



What's New Creo Parametric 5.0

Creo 5.0 Sneak Peek

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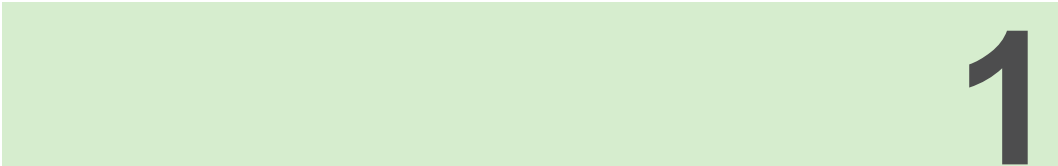
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Installation

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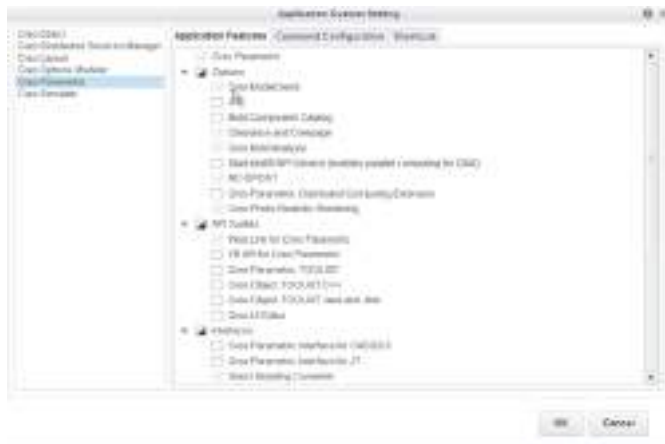
Viewing Tooltips During Installation

Tooltips are available for all options during installation.

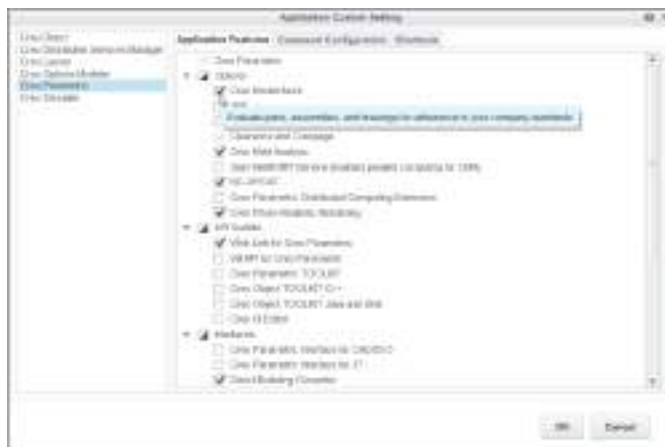
What is the benefit of this enhancement?

When installing Creo applications, all options available during the installation have tooltips to guide you and to help you better configure your installation.

For example, in the **Application Custom Setting** dialog box for Creo 4.0 and earlier releases, there are no tooltips for the options, making it difficult to understand which customization to select.




When installing Creo 5.0, each option has a tooltip to guide you.



Additional Information

Tips:

Place your pointer on an option to see the tooltip. For additional information at any time during the installation, click  to open the Installation Help Center.

Limitations:

No known limitations.

Does this replace existing functionality? No

Configuration options associated with this functionality: None

Silent Uninstall for Creo

When you execute a silent uninstall, you can see the progress of the uninstall.

User Interface Location: From the Creo load point, in the Installmanager directory, click `Silent-Group_uninstall.bat`.

What is the benefit of this enhancement?

The file `Silent_Group_uninstall.bat`, located in the `InstallManager` directory was introduced in Creo 4.0. When the `.bat` file is executed in Creo 4.0, there is nothing to indicate if the process is running successfully. In Creo 5.0 you can see the progress of the uninstall.

When `Silent_Group_uninstall.bat` is executed you can see the progress of the uninstall.



Additional Information

Tips:

Creo Platform services are uninstalled after all components that require these services are uninstalled.

Limitations:

Does not uninstall Creo Schematics

Does this replace existing functionality?:

No

Uninstalling Creo Applications

You can uninstall one or all Creo applications at the same time.

User Interface Location: From the Windows control panel, select **Programs and Features** to open the **Uninstall or change a program** dialog box.

What is the benefit of this enhancement?

In Creo 4.0 and earlier releases each application must be uninstalled separately. The uninstall process can involve completing up to nine uninstall tasks, making the process time consuming. In Creo 5.0 you can uninstall all Creo applications of the same version, at the same time.

In Creo 4.0 and earlier releases, each application must be uninstalled separately.



In Creo 5.0, when you select a Creo application to uninstall, a dialog box opens. From this dialog box, you can select to uninstall only the application you selected or to uninstall all applications and utilities of the same version.



Additional Information

Tips:

Creo Platform services are uninstalled after all components that require these services are uninstalled.

Limitations:

- Does not uninstall Creo Schematics
- You cannot select multiple applications to uninstall. You must select one application or all of them.

Does this replace existing functionality?

No



2

Piping

| | |
|-----------------------------|---|
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Removing Pipe Segments

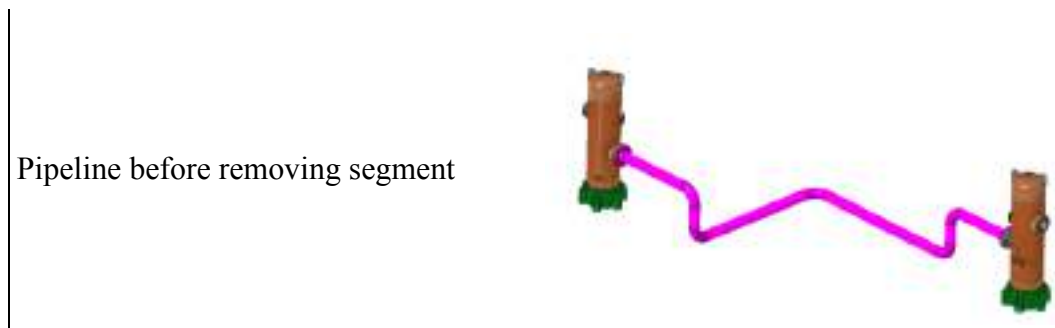
Use the **Remove Pipe Segment** tool to reduce the number of pipe segments, cuts, and welds.

User Interface Location: Click **Tools** ▶ **Remove Pipe Segment**.

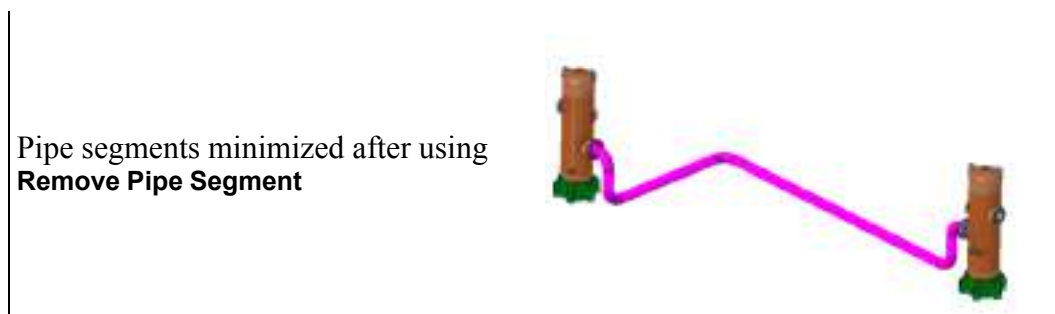
Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

The pipeline in the image below has 12 joints requiring 12 weld operations, 6 pipe cuts, and 6 pipes. After the pipeline is routed and the design is close to final, you may want to minimize the number of pipes, pipe cuts, and welds. This minimization helps to reduce costs, optimize manufacturing time, and reduce part numbers.



When you use the **Remove Pipe Segment** tool, the design is reduced to 4 pipes, 10 welds, and 4 pipe cuts, as you can see in the image below. In addition, you may be able to reduce the count to 2 pipes, 9 welds, and 2 pipe cuts.



Sometimes removing a pipe segment requires the rotation of a fitting to accommodate the change. When it is required, fittings are automatically rotated. Click **Remove Pipe Segment** ▶ **Options** ▶ **Enable fitting rotation** to change the default setting.

Elbows are automatically rotated.



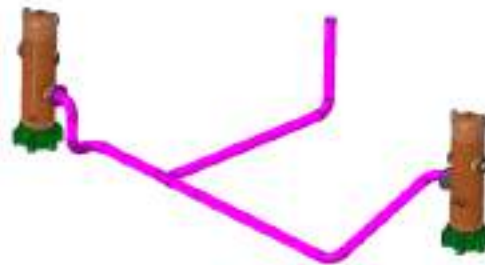
You do not need to completely remove a pipe segment. Click **Remove Pipe Segment** ► **Edit the pipe segment length**.

Partially removed pipe segment

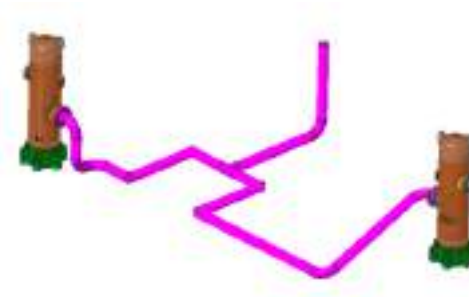


Where there is a break in the pipe, such as when the tee fitting needs to be moved, you can automatically create a connection joint. Click **Remove Pipe Segment** ► **Options** ► **Enable automatic connect**.

Tee fitting prior to move, with pipe break on either side



Tee fitting moved with resulting automatic pipe connection



Additional Information

| | |
|---|--|
| Tips: | When selecting the boundary of the pipe segment that you want modify, press SHIFT and click to define the boundary extents. Alternatively, click Remove Pipe Segment ► References ► Details . |
| Limitations: | You can select only one pipe segment to remove at a time. |
| Does this replace existing functionality? | No. This is a new tool. |
| Configuration options associated with this functionality: | None |

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Assembly Design

| | |
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IFX Supports Inserting Heli-coils

Intelligent Fastener (IFX) supports the insertion of heli-coils.

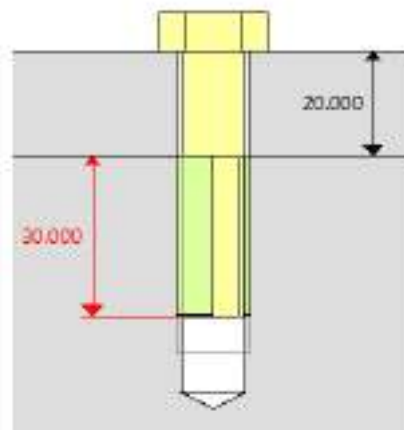
User Interface Location: Open the **Screw Fastener Definition** dialog box.

Watch a video that demonstrates this enhancement:

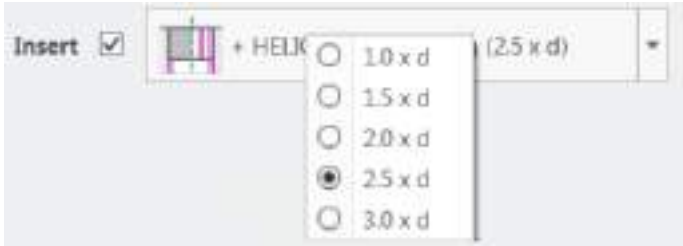
What is the benefit of this enhancement?

A heli-coil inserts provides protection and strengthening for tapped threads in any material. Bolt tensile strength can be balanced against parent material shear strength, ensuring bolt failure rather than damage to the parent material.

30 mm heli-coil preview



Additional Information

| | |
|---|---|
| Tips: | Right-click the heli-coil, to change the heli-coil length. |
| |  |
| Limitations: | None |
| Does this replace existing functionality? | No. This is a new tool. |
| Configuration options associated with this functionality: | hole_parameter_file_path—Specifies the directory path in which to load a standard screw-size parameter (.hol) file. |

IFX Supports Collapsible Lists

IFX supports collapsible lists for fastener selection.

User Interface Location: Open the **Screw Fastener Definition** dialog box.

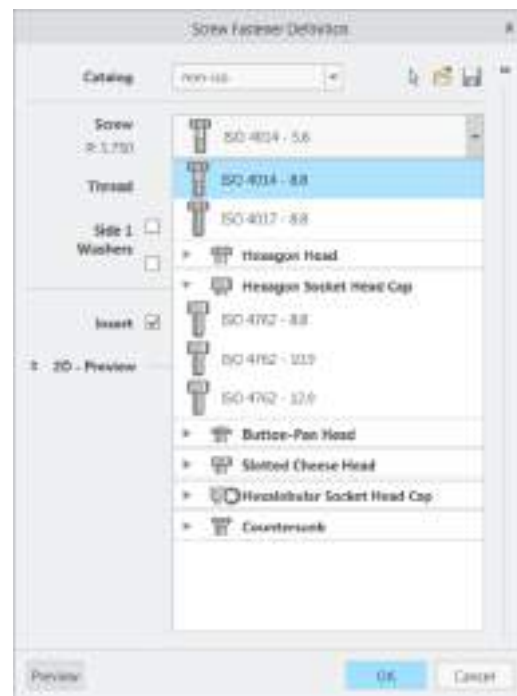
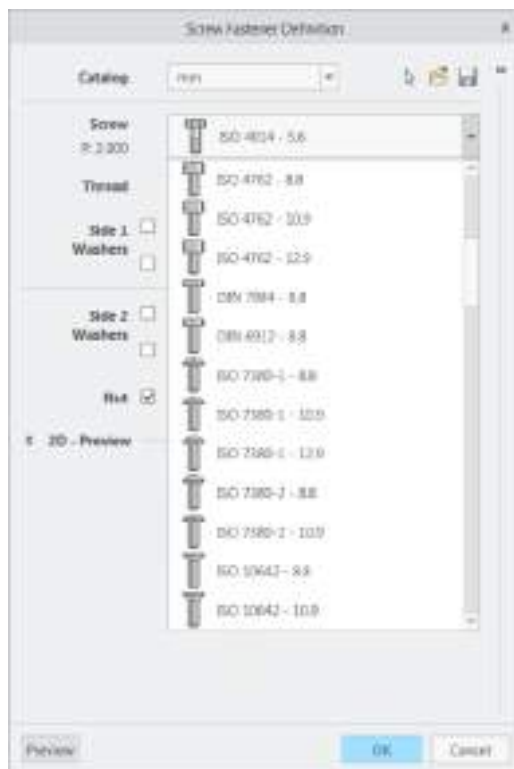
Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

A fastener library catalog can be large, resulting in a need to scroll to find the fastener that you want. With a collapsible list, you can define which lists are collapsed by default and which fasteners appear at the top of the list. This makes access to fasteners quicker and more efficient.

Creo Intelligent Fastener 4.0

Creo Intelligent Fastener 5.0

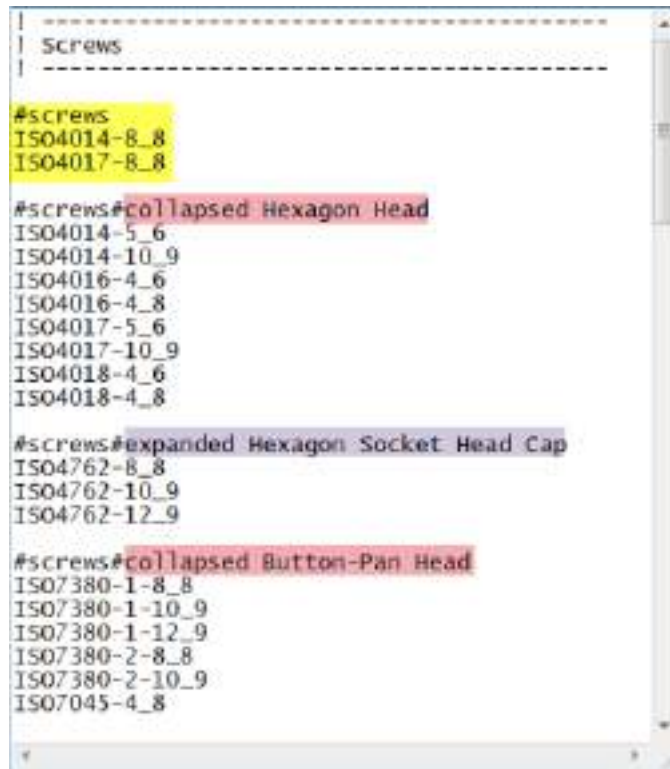


Additional Information

Tips:

From the `ifx_catalogs` directory (`<installation location>\common files\afx\part\ifx_catalogs`) you can open a `catalog.txt` file, such as `mm-iso.txt`, and set the following:

- Favorites. See yellow highlight.
- Groups to be collapsed by default, using the word `collapsed`. See pink highlight.
- Groups to be expanded by default, using the word `expanded`. See purple highlight.



```
| -----  
| Screws  
| -----  
  
#screws  
ISO4014-8_8  
ISO4017-8_8  
  
#screws#collapsed Hexagon Head  
ISO4014-5_6  
ISO4014-10_9  
ISO4016-4_6  
ISO4016-4_8  
ISO4017-5_6  
ISO4017-10_9  
ISO4018-4_6  
ISO4018-4_8  
  
#screws#expanded Hexagon Socket Head Cap  
ISO4762-8_8  
ISO4762-10_9  
ISO4762-12_9  
  
#screws#collapsed Button-Pan Head  
ISO7380-1-8_8  
ISO7380-1-10_9  
ISO7380-1-12_9  
ISO7380-2-8_8  
ISO7380-2-10_9  
ISO7045-4_8
```

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Electrical Design

| | |
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|---|----|

Applying Stripes to Cables and Wires

In Creo Parametric 4.0 and earlier, cables are restricted to one color. In Creo Parametric 5.0 you can create a library of horizontal and vertical stripes to apply to wires and cables. Use the **Cable Stripes** tool to create a library of cable stripes.

User Interface Location:

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

With the increased number of wires in a modern product, when your harness has more than 10 conductors, it is crucial to be able to identify the wires quickly and efficiently. With wire striping, cable identification is easy. Striped wire saves you time when locating wires during installation and maintenance. Striped cable saves you not only time, but also money.

Vertical stripe



Control the spacing from one end.



Make the stripes symmetrical.



Control the width of the stripe.



Control the gap between stripes.



