**CREO 8.0** 

ENHANCEMENT SUMMARY DECK

CREO PARAMETRIC, EXTENSIONS AND NEW PRODUCTS







- <u>System</u>
- Core Modeling
- Multibody Design
- Sheet Metal Design
- Weld Design
- Warp
- Inseparable Assembly
- 2D/3D Annotation and MBD
- Advanced Assembly
- Advanced Framework
- Piping and Cabling
- Design Exploration

- Interactive Surface
   Design
- Generative Design
- Render Studio
- GD&T Advisor
- Ansys Powered
   Solutions
- Subtractive Manufacturing
- Additive
   Manufacturing
- ModelCHECK
- Unite Technology
- Windchill Interaction



### SYSTEM

#### SYSTEM



#### Creo 8.0 Enhancement Summary

#### Update Chromium Browser Version Used by Creo

- Creo has updated the Chromium browser used in Creo for the embedded browser to version 85
  - This update addresses several security vulnerabilities with the old version and aligns with Windchill's browser support matrix
  - PTC will continue to monitor release cadence of the Chromium browser and will update Creo's support accordingly
- Creo's default embedded browser is still Internet Explorer, but users can change to leverage Chromium using the config option: windows\_browser\_type



#### Creo Security Updates

- PTC has implemented a strict Security Policy for all of PTC's products to evaluate potential security vulnerabilities that might exist
  - Through regular monitoring using industrial tools, PTC works with our extended development team and partners to continuously update open-source and proprietary code to the latest versions to maintain a secure environment in accordance with PTC's Security Policy
  - These updates include images, libraries, and systems
- Some of these updates include:
  - Update all open-source components and vulnerable DLL's
  - Update Creo.JS to V8
  - Update Creo license server to Flexera V11.17.1.0
  - Remove unused ProTK ports and bind ports to loopback address

#### SYSTEM

#### Creo 8.0 Enhancement Summary

#### Update Creo to Support Java 11

- Creo has updated it support of Java to version 11 to address several security vulnerabilities with the old version and to align with Windchill's Java support matrix
  - Customers can choose to desired Java JRE distribution from either Oracle or Amazon Corretto
- Creo will only require Java 11 JRE if users are accessing customization through J.Link or Java Object Toolkit, Creo Product Insight connected to Thingworx or Windchill PLM Connector
  - Regular communication with Windchill will not require users to have Java installed







ptc

#### Multi-monitor Display Improvements

- New config option to adjust screen resolutions when working with multiple monitors
  - Define primary monitor for Creo
  - Define dpi scaling for Creo independent from OS
  - Use high dpi text display
  - Control text height in graphics
  - Configuration options:
    - PRIMARY\_MONITOR
    - DPI\_SCALE
    - HIGH\_DPI\_TEXT\_ENABLED
    - TEXT\_HEIGHT\_FACTOR



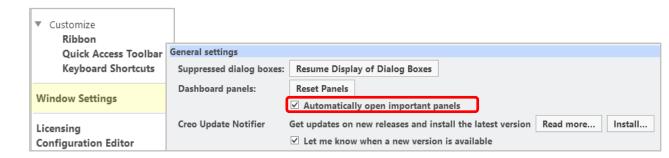


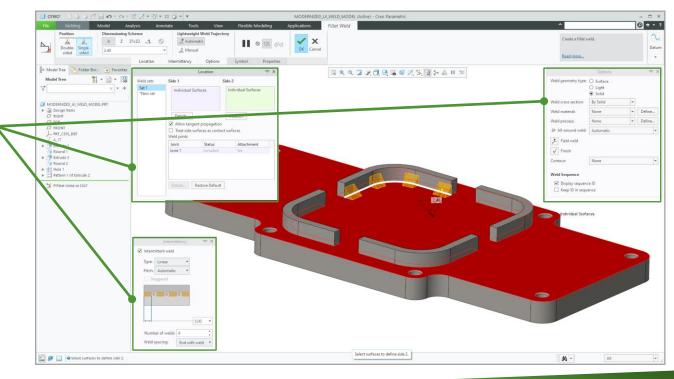


#### Creo 8.0 Enhancement Summary

#### Modernized UI – Dashboards and Skins

- Improved dashboard layout
  - Command ribbon organization improves workflow, usability and user experience
- Optimized UI
  - New option to Automatically open important panels
    - Auto expand panels when entering feature definition workflow
  - Detach multiple panels from the dashboard and display them simultaneously
    - Drag to panels to preferred position on screen
    - Panel positions are remembered and restored
    - Easy access and visibility to feature creation options and defining references
  - Detachable panels simplify interaction and streamline workflows when defining features with a rich set of options





Creo 8.0 Enhancement Summary

#### Shaded 3D Datum Plane Display

- Modernized display of datums using semi-transparent 3D object
  - Datum config options: 3D\_datum\_display\_fill, 3D\_datum\_planes\_transparency
  - Easier to visualize datums and intersections with geometry
- Transparent fill is switched off programmatically
  - Sketcher, HDR display (hidden, no hidden, wireframe)
- Toggle Plane Fill Display from graphics toolbar



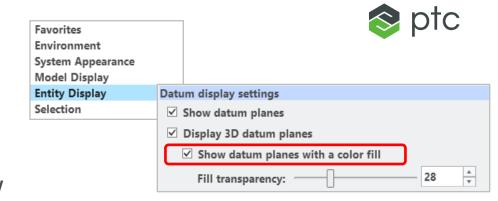
– CSYS config option: csys\_color\_rgb

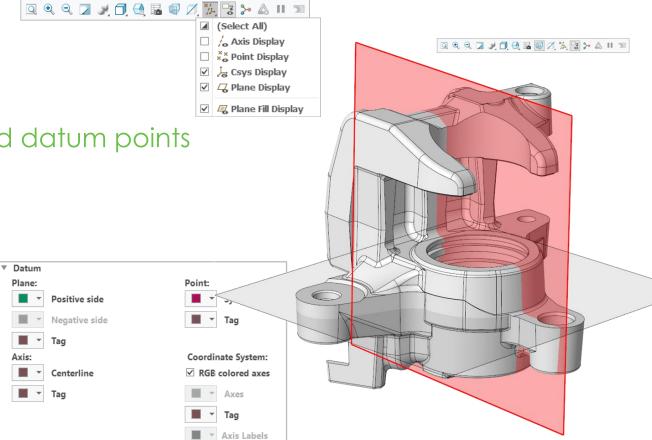
#### Customize display settings (i.e., colors)

- File > Options > Systems Appearance > Entity Display
- File > Options > Entity Display > Datum display settings

### Legacy 2D datum planes can improve performance in some areas

Config option: enable\_legacy\_datum\_planes (Yes)







#### Creo 8.0 Enhancement Summary

#### **Enhanced Visibility Control**

- Transparency Control for bodies, quilts and tessellated geometry
- Easy access to display from Graphics toolbar
  - Toggle display off/on
  - Adjust transparency of Solid Bodies, Quilts and Tessellated Geometry

Selection Options:

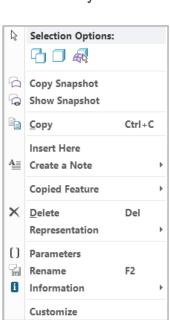
#### Selection Improvements

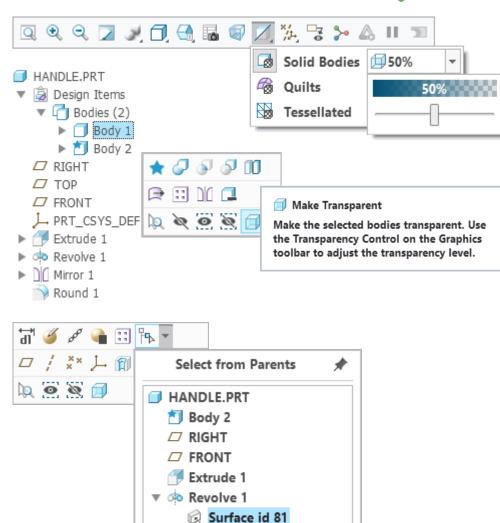
- Select surfaces of all bodies
- Select body surfaces
- Select Quilt or Body
- Quickly "upgrade" selection to the container (quilt, body) that contains the selected geometry

#### Enhancement to "Select from Parents"

- Include parent quilts in listing
- Improve window behavior when "pinned"

Suppress and Resume for "mixed" selection







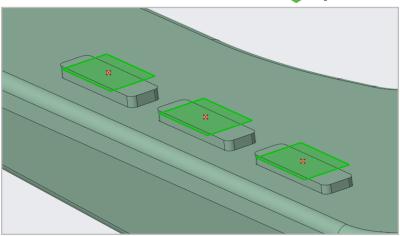
#### Creo 8.0 Enhancement Summary

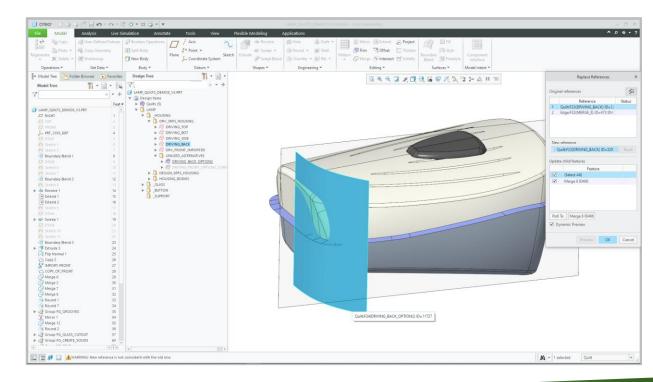
#### Options to Control Default Datum Plane Size

- Additional Options To Control Datum Plane Default Size
  - Default size is automatically calculated to fit the model outline (as previously)
  - Default size is automatically calculated to fit the size of the display reference
  - Default size is set to pre-configured
- Additional logic for point and vertex placement references
- Configuration options:
  - datum\_display\_default
  - datum\_outline\_default\_height
  - datum\_outline\_default\_width

#### Enhanced "Replace References"

- Simplified reference replacement workflow when replacing quilt geometry
- New option "Recursive References"
  - For Quilt reference replacements
  - Recursively list potentially impacted references in the Replace References dialog and replace them
- New option "Dynamic Preview"
  - Automatically trigger regeneration and move to next missing reference





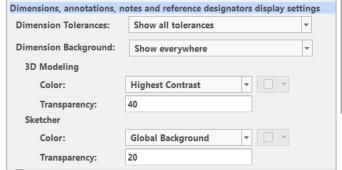
#### ptc

#### Creo 8.0 Enhancement Summary

#### Improved Visibility Controls in Sketcher

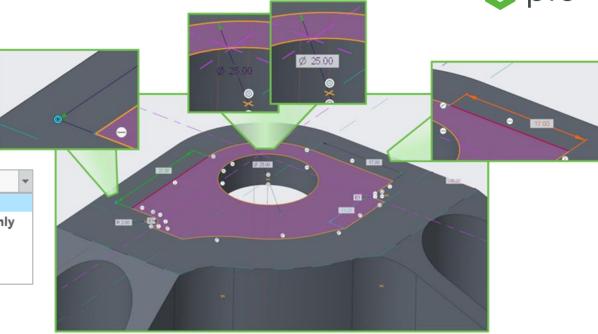
Control of dimension background appearance

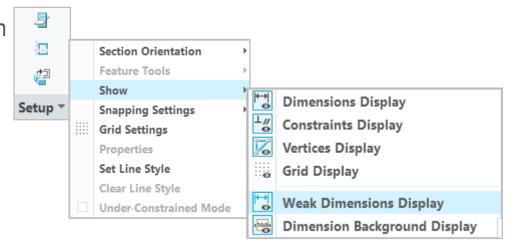
Color and transparency





- More pronounced dimension display during selection and pre-selection
  - Sketch\_thick\_dim\_line\_highlight (yes)
  - Dimension and sketch entity are highlighted when cursor is placed over dimension
- Modernized dimension dragger
  - Dragger icon is easier to see, select and drag
- Easy access to sketch display options
  - Toggle display of Weak Dimensions and Dimension Background





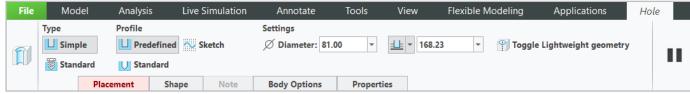


#### Creo 8.0 Enhancement Summary

#### Enhanced Hole Feature UI and Usability

- Improved Command Ribbon organization and UI layout
  - Simplified workflow with improved clarity and access to hole feature creation tools
- Easy to understand icons and labels
- New Tokens for hole note call-outs
  - HOLE\_INSTANCE\_COUNT (number of instances as in PATTERN\_NO)
  - HOLE\_INSTANCE\_LEFT\_BRACKET (evaluates to "(")
  - HOLE INSTANCE RIGHT BRACKET (evaluates to ")")
  - HOLE\_INSTANCE\_HYPHEN\_SYMBOL (evaluates to "-")
  - HOLE\_INSTANCE\_X\_SYMBOL (evaluates to "x")
  - HOLE\_INSTANCE\_SPACE (evaluates to " ")
  - Improved call-out control to differentiate single hole and multiple holes
  - Enhanced compliance to MBD and drawing standards
- Medium-fit options for metric clearance holes
- Standard compliance (selected hole chart corrections)

#### **Creo 7.0**

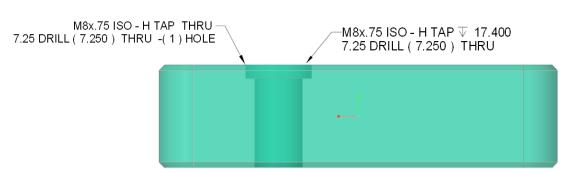


#### Creo 8.0



**Creo 7.0** 

Creo 8.0



HOLE\_INSTANCE\_COUNT (number of instances as in PATTERN\_NO)
HOLE\_INSTANCE\_LEFT\_BRACKET (evaluates to "(")
HOLE\_INSTANCE\_RIGHT\_BRACKET (evaluates to ")")
HOLE\_INSTANCE\_HYPHEN\_SYMBOL (evaluates to "-")
HOLE\_INSTANCE\_X\_SYMBOL (evaluates to "x")
HOLE\_INSTANCE\_SPACE (evaluates to " ")

The tokens evaluate to real values when:

- The hole feature placement is based on a sketch and results in more than one hole instance
- The hole feature is a pattern leader



#### Creo 8.0 Enhancement Summary

#### Multi-hole Feature

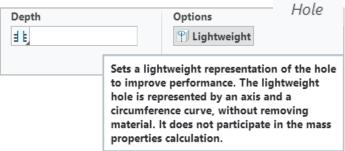
- New sketch-based placement workflow option in hole feature
  - Option to place holes on sketched points and endpoint and midpoints of sketched lines
- Supporting sketched holes with drawing hole tables
- Reference pattern support
  - Multi-hole feature can be used as a Ref Pattern for subsequent geometry
- Hole note pattern number summation

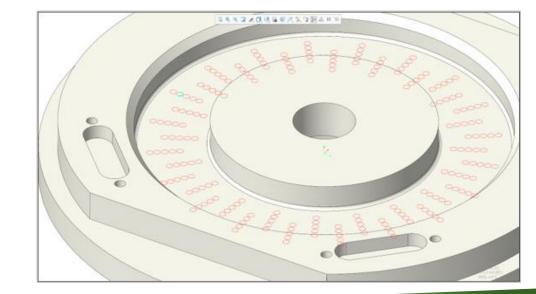
# Type Sketched Sketch Internal Section 2 Place on: X Ø Ø Click here to add item Dimension orientation reference:

#### Lightweight Representation Support for All Hole Types

- Option to set lightweight representation of hole to improve performance
  - Lightweight representation propagates to all pattern members

Previous functionality limited lightweight representation to simple hole types with rectangular profile







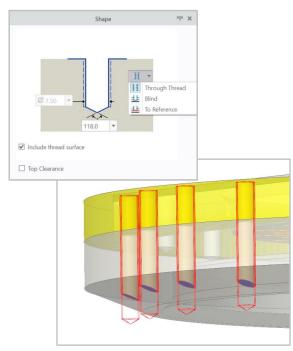
#### Creo 8.0 Enhancement Summary

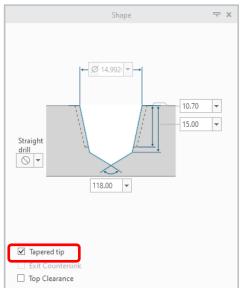
#### Enhanced Support for Tapped and Tapered Holes

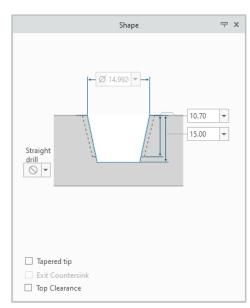
- New options for NPT, NPTF, and ISO-7 standard holes
- Combine straight drilled hole with the tapped, tapered hole section
- Full range of depth options for straight drill section
- Updated hole charts (.hol files) to drive thread depth and straight drill diameter
- Additional geometric shape via new "tapered tip" control option

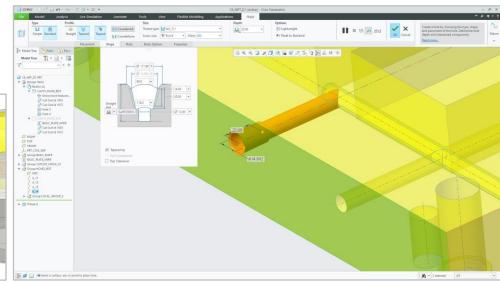
#### **Enriched Thread Handling Options**

