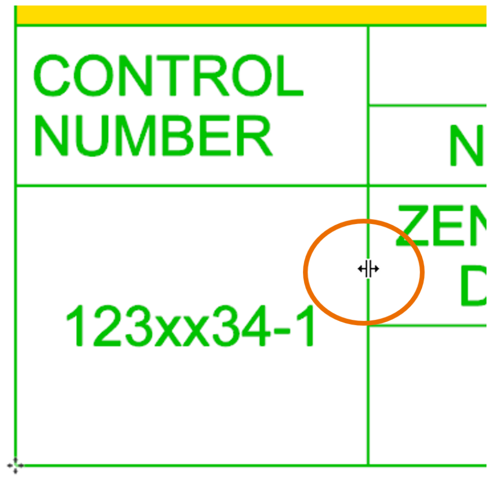
Using the draggers, perform the following tasks:

- Table selection—Hover the pointer over table to pre-highlight it. The top left corner shows a selection handle for quick table selection.

- Table movement—Select the table or a cell. Use one of the three corner draggers to move the table. You can move the table in single direction (horizontally or vertically) by holding the SHIFT key while dragging the table. Without the SHIFT key, the table will move freely in any direction.
- Table scaling—For freestyle scaling, use the scaling dragger to scale the table. For proportional scaling that maintains consistent table width, height, and text size, press and hold the Shift key while dragging the table.



- Width or Height adjustment for table rows, columns, or cells—Drag the on-screen table borders or use the commands available on the ribbon tab. For a quick width adjustment, double-click width dragger to fit column width to text.



Benefits

This enhancement provides an easy and quick way for on-screen table interaction.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Text Editing Modes for Tables

Creo Parametric 11.0.0.0

User Interface Location: Select the cell in the table, and then on the **Format** tab, click  **Text Editor**.

Videos

See the video on the Learning Connector.

Description

Quick and intuitive options to edit the contents of the tables:

- On-screen editing mode
  - Double-click the table cell to activate the edit mode and start entering text.
  - Single-click the next cell to edit its content.
  - Navigate between the previous and next cells using the Tab and Shift + Tab keys.
- Text editor mode
  - Use the **Text Editor** in the **Format** tab to add text in the selected cell.
  - Move to the next cell to edit its contents while in text editor mode.

Benefits

Intuitive text editing methods and easy navigation across table cells in edit mode.

Additional Information

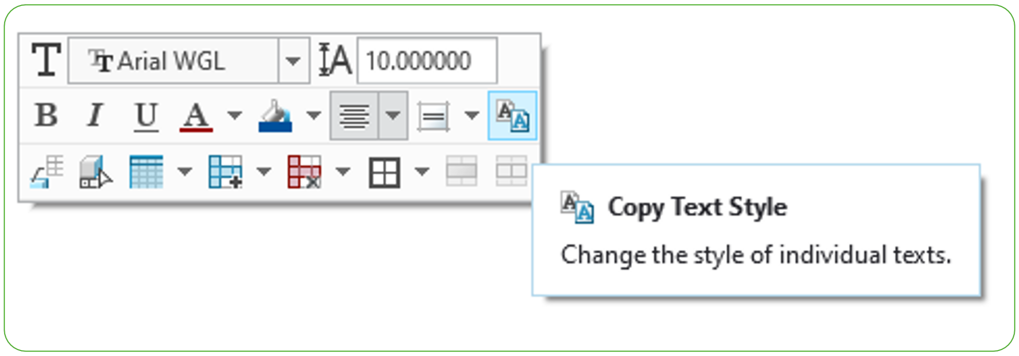
Tips:	Use the F2 key to edit the content of a selected table cell.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Leverage Reference Formatting of Text Styles for Tables in MBD

Creo Parametric 11.0.0.0

User Interface Location:

- In the **Format** tab, click **Copy Text Style**.
- Select a cell, in the shortcut menu click **Copy Text Style**.



Videos

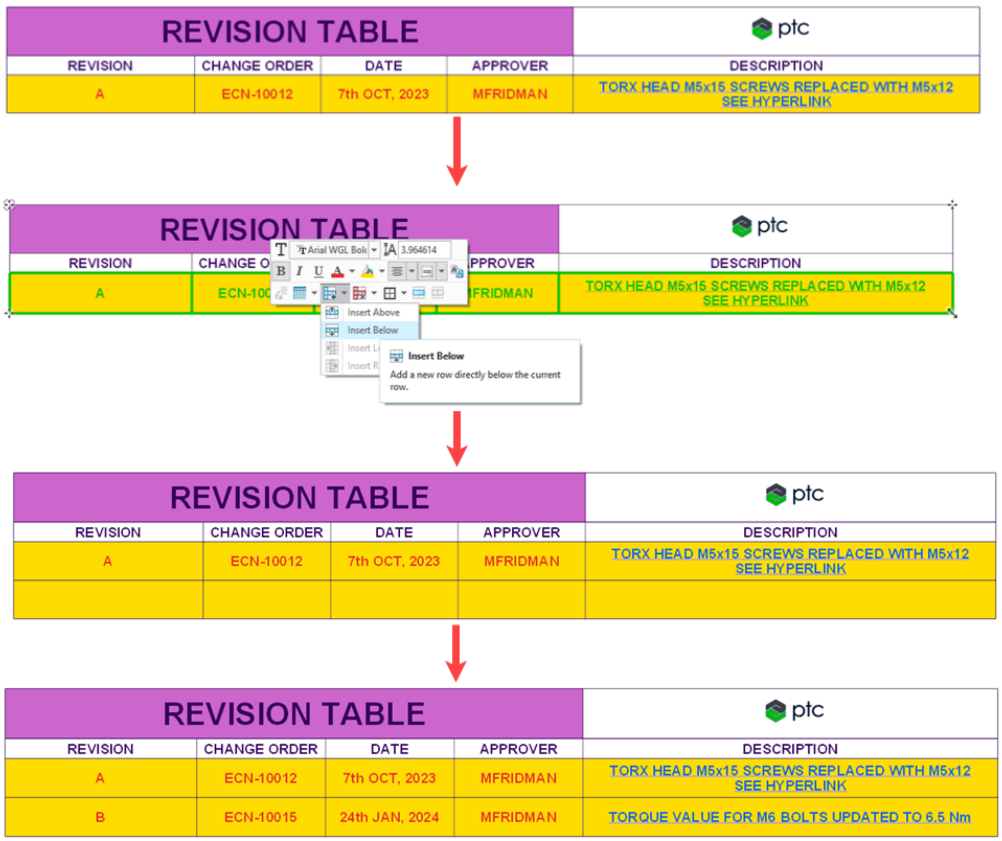
See the video on the Learning Connector.

Description

The new enhancement for reference formatting simplifies table formatting in MBD. With this enhancement, you can copy the current text style from reference cells to the target cells ensuring consistency in the text style within the table.

Copying the text style:

- When inserting new rows or columns in a table, the text formatting of the referenced row or column is copied to the new table object.
- Select multiple cells, rows, and columns and copy the text style from another reference cell using **Copy Text Style**.



Benefits

- Quick way to copy formatting from a reference cell.
- Less time spent on reapplying formatting to newly added rows or columns.

Additional Information

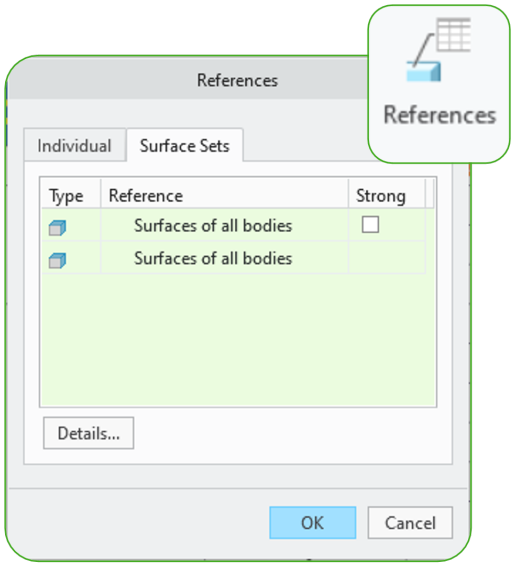
Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Semantic Query Definition for Tables

Creo Parametric 11.0.0.0

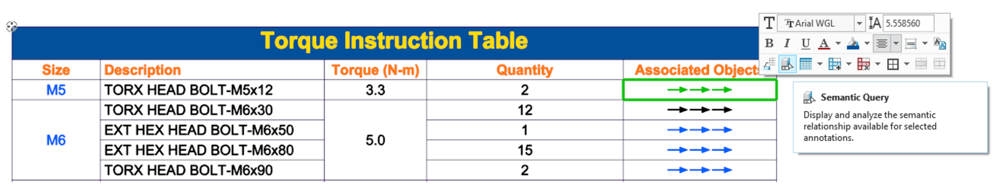
User Interface Location:

1. Select the table or a table cell.
2. On the **Table** tab, click **References**.



or

1. Select the table or a table cell.
2. Click **Semantic Query** in the shortcut menu.

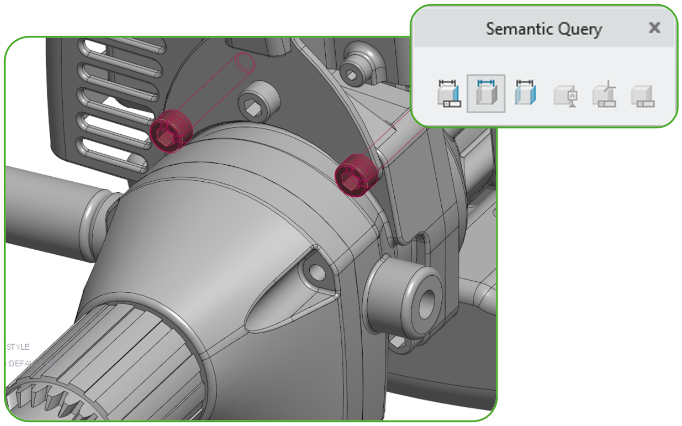


Videos

See the video on the Learning Connector.

Description

You can add semantic references to a table or to individual table cells. Using the semantic query tool, you can query the semantic associativity between tables and model surfaces. The corresponding model surfaces will be cross-highlighted in the graphics window.



Benefits

The semantic behavior and cross-highlighting of associated surfaces provide a better experience for users and enhance machine readability in MBD.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

GD&T Advisor Enhancement: Combined Simplified Hole Callouts for ISO Models

Creo Parametric 11.0.0.0

User Interface Location: N/A

Videos

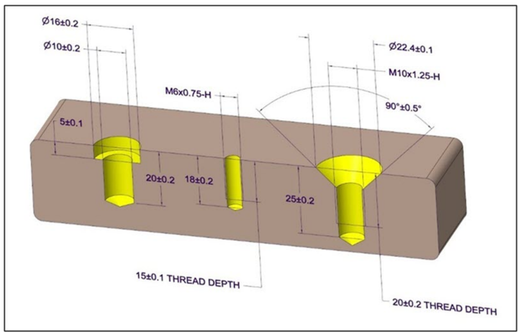
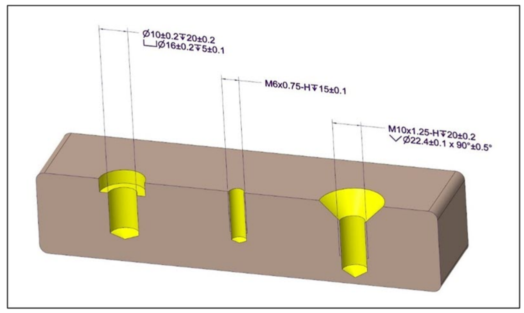
See the video on the Learning Connector.

Description

The ISO GPS standard supports both full and simplified dimensioning schema for hole callouts in multi-element holes (for example, counter bore/sink or threads). GD&T Advisor now supports the simplified schema for ISO models (already supported for ASME models).

The combined simplified hole callout is enabled using a new XML tag `Combined_ISO_hole_notes` set to `YES` in the application options file `GDTAAppOptions.xml`.

Using this option, ISO users will be able to specify callouts for their hole features while making sure that the multiple specifications are shown as combined.

<table>
<tr><th>Combined_ISO_hole_notes = No</th><th>Combined_ISO_hole_notes = Yes</th></tr>
<tr><td>
</td><td>
</td></tr>
</table>

Benefits

Improved readability of hole callouts, reduced clutter of annotations in the Model-Based Definition model, and improved compliance with the ISO standards.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	Legacy models with existing hole callouts continue to show the full dimensioning schema if the new XML option `Combined_ISO_hole_notes` is set to `NO`. When it is set to `YES`, GD&T Advisor updates the hole callout schemas during its verification.

GD&T Advisor Enhancement: Slab and Slot Features for Disjoined Coplanar Surfaces with Opposing Planes

Creo Parametric 11.0.0.0

User Interface Location:

1. On the **GD&T Advisor** ribbon, click **Tolerance Feature**.

2. Select any disjoined coplanar surfaces along with an opposing plane, which corresponds to a definition of a slab or a slot feature.

Videos

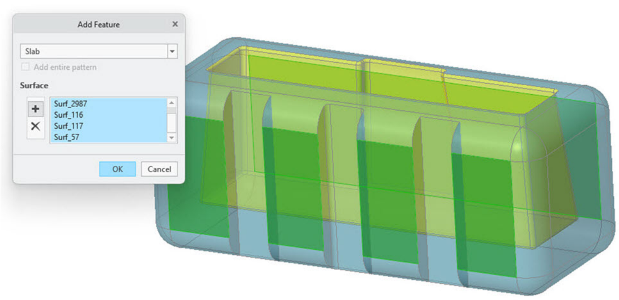
See the video on the Learning Connector.

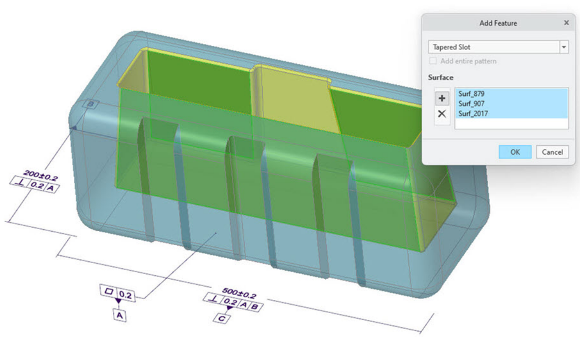
Description

GD&T Advisor now supports disjoined coplanar surfaces with opposing planes defined as slab or slot features.

This enhancement supports the following features:

- Slab
- Slot
- Tapered slab
- Tapered slot





Benefits

This enhancement improves productivity when working with the slab or slot feature in the GD&T Advisor and expands its support for additional use cases.

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

GD&T Advisor Enhancement: Support of ISO 22081 for General Tolerances

Creo Parametric 11.0.0.0

User Interface Location: Click **Edit Properties** in GD&T Advisor.

Videos

See the video on the Learning Connector.

Description

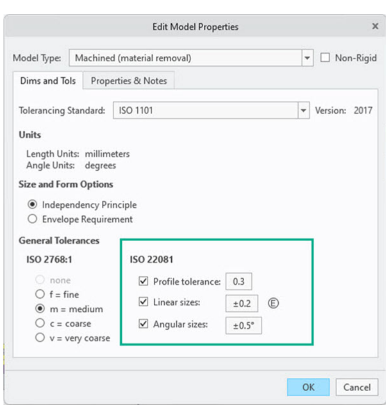
GD&T Advisor was updated to support ISO GPS 22081:2021 to indicate the general size and geometrical tolerance specifications.

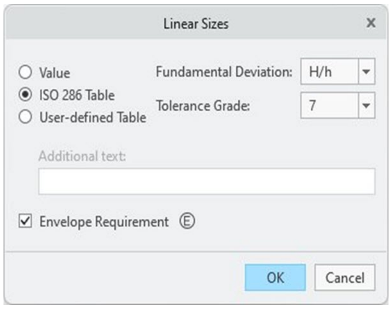
This enhancement includes the following general tolerances:

- General profile tolerance
- General linear sizes tolerance
- General angular sizes tolerance

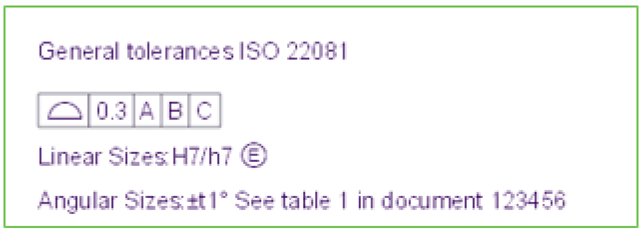
Now, once the tolerance is applied, a corresponding note is displayed in the model to indicate the used tolerances, and a collection of relevant semantic references for the general profile tolerance is associated with this geometrical specification.

You can specify the details of the general tolerances according to the latest ISO standards used by the model.





A generated system note is displayed as follows:



Benefits

With this enhancement, the ISO-compliant specification of general tolerances is now supported in the GD&T Advisor.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

GD&T Advisor Enhancement: New Contextual Commands for Improved Productivity

Creo Parametric 11.0.0.0

User Interface Location: Right-click any feature in the feature tree to delete the selected geometrical tolerance and to reset the user-specified properties added to an annotation.

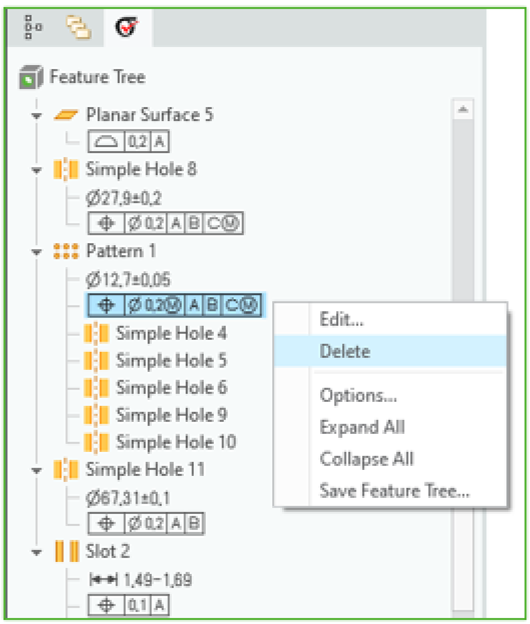
Videos

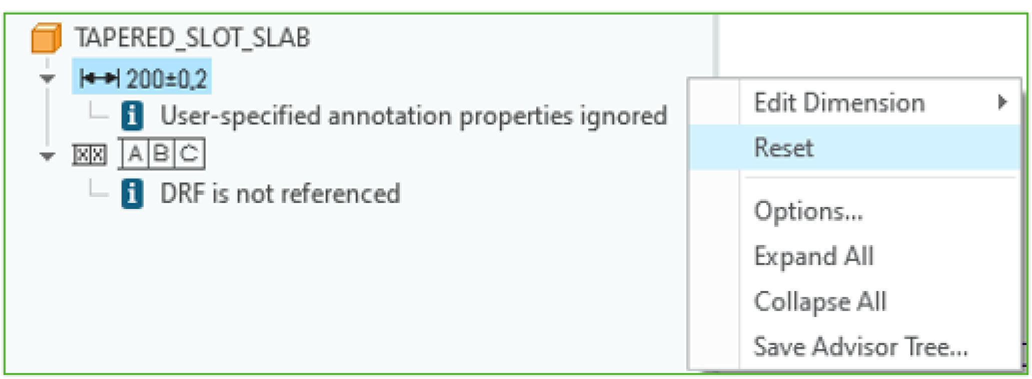
See the video on the Learning Connector.

Description

Two new commands were added to the contextual shortcut menu of GD&T Advisor when a feature is selected in the advisor tree:

- **Delete**—Delete the selected geometrical tolerance from the owner functional feature without having to first redefine the feature and delete it from the dashboard.
- **Reset**—Reset the user-specified properties added to an annotation and reset them to the default specification.





Benefits

With this feature, you can reset user-defined properties or delete geometrical tolerances in less time and with fewer mouse clicks.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

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Part Modeling

Extend: New Extrapolate Option

Creo Parametric 11.0.0.0

User Interface Location: Click **Model** ▸ **Editing** ▸ **Extend** ▸ **Options** tab.