Add multicolored stripes.



Control spacing, width, and start position for each stripe, independently.



Add horizontal stripes.



Additional Information

Tips:

- Stripes are defined in a library which is often controlled by the administrator. You can create your own pattern and submit it to the administrator for approval. If the pattern library file cannot be found, the cables appear in the default color.
- To control whether the section definition is calculated by units or percentage, toggle the percentage icon.

Limitations:

- You can add up to six different stripe sections.
- Helical stripes are not supported.

Does this replace existing functionality?

No. This is a new tool.

Configuration options associated with this functionality:

- `cable_stripes_pattern_file`—Sets the path to the location of the striped appearance `.csv` file.
- `edit_cable_stripes_pattern_file`—When set to `yes`, you can define new patterns and edit and save patterns locally. When set to `no`, you can use patterns created and defined by the administrator.




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Creo Advanced Framework (AFX)

Element Definition User Interface in AFX Is Improved.....21

Element Definition User Interface in AFX Is Improved

In Creo Advanced Framework, the **Element Definition** dialog box is easier to use.

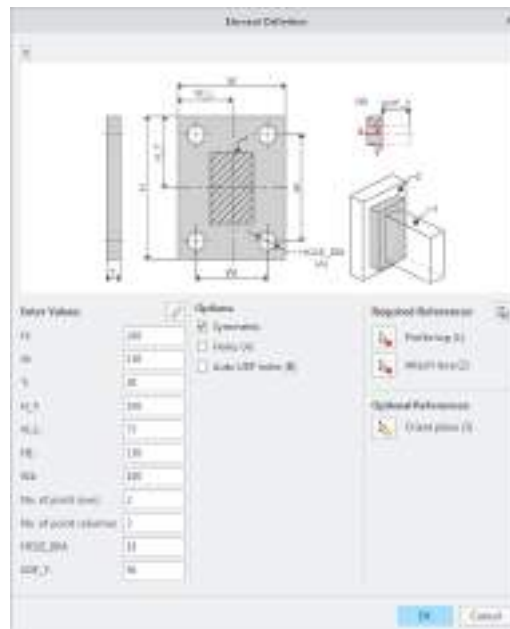
User Interface Location: Open the **Element Definition** dialog box.

Watch a video that demonstrates this enhancement:

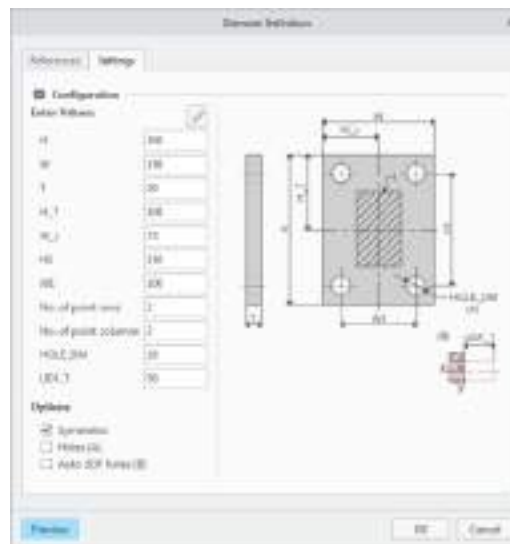
What is the benefit of this enhancement?

In Creo Parametric 4.0 and earlier the **Element Definition** dialog box contains all settings and references in one area and it does not include a preview. In Creo Parametric 5.0 to improve clarity and efficiency, the **Element Definition** dialog box is separated into two tabs, **References** and **Settings**. From this dialog box, you can also preview the proposed changes to the connector element and assess the impact before making the change. This helps to identify potential problems and simplifies the workflow.

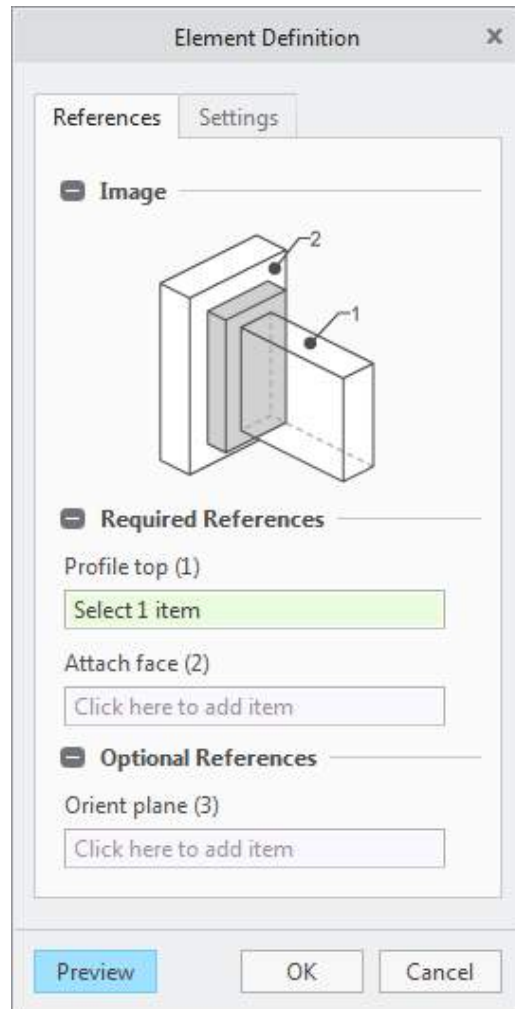
Element Definition dialog box in Creo Parametric 4.0



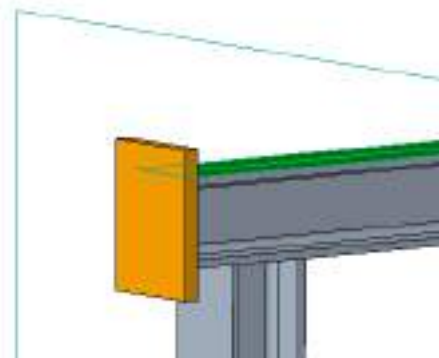
Settings tab in the **Element Definition** dialog box in Creo Parametric 5.0



References tab in the **Element Definition** dialog box in Creo Parametric 5.0



Preview of the connector element with the references highlighted in the **Element Definition** dialog box in Creo Parametric 5.0



6

Surfacing

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Loft Improvements in Style

You can flip the normal direction of a loft surface from one side of a boundary curve to the other side.

User Interface Location: Right-click the normal connection symbol on the loft surface and select **Flip Direction**.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

In Creo Parametric 4.0 and earlier you cannot control the direction of the normal connection of a loft surface where the boundary curve is planar. As a result, the surface may not be created as you intended. In Creo Parametric 5.0, the **Flip Direction** command gives you the needed control and allows you to flip the direction of the normal connection of the loft surface. To flip the loft surface to the other side of the boundary curve, right-click the normal connection symbol on the loft surface and select **Flip Direction**.

Start by creating loft surfaces between the defined curves, such as the two boundary curves you can see in the picture to the right.

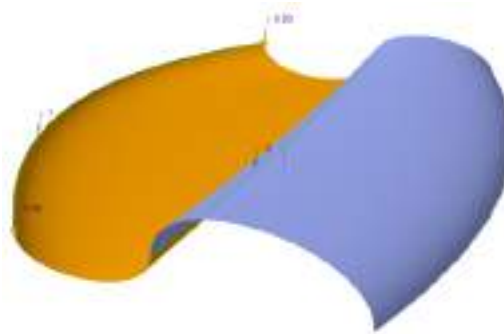
The loft surfaces are created incorrectly as the intent was for them to have a smooth transition.



Flip the loft surface to the other side of the outside boundary curve.



Flip the loft surface to the other side of the middle boundary curve.



Additional Information

| | |
|---|--|
| Tips: | None. |
| Limitations: | The following conditions are required to flip a loft surface: <ul style="list-style-type: none">• You must have a loft surface• The Normal connection type must be available. A normal connection type is a boundary curve with points and endpoint tangents that are coplanar.• The Surface or Surface Connection tab in Style is open. |
| Does this replace existing functionality? | No, this is new functionality. |
| Configuration options associated with this functionality: | None. |

Mini Toolbar in Style

A context-sensitive mini toolbar containing the commands that are relevant to the selected item, is added to Style.

User Interface Location: When you select a curve, surface, or datum, the mini toolbar appears in the graphics window or the Style Tree.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

The mini toolbar appears close to the pointer when you select a curve, surface, or datum. This gives you quick access to commands with fewer clicks. Commands on the mini toolbar vary, based on the type of item you select.

You can access commands on the mini toolbar from either the graphics window or the Style Tree. For the selected item, the same mini toolbar appears regardless of where you select the item. You can suit your specific requirements by customizing the commands on the toolbar. When the mini toolbar appears, a shortcut menu also opens. From the shortcut menu click **Customize**.

See examples of mini toolbars for Style below:



Additional Information

| | |
|---|--|
| Tips: | <ul style="list-style-type: none"> Some commonly-used commands, such as Edit Definition and Suppress, are moved from the shortcut (right-click) menu to the mini toolbar. For most commands, the mini toolbar provides an additional method of accessing commands that are also on the tabs. |
| Limitations: | None. |
| Does this replace existing functionality? | The mini toolbar is new for Style. |
| Configuration options associated with this functionality: | None. |

Slice of Shapes by Plane Is Available in Freestyle

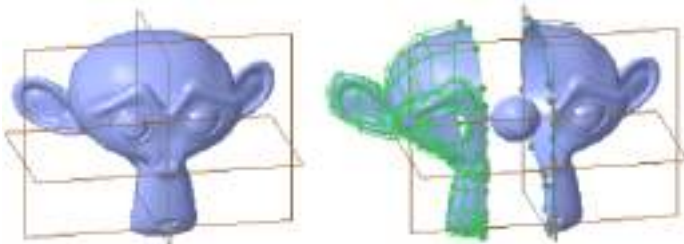
Use the **Mesh Slice** command to slice shapes by a datum plane.

User Interface Location:

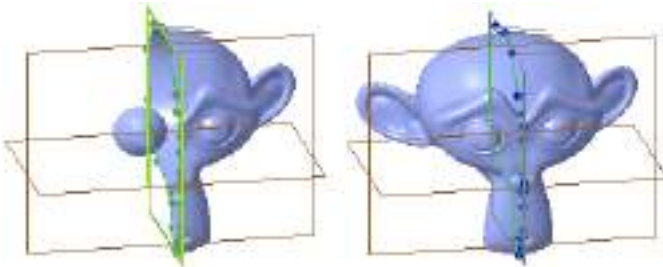
Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

In addition to the existing mesh slicing functionality, you can slice one or more shapes by an existing datum plane or by creating a datum plane on the fly. The selected plane must intersect with the shapes that you want to slice.



The imported datum objects can be made symmetrical using the **Mesh Slice** and **Mirror** commands. After slicing, delete the imperfect sliced shape, and then mirror the perfect sliced shape to achieve symmetry.



Additional Information

| | |
|--------------|---|
| Tips: | You can select shapes on the Freestyle Tree or on the graphics window. Select Shape in the search filter to select shapes on the graphics window. You can select multiple shapes for slicing. |
| Limitations: | <ul style="list-style-type: none">• The hidden shapes are not sliced.• Multilevel changes are lost while slicing a shape by plane. You are prompted when such a conflict occurs. |

| | |
|---|---|
| Does this replace existing functionality? | No. You can use the Mesh Slice command for all slicing operations. |
| Configuration options associated with this functionality: | None |

Preview of Imported Objects in Freestyle

Preview the objects before importing them into Freestyle.

Watch a video that demonstrates this enhancement:

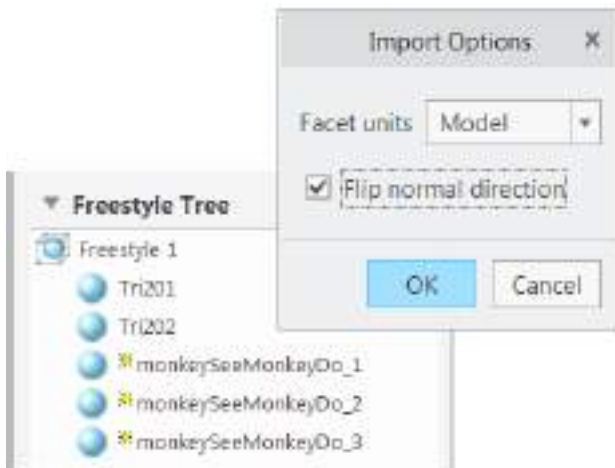
What is the benefit of this enhancement?

You can preview the object before importing OBJ files in Freestyle. In addition, in the **Import Options** dialog box, you can define the following properties:

- **Facet units**—Select the facet units from Model, Inch, Foot, Millimeter, Centimeter, and Meter.
- **Flip normal direction**—Flip the direction of the face normals.



The arrows pointing outside of the mesh indicate the direction of the face normals. You can flip the direction by selecting the **Flip normal direction** check box. The Freestyle Tree lists all shapes in the OBJ file being imported, with an indicator to distinguish them from other shapes. You can also rename these shapes before importing the object file.



Additional Information

| | |
|---|--|
| Tips: | If you do not want to proceed with the object file, click Cancel in the Import Options dialog box. |
| Limitations: | None |
| Does this replace existing functionality? | No. This is new functionality. |
| Configuration options associated with this functionality: | None |

Freestyle Surfacing in Box Mode Is Available

Toggle between standard and box modes to rapidly design your freestyle surfaces.

User Interface Location: In-graphics toolbar

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

In Box mode, the control mesh is shaded and not the actual B-spline surface. Along with the B-spline surfacing, the Box mode surfacing helps you rapidly create the features, when used effectively.

Surfaces are generated faster in Box mode than the standard mode. As a result, you do not see any lag in surface creation when you manipulate the control mesh. This allows you to efficiently perform complex manipulations.



A new icon in the in-graphics toolbar helps you toggle between the modes.



Additional Information

| | |
|---|--|
| Tips: | If you modify the crease of a mesh in Box mode, toggle back to standard mode to view the modified mesh |
| Limitations: | None. You can perform all surface operations in both the modes. |
| Does this replace existing functionality? | No. You can toggle between the modes and choose the appropriate mode when required. |
| Configuration options associated with this functionality: | None |

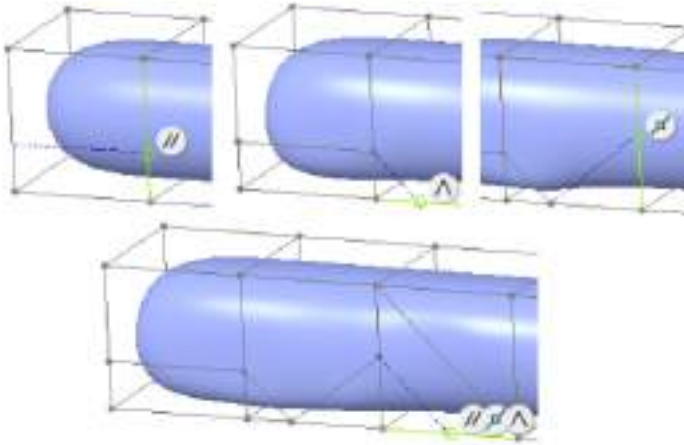
Snapping When Adding Edges Is Improved in Freestyle

With the help of snapping references, you can create accurate mesh in Freestyle.




Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

When adding a new edge, the target edge on which you place the pointer, is highlighted. You can select any point on the target edge to complete the new edge. The snapping references help you to select the precise points on the target edge.



If the point to be selected satisfies any of the following conditions, one or all three of the following unique snapping references are displayed:

| | |
|---|---|
|  | Indicates the center point on the target edge. |
|  | Indicates that the new edge is parallel to an adjacent edge on the same face. |
|  | Indicates that the new edge completes an isosceles triangle on the same face. |

Additional Information

| | |
|---|---|
| Tips: | Follow the action-object method to view the snapping references on the initial or the leading edge that you select to add a new edge. |
| Limitations: | There are no known limitations. |
| Does this replace existing functionality? | No. This enhancement improves the current functionality by providing you more control over selecting points when adding edges. |
| Configuration options associated with this functionality: | None |

Align Curvature Is Available in Freestyle

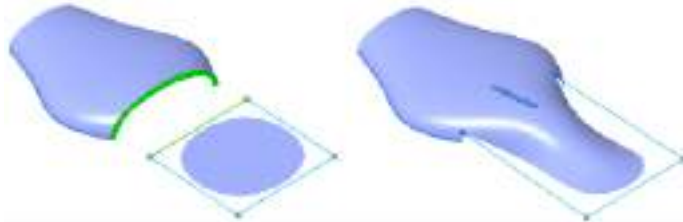
Use **Align Curvature** to align shapes without losing the curvature continuity.

User Interface Location: Click **Freestyle** ► **Align** ► **Align Curvature**.

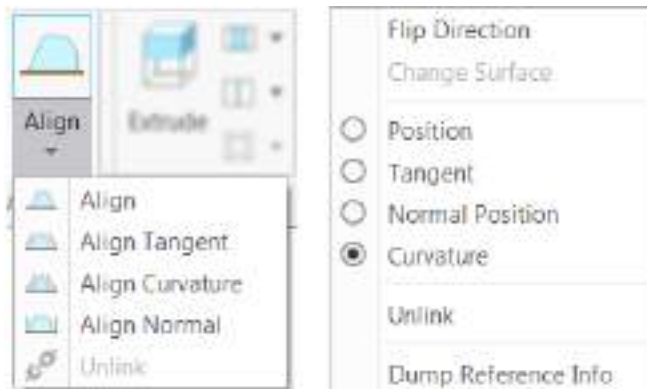
Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

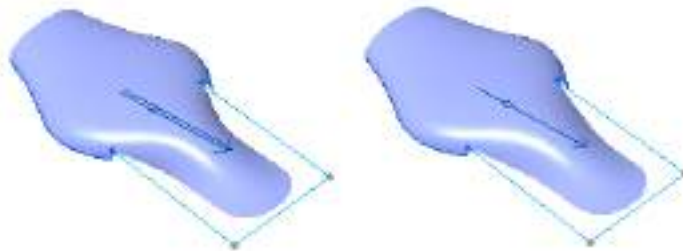
Use **Align Curvature** to maintain the curvature continuity of the aligned shapes.



Click **Align Curvature** when aligning surfaces or right-click in the graphics window and select **Curvature** to change the noncurvature connection to a curvature connection.



