

TOP 10

John SWEENEY

Senior Director, SOLIDWORKS Product Development

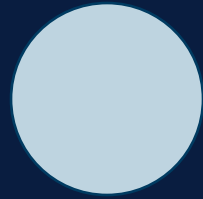
Bruce HOLWAY

Senior Director, SOLIDWORKS Product Definition

The logo for 3DEXPERIENCE WORLD is displayed on a dark blue background. The text "3DEXPERIENCE" is in a smaller, white, sans-serif font, with a registered trademark symbol. Below it, the word "WORLD" is written in a much larger, bold, white, sans-serif font. The background features a network of glowing blue nodes connected by thin lines, and a circular graphic in the top right corner with concentric circles and a red and blue color scheme.

3DEXPERIENCE[®]
WORLD

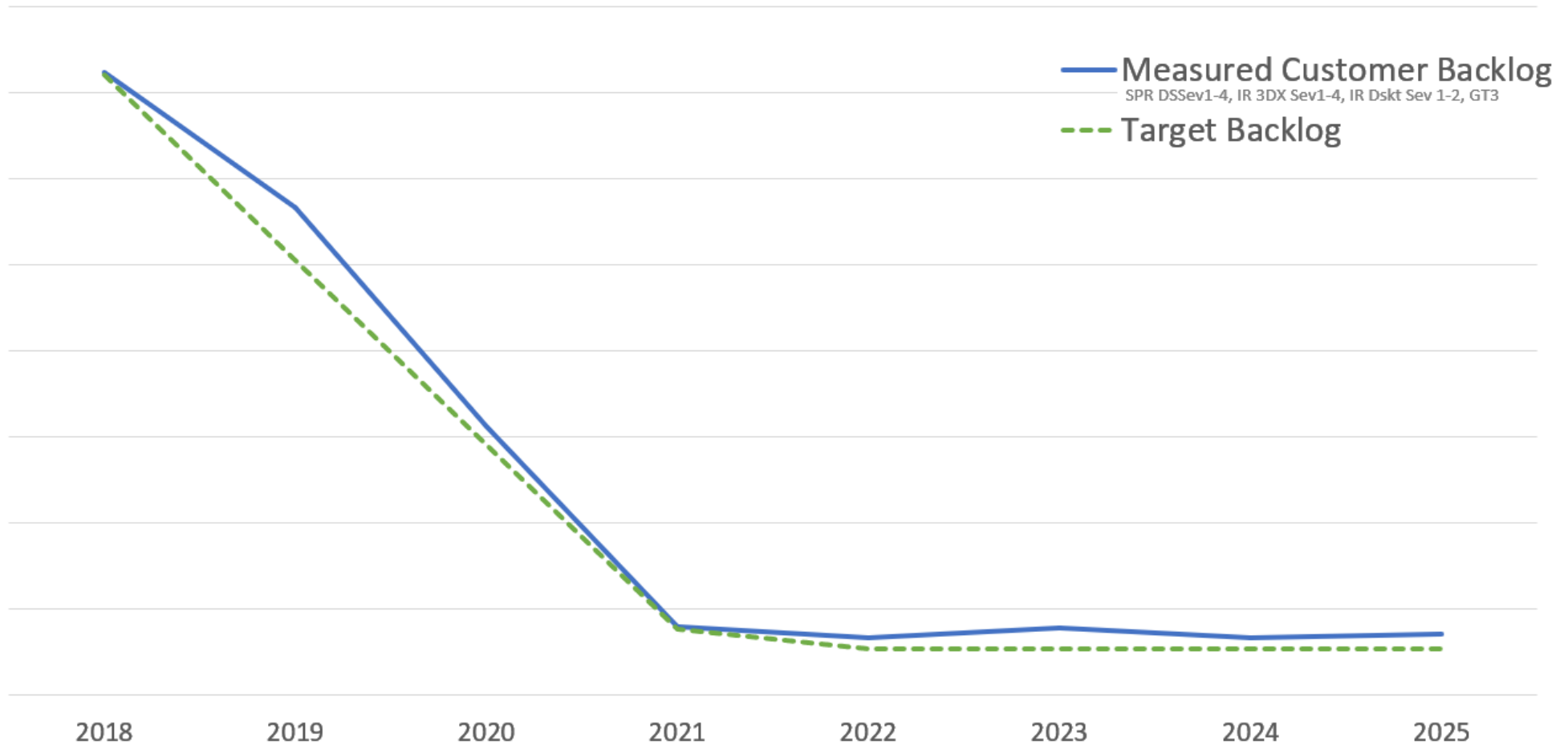
R&D
Focus



Quality & Performance Efforts

Quality

- *Bug Backlog*
- *Bug Inflow = Lowest in 10 years!*



Performance Initiative

Real-world customer models



Tire manufacturing



Construction equipment



Automotive manufacturing



Automation



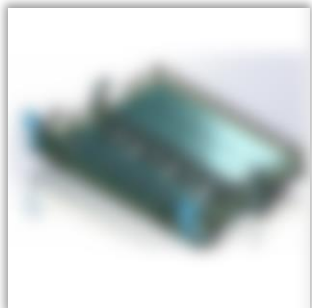
Medical device



Research laboratory



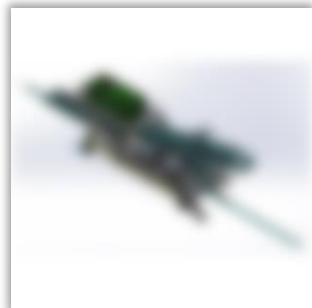
Packaging equipment



Manufacturing equipment



Packaging equipment



Cement plant



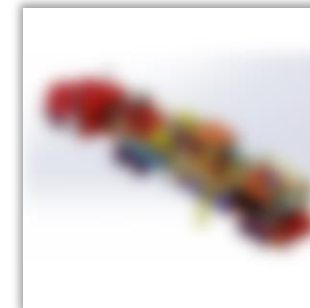
Industrial laundry



Semi-conductor manufacturing



Medical device



Oil and gas equipment

Performance Initiative – Workflow Focus

Broad range of real-world workflows...

Specific operations common to each workflow...

Customer Workflow

- | |
|--|
| Workflow #1 - Assembly Structure |
| Workflow #2 - Mates |
| Workflow #3 - Configurations |
| Workflow #4 - Modeling & Sketching |
| Workflow #5 - Multi Window |
| Workflow #6 - Selections, UI Interactions & Tools |
| Workflow #7 - Drawing Creation |
| Workflow #8 - Editing Large Multi-Body Parts |
| Workflow #9 - Platform Open, Insert, Save |
| Workflow #10 - Platform Modify Structure Open and Save |
| Workflow #11 - Platform Open & Save of Many Configurations |
| Workflow #12 - Identify files that get marked dirty on open |
| Workflow #13 - Rebuild Drawing Views |
| Workflow #14 - Build an Assembly with LDR Edit |
| Workflow #15 - Large Multi-Body Parts in Assembly & Drawings |



Workflow 1 - Structure

1. Open assembly (a) in lightweight mode
2. Insert New Part to top level
3. Insert Existing Part (c) to top level
4. Undo
5. Redo Insert Existing Part (c) to top level
6. Suppress part inserted at step 3 5
7. Unsuppress part inserted at step 3 5
8. Delete Part inserted at step 3 5
9. Edit subassembly (b) (depth TBD)
10. Insert New Part in context
11. Insert Existing Part (c) in context
12. Undo
13. Suppress part inserted at step 10
14. Unsuppress part inserted at step 10
15. Redo?
16. Delete part (step 10)
17. Undo
18. Edit Top assembly
19. Delete part inserted at step 3 5 ?
20. Undo
21. Save
22. Close

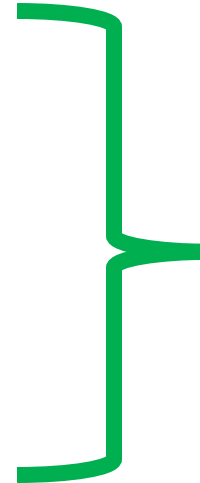
Work In Progress

Performance Initiative – Workflow Focus

Broad range of real-world workflows...