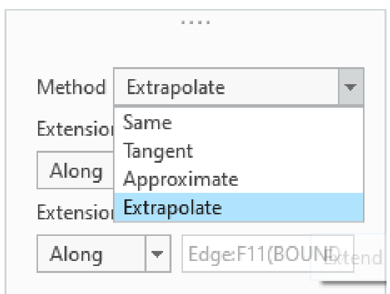
Description

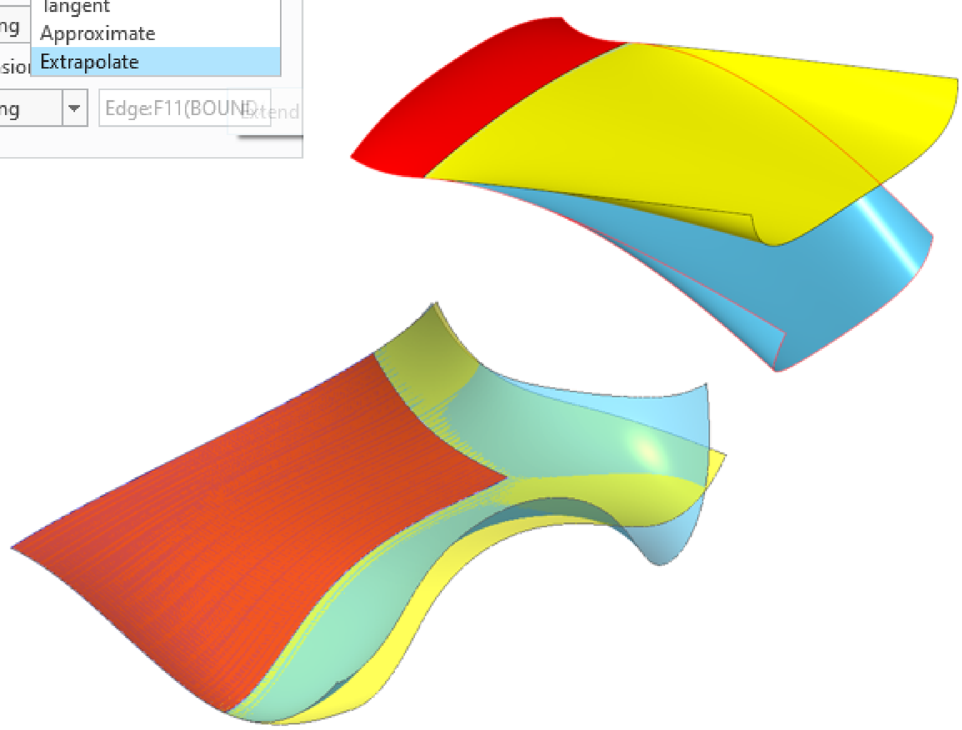
In Creo Parametric 11.0, the **Extend** feature offers a new **Method** option called **Extrapolate**.

With this method, each Surface will be replaced by a new extrapolated surface that coincides with the original surface within its original domain. This produces a less wavy surface extension. The extrapolated surface will get a new Surface ID.

The Extrapolate method can help achieve better results when the **Same** surface extension method produces undesirable results, such as inflections or folding, or fails.

This broader range of geometric extension methods provides increased flexibility and productivity to create the desired geometry.





When using the **Extrapolate** option, all surface types except planes will be extrapolated and become a B-Spline or Spline surface. If the original surface is not of type Spline or B-spline, the extrapolated surface will be the close approximation of the original surface within its original domain. Planes will remain planes. Use the extension method **Same** (surface) to extend other analytic surfaces so they remain analytic.

Benefits

Increased flexibility to create desired geometry

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Improved Feature Dimension Handles

Creo Parametric 11.0.0.0

User Interface Location: Click **Edit Definition** or **Edit Dimensions**.

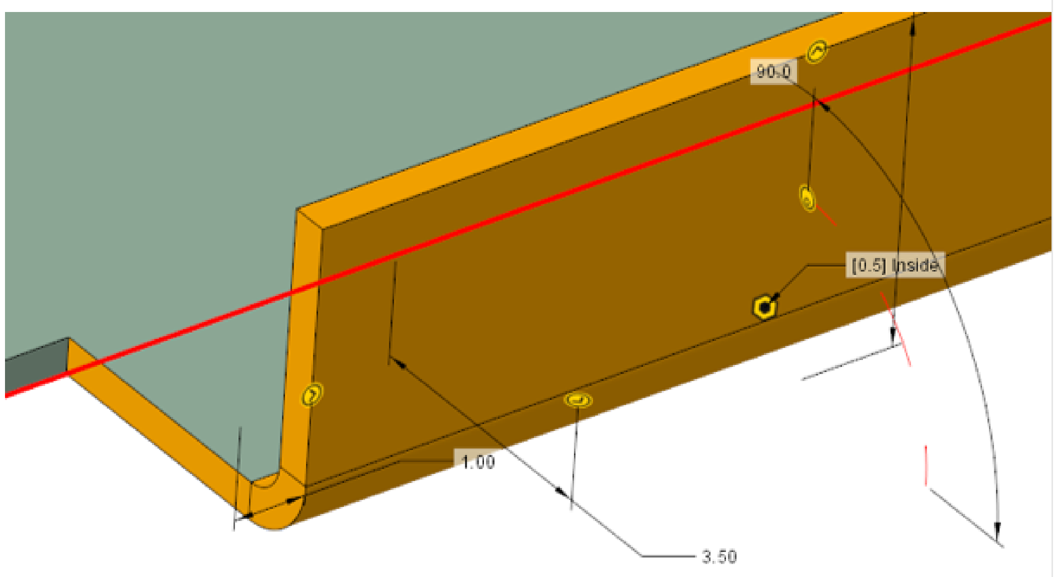
Videos

See the video on the Learning Connector.

Description

The drag handles for feature dimensions have been modernized in regular Part mode and Sheet Metal mode. The improved handles make it easier to differentiate between different types of dimensions and interaction possibilities in features. This in turn makes it easier to identify controls, especially in complex features. The handles are self-orienting in 3D space, adjusting the display as you work with your model.

Previously, all handles were flat to the screen, which made it harder to identify and interact with the right handle.



Benefits

Easier identification of controls for complex features

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Assign Commands to Quick Access Toolbar from within Command Search

Creo Parametric 11.0.0.0

User Interface Location: Click Title bar > Command Search.

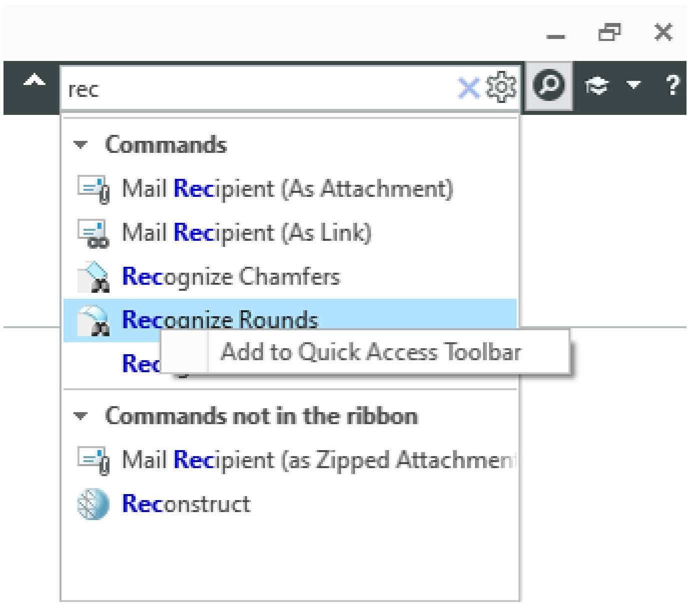
Videos

See the video on the Learning Connector.

Description

You can now quickly search for and find commands and easily add them to the Quick Access Toolbar (QAT).

When you use command search, use **Add to Quick Access Toolbar** from within the context menu of a search result item. This adds the command to the Quick Access Toolbar without the need to go through other user interface customization dialogs and steps.



Previously, after searching for and finding a command, it was necessary to leave the search dialog to invoke Quick Access toolbar customization workflows.

Benefits

Faster customization workflow

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

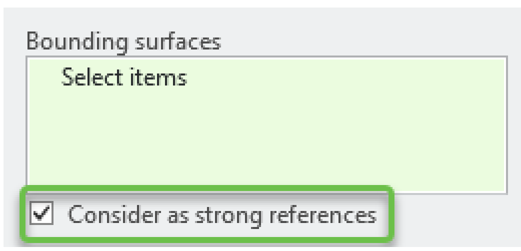
Enhancement: Control Reference Type in Seed and Boundary Surface Selection

Creo Parametric 11.0.0.0

User Interface Location: Surface collection.

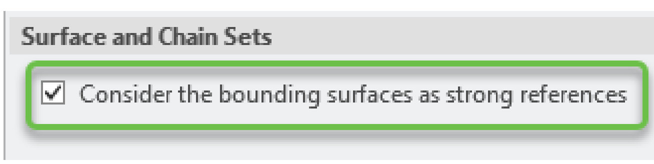
Description

In Creo Parametric 11 you can control the reference type for boundary surface references in Seed and Boundary surface selection.



Select the new option **Consider as strong references** to mark boundary references in the surface collection definition as strong references. In case they are no longer present, Creo will report them as missing strong references, and the surface collection fails.

If the option is not selected, then boundary references are considered weak references. This corresponds to the behavior prior to Creo Parametric 11.

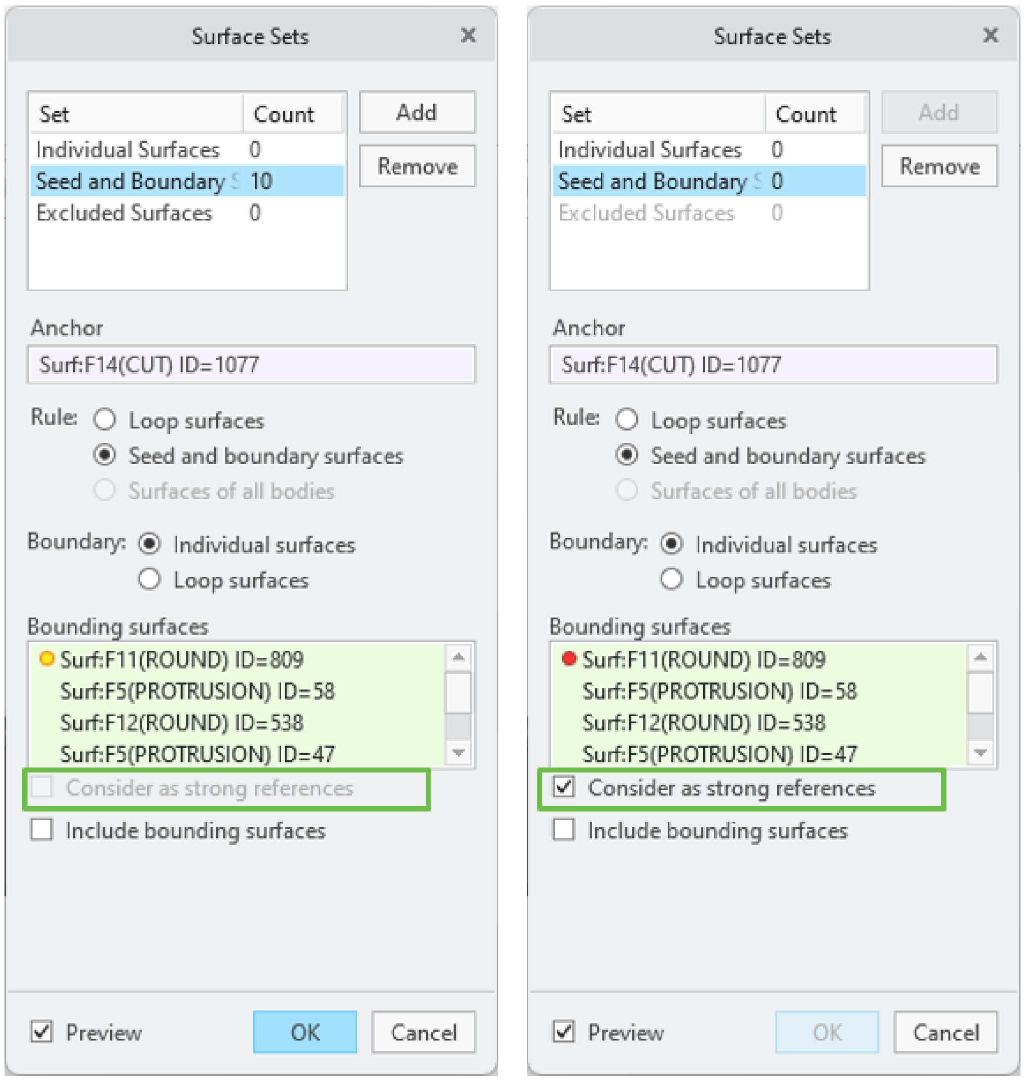


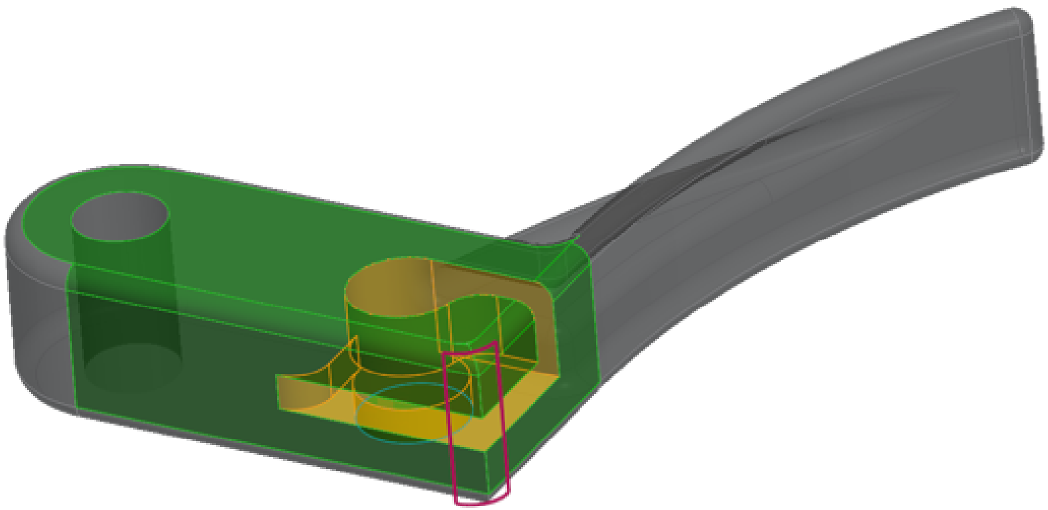
You can define the default behavior from within the **Creo Parametric Options** dialog box **Creo Parametric Options ▸ Global ▸ Selection ▸ Surface and Chain Sets**. By default, the boundary surfaces are now considered strong references starting with Creo Parametric 11.

Strong and weak references behavior for Boundary surfaces each have their pros and cons. Previously you could not tune the behavior according to your preference.

- New option: **Consider as strong references**
  - Weak (existing behavior)
  - Strong (new)
- Controls regeneration behavior
  - When set to strong and with boundary references missing, the surface collection will fail even if it could return a surface collection result

- Configuration option
  - `default_boundary_refs_strong` yes*, no
  - Also accessible from **Creo Parametric Options ▸ Global ▸ Selection ▸ Surface and Chain Sets**





Benefits

- Improved flexibility to control regeneration behavior according to preferences

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.

Configuration option associated with this functionality:	`default_boundary_refs_strong`
	`yes*`, `no`
	In Seed and Boundary sets, considers the bounding surfaces as strong references by default.

Feature's Diagnostics Reporting

Creo Parametric 11.0.0.0

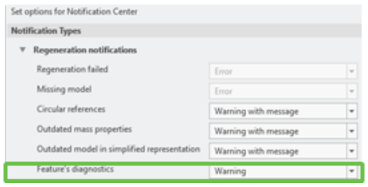
User Interface Location: Status bar, Notification Center flag.

Videos

See the video on the Learning Connector.

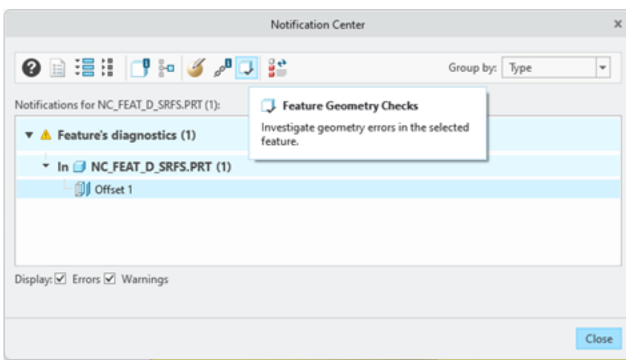
Description

Creo 11 enhances reporting of feature diagnostics within the Notification Center. Creo Parametric can be configured to report all features that reported diagnostic information, Geometry Checks, during regeneration. The setting is controlled from the Notification Center options under Feature’s diagnostics.

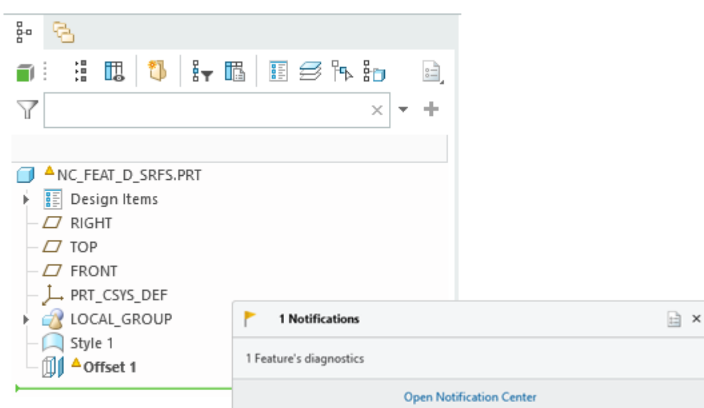


This notification allows you to immediately see whether the regeneration produced diagnostic information without the need to switch to the tools tab to investigate the **Geometry Checks** command.

Features that report diagnostics are flagged in the Model Tree and listed in the Notification Center under the category Feature’s diagnostics. You can click to access the **Troubleshooter** dialog box containing diagnostic details and troubleshooting recommendations for it.



In the Model Tree, Select the feature and click **Information** ▸ **Feature Geometry Check** to open the **Troubleshooter** dialog box.



Benefits

Personalize and optimize Notification Center reporting so that you do not miss geometric diagnostics information.

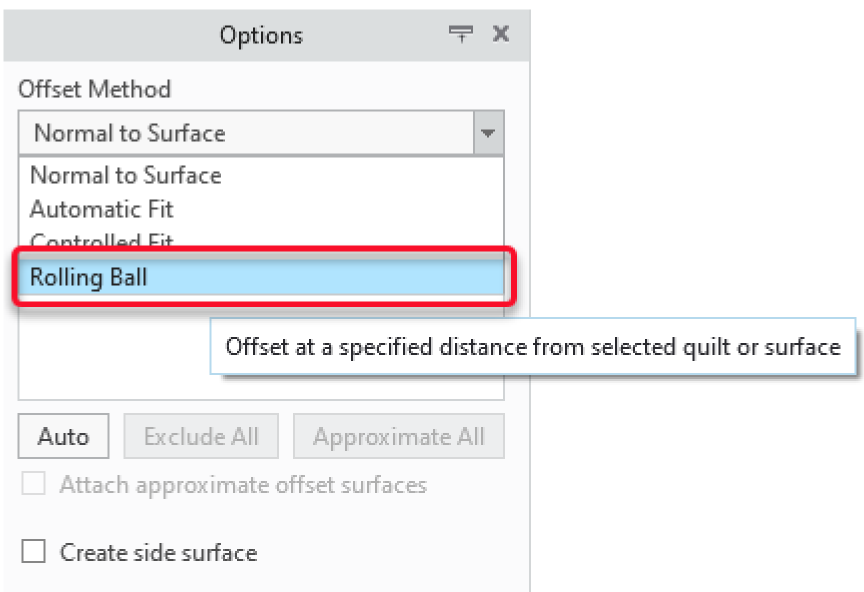
Additional Information

Tips:	None.
Limitations:	If a feature fails, it is reported as part of the notification Regeneration Failed.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	`nmgr_geom_checks`—Show notifications for features that have geometry checks.

Offset: Rolling Ball Enhanced

Creo Parametric 11.0.0.0

User Interface Location: Click **Model** ▸ **Editing** ▸ **Offset** ▸ **Options** ▸ **Offset Method** ▸ **Rolling Ball**.



Videos

See the video on the Learning Connector.

Description

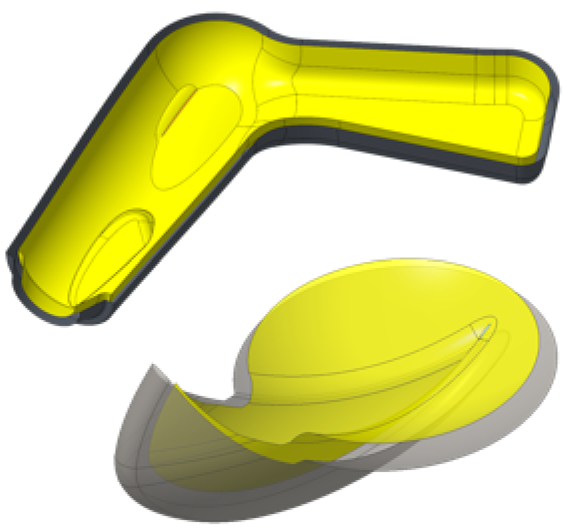
Creo Parametric 11 further enhances the **Rolling Ball** option in **Offset** that was initially released in Creo Parametric 10.0.1.0.

This option provides a noticeable performance increase, both in general, and in feature regeneration workflows without the user interface, or when pressing **OK** after an attached/unattached preview was calculated.

The Rolling Ball method increases the success rate of offset geometry creation for situations where the **Normal to Surface** offset method fails.

Benefits

- Improved productivity - Perform offset operations on complex models faster
- Faster regeneration of **Rolling Ball** Offset
- Increase success rate
- Improved diagnostics information



Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

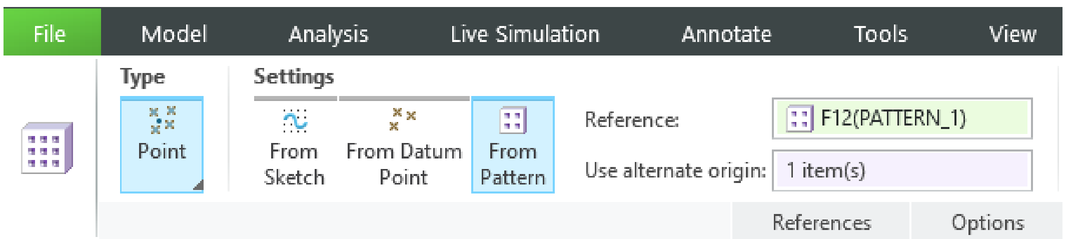
Pattern: Enhanced Point Pattern Flexibility and Performance

Creo Parametric 11.0.0.0

User Interface Location: Click any of the following:

- **Model** ▸ **Pattern**
- **Model** ▸ **Pattern** ▸ **Geometry Pattern**
- **Flexible Modeling** ▸ **Flexible Pattern**

Under **Type**, select **Point**. Under **Settings**, select **From Pattern**.