For every surface connection there is a unique glyph or connection icon. Click the icon to flip the direction of the connection.



Additional Information

| | |
|--------------|----------------------|
| Tips: | None |
| Limitations: | No known limitations |

| | |
|---|----------------------------------|
| Does this replace existing functionality? | No. This improves functionality. |
| Configuration options associated with this functionality: | None |

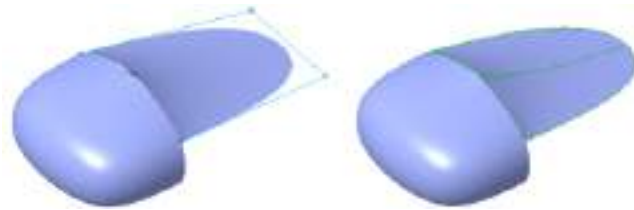
Align to Non-G2 Chains in Freestyle

Use the **Align** command to align **Freestyle** edges to external curves or edges with G0, G1, G2, or G3 connections.

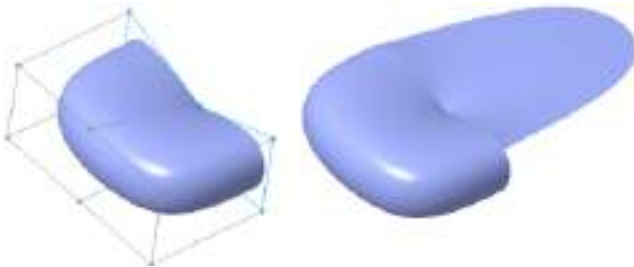
Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

In **Freestyle**, you can align one-sided open edges to chains of curves or edges of an external geometry with G0 or G1 surface connections. You can view the resulting geometry only after completing the **Freestyle** feature.



If you modify the external geometry, the features are redefined and regenerated.



Additional Information

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| Tips: | You can view the final geometry only after completing the Freestyle feature. |
| Limitations: | No known limitations |

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| Does this replace existing functionality? | This improves the existing functionality by allowing you to align to curves with non-G2 surface connections |
| Configuration options associated with this functionality: | None |


7

Part Modeling

| | |
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New Sweep Tool

Use the new volume sweep and helical sweep tools to create accurate 3D geometric representation for parts that are machined with cutting tools.

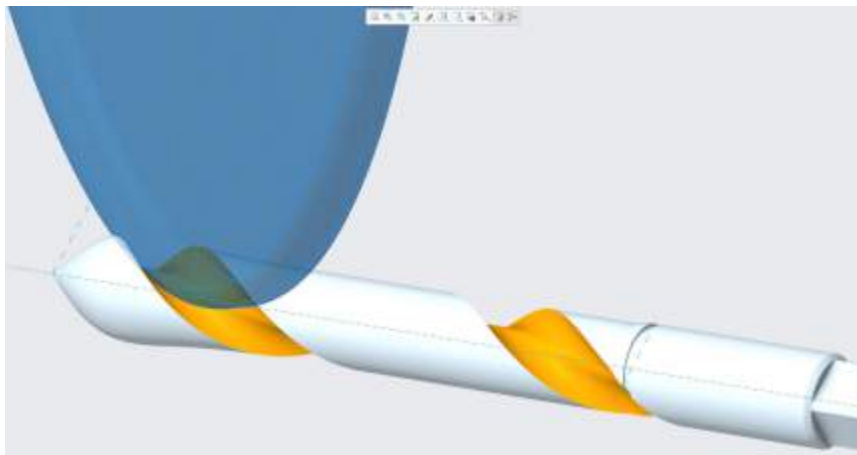
User Interface Location: Click **Model** ► **Sweep** ► **Volume Helical Sweep**.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

Creo Parametric 5.0 broadens sweep functionality beyond the 2D sweep operations that are available in Creo Parametric 4.0 and earlier, by introducing **Volume Helical Sweep**. You can accurately model geometry resulting from cutting tools used in manufacturing operations. Supported use cases cover the grinding wheel and the screw conveyor scenarios where, in both cases, the sweep is performed along a helical curve. In both cases, the cutting tool is represented by an axis-symmetric shape defined by a revolved section. Material is removed where it intersects the part along its trajectory. From the **Volume Helical Sweep** dashboard, you can easily define and control the parameters of the helical trajectory. You can also easily select, create, or edit the 2D section that defines the cutting tool, and control available dimensions.

Volume Helical Sweep



From the **Volume Helical Sweep** tab, you can turn on the display of the sweep frame and revolved cutting tool. Drag the 3D cutting tool visualization along the sweep trajectory to easily visualize and quickly understand its movement and orientation. Unattached and attached previews are available. You can use the previews to visualize the calculated tool path envelope geometry subtracted from the part as well as the resulting geometry. With the new **Volume Helical Sweep** capabilities in Creo Parametric 5.0, you can model geometry that could previously only be modeled inaccurately. As a result, there were often downstream issues.

Additional Information

Tips:

- You can also use **Volume Helical Sweep** in a screw conveyor scenario. Define a helix with constant or variable helix pitch and apply an adjustment angle of $RX=90$ degrees. You can see the results in the example below:



- You can use two workflows to define the section of the cutting tool.
 - Define it within the **Volume Helical Sweep** feature.
 - Reference an existing sketched section and specify origin and rotation axis. This automatically transforms the referenced section in 3D space to conform with the overall feature setup such as for the trajectory, adjustment angles and so on.

Limitations:

- **Volume Helical Sweep** only provides geometric operations to remove material. You cannot create or add material.
- The section that defines the cutting tool may only contain lines and arcs forming a convex shape.
- Geometric conditions leading to self-intersections of the calculated envelope inside the cutting area are not supported as described in the following cases:
 - When trajectory radius is smaller than the tool radius
 - For setup and geometry-dependent ranges of adjustment angles
- The success of the geometric operation may be

| | |
|---|--|
| | dependent on part accuracy. It is recommended to use absolute accuracy with values ranging between 0.01mm and 0.001mm. |
| Does this replace existing functionality? | No. |
| Configuration options associated with this functionality: | None. |

Using Sketch Regions

You can use **Sketch Region** selection to quickly create geometry with selected sketch-based features.

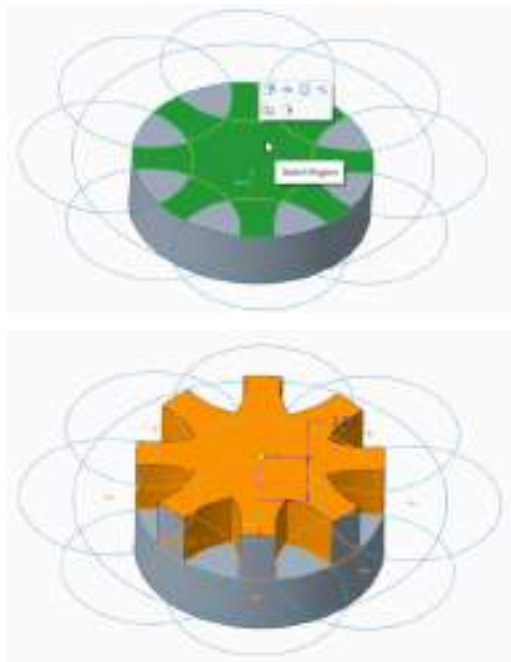
User Interface Location: Set the selection filter to **Sketch Regions**.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

In Creo Parametric 5.0 there is a new workflow to quickly create geometry with selected sketch-based features based on **Sketch Region** selection. A **Sketch Region** is a closed contour defined by sketched entities and their intersection with coplanar 3D edges in the part geometry. Sketch-based feature geometry creation is faster and easier. Using **Sketch Region** reduces the need to perform **Project** and trim operations within Sketcher. It also offers a flexible way to use portions of a single sketch as the basis for several sketch-based features.

Set the selection filter to **Sketch Region** to quickly select one or more sketch regions. 2D box-selection is available for sketch regions. Sketched entities that are involved can belong to different sketches of the same model. After you make your selection, a context-sensitive mini toolbar provides direct access to the features that support **Sketch Region** input: **Extrude**, **Revolve**, **Fill**, and **Sketch**. Sketch regions are supported only for object–action workflow.



Additional Information

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| Tips: | <ul style="list-style-type: none"> • Sketch regions provide a faster way to create sketch-based features without the need to prepare the sketch with Project and trimming operations inside sketcher. • Press SHIFT+S to switch to the Sketch Region selection filter. • Press SHIFT+G to switch back to the Geometry selection filters. |
| Limitations: | Sketch Region is currently supported in object–action workflows only. Sketch regions do not represent objects that can be regenerated. Sketch regions provide a faster way to create sketch-based features without the need to prepare the sketch with Project and trimming operations inside sketcher. |
| Does this replace existing functionality? | No |
| Configuration options associated with this functionality: | None |

Round Handling in Draft

You can easily apply drafts to design models containing rounds and chamfers.

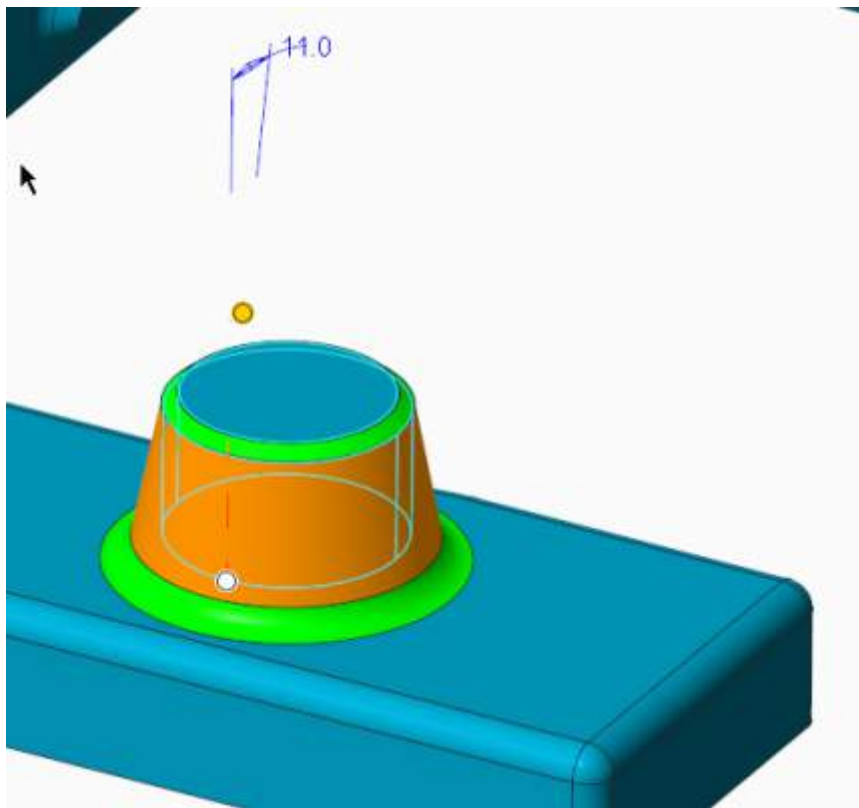
User Interface Location: Click **Model** ► **Draft**.


Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

You can easily apply drafts to design models containing rounds and chamfers. This addresses the difficulty in Creo Parametric 4.0 and earlier releases in applying drafts to rounded-base part models from the design department or to imported models containing rounds and chamfers.

You can select surfaces as references in the **Draft surfaces** collector of the draft feature even if they have adjacent rounds or chamfers. Rounds and chamfers are automatically detected and highlighted in a different color. Rounds and chamfers are then handled as they are in the Creo Parametric Flexible Modeling environment. For example, they are implicitly removed before and recreated after the geometric modification. This allows you to apply drafts to models that already contain rounds or chamfers at the boundaries of the to-be-drafted surfaces.








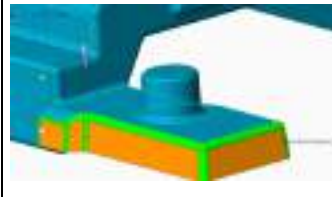
This enhancement increases productivity through faster creation of drafted surfaces having adjacent rounds or chamfers.


Additional Information

Tips:

- If the geometry selection contains inlying rounds, you can control their inclusion or exclusion from the draft operation by selecting  on the **Draft** tab. It is typically faster and easier to include the inlying rounds in the selection first, and then to unselect  to exclude them from the drafted geometry.

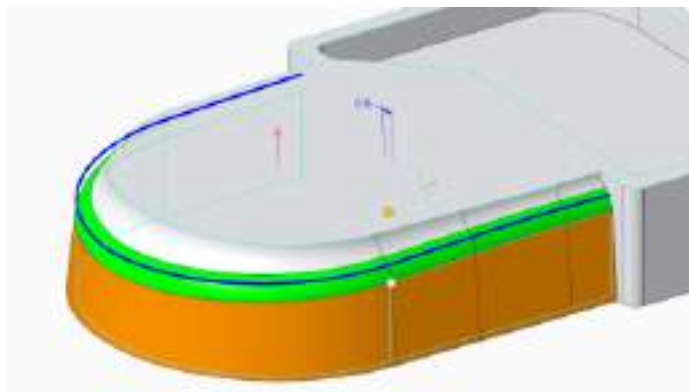
Select  on the **Draft** tab to exclude inlying rounds. Inlying rounds then appear in green and are treated as rounds.



Click to unselect  on the **Draft** tab to include inlying rounds. Inlying rounds appear in orange, the modified geometry color, and are included in the draft.



- In addition, you can specify a round surface chain as a hinge for the draft operation. This can be helpful for geometric situations such as in the example below:



- **Options** on the **Draft** tab also includes the **Create round/chamfer geometry** check box. When selected, rounds and chamfers are recreated after the draft operation. When this check box is cleared, rounds and chamfers are removed.

| | |
|---|---|
| Limitations: | <ul style="list-style-type: none"> This enhancement enables the handling of rounds and chamfers based on previously available capabilities in Draft. It does not extend the general capabilities of the draft feature, such as to draft previously drafted surfaces. This enhancement only supports round and chamfer types that are currently supported within Creo Parametric Flexible Modeling. |
| Does this replace existing functionality? | Draft tangent surfaces moves from under Options to the Draft tab. Draft features created in Creo Parametric 4.0 and earlier maintain legacy regeneration and user interface. |
| Configuration options associated with this functionality: | <ul style="list-style-type: none"> <code>draft_tan_propagation_default</code>—Determines if draft is automatically propagated along tangent surfaces. Values are <i>yes</i> or <i>no</i>. The default is <i>yes</i>. <code>draft_preserve_inlying_rounds</code>—Determines if inlying round and chamfer surfaces are preserved and not to be drafted. Values are <i>yes</i> or <i>no</i>. The default is <i>no</i>. |

Point Pattern Workflow Is Improved

Point pattern workflows are improved when the definition of an alternate origin is required.

User Interface Location: Click **Model** ► **Pattern**.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

When the definition of an alternate origin is required, point pattern workflows for feature patterns are improved, such as in the examples listed below:

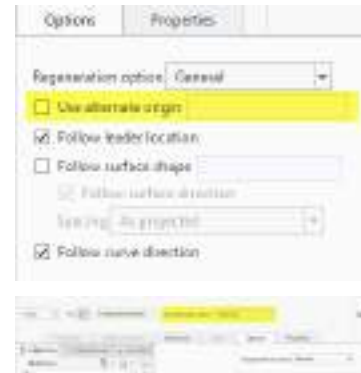
- Point pattern of threaded hole features on a slanted surface referencing a point of the point pattern array
- Point pattern of sketch-based features referencing a point of the point pattern array

When you specify the pattern point array during pattern definition, typical alternate origins are automatically detected. In cases where a point in the point array is referenced by the pattern leader, this point is automatically set as the alternate origin reference. The **Use alternate origin** collector appears on the **Pattern** tab. Where autodetection applies, you can see that the **Use alternate origin**

collector fills automatically. This enhancement provides a smarter default solution for point pattern workflows and helps you to correctly define the point pattern feature.

Additional Information

| | |
|---|---|
| Tips: | To learn more about the background of this enhancement and previous issues addressed with this enhancement, see Support article CS152954 . |
| Limitations: | This workflow only applies to point pattern of features. You may need to explicitly set the Use alternate origin collector for a geometry pattern and a flexible pattern where required. There are no feature references to the point array. |
| Does this replace existing functionality? | Creo Parametric 4.0 Pattern user interface. Creo Parametric 5.0 Pattern user interface. The Use alternate origin collector is no longer under Options . |
| Configuration options associated with this functionality: | None. |



Mirror Is Enhanced

The Mirror workflow provides an intuitive preview and increased flexibility during redefinition.

User Interface Location: Click **Model** ► **Mirror**.

Watch a video that demonstrates this enhancement:

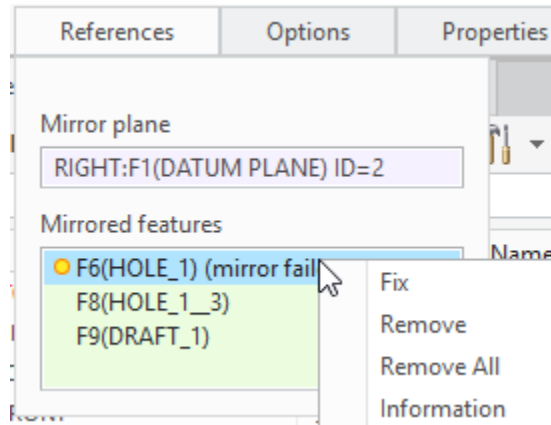
What is the benefit of this enhancement?

Enhancements to the **Mirror** tool in Creo Parametric 5.0 result in faster definition and redefinition of the **Mirror** feature. A preview in the graphics window is provided during feature creation or redefinition for immediate visual feedback on the geometry. In addition, a feature collector is added which alerts you if source features are missing. When you select the **Reapply Mirror** check box, missing features are removed from the definition. Furthermore, the feature collector gives you the freedom to add or remove features when reapplying the mirror operation.

Additional Information

Tips:

- You can easily add or remove features from the mirror operation.
 - Select the **Reapply Mirror** check box on the **Mirror** tab. With **Mirrored features** active, press CTRL and select features from the graphics window to be removed or added.
 - If source features are deleted, they are not included in the **Mirrored features** list.
 - If target features are previously modified, modifications are lost.
- When references of mirrored features need to be redefined, they are visible in the feature collector. See the example below:



- From the shortcut menu, use **Fix** to access the corresponding feature to replace references where needed.
- It is recommended to first add and remove features to the **Mirrored features** collector before using **Fix**.