- Enriched thread depth control options:
  - Thru Thread
  - Blind
  - To Selected
- Thru Thread" option is enabled for
  - Drill up to next surface
  - Drill to intersect with selected surface
  - Drill to selected object
  - Drill to intersect with all surfaces (in part mode only)







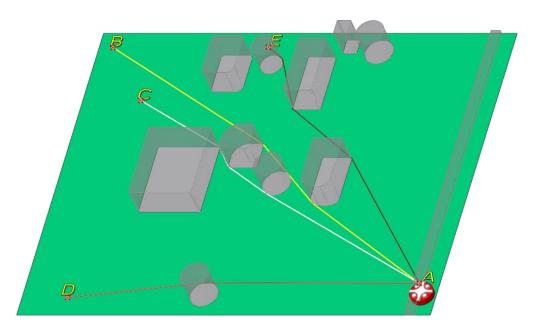




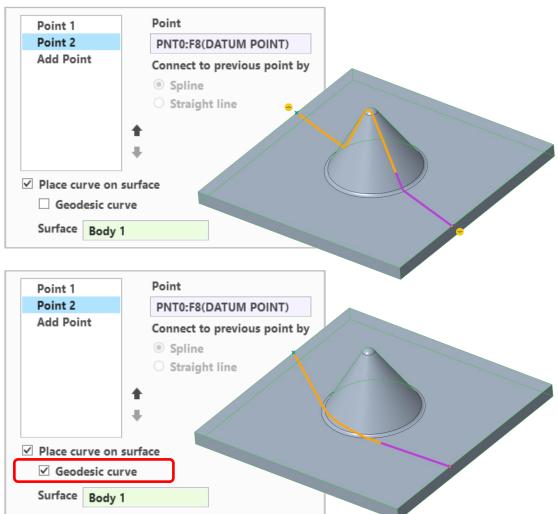
Creo 8.0 Enhancement Summary

#### Geodesic Curve Between 2 Points

- Enhanced Curve Through Points
  - Automatically create curve using "shortest distance" curve-on-surface
  - Specify points and surface(s)









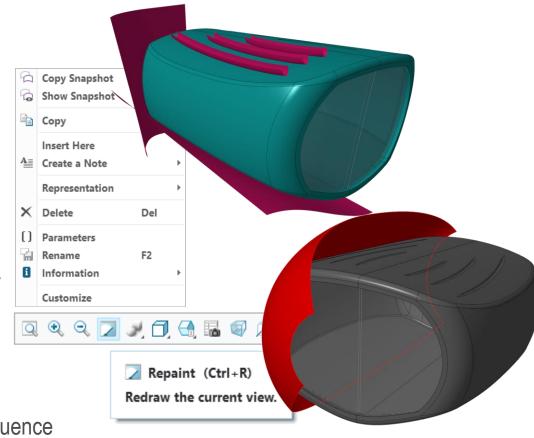
#### Creo 8.0 Enhancement Summary

#### Preview of the Quilt/Body Snapshot

- "Show Snapshot" provides a preview of the quilt/body snapshot
  - RMB quilt/body or contributing features "Show Snapshot"
- Show Snapshot improves understanding and editing
  - For contributing features
    - Shows a snapshot preview of the quilt/body geometry at the point in time after the regeneration of the selected feature.
  - For active bodies/quilts
    - Shows a snapshot preview of the current quilt/body geometry
  - For consumed bodies/quilts
    - Shows the geometry before consumption
- Clear graphics using Redraw

#### Copy the Quilt/Body Snapshot

- Create a copy of the quilt/body at a selected point in the regeneration sequence
  - For contributing features
    - Copies the quilt/body geometry by inserting a copy feature immediately after the selected feature
  - For active bodies/quilts
    - Copies the quilt/body geometry immediately after the last contributing feature
  - For consumed bodies/quilts
    - Copies the quilt/body before the respective consuming feature





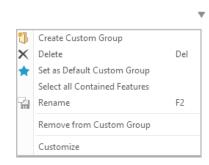
#### Creo 8.0 Enhancement Summary

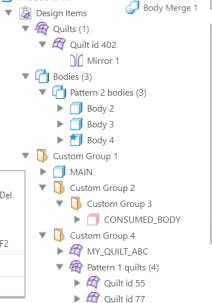
#### New Design Tree (part mode)

- Design Tree helps to organize and document design intent
  - "Design Items" folder organizes bodies, quilts, materials and annotation elements
  - "Quilts node" list individual quilts with contributing features in the Model Tree
    - Similar function to "Bodies Node"
- Second level grouping for bodies and quilts
  - Groupings provide visual clarity of pattern, mirror and import geometry
- Auto Locate in Tree
  - Geometry selection highlights items in Model Tree and Design Tree

#### Custom Groups (model tree)

- Improved flexibility to organize and document design intent
  - Manage quilts and bodies based on "design function"
  - Organization is not dependent on regeneration order
- Custom Group operations
  - Create / Delete / Re-name custom groups
  - Support for nesting, re-order and restructure
- Default Custom Group
  - If set, new quilts/bodies will automatically be created in the Default Custom Group





PRT0001.PRT ▼ 👸 Design Items

▼ @ Quilts (6)

Pattern 1 quilts (4)

Quilt id 55

▶ Ø Quilt id 77

▶ Ø Quilt id 95

▶ Ø Quilt id 113

Extrude 2

Round 1

Round 2

Pattern 2 bodies (3)

▶ ■ Body 2

▶ ¶ Body 3

▶ M Body 4

Extrude 4

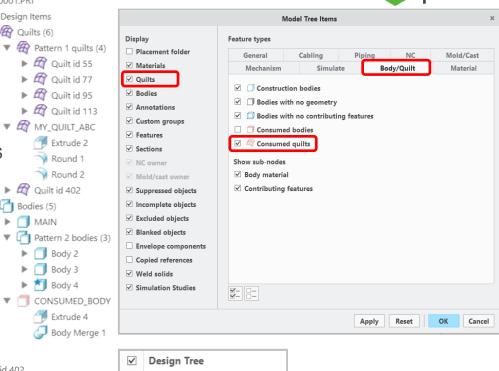
▼ MY QUILT ABC

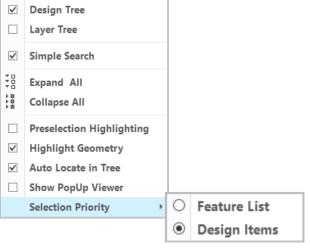
▶ Ø Ouilt id 402

▼ Podies (5)

PRT0001.PRT

► MAIN







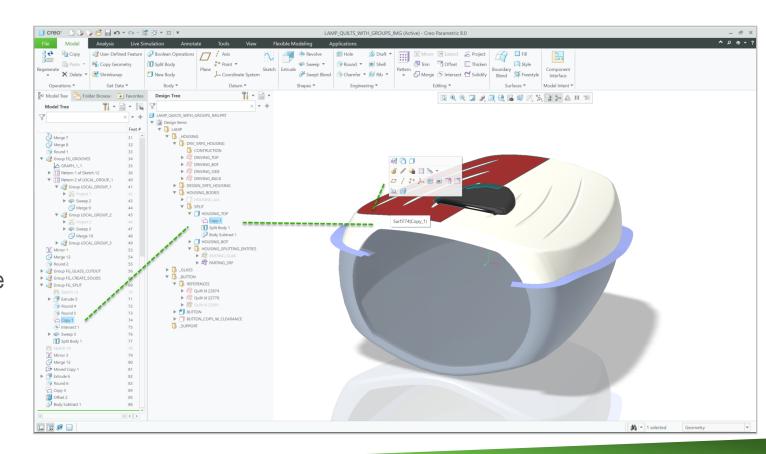
#### Creo 8.0 Enhancement Summary

#### Side-by-side positioning of Model and Design Tree

- Easy visualization of design features and design intent
- Secondary trees (i.e., layer tree, design tree) can be docked to various screen locations
  - Within or outside navigation bar
- Separate configuration settings and "Simple Search"
- Auto Locate in Tree
  - Design items in Design Tree And
  - Features in Model Tree

#### Limitations

- The Design Tree / Design Items folder is not available in assembly mode
- Display of the new "Design Tree" in side-by-side tree configuration is only supported in selected situations in part mode (not within the Flexible Modeling tab or in assembly and related modes)





Project

☐ Thicken

Mirror Extend

☐ Trim ☐ Offset

Editing \*

Swap

Details...

Merge Intersect M Solidify

#### Creo 8.0 Enhancement Summary

#### Action/Object Editing Workflows

- Improved Action/Object Workflows for "Editing Group"
  - i.e., Project, Trim, Offset, Thicken, Merge, Intersect, Solidify
- Ability to switch between feature "Type" during initial feature creation (i.e., Quilt, Curve)
- Increase ease-of-learning and ease-of-use through improved UI consistency

# Type Settings Quilt Curve Trimming object: Click here to add item Silhouette Trim References Options Properties

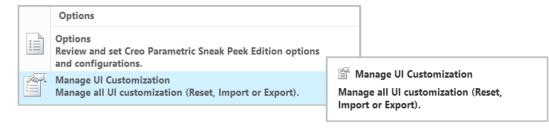
Trimmed quilt

Select 1 item

Trimming object Click here to ad

#### Import/Export Filters for the UI File

- New command "Manage UI Customization"
  - Easier administration of Creo Parametric customizations
    - Review the existing settings
    - Reset a range (or all) of the settings
    - Export a specific range (or all) of the settings
    - Import new settings with a control on what to import





# CORE MODELING MULTIBODY DESIGN

#### CORE MODELING - MULTIBODY DESIGN



#### Creo 8.0 Enhancement Summary

#### Multibody Design – Part Level BOM

- New command to generate part level BOM
  - List designated model items including body objects
  - Enables the designation of model items for exposure to Windchill

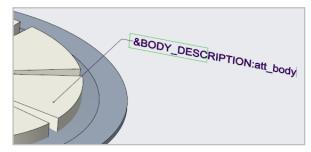
Note: The enhancement covers BOM control for designated items, not limited to multibody design

- Accessible in part environment from tools tab
- New "Part" selection option in Bill of Materials within assembly environment

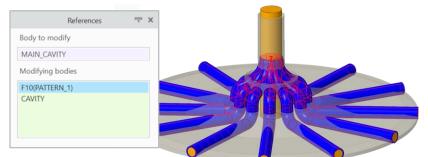
#### Pattern References in Boolean Operations

- Collectors in Boolean operations that allow selection of multibodies now support selection of geometry pattern feature and patterns of bodies
  - Boolean operations that involve body patterns are now parametric

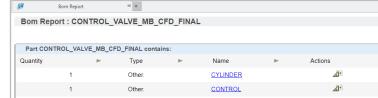
#### Support callout &<param\_name>:att\_body for notes attached to bodies

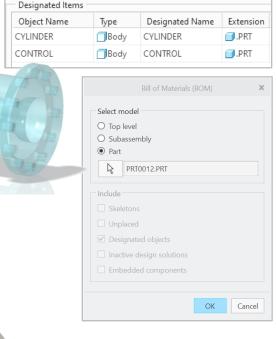














### CORE MODELING SHEET METAL

#### CORE MODELING - SHEET METAL

#### ptc

✓ Add miter cuts

3-bend corner relief type:

#### Creo 8.0 Enhancement Summary

#### Multi-Flat Wall Feature

- Fast sheet metal design with fewer flat wall features
- Multi Wall Support
  - Create multiple flat walls together based on a common definition
- Inner & outer corner detection & management
  - Intelligent default settings for corner and seam handling
- Contextual dragger placement

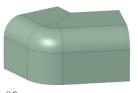
#### Enhanced Multi-Flat Wall Miter support

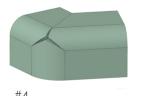
- Basic Miter support
  - For 3-bend corners in Flat Wall tool
  - Not yet for FMX tools
- Miter options
  - Trim walls and create cut
  - Control gap width and position
- Bend corner relief types
  - Tangent
  - Open
  - Closed
  - Rip

- 1. Tangent "tangent" corner opening
- 2. Open "minimum" corner opening
- 3. Closed closed corner area
- 4. Ripped corner area
- 5. Open with gap
- 6. Closed with gap
- 7. No miter cut















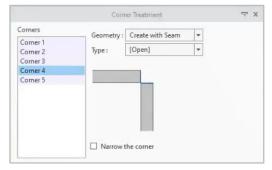
#### CORE MODELING - SHEET METAL

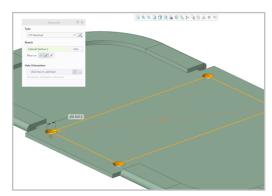


#### Creo 8.0 Enhancement Summary

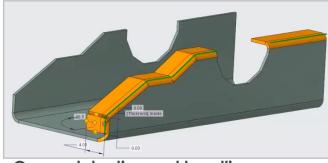
#### Sheet Metal Productivity Enhancements

- Automatic corner detection
  - Special handling for inner and outer corners
- Improved Bend relief for flushed hems:
  - Bend Relief Handling in Flange Wall works for all bend position settings
- New Lightweight Hole representation support for standard holes is available in Sheetmetal
- New Sketched Multi-Hole capability is available in Sheetmetal
- Selected Flexible Modeling design objects support cross-highlighting in the graphics
- User configurable default name for flat simplified representation
  - New configuration option User\_flat\_simp\_rep

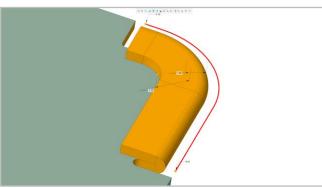




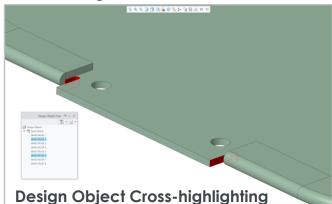
Sketched multi-hole in Sheet Metal



Corner detection and handling



Flushed hem with bend position offset and oblong bend reliefs





### CORE MODELING WELD DESIGN

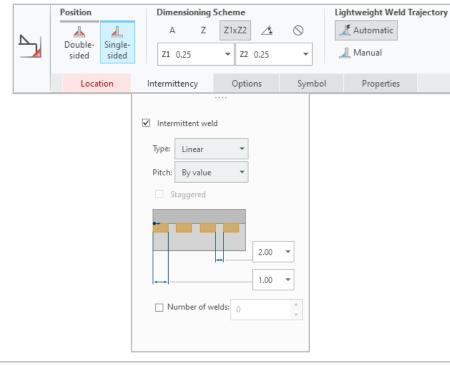
#### CORE MODELING - WELD DESIGN

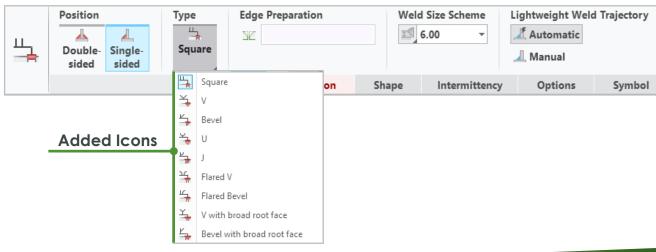
#### ptc

#### Creo 8.0 Enhancement Summary

#### Enhance Weld Design Dashboard

- Overhauled UI improves command organization and ease of use
  - Improved weld creation workflow and interaction
  - Simplified panels and dialogs present clear actions to user
- Fillet Weld Dashboard Update
  - Dim Scheme as radio group
- Groove/Butt
  - Dim Scheme as icons/images (previously just text)
- All Dashboards
  - Labels on single or double sided
  - More Labels for all respective weld types
    - Labels on single and double-sided welds
    - Edge Preparation and Weld Size
    - Lightweight weld trajectory
    - Referencing scheme
    - Labels on all respective controls
    - Updated images in shape slide out panels
  - Moved intermittent welds to slide out panel
  - Improved images in weld preferences





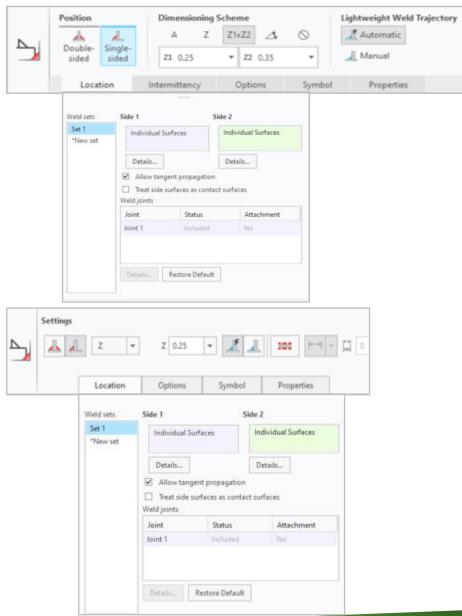
#### CORE MODELING – WELD DESIGN

#### Creo 8.0 Enhancement Summary

#### ISO Fillet Weld Improvements

- Z1 x Z2 dimensioning scheme is now supported
  - Provides the ability to define each leg length by value
  - Z1xZ2 dimensioning scheme was needed for compliance with ISO standard
  - ISO Fillet Weld improvements increase the usability of Creo Weld
- New intermittent panel for Fillet & Groove in both ANSI & ISO
- Shape tab in the Preferences dialog has new diagrams and same dim scheme UI as in the feature dashboard UI
- New parameter DEFAULT\_SCHEME for fillet weld
- New parameter LEG\_LENGTH\_2.
  - Used to store the value of the Z2 dimension in the Z1 x Z2 dim scheme
- Default value for A dimensioning scheme is greater than zero
  - Enhancement covers the typical ISO Fillet weld feature for companies that apply welds to their models in Creo
- Much clearer definition with labels