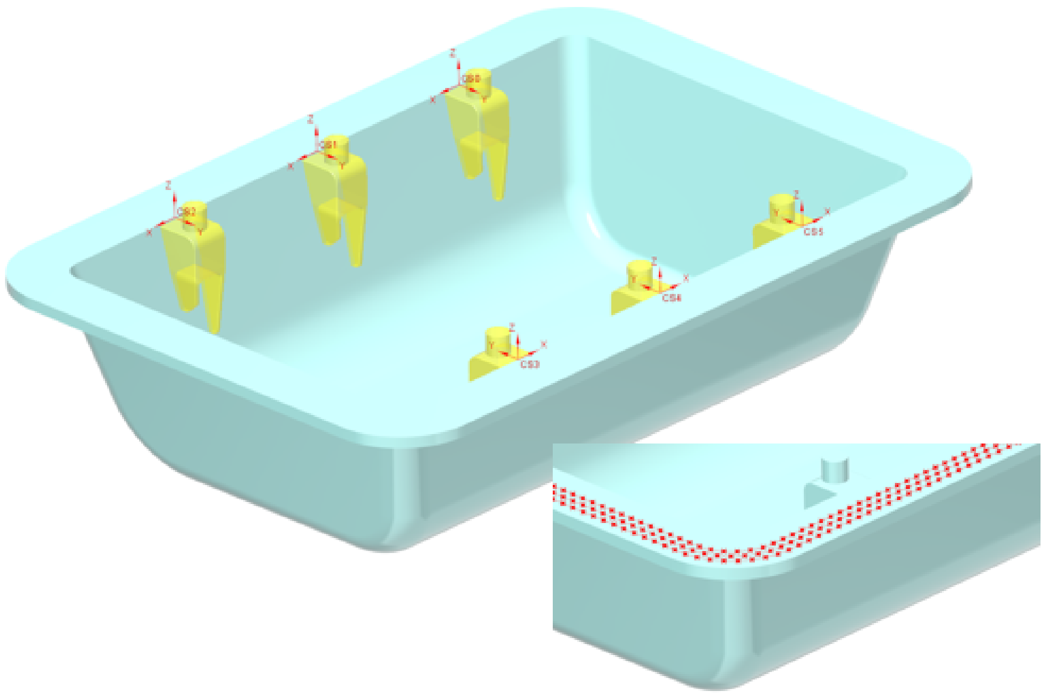
Videos

See the video on the Learning Connector.

Description

A new **From Pattern** reference option is introduced for patterns of type **Point**. This allows defining a point pattern based on:

- Point patterns
- Coordinate system patterns
- Pattern that contains points or coordinate systems



When the patterned geometry has no dependency on the leader point/coordinate system, it can be beneficial to specify the leader as the alternate origin.

An example use case for coordinate system references is a body pattern where you can control the location and orientation of the bodies through the coordinate systems that are referenced through the new **From Pattern** option.

In addition to increased design flexibility, this can also help to speed up pattern regeneration, as the Point Pattern type within Geometry Pattern can offer the full choice of regeneration options (Identical, Variable, General) depending on the geometric situation. This allows to optimize pattern performance where applicable.

Benefits

- Increased design flexibility for patterns that reference patterns of points or coordinate systems
- Easier to create body patterns by following a pattern of coordinate systems

Additional Information

Tips:	None.
Limitations:	A pattern under an External Inheritance feature cannot be selected as a reference during the creation of a point pattern with reference to a pattern of points or coordinate systems.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Enhanced Remove Body Feature

Creo Parametric 11.0.0.0

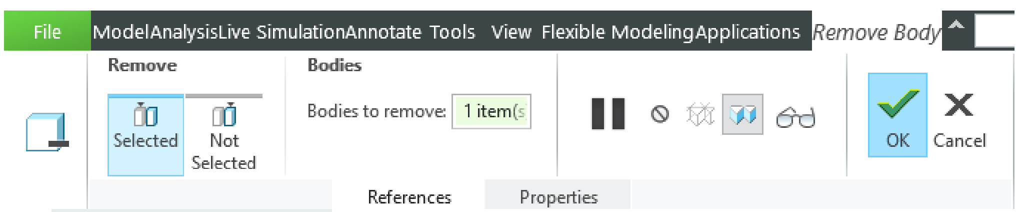
User Interface Location: Click **Model** ▸ **Body** ▸ **Remove Body**.

Videos

See the video on the Learning Connector.

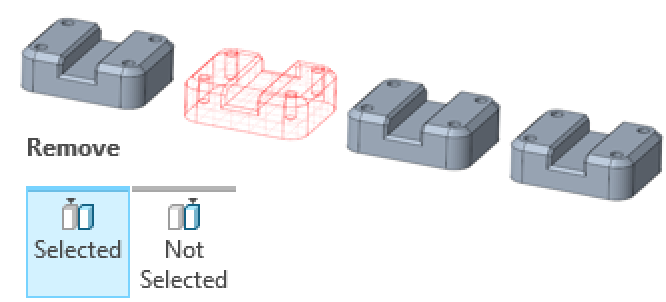
Description

- In Creo Parametric 11, the **Remove Body** feature provides an additional option that allows you to toggle between two definition schemes:



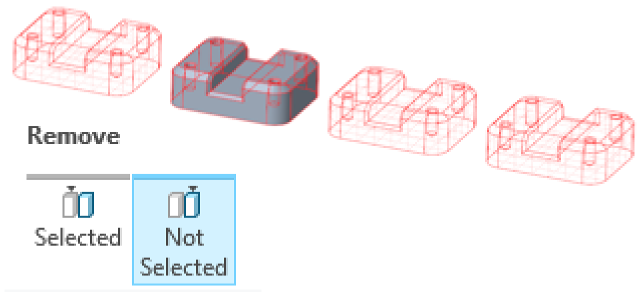
  - **Selected**

    Identifies bodies to remove



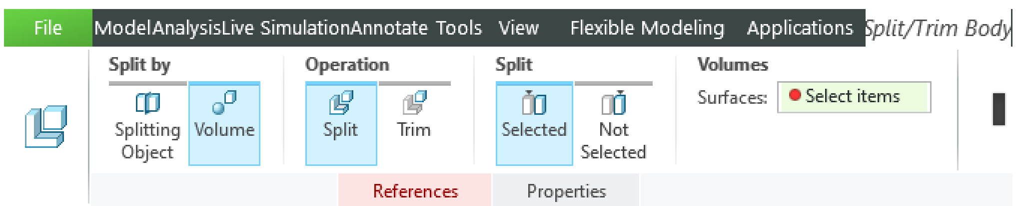
  - **Not Selected** (new)

    Identifies bodies to keep, while all other bodies are removed



Depending on the situation, the new definition scheme can help to define the desired outcome with fewer selections. It can help to define the feature in a more robust way to better handle situations in which the number of bodies can change.

- The **Split/Trim Body** feature is updated in alignment with Remove Body. The new **Selected** and **Not Selected** options replace the previous flip option.



Benefits

Improved productivity due to a more flexible feature definition to achieve more desired, parametric behavior.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Control Selection Priority for Quilts

Creo Parametric 11.0.0.0

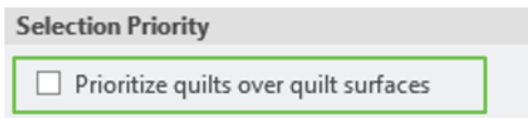
User Interface Location: Click **File** ▸ **Options** ▸ **Global** ▸ **Selection** ▸ **Selection Priority**.

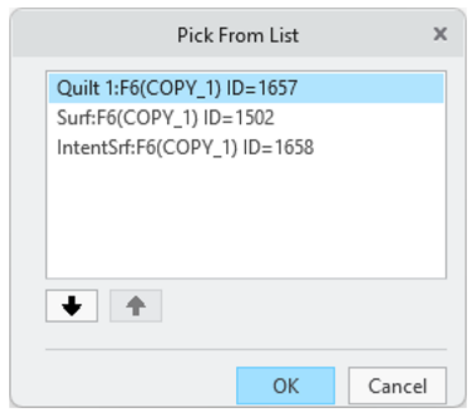
Description

A new option has been added that gives you more control over the priority of selection for surfaces and quilts. You can now chose between the previous selection priority order, or give a higher selection priority to quilts over surfaces.

If you typically need to select quilts as the more stable reference in regeneration, the new option can reduce the number of clicks required during selection, and reduce the likelihood to select the surface instead of the quilt.

Select the option **Prioritize quilts over quilt surfaces** to activate the higher selection priority for quilts. Selection workflows such as **Pick From List** will then list the Quilt object before the listing the nearest corresponding surface under the pick point.





Benefits

- Option to pick an easier workflow to ensure the more stable quilt reference can be selected.
- Fewer clicks required to select quilts.
- Reduced likelihood to select the surface instead of the quilt when the quilt consists of a single surface.

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	selection_prioritize_quilts yes, no* yes—Give quilt objects a higher selection priority than quilt surfaces. no—Quilt surfaces will have a higher selection priority than quilts.

Enhancement: Streamlined Placement of Legacy UDFs (User-Defined Features)

Creo Parametric 11.0.0.0

User Interface Location: Click **Model** ▸ **User-Defined Feature**.

Description

In Creo Parametric 11.0 the placement of user-defined features (UDFs) is streamlined for UDFs that reference just a single body. The enhanced UDF placement workflow for these UDFs now automatically populates the body reference collector with the default body, if it is valid. This is always the case for UDFs created in Creo Parametric 7.0 or earlier (before the multibody capabilities had been introduced), but it also applies to UDFs created in newer versions of Creo.

This solves the problem that with the introduction of multibody capabilities, an additional selection step to select the body reference was required.

The body reference is not automatically populated in the following cases:

- The body reference is the single reference exposed by the UDF.
- The current default body is not a valid reference for the UDF body reference to be populated.

Benefits

Increased productivity when placing UDFs.

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

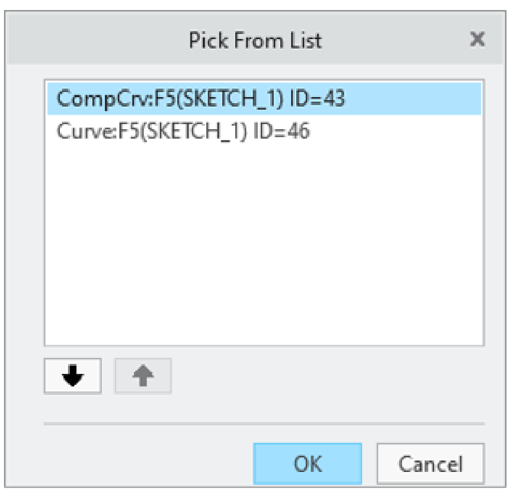
Improved System Feedback for Composite Curve Selection

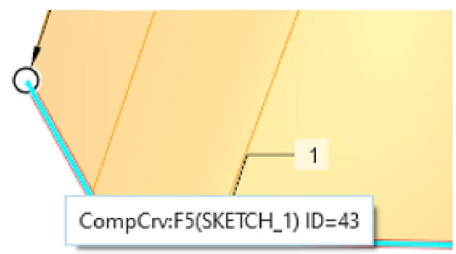
Creo Parametric 11.0.0.0

User Interface Location: graphics window and collectors.

Description

In Creo Parametric 11.0, there is a clearer indication whether a curve selection contains the composite curve object, or only a curve segment of the composite curve. Previously, both were indicated with a Curve label. Understanding the difference is important, as the composite curve object is typically the preferred reference in selection to ensure regeneration stability upon geometric changes.





In Creo Parametric 11.0, when a composite curve object is selected, the label is now changed to CompCrv, replacing the previously used Curve label. The label can be seen during selection workflows in the tooltip shown in the graphics area, in the **Pick From List** dialog, and in collectors.

Previously there was no obvious indication whether a curve selection contained the composite curve object, or only a curve segment of the composite curve.

No change was made for situations that do not show a Curve label at all.

Benefits

Easier workflow to ensure the more stable curve reference can be selected.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Enhancement: Fast Bounding Box Calculation

Creo Parametric 11.0.0.0

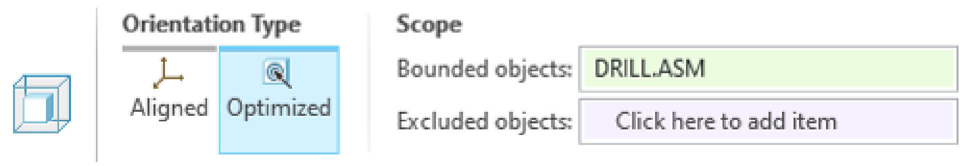
User Interface Location: Click **Model** ▸ **Surfaces** ▸ **Enclosure Volume**.

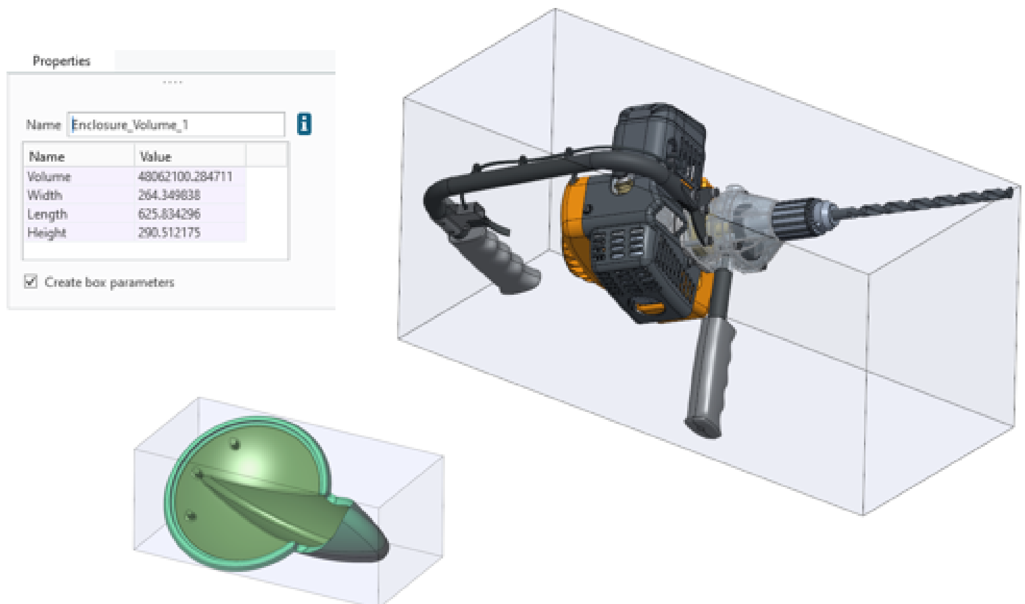
Videos

See the video on the Learning Connector.

Description

The Enclosure Volume feature has been enhanced to optionally calculate an orientation-optimized, minimal bounding box.





The **Optimized** option calculates the minimal bounding box that can enclose the geometry, regardless of the orientation of the model. You can use the **Create box parameters** option to add feature parameters for the bounding box dimensions and volume, and create a coordinate system showing the optimized orientation.

Objects can be excluded from the bounding box by selecting them. You can also choose to exclude all quilts, and exclude all construction bodies.

The Enclosure Volume feature is automatically regenerated when geometry changes in Part mode. As part of the regeneration, you can decide what you want to recalculate or not, with these options:

- **Update optimization and dimensions**—Recalculates the optimized orientation and box volume.
- **Freeze optimization**—Only updates the box volume.
- **Freeze optimization and dimensions**—No update (legacy feature behavior).

You can offset the box from the geometry by a uniform distance on all sides, or you can specify a unique offset for each of the six sides of the box.

The Enclosure Volume and Internal Volume features are available in core modeling and no longer require an additional license.

Legacy Enclosure Volume features are updated upon **Edit Definition**.

Benefits

- Parametric update of Enclosure Volume in Part mode
- Easy and intuitive creation of a bounding box for various use cases such as packaging optimization

Additional Information

<table>
<tr><td>Tips:</td><td>None.</td></tr>
<tr><td>Limitations:</td><td>
In Assembly

In assemblies, the regeneration options for automatic recalculation of the Enclosure Volume orientation and dimensions are not supported. However, automatic recalculation and update is done upon Edit Definition.
For components added to the Excluded objects collector, all occurrences of the component will be excluded automatically.
Upper-level components/objects should not be referenced by a lower-level Enclosure Volume feature (external references). Selecting references outside this scope could lead to unpredictable results, such as the wrong orientation or dimensions of the box.

There are situations that don’t trigger regeneration where a manual update via the Edit Definition workflow is required to update the Enclosure Volume feature.

Example: Conversion of the Construction status of a body

ThingMark and Spatial Target features for Augmented Reality are considered to be part of the model geometry. They cannot be selected individually as excluded objects. Spatial Target features can only be excluded by selecting the Exclude quilts check box.
</td></tr>
</table>

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Project Sketched Points

Creo Parametric 11.0.0.0

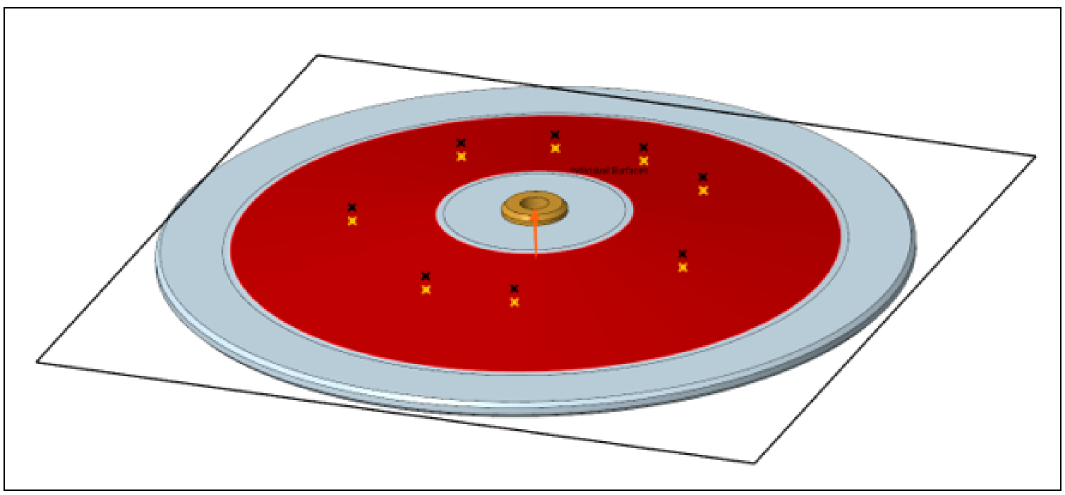
User Interface Location: Click **Model** ▸ **Project**.

Videos

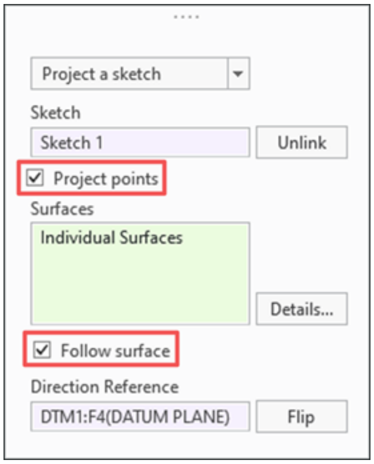
See the video on the Learning Connector.

Description

In Creo 11.0, the **Project** feature is enhanced to project sketched points, allowing you to project multiple points in a single operation. Previously, only a single point could be projected onto a geometric target object in a single operation.



When the **Sketch** option is selected, the new **Project points** checkbox is available. It allows you to project points contained in the sketch to the selected target, in addition to sketched curves that are present in the sketch. Like construction curves, construction points are ignored and not projected. The resulting feature points can be used as location references in a spot-weld feature, or in a point-based pattern where the **From Feature** option supports referencing the **Project** feature containing projected points.



In the sheet metal environment, the **Follow surface** option is enhanced and applies to all projected sketch points.

Benefits

Faster and more flexible definition of multiple projected point references as input to repetitive features.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Control Locks Display in Sketcher

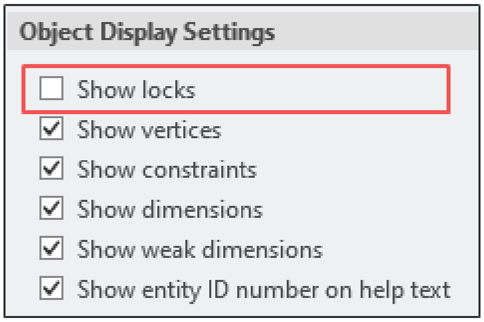
Creo Parametric 11.0.0.0

User Interface Location: Click **File** ▸ **Options** ▸ **Sketcher**.

Description

In Creo 11.0 there is a new configuration option to control the display of locked objects. It allows you to control the default state of the **Locks Display** option in the graphics toolbar.