Customer Workflow

- | |
|--|
| Workflow #1 - Assembly Structure |
| Workflow #2 - Mates |
| Workflow #3 - Configurations |
| Workflow #4 - Modeling & Sketching |
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46.3% Faster

4.1 hours → 2.2 hours

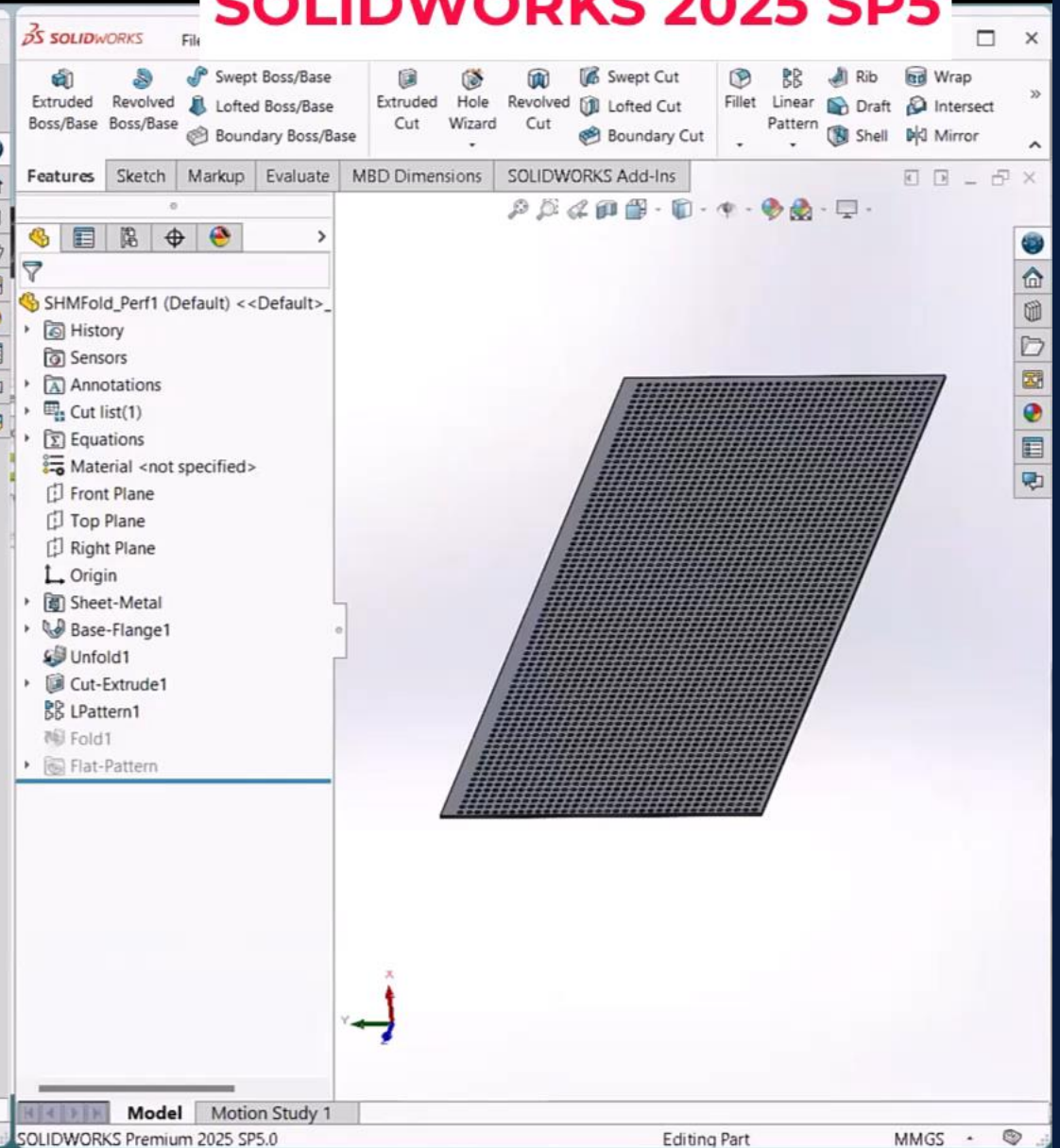
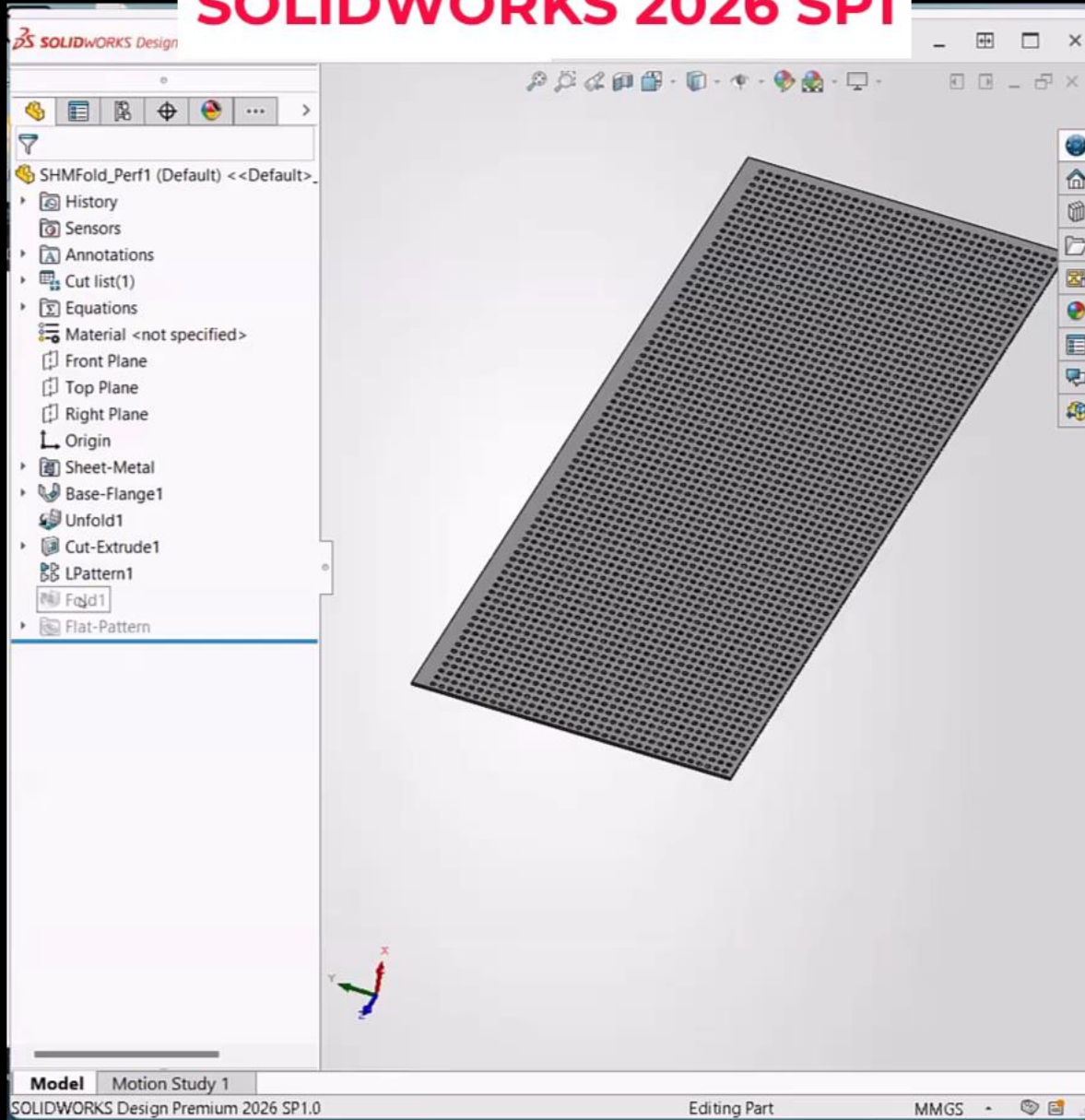
SW2023 sp3 to SW2026 sp0

Performance Efforts - Beyond the “Workflow Focus”