Limitations:

This workflow enhancement is only available in Creo Parametric part mode and is not supported in combination with the option **Fully dependent with options to vary**.

Does this replace existing functionality?

No, this is new functionality for **Mirror**.

Configuration options associated with this functionality:

None




Model-Based Definition

| | |
|--|----|
| Mini Toolbars for 3D Annotations | 49 |
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| Enhanced Workflow for Radial Dimensions | 57 |

Mini Toolbars for 3D Annotations

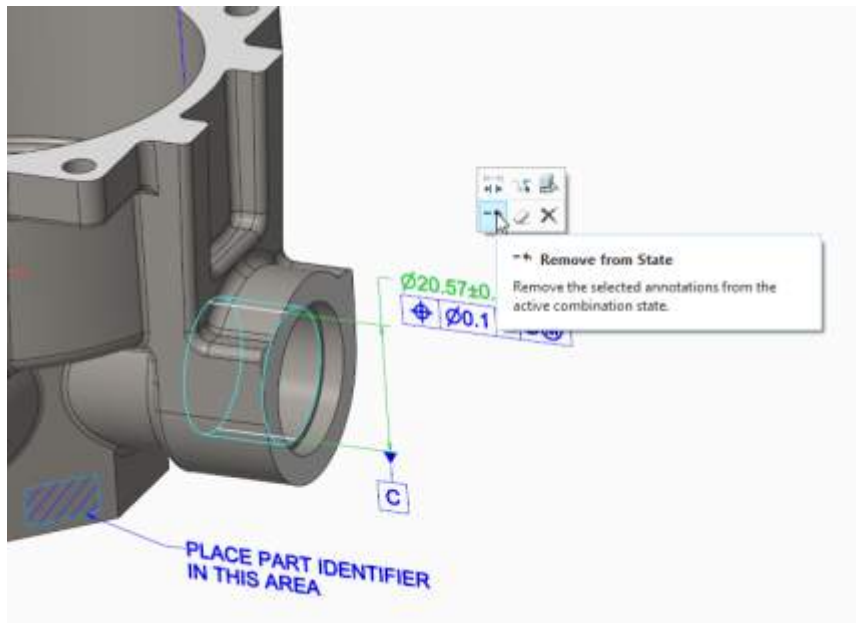
Mini Toolbars are introduced for 2D and 3D Annotations.

User Interface Location: Click the **Annotate** tab.

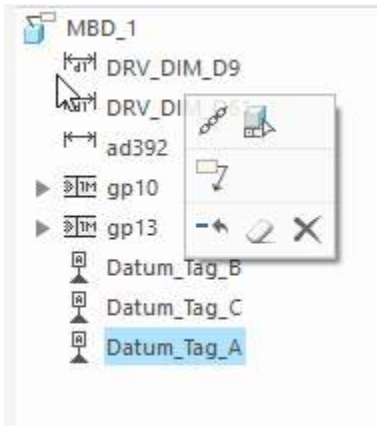
Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

Mini toolbar support in Creo Parametric is broadened. There is full support for mini toolbars in 3D annotations. As a result, there is faster and more intuitive command access in object-action workflows. Click a 3D annotation in the graphics area to see the corresponding context-sensitive mini toolbar. You can then access available actions without scrolling through the shortcut menu. When you right-click an annotation both the mini toolbar and the shortcut menu appear.



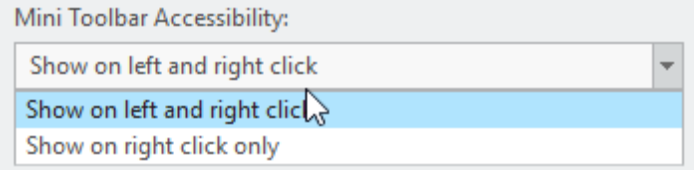
Annotation mini toolbars are also available when you select annotations in the Model Tree or Detail Tree.



You can customize the mini toolbar in either of the ways described below:

- Right-click an annotation and select **Customize** from the shortcut menu. Drag commands to one of the four rows of the **Mini Toolbar – Annotation** dialog box to add commands to the mini toolbar. You can also drag commands off the mini toolbar.
- Click **File ► Options ► Customize ► Shortcut Menus** and then select one of the Annotation names.

Additional Information

| | |
|---------------------|---|
| <p>Tips:</p> | <p>To customize the accessibility of mini toolbars, click File ► Options ► Customize ► Shortcut menus</p>  <p>Note</p> <p>Some commands are moved from the shortcut (right-click) menu to the mini toolbar.</p> |
| <p>Limitations:</p> | <p>If you customize the mini toolbar for annotations in the standard application, the mini toolbar for annotations in other applications, such as Welding, does not change. However, if you customize the mini toolbar for annotations in Welding, for example, the mini toolbar for the standard application also changes.</p> |

| | |
|---|--|
| Does this replace existing functionality? | This is new functionality for Model-Based Definition 3D annotations. |
| Configuration options associated with this functionality: | None |

Semantic Query in Model-Based Definition

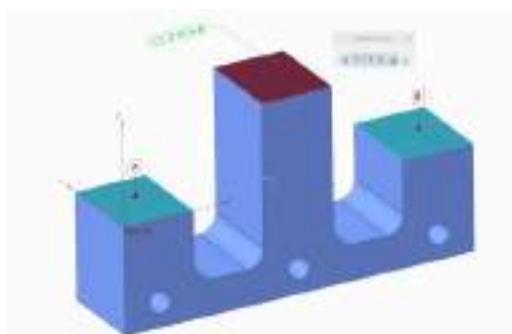
You can query models per ASME and ISO standards and analyze semantic information related to annotations.

User Interface Location: **Semantic Query** is available in the **Query** group of the **Annotate** tab. **Semantic Query** is also available from the mini toolbar for annotations.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?


Semantic Query is a special analysis mode that helps you to obtain the semantic information related to annotations. Click **Annotation** ► **Semantic Query** to open **Semantic Query** mode. In this mode, you can query annotations and analyze the complex relationships between the annotations and the geometry in a Model-Based Definition (MBD) environment.



Query-Response Options

- **Annotation > Geometry**
 - Semantic references for selected annotation are highlighted with shading
- **Geometry > Annotation**
 - All associated annotations are highlighted for selected surfaces
- **GTOL > Datum Feature Symbol and Datum Target**
 - All associated datum symbols are highlighted for selected GTOL
- **GTOL > DRF Coordinate System**
 - Associated DRF is highlighted for selected GTOL
- **Datum Feature Symbol and Datum Target > GTOL**
 - All associated GTOLs are highlighted for selected datum symbols
- **Similar annotations**
 - All annotations are highlighted with same semantic references as selected annotation

Additional Information

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|---|--|
| Tips: | When highlighting annotations in Semantic Query mode, you can right-click and then from the shortcut menu, select Add to Combination State to add them to a combination state. |
| |  Note Semantic Query mode is available in Creo Parametric 4.0 M020 and later. |
| Limitations: | No known limitations |
| Does this replace existing functionality? | This is new functionality as of Creo Parametric 4.0 M020. |
| Configuration options associated with this functionality: | None |

Improved Undo and Redo Support in Model-Based Definition

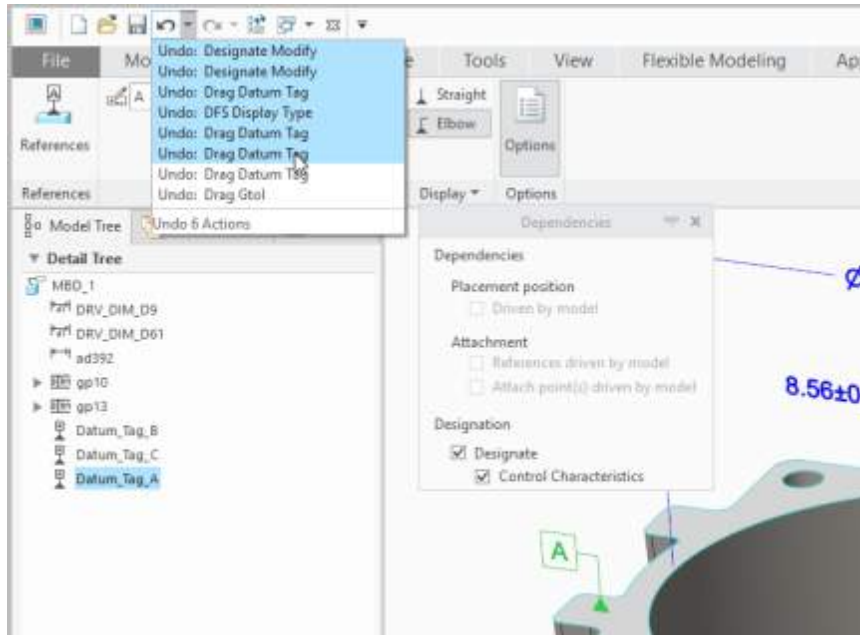
Support for **Undo** and **Redo** is significantly improved for standalone semantic annotation types, such as dimensions, geometric tolerances (GTOLs), datum feature symbols and datum targets. You can reliably and easily revert to previous work states during annotation creation and modification workflows.

User Interface Location: Click **Undo** or **Redo**.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

Reverting to previous work states during standalone annotation creation and modification workflows is more reliable and easier. **Undo** and **Redo** for commands relating to standalone semantic annotation types is more robust. For example, you can use **Undo** and **Redo** when creating and deleting standalone annotations and performing graphical modifications such as movements. You can also use **Undo** and **Redo** when making semantic modifications, such as changing value and text fields, references, designation attributes, and so on.



Additional Information

| | |
|---|---|
| Tips: | None |
| Limitations: | Enhancements do not include format and actions related to text style. |
| Does this replace existing functionality? | This is an improvement to existing functionality. |
| Configuration options associated with this functionality: | None |

Improved Failure Notifications for 3D Annotations

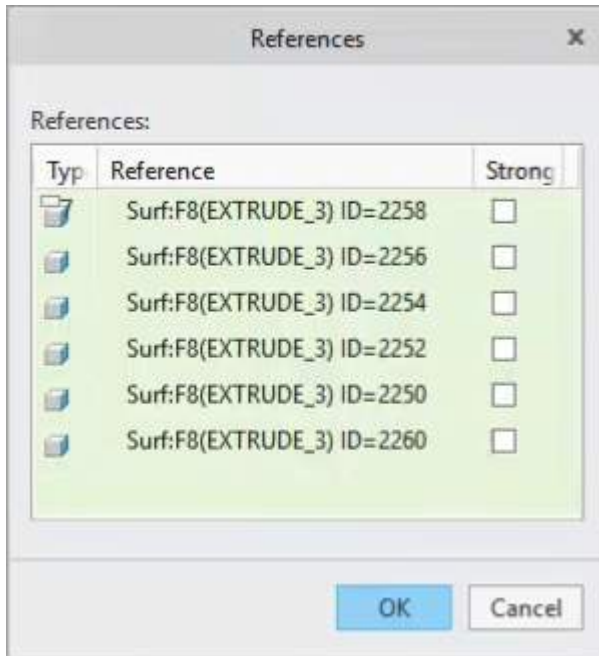
Identify and diagnose failing semantic 3D annotations faster and more intuitively

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

Failure notifications is improved for semantic 3D annotations, such as dimensions, geometric tolerances (GTOLS), datum feature symbols (DFS) and datum targets (DTD). The improved graphical display of those 3D annotations and missing references makes it easier to identify the failure and diagnose of what is causing the failure. The graphical annotation display is color coded and indicates whether

strong or weak references are missing. Additionally, there are icons in the Model Tree and Detail Tree indicating the failing annotation. With the improved graphical display and diagnostics, you can identify and fix failing 3D annotations faster and more intuitively.



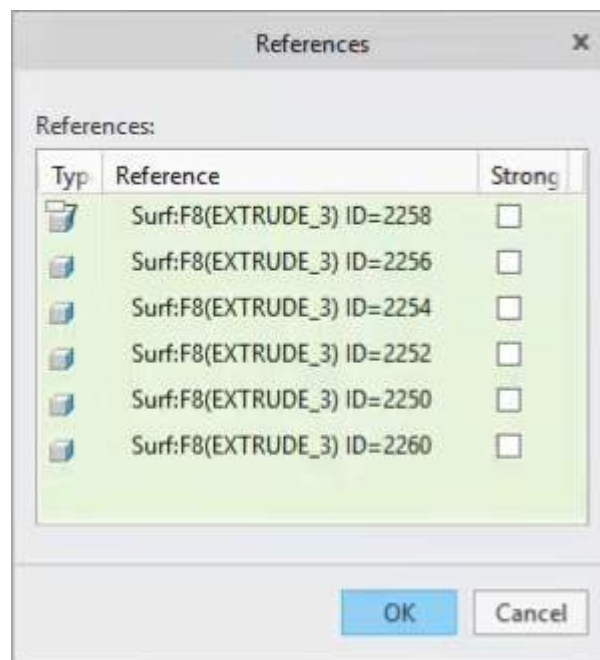
Additional Information

Tips:

A 3D annotation can fail when it loses a reference or when it may be otherwise unable to regenerate. The diagnostic display of failing annotations is color coded in the following way:

- Orange—Only weak references are missing.
- Red—At least one strong reference is missing.

You can mark references as strong or weak from the **References** dialog box for 3D Annotations.



Limitations:

No known limitations.

Does this replace existing functionality?

This is an improvement to existing functionality.

Configuration options associated with this functionality:

Set the configuration option, `highlight_failed_3d_annotations` to `yes` to control the new graphical display of failure notifications for 3D Annotations. This is also useful when printing MBD combination states.

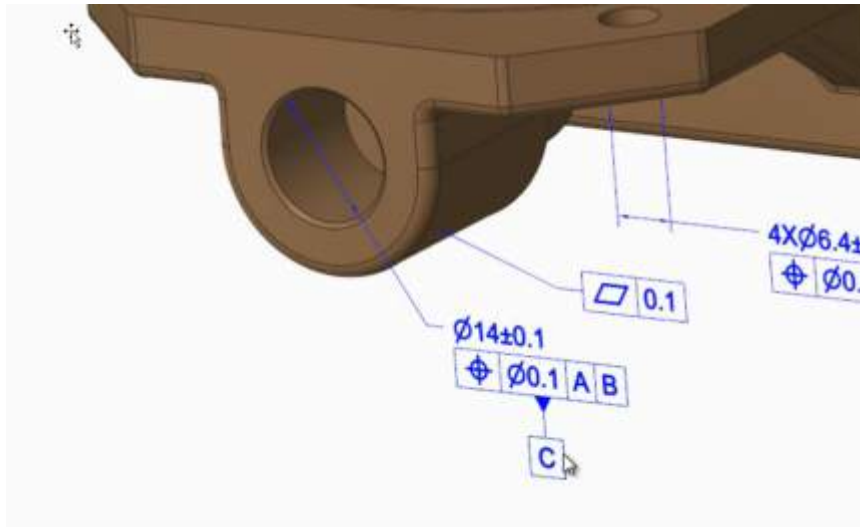
Enhanced Datum Feature Symbol Attachment Option

You can attach a Datum Feature Symbol (DFS) to a GTOL that is placed on a dimension.

Watch a video that demonstrates this enhancement:

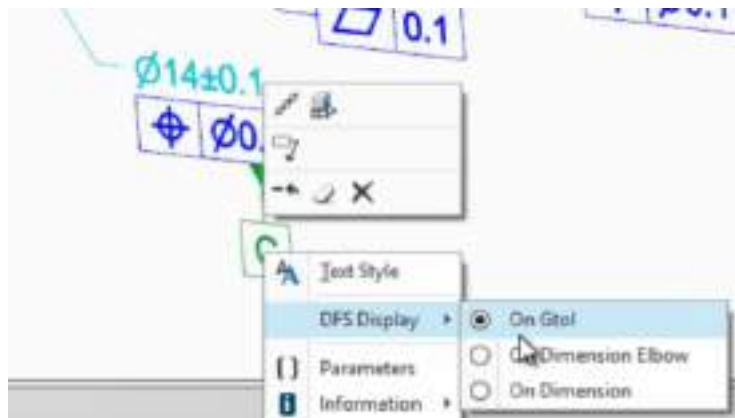
What is the benefit of this enhancement?

You can attach a Datum Feature Symbol (DFS) to a GTOL that is placed on a dimension. This improves DFS placement compliance to the relating ISO 5459-2011 and ASME Y14.41-2009 standards.



Additional Information

Tips: Right-click the datum feature symbol and select other attachment options.



Limitations: None
Does this replace existing functionality? No
Configuration options associated with this functionality: None

Enhanced Workflow for Radial Dimensions

The workflow for creating radial dimensions is enhanced. Defining radial dimensions is faster, easier, more visual, and intuitive.

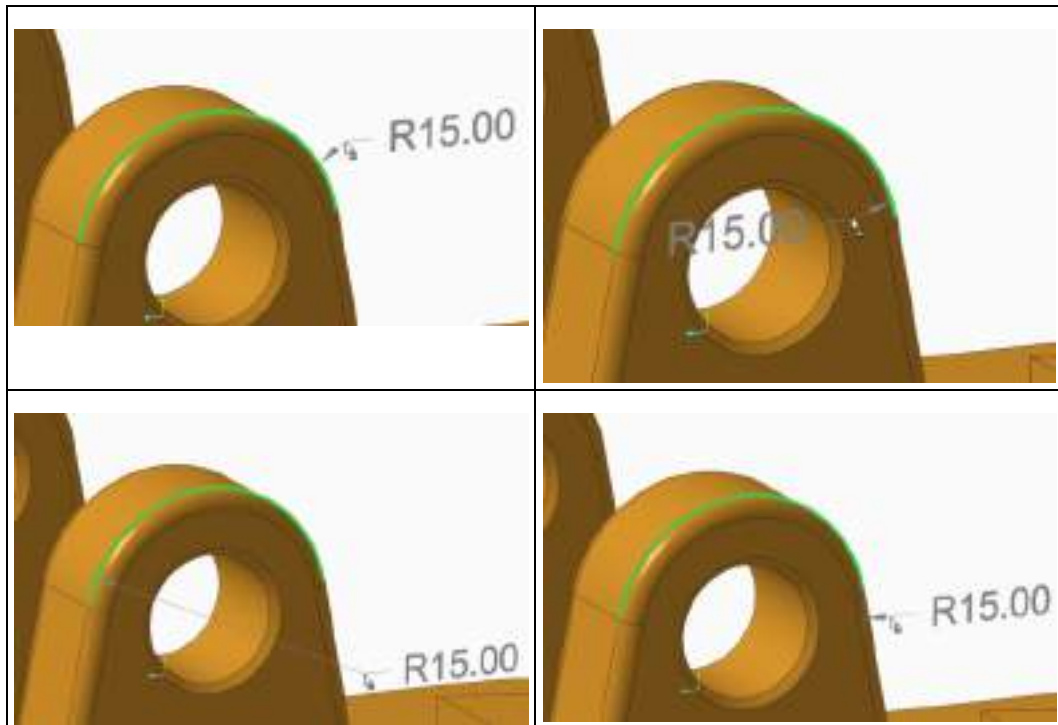
Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

Typical creation of a radial dimension provides for up to four flip states that represent different witness line and arrow configurations.