You can set the default behavior inside the Creo Parametric **Options** dialog under **Sketcher ▸ Object Display Settings** by checking **Show locks**.



Benefits

Increased flexibility to control default system behavior.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.

Configuration option associated with this functionality:	`sketcher_disp_locks`
	`yes, no*`
	Controls whether or not locks are visible by default in Sketcher.

Offset Supports Edge Chain References in Sketcher

Creo Parametric 11.0.0.0

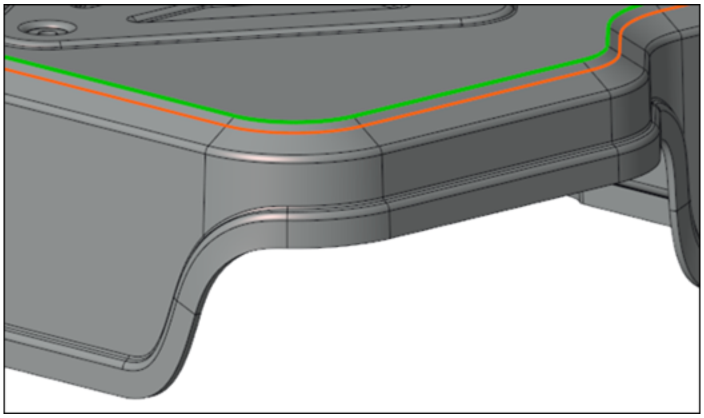
User Interface Location: Click **Model ▸ Offset**.

Videos

See the video on the Learning Connector.

Description

In Creo 11.0 there is additional flexibility in defining the**Offset** feature. In addition to selecting curves or intent datum curves, the **Offset** feature now supports directly selecting an edge, edge chain, or intent chain. This eliminates the need to first create a curve from an edge chain before creating an offset.



Benefits

Improved productivity by eliminating an additional step when offsetting edges, edge chains, or intent chains.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Trim Self-Intersecting Composite Curves in Sketcher

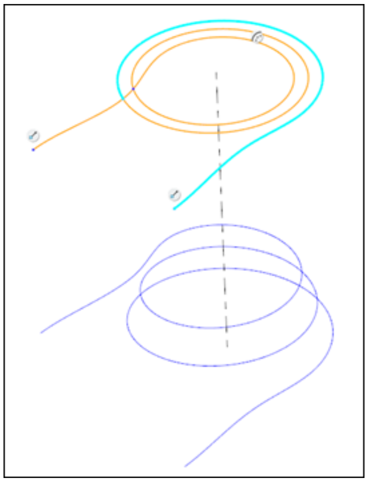
Creo Parametric 11.0.0.0

User Interface Location: Click **Model** ▸ **Sketch**.

Description

In Creo 11.0 you can now trim self-intersecting composite curves in Sketcher. This provides additional flexibility to further modify a composite curve which was created using the**Project** or**Offset** tool. Previously, self-intersecting composite curves could not be trimmed.

For example, in the image below, the upper curve self-intersects and was created by projecting the lower curve. The curve can now be trimmed, using a point of self-intersection to determine the segment to trim.



Benefits

Increased productivity by removal of limitation when trimming composite curves that self-intersect.

Additional Information

Tips:	None.
Limitations:	Sketch must be created in Creo 11.0
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Control Automatic Scaling of Palette Shapes in Sketcher

Creo Parametric 11.0.0.0

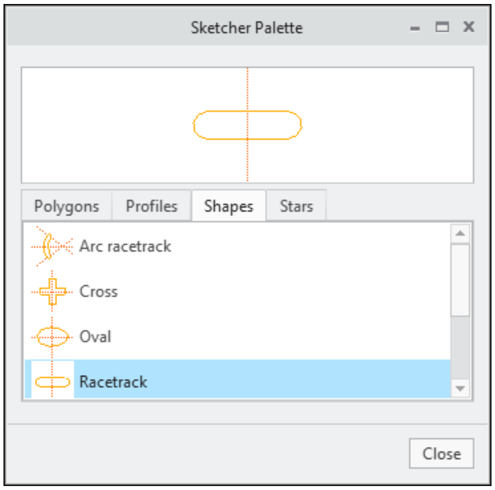
User Interface Location: Click **File ▸ Options**.

Description

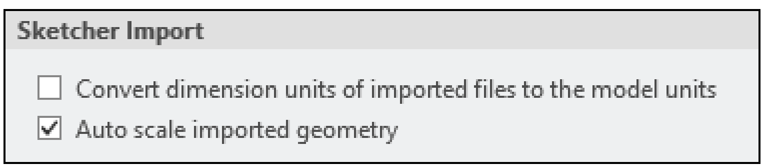
In Creo 11.0 there is a new configuration option to control the default scaling for imported geometry, including:

- Shapes imported using the **Palette** command
- Geometry from drawings or other files imported using the **Import** command

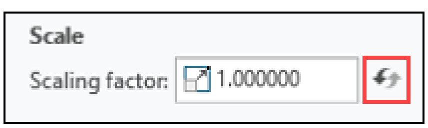
Often users create custom palette shapes such as O-rings, grooves, or slots at a 1:1 scale, meaning in the precise size that is needed. You can now choose to turn off the automatic scaling for palette shapes and maintain the defined 1:1 scale as the default scale during shape placement.



The default behavior can be controlled using the new **Auto scale imported geometry** setting under **File** ▸ **Options** ▸ **Sketcher** ▸ **Sketcher Import**.



When you change the scaling factor from the default value to a different value, you can also reset it back to the default value using the new reset button next to the scaling factor. Depending on the default method, it will reset the dimension value to the calculated auto scale value or to a value of 1.



Benefits

Increased flexibility to control system default behavior, resulting in improved efficiency.

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.

Configuration option associated with this functionality:	`sketcher_import_autoscale`
	`yes*`, `no`
	When importing shapes and other geometry using the **Palette** and **Import** commands, controls whether to auto scale the geometry or to set the default value of the scaling factor to 1.

19

Sheetmetal

Sheetmetal Multibody Overview

Creo Parametric 11.0.0.0

User Interface Location: N/A.

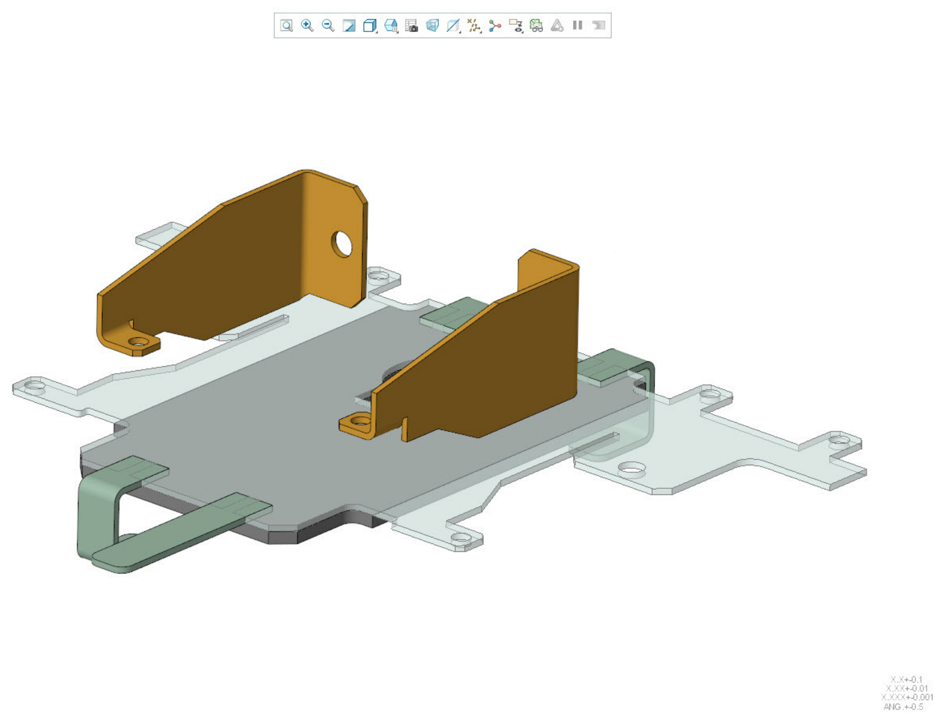
Videos

See the video on the Learning Connector.

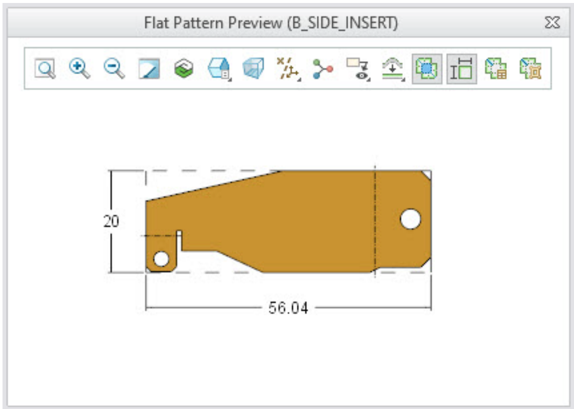
Description

The Sheetmetal design environment now has full multibody support. You can create multiple sheet metal bodies as well as the multiple solid bodies you could create before this enhancement. It is now possible to design multibody parts and then separate the bodies into different parts. Different bodies can have different thickness. Sheet metal parameter handling has changed.

Multibody support for sheet metal bodies includes typical body operations such as Boolean Operations, Split, Trim, Remove, Copy, Pattern, Mirror, and more.



You can create flat states for each body, visualize them together, and create flat instances or simplified representations for downstream usage such as drawing creation.



Benefits

Increase user productivity and design efficiency.

Use of multibody design in sheet metal parts and assemblies allows you to do the following:

- Easily and reliably design a single part that contains repetitive or mirrored geometric shapes.
- Design in context by applying the master model methodology for sheet metal designs.

Additional Information

Tips:	None.
Limitations:	Limitations are described in the Online Help.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Basic Multibody Part Creation and Workflow

Creo Parametric 11.0.0.0

User Interface Location: N/A.

Videos

See the video on the Learning Connector.

Description

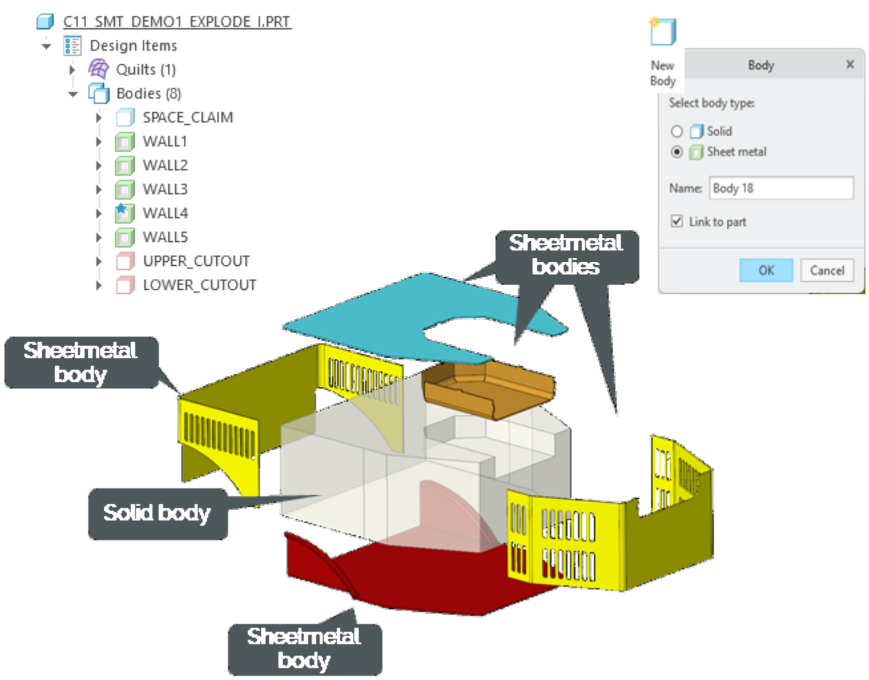
Full multibody support for Sheetmetal design. You can create multiple sheet metal bodies in addition to multiple solid bodies. Sheet metal characteristics (preferences) are now defined and driven by the body. For example, different bodies can have different thicknesses.

A new concept, specific to sheet metal bodies, enables you to link a body to the part. When the body is linked to the part, the sheet metal parameters of the body, for example SMT_Thickness, are driven by the part parameters.

Sheet metal bodies support the same body concepts such as the Default body and the Construction body attributes.

New and refined body creation workflows for sheet metal bodies include:

- The **New Body** command enables you choose the type of body to create: a solid body or a sheet metal body.
- There is now a **Body Options** tab in features that create unattached walls.
- You can convert a solid body to a sheet metal body.
- Body operations are available for sheet metal bodies.
- When you place user-defined Sheetmetal features, there is a **New Body** option.



Benefits

These enhancements increase user productivity and design efficiency.

With improved multibody design methodologies, you can now:

- Easily and reliably design a single body part that contains repetitive or mirrored geometric shapes.
- Design in context by applying the master model methodology for sheet metal designs.

Additional Information

Tips:	None.
Limitations:	• At least one sheet metal body must exist in the design. • The First Wall feature that creates the initial geometry of a sheet metal body cannot be suppressed.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Boolean and Body Operations in Multibody Sheetmetal

Creo Parametric 11.0.0.0

User Interface Location: Click **Sheetmetal** ▸ **Boolean Operations**.

Click **Sheetmetal** ▸ **Split/Trim Body**.

Click The arrow next to **Body** and click **Remove Body**.

Use the commands in the **Operations** group.

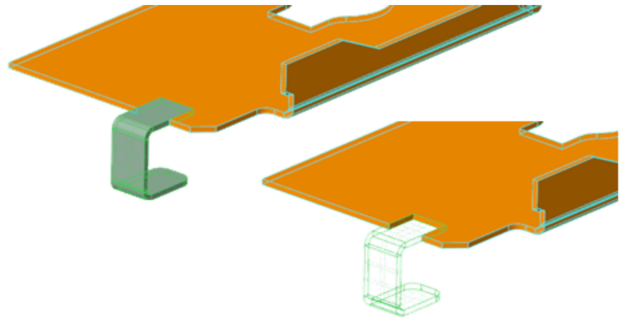
Videos

See the video on the Learning Connector.

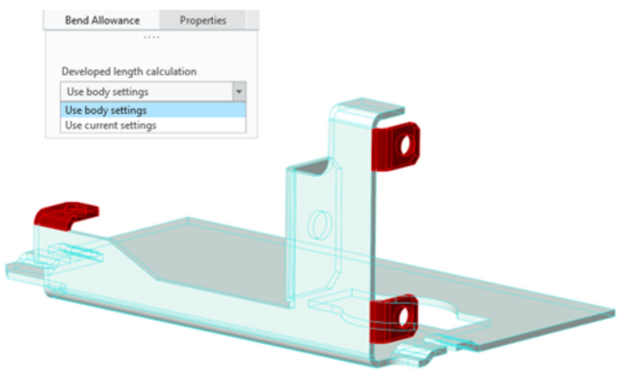
Description

With full multibody support, **Boolean Operations** and **Split** and **Trim** are available for sheet metal bodies.

- Boolean Operations (merge, subtract, and intersect) options specific to Sheetmetal
  - For subtract and intersect the **Normal to Surface** option controls material removal.

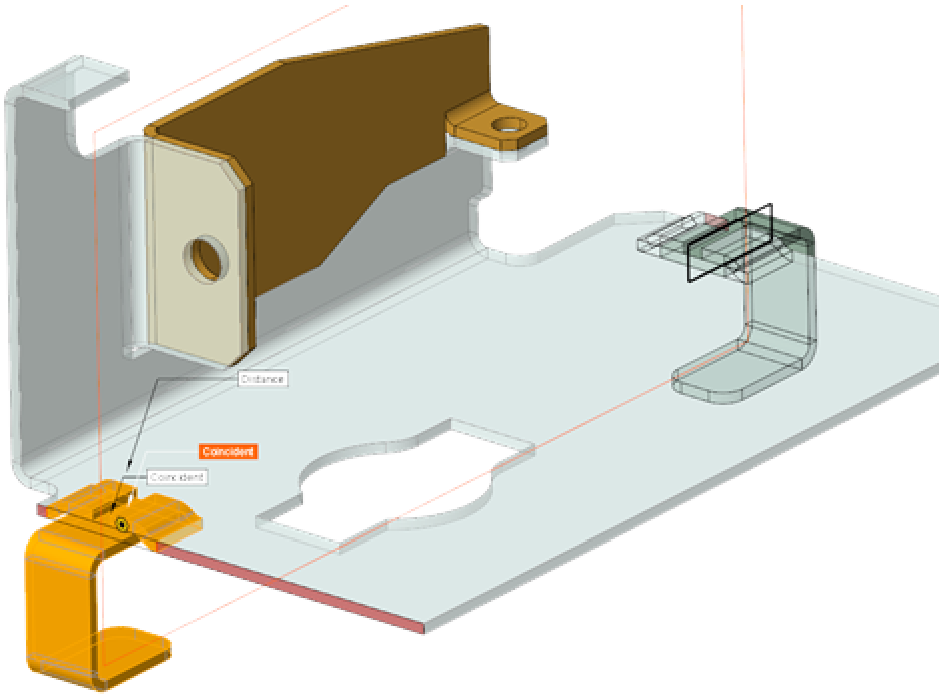


- For merge there is the option to change bend allowance values for bends in modifying bodies when you select **Use body settings**, or to keep the bend allowance values of the modifying bodies when you select **Use current settings**.

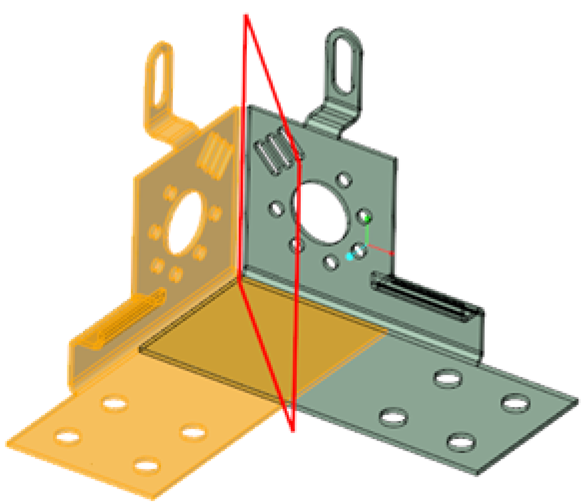


- Split body options specific to Sheetmetal:
  - The option to use a sheet metal cut (an extruded cut) or a solid cut.
  - **Split by Volume** allows you to split out individual distinct volumes, previously known as distinct pieces, from a sheet metal body to a new sheet metal body.
- Other body operations include, **Remove Body**, **Copy**, **Paste**, **Paste Special**, **Mirror**, **Pattern**, Flexible Modeling commands, and **Copy Geometry**. A local copy geometry feature creates a new sheet metal body, and an external copy geometry feature creates a regular solid body.

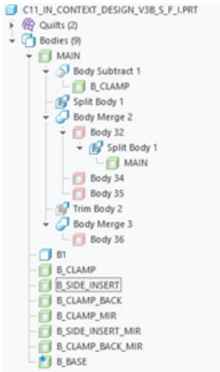
  An example of a flexible move operation:



An example of a mirror operation on a body:



An example of the Quilt Body Evolution Tree with body operations.



Benefits

These enhancements increase user productivity and design efficiency.

With improved multibody design methodologies, you can now:

- Easily and reliably design a single body part that contains repetitive or mirrored geometric shapes.
- Design in context by applying the master model methodology for sheet metal designs.

Additional Information

Tips:	None.
Limitations:	Remove Body—You cannot remove the last remaining sheet metal body in the part. Bend allowance—When you use **Use body settings**, you can only merge bodies that do not contain flattened bends. Split Body—In some situations, only the solid cut can successfully produce the resulting geometry required. Boolean Intersect and Subtract— **Copy Surface Appearance** and **Update References** are only available when the cutting option **Normal to Surface** is not the active option.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Multibody Sheetmetal Convert Workflow and Using Sheet Metal Parameters and Preferences

Creo Parametric 11.0.0.0

User Interface Location: Right-click the body and select **Convert to Sheetmetal**.

Videos

See the video on the Learning Connector.

Description

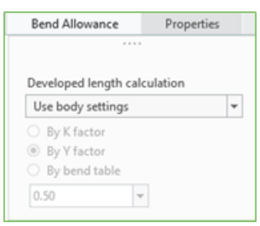
Link to Part

A new concept specific to sheet metal bodies, enables linking a sheet metal body to the part.

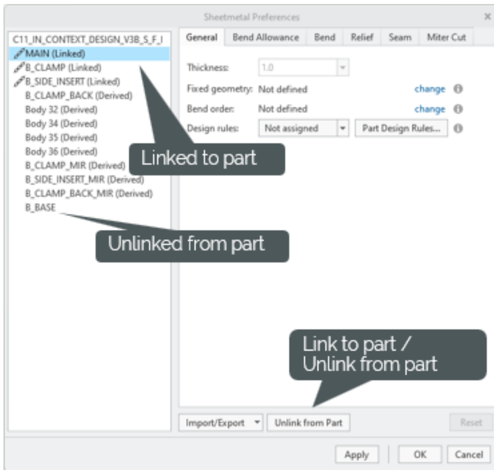
- When a body is linked, the sheet metal parameters are driven by the part-level preferences and parameters.
- When a body is unlinked, the body can have different thickness and other sheet metal parameters.
- Derived bodies, bodies create by a copy, split, or similar operation, have a linked thickness value.