Area	Function Id	Func Title	Program Title
Electrical Schematic	FUN116162	Performance and accuracy: Contact penalty stiffness improvement for shells	SOLIDWORKS_Desktop_2025_SP01-FD01
Drawings, MBD, Inpsection	FUN139915	Improve performance of Export of Draft Quality views to DXF/DWG	SOLIDWORKS_Desktop_2025_SP01-FD01
Parts	FUN140914	IFC Import Filtering (workflow performance)	SOLIDWORKS_Desktop_2025_SP01-FD01
Parts	FUN141409	Performance: Ability to import models as a background process	SOLIDWORKS_Desktop_2026_Beta2
Visualize	FUN144889	Visualize Import Performance Improvement	SOLIDWORKS_Desktop_2025_SP01-FD01
Draftsight	FUN149802	xDraftSight - Performance improvements by Graebert	3DEXPERIENCExDraftSight-1.4.0
Assemblies	FUN151184	Seamless transition between LDR and editing parts in context (performance)	SOLIDWORKS_Desktop_2025_SP04-FD04
UX	FUN151680	[HMA][Desktop]Migrate Assembly Performance Evaluation Dialog to WPF	SOLIDWORKS_Desktop_2025_SP01-FD01
Routing	FUN152715	Edit Flattened Route - Performance	SOLIDWORKS_Desktop_2025_SP01-FD01
Parts	FUN152983	Performance: Selection filter for large number of feature bodies.	SOLIDWORKS_Desktop_2026_Beta2
Sheet Metal/Weldments	FUN153179	[QUALITY]Sheet Metal - Quality & Performance	SOLIDWORKS_Desktop_2026_Beta2
Systems	FUN153379	[SW2025 Desktop SP01] Improve Incremental/Full build performance	SOLIDWORKS_Desktop_2025_SP01-FD01
Parts	FUN153487	Workflow 14 Part Performance	SOLIDWORKS_Desktop_2025_SP01-FD01
SW Plastics	FUN153579	Mesh performance improvements	SOLIDWORKS_Desktop_2025_SP01-FD01
Power'By	FUN154517	Performance improvement for connectors	SOLIDWORKS_Desktop_2026_Beta2
API	FUN156194	Performance Improvement in retrieving Annotations	SOLIDWORKS_Desktop_2025_SP02-FD02
Power'By	FUN156750	SW Power'By - Performance log utility - [UES][CONNECTED]	3DEXPERIENCER2025x_FD03
Drawings, MBD, Inpsection	FUN157375	Improve ballooning performance for compound holes in add-in	SOLIDWORKS_Desktop_2025_SP03-FD03
PDM	FUN160348	Improve performance, stability and quality	3DEXPERIENCER2025x_FD03
Parts	FUN160564	Performance: Address performance degradation when loading a large assembly	SOLIDWORKS_Desktop_2026_Beta2
Parts	FUN160580	Performance: Apply Parasolid Guards to improve productivity	SOLIDWORKS_Desktop_2026_Beta2
API	FUN161635	Research Performance improvements for document manager using Rapid XML	SOLIDWORKS_Desktop_2026_Beta2
PDM	FUN162961	Jobs reliability and performance	3DEXPERIENCER2025x_FD04
Electrical	FUN117125	[TEW-52390] Improve application startup performance	3DEXPERIENCER2026x_FD03
Drawings, MBD, Inpsection	FUN139917	Improve performance of Detail/Crop/Break view Trimming	SOLIDWORKS_Desktop_2026_SP03-FD03
Draftsight	FUN149802	xDraftSight - Performance improvements	3DEXPERIENCExDraftSight-1.4.0
Assemblies	FUN151184	Seamless transition between LDR and editing parts in context (performance)	SOLIDWORKS_Desktop_2026_SP02-FD02
Power'By	FUN156750	SW Power'By - Performance log utility	3DEXPERIENCER2026x_FD01
Routing	FUN162541	Build model performance analysis	SOLIDWORKS_Desktop_2026_SP01-FD01
Parts	FUN164011	Performance: Ability to import models as a background process	SOLIDWORKS_Desktop_2026_SP01-FD01
Parts	FUN164237	Performance: Selection filter for large number of feature bodies.	SOLIDWORKS_Desktop_2026_SP01-FD01
Sheet Metal/Weldments	FUN166396	[Performance]Pattern performance improvements with large number of pattern cuts	SOLIDWORKS_Desktop_2026_SP01-FD01
Sheet Metal/Weldments	FUN166398	[Performance]Improve Sheet Metal Fold performance with larger number of pattern instance cuts	SOLIDWORKS_Desktop_2026_SP01-FD01
Sheet Metal/Weldments	FUN166400	[Performance]Improve Cut-list update performance for Sheet Metal parts with large number of pattern cuts	SOLIDWORKS_Desktop_2026_SP01-FD01
Graphics	FUN166802	Large multibody part highlighting performance	SOLIDWORKS_Desktop_2026_SP01-FD01
Parts	FUN169162	[Performance] Point Cloud import performance	SOLIDWORKS_Desktop_2027_Beta2
Parts	FUN169459	[Performance] Do not update doc modify on cancel feature edit	SOLIDWORKS_Desktop_2026_SP02-FD02
Sheet Metal/Weldments	FUN169871	[Performance]Performance improvements for Area fill and other pattern types	SOLIDWORKS_Desktop_2026_SP02-FD02
Assemblies	FUN170315	Always-on performance evaluation	SOLIDWORKS_Desktop_2026_SP02-FD02
Power'By	FUN170425	[HMA] CircuitWorks performance — Phase 2	SOLIDWORKS_Desktop_2026_SP02-FD02

SOLIDWORKS 2026 SP1

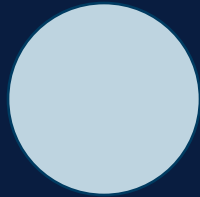
SOLIDWORKS 2025 SP5



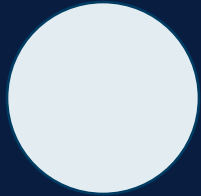
00:00

00:00

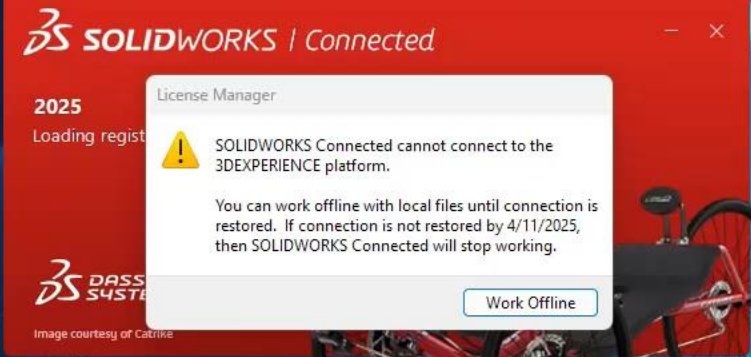
R&D Focus



Quality & Performance Efforts




Powerful & Flexible Platform Additions



SOLIDWORKS | Connected

2025
Loading regist...

License Manager

 SOLIDWORKS Connected cannot connect to the 3DEXPERIENCE platform.