In Creo Parametric 5.0 the definition of flip states is improved to provide better default states, depending on the position of the pointer. As a result the desired solution is provided faster and with fewer clicks. Visual feedback reduces the need to perform subsequent flip operations to get to the desired state.

When creating the radial dimension, move the pointer to different locations to see an update of the default arrow state. Middle-click to place the dimension. See the examples below:



Additional Information

Tips:

You can flip to different arrow states using the methods described below:

- Click **Flip** on the **Dimension** tab.
- Select the dimension and then right-click to toggle through the states.
- Click **Flip Arrows** on the mini toolbar.



Limitations:

None

Does this replace existing functionality?

This replaces the workflow for creating radial dimensions in Creo Parametric 4.0 and earlier.

Configuration options associated with this functionality:

None

9

Sheet Metal

| | |
|--|----|
| New Types of Corner Reliefs | 60 |
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| Closed Section for Bend Relief | 67 |
| Improved Rounds and Chamfers for Flexible Modeling | 69 |

New Types of Corner Reliefs

New types of corner reliefs provide more geometrical solutions.

User Interface Location:

- In the **Shapes** group click **Flange** ▶ **Relief** and then select **Corner Relief**.
- Click **Flexible Modeling** ▶ **Sheet Metal Objects** ▶ **Edit Corner Relief**.
- Click **Flexible Modeling** ▶ **Sheet Metal Objects** ▶ **Recognize Corner Reliefs**.

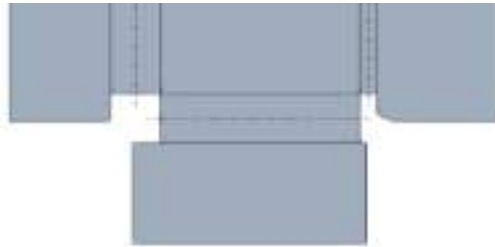
Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

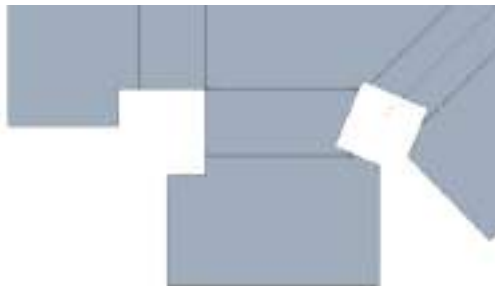
You have more control when creating or modifying corner reliefs. When defining a corner relief using **Flange** or modifying a corner relief with the **Edit Corner Relief** tool when working in Flexible Modeling, you can select additional shapes such as **Normal** and **Square**. For orientation of the shape, **Bisector** and **Diagonal** are added. The **Origin** options, **Corner point** and **Bend lines intersection**, help you to position the shape. The **Orientation** and **Origin** options are also available for other shapes such as **Circular**, **Rectangular** or **Obround**.

New shapes are available when creating or modifying corner reliefs:

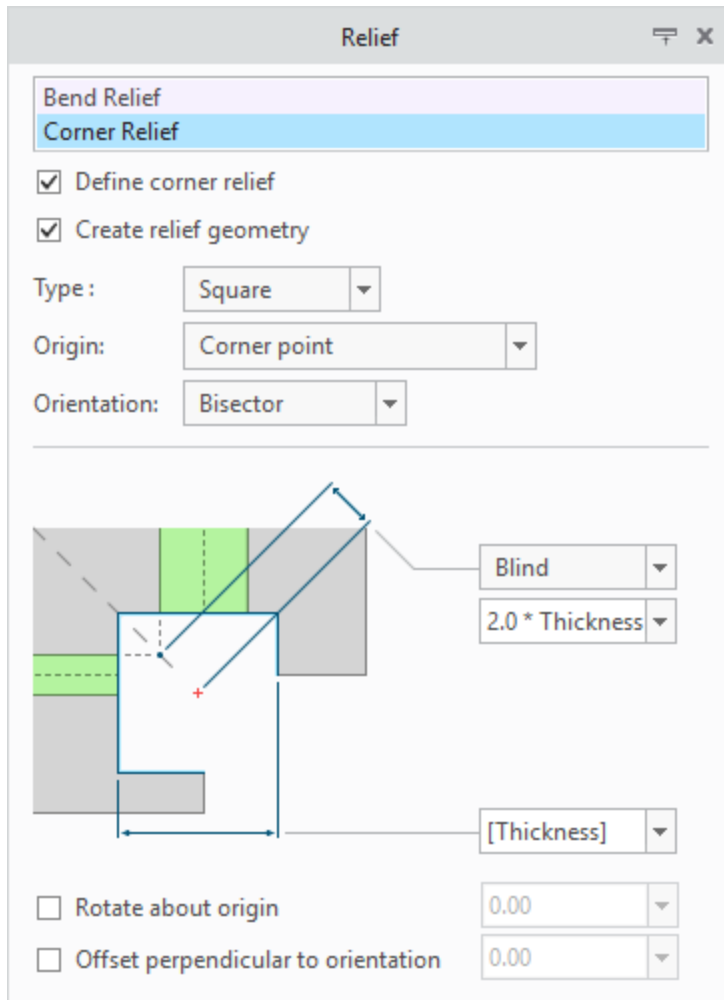
Normal—Normal creates a cut from the corner and up to and normal to the bend end.



Square—Square creates a cut which is concentric with the relief anchor point reference and its section diagonal is parallel to the relief orientation reference.

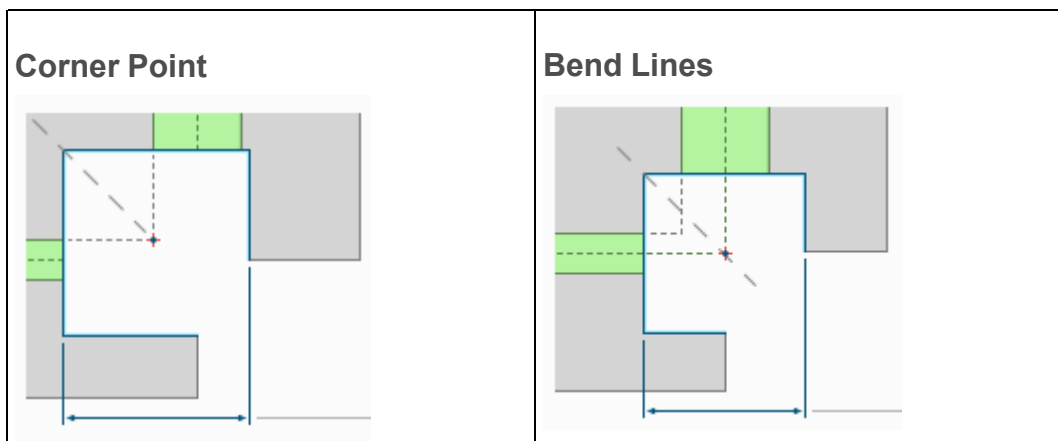


See the **Relief** dialog box below for an example of new shape types:

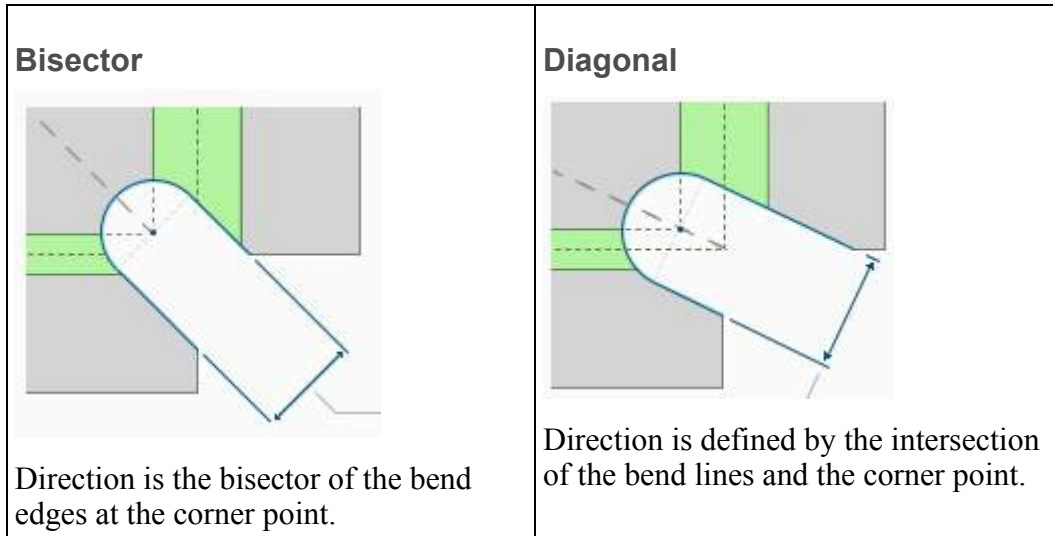


Draggers are added to control the length, position, and offset value when defining a square-shaped corner relief.

Origin Options



Orientation Options



The **Edit Corner Relief** tool gives you more flexibility to change corner reliefs in a sheet metal part in an intuitive and easy way, independent of how the corner relief was created.

In addition, the recognition of corner reliefs also includes the recognition of **Orientation** and **Origin** options independent of how the part was created. This also applies to parts created in another CAD system.

Additional Information

| | |
|---|---|
| Tips: | In complex corner cases, when a normal shape is not possible, a V Notch relief is created instead. |
| Limitations: | None |
| Does this replace existing functionality? | This enhances the functionality for existing tools. |
| Configuration options associated with this functionality: | None |

Flattened Representation of Sheet Metal Part Is Improved

Defining a flat pattern representation of a sheet metal part is improved and simplified.

User Interface Location: In the Graphics toolbar click **Flat Pattern Preview**.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

The process of defining a simplified representation of a sheet metal part in the flattened state, while having the master representation in the formed (bent) state is simplified. You can easily switch from a bent to a flattened state when modeling a sheet metal part. No additional Windchill business object is created or required.

The process is outlined below:

Create a flat pattern preview.



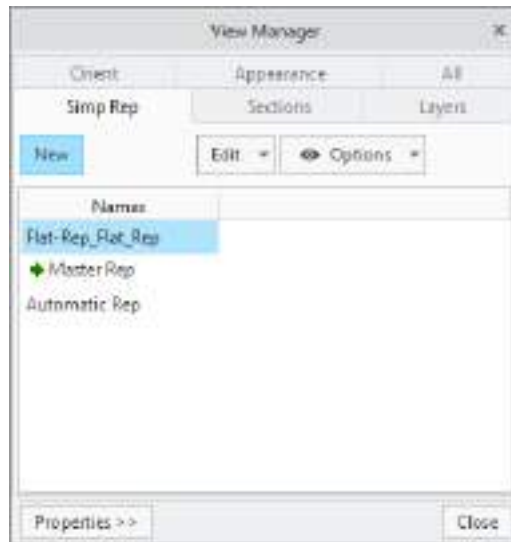
Create a representation.



Name the representation.



Select the newly created representation.



Activate the new simplified representation.



Additional Information

| | |
|---|--|
| Tips: | You can create only one flat pattern representation. |
| Limitations: | No known limitations. |
| Does this replace existing functionality? | This is an enhancement to existing functionality. |
| Configuration options associated with this functionality: | None. |

Conversion Is Improved

The conversion tool is improved to support conversion of solid parts into sheet metal parts even if the parts contains nonuniform thickness.

User Interface Location: Click **Model** ► **Operations** ► **Convert to Sheetmetal**.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

When you use the enhanced **Convert to Sheetmetal** tool, you can convert solid parts into sheet metal parts and enforce uniform thickness. The improvements support the import of sheet metal parts from legacy systems and the conversion into valid Creo Parametric sheet metal parts.

The enhancements give you more control over the conversion. After defining the first wall and using automatic thickness detection, you can perform the following actions:

- Manually overwrite the calculated thickness
- Include additional surfaces or exclude surfaces
- Keep, remove, or ignore the adjacent rounds and chamfers
- Keep nonclassified surfaces as separate quilts
- Troubleshoot for additional guidance

See the different stages of the part below:

Solid part in Creo Parametric



Result after selecting the driving surface and first wall



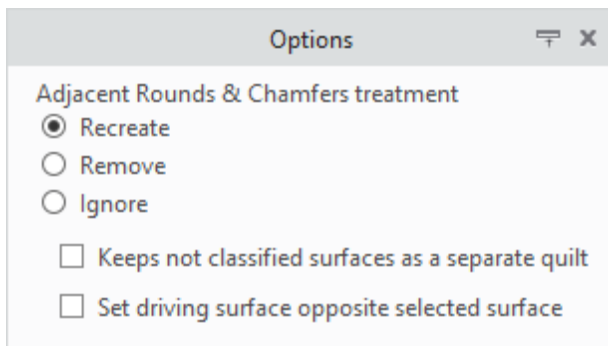
Visual feedback after adding more surfaces



Final Result

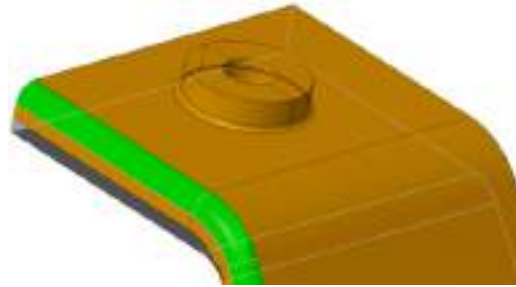


Under **Options** you can make selections regarding the treatment of adjacent rounds and chamfers.

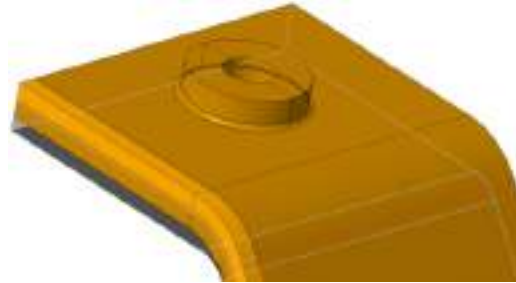


See descriptions of these options below:

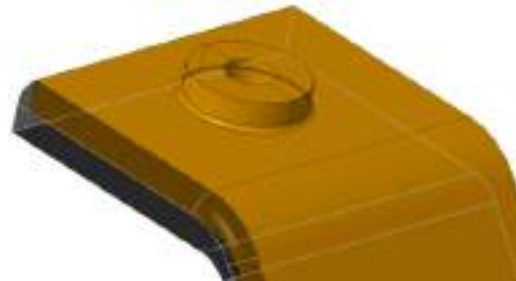
Recreate—Rounds and chamfers are recognized and recreated.



Remove—Rounds and chamfers are removed and a sharp edge is created.



Ignore—Rounds and chamfers are recognized and side faces are trimmed back by the size of the round or chamfer.



Additional Information

| | |
|---|---|
| Tips: | There is no support for flushed hem and piercing forms. |
| Limitations: | No known limitations. |
| Does this replace existing functionality? | This is an enhancement to existing functionality. |
| Configuration options associated with this functionality: | None. |

Closed Section for Bend Relief

You have more control when creating closed sections for bend reliefs.

User Interface Location:

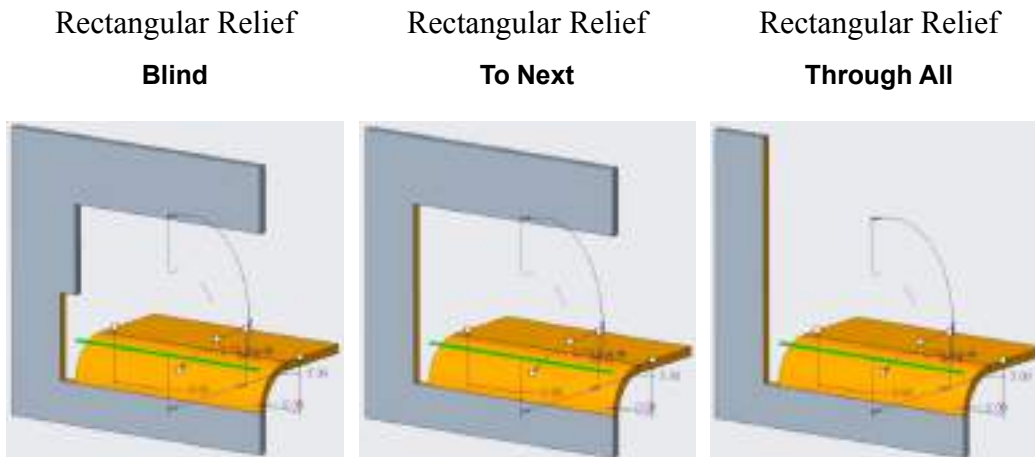
- Click **Model** ▶ **Flange** ▶ **Relief**.
- Click **Model** ▶ **Flat** ▶ **Relief**.
- Click **Model** ▶ **Editing** ▶ **Join** ▶ **Relief**.
- Click **Flexible Modeling** ▶ **Edit Bend Relief** ▶ **Shape**.

Watch a video that demonstrates this enhancement:

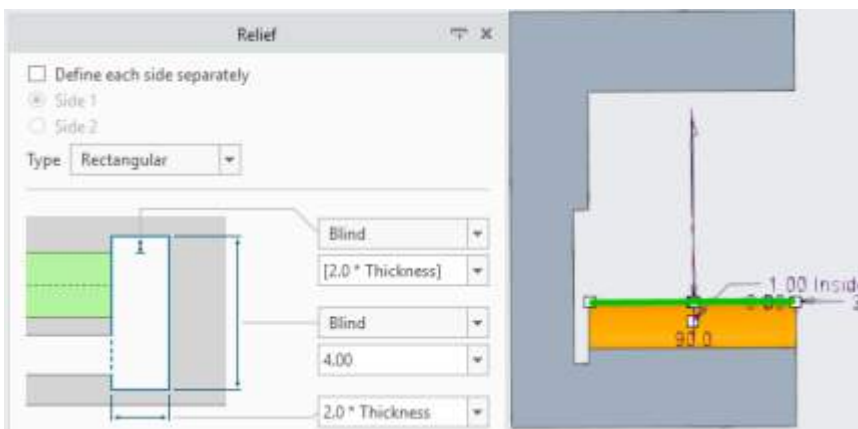
What is the benefit of this enhancement?

You have more control when creating or modifying bend reliefs. When defining a bend relief with a rectangular or oblong shape, you can control the length of shape. In Creo Parametric 5.0 there are more geometrical solutions available than in earlier releases.

There are three new options to control length:



See an example of the **Relief** dialog box below:



Draggers are added to control the length of the rectangular or oblong shape. Recognition of bend reliefs (**Flexible Modeling** ► **Sheet Metal Objects** ► **Recognize Bend Reliefs**) includes **Blind**, **To Next**, and **Through All**. Recognition is independent of how the part was created. This also includes parts imported from another CAD system.

Additional Information

| | |
|---|--|
| Tips: | When defining the length of a bend relief, ensure the length is greater than the relief depth. |
| Limitations: | There are no known limitations. |
| Does this replace existing functionality? | This is an enhancement to existing functionality. |
| Configuration options associated with this functionality: | None |

Improved Rounds and Chamfers for Flexible Modeling

The treatment of rounds and chamfers during **Flexible Modeling** operations in Sheetmetal is improved.

User Interface Location:

- Click **Flexible Modeling** ► **Pull Wall** ► **Adjacent Conditions** ► **Create rounds/chamfer geometry**.
- Click **Flexible Modeling** ► **Edit Bend** ► **Adjacent Conditions** ► **Create rounds/chamfer geometry**.
- Click **Flexible Modeling** ► **Edit Corner Seam** ► **Options** ► **Create rounds/chamfer geometry**.

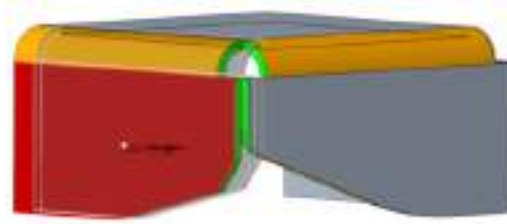
Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

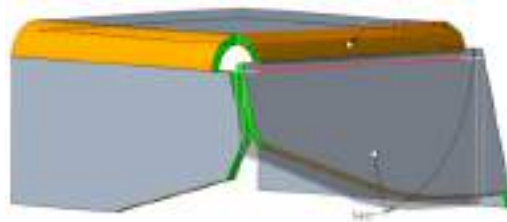
You have more control when working with **Flexible Modeling** operations affecting corner seams with adjacent rounds and chamfers. The automatic recognition of design intent provides the most obvious results. For additional control, use **Create round/chamfer geometry** when working with **Flexible Modeling**, **Pull Wall**, **Edit Bend**, and **Edit Corner Seam**.

See the examples below of using **Create round/chamfer geometry** with **Flexible Modeling** operations:

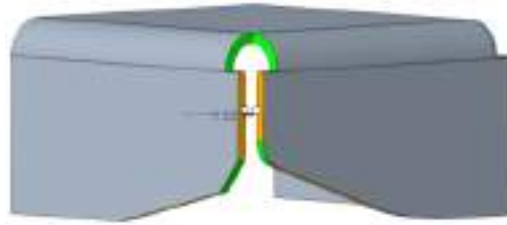
Pull Wall



Edit Bend



Edit Corner Seam



Additional Information

| | |
|---|---|
| Tips: | None |
| Limitations: | No known limitations |
| Does this replace existing functionality? | This is an enhancement to existing functionality. |
| Configuration options associated with this functionality: | None |

10

Data Exchange

| | |
|---|----|
| Transferring Configurations from Creo Elements/Direct to Explode States | 72 |
| Improved Associative Drawing Import from Creo Elements/Direct | 73 |

Transferring Configurations from Creo Elements/Direct to Explode States

When importing a Creo Elements/Direct 3D model to Creo Parametric, existing configurations in Creo Elements/Direct are automatically transferred to explode states in Creo Parametric.

User Interface Location: Click **Open** and then in the **Type** box, select Creo Elements Direct.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

You can use additional data from Creo Elements/Direct in Creo Parametric when converting 3D models. You can use operations from Creo Elements/Direct in Creo Parametric. The conversion of a configuration into to Creo Parametric supports the following:

- Alternative positions of components in an assembly explode state
- Orientation of the configuration
- Showing and hiding of components of a configuration
- Transfer of configurations in Creo Elements/Direct to an explode State or combined state
- Support of multiple configurations for an assembly

The following shows a configuration in a Creo Elements/Direct model imported to Creo Parametric:

Default state in Creo Parametric



Explode state in Creo Parametric



Additional Information

| | |
|---|---|
| Tips: | None. |
| Limitations: | When hiding or showing multiple levels, such as an assembly and its components, the results may not exactly match with the results in Creo Elements/Direct. |
| Does this replace existing functionality? | This is an enhancement to existing functionality. |
| Configuration options associated with this functionality: | None |

Improved Associative Drawing Import from Creo Elements/Direct

User Interface Location: Click **File** ► **Open** and then in the **Type** box select Creo Elements Direct Drawing.

Watch a video that demonstrates this enhancement:

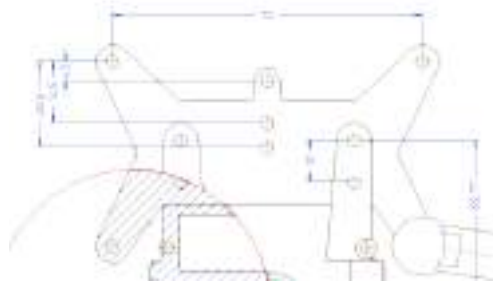
What is the benefit of this enhancement?

Associative drawing import saves time when creating drawings in Creo Parametric for imported Creo Elements/Direct 3D models with associative drawings. This improvement strengthens the transfer of more complex drawings, associatively. Improvements are listed below:

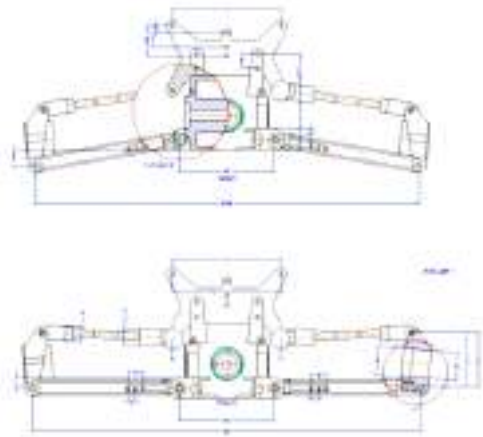
- Centerlines in Creo Elements/Direct drawings are transferred as an axis in Creo Parametric along with the associative references for dimensions.
- Support for views containing Creo Elements/Direct Configurations.

See examples of these improvements below:

Creo Elements/Direct drawing containing centerlines transferred as an axis in Creo Elements/Direct



Creo Elements/Direct drawing with a front view, with the front view containing an exploded state



Additional Information

| | |
|---|---|
| Tips: | None |
| Limitations: | No known limitations |
| Does this replace existing functionality? | This is an enhancement to existing functionality. |
| Configuration options associated with this functionality: | Set <code>enable_rt_6400852</code> to yes for support for views containing Creo Elements/Direct configurations and explode states. This configuration option is required for Sneak Peek only. |

Manufacturing

| | |
|---|----|
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Simplified Visualization and Mass Properties Calculation in Additive Manufacturing

Additive Manufacturing Extension (AMX) supports an improved symbolic representation of lattices.

User Interface Location: On the **Lattice** tab, select **Simplified**.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

In Creo Parametric 4.0 the simplified representation of a beam-based lattice is constructed by lines connecting each node, and the material properties calculation is not supported for simplified lattices. In Creo Parametric 5.0 the simplified representation is improved to more accurately represent the lattice, as you can see in the image below:



This enhancement expands the capabilities of the simplified lattice. The simplified lattice represents a method to quickly run design analysis cycles to speed up the development of light-weight parts that fulfill structural and thermal requirements.

Additional Information

| | |
|--------------|--|
| Tips: | Use this simplified representation to run all the initial simulation experiments. You can switch to a full representation, later in the process. |
| Limitations: | The mass properties calculated by this method for the simplified lattice is an approximation. Run the mass properties calculation on the full lattice representation to get the accurate result. |

| | |
|---|---|
| <p>Does this replace existing functionality?</p> <p>Configuration options associated with this functionality:</p> | <p>Yes. This replaces the previous simplified–representation–for–lattice functionality.</p> <p>None</p> |
|---|---|

Support for Conical Beams from Additive Manufacturing in Creo Simulate

There is support in Creo Simulate for conical beams that are defined in symbolic representations in Additive Manufacturing.

User Interface Location: On the **Lattice** tab select **Simplified**. Then, create a variable density lattice with conical beams.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

In Creo Parametric 5.0 there is support in Additive Manufacturing for the simplified representation of conical beams to be automatically transferred to Creo Simulate.

Simplified Representation of Conical Beams in Additive Manufacturing



Transferred to Creo Simulate



This enhancement expands the capabilities of the simplified lattice. The simplified lattice represents a method to quickly run design–analysis cycle, to speed up the development of light–weight parts that fulfill structural and thermal requirements.

Additional Information

| | |
|---|---|
| Tips: | Use this simplified representation to run all the initial simulation experiments. You can switch to a full representation, later in the process |
| Limitations: | Analyses calculated by this method for the simplified lattice is an approximation. Run the analysis calculations on the full lattice representation to get the accurate result. |
| Does this replace existing functionality? | Yes. This replaces the previous simplified–representation–for–lattice functionality. |
| Configuration options associated with this functionality: | None |

Support for Truss Lattice in Additive Manufacturing

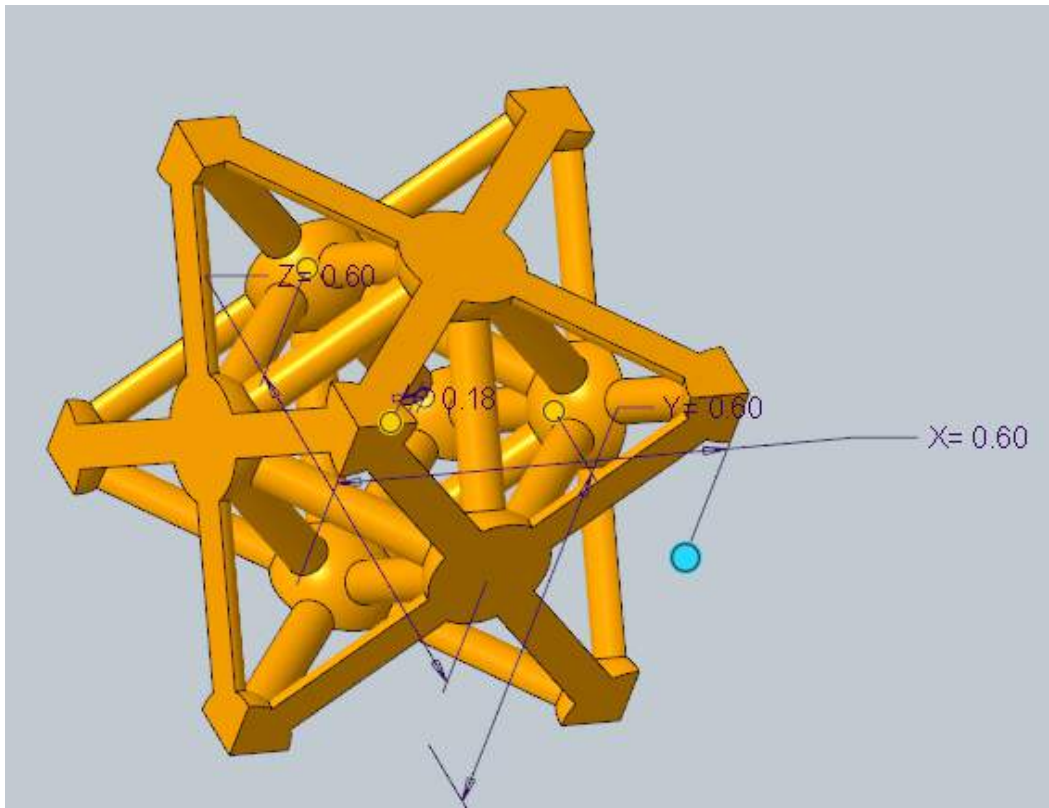
Additive Manufacturing Extension (AMX) supports the Truss cell type.

User Interface Location: On the **Cell** tab, in the **Cell Configuration** box, select **Truss**.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

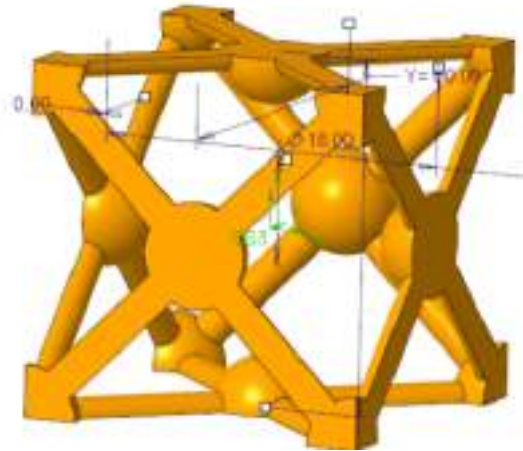
This enhancement expands the capabilities related to additive manufacturing. In Creo Parametric 4.0 and earlier, Truss cell types are not available. In Creo Parametric 5.0, requirements for stiffness and strength-to-weight ratio are met with the addition of the truss lattice.



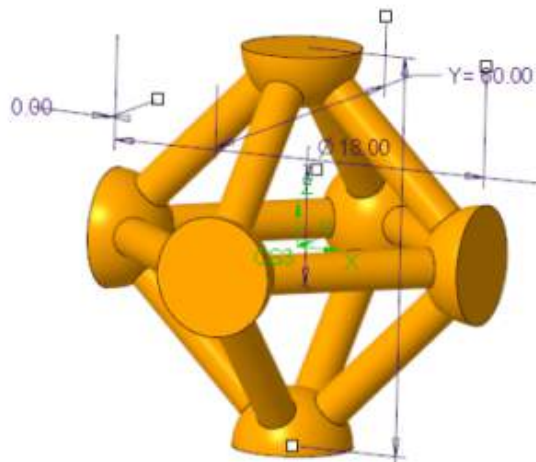
Additional Information

Tips: You can select inner truss beams, outer truss beams, or both:

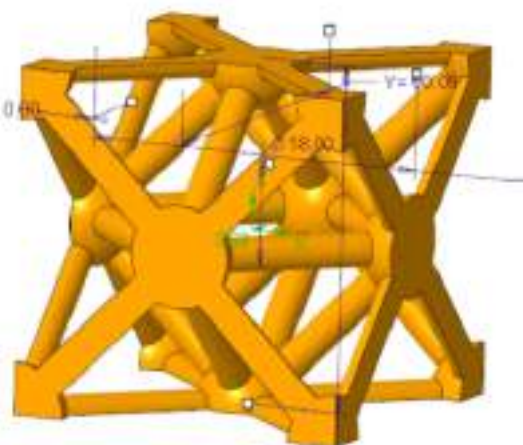
Outer



Inner



Both



Limitations: No known limitations.

Does this replace existing functionality? This is a new tool.

Configuration options associated with this functionality: None.

Support for Thermal Loads in Additive Manufacturing

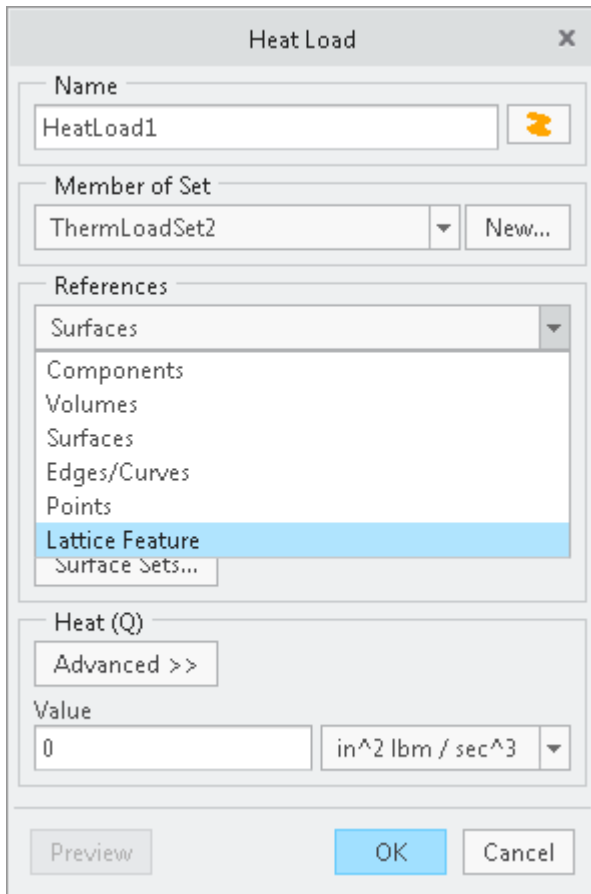
There is support for thermal loads on lattices that are defined in symbolic representations.

User Interface Location: In Creo Simulate thermal mode, click the **Heat Load** tab. In the **Heat Load** dialog box, from the **References** list, select **Lattice Feature**.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

In addition to the improved support for the symbolic representation of lattices, there is also support for thermal loads on lattices defined in a symbolic representation. The support for thermal loads is visible when you are working in Creo Simulate. The simplified representation of lattices supports thermal loads when transferred to Creo Simulate. See the image below:



This enhancement expands the capabilities of the simplified lattice. The simplified lattice represents a method to quickly run design-analysis cycles to speed up the development of light-weight parts that fulfill structural and thermal requirements.

Additional Information

| | |
|---|--|
| Tips: | Use this simplified representation to run all the initial thermal simulation experiments. Later in the process, switch to a full representation. |
| Limitations: | All analyses calculated by this method for the simplified lattice are an approximation. Therefore, you should run the analysis calculations on the full lattice representation to get the accurate result. |
| Does this replace existing functionality? | This replaces previous simplified representations for lattices. |
| Configuration options associated with this functionality: | No configuration options. Requires an Additive Manufacturing Extension (AMX) license and a Creo Simulate license that includes Thermal Simulation. |

Highlighting Toolpaths in Subtractive Manufacturing

In Creo machining extension, the toolpath, references, or both are highlighted when you click in the Model Tree.

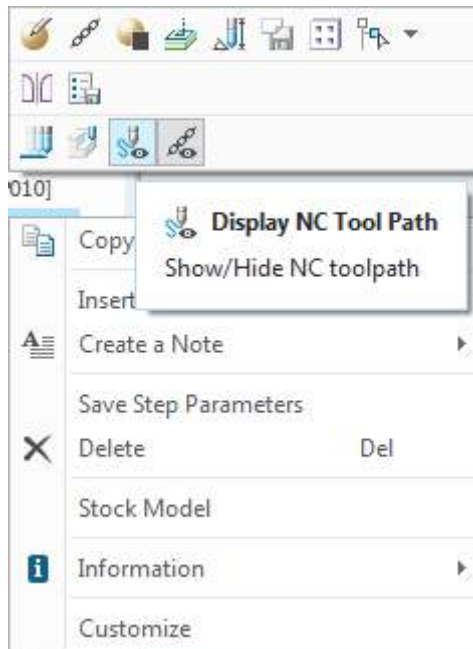
User Interface Location: In the Model tree, right-click a toolpath and then select one or both of the icons.

Watch a video that demonstrates this enhancement:

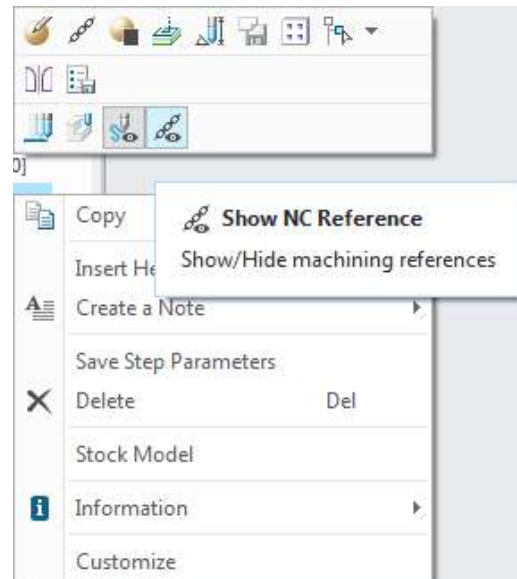
What is the benefit of this enhancement?

When you click in the Model Tree, the corresponding toolpath, reference, or both are highlighted in the graphics area. The selected toolpath or reference remain highlighted until you click the icons in the Model Tree again. In previous releases, many clicks are required to identify toolpaths and references. In Creo 5.0, toolpaths and references are easy to identify.

Display NC Toolpath



Show NC Reference



Additional Information

| | |
|--------------|--|
| Tips: | None |
| Limitations: | You must click the operation and execute the playpath command on all the toolpaths included. Then, you can execute this command. |

| | |
|---|--|
| Does this replace existing functionality? | No. This is a new tool. |
| Configuration options associated with this functionality: | No configurations options are required. A Creo NC license is required. |

Predefined NC Parameters in Subtractive Manufacturing

In Creo machining extensions you can automatically calculate NC parameters based on the geometric values for the tool.

User Interface Location: Formulas are included by default in Creo NC.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

It is easy to create NC toolpaths. Select the tool and references to see an NC toolpath in the graphics area.

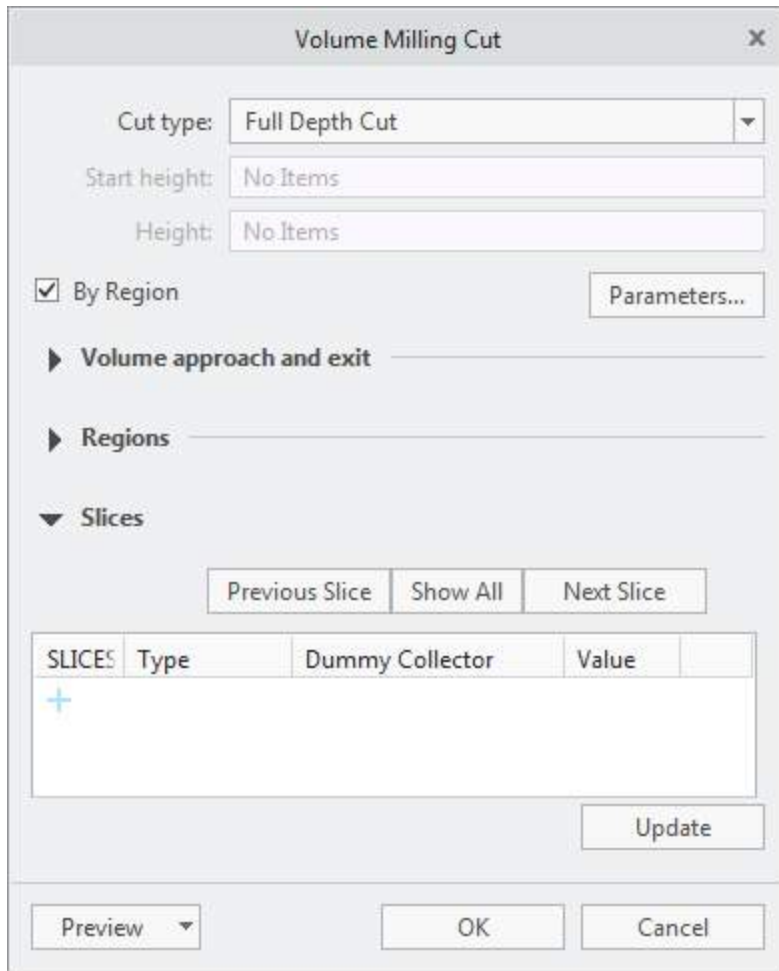
Additional Information

| | |
|---|---|
| Tips: | None |
| Limitations: | Only tool information is used to feed the formulas and to calculate the parameter values. Calculation of turning parameters is limited. |
| Does this replace existing functionality? | No. This is a new tool. |
| Configuration options associated with this functionality: | No configurations options are required. A Creo NC license is required. |

Slice by Slice in Volume Milling (Subtractive Manufacturing)

The volume milling toolpath sequence is improved. In addition to machining region by region in a multipocket toolpath, you can also machine slice-by-slice.

User Interface Location: In volume milling toolpath, click **Tool Motions** to open the **Volume Milling Cut** dialog box. In the **Cut type** list, select **Full Depth Cut**.



Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

In the **Volume Milling Cut** dialog box, the **By Region** check box is selected by default. Clear this check box to enable slice-by-slice in a multipocket scenario.

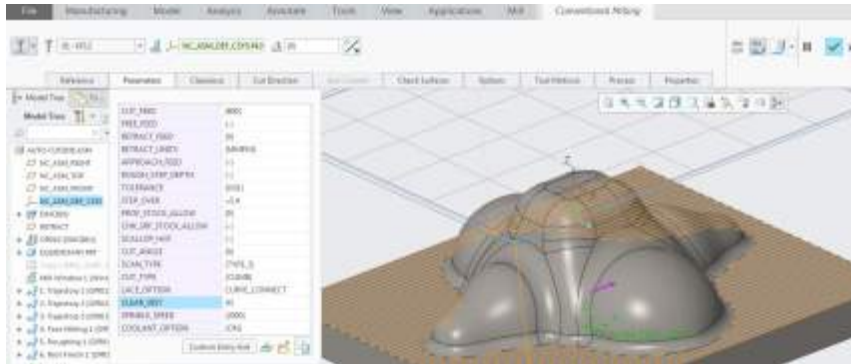
Additional Information

| | |
|---|-------------------------|
| Tips: | None |
| Limitations: | No known limitations. |
| Does this replace existing functionality? | No. This is a new tool. |
| Configuration options associated with this functionality: | None |

Modern User Interface for Conventional Milling (Subtractive Manufacturing)

The user interface for the toolpath sequence for conventional milling is in a ribbon user interface

User Interface Location: Click **Mill** ► **Milling** ► **Conventional Milling**.



Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

In Creo 5.0 there is a modernized **Conventional Milling** ribbon making it easier and more intuitive to perform conventional milling actions. This new user interface corresponds to the Straight Cut command on the menu manager in earlier releases.

Additional Information

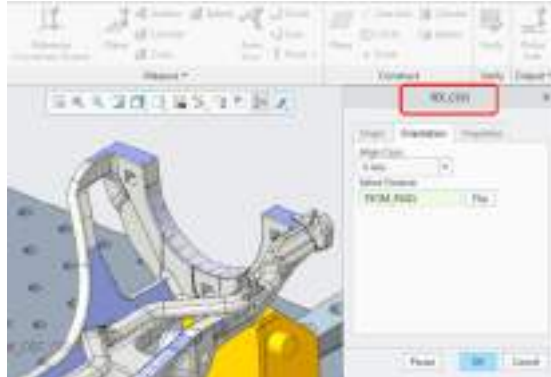
| | |
|---|--|
| Tips: | None |
| Limitations: | In Creo 5, only the Straight Cut command is converted to the ribbon. |
| Does this replace existing functionality? | This replaces the previous surface milling, straight cut command. |
| Configuration options associated with this functionality: | None. |

Modern User Interface for CMM (Subtractive Manufacturing)

In Coordinate Measuring Machines (CMM), the reference coordinate system and CMM Construct Step functionality is accessible from a new ribbon.

User Interface Location:

- Click **Inspect** ► **Reference Coordinate System**.



- Click **Inspect** ► **Construct**.



Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

In previous releases, this functionality is accessible from the Menu Manager. This improvement to the user interface provides a more intuitive and efficient way to perform CMM tasks.

Additional Information

| | |
|---|---|
| Tips: | None |
| Limitations: | This conversion to the ribbon for CMM is for Ref Csys and Construct only. |
| Does this replace existing functionality? | The functionality is the same but with a new user interface. |
| Configuration options associated with this functionality: | None |




12

Fundamentals

| | |
|---|----|
| Additional Commands for Showing and Hiding..... | 89 |
|---|----|

Additional Commands for Showing and Hiding

There are additional commands for showing and hiding objects.

User Interface Location:

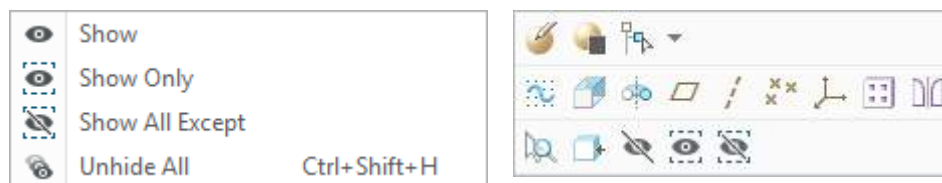
- Right-click an object in the Model Tree or Graphics window and select **Show All** or **Show All Except** from the shortcut menu or the mini toolbar.
- Click **View** and then in the **Show** box select **Show All** or **Show All Except**.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

The commands **Show All** and **Show All Except** are introduced:

- **Show Only**—Shows the selected object. All other objects of the same type are hidden.
- **Show All Except**—Does not show the selected object. All other objects of the same type appear.



These commands are available in Part and Assembly and provide easier control over visibility, by the type of object.

Additional Information

| | |
|---|---|
| Tips: | Click View ► Reset Status to reset to the original visibility configuration. |
| Limitations: | No known limitations. |
| Does this replace existing functionality? | This is additional show and hide functionality. |
| Configuration options associated with this functionality: | None |

13

Creo Simulate

| | |
|--|-----|
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Diagnosing Failures in Mechanism

Diagnostics and suggested resolutions are available during Mechanism failures.

User Interface Location: Access reports from the **Notification Center** or from the **Mechanism Disconnected** warning.

Watch a video that demonstrates this enhancement:

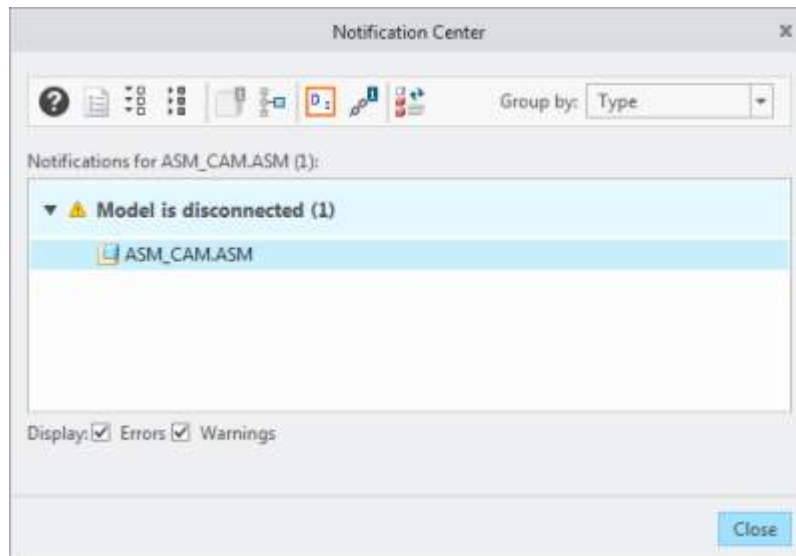
What is the benefit of this enhancement?

During a Mechanism failure you receive messages reporting detailed diagnostics and suggestions for resolution so that you can better understand the issue. These messages appear when any of the following occurs:

- The model is disconnected as a result of regeneration.
- The mechanism analysis fails to connect.
- The mechanism fails during dragging.

When a failure occurs, a `pmech_conn_regen.err` or `pmech_conn.err` file is created. These files contain the error information. You can access these files through the **Notification Center** or by clicking **Details** in the **Mechanism Disconnected** warning.

- `pmech_conn_regen.err` file—Contains errors resulting from regeneration. To view this file, click the diagnostic icon in the **Notification Center**.



- `pmech_conn.err`—Contains all other errors resulting from a drag failure, analysis failure, and so on. To view this file, click **Details** on the **Mechanism Disconnected** warning.



Additional Information

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

Automatic Meshing Refinement

In Creo Simulate, the adjustment for the refinement of meshing for shell elements is automatic.

User Interface Location: Click **Refine Model** ► **AutoGEM** ► **Create**.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

Automatic refinement is the default. This enhancement provides a way to calculate smoother test results automatically. See the image below comparing results in earlier releases to Creo Simulate 5.0.



Additional Information

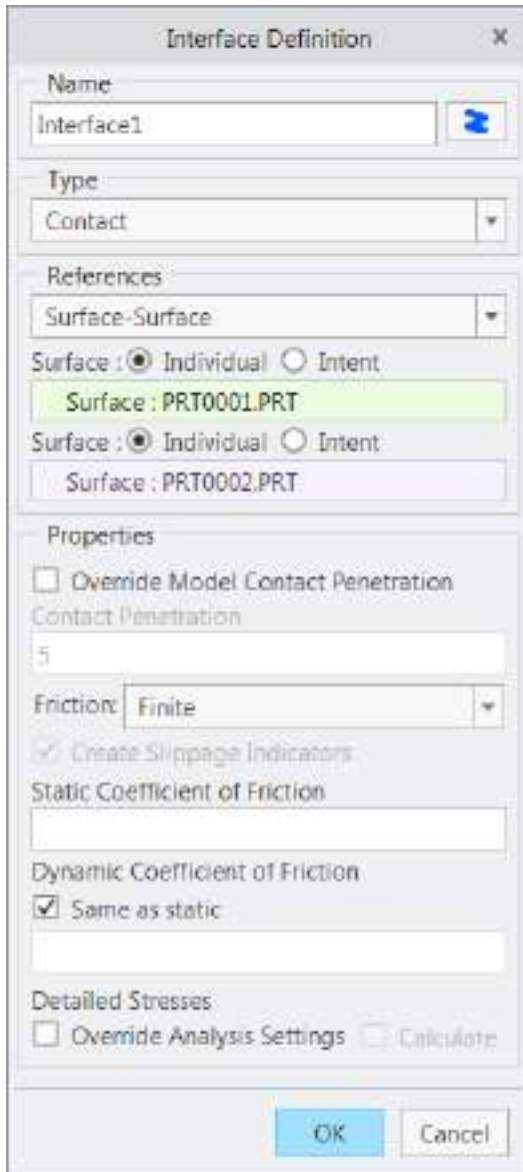
| | |
|--------------|----------------------|
| Tips: | None |
| Limitations: | No known limitations |

| | |
|---|------------------------|
| Does this replace existing functionality? | No. This is a new tool |
| Configuration options associated with this functionality: | None |

Support for Detailed Stresses

There is support for detailed stresses for a given contact interface.

User Interface Location: In Structure mode, click **Refine Model** ► **Connections** ► **Interface**. In the **Interface Definition** dialog box, in the **Type** box, select **Contact** and in the **Friction** box, select **Finite**.



Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

Detailed Stresses is supported only for finite friction interfaces. When **Detailed Stresses** is set, it overrides the global model and analysis-level detailed stresses setting for this given interface. This enhancement gives you the option of using a detailed stresses setting on selected important interfaces. This setting helps to improve the quality of the stress result on a given interface.