Parameters and Properties

Preferences defined for each body drive the design. Features such as a Flat wall, by default, use the bend allowance set by the body settings.

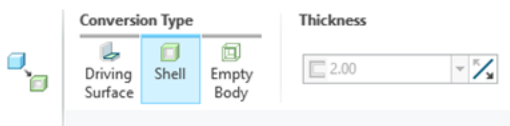


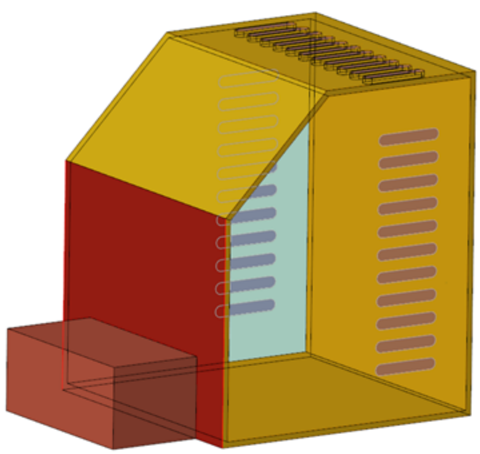
All bodies and preferences for body parameters in the design are listed in the **Sheetmetal Preferences** dialog box.



Conversion

When converting a part to sheet metal, the default body is the first body converted.





When you convert the first body, setting a thickness in the **Convert** tab results in the following:

- Sets the body thickness
- Sets the part thickness parameter if it does not exist for the part
- Automatically links the body to the part settings

When converting secondary bodies there is the choice to keep model parameters or keep parameter values. When the thickness is the same as the part thickness, the new bodies are linked to the part as well.

Benefits

- Ability to convert a single solid body to a sheet metal body
- Increased productivity and design efficiency

Additional Information

Tips:	None.
Limitations:	Limitations are documented in the Online Help.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Master Model Methodology in Sheetmetal

Creo Parametric 11.0.0.0

User Interface Location: Right-click a body and select **Create Part from Body**.

Videos

See the video on the Learning Connector.

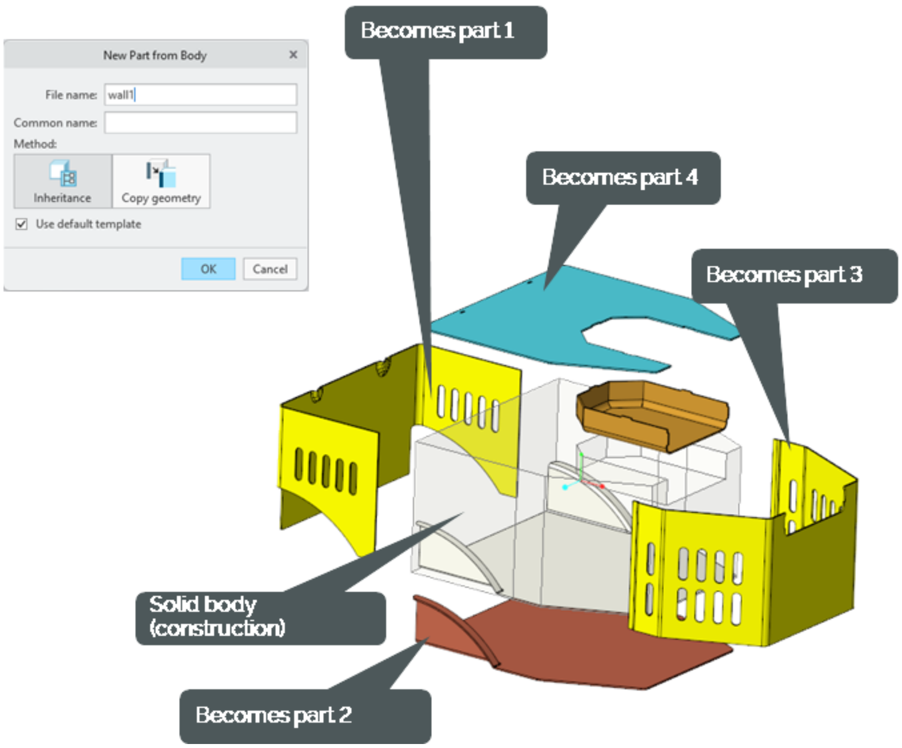
Description

Creo 11 introduces full multibody support for the Sheetmetal Design environment. This enables the master model design use case where parts are designed in the context of a multibody part to then extract the individual bodies into separate parts. Use the **Create Part from Body** command to create a new sheet metal part from a sheet metal body.

- Use the **Inheritance** command to extract a sheet metal body.
  - Creates a new sheet metal part using the Sheetmetal template.
  - Creates an external inheritance feature adding all bodies to the reference model.
  - Adds a **Remove Body** feature to remove all bodies except the selected body.
  - Drives all body parameters by the inheritance feature.
  - Sets the sheet metal part parameters to the parameters of the selected body, but does not link them.
  - Supports the use of a regular flat pattern feature in the inheritance part.

- Use the **Copy Geometry** command to extract a solid body.

  Creates an external copy geometry feature.



Benefits

Increased productivity and design efficiency by supporting the master model methodology for the design of sheet metal parts and assemblies.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Model Check Support for Multibody in Sheetmetal

Creo Parametric 11.0.0.0

User Interface Location: Click **File** ▸ **Prepare** ▸ **Model Check**.

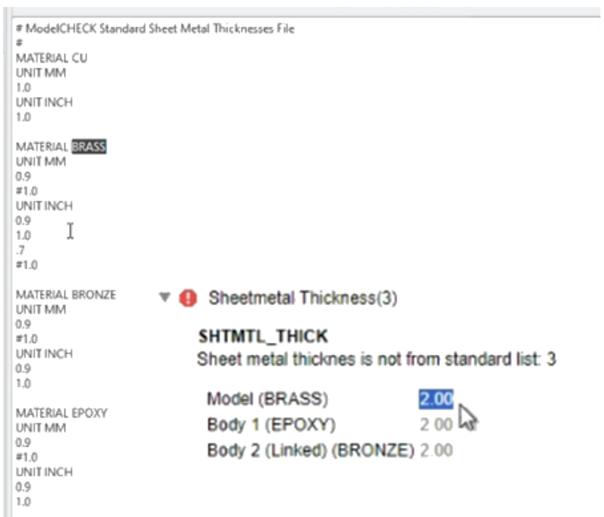
Videos

See the video on the Learning Connector.

Description

Model Check capabilities are enhanced to accommodate full multibody support for sheet metal bodies. The following checks are added to the model check:

- SHTMTL_THICK—The thickness table now includes allowed thickness values per material. Model Check reports incorrect thicknesses per body. There is support of regular expressions for material names.



- SHTMTL_BENDTAB—Reports bend table assignments not from the standard list. Check is performed for each body and is reported for all failing bodies.

- SHTMTL_YFACTOR, SHTMTL_KFACTOR—Reports the Y-Factor and K-Factor per body.
- SHTMTL_UNBENDS—Reports the number of consecutive unbend/bend-back features, regardless whether they are from the same body.
- SHTMTL_FLAT—Checks for the presence of flat pattern feature. Reports the number of bodies with a missing flat pattern feature.

Benefits

Model check can now be configured to fully cover sheet metal multibody designs.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Configuration Option to Control Appearance of Flat Pattern Commands

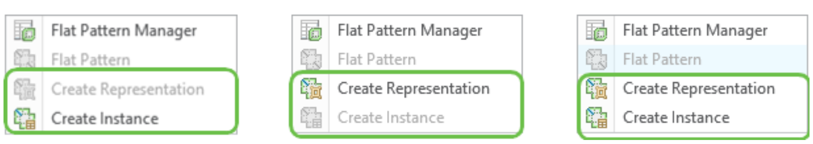
Creo Parametric 11.0.0.0

User Interface Location: Click **File ▸ Options** under **Core** click **Sheetmetal**.

Description

Configure the user interface to present the Flat Pattern drop-down options based on your company's preferred method to manage bend states and flat states of the model.

By default, options for both Family Table instances and Simplified Representations are available. Based on your preferences, you can now configure only one of these options to be accessible making it easier for your users to follow a standard practice.



The same behavior applies to the corresponding commands in the **Flat Pattern Preview Window** as well.

Benefits

Better user guidance to the preferred flattening method and tool of choice

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	`smt_flat_rep_inst_option`

Unbending and Creating Flat Patterns

Creo Parametric 11.0.0.0

User Interface Location: Click **Sheetmetal** ▸ **Unbend**.

Click **Sheetmetal** ▸ **Flat Pattern** ▸ **Flat Pattern Manager**.

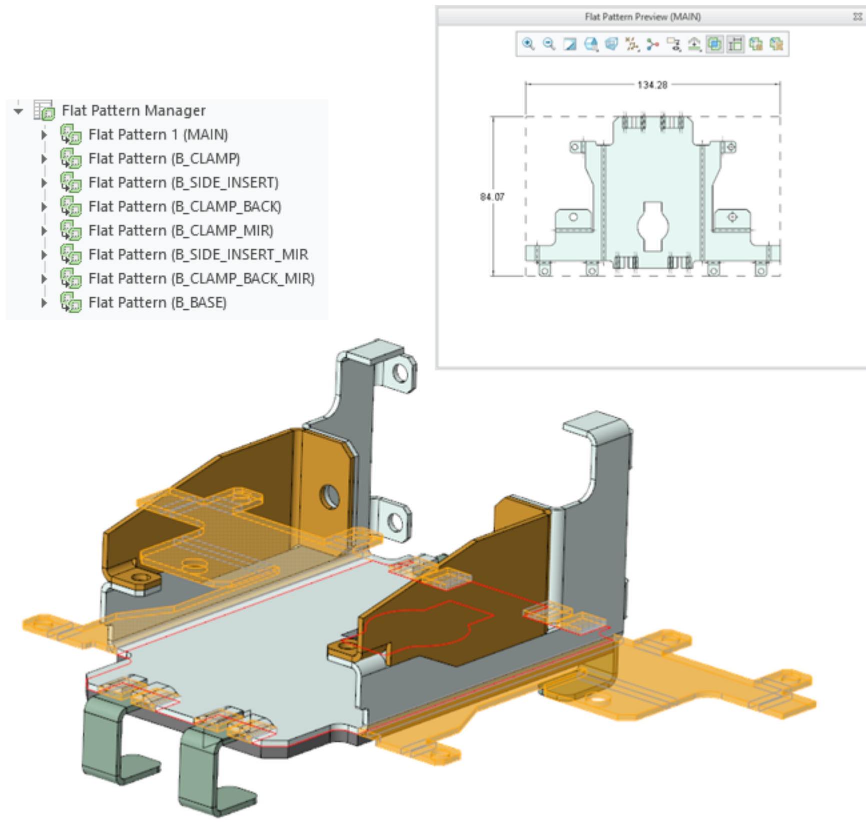
Videos

See the video on the Learning Connector.

Description

Multibody support has been extended to sheet metal bodies enabling you to unbend and create flat patterns for multiple sheet metal bodies. Both the unbend and the flat pattern features are used for individual sheet metal bodies. A part with multiple sheet metal bodies may have several of these features, one for each body.

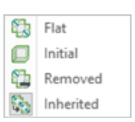
Define fixed geometry and bend order for each body in the **Sheetmetal Preferences** dialog box.



A flat pattern preview is available for each body. The **Flat Pattern** feature continues to be supported for a single sheet metal body. When flat patterns are required for multiple sheet metal bodies, use the **Flat Pattern Manager**. It automatically creates and manages flat patterns for each of the sheet metal bodies in the part. The Flat Pattern Manager state parameter can be one of:

- Flat—Flattened state
- Initial—As designed state
- Removed—Body is removed

Individual flat pattern features for each body have an additional state parameter called Inherited, which indicates that the state of the flat pattern was inherited from the state of the Flat Pattern Manager. This is the default state. Flat state instances and flat state representations set the states of the flat pattern features of individual bodies upon their creation. For an empty body, the flat pattern feature is hidden in the Flat Pattern Manager.



Benefits

The Flat Pattern Manager can set the flat state for all sheet metal bodies, or the flat state can be set for each sheet metal body individually.

Additional Information

Tips:	None.
Limitations:	Swept flanges that are split with a split body operation cannot be unbent simultaneously.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	`smt_flat_instance_name-format`—Specifies the naming format of the flat state instance. `smt_flat_simp_rep_name_format`—Specifies the naming format of the flat state representation. `smt_flat_inst_index`—Removes gaps in index numbers of flat pattern Family Table instance names.

20

Simulation

Conjugate Heat Transfer Studies in Creo Simulation Live

Creo Parametric 11.0.0.0

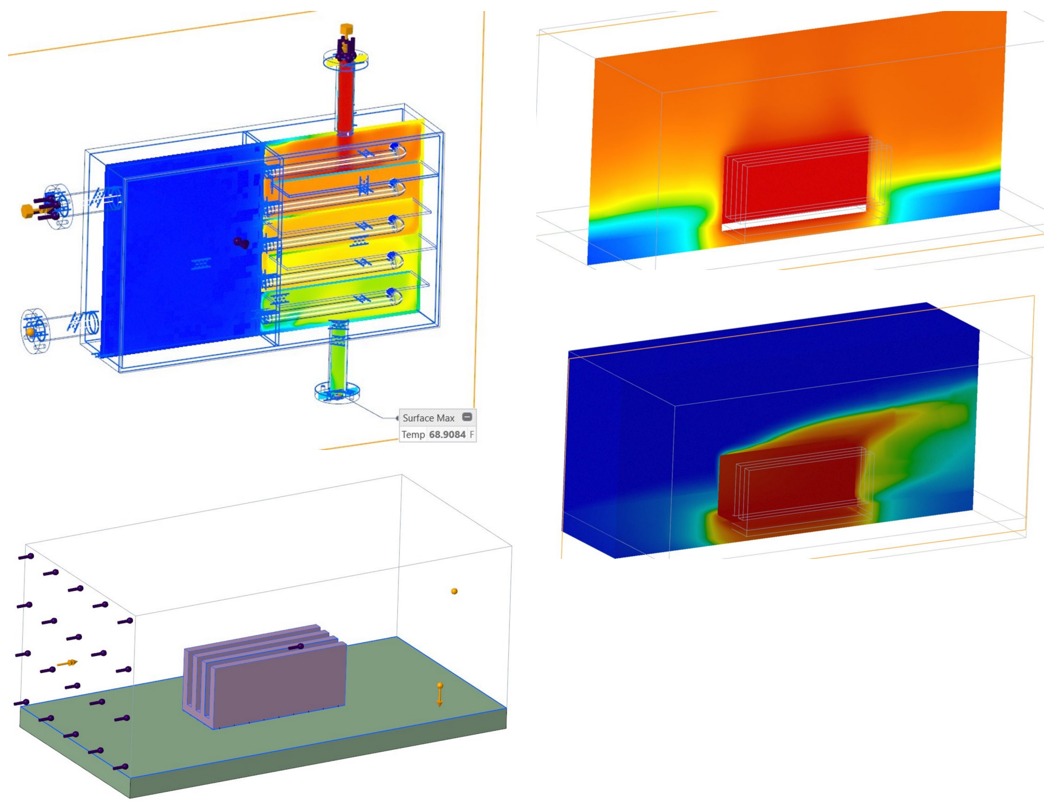
User Interface Location: For a fluid simulation study, select the **Conjugate Heat Transfer** check box from the overflow menu of the **Study** group on the **Live Simulation** tab. **Live Simulation** ▸ **Study** ▸ **Conjugate Heat Transfer**

Videos

See the video on the Learning Connector.

Description

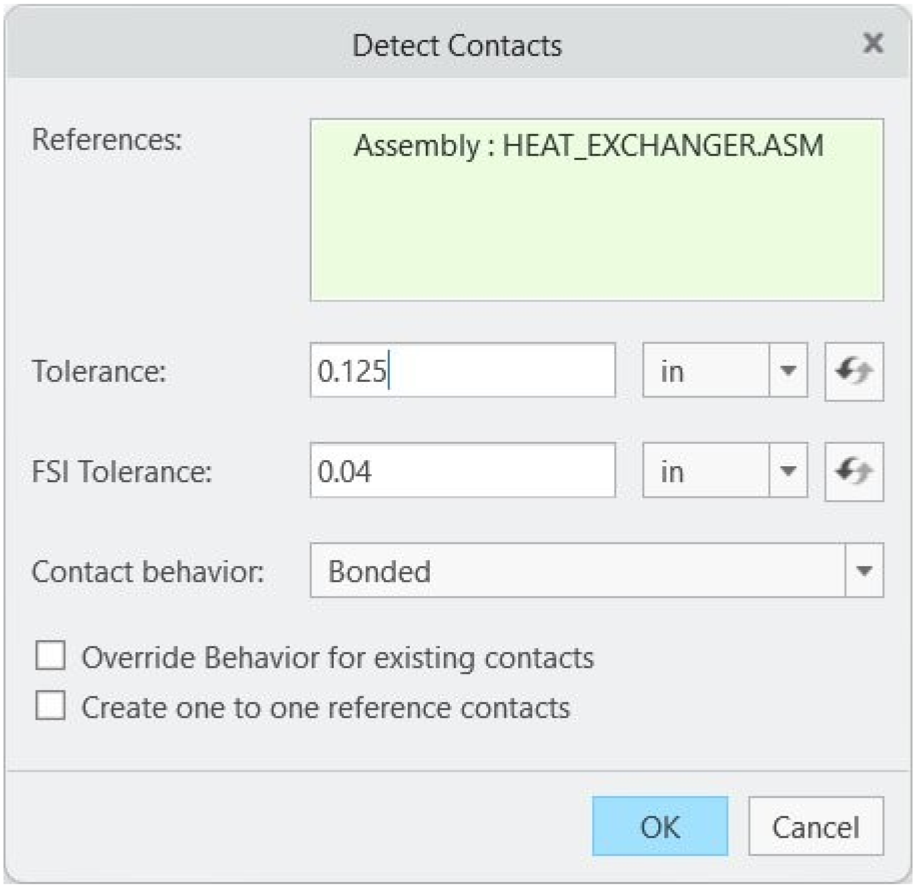
With this enhancement you can simulate the heat transfer between solids and fluids for both internal and external studies. Conjugate heat transfer can be simulated using a forced flow or a natural convection situation where heat is lost into the surrounding fluid body. A single study can have multiple fluid types. Multiple fluid domains must be separated by a solid wall.



There is improved contact control implemented as follows:

- Automatic contact detection includes solid-fluid connections.

- There is a new ‘FSI’(Fluid-Solid Interface) tolerance option for contact detection between solids and fluids.



Benefits

- Incredible speed in solving complex studies
- Accurately predict heat transfer of combination of solids and fluid flow
- Optimize designs from CHT results

Additional Information

Tips:	None.
Limitations:	External fluid domains must be solid domains only. You cannot use the quilt based enclosure volume feature in Creo to create an external fluid domain for a conjugate heat transfer study.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Transient Structural Studies in Creo Ansys Simulation

Creo Parametric 11.0.0.0

User Interface Location: For an active structural study, select **Ansys Simulation ▸ Setup ▸ Transient Mode**.

Videos

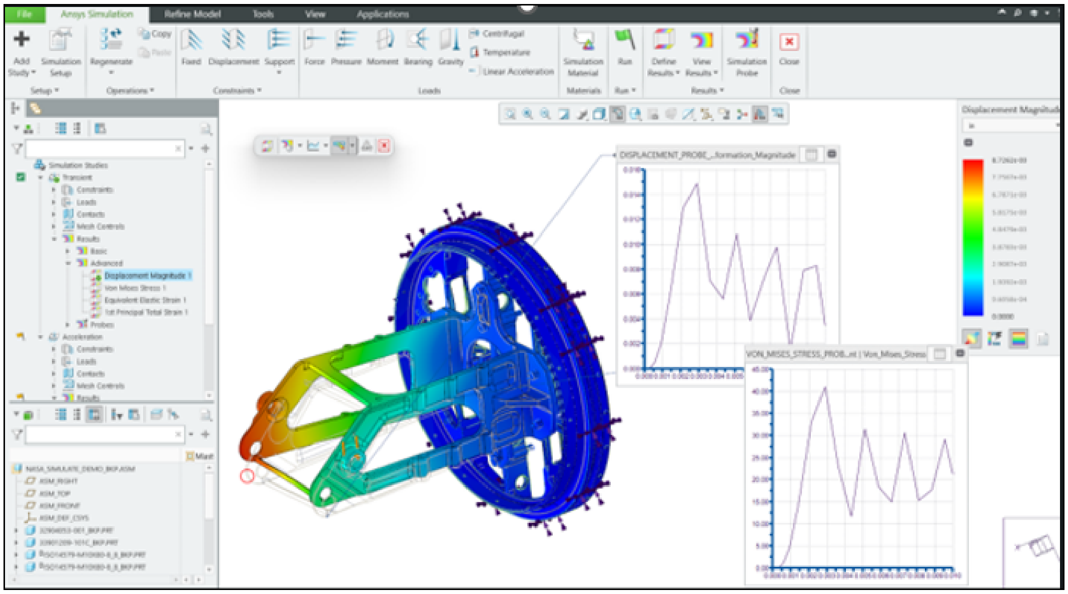
See the video on the Learning Connector.

Description

You can now run transient structural analyses in Creo Ansys Simulation to study the dynamic response of a structure when a time-varying load is applied to it. You can define loads for transient studies as a function of time using a table function and study the effects of this load on the resultant displacements, deformation, stresses and strains in the model. Constraints can also be defined as a function of time.