#### CORE MODELING - WELD DESIGN

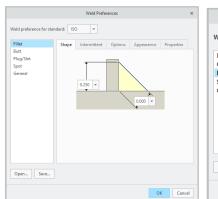
#### Creo 8.0 Enhancement Summary

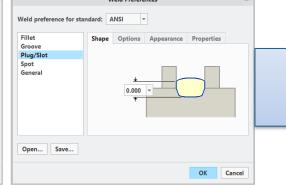
#### Intermittent Weld Enhancements

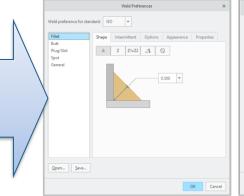
- New Slide Out panel for Intermittent welds for fillet and Groove/Butt welds
  - Weld panel dialog includes images to help the user clearly define weld specifications
    - i.e., pitch, number of welds, weld spacing
- Intermittent is added to Weld Preferences dialog
- More precise reporting for capturing parameter information
  - Improved report of other side numbers for staggered welds
  - Improved report for WELD LENGTH parameter for all weld pieces

#### Improved Weld Preferences

- Shape tab in the Preferences dialog has new diagrams and same dim scheme UI as in the feature dashboard UI
- Improved images in weld preferences

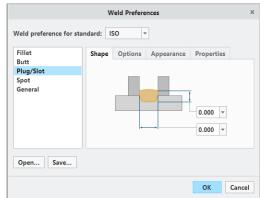






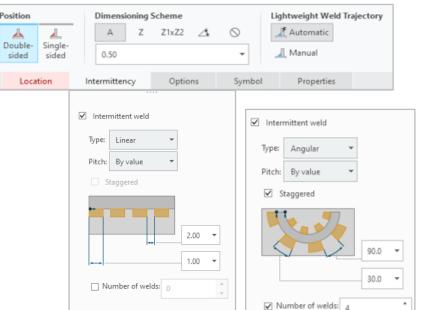
Position

sided



**Creo 7.0 Weld Preferences** 

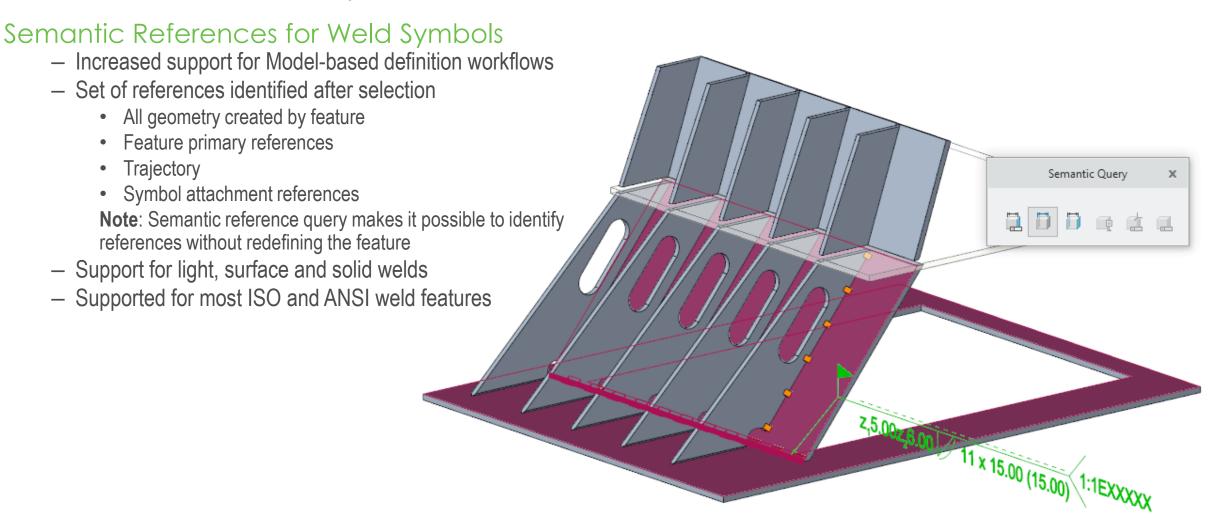
**Creo 8.0 Weld Preferences** 



#### CORE MODELING - WELD DESIGN



#### Creo 8.0 Enhancement Summary





# CORE MODELING WARP

#### CORE MODELING - WARP

Creo 8.0 Enhancement Summary

Tools

Bend

Transform

#### Overhauled Command Ribbon UI for Warp Feature

Stretch

 Style dashboards have been updated to conform with the new dashboard layout guidelines used throughout Creo

The new layout provides better use of space with an improved workflow

**Sculpt Settings** 

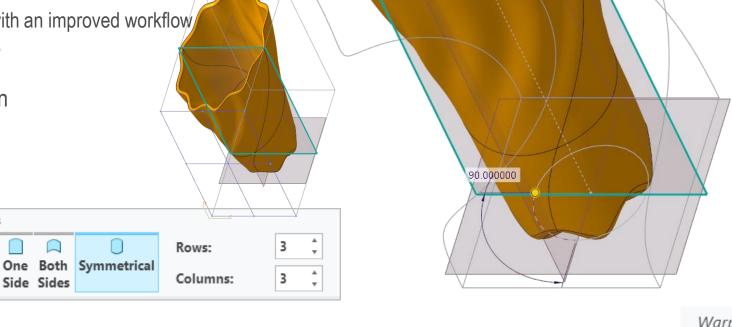
Next Axis

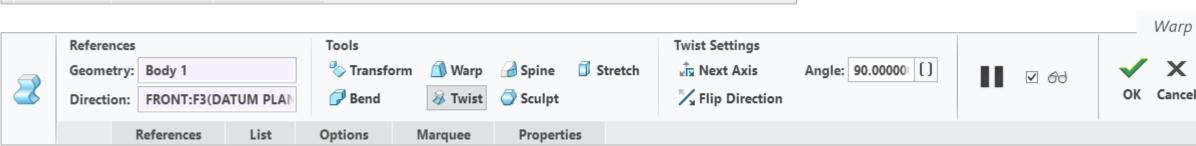
Side Sides

- Reduce the need to access sub-panels or tabs
- Improved tooltips, labels and icons

Marp Brine

Command workflow and command organization





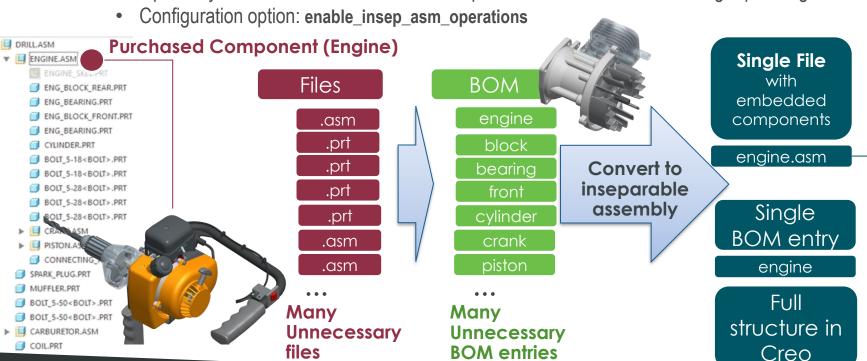


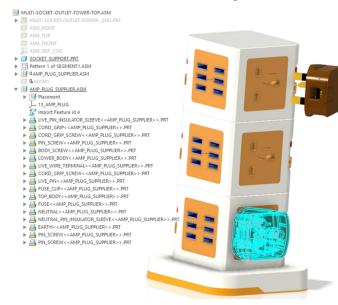


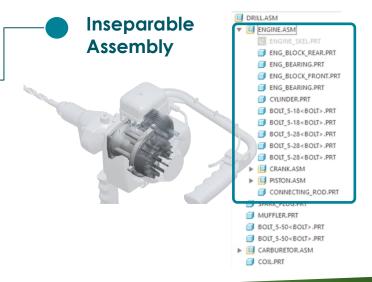
#### Creo 8.0 Enhancement Summary

#### New Inseparable Assembly

- Support for "purchased component assemblies" and embedded component concept
  - Embed component or make inseparable
  - Extract, make separable
  - Open/Import non-native embedded components
  - Create component as embedded
  - Optionally include/ exclude embedded components in Creo BOM & Drawing repeat regions





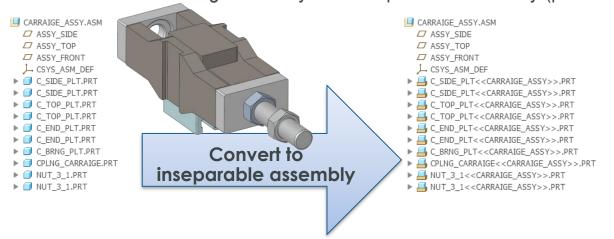




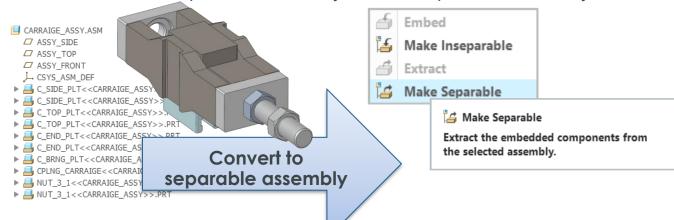
#### Creo 8.0 Enhancement Summary

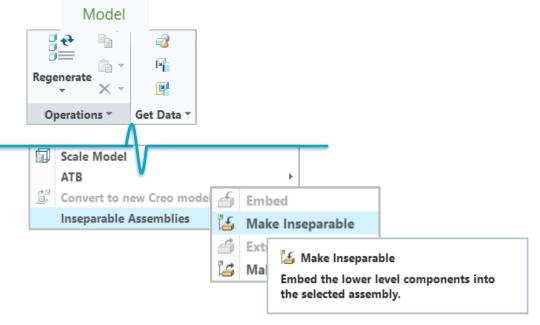
#### New Inseparable Assembly

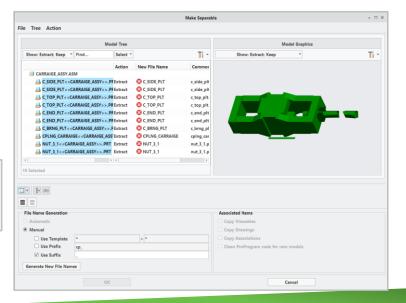
Convert existing assembly into inseparable assembly (partial or full)



Convert inseparable assembly back to separable assembly





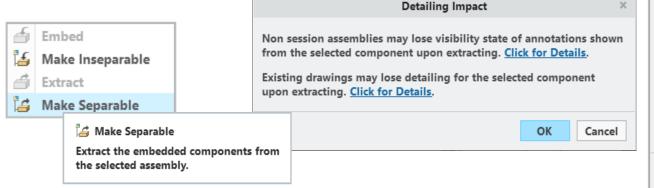


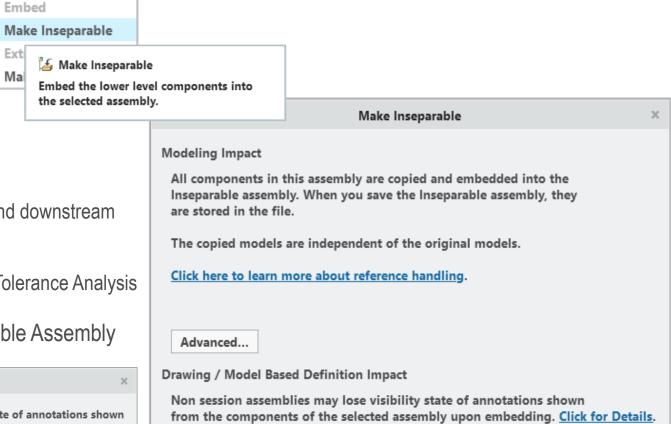


#### Creo 8.0 Enhancement Summary

#### New Inseparable Assembly

- Warning dialog when creating inseparable assembly
  - Embedded components are copies of the original models
  - Using copies of components can produce consequences due to reference handling
    - Refer to Help documentation for details
  - Additional consequences can impact modeling, drawing and downstream deliverables and applications
    - Drawing / Model Based Definition Impact
  - Embedded components are also not support by Creo EZ Tolerance Analysis and GD&T Advisor
- Additional Warning dialog when changing back to Separable Assembly





upon embedding. Click for Details.

and GD&T Advisor.

Existing drawings may lose detailing for the components of the selected assembly

Embedded components are not fully supported by Creo EZ Tolerance Analysis

Cancel

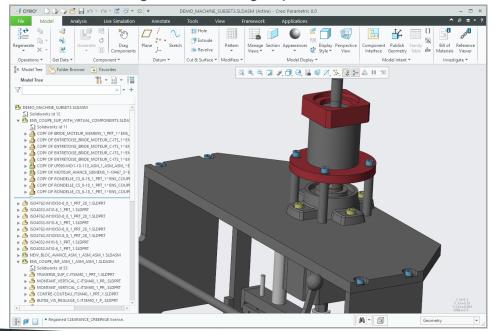
OK

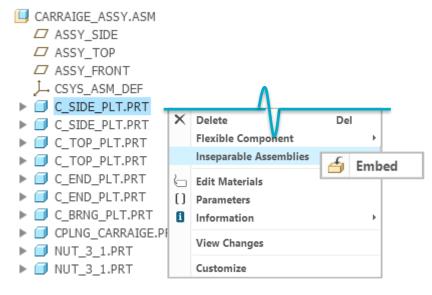


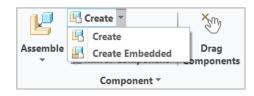
#### Creo 8.0 Enhancement Summary

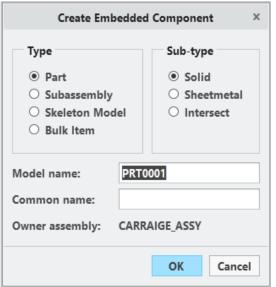
#### New Inseparable Assembly

- Convert assembled component to embedded component
  - Rapidly define Embedded components using RMB > Inseparable Assemblies > Embed
- Create "new" embedded component
  - Embedded component creation within the context of the assembly
- Ability to import and work with non-native files that support "inseparable assemblies"
  - Example: SolidWorks "virtual assemblies" are imported maintaining "embedded" design intent during import













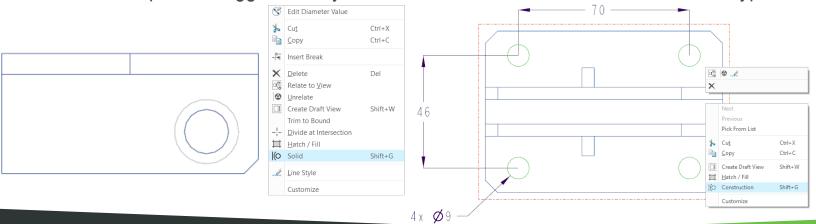
#### Creo 8.0 Enhancement Summary

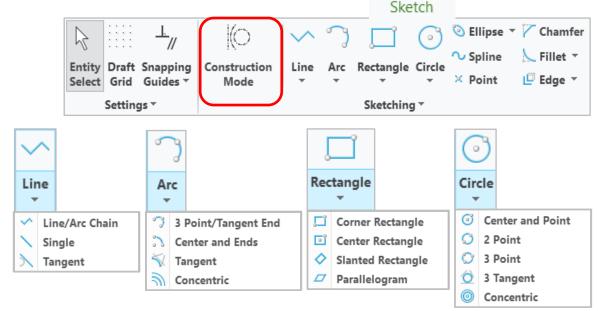
#### Modernized Sketching Tools

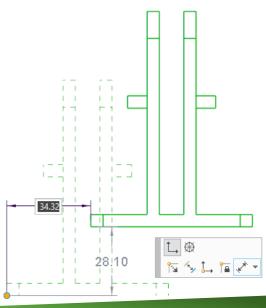
- Set of modernized drawing sketching tools for creation of non-parametric sketches
  - i.e., lines, arcs, rectangles, circles, ellipses, splines, points.
- Intuitive dragging behavior with on-the-fly dimension definition
- Dimension toolbar for setting dimension scheme and references.
- Access to legacy sketch is available

#### Improved Creation of Construction Entities

- Construction mode for sketching reference geometry with phantom lines
- New geometrical shapes that supports construction mode
- Shortcut menu option to toggle already sketched entities between construction/solid types









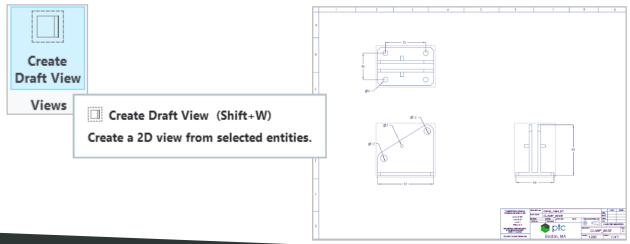
Creo 8.0 Enhancement Summary

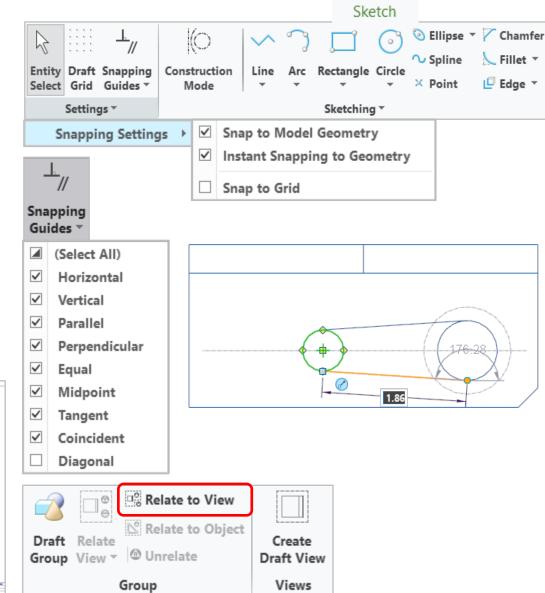
#### Control Over Geometry Snapping Guides