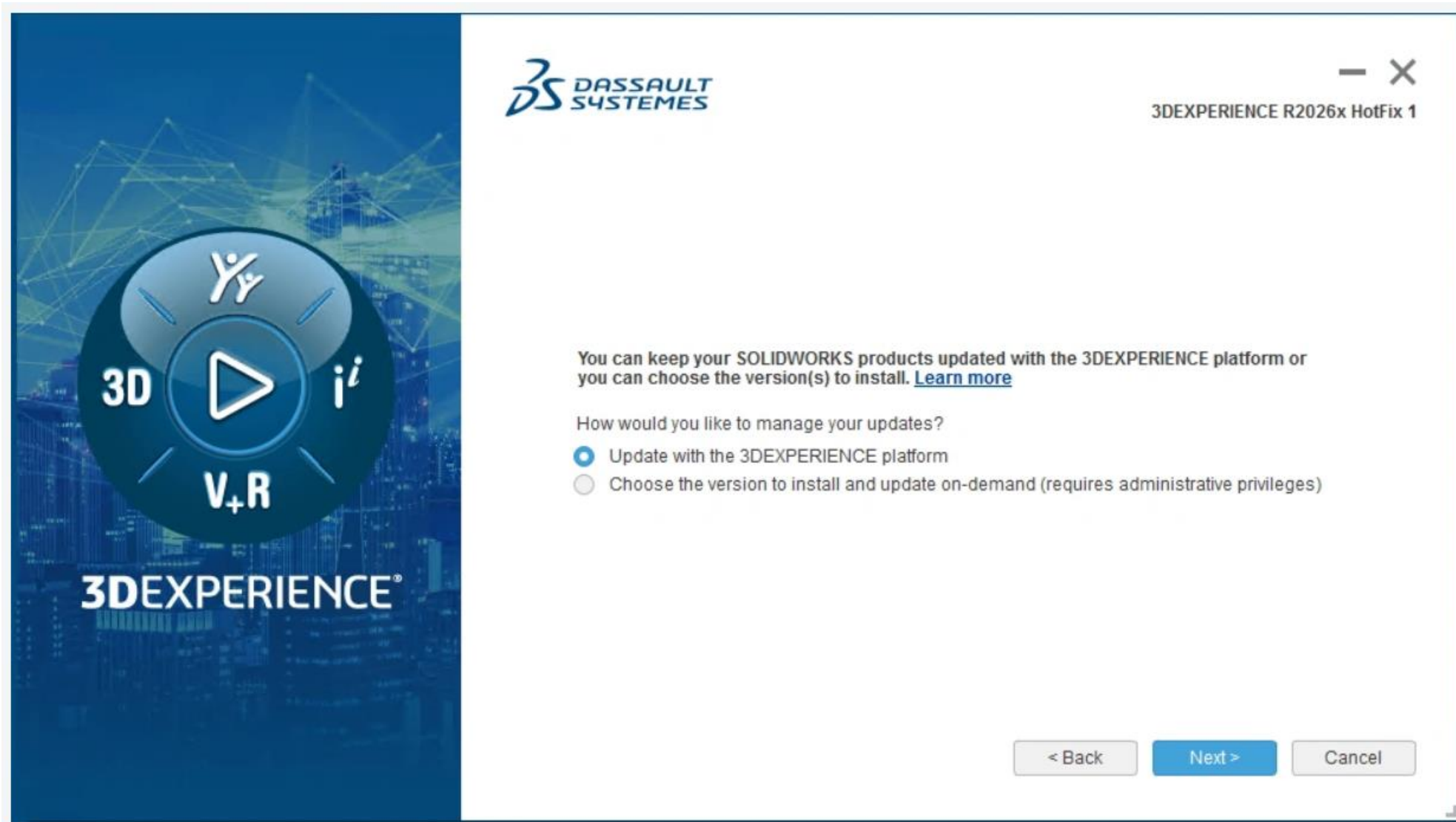
You can work offline with local files until connection is restored. If connection is not restored by 4/11/2025, then SOLIDWORKS Connected will stop working.

[Work Offline](#)

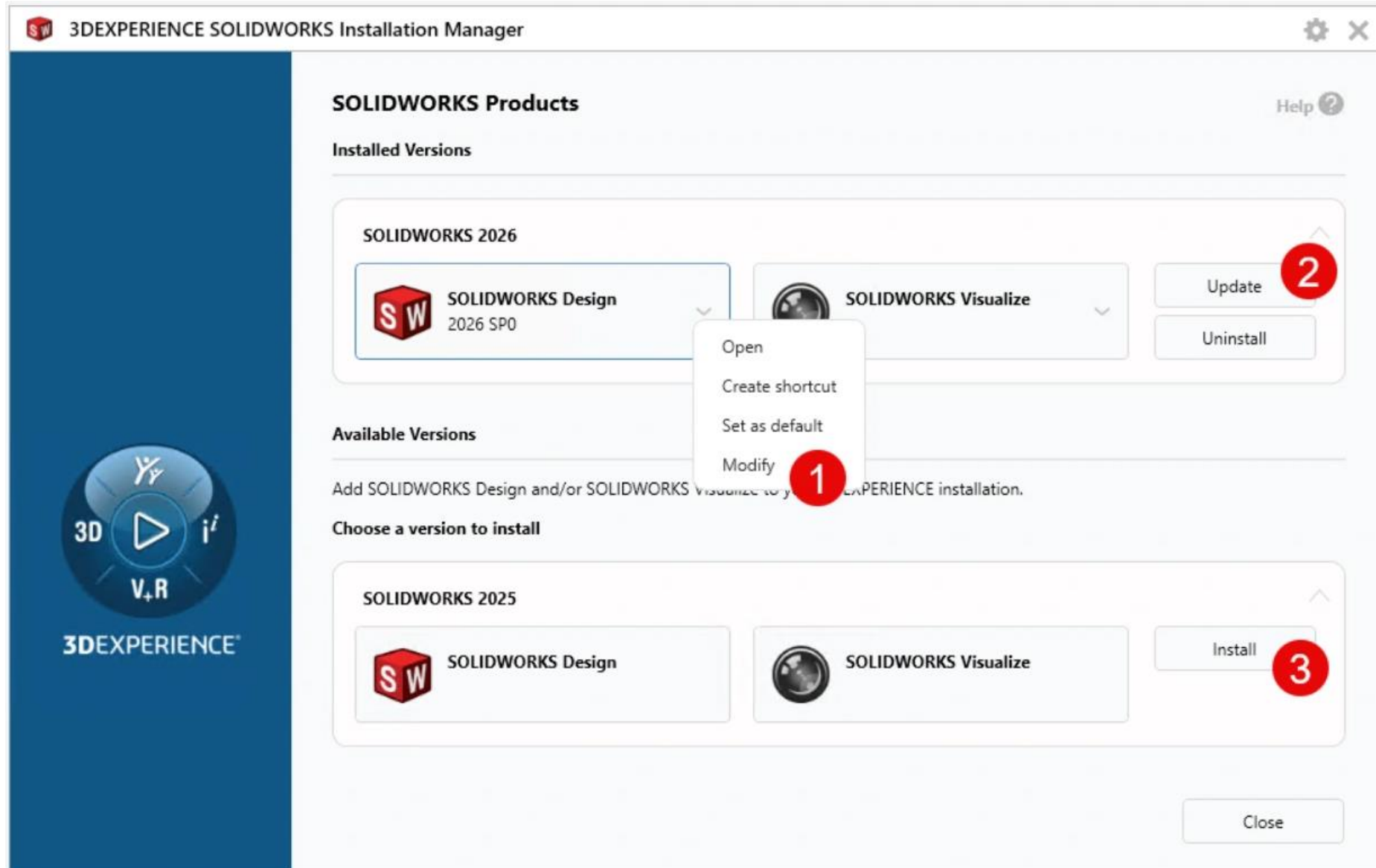
DASSAULT SYSTEMS

Image courtesy of Catrine

Installation Flexibility



Installation Flexibility



SELECTIVE LOADING - HANDLING LARGE ASSEMBLY WITH PRECISION



Product Explorer
(Browser)



Define Filter

Selects part/assembly via various methods to define open scope



Open Filter in
SOLIDWORKS

Open only what is needed not the entire assembly

The screenshot displays the 3DEXPERIENCE 3DDashboard interface for the 'Trakka Project'. The top navigation bar includes the 3D logo, '3DEXPERIENCE | 3DDashboard Trakka Project', a search bar, and user information 'Ed Engineer'. The main interface is divided into two panes:

- Product Structure Explorer (Left Pane):** A tree view showing the assembly structure. The root is 'ENOVIA - Product Structure Explorer - TRAKKAWAY 860 - ALKO KL1463994-2'. Below it is a table of parts:

Title	CA...	Enterprise Item Nu...	Title	Revi...	Matu...	Lock	P	Description	Menu
TRAKKAWAY 860 - ALKO KL...	SP			A	In Work				Eye
TRAKKAWAY 860 - TRAK...	SP		TRAKKAWAY 860 - ...	A	In Work				Eye
Floor Frame - Trakka...	SP		Floor Frame - Trakka...	A	In Work				Eye
Front Lounge Assem...	SP		Front Lounge Asse...	A	In Work				Eye
Cabinet Assembly - R...	SP		Cabinet Assembly - ...	A	In Work				Eye
Cabinet Assembly - L...	SP		Cabinet Assembly - ...	A	In Work		Default		Eye
False Floor - Rear Be...	SP		False Floor - Rear ...	A	In Work				Eye
Wall Assembly - Trakk...	SP		Wall Assembly - Tra...	A	In Work				Eye
Tank Water Grey - Tra...	SP		Tank Water Grey - T...	A	In Work				Eye
Tank Water Fresh - Tr...	SP		Tank Water Fresh - ...	A	In Work				Eye
RWR Aux Tank - 50L...	SP		RWR Aux Tank - 50L...	A	In Work		Trakkaway 860 - R...		Eye
Shower Assembly - Tr...	SP		Shower Assembly - ...	A	In Work				Eye
Front False Floor - Tr...	SP		Front False Floor - ...	A	In Work				Eye
Sink-To-Tank	SP		Sink-To-Tank<1>	01	In Work				Eye
ALKO TANDEM - KL1463...	SP		ALKO TANDEM - K...	A	In Work		FPC 3T NR/WR		Eye

The bottom pane shows 'ENOVIA - 3D Navigate - TRAKKAWAY 860 - ALKO KL1463994-2' with a 3D model of the motorhome. The interface includes a search bar, navigation icons, and a toolbar at the bottom with 'View' and 'Tools' tabs. A user profile picture is visible in the bottom left corner of the interface.