The following additional simulation set up options are available:

- Initial Velocity
- Damping
- Step Duration
- Time-based sub-stepping



You can view result graphs over time for all contour plots and probes.

Benefits

The following are the benefits of this enhancement:

- Determines the dynamic response under time-history loading
- You can use transient structural studies to predict structural integrity, natural frequencies, and modes of vibration for a model.
- You can define and run multi-step, time dependent structural simulation.

Additional Information

Tips:	None.
Limitations:	• When defining time-varying load functions you can only use table functions. Formula or symbol based (expression) functions are not available. As a workaround you can create a table using a formula in Excel and then import the date as a table function. • Basic results for a transient structural study display the value of the quantity for the last time value (static). • Animation of results is not available for a transient structural analysis as deformation results are not available. This will be available in a later release. • Transient structural studies consume considerable

	computing resources and hence they take a longer time to execute. Display of the results graph against time also takes a long time.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Creo Simulation Live and Creo Ansys Simulation—Upgraded to Ansys 23R2 Solver

Creo Parametric 11.0.0.0

Description

Both Creo Simulation Live and Creo Ansys Simulation are upgraded to the latest Ansys 23R2 solver. This upgrade provides faster and more accurate solutions for all types of physics by using the most current Ansys technology.

Benefits

- Better GPU performance.
- Higher accuracy in results.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Expanded Results for Creo Simulation Live

Creo Parametric 11.0.0.0

User Interface Location:

Changes to Simulation Probe—**Live Simulation** ▸ **Simulation Probe**.

Additional dynamic query of live simulation results—**Live Simulation** ▸ **Simulation Query**

More than 6 modes available for Modal Studies: Click **Live Simulation** ▸ **Study** ▸ **Simulation Options** and select upto 12 modes for a modal study.

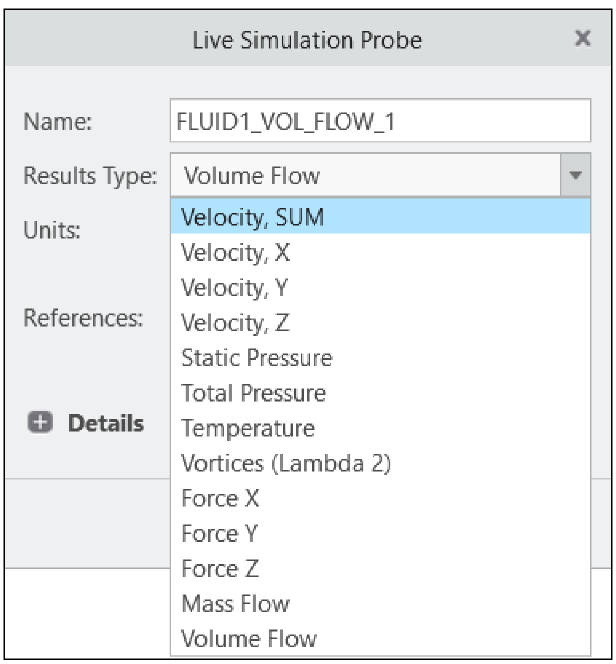
Videos

See the video on the Learning Connector.

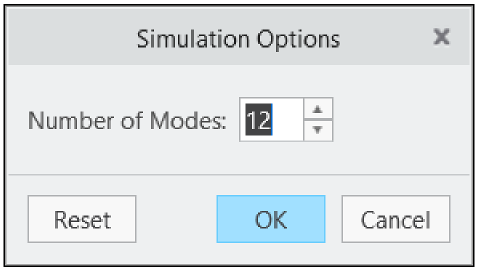
Description

The following enhancements to results in Creo Simulation Live are implemented:

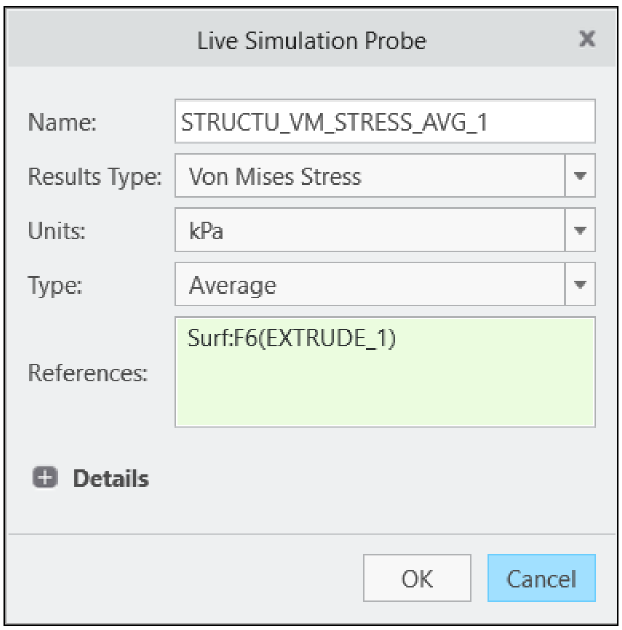
- You can now probe result quantities that are available on the results legend as well as additional result quantities such as mass flow, volume flow, heat, natural frequency. You can now create probes before running the simulation study.



- You can select 12 mode shape values (earlier the maximum number of modes was 6). The first six mode shapes are the rigid body modes for an unconstrained model. You can probe the modal frequency for the selected mode.



- You can probe the average value of result quantities such as stress, strain, temperature, mass flow, etc. for a selected reference.



- Additional result quantities are available for static and total pressure.
- Bodies are now selectable for results and probes, for all types of studies.
- You can select multiple references of the same type when creating a probe for a result quantity. For example, you can select multiple surfaces over which to probe the average value of stress.
- A dynamic query similar to that available in Creo Ansys Simulation allows you to query the value of a result quantity selected on the Results legend, at any point on the model.

  The dynamic query can be converted to a probe and saved.

Benefits

- Many more result options are available to users.

  All new probes can be used in Behavioral Modeling studies.
- Users can dynamically query the value of a quantity at any point on the model.
- The pressure drop use case is now possible with the static and total pressure results available.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	The dynamic query replaces the simulation probe (provides only value at point). The new simulation probe is similar to the probe available in Creo Ansys Simulation.
Configuration option associated with this functionality:	None.

21

Surfacing

Curve Edit: In View Point Move

Creo Parametric 11.0.0.0

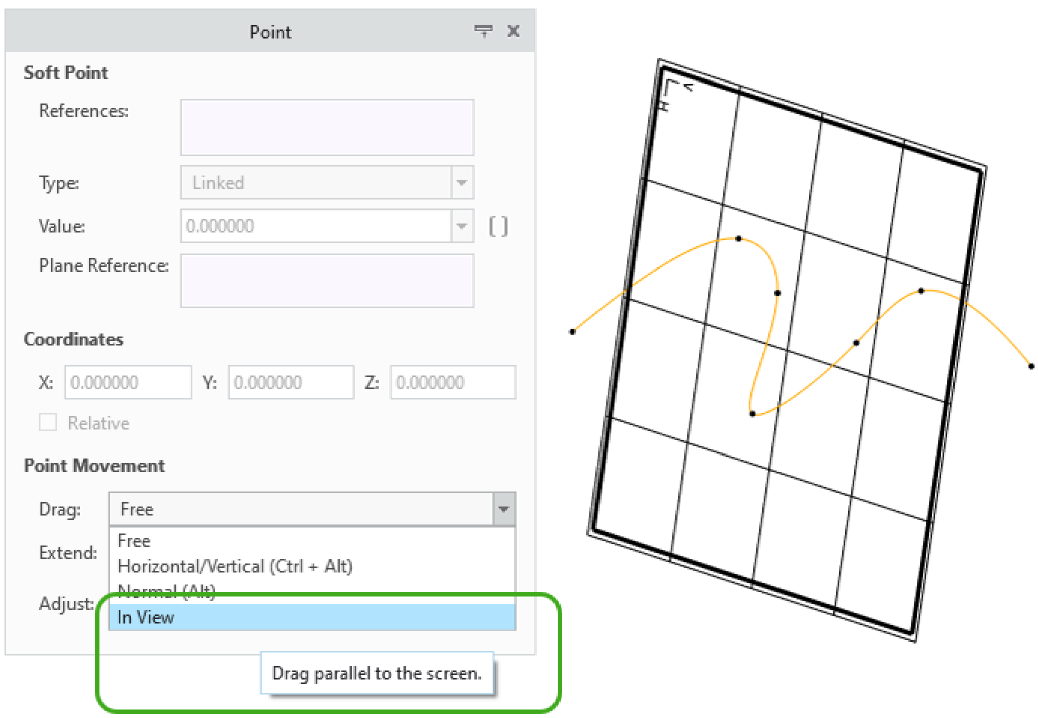
User Interface Location: Click **Style** ▶ **Curve Edit** ▶ **Point** tab, **Style** ▶ **Move**, or **Style** ▶ **Curve** ▶ **Copy**, and in the **Drag** list, select **In View**.

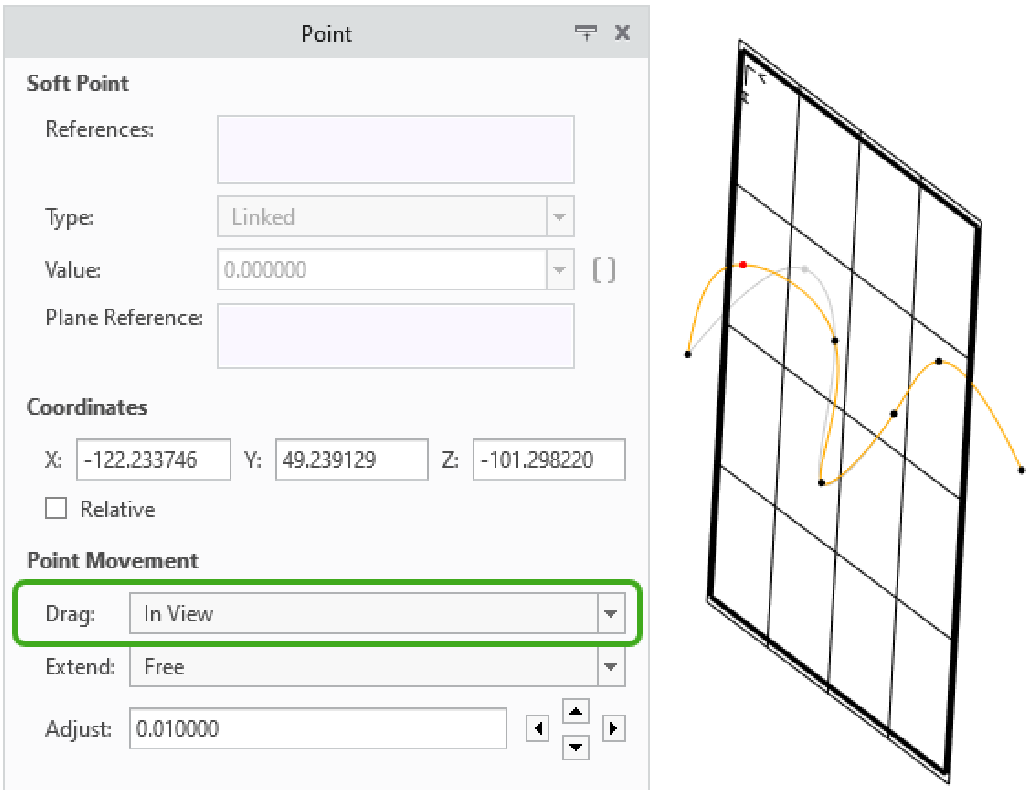
Videos

See the video on the Learning Connector.

Description

The new **In View** method of moving points lets you move free curve points parallel to the computer screen, and not in relation to the curve's reference plane or the current active plane, when you edit, move, or copy curves.





Benefits

The **In View** method of moving points gives you more control to easily move free curve points parallel to the computer screen.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Curve from Surface: Improved Quality

Creo Parametric 11.0.0.0

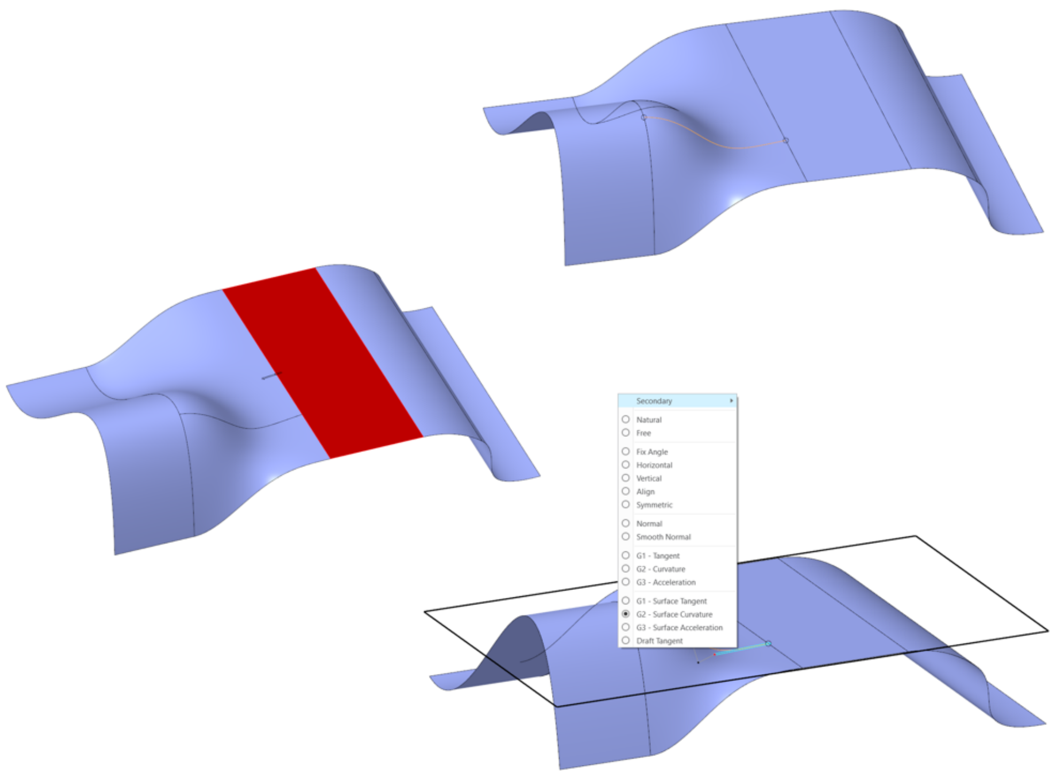
User Interface Location: Click **Style** ▸ **Curve from Surface**.

Videos

See the video on the Learning Connector.

Description

Curves from Surface now have improved curve quality by constructing the curve so it inherits the same connections on its endpoints as its parent surface. The Curve from Surface will inherit the surface definitions from the parent surface, such as the same degree, knot vector, and number of control points as its parent surface.



Benefits

- Improved curve quality for Curves from Surface
- Curve from Surface maintains the design intent from the parent surface

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Style: Improve Curve Quality with Natural Tangency

Creo Parametric 11.0.0.0

User Interface Location: Click **Style** ▸ **Curve**.

Videos

See the video on the Learning Connector.

Description

Style has been improved to enhance the curve quality of single natural tangency curves without internal knots.

Previously two-point curves without internal knots had distortion. Without internal knots, there was inflection in the curve’s free end when the other end's tangency was defined.

In Creo Parametric 11, the algorithm of natural tangency calculation for a curve with only two interpolation points and a single tangency was improved, to result in a more natural curve with no inflection.

Benefits

Improved curve quality for single natural tangency curves without internal knots.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Style: Isoline Reference Datum Point

Creo Parametric 11.0.0.0

User Interface Location: Click **Style** ▸ **Curve from Surface** ▸ **Isoline** ▸ **By Reference**.

Videos

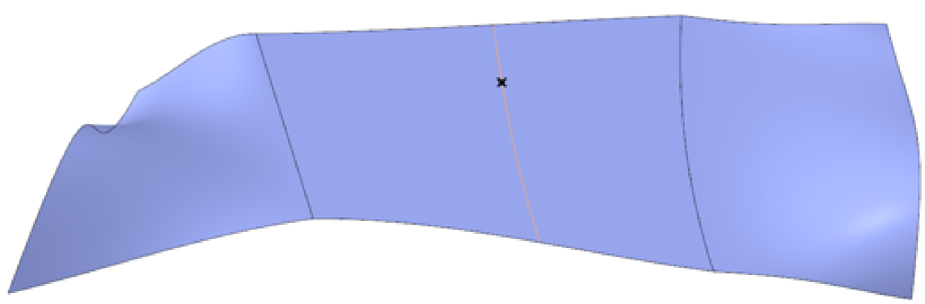
See the video on the Learning Connector.

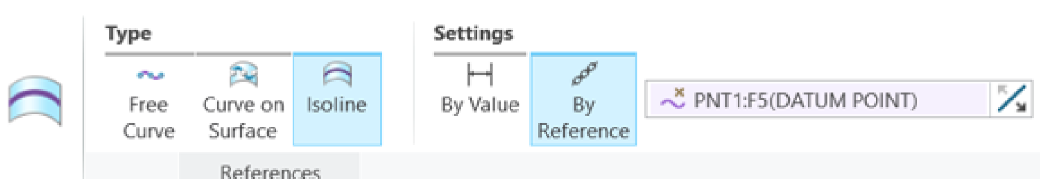
Description

Starting in Creo Parametric 11, a datum point can be used as a reference for defining an isoline curve defined using **By Reference**. The datum point must lie on the same surface as the curve.

This allows you to control the location of the isoline with the placement of a datum point. When the datum point is updated, the isoline curve will also be updated to reflect the new reference location, as long as it is on the reference surface.

Previously, only a curve could be used as a reference for defining an isoline curve.





Benefits

Broader range of references for creating isoline curves

Additional Information

Tips:	None.
Limitations:	If the reference datum point is no longer on the specified surface, the isoline will fail until there is a reference on the specified surface.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Style: Low Degree Curves

Creo Parametric 11.0.0.0

User Interface Location: Click **Style** ▸ **Curve** or **Curve Edit** ▸ **Control Points**.

Videos

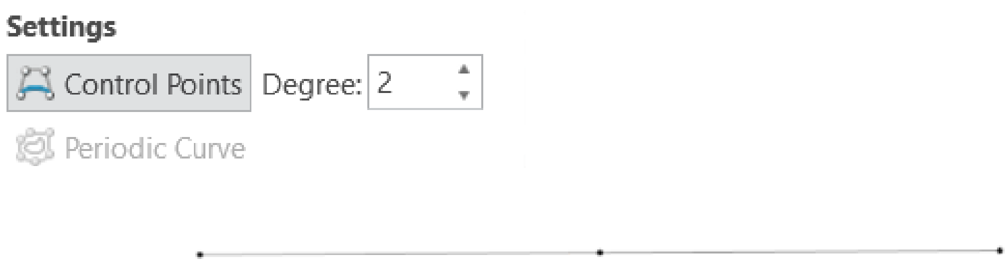
See the video on the Learning Connector.

Description

You can now create curves of degree 1 or 2 when in Control Point mode. This results in a more natural curve with no inflection in Control Point mode.

A curve of degree 1 can have two control points. A curve of degree 2 can have a maximum of three control points.

These minimal allowed degrees will depend on the curve connection. The default remains as degree 3, but you can reduce it to degree 2 or 1.



Benefits

Improved curves in Style and greater control over low degree curves.

Additional Information

Tips:	None.
Limitations:	Curves of degree 2 or 1 can only be created in Control Point mode. If a curve of degree 2 or 1 is converted to Interpolation mode, the degree will automatically change to degree 3. Additionally, the curve tangency is changed from Natural in low degree control point mode, to Free in 3rd degree Interpolation mode.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Style: Tooltips on Curve Tangents

Creo Parametric 11.0.0.0

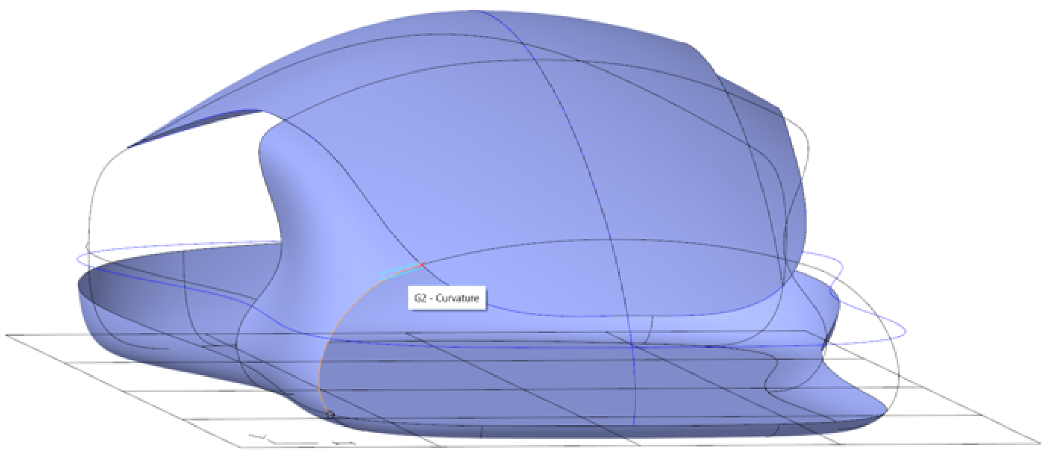
User Interface Location: Click **Style** ▸ **Curve Edit**, and hover over the tangent.