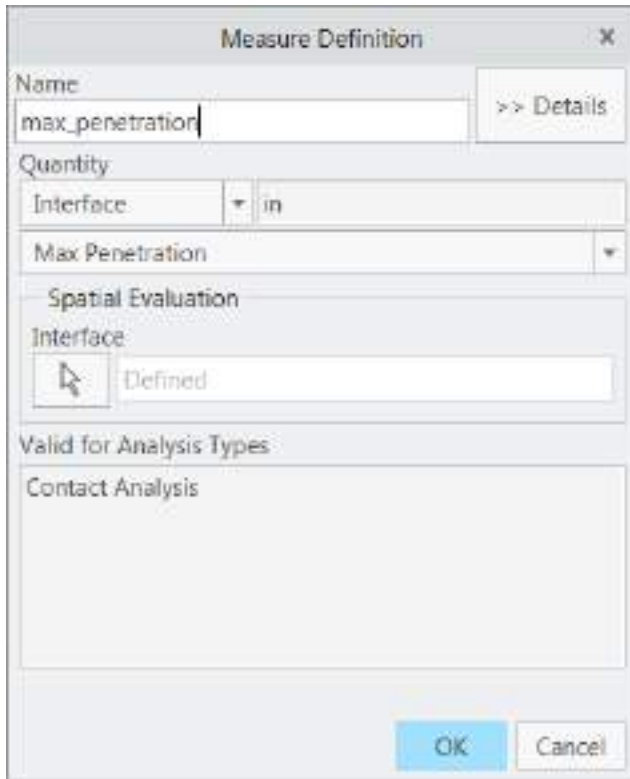
Additional Information

| | |
|---|---|
| Tips: | When Detailed Stresses is set, the engine uses the interface-level detailed stresses setting instead of the global model-level detailed stresses setting. Detailed stresses require more computation time so be selective when using it. |
| Limitations: | Using this setting on many interfaces will result in longer run time. |
| Does this replace existing functionality? | No |
| Configuration options associated with this functionality: | None |

Support for User-Defined Output Measure

There is support for a user-defined output measure to output maximum penetration for an engine on a given interface.

User Interface Location: In Structure mode, click **Home ► Run ► Measures**. In the **Measure** dialog box click **New**. In the **Measure Definition** dialog box, under **Quantity**, select **Interface** and **Max_Penetration**.



Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

This enhancement gives you more information and visibility about maximum penetration an engine uses on a given interface. You can check whether this actual penetration by the engine is acceptable from a design point and then take appropriate actions.

Additional Information

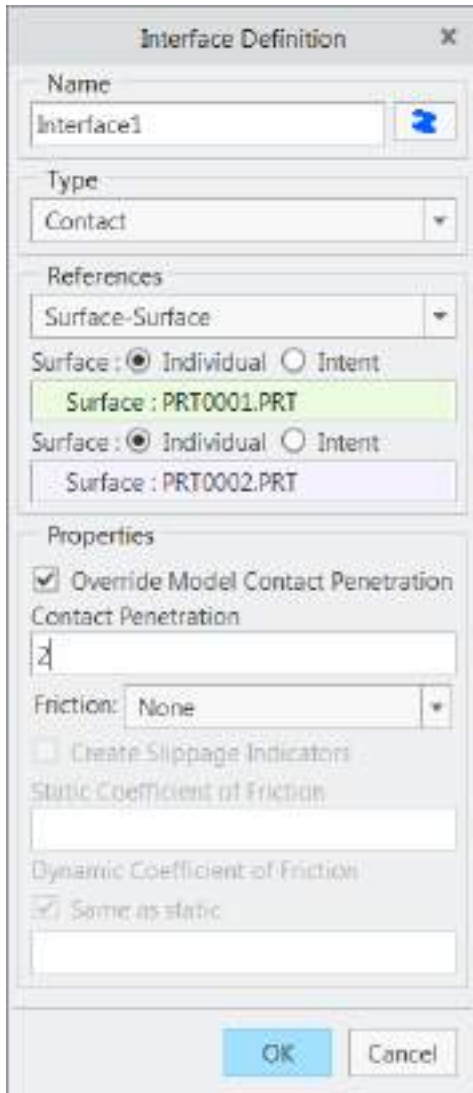
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| Tips: | Use this measure when you want to review the level for maximum penetration on a given interface. If penetration is more, you can tighten the input maximum for the penetration that you specify on this interface. |
| Limitations: | The maximum penetration the engine uses may override the under-input levels for maximum penetration. The engine may override maximum penetration guidelines that you specify, based on valid convergence requirements. |

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| Does this replace existing functionality? | No |
| Configuration options associated with this functionality | <code>sim_contact_penetration</code> —Specify the value of depth for contact penetration depth in a percentage greater than 0.0 through 100.0. The default is 5.0. |

Support for User-Defined Level for Contact Interface

There is support for a user-defined level for maximum penetration, for a given contact interface. This interface property is supported for all contact interfaces: **None**, **Infinite Friction**, and **Finite**. If used, this overrides the global model-level contact penetration for the given interface.

User Interface Location: In Structure mode, click **Refine Model** ▶ **Connections** ▶ **Interface**. In the **Interface Definition** dialog box, under **Type**, select **Contact**.



Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

You can specify the maximum contact-penetration guideline on selected important interfaces. This interface-specific level of penetration helps to tune your contact models closer to reality.

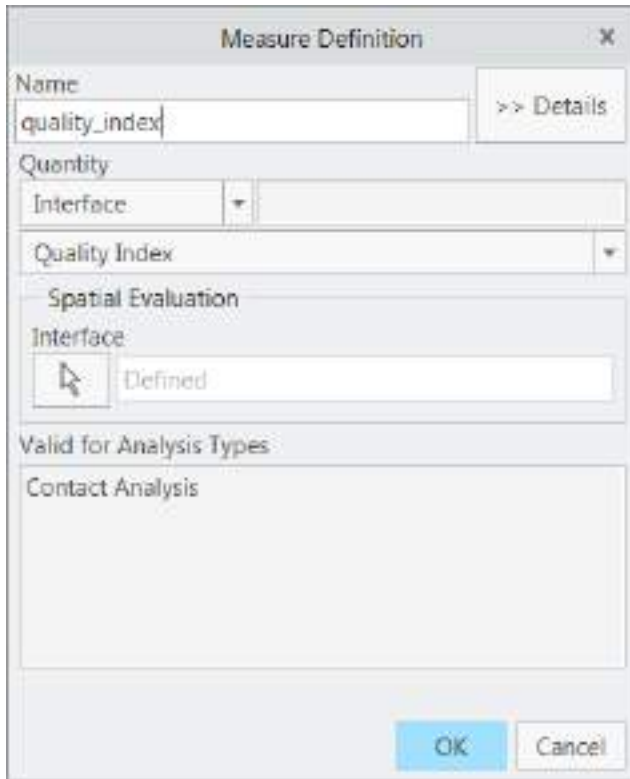
Additional Information

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| Tips: | When you specify the level for maximum penetration, for a given contact interface, the engine uses your specification instead of the global model-level guidelines for maximum penetration. You should make this specification on important interfaces. |
| Limitations: | The tighter levels for penetration on any interface may result in convergence difficulties. |
| Does this replace existing functionality? | No |
| Configuration options associated with this functionality: | <code>sim_contact_penetration</code> —Specify the value of depth for contact penetration depth in a percentage greater than 0.0 through 100.0. The default is 5.0. |

Support for User-Defined Output Measure for Quality Index

There is support for user-defined output measure to output the quality index of the interface.

User Interface Location: In Structure mode, click **Home** ► **Run** ► **Measures**. In the **Measure** dialog box, click **New**. In the **Measure Definition** dialog box under **Quantity**, select **Interface** and **Quality Index**.



Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

This enhancement gives you more information and visibility about the quality of the interface. You can check whether this interface is adequately meshed to give quality results. This measure reports a value in the range of zero to one (0-1). Zero (0) is a poor mesh and one (1) is a good mesh.

Additional Information

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| Tips: | Use this measure when you want to review mesh quality on a given interface. If the value of this measure is < 1 , then you should consider refining the mesh on this interface by adding the appropriate AutoGEM control. Or, you can improve the mesh quality by using a mapped mesh. |
| Limitations: | The overall interface quality depends on the mesh quality and on the nature of contact. A contact between curved edges or surfaces that results in a contact area near zero, can also impact the interface quality. |

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| Does this replace existing functionality? | No |
| Configuration options associated with this functionality: | None |

Solver Accuracy Settings for Contact Analysis

There is support for different solver accuracy levels for contact analysis. For small deformation analysis (SDA) contact analysis, there are two accuracy levels: **Medium** and **High**. For large deformation analysis (LDA) Contact Analysis, there are three accuracy levels: **Low**, **Medium**, and **High**.

User Interface Location: In Structure mode click **Home** ► **Analyses and Studies**.

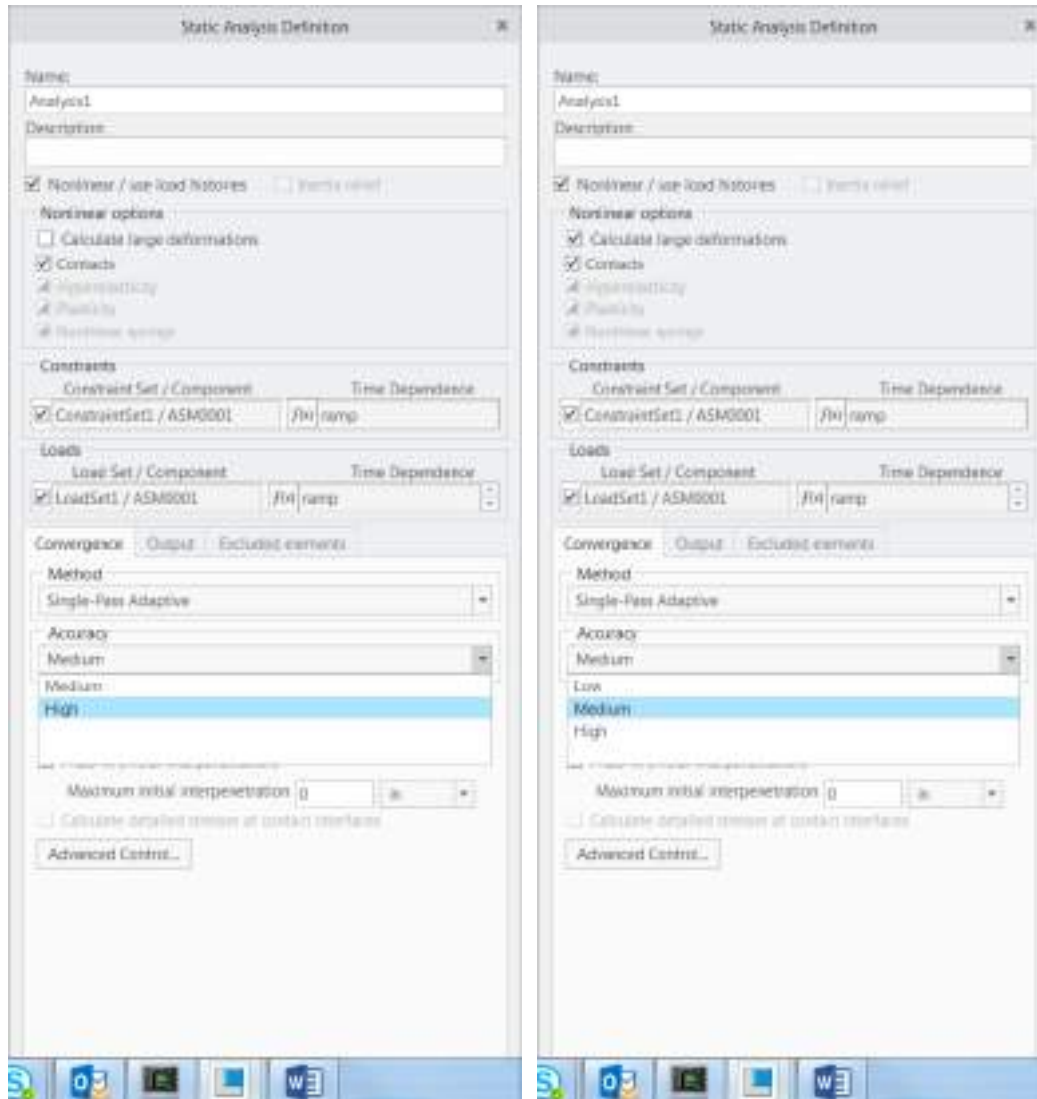
Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

This enhancement gives you more control over the solution accuracy of contact analysis. In initial design phases, when seeking approximate solutions, you can use **Low** or **Medium** accuracy. When finalizing designs, you can then switch to **High** accuracy. This helps to save time and cost.

SDA Contact Analysis

LDA Contact Analysis



Additional Information

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| Tips: | Use this contact solver accuracy setting to help save solution time in initial design stages. The default medium accuracy in LDA and SDA contact analysis is sufficient and results are reasonable. |
| Limitations: | The High accuracy settings for contact solver may require a longer solution time. The solution may not converge if you have tight penetration requirements on important interfaces. Also, the models with a low measure value for the interface quality index, may have |

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| | difficulty during convergence. |
| Does this replace existing functionality? | No |
| Configuration options associated with this functionality: | None |




14

Creo Direct

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| Round Handling in Draft in Creo Direct..... | 105 |
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Round Handling in Draft in Creo Direct

You can easily apply drafts to design models containing rounds and chamfers.

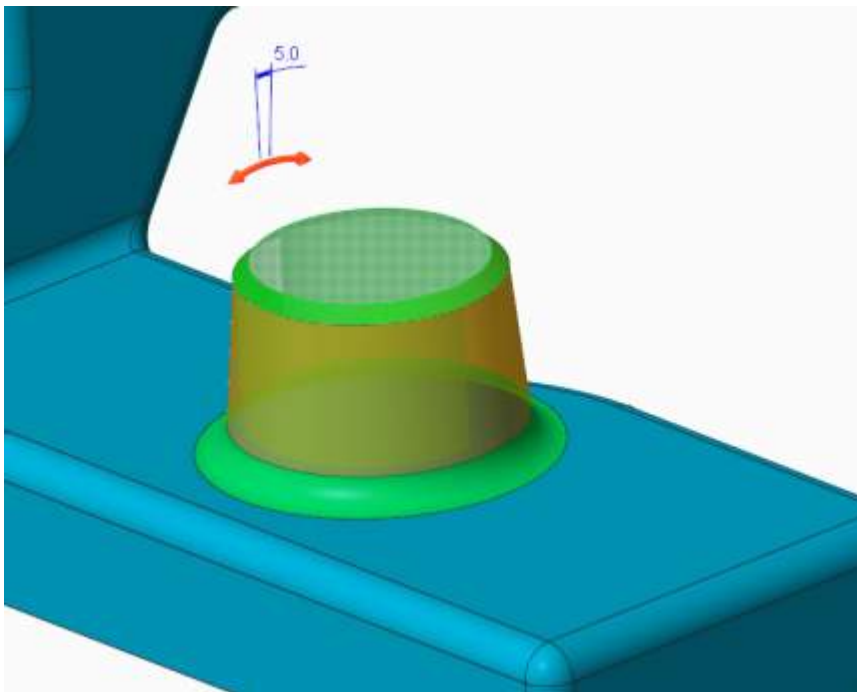
User Interface Location: Click **Home** ► **Draft**.

Watch a video that demonstrates this enhancement:

What is the benefit of this enhancement?

You can easily apply drafts to design models containing rounds and chamfers. This addresses the difficulty in Creo Direct 4.0 and earlier releases in applying drafts to rounded-base part models from the design department or to imported models containing rounds and chamfers.



You can select surfaces as references for the **Draft** tool even if they have adjacent rounds or chamfers. Rounds and chamfers are automatically detected and highlighted in a different color. Rounds and chamfer handling is then applied as it is for other Creo Direct modeling tools such as **Move** and **Offset**. This allows you to apply drafts to models that already contain rounds or chamfers at the boundaries of the to-be-drafted surfaces.




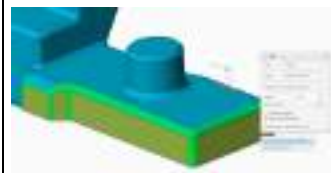
This enhancement increases productivity through faster creation of drafted surfaces having adjacent rounds or chamfers.


Additional Information

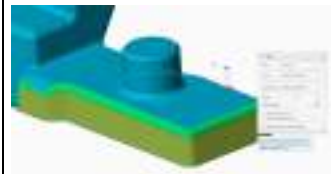
Tips:

- If the geometry selection contains inlying rounds, you can control their inclusion or exclusion from the draft operation by selecting  on the floating dashboard. It is typically faster and easier to include the inlying rounds in the selection first, and then to select  to exclude them from the drafted geometry.

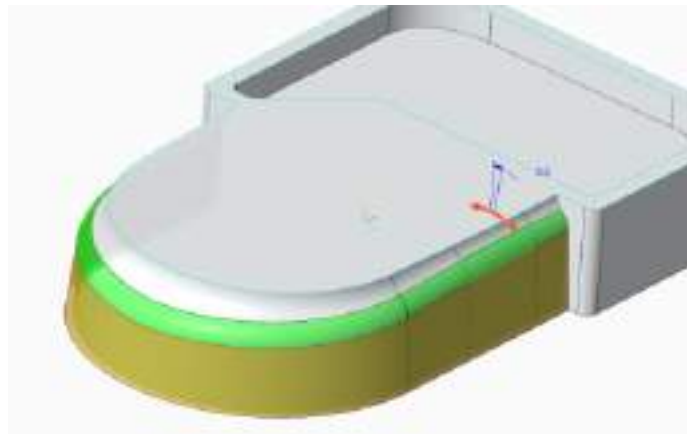
Select  on the floating dashboard to exclude inlying rounds. Inlying rounds then appear in green and are treated as rounds.



Click  on the floating dashboard to stop the select operation and to include inlying rounds. Inlying rounds appear in orange, the modified geometry color, and are included in the draft.



- You can specify a round surface chain as a hinge for the draft operation. This can be helpful such as in the example below:



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| <p>Limitations:</p> | <ul style="list-style-type: none"> • Click or clear the Create round/chamfer geometry check box on the floating dashboard. When this check box is selected, rounds and chamfers are recreated after the draft operation. When this check box is cleared, rounds and chamfers are removed. • This enhancement enables round and chamfer handling based on previously available functionality of the Draft feature. It does not extend the general functionality of the Draft feature, such as to drafting previously drafted surfaces. • This enhancement supports round and chamfer types that are currently supported within Creo Direct. |
| <p>Does this replace existing functionality?</p> | <p>No</p> |
| <p>Configuration options associated with this functionality:</p> | <p>To be determined.</p> |