- Control over various types of snapping guides to draft entities or model geometry (config.pro based options)
  - Full control over snapping guides

#### Creation of 2D Views from Draft Entities

- New option to "Create Draft View"
  - Create a 2D draft view from sketched and imported entities
- Align draft views and define projection relationship between draft view and projection view
- Draft views can be scaled as required







#### Creo 8.0 Enhancement Summary

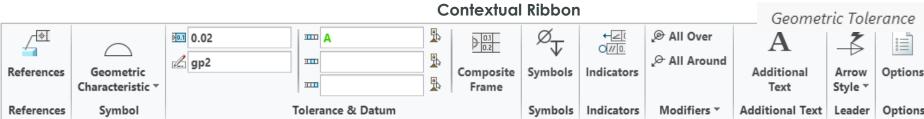
#### Symbols Modernization

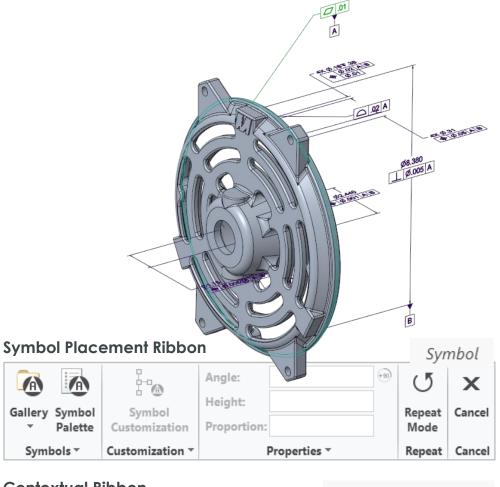
- Symbol Placement Ribbon
  - Easy browsing and selection of symbols from gallery or palette
  - Easily access to customize and input properties
- Immediate preview and placement options
- Contextual Ribbon provides easy editing of existing symbols
  - Symbol properties are automatically populated with references, properties, etc.

#### Streamlines Placement of Symbols

- Symbols can be placed automatically based on selected attachment type or using one of the special placement types from the shortcut menu
- The offered placement options are based on symbol definition attributes
- The 'With leader-free point' placement option can be added with the below config.pro option: symbol\_free\_with\_leader yes/no\*









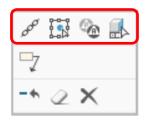
#### Creo 8.0 Enhancement Summary

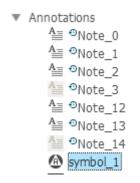
#### Contextual Command for Symbol Placement and Editing

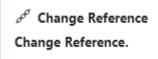
- Contextual commands have been added to mini toolbar and shortcut menu
  - Select symbol and RMB to access editing options from mini toolbar
    - Mini toolbar: Change Reference, Transform, Repeat, Semantic Query
    - Shortcut menu: Add leader, Update Symbol, Parameters, Designation

#### Improved Placement and Editing Workflows for Symbols

- Configuration Option: attach\_last\_symbol\_to\_cursor y/n
  - New Configuration option controls whether the last placed symbol is attached to the cursor when you place a new symbol
- Configuration Option: allow\_color\_change\_for\_symbols y/n
  - New configuration option allow\_color\_change\_for\_symbols defines whether the color command is active for the symbols placed in a model or drawing
    - Option is used to maintain/enforce company policy for symbol colors
    - This option should be added to the config.sup file.
- Configuration Option: symbol\_repeat\_mode y/n
  - N configuration option defines the default behavior for the repeat mode for a symbol
- Configuration Option: symbol\_free\_with\_leader y/n
  - New configuration option allows you to define if the With Leader-Free Point placement option is available in the shortcut menu when you place symbols in Drawing mode

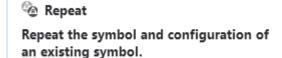


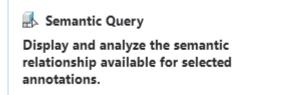


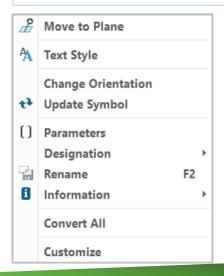


Transform

Apply scaling, rotation, and origin change.







All Symbols ▼

user symbols



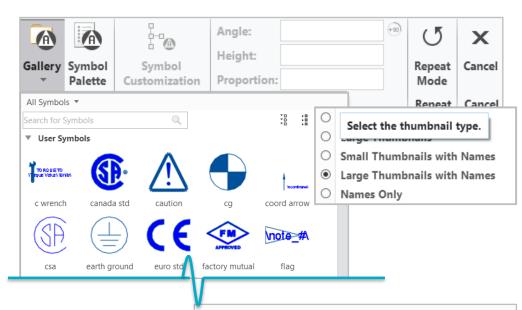
#### Creo 8.0 Enhancement Summary

#### New Gallery for Symbols

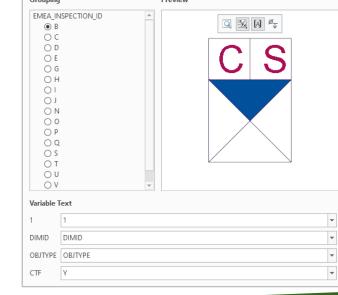
- New gallery layout for browsing all system/user defined symbols in one place
  - Each section is a separate folder in your symbols directory
  - Sections can be searched and collapsed
- Customizable display options
  - Option to change size and display name
  - Default thumbnail type is defined with a config.pro: default\_symbol\_thumbnail\_type
- Ability to pin favorite symbols for easy access and future use

#### **Enhanced Symbol Grouping Customization**

- Single detachable panel shows all relevant customization options and variable text input
  - Easy customization of symbols and symbol instances that have already been placed
  - Context based expansion of variable text section
  - Enhanced preview section with visibility controls





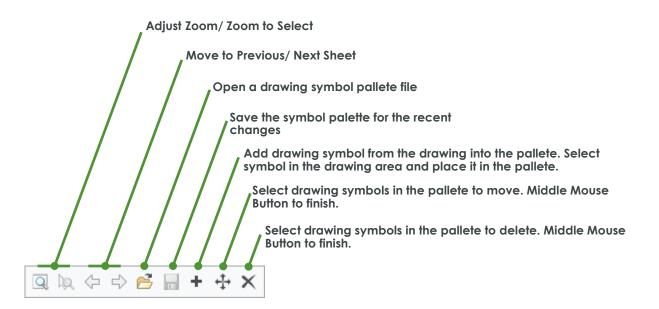


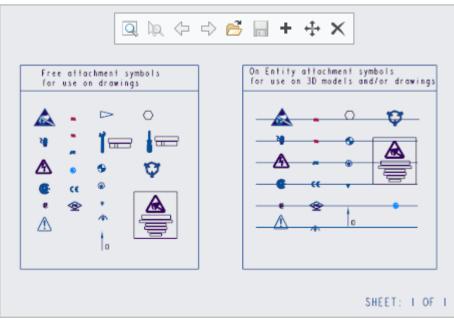


#### Creo 8.0 Enhancement Summary

#### Improved Symbol Palette Interaction

- The new in-graphics toolbar allows you to easily navigate and edit the symbol palette
  - Zoom to fit and zoom to selected
  - Navigation of multi-sheet drawing palettes
  - Easy editing of the symbols that are included in the palette
  - The configuration option: restricted\_symbol\_palette allows the CAD Admin to define whether the users are able to modify symbol palette







Creo 8.0 Enhancement Summary

#### Semantic Definition Support for Symbols

Reference collector for standalone symbols can collect multiple semantic references

- Attachment reference is automatically added to the reference collector
- Symbols are now supported by the semantic query tool

#### Semantic References for Weld Symbols

Semantic references of weld features are propagated to its weld symbols

 Primary references as well as the resulting welding geometry surfaces, are transferred and highlighted on query



References

References

0.02

∠ gp2

Geometric

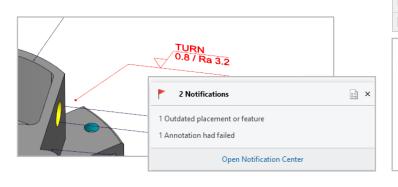
Characteristic \*

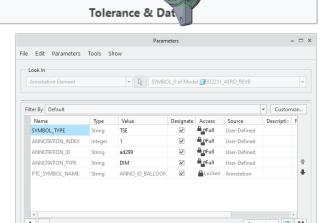
Symbol

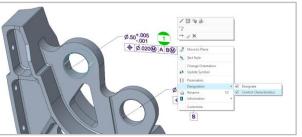
#### Enhanced Standalone Annotation capabilities for Symbols

Standalone symbols supports the creation of associated parameters

- Symbols can be designated as control characteristic to Windchill
- Issue for failing references of symbols are reported in the notification center
- Symbols can be set to be excluded when their parent reference is suppressed, excluded or deleted







0.1

Composite



#### Creo 8.0 Enhancement Summary

#### Advanced Surface Collection for Annotations

New "Surface sets" tab in reference collector of standalone annotations

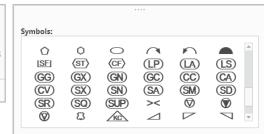
 Quick collection of semantic references using the following advanced methods:

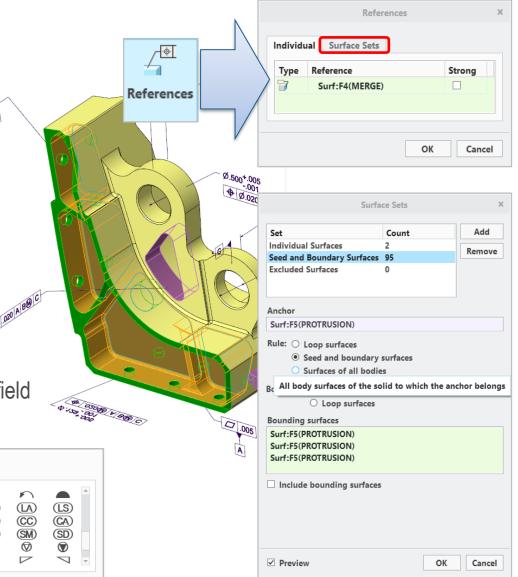
- Surface of all bodies
- Seed and bounds
- Loop surfaces
- Intent surfaces
- Exclusion of specific references from a set
- Supported by all types of standalone annotations (except surface finish)

#### Text Symbols Pallet for Datum Feature Symbols

- Text symbols can be easily added to datum feature symbol's additional text field
  - Improved compliance with detailing standards









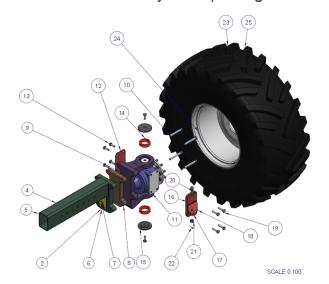
#### Creo 8.0 Enhancement Summary

#### New Leader Note Callout for Bodies

- Values of user defined body parameters can be called out into leader notes
- The callout is reported for the attached body
- Note callout syntax: "&<parameter\_name>:att\_body"

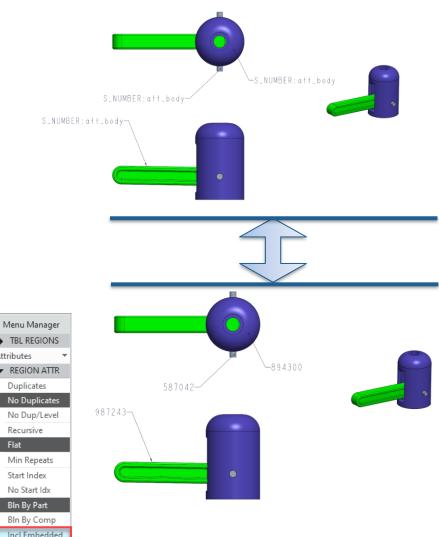
#### Repeat Region Support for Listing Embedded Components

- Users can choose if they want to include/exclude embedded components from their repeat region tables
  - Enhanced flexibility for reporting embedded components in drawings



NO	TTPE	PART NAME	PART NUMBER	OIT
	ASSEMBLY	AXLE_WITH_TIRES	531000	- 1
2	ASSEMBLY	AXLE_ASSEMBLY	532000	1
3	ASSEMBLY	CONNECTING_BAR	532010	I
4	ASSEMBLY	BAR_ASM	532003	1
5	PART	INTERNAL_BAR	230012	1
6	PART	FIRST_BRACKET_AXLE	532001	1
7	PART	RIB	532002	2
8	PART	SECOND_BRACKET_AXLE	532004	1
9	PART	THIRD_U_BRACKET	532006	1
10	PART	MFHS08109	204040	4
$\Box$	PART	AXLE_HOUSING	138603	1
12	PART	SIDE_HOUSING_BRACKET	138604	1
13	PART	MFHS08101	204050	6
14	PART	SEAL_ADAPTER	204060	2
15	PART	SEAL	204070	2
16	ASSEMBLY	TOP_HOUSING_BRACKET_ASM	532005	1
17	PART	TOP_HOUSING_BRACKET	532006	1
18	PART	STEERING_CONNECTOR	532007	1
19	PART	MFHS08102	104012	4
20	PART	CONNECTING_PIN	104015	1
21	PART	CASTLE_NUT	104088	1
22	PART	SPLIT_PIN	104089	I
23	ASSEMBLY	WHEEL_ASSEMBLY	532008	
24	PART	WHEEL < < WHEEL _ ASSEMBLY >>	N / A	T I
25	PART	RUBBER_TIRE< <wheel_assembly>&gt;</wheel_assembly>	N/A	

No Embedded Cable Info No Cbl Info





Creo 8.0 Enhancement Summary

#### Improved UX for Varied Items Table

 Enhanced understanding when working with Flexible Components and Inheritance Features

- Easily identify varied item types and quantities
  - Dialog Name shows total quantity
  - Tabs show Type and Quantities (i.e, dimensions, features)
- Cross highlighting of varied items
  - Selecting varied item from table cross-highlights varied item in model tree and graphics area

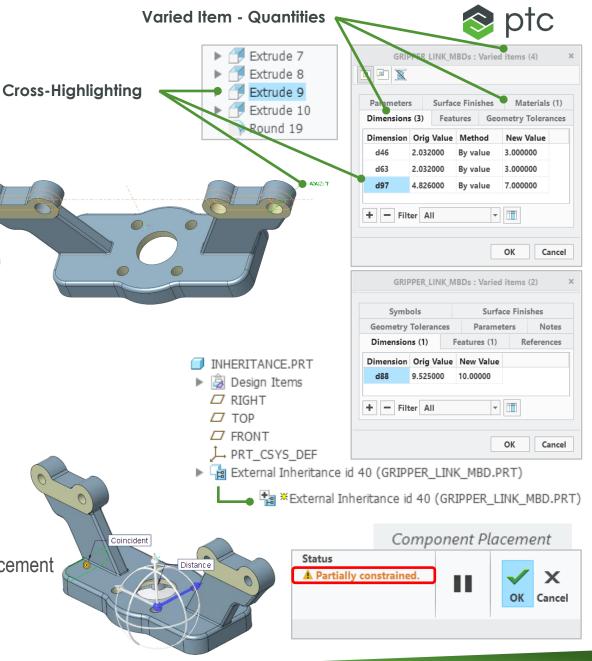
#### New Inheritance Icon Modifier

- Inheritance Model Icon updates to indicate the addition of Varied Items
  - Clear understanding of varied items without explicit action

#### Component Placement Status in Dashboard

- Placement Status now provides detailed placement information
  - Status indicates what's needed or required to redefine component placement (i.e., missing or invalid placement references)

**Note**: Core functionality (does not require Creo AAX)



#### Creo 8.0 Enhancement Summary

#### Expansion and Improvement of Automatic Rep Support

- Smoother workflows working with Automatic Reps (lighter and faster)
- Support for Mass properties and Interference analysis
  - On-demand retrieval of data recognized by system
  - For interference checks, the system performs a "rough" check and then performs on-demand retrieval of the required objects

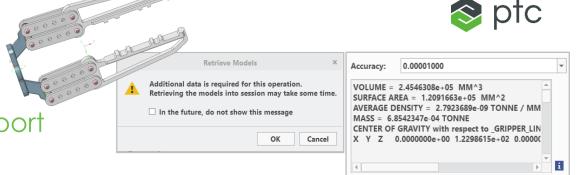
#### Enhanced External Copy Geometry (from Assembly)

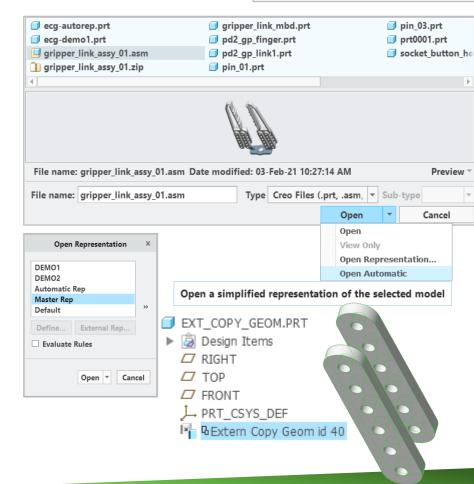
- Create External Copy Geometry from an assembly representation
  - No need to open the master assembly model
  - Open a User Defined or Automatic Rep as the source of reference geometry
- Redefining the External Copy Geometry will perform on-demand retrieval of reference geometry
  - Redefine does not require assembly retrieval, only reference objects
  - Easily redefine reference geometry

**Note:** If during redefinition any simplified representation is in session, the Open Rep dialog will be presented

- Ability to change "scope" during redefine
  - Remove existing reference, add new reference object (component) from secondary model tree







#### Creo 8.0 Enhancement Summary

#### Enhanced External Shrinkwrap (from Assembly)

- Create External Copy Geometry from an assembly representation
  - No need to open the master assembly model
  - Open a User Defined or Automatic Rep as the source of reference geometry

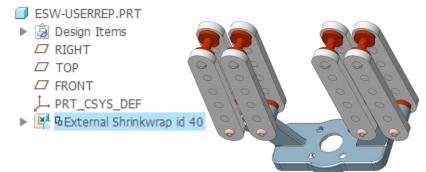
Open a simplified representation of the selected model

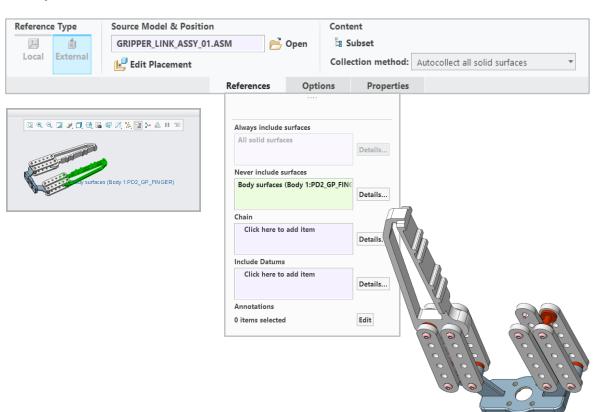
- Redefining the External Copy Geometry will perform on-demand retrieval of reference geometry
  - Redefine does not require assembly retrieval, only reference objects
  - Easily redefine reference geometry

**Note:** If during redefinition any simplified representation is in session, the Open Rep dialog will be presented

- Ability to change "scope" during redefine
  - Remove existing reference, add new reference object (component) from secondary model tree







#### Creo 8.0 Enhancement Summary

#### Reference Model Access

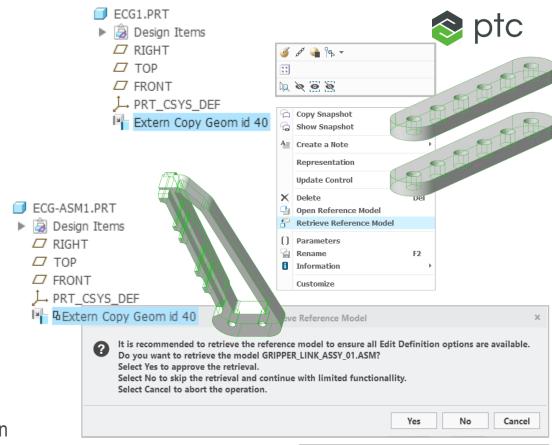
- Aligned reference model access for Data Sharing Features
  - Open Reference Model
  - Retrieve Reference Model
  - (External) Merge/Inheritance
  - (External) Copy Geometry
  - Shrinkwrap
- Easy access from the RMB menu

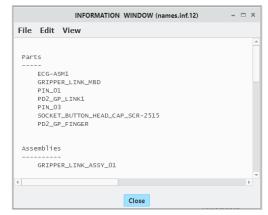
#### **External Copy Geometry**

- When referencing a component through an assembly
  - Open reference model will only bring the minimum required data, objects in the referenced path
  - Retrieve reference model will only bring the minimum required data, objects in the referenced path

#### External Shrinkwrap

- Open reference model will open the assembly in automatic representation
- Retrieve reference model will only bring the minimum required data, objects in the referenced path





Creo 8.0 Enhancement Summary

#### Publish Geometry Feature

- Publish Geometry feature is created in the Footer by default
- Aligning Publish Geometry creation with the feature's evaluation, by creating it in the Footer by default



Published Geometry id 14215

#### Legacy Publish Geometry can be manually added to the Footer

- Once in the Footer they cannot be moved out
- If needed it can be dragged into the Model Tree
  - keep\_publish\_geom\_in\_footer yes\*/no







## ADVANCED FRAMEWORK (AFX)

## ADVANCED FRAMEWORK (AFX)



#### Creo 8.0 Enhancement Summary

#### Support for Embedded Component Workflows

Smooth experience with AFX when using embedded components

#### **Limitations:**

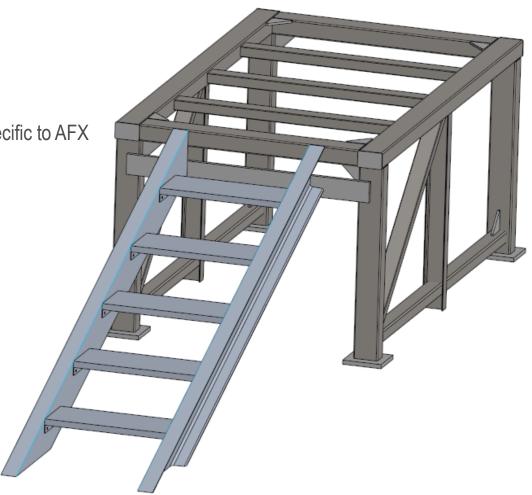
You cannot copy embedded components for reuse

• This is a limitation for embedded component functionality, not a limitation specific to AFX

- Pro/Program is not supported
  - As a workaround, users can manually resume/suppress features that are controlled by Pro/Program

#### Improvements to Copy Component Definition

- Better control when naming copies
- Copy Component Definition dialog box now also shows all the framework components in the assembly in tree form
  - In prior releases, the Copy Component Dialog did not show all components in the framework if multiple instances existed
- Improve navigation when selecting components to copy





## PIPING AND CABLING (PCX)

PIPING AND CABLING (PCX)



**■** ◎ 数 60

#### Creo 8.0 Enhancement Summary

#### Branch Tape Feature

- Represents an electrical tape wrapped around wires and junctions
  - A branch tape can be wrapped around wires, cables, bundles, and stiff shrinks
- Specify an overlap factor parameter that defines a width % for each wrap of a tape
- Use a tape spool to define the following parameters:
  - Length of wrapped tape on one bundle, Color, Thickness, Tape width, Tape thickness and Mass per length

#### Stiff Shrink Feature

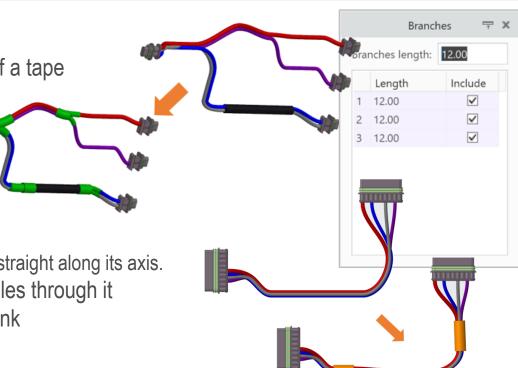
- Stiff Shrink sheath feature allows user to route multiple cables and wires through it
  - Unlike the Bundle feature, the Stiff Shrink feature does not bend and remains straight along its axis.
- Stiff Shrink feature is created using coordinate system and routing new cables through it
- Alternatively, you can select an existing location, and then place a Stiff Shrink feature on all cables that go through that location

#### Slots in ECAD Boards

- Create, edit, import, and collaborate on ECAD cuts and slots (slot is a specific type of ECAD cut with an attribute)
  - Option to import and export ECAD cuts and slots with an IDX file (EDMD version 3 and above)
  - Slots are imported as a separate cut feature along with attributes that identify them as ECAD Slots
  - Changes to a cut or a slot are fully supported in ECAD-MCAD collaboration (getting, proposing, and sending changes).

Placement

Location: P 1 item(s)



Properties

**Tape Spool Settings** 

Spool: TAPE\_SPOOL
Overlap Factor: 0.20

Branches



## DESIGN EXPLORATION EXTENSION (DEX)

### DESIGN EXPLORATION

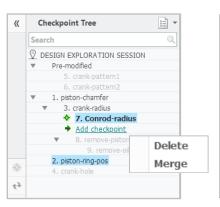
### ptc

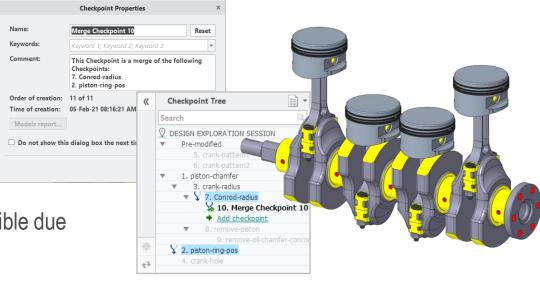
#### Creo 8.0 Enhancement Summary

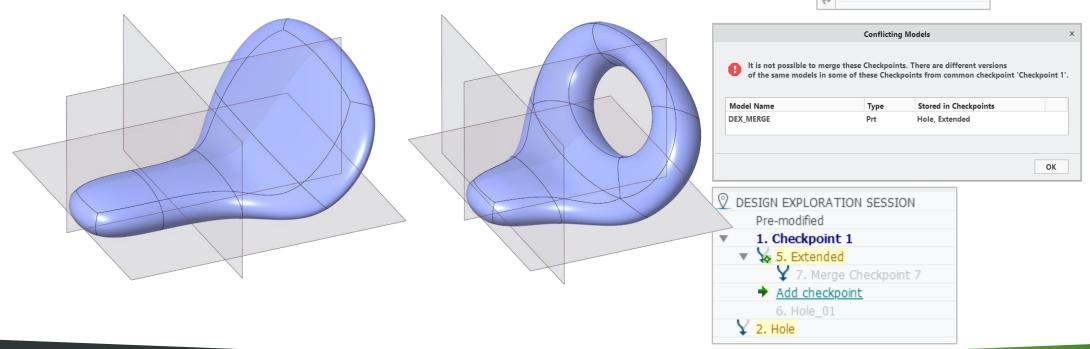
#### Merging of Checkpoints

- DEX checkpoint handling now allows merging
  - 2 or more check points can be merged
- Visual indication in the Checkpoint Tree
- More flexibility using Design Exploration

System will provide "Conflicting Models" warning when "merge" is not possible due to conflicting models









## INTERACTIVE SURFACE DESIGN (ISDX)

## INTERACTIVE SURFACE DESIGN (ISDX)



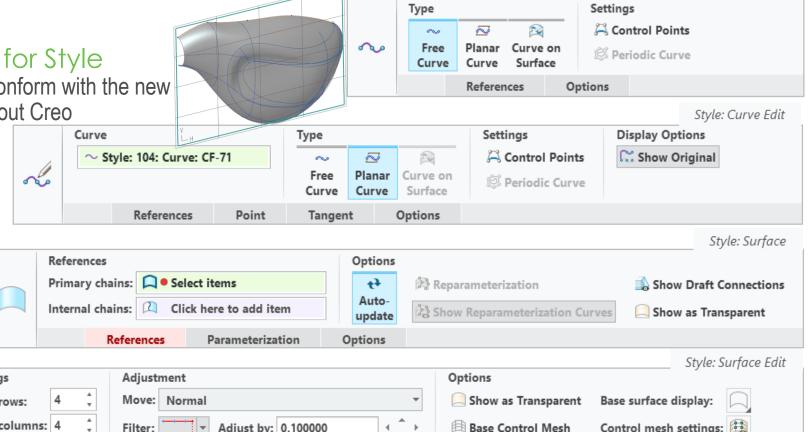
Style: Curve

Creo 8.0 Enhancement Summary

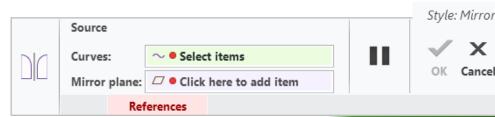
#### Overhauled Command Ribbon UI for Style

 Style dashboards have been updated to conform with the new dashboard layout guidelines used throughout Creo

- The new layout provides better use of space with an improved workflow
- Reduce the need to access sub-panels or tabs
- Improved tooltips, labels and icons
- Command workflow and command organization







Cancel





#### Creo 8.0 Enhancement Summary

#### Support for Creo Process Manager

- Creo Process Manager list all optimization (single or multiple) performed in the current session of Creo
- Process Manager notifies user when optimization are started, finished or failed
- Opening the Creo Process Manager will show the status for each optimization and provide additional commands to the user
  - Pause/Resume, Stop, Open Study or Dismiss



## Structure\_Study\_1 | TUTORIAL\_DESIGN\_REDEFINE.PRT In progress | 0% completed | Started: 15-Jan-21 02:21:15 PM | Elapsed time: 0h 0m 10s Structure\_Study\_1 | BRACKET\_01\_DRAFTED\_CREO\_KEEPIN.PRT In progress | 90% completed | Started: 15-Jan-21 02:20:59 PM | Elapsed time: 0h 0m 25s

Creo Process Manage

1 Process in session

Structure\_Study\_1 |

#### Supports for Asynchronous Processing

- Users can now work on other models in the Creo session while an optimization is running
- Switching to another Creo window will not pause or terminate a current optimization
- Creo Process Manager is used to check the optimization status and open a study



#### Creo 8.0 Enhancement Summary

#### **Automatic Envelope**

- Generative Design (GTO) will automatically create the Starting Geometry envelope based on the preserved geometry defined in the optimization setup
  - Automatic envelope is ONLY created when the starting geometry hasn't been manually defined
  - Automatic envelope is displayed in the graphics window as a transparent boundary body offset from the preserved geometry
    - Users are not able to control or modify the size of the auto envelope
  - Users can only select suitable mass units for the design goal in the design criteria
  - A new body will be created for the auto envelope as part of the reconstruction process
- Additional "exclude bodies" may be needed to define the optimization
- Automatic envelope increases engineering productivity by removing the need to create/define the starting geometry
  - Faster exploration of design alternatives
  - Eliminate time required to manually define starting geometry







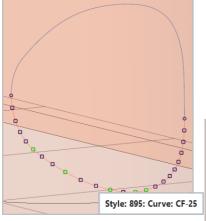




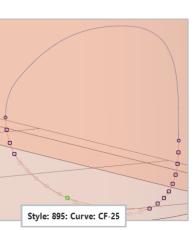
#### Creo 8.0 Enhancement Summary

#### Improved Usability of "Heavy" Style Curves

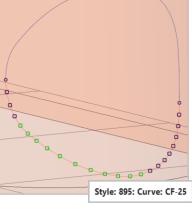
- Style curve editing has been improved to speed and simplify the editing of boundary curves that are generated during the reconstruction process
  - Editing style curves allows the user to modify the defining reconstruction curves to change topology of reconstruction and resolve reconstruction failures
- Easily edit "heavy" style curves by removing unwanted points
  - Select and delete curve points (single or multiple) by pressing the delete keyboard button
  - Use 'from-to' selection using the Shift key to select a range of points between the starting and end locations for point selection
- Soft-snapping of curve points allows user to snap curve point locations back to the original curve that is shown as 'ghost curve'
- Curve editing enhancements are supported in Generative Design and Style (Creo ISDX)
  - Curve editing capabilities apply to Style curves created during reconstruction in Generative Design and Style curve editing in Creo ISDX



Delete Selected Points



Snap points to "ghost curve"



"From-To" Point Selection



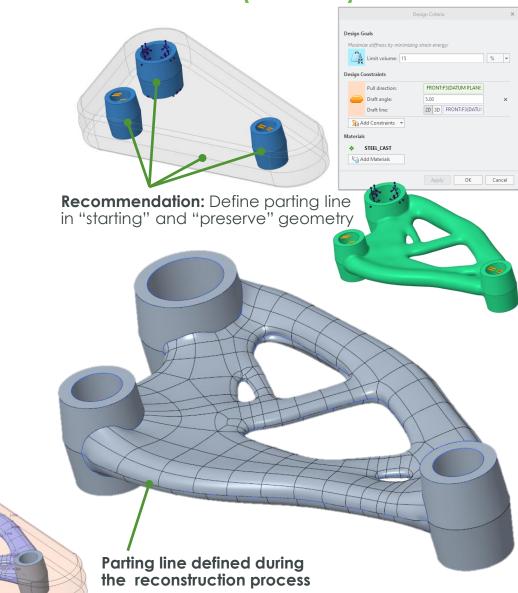
Creo 8.0 Enhancement Summary

#### Improvements to Reconstruction Result

Ongoing development and support for B-Rep reconstruction

#### Reconstruction Improvements for Parting Line Constraint

- The reconstruction process has been improved for studies including Split Constraint
- Creo will create a defined parting line during the reconstruction
  - **2D Parting Line**: A Planer Style Curve through the intersection of the Optimized shape and the draft plane
  - **3D Parting Line**: A Free Style Curve through the computed parting line from the solver
- Freestyle surfaces are created on either side of the parting line and aligned to the parting line with a defined draft angle
  - Local draft alignment for each curve at the boundary
  - Users can modify the angle when editing the freestyle feature, but for global change, user should re-run the optimization
- To improve reconstruction success rate, it is recommended users draft the preserved geometry
  - This draft feature is used to avoid needing Freestyle to create geometry around the preserved bodies