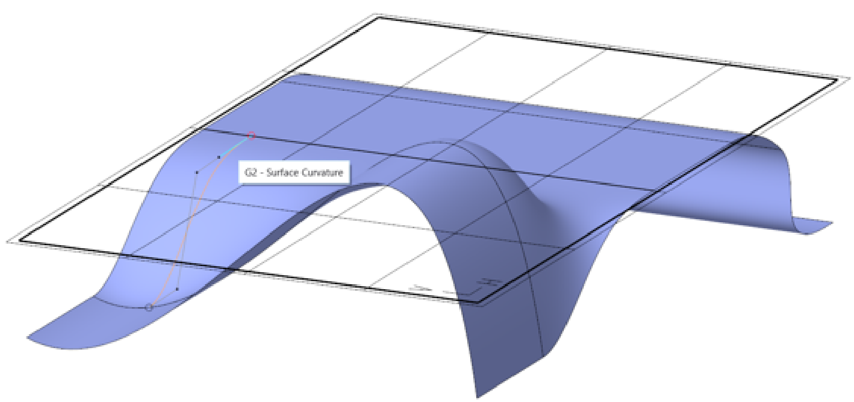
Videos

See the video on the Learning Connector.

Description

Tooltips are now provided when you hover over a curve tangent in Style. These tooltips show the connection type, removing the need to right-click the curve tangent to know the status of the connection type.





Benefits

Improved user experience when working with curves in Style

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Style and Warp: Updated Draggers

Creo Parametric 11.0.0.0

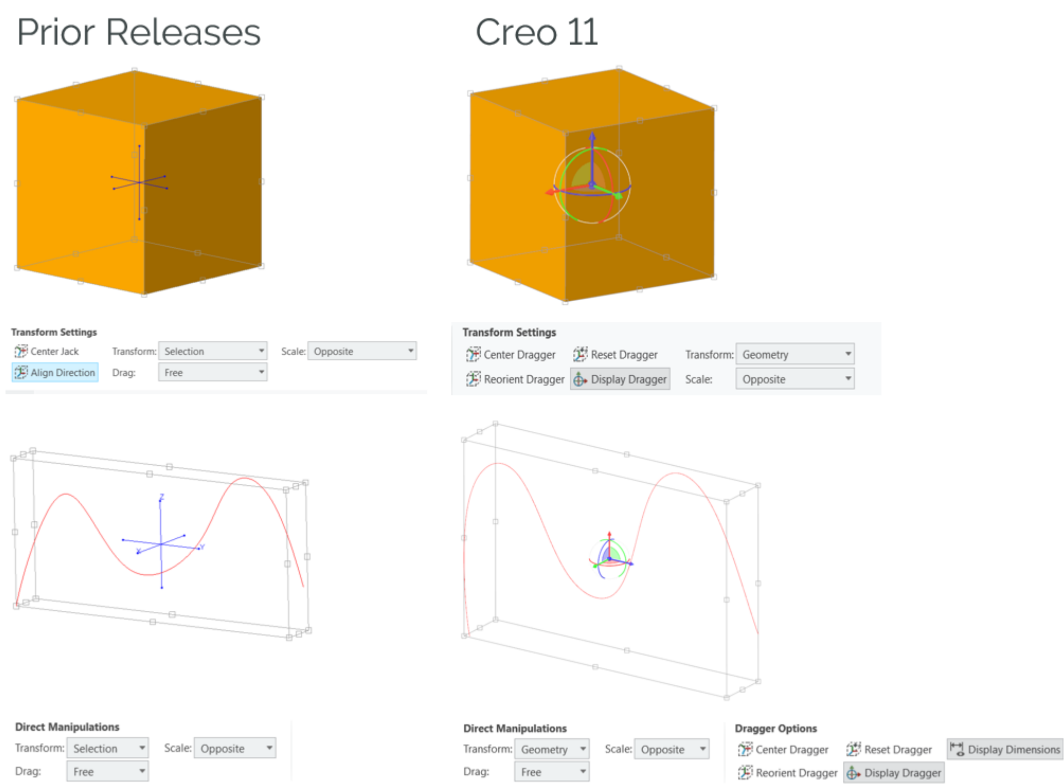
User Interface Location: Click **Model** ▸ **Style** or **Model** ▸ **Editing** ▸ **Warp**.

Description

The dragger in the Style **Move** and **Copy** tools has been updated to the more modern 3D dragger used elsewhere in Creo. The dragger in Warp has also been updated to the 3D dragger.

The feature tabs in Style and Warp were updated to reflect the options available with the 3D dragger. **Reset Dragger**, **Display Dragger** and transform and scale options were added to the tabs.

The Move and Rotate controls were placed on the **Options** tab. The active handle field was removed, as it is now easy to select the desired handle or arc on the color-coded dragger.



Benefits

- Updated 3D dragger with enhanced functionality
- Style and Warp now aligned with other Creo tools that use a 3D dragger

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Warp: Improved Dimensions Handling

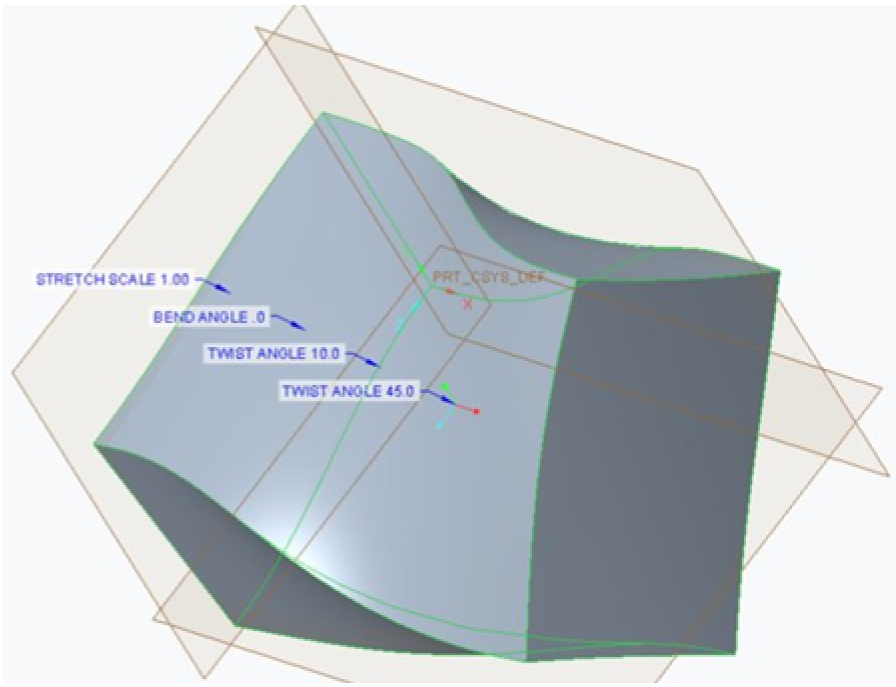
Creo Parametric 11.0.0.0

User Interface Location: Click **Model** ▸ **Editing** ▸ **Warp**.

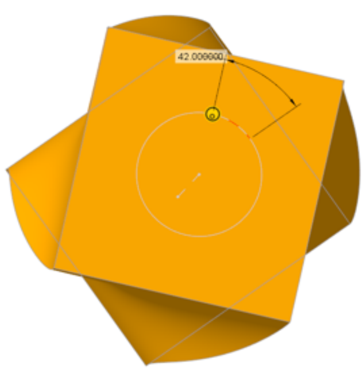
Description

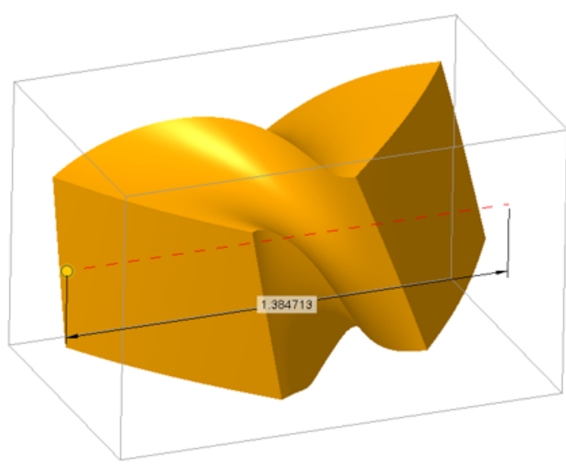
The dimensions in Warp have been improved for better usability during editing. Each dimension now appears close to the relevant geometry it controls. Additionally, there is a new angular appearance for angle dimensions during editing in Warp.

Dimensions in prior releases:



Dimensions in Creo Parametric 11:





Benefits

Improved usability when editing in Warp

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Warp: Improved Performance with Multithreading

Creo Parametric 11.0.0.0

User Interface Location: Click **Model ▸ Editing ▸ Warp**.

Description

Warp has been improved to use parallel calculations, or multithreaded calculations, during approximation of surfaces in Warp regeneration. This reduces the time it takes to regenerate, and improves the shape's response during the preview and drag operations.

Benefits

Improved performance when editing in Warp

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Style: Surface Connections Table

Creo Parametric 11.0.0.0

User Interface Location: Click **Style ▸ Surface ▸ Constraints** tab.

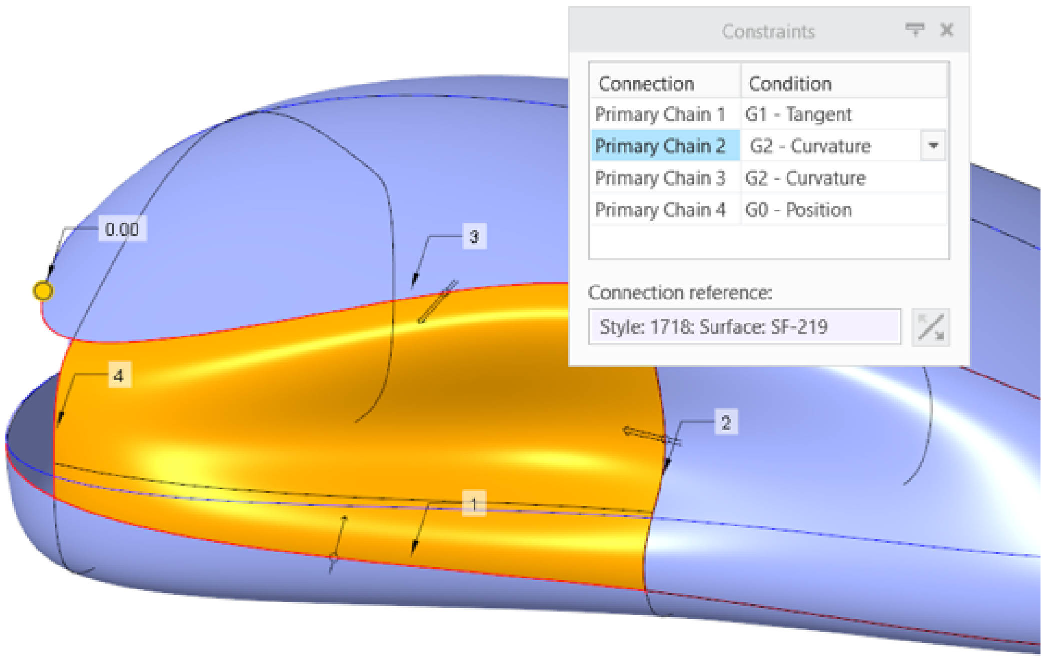
Videos

See the video on the Learning Connector.

Description

A new connections table has been added to the Surface tool in Style. For each surface, the table displays the connections that can be fully defined in the current surface definition, the connection type, and the reference for the connection.

You can edit the connection using the table, but surface connections cannot be edited in Reparameterization mode.



Benefits

The new surface connections table provides convenient access to the connection definitions for each surface. This allows you to investigate and fix surface failures more easily.

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Freestyle: Bevel Command

Creo Parametric 11.0.0.0

User Interface Location: In Freestyle, click **Create ▸ Bevel**.

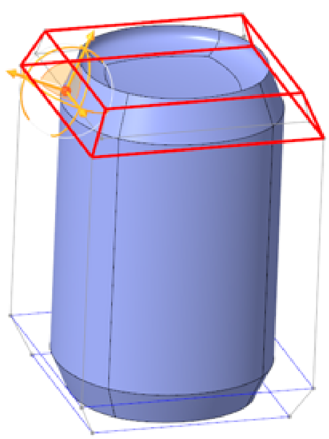
Videos

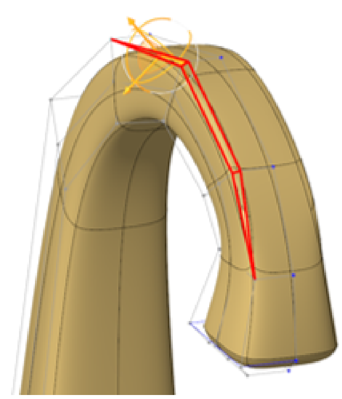
See the video on the Learning Connector.

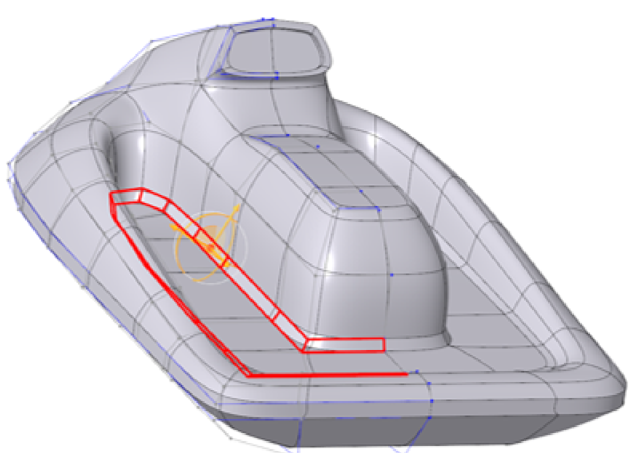
Description

The new **Bevel** command creates a bevel along an edge or within a face. It creates two new edges and a face.

This enhancement is useful when modifying the sharpness of edges, adding new features to a face, and adding a chamfer to the edge of a part.







Benefits

This new freestyle modeling capability helps you better capture your design intent.

Additional Information

Tips:	None.
Limitations:	The bevel operation cannot be performed on a single edge of the mesh. It requires two or more edges of the mesh.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Freestyle: Mesh Cut Command

Creo Parametric 11.0.0.0

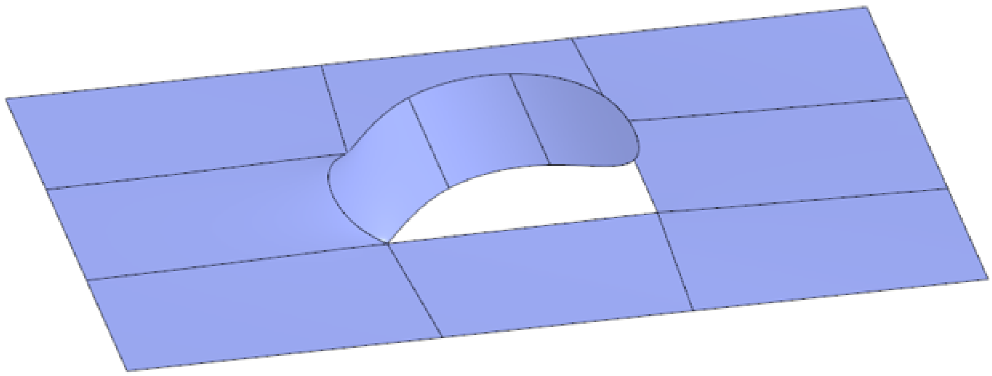
User Interface Location: In Freestyle, click **Create** ▸ **Mesh Cut**.

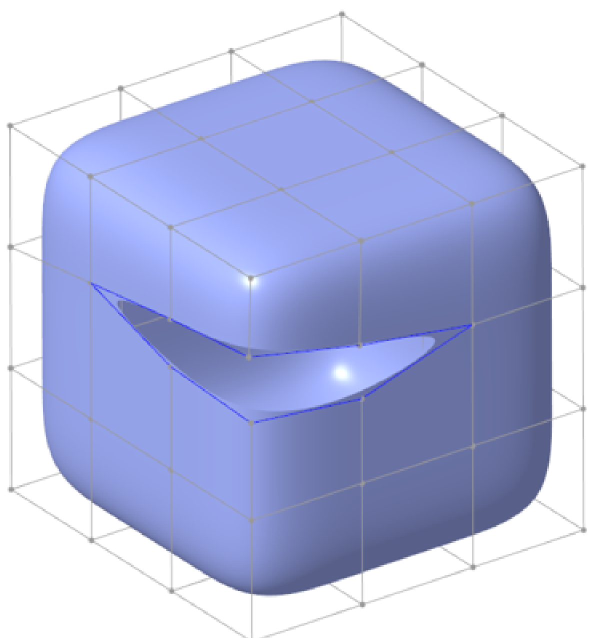
Videos

See the video on the Learning Connector.

Description

You can now use the **Mesh Cut** command to rip a shape open along a chain of edges. Previously, you could cut only closed boundaries of a mesh.





Benefits

This enhancement provides you with a better control over the mesh of a freestyle shape.

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	The **Mesh Cut** command replaces and extends the capabilities of the previous **Mesh Slice** command.
Configuration option associated with this functionality:	None.

Freestyle: Enhanced Resolution Level Usability

Creo Parametric 11.0.0.0

User Interface Location: In the Freestyle Tree, right-click a shape and select any of the resolution levels.

Videos

See the video on the Learning Connector.

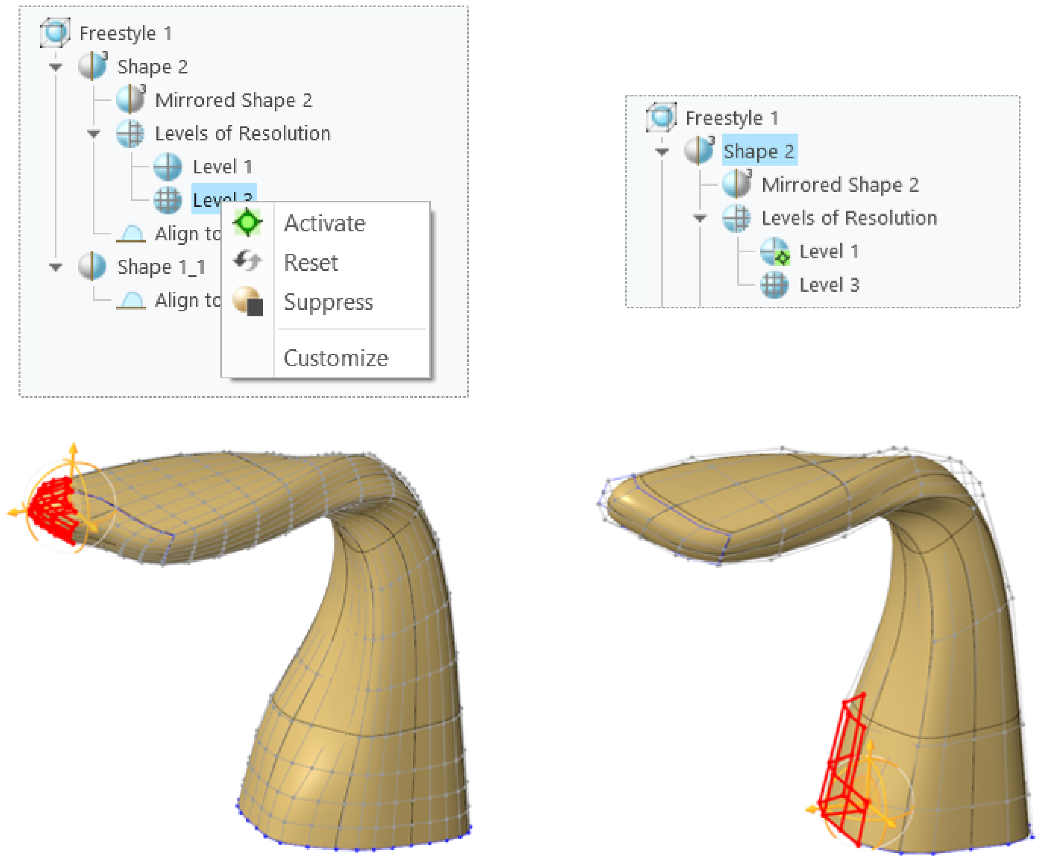
Description

You can now easily access and work with different resolution levels in Freestyle.

With the newly added commands, you can control the resolution without entering any resolution level:

- The **Reset All** command resets all resolution levels.
- The **Suppress All** command suppresses or disables all resolution levels without deleting them.
- The **Reset** and **Suppress** commands reset and suppress that specific resolution level.

The new **Levels of Resolution** subnode under a shape in the Freestyle Tree contains information related to the changes in each resolution level.



Benefits

This enhancement simplifies working with resolution levels.

Additional Information

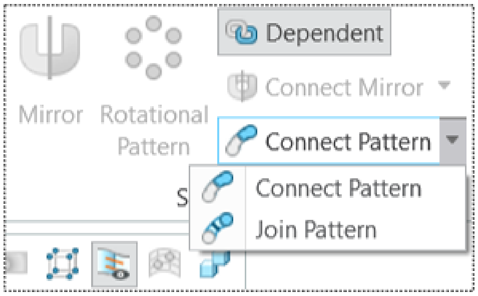
Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Freestyle: Connect Pattern and Join Pattern Commands

Creo Parametric 11.0.0.0

User Interface Location:

- In Freestyle, click **Symmetry** ▸ **Connect Pattern**.
- In Freestyle, click **Symmetry** ▸ **Join Pattern**.