R&D Focus



Quality & Performance Efforts

Powerful & Flexible Platform Additions

AI

AI IN SOLIDWORKS

AUTOMATE

Smart automation
that help users
work faster

ASSIST

AI anticipating your
needs, suggesting
next steps

CO-CREATE

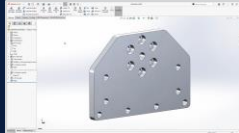
AI creating designs &
documentation with
you, not just for you

AI IN SOLIDWORKS

AUTOMATE



Repairing Missing Mate Reference (SW23)



Repair Sketch Relations (SW25xGA)



Pattern Driven Component (SW23)



Selection Accelerators (SW25xGA)

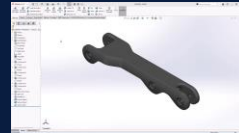


Driven Silhouette Part Defeature (SW25xGA)



Auto Generate Drawing (25xFD03)

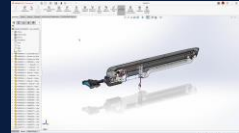
ASSIST



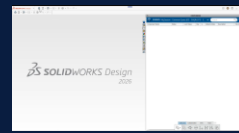
Command Predictor (SW25xGA)



Best Practices Companion (SW26xFD01)



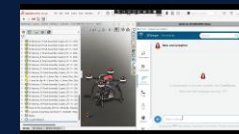
Auto Fastener Recognition (SW25xFD03)



Error Assistant (SW26xFD01)



Assembly Performance Doctor (26xFD01)



Model Insights (26xFD02)

CO-CREATE

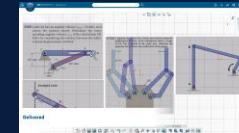


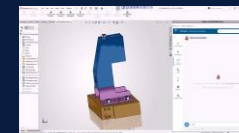
Image to Sketch to 2D Mechanism (X25 FD03)



Generative Rendering (Visualize25)



Assembly Structure Design (26xFD01)



Drawing Creation (26xFD01)



THE TOP TEN LIST

User Community Driven Enhancements

A large circular graphic on the right side of the page. It has a dark blue background with a glowing blue network of nodes and lines. In the top right corner of the circle is a circular logo with a blue top half and a red bottom half, and a white spiral in the center. The text '3DEXPERIENCE' is in white, all-caps, sans-serif font, and 'WORLD' is in a much larger, white, all-caps, sans-serif font. Below this, a red rounded rectangle contains the text 'FEBRUARY 1-4, 2026 | HOUSTON, TX' in white, all-caps, sans-serif font.

3DEXPERIENCE[®]
WORLD

FEBRUARY 1-4, 2026 | HOUSTON, TX

WE'RE ALWAYS LISTENING

The Top Ten List on 3DSwym

The screenshot shows the 3DEXPERIENCE 3DSwym interface. At the top, the page title is "The Top Ten List - 3DEXPERIENCE World 2026". A search bar and navigation icons for Post, Media, Idea, and Wiki are visible. A sidebar on the left contains navigation options like Conversations, Communities, Beta, AURA, Calls, Me, and Network.

The main content area features a "Write a Post" input field. Below it is a blue announcement banner titled "Top Ten List Closing Soon! Get Final Votes in Now". The banner includes a graphic for "3DEXPERIENCE WORLD 2026 Top Ten List" and text stating: "Due to some last minute changes, the Top Ten List will close in a couple hours. Please make sure to review and vote for the ideas you would like to see implemented in SOLIDWORKS. Thank you all for your ideas and participation! We are gathering Top Ten List ideas for the following products: ...See more".

Below the announcement is a post titled "Ordinate Dimensions Toolbar: Jog, Re-Jog... right click commands" with a "Gathering Support" tag. The post is by Kevin Chandler, dated 1 month(s) ago, and is moderated. The text of the post reads: "Please create an Ordinate Dim toolbar that contains all of the right click ord dim commands (some of which are nested 2 levels down). People who use ordinate dimensioning I would think do so out of policy and work with ord dims frequently relative to other dimensioning schemes. Having a toolbar for the numerous ord dim adjustments would be handy and for the non-ord dim people, they ...See more". The post has 14 likes, 3 comments, and a share icon.

Below the main post, there are two comments. The first is from Jason Capriotti, dated 3 week(s) ago, with the text: "Maybe they should add to the dimension popup when you select a dimension, and also to the property editor? It has the other display options like 'Parentheses' and 'Show as Inspection' already." This comment has 2 likes and 0 replies. The second comment is from Top Ten List Admin, dated 3 week(s) ago, with the text: "Status changed to: Gathering Support".

WE'RE ALWAYS LISTENING

How We Gather Feedback From Users

- *Alpha Testing & Usability Testing (PD & UX)*
- *Enhancement Requests (thousands per year)*
- *Forums (thousands of discussions)*
- *Customer Visits*
- *User Group Meetings*
- *VARs, Meetings with VAR AEs, etc.*
- *Tech Support*
- *The Top Ten List*

WE'RE ALWAYS LISTENING – HOW WE MINE THIS VAST DATA

Phase 2: Data Loading

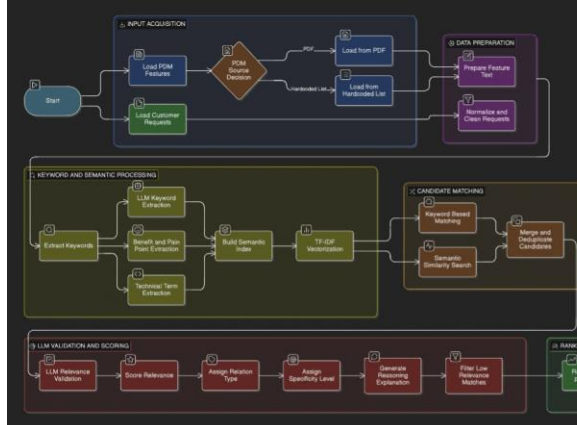
[Loading Historical Data: 2011-2024]
Mode: COMPREHENSIVE (fetching from 30Days)

- Loading 2011 (old format) ...
- Loading 2012 (old format) ...
- Loading 2013 (old format) ...
- Loading 2014 (old format) ...
- Loading 2015 (old format) ...
- Loading 2016 (old format) ...
- Loading 2017 (old format) ...
- Loading 2018 (old format) ...
- Loading 2019 (old format) ...
- Loading 2020 (new format) ... [fetching from 30Days]
- Loading 2021 (new format) ... [fetching from 30Days]
- Loading 2022 (new format) ... [fetching from 30Days]
- Loading 2023 (new format) ... [fetching from 30Days]
- Loading 2024 (new format) ... [fetching from 30Days]

Total Ingested: 6989 requests
With descriptions: 6886 (99.0%)
Titles only: 34
Loaded: 6700 requests

Phase 3: Feature Extraction & Keyword Generation

Generating keywords: 65
Extracting keywords for Feature 1...
L1: 36 keywords
Benefits: 5 keywords
Pain points: 23 keywords
Technical: 13 keywords
Total: 30 keywords



Tier	Score Range	Specificity	Meaning	Example
EXCELLENT	80-100	HIGH	User describes SPECIFIC friction with concrete scenario. 2026 feature DIRECTLY solves the exact pain point.	"Why the hiccup when logging into PDM?" → Kerberos auth feature (addresses specific login delay)
STRONG	70-79	HIGH/MEDIUM	Clear connection to user pain. Feature addresses the problem area, though maybe not the exact solution user proposed.	"Can't recover files after crash" → Automated backup feature (directly prevents data loss)
MODERATE	40-69	MEDIUM	Related pain point but less specific. Same functional area but different angle. Feature solves root cause of symptom.	"Need better login experience" → Kerberos auth (improves login but vague request)
MINIMAL	15-39	LOW	Generic/architectural request. Theoretical connection. Same problem space but vague. No concrete user friction described.	"License by user/group not workstation" → Auth feature (related to user mgmt but no pain point)
WEAK	0-14	N/A	Different domain entirely. Pure speculation. No connection to feature. (Rarely shown - filtered out)	"Add CAD modeling tools" → PDM authentication feature (completely unrelated)

ID	Score	Location	Request Details	Feature Manager
1	50	2013 2024	19. Disabling Custom Triggers before Database Upgrade	20. Named BOM and File Details in the Web Client
2	50	2013 2024	20. MODERATE (HIGH) (Weak) EPDM State Change Show Conditions During Transition	21. Data Encryption Standard
3	50	2013 2024	21. MODERATE (HIGH) (Weak) Improve "Add File" Function to better replicate the Toolbar tabs and PM	22. Support for the Kerberos Windows Authentication
4	50	2013 2024	22. The request is a specific pain point about transition conditions, but the 2026 feature about disabling triggers for upgrades does not address it	
5	50	2013 2024	23. MODERATE (HIGH) (Weak) Reduce Saving Time for Drawings with Detailed Mode Data	
6	50	2013 2024	24. MODERATE (MEDIUM) (Weak) Admin System Option Settings	
7	50	2013 2024	25. MODERATE (HIGH) (Weak) Improve the "Check" Command	
8	50	2013 2024	26. MODERATE (HIGH) (Weak) Better Analysis of circular references	
9	50	2013 2024	27. MODERATE (HIGH) (Weak) The ability to mirror your own drawing into a separate workspace	
10	50	2013 2024	28. MODERATE (LOW) (Weak) Design Checker needs to be improved	
11	50	2013 2024	29. MODERATE (HIGH) (Weak) Multiple Worksheet Cut Lists Combined into One Table	
12	50	2013 2024	30. MODERATE (HIGH) (Weak) Add ability to create and apply 3D texture to parts	
13	50	2013 2024	31. MODERATE (HIGH) (Weak) Hide Configuration Tabs in Print Data Card	

ID	Score	Location	Request Details	Feature Manager
14	558	2648	Free Basic SolidWorks version for 3D Printing	
15	559	2679	Ability to have mid plane extrusion as a 2679 default	
16	561		This list was exported on 1/4/21 from SolidWorks World 2014 and the Top Ten List space	
17	562			
18	563			

to get something to be the exact shape/size that you want it to be. It would be nice if the files were compatible with the full version, although they could probably be limited similar to the Student Version. The files could be saved in the standard SW format, with an option to export to STL.

Sketch mode could be limited to 2D sketches with lines, arcs/circles, ellipses, and points. You would probably need all of the relations available, as well as the Trim command, but you could leave out things like Splines, Convert/Offset Entities, Sketch Patterns, and any of the other more advanced sketch capabilities. This has probably been discussed before, but I would really benefit from this.

ID	Score	Location	Request Details	Feature Manager
19	100	2018 4770	1. Hiding and Showing Manager Pane Tabs	2. Viewing Diminished Messages
20	100	2018 6380	3. Usability	4. Selection Filters
21	50	2013 1286	5. MODERATE (HIGH) (Weak) Display Status for drawing BOMs (BOM Views)	
22	50	2016 4736	6. MODERATE (LOW) (Weak) Make assembly component reference data easier to access in drawings	
23	50	2011 1084	7. MODERATE (HIGH) (Weak) Eliminate the need to create copied standards in Toolbars and clean up the data	
24	50	2011 2074	8. MODERATE (HIGH) (Weak) Customize Toolbar Wizard	
25	50	2012 1849	9. MODERATE (HIGH) (Weak) EPDM state and editor tabs controlled by variables instead of user preferences	

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HISTORY OF THE TOP TEN LIST

2001



Orlando

Rank	Idea
1	More flexible BOM
2	Pattern a component pattern
3	Measure model in section view
4	Wrap sketch/text on surface
5	Unfold complex geometry
6	Explode lines/trails
7	Better surfacing tools
8	Measure in eDrawings
9	Interrupt rebuild
10	Improve assembly performance

1
2
3
4
5
6
7
8
9
10



HISTORY OF THE TOP TEN LIST

2002



Las Vegas

Rank	Idea
1	Stop button to abort lengthy operations
2	Pattern a component pattern
3	Measure model in section view
4	Wrap sketch/text on surface
5	Unfold complex geometry
6	Explode lines/trails
7	Better surfacing tools
8	Measure in eDrawings
9	Interrupt rebuild
10	Improve assembly performance

1	1
2	2
3	3
4	4
5	5
6	6
7	7
8	8
9	9
10	10



HISTORY OF THE TOP TEN LIST

2003



Orlando

Rank	Idea
1	Improve drawing performance
2	Improve assembly performance
3	More flexible BOM
4	Limit mates
5	Save as previous version of SOLIDWORKS
6	Create a PDF from SOLIDWORKS
7	Mate more than 2 components at a time
8	Hole charts in drawings
9	Multiple colors/fonts/styles in a single note
10	Material library for density/crosshatch/etc.

1	1	1
2	2	2
3	3	3
4	4	4
5	5	5
6	6	6
7	7	7
8	8	8
9	9	9
10	10	10



HISTORY OF THE TOP TEN LIST

2004



Boston

Rank	Idea
1	Dynamic section view
2	Mold tools
3	Improved assembly mates
4	User interface improvements
5	Weldments

1	1	1	1
2	2	2	2
3	3	3	3
4	4	4	4
5	5	5	5
6	6	6	
7	7	7	
8	8	8	
9	9	9	
10	10	10	



HISTORY OF THE TOP TEN LIST

2005



Orlando

Rank	Idea
1	Copy assembly, its components, and drawings of components all at once
2	Break link of mirrored/derived part and keep features of original part
3	Import dimension of the parent part into the drawing of a mirrored part
4	Display any custom property in a balloon on an assembly drawing
5	Propagate assembly features back into parts as part features
6	Extrude up to multiple set of faces or surfaces
7	Multi-jog leader for all annotation types
8	Mate more than two parts at once
9	Insert tolerances in addition to dimensional value into notes
10	Create more than once exploded view per assembly configuration

1	1	1	1	1
2	2	2	2	2
3	3	3	3	3
4	4	4	4	4
5	5	5	5	5
6	6	6		6
7	7	7		7
8	8	8		8
9	9	9		9
10	10	10		10



HISTORY OF THE TOP TEN LIST

2006



Las Vegas

Rank	Idea
1	Recreate all mates when mirroring components
2	Clearance check in assemblies similar to interference detection
3	Select subassembly via right-click on one of its parts
4	Represent items such as grease, adhesive, and paint in bill of materials
5	Link balloons parametrically to text in a note
6	Copy entire drawing sheet, or a portion of it, into the same drawing or a different one
7	Create component patterns based on a curve/sketch-driven pattern
8	Select cylindrical surface to specify axis of a circular pattern
9	Provide more options to create reference planes and axes
10	Provide same printing options in Task Scheduler as in SOLIDWORKS

1	1	1	1	1	1
2	2	2	2	2	2
3	3	3	3	3	3
4	4	4	4	4	4
5	5	5	5	5	5
6	6	6		6	6
7	7	7		7	7
8	8	8		8	8
9	9	9		9	9
10	10	10		10	10



HISTORY OF THE TOP TEN LIST

2007



New Orleans

Rank	Idea
1	Recreate all mates when mirroring components
2	Propagate assembly par
3	Ability to rename components (filenames) from the assembly
4	Input dimensions in drawing views of a mirrored or derived part
5	Slot feature or slot option in Hole Wizard
6	Flat pattern of a mirrored sheet metal part
7	Title Block Wizard – dialog box to input all title block info
8	Provide support for dual monitors
9	Mirror assembly features
10	Clearance check in assemblies similar to interference detection

1	1	1	1	1	1	1
2	2	2	2	2	2	2
3	3	3	3	3	3	3
4	4	4	4	4	4	4
5	5	5	5	5	5	5
6	6	6		6	6	6
7	7	7		7	7	7
8	8	8		8	8	8
9	9	9		9	9	9
10	10	10		10	10	10



HISTORY OF THE TOP TEN LIST

2008



San Diego

Rank	Idea
1	Allow adding coordinates in detail view based off of the zero in the parent view
2	Fixed length spline
3	Create more than one exploded view per assembly configuration
4	Ability to create a point at the center of mass
5	Mirror assembly features
6	Ability to lock a configuration
7	Clearance check in assemblies similar to interference detection
8	Undo should have a corresponding Redo
9	Recreate all mates when mirroring components
10	Provide support for dual monitors

1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5
6	6	6	/	6	6	6	6
7	7	7		7	7	7	7
8	8	8		8	8	8	8
9	9	9	9	9	9	9	9
10	10	10	10	10	10	10	10



HISTORY OF THE TOP TEN LIST

2009



Orlando

Rank	Idea
1	Increase program stability
2	Clear memory when file is closed
3	SOLIDWORKS backward compatibility
4	Ability to rename configurations while referenced
5	Faster rebuilds on complex models
6	On the fly equation creation & editing
7	Dual monitor support
8	Add lock points for model rebuild
9	Abort any running commands by pressing ESC
10	Excel behavior in tables

1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6
7	7	7		7	7	7	7	7	7
8	8	8		8	8	8	8	8	8
9	9	9		9	9	9	9	9	9
10	10	10		10	10	10	10	10	10



HISTORY OF THE TOP TEN LIST

2010



Anaheim

Rank	Idea
1	<i>SOLIDWORKS should cleanly uninstall itself</i>
2	<i>Increase stability</i>
3	<i>File compatibility between versions</i>
4	<i>On-the-fly equations in dialog boxes</i>
5	<i>Better utilization of processors</i>
6	<i>Allow more types of assembly features</i>
7	<i>Graphical map of references</i>
8	<i>Option to dangle children instead of deleting</i>
9	<i>Exploded views for weldments</i>
10	<i>Simplify video cards requirements</i>

1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6
7	7	7		7	7	7	7	7	7
8	8	8		8	8	8	8	8	8
9	9	9		9	9	9	9	9	9
10	10	10		10	10	10	10	10	10



HISTORY OF THE TOP TEN LIST

2011



San Antonio

Rank	Idea
1	Angle mates that don't flip at random
2	Better utilize processor cores
3	Option to dangle children instead of deleting them
4	SOLIDWORKS should cleanly uninstall itself
5	Freeze features
6	Enhanced equations input, linking, and editing
7	Convert a solid body into a surface body without having to delete, offset, or knit faces
8	Make the 'ideas' section of the forum available all year, not just for SWW
9	File compatibility between versions
10	Pressing ESC key should immediately return control of the interface

1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6	6
7	7	7		7	7	7	7	7	7	7
8	8	8		8	8	8	8	8	8	8
9	9	9		9	9	9	9	9	9	9
10	10	10		10	10	10	10	10	10	10



HISTORY OF THE TOP TEN LIST

2012



San Diego

Rank	Idea
1	Start using more CPU cores whenever possible
2	Mass properties should include option to automatically put a point at center of gravity
3	Don't delete your children, dangle them instead!
4	Address the shaded with edges bleed-through issues
5	Reduce the bounding box of drawing views to visible components only
6	Ability to control a plane's front and back side orientation
7	Dimensions should stay where you place them
8	ESC key should immediately return control of the interface
9	Add 'Lock Rotation' option to cylindrical mates
10	Thread creation wizard

1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6	6	6	6
7	7	7		7	7	7	7	7	7	7	7	7
8	8	8		8	8	8	8	8	8	8	8	8
9	9	9		9	9	9	9	9	9	9	9	9
10	10	10		10	10	10	10	10	10	10	10	10



HISTORY OF THE TOP TEN LIST

2013



Orlando

Rank	Idea
1	Make rebuild time faster
2	True backward compatibility
3	Slot mate
4	Cylindrical mates need an option to lock rotation
5	Provide a version of the eDrawings app for Android
6	Option for equal spacing on linear patterns
7	Default concentric mate for axis to cylindrical surface
8	External thread wizard for all thread types
9	Draw a line segment starting from the midpoint
10	Create auxiliary line for dimensioning angles

1	1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6	6	6	6	6
7	7	7		7	7	7	7	7	7	7	7	7	7
8	8	8		8	8	8	8	8	8	8	8	8	8
9	9	9		9	9	9	9	9	9	9	9	9	9
10	10	10		10	10	10	10	10	10	10	10	10	10



HISTORY OF THE TOP TEN LIST

2014



San Diego

Rank	Idea
1	Equal spacing options for linear patterns
2	Rename components from the FeatureManager tree
3	Draw line from midpoint
4	ESC key should always return control to the UI
5	Unlimited Undo/Redo
6	Redefine angular dimensions after they have been places
7	Redefine a fillet to a chamfer from within the same command (and vice versa)
8	List quantity of identical holes in a hole callout
9	Provide a Purge Data command to reduce file size
10	Auto-hide assembly components while mating new components

1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6	6	6	6	6	6
7	7	7		7	7	7	7	7	7	7	7	7	7	7
8	8	8		8	8	8	8	8	8	8	8	8	8	8
9	9	9		9	9	9	9	9	9	9	9	9	9	9
10	10	10		10	10	10	10	10	10	10	10	10	10	10



HISTORY OF THE TOP TEN LIST

2015



Phoenix

Rank	Idea
1	Support zero thickness geometry
2	Bidirectional sweep
3	Export sheet metal parts of an assembly to DXF/DWG in flat state
4	Selection list box is too small
5	Display the exact item causing the error
6	Ability to make inferred relationships permanent in sketch
7	Selecting inner loops of selected face
8	Both direction pattern
9	Topology optimization from Tosca for all engineers
10	Variable pull direction draft

1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10	10	10	10	10	10	10	10	10	10	10	10	10



HISTORY OF THE TOP TEN LIST

2016



Dallas

Rank	Idea
1	Create a 'classic' mode GUI option in SOLIDWORKS 2016 to emulate SOLIDWORKS 2015
2	Ability to export BOM table with thumbnail images of components
3	Use ALT to temporarily hide pointed face during mating
4	Move assembly sketch to components
5	Bounding box values for any parts and assemblies
6	Enhance chamfer and other functions for multibody parts
7	Automate the explode lines
8	Ability to use endpoints of sketch line for Hole Wizard holes
9	Automated fastener BOM for large assembly
10	Revert to eDrawings 2013 engine and interface for desktops

1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10



HISTORY OF THE TOP TEN LIST

2017



Los Angeles

Rank	Idea
1	Focus more on fixing bugs
2	Substantially increase stability in SOLIDWORKS
3	Improve the performance and quality when importing IGES and STEP files
4	Allow zero-thickness geometry
5	Completely release memory/GDI handle when file is unloaded
6	Export BOM with model thumbnail included in Excel file
7	External references graphical display (like Treehouse)
8	Bounding box value for any parts and assemblies
9	Automatic fastener BOM creation without physically insert the fasteners into the assembly
10	Fully integrated sub-D modelling tool

1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7		7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8		8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9		9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10		10	10	10	10	10	10	10	10	10	10	10	10	10



HISTORY OF THE TOP TEN LIST

2018



Los Angeles

Rank	Idea
1	Allow equations (including IF/THEN statements) in defining custom properties
2	Coatings feature
3	Option for Offset Surface to offset all possible faces
4	Unbend any profile (tube, pipe, bent component), similar to the sheet metal functionality
5	The "What's Wrong" dialog should have links to each specific problems and better descriptions
6	Enhance Cut List table functionality to math BOM table functionality
7	Add ability to project any type of entity (face, component, body, etc) on a plane or surface
8	Make patterns consistent across the board
9	Close parent on "Save as Copy and Open"
10	Add option to limit Coincident Mate to area of face selected, for all types of faces

1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7		7	7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8		8	8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9		9	9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10		10	10	10	10	10	10	10	10	10	10	10	10	10	10



HISTORY OF THE TOP TEN LIST

2019



Dallas

Rank	Idea
1	Flip the new mate, not the old mates
2	Multicore workflow for large assemblies and drawings
3	The "What's Wrong" dialog box should have links to each specific problem & better descriptions
4	Provide better controls to prevent tangent and other mates from flipping
5	Allow SOLIDWORKS to kill a stalled file without having to close SOLIDWORKS
6	Accurate and correct Solid Body Sweep
7	Allow any drawing to be loaded with or without the model
8	Undo/Redo – unlimited/unrestricted
9	FIX Top Ten List, Enhancement Request, and Knowledge Base systems
10	Automatic direction change on Extruded Cut

1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7		7	7	7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8		8	8	8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9		9	9	9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10		10	10	10	10	10	10	10	10	10	10	10	10	10	10	10



HISTORY OF THE TOP TEN LIST

2020



Nashville

Rank	Idea
1	Focus on fixing bugs and increasing stability
2	Ability to set up print orientation automatically
3	PDM should manage SW Material database
4	PDM – Show Cut List items in assembly BOM
5	Show PDM Workflow stages to users, like Manage
6	SW File RMB – Rename: “Rename assy/part with drawing”
7	Toolbox – Need warning about the function of “Make this folder the default search location for toolbox components” option
8	Allow SOLIDWORKS the ability to view/edit/create DXF
9	Markups: Use Design Library for markup reuse
10	Accurate and correct Solid Body Sweep
11	Reset Markup view

1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7		7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8		8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9		9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10		10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10
																			11



HISTORY OF THE TOP TEN LIST

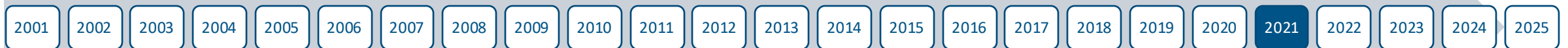
2021



Virtual

Rank	Idea
1	ESC Key should stop the current process and revert to previous state
2	Ability to open a subassembly in Large Design Review mode
3	Variable Angle Draft
4	Drawing Detailing Mode form assembly Large Design Review mode
5	Improve "Re-attach Dimensions"
6	Make property tabs file-type agnostic
7	Add markups to the Large Design Review mode
8	Add the option "Mark for MBD"
9	Make Markups shortcuts for drawing sheets
10	UV mapping for SOLIDWORKS Visualize

1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7		7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8		8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9		9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10		10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10
																				11



HISTORY OF THE TOP TEN LIST

2022



Nashville

Rank	Idea
1	When parent features are suppressed, children should have the option to remain unsuppressed and dangling or having the external relations frozen
2	More functionality for breadcrumbs
3	Assembly Cut-Extrude to have the same end conditions as a Cut-Extrude in Parts
4	Provide a better way of re-attaching dangling dimensions
5	User Interface: Show/Hide all selected entities/features in same step
6	Extruded Cut direction should be where there is material
7	Linear Pattern – Allow staggers
8	Bidirectional Linear Patterns
9	Allow more options for attaching entities to other entities in drawings
10	Break out counterbores to avoid zero thickness geometry

1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7		7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8		8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9		9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10		10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10
																					11



HISTORY OF THE TOP TEN LIST

2023



Nashville

Rank	Idea
1	AutoSolve Dangling Relations and Dimensions in Sketches and Drawings
2	Ability to toggle a selection filter that only allows selections (faces, edges, etc) from a specific component - useful to restrict selections when working in complex assembly
3	Ability to reattach dangling DimXpert Annotation in MBD
4	Ability to create MBD Reference Dimensions between a cylinder and a plane
5	Tell us which file/model is at fault (when presenting an error message)
6	Enhance Markups (anchor to a drawing entity, view, or sheet.)
7	Assembly created MBD (Model Based Definition)
8	Treehouse Improvement: PDM Integration
9	Indicate if Folder Contains Under Defined
10	Features available in assemblies (Make all features that are available for parts, also available for assemblies)

1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7		7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8		8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9		9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10		10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10
																						11



HISTORY OF THE TOP TEN LIST

2024



Dallas

Rank	Idea
1	ESCAPE Key should always stop the current process and revert to the state before it
2	MBD: Ability to create ordinate dimensions in MBD
3	MBD: separate positional tolerance for length and width of a slot
4	MBD: Allow Creating width dimension between Edges/Points (Drafted Features)
5	MBD: Ability to cancel Instant Count for Gtol item
6	Convert Drawing View Dimensions and Annotations to MBD Model Annotations (3D Views)
7	Manage Display States with Tables
8	Mark dimension as MBD Dimension
9	Reduce Saving Time for Drawings with Detailing Mode Data
10	Filter Modified Components in Resolved and LightWeight Modes

1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7		7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8		8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9		9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10		10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10
																						11



HISTORY OF THE TOP TEN LIST

2025



Houston

Rank	Idea
1	MBD: Be able to assign Note and Surface Finish, to a group of faces or a pattern
2	MBD: Add PMI list module in 3D PDF editor
3	Electrical Schematic Interconnect
4	Coating/Treatment object in FeatureManager Tree
5	Material Database Interface: Ability to link/attach a file (.pdf, .doc, etc.) to a material
6	Electrical Schematic Drawings vs 3D Flattened
7	Electrical & PDM – Better link between the two
8	Make Electrical an everything Schematic Tool
9	MBD: Dynamically link to faces defined in compound plane
10	Electrical Splines

1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	
2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7		7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8		8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9		9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10		10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10
																								11

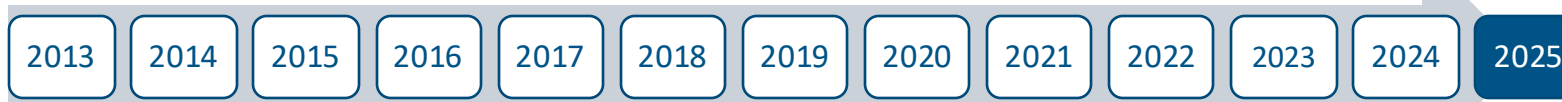


HISTORY OF THE TOP TEN LIST

2025

Rank	Assemblies
1	<i>New BOM Table Editor</i>
2	<i>Mate option "Use for positioning only" in the Quick Mates pop-up tool bar</i>
3	<i>Color settings for components excluded from BOM</i>
4	<i>Additional Rollback Bar in Assemblies</i>
5	<i>Minimize window only, do not update</i>

1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10	10	10	10	10	10	10	10	10	10	10
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HISTORY OF THE TOP TEN LIST

2025

Parts

1 *Patterns features: ability to create pattern based on body AND feature in a single pattern feature*

2 *Cosmetic Knurls*

Assemblies

1 *New BOM Table Editor*

2 *Mate option "Use for position"*

3 *Color settings for components*

4 *Additional Rollback Bar in dialog*

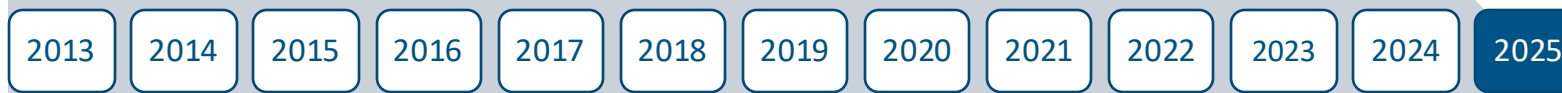
5 *Minimize window only, do not close*

3 *Automate workarounds with export and re-import*

4 *Override Mass Properties Dialog Box – Override Density*

5 *Multi-body Feature*

1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10	10	10	10	10	10	10	10	10	10	10
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HISTORY OF THE TOP TEN LIST

2025

Drawings

1 Add center mark to detailed views if the view does not contain a center mark

2 Improve selection of lines in drawings