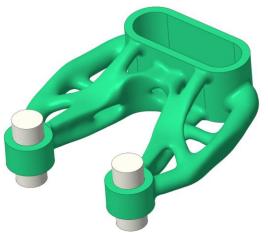




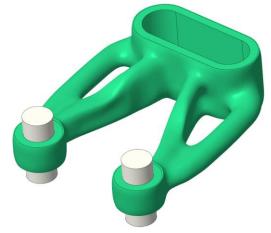
#### Creo 8.0 Enhancement Summary

#### Minimum Crease Radius

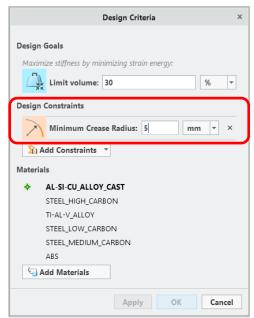
- New Minimum Crease Radius design constraint has been added to the Design Criteria
  - Minimum Crease Radius constraint smooths the optimization result and reduces the webbed topology by maintaining surface curvature as defined by the minimum crease radius
    - New constraint provides additional control to explore design alternatives
  - New constraint is added to the Geometric Constraint list
    - Selecting the option prompts the user to set the minimum crease radius value and unit
    - The entered value should be greater than the minimum element size

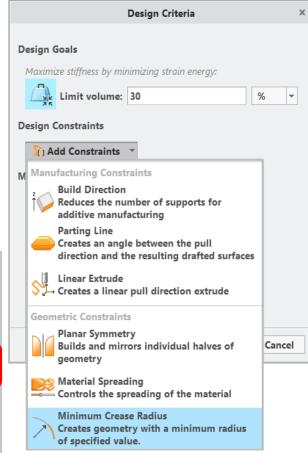


No Minimum Crease Radius



Minimum Crease Radius = 3 mm







## RENDER STUDIO

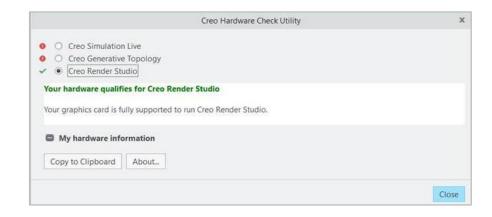
### RENDER STUDIO



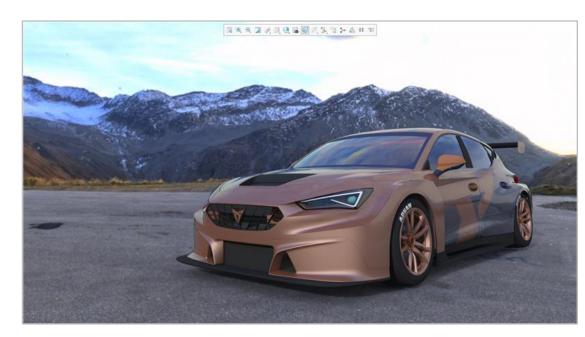
#### Creo 8.0 Enhancement Summary

#### Render Studio now supports Keyshot 9.3 rendering engine

- Users can now leverage GPU processing which will provide improved quality and performance needed for greater realism with real-time rendering
  - A new icon called '**Denoise**' is available in ribbon when in GPU mode which removes the noise from the rendering to speed up the rendering workflow
- Creo will automatically select GPU mode if a supported NVIDIA RTX graphics card and appropriate driver are available
  - Without the supported hardware, Creo will default to CPU mode
    - PTC Hardware Check Utility has been updated to check hardware for Creo Render Studio to support GPU mode
  - Users can control the graphics mode through the config option: advanced\_rendering\_mode









## GD&T ADVISOR

### **GD&T ADVISOR**

### ptc

#### Creo 8.0 Enhancement Summary

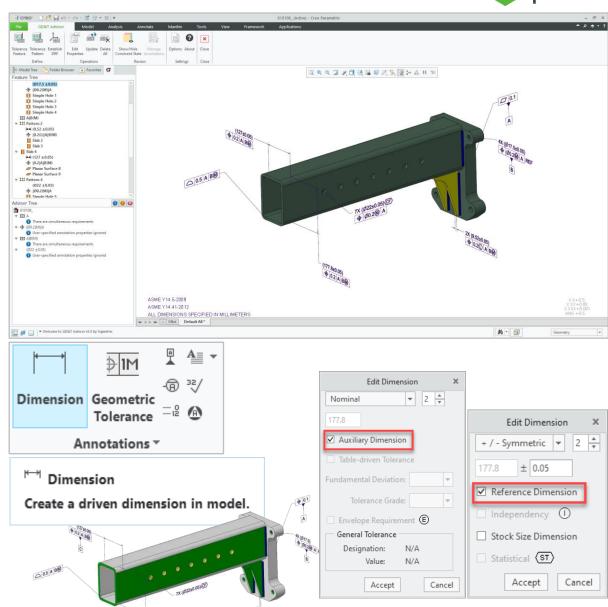
#### GD&T Advisor Support for Assemblies

- Quick creation of syntactically correct, feature centric GD&T annotations for assemblies
- Validates existing Creo annotations
- Extensive help content and advisor messages to educate the user on the state of the applied GD&T
- Color based constraint state mode

Note: Requires GD&T Advisor Plus license

#### **Enhanced Dimension Creation Workflow**

- Dimensions can be created easier when working with GD&T Advisor
  - Prior to Creo 8.0, user was required to exit GD&T Advisor, select the annotate tab and create the dimension
  - New workflow does not require use to close GD&T Advisor
- Dimensions can be indicated as reference/auxiliary type when required (for assembly mode only)



#### GD&T ADVISOR



Flexible Modeling

#### Creo 8.0 Enhancement Summary

#### Mismatched Annotations Manager

New tool for reporting and managing the updates to annotations

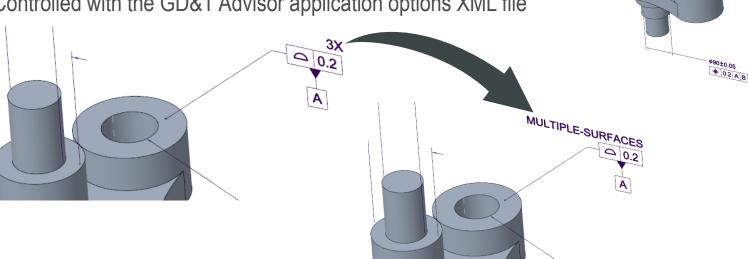
Review updates identified by GD&T Advisor and choose what update are applied

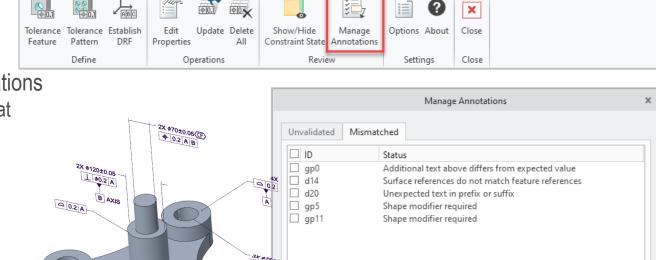
- Full report of mismatched annotations
- Visibility and control over expected updates

#### Count of multi-surface Feature Annotations

 Users can choose whether multi-surface features will report the count of surfaces or will report a custom text

Controlled with the GD&T Advisor application options XML file





Annotate

Tools

Refresh List

View

Model

Analysis

GD&T Advisor

Close



# ANSYS POWERED SOLUTIONS CSL AND CREO ANSYS SIMULATION

## CREO ANSYS SIMULATION

## ptc

## Creo 8.0 Enhancement Summary

## Solver Settings Reset

Reset option has been added to the Solver Settings dialog box

## Control for large deflection (off or on)

- Option to control Large Deflection added to the Solver Settings dialog
  - CSL includes large deflection by default when condition is detected
  - Users can now turn the option off when desired

#### Automatically grabbing of CAS license

Eliminate need to manually acquire license (File > Options > License menu)

#### Reset Mesh settings to default

- Mesh control settings are now recorded in the Simulation Tree
  - Deleting mesh controls returns to mesh control to default values
  - Changing mesh resolution removes all mesh control settings

## Mesh diagnostics in Process Manager

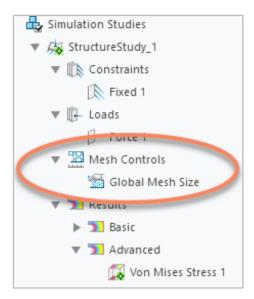
Number of nodes and elements reported

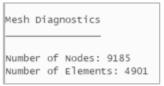
#### dcsodat > casdat

PTC specific format introduced to export CAS data to Ansys Mechanical

#### Automatic contact detection Enhancement

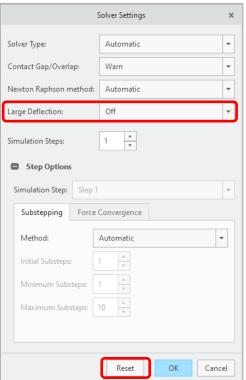
Skip for models with one solid body











**Note**: Creo Ansys Simulation has defaults for all simulation types (modal, structure and thermal)

"Reset" option makes it easy to return to default settings

## CREO SIMULATION LIVE (CSL)



## Creo 8.0 Enhancement Summary

#### Ability to Redefine Probes

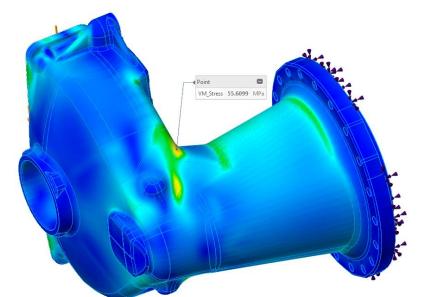
- New functionality to redefine probes
  - Previous functionality required user to delete and recreate probes
- Easily redefine and location and probe type

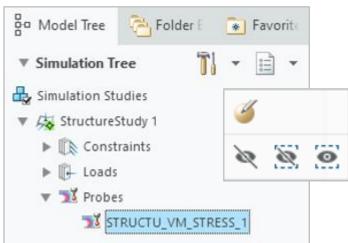
## Removal of Selection Highlighting

- Selected surface does not highlight
  - Easier visibility of selected area
- No more highlighting of surface when defining prope
  - Fringe is completely visible when selecting location

## Upgrade to latest Ansys 2021 R1 release

- Upgrade impacts CSL and Creo Ansys Simulation
- Improved robustness and solving using latest release
- No UI differences







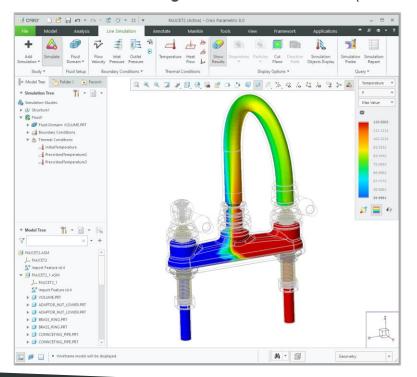
## CREO SIMULATION LIVE PLUS (CSL)

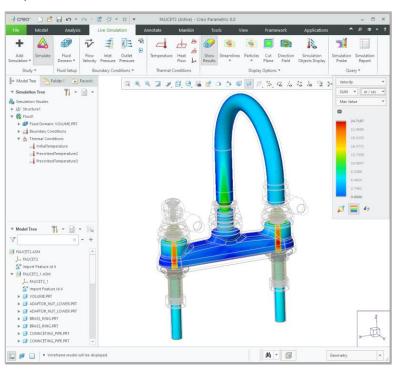


## Creo 8.0 Enhancement Summary

### Steady State Fluids

- New functionality to support steady state fluids in CSL CFD
  - Steady state is default for CSL Fluids Study
- User can switch to transient study if desired time dependent solution
- All probes, legend values and results switch when changed between steady state and transient
- Works with existing CSL Fluids license (CSL Plus)







## SUBTRACTIVE MANUFACTURING

## SUBTRACTIVE MANUFACTURING SOLUTION SUITE



#### Pre-Creo 8.0 Subtractive Manufacturing Solution suite



Specialized capabilities to speed 3 axis mold machining

#### Creo Prismatic & Multi-Surface Milling



Multi-Surface 3-axis milling with highspeed machining support

## Creo Production Machining Extension



Added support for 4-axis turning and

## Creo Complete Machining Extension



2.5- to 5-axis milling, multi-axis turning, multi-task machining and 4-axis wire EDM

#### **Creo NC Sheetmetal**



NC programming for turret punch presses, contouring laser/flame machines, nibbling and shearing

#### **Creo 8.0 Solution Suite**

## Creo High Speed Milling (3-axis)



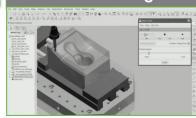
New Offering supports 3-axis highspeed machining

## Creo High Speed Milling Plus (3 to 5-axis)



Specialized capabilities to speed 2 and 5-axis high speed machining

#### Creo Prismatic & Multi-Surface Milling



Multi-Surface 3-axis milling with highspeed machining support

## No Changes



Added support for 4-axis turning and wire EDM

# Creo Complete Machining Extension

2.5- to 5-axis milling, multi-axis turning, multi-task machining and 4axis wire EDM



NC programming for turret punch presses, contouring laser/flame machines, nibbling and shearing

## Creo High Speed Milling Plus replaces Creo Mold Machining Extension

- Same part #, adds additional HSM capabilities to support simultaneous 5-axis and 3+2-axis high speed machining

## **Creo High Speed Milling (New Offering)**

Specialized capabilities to support 3-axis high speed machining

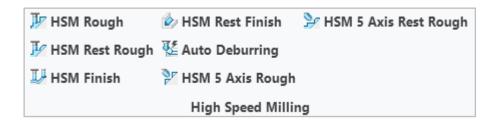
## 5-AXIS HIGH SPEED MACHINING (HSM)

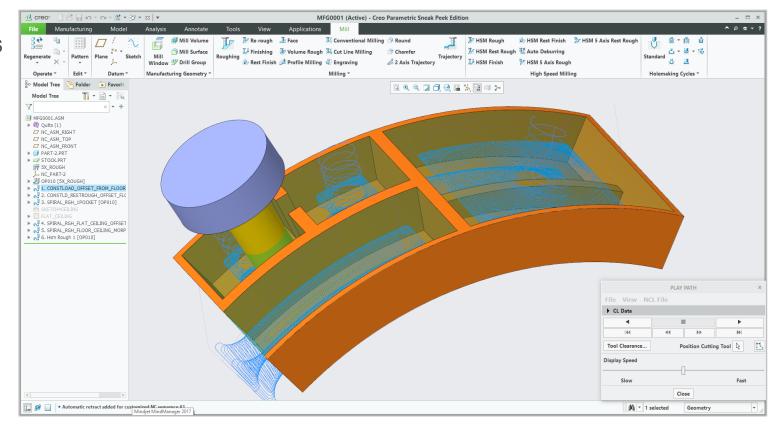


## Creo 8.0 Enhancement Summary

## Creo High Speed Milling Plus

- 5-axis dedicated Creo machining module that simplifies 5-axis programming
  - Solution builds on the 3-axis HSM machining capabilities introduced with Creo 5.0
- Easy-to-use simultaneous 5-axis and 3+2-axis collision free toolpaths
  - Users with 3-axis HSM machining experience will find it easy to learn and use
- Functionality supports the HSM of mold & die and production parts
- Solution complements existing 5-axis Creo finish toolpaths in the Production and Complete Machining packages





## CREO HIGH SPEED MACHINING PLUS



## Creo 8.0 Enhancement Summary

## 3 New 5-axis Toolpaths

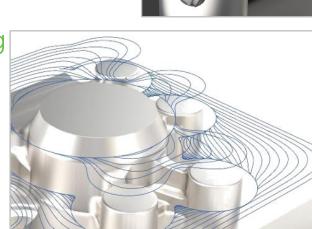
- HSM 5-Axis Rough
- HSM 5-Axis Rest Rough
- Auto Deburring



- HSM Rough
- HSM Rest Rough

#### 3 to 5-Axis Conversion

- HSM Finish
- HSM Rest Finish



## 7 New 5-axis Toolpaths



## Creo Mold Machining Extension was introduced with Creo 5.0

## ModuleWorks

Specialized 3-axis HSM machining powered by ModuleWorks (4 HSM Toolpaths)



#### New name: Creo High Speed Machining Plus



## **Module Works**

Expanded partnership with ModuleWorks Includes existing 3-axis HSM capabilities Adds 7 new 5-axis HSM toolpaths



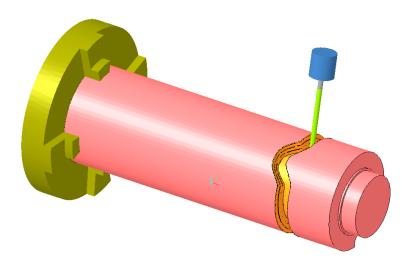
## Creo 8.0 Enhancement Summary

## HSM 5-axis Rough

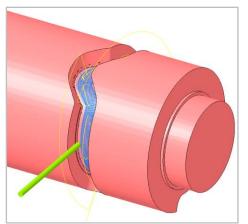
- Support for HSM 5 Axis Roughing toolpaths
  - Rapid manufacturing complex shapes with fewer setups and better surface finish
- Multiple Slice Generation Strategies
  - Offset from Floor
  - Offset from Ceiling
  - Morph
- Ability to select individual containment loops

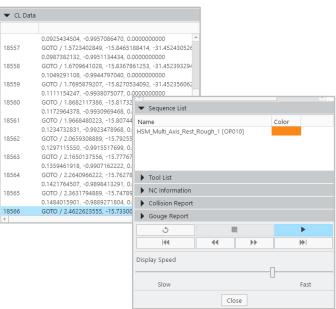
## HSM 5-axis Rest Rough

- Support for HSM 5 Axis Rest Roughing toolpaths
- Multiple Slice Generation Strategies
  - Offset from Floor
  - Offset from Ceiling
  - Morph
- Slice generation strategies use the rough stock file which is used to calculate the rest material
- Ability to select individual containment loops







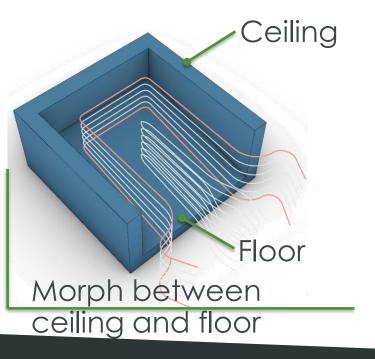


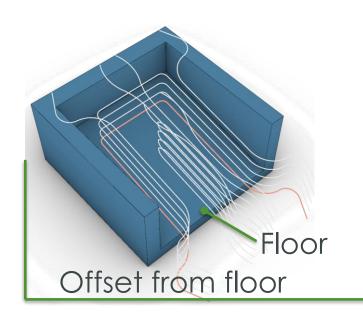


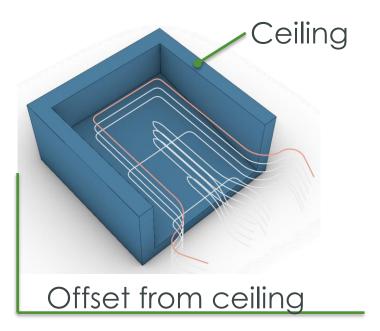
## Creo 8.0 Enhancement Summary

## 5-axis Rough and Rest Rough, high efficiency slice generation strategies

- Morph between ceiling and floor
  - Good for machining steep areas for mold machining
- Offset from floor
  - Defines pattern and creates offset cuts from floor surface
- Offset from ceiling
  - Defines pattern and creates offset cuts from floor ceiling







Creo 8.0 Enhancement Summary

## HSM Rough Scan Types: Constant Load (Default)

- Default setting computes HSM Roughing using Constant Load
- Constant load ensures constant cutting conditions
  - Stable tool load
  - Higher feedrate
  - Reduced tool-wear
- Constant Load reduce milling times by 40%-60% compared to conventional roughing
- Functionality is comparable to that offered in high priced competitive offerings
   Important note: Constant Load scan type should be used with solid carbide tools where full flute length is engaged for machining

## HSM Rough Scan Types: Spiral

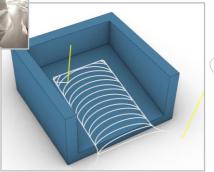
- Efficient, collision free and reliable roughing toolpaths using advanced smoothing
  - Reduced machining time for large volumes of stock
  - Stable machine kinematics
  - Enhanced tool-life

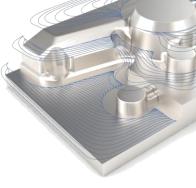
**Important note**: Spiral scan type should be used with ndexable insert tools on larger die and mold parts with less dynamic machines

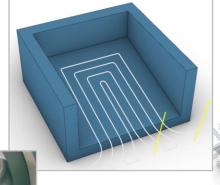




Constant Load









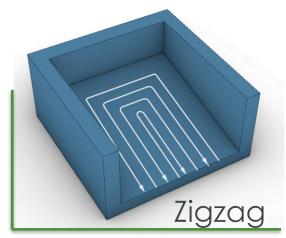


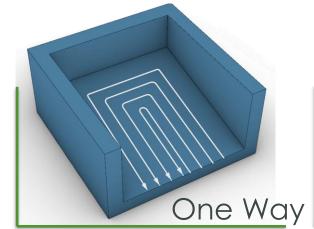


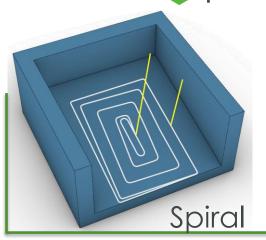
Creo 8.0 Enhancement Summary

## 5-axis, cutting motions

- Zigzag
- One way
  - Upcut, climb
- Spiral







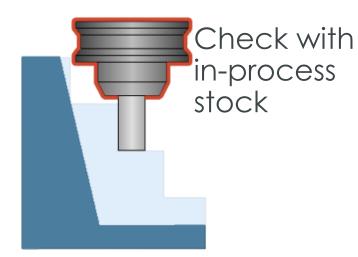
## Highly Automated tool axis control

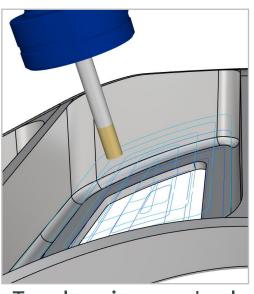
Tool is automatically oriented normal to floor surface

## Automatic collision checking with machining surfaces

Check with Machining surfaces







Tool axis control

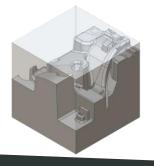


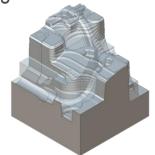
## Creo 8.0 Enhancement Summary

## Automatic 3+2 Machining

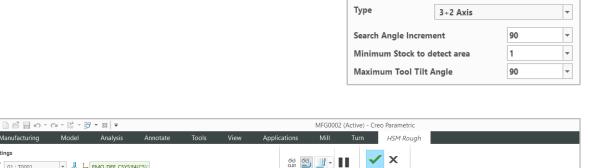
- Create 3+2 HSM Rough and HSM Rest Rough toolpaths
  - Automatic calculation if tool rotations and machining directions to minimize tool orientations and reach all material
  - Automatic collision detection with in-process stock and part
  - Works with complex parts including those with undercut areas
- Simplified creation
  - Tool definition, stock and part as input surfaces
  - Axis Control
    - Search Angle Increment
    - Minimum Stock to detect area
    - Maximum Tool Tilt Angle
- Maximum tool rotation of 90 degrees

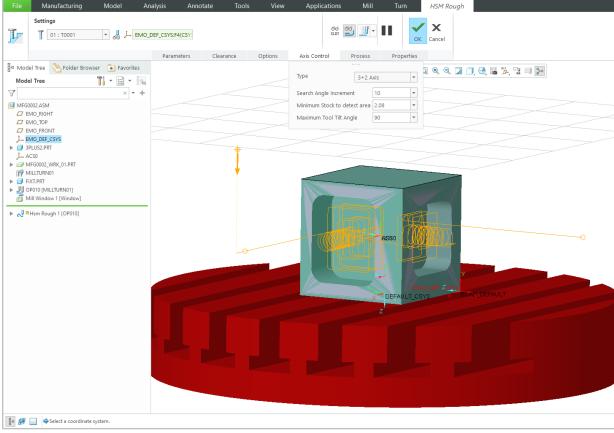
**Note**: 3+2 Machining computes toolpaths that are optimized to remove material using shorter tools to improve machining quality









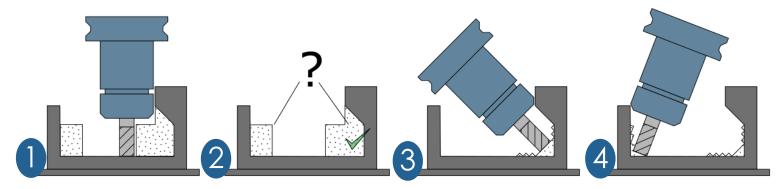




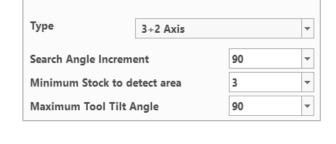
## Creo 8.0 Enhancement Summary

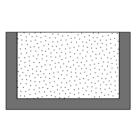
## Automatic 3+2 Axis Rough: How it works

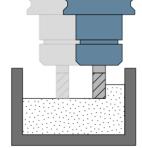
- 1. Automatically checks for the max rest machining area and selects the biggest to be machined along Z axis
- 2. Automatically selects the tool direction with best access
- 3. Machines the area (automatic boundary is being created)
- 4. This repeats until the stop condition set by the user is met

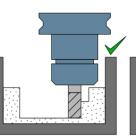


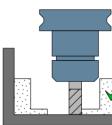
- Dynamic holder collision checking is a key element to this toolpath
- This protects the holder from collisions against:
  - the in-process stock model (that changes continuously during machining)
  - the actual workpiece











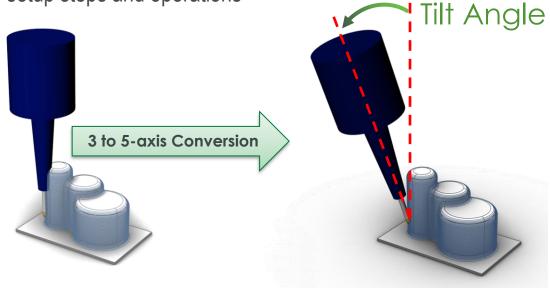


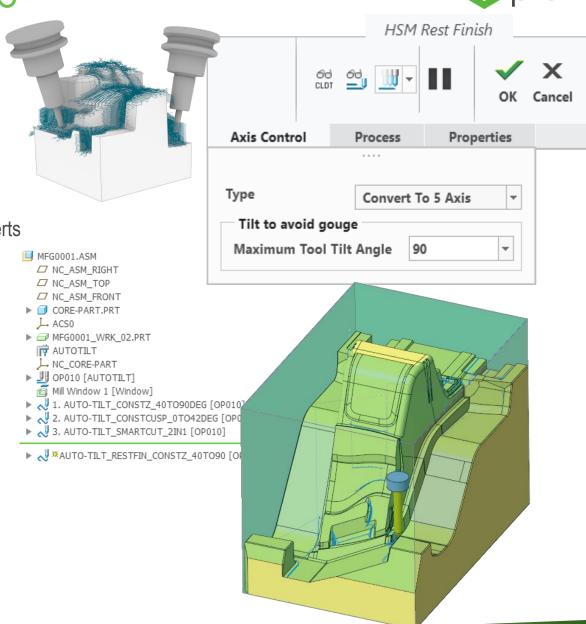
## Creo 8.0 Enhancement Summary

#### HSM 3 to 5-axis Conversion

- Automatically convert 3-axis HSM Finish and Rest Finish toolpath to 5-axis Finish and Rest Finish toolpath
  - Goal is to use a shorter tool to improve machining quality of the finishing toolpaths (less tool chatter and deflection)
  - 3 to 5-axis conversion auto detect collision with tool holder and converts to 5-axis to enable tool tilt and avoid gouging
    - Maximum Tool Tilt Angle (control)

**Note**: Conversion to 5-axis can produce more complex parts with less setup steps and operations



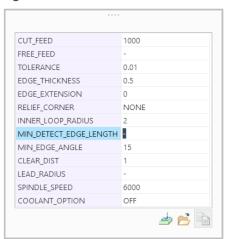


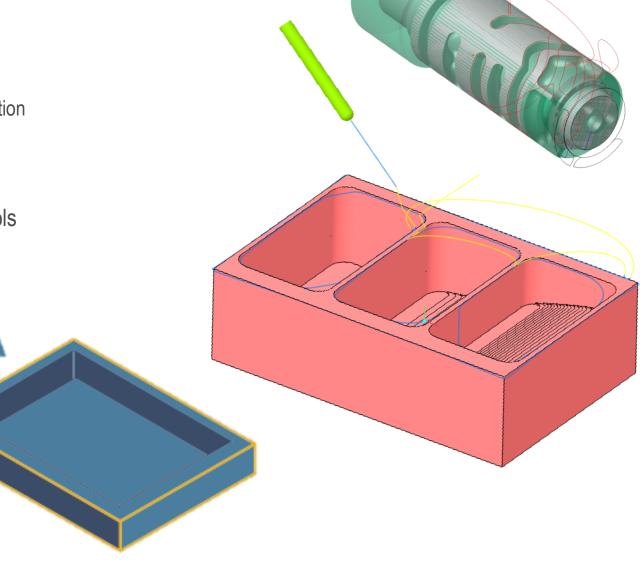
## Creo 8.0 Enhancement Summary

### **Auto Deburring Toolpath**

- Automatic creation of deburring toolpaths
  - Simplified setup using part selection and automatic edge detection
  - Option to exclude edges
- Supports 3 and 5-axis machining strategies
- Automatic tilting, linking and collision checking
- Option to support manual selection and provide explicit controls
  - manual edge selection for inclusion or exclusion
  - minimal length edge detection
  - minimal edge angle edge detection









## Creo 8.0 Enhancement Summary

## **Auto Deburring Chamfer Geometry**

Geometry of the chamfer can be approximated by Edge\_Thickness parameter

## Approximate chamfers

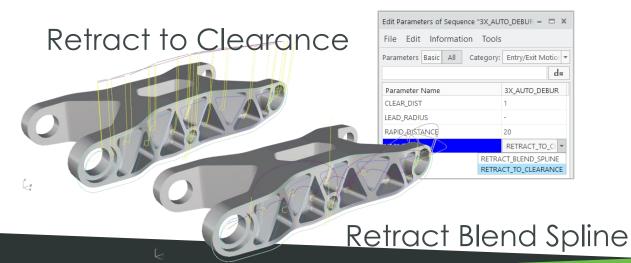
Number\_Cuts parameter can approximate edge chamfers

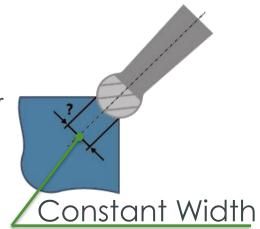
## **Auto Deburring Link Optimization**

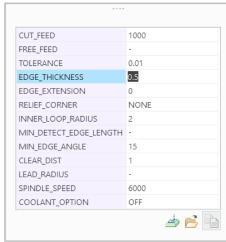
Retract\_Blend\_Spline parameter creates collision free blend spline

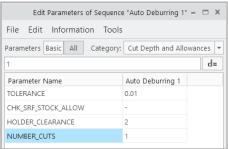
#### **Auto Collision Avoidance**

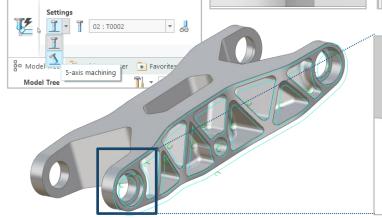
Automatic tilting to avoid collision with part, fixture surfaces and tool

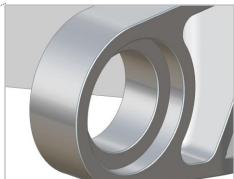














## ADDITIVE MANUFACTURING

## ADDITIVE MANUFACTURING SOLUTION SUITE



## Creo 8.0 Additive Manufacturing Solution suite



Connectivity with polymer printers and service bureaus



Parametrically controlled lattice structures and data-managed tray assemblies



Creo AM functionality plus Support structures for metal and build processors

## Creo Users have expressed their needs of Additive Manufacturing

Lightweight design – Different methodologies with an integrated design-analysis process backbone

- Lattice modeling and generative design
- Simulation results update dynamically as user makes geometry or setup changes

**Self-supporting geometries** – Minimize or eliminate the need of support structures for printing **Enable bio-mimicry** – Both lattice and generative design enable modeling the best designs found in nature

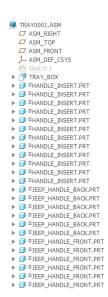
To solve structural problems, sound or impact absorption, thermal transfer, vibration control...
 Enable interoperability – Cover a wide array of ways to send the 3D Models to 3D Printers



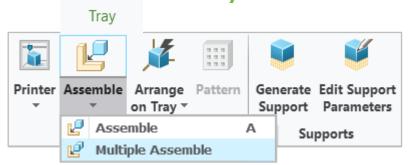
## Creo 8.0 Enhancement Summary

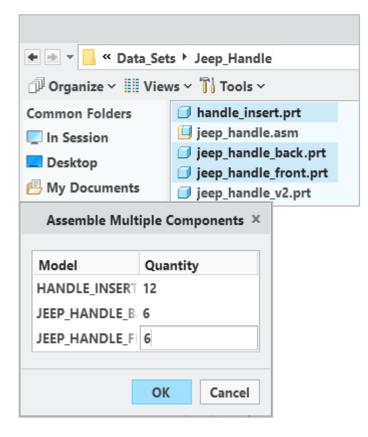
#### Multiple Component Print Tray Assembly

- New Assembly Dialog option to insert multiple Quantities of components
  - Choose model and specify "Quantity" to assemble
  - Eliminate the need for workaround using pattern functionality
- Ability to select multiple occurrences of the same model or of different models
  - Automatic arrangement on tray
  - Arrangement based on build direction if available
- Support for parts and assemblies











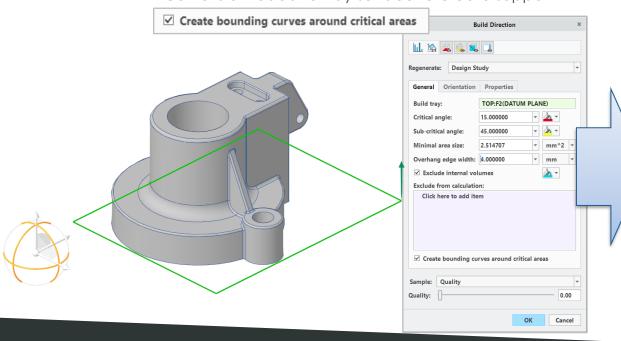
64.00 29.22%

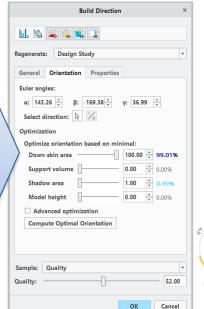
22.00 10.05%

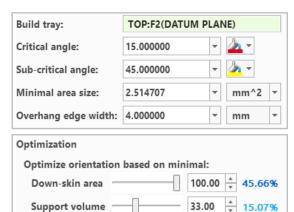
## Creo 8.0 Enhancement Summary

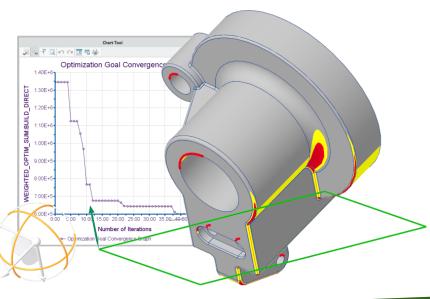
#### Improvements to Build Direction Optimization

- Simplified workflow
- Recognition of overhang edges and corners
- Optimize Build orientation based on weighted optimization criteria
  - i.e., Down skin area, Support volume, Shadow area, Model height
- Minimal Edge Width option
  - Define minimal width for support structures
  - Option to create bounding curves around critical areas
    - Curve is extruded to tray surface to create support











## Creo 8.0 Enhancement Summary

## Solid Representation support for Stochastic Lattices

- Full geometry representation for stochastic lattice
  - Accurate representation and calculation of mass properties
  - Previous versions displayed simplified representations and needed to be printed using 3MF file format
- Option to toggle "full geometry" representation
  - Solid geometry display increases computation time

Note: The recommendation is to continue using simplified representation

- CSL does not require Full Geometry for simulation
- At the end of process, convert to Full Representation

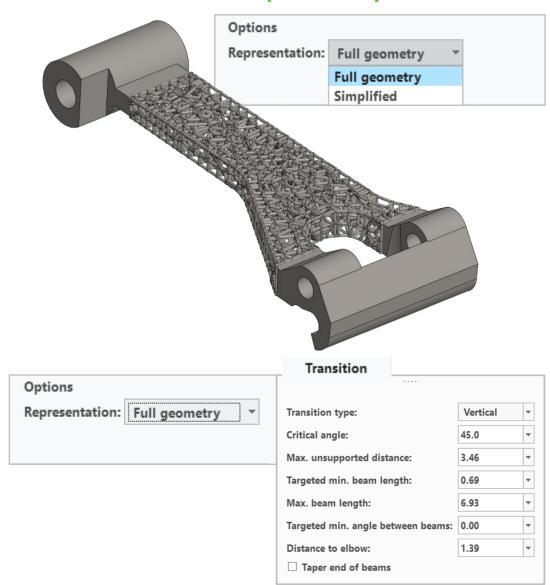
#### Solid Representation for Transitions

Full Representation support for all transition types and options

#### Delete individual beams in lattice

 Remove command supports selective removal of unwanted lattice geometry i.e., beams, balls, dangling beams







## Creo 8.0 Enhancement Summary

Support for Trabecular shape Stochastic Lattices

Trabecular Shape option reduces shape deviation of cells providing shape consistency

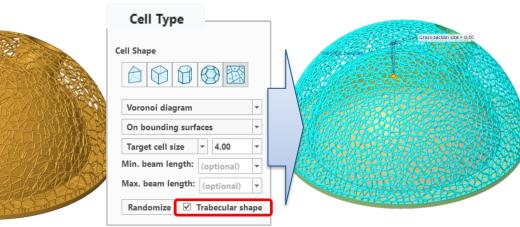
- Trabecular Shape control mimics that of human bone cells
  - Broad application within the medical industry

#### Replacement of Formula-Based algorithm

- Formula-driven lattice creation uses a newer, more accurate technology
  - Option to "convert to new technology" when opening existing models
  - Higher quality lattice improves the intersection (interface) between lattice and shell geometry
- Support for gyroid, primitive and diamond Cell Types
- Option to create complementary bodies
  - Complementary bodies support the use case in which bodies can represent fluid or material assignment (3D print)





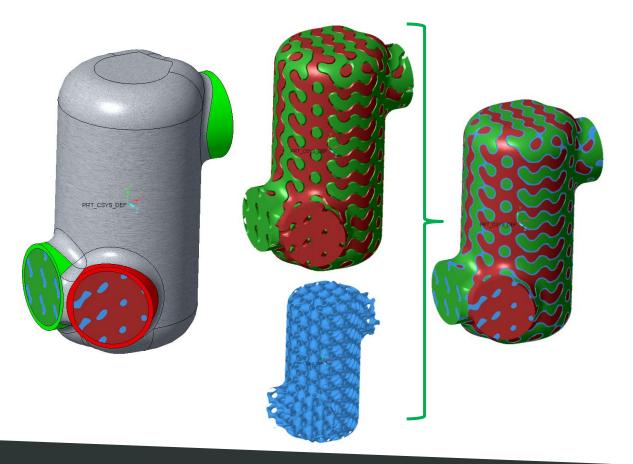




## Creo 8.0 Enhancement Summary

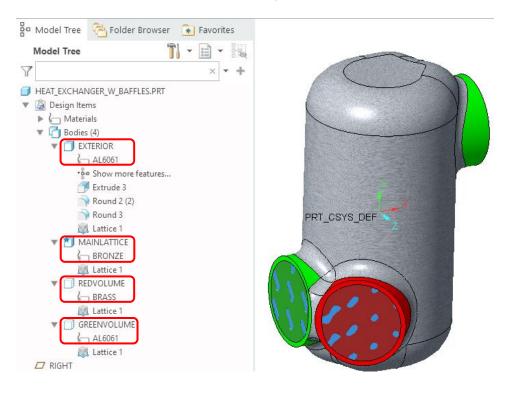
#### Replacement of Formula-Based algorithm (Use case positioning)

- Heat Exchanger application
  - Automatic creation of baffles to create a barrier between the two fluid domains on the TPMS lattice



#### Structural application

- Possibility to assign different materials to each complementary volume and to the lattice itself.
- Complete control on material assignment



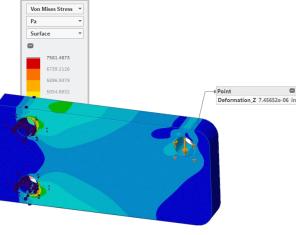


Creo 8.0 Enhancement Summary

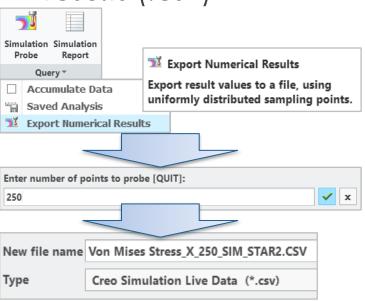
#### Lattice Variability based on simulation results

- 3<sup>rd</sup> phase to provide simulation-driven lattice adjustment and refinement
  - Simulation results are imported and used as an overlay to map areas of stress and adjust lattice density
- New workflow adds variability to lattices using simulation results data
  - Create variable density stochastic lattices
  - Create classic rectangular lattices with variable beam size

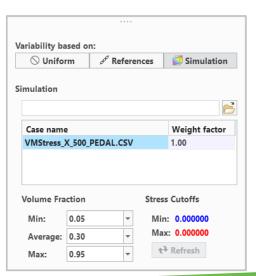




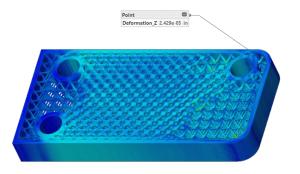
2 Export Simulation results (.csv)



Define lattice density from Simulation results



4 Create simulationdriven lattice and validate results

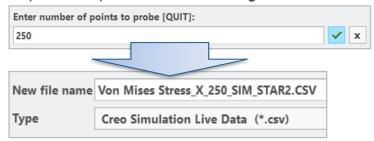




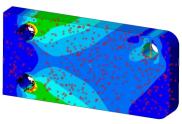
## Creo 8.0 Enhancement Summary

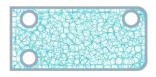
## Lattice Variability based on simulation results

- Works with simulation data from Creo Simulation Live and Creo Simulation
  - Creo Simulate Study (folder)
  - Creo Simulation Live Data (\*.csv)
  - Density Map Tab Delimited (\*.txt)
- When using results from Creo Simulation Live, the user specifies the number of points to probe when creating the Creo Simulation Live \*.csv file

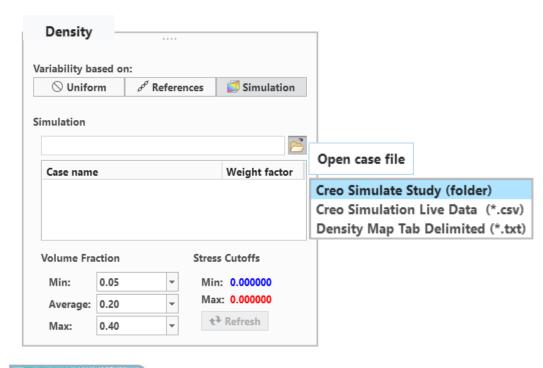


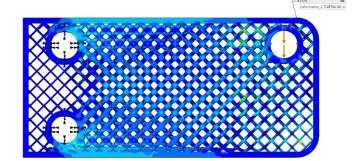
- For Stochastic Lattices
  - · Cell density increases in areas of high stress
  - · Diameter of cell beams and balls does not vary
- For Uniformly Distributed Lattices
  - Diameter and beams and balls increases in areas of high stress
  - The aspect ratio between beam and ball is maintained













## Creo 8.0 Enhancement Summary

#### Lattice Variability based on simulation results

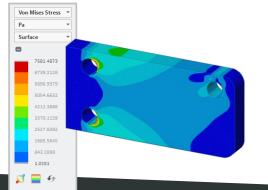
- Volume Fraction controls lattice beam and ball size
  - Volume Fraction is the % of mass of the original body
- Stress Cutoffs
  - Absolute stress value
  - In regions where stress is below the stress cutoff is value, the lattice density will correspond to the minimum volume fraction

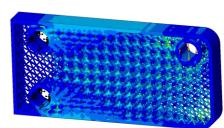
## Seamless integration with Behavioral Modeling

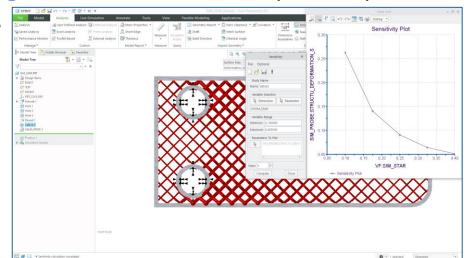
Sensitivity Analysis can be used to identify the Volume Fraction to achieve design goals (i.e., deflection)



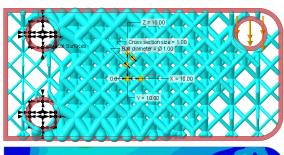






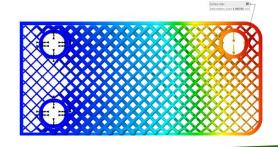








4 Use Sensitivity results to obtain volume fraction





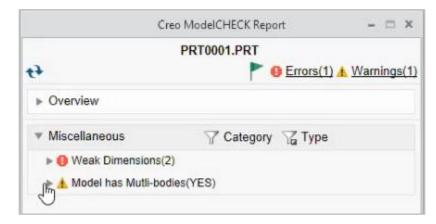
# MODELCHECK

## MODELCHECK

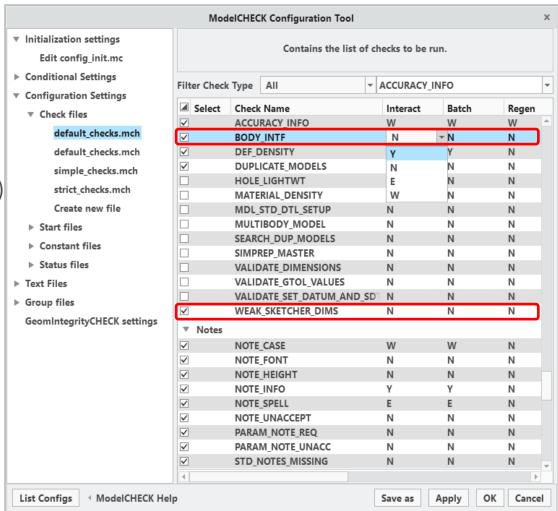
## Creo 8.0 Enhancement Summary

#### **New Checks**

- Body Interference: BODY\_INTF (Yes/No/Error/Warning)
  - Reports any bodies that have intersecting geometry, including information on which bodies have interference
- Weak Dimensions: WEAK\_SKETCHER\_DIMS (Yes/No/Error/Warning)
  - Reports any weak dimensions in sketches









# DATA EXCHANGE UNITE TECHNOLOGY

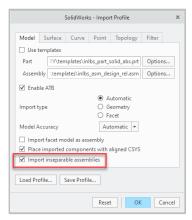
## UNITE TECHNOLOGY – DATA EXCHANGE

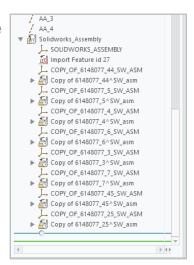


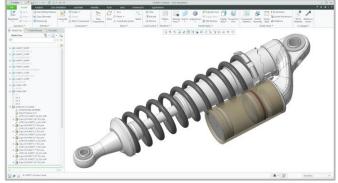
## Creo 8.0 Enhancement Summary

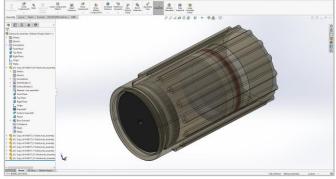
## Standard Import Support for Inseparable Assemblies

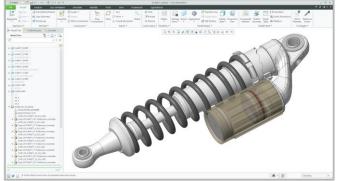
- New option for importing inseparable assemblies is now available in the standard import profile dialog
  - Inseparable structure will be preserved when importing assemblies into Creo
  - Component glyph will reflect if it is an inseparable component of the assembly
- Users cannot change the inseparable status manually
  - Ownership of the inseparable status belongs to ATB update only
  - Breaking the ATB link (Make Independent) will make it possible to extract the components to their own file
- Users can update their imported assembly through ATB if the original source has been modified
- Creo 7.0 provided the support for Creo Unite to open assemblies defined as inseparable assemblies
  - Note: Converting the Creo Wrapper to Creo models will extract the inseparable assembly to individual Creo files
- Creo does not currently support exporting inseparable assemblies to other formats











## UNITE TECHNOLOGY - DATA EXCHANGE



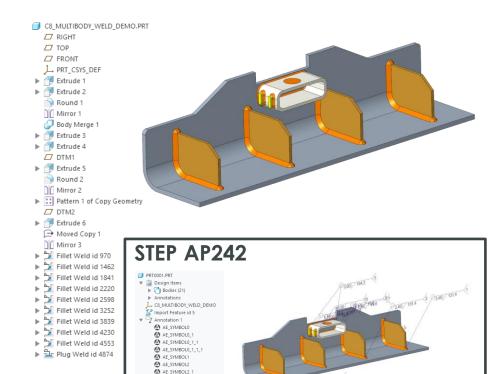
## Creo 8.0 Enhancement Summary

## **Export Solid Weld Geometry**

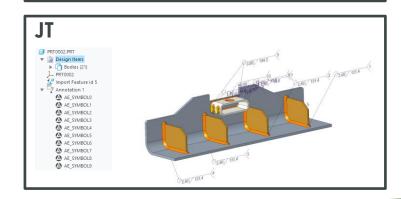
- Exported parts containing solid welds as 3D Precise Geometry to STEP, JT or Neutral formats will be exported as multibody parts
  - Each weld feature will be defined as an additional solid body of the part
- Exporting parts containing solid welds as Tessellated Geometry to STEP or JT will represent the solid weld as additional facet geometry
- Including Annotations and Rich Content when exporting to STEP AP242 and JT will include the solid weld PMI information
  - Geometric references will be export as semantic PMI of the weld symbol
  - Selecting the weld symbol of the imported file will highlight the Geometric references (surface and datum) for the given weld

#### Note:

- Part-level Surface or Light-weight welds there will not be impacted by this enhancement and follow current behavior
- Assembly-level solid welds will continue to be exported as separate part components of the exported assembly



♠ AE\_SYMBOL3\_1
 ♠ AE\_SYMBOL4\_1





# WINDCHILL INTERACTION

## CREO WINDCHILL INTERACTION



## Creo 8.0 Enhancement Summary

## Content pop-up during custom check in

- Custom Check-in made easier
- When "Custom Check In" is used, the user is no longer asked what type of content to be checked in
- The default dialog setting is controlled by Windchill Preferences
- If needed the pop-up dialog (Custom Check In) can be shown
  - dm\_custom\_checkin\_prompt no\*/yes