Videos

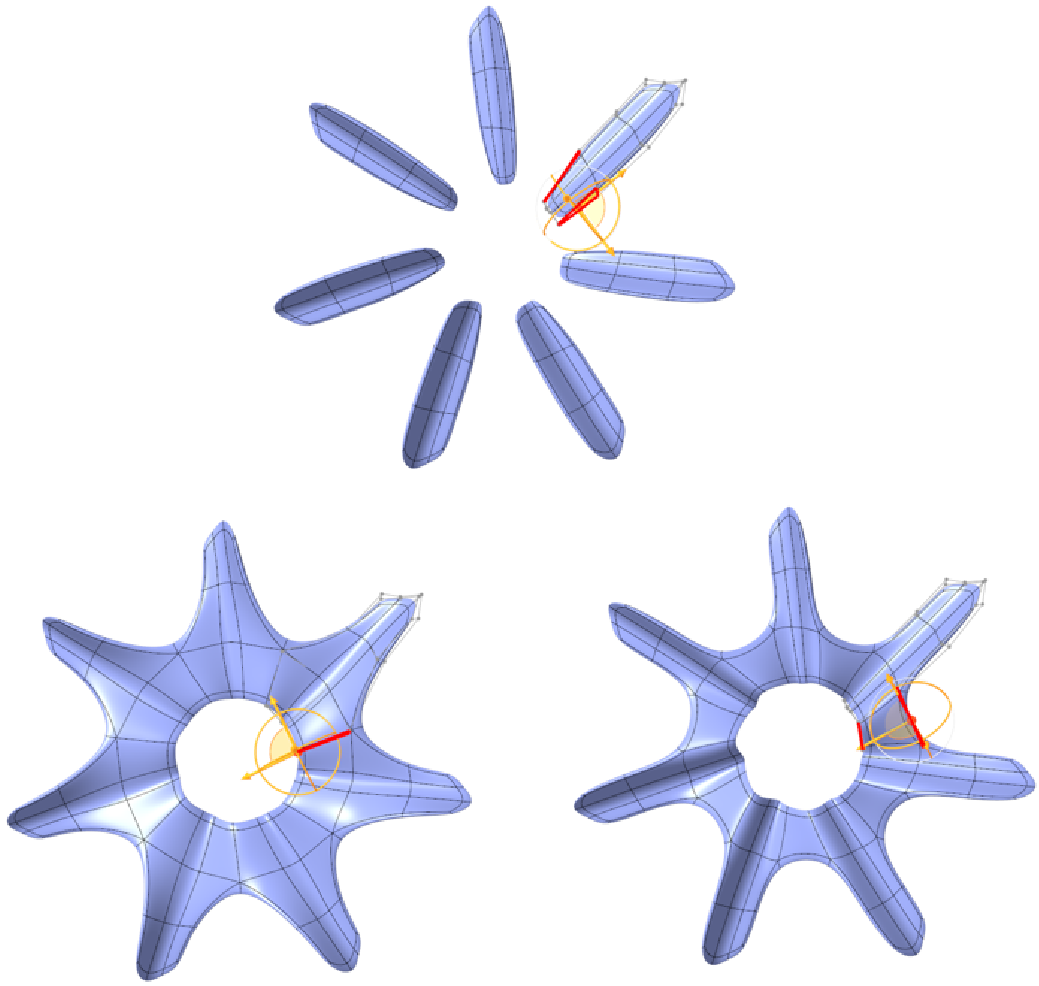
See the video on the Learning Connector.

Description

You can now use the new **Connect Pattern** and **Join Pattern** commands to connect or join distinct rotational pattern shapes and create a single rotationally symmetric shape. The workflows of these commands are similar to the workflows of the **Connect Mirror** and the **Join Mirror** commands.

During the Connect Pattern and Join Pattern operations, the connection is determined by the target elements of the mesh on the leader shape.

The Connect Pattern operation creates a new shape and mesh elements to bridge the gap between two elements, whereas the Join Pattern operation brings the target elements of the mesh together.



Benefits

This enhancement enables you to quickly and easily create a single rotationally symmetric shape.

Additional Information

Tips:	None.
Limitations:	You cannot modify the pattern elements after the pattern is connected. To pattern the leader shape again, you need to delete the connected pattern.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Freestyle: Rotational Pattern as a Reference Pattern

Creo Parametric 11.0.0.0

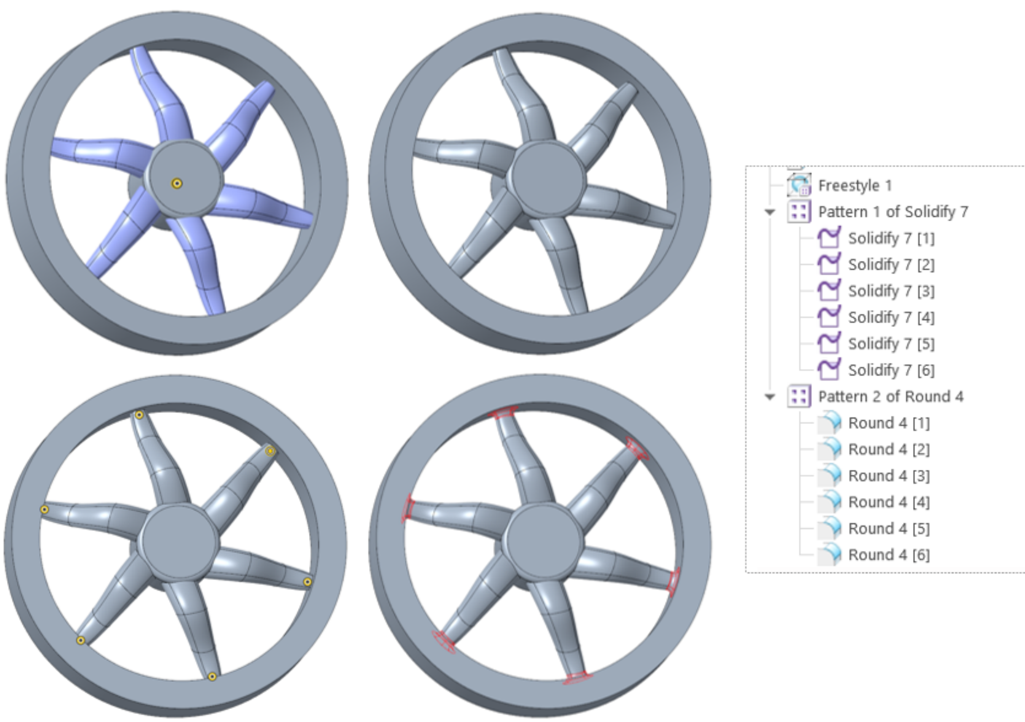
User Interface Location: Click **Pattern ▸ By Reference**.

Videos

See the video on the Learning Connector.

Description

You can now use a rotational pattern defined in Freestyle as a reference pattern outside the Freestyle environment.



Benefits

This enhancement improves the usability of rotational patterns in Creo.

Additional Information

Tips:	None.
Limitations:	No known limitations.

Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

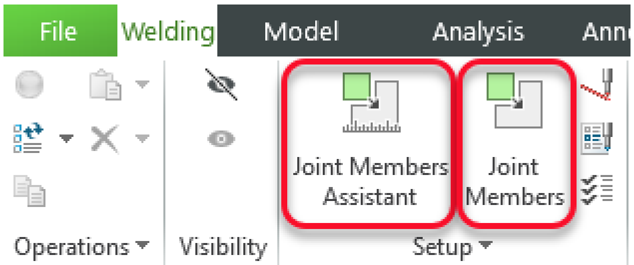
22

Welding

Welding: Joint Members

Creo Parametric 11.0.0.0

User Interface Location: Click **Applications** ▸ **Welding** ▸ **Joint Members** or **Joint Members Assistant**.



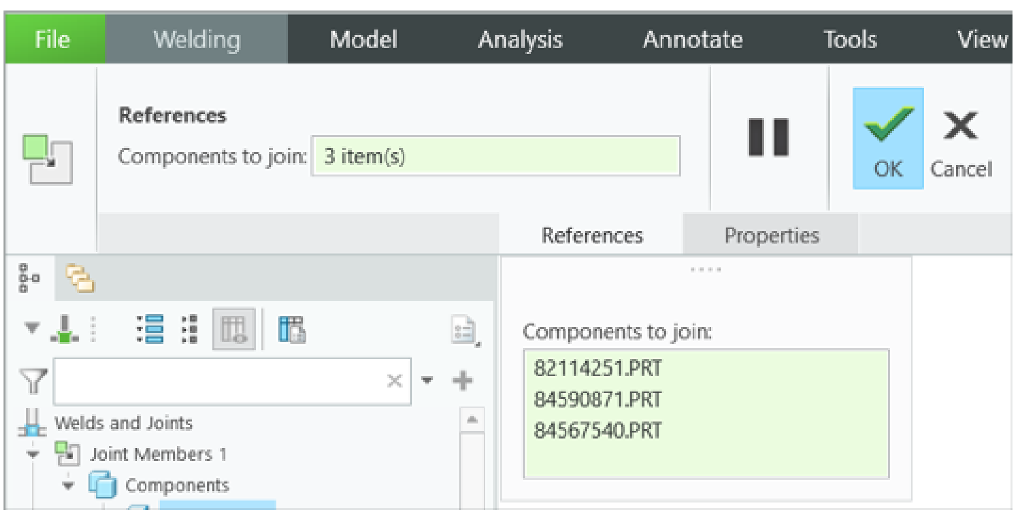
Videos

See the video on the Learning Connector.

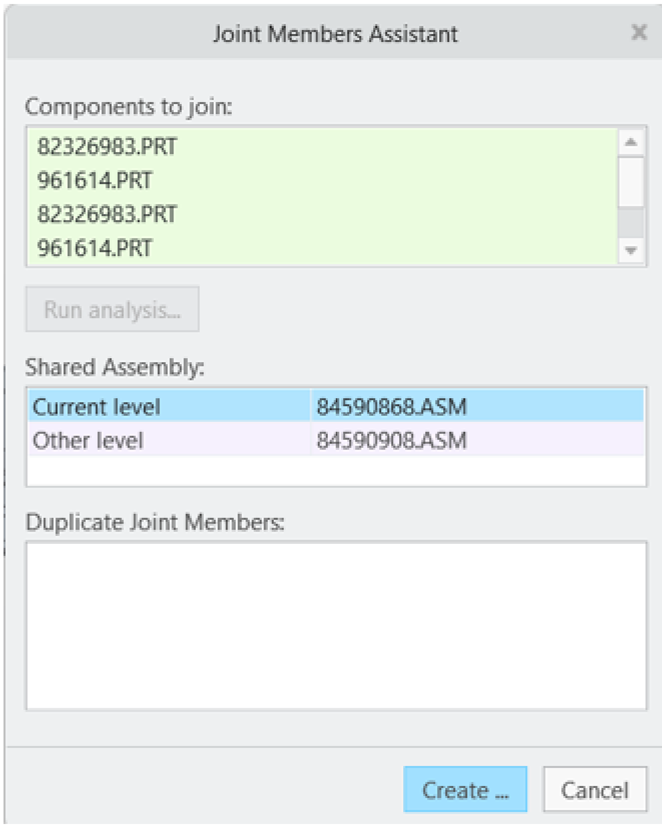
Description

The Joint Members feature has been added to Creo Welding so you can define which components are being joined with Weld features. With complex assemblies and hundreds of welds, it is an advantage to define which components are being joined during the process.

- Manual selection method using **Joint Members**



- Assistant method using **Joint Members Assistant**
  - Top common level assembly
  - Can choose Joint Members feature location



- Activate Joint Members features in the **Weld and Joint Tree**
- Transparent view for active Joint Members
- Optional step in welding process

Benefits

- Clearly visible which parts are being joined
- Supports downstream xMCF standard

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Welding: Spot Weld Enhancements

Creo Parametric 11.0.0.0

User Interface Location: Click **Applications** ▸ **Welding** ▸ **Spot Weld**.

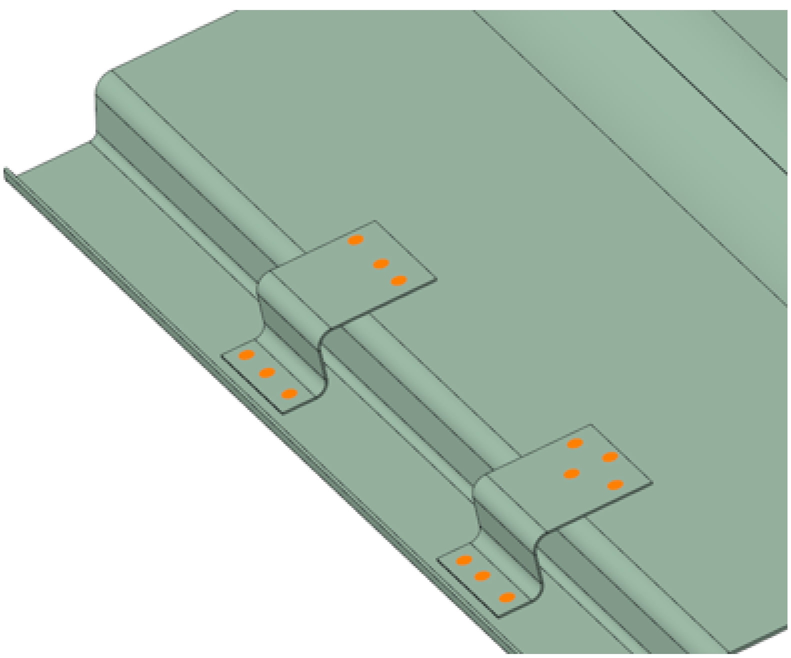
Videos

See the video on the Learning Connector.

Description

The Spot Weld feature has been enhanced to be able to use all datum points to define spot welds, along with some core features which were added to Creo Parametric 11 to support datum point creation. Previously, only datum points created a specific way were available for spot welds.

- All datum points now available for spot weld
  - Sketched, projected, on curve, offset, field, imported, all
- Pattern of points supported
- Additional core features related to Weld
  - Project Sketch supports points
  - Offset Curve



Benefits

- Efficiency in creation of spot welds

- No limitations for how points are created

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Welding: Weld and Joint Tree

Creo Parametric 11.0.0.0

User Interface Location: Click **Applications ▸ Welding**.

Videos

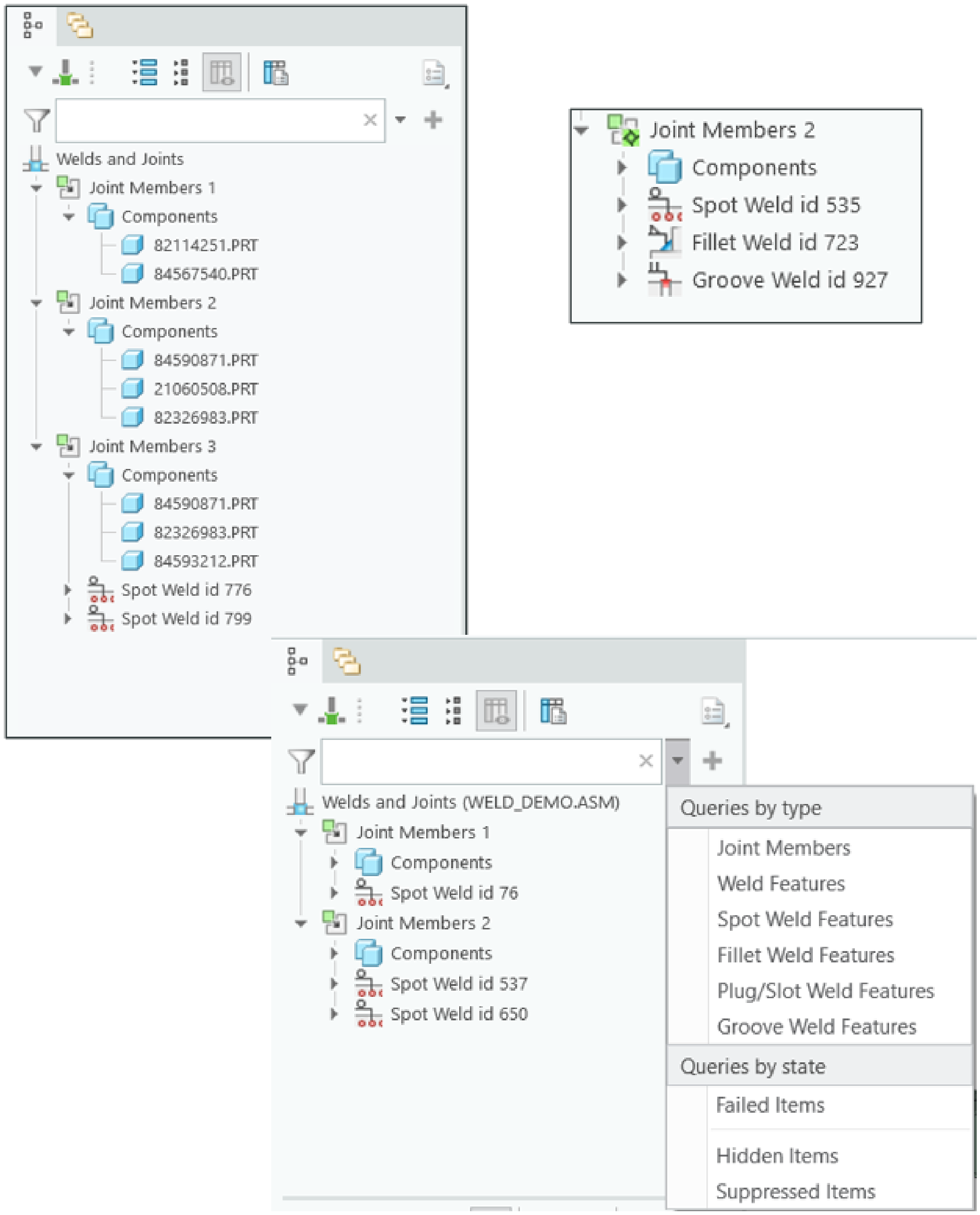
See the video on the Learning Connector.

Description

A new Weld and Joint tree has been added to Creo Welding. It provides an organized view of all Welding features in the model. The tree appears when you enter the Creo Welding environment.

The Weld and Joint tree supports all welding features. It supports all actions, like Insert Here, Suppress, Edit Colors, Parameters, Delete, Blank, Convert, and more. Search and filters are available in the tree.

Previously, welding features only appeared in the Model Tree as features inside parts or assemblies.



Benefits

- Improved organization for all weld features
- Tree that is consistent with other Creo applications
- Right mouse button shortcut menu for quick actions

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.

Welding: xMCF Export

Creo Parametric 11.0.0.0

User Interface Location: Click **Applications** ▸ **Welding** ▸ **Info** ▸ **Export**.

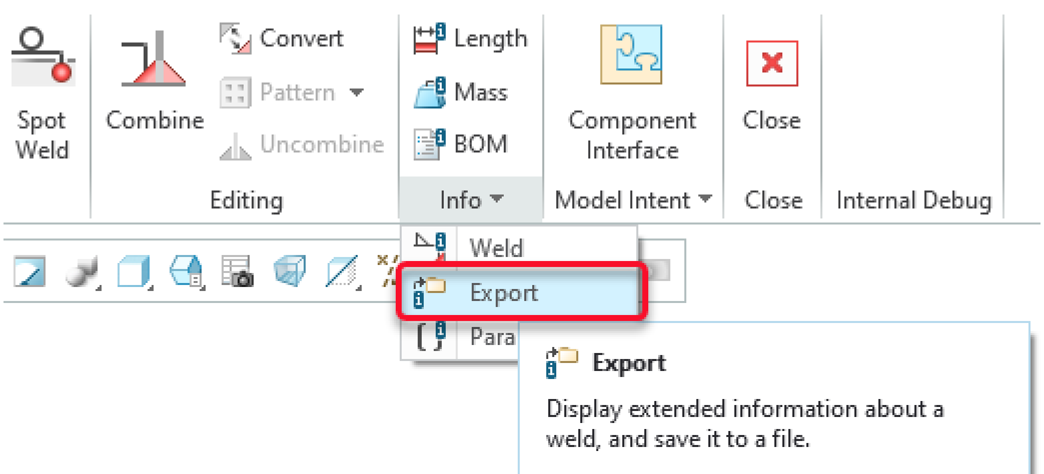
Videos

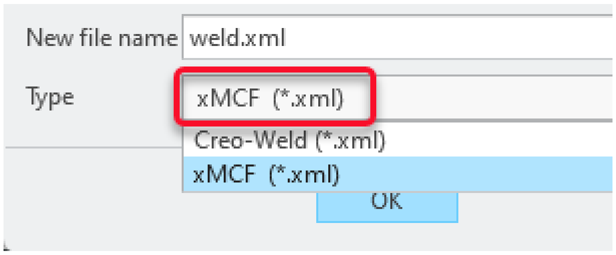
See the video on the Learning Connector.

Description

You can now export from Creo to an xMCF (Extended Master Connection File) file, to be used by downstream processes.

- xMCF support (ISO 10303-242)
- Currently supported for all spot weld features
- Will be expanded to all weld features in Creo Parametric 12.0





Benefits

- ISO standard support
- Can automate processes downstream related to welding

Additional Information

Tips:	None.
Limitations:	No known limitations.
Does this replace existing functionality?	No.
Configuration option associated with this functionality:	None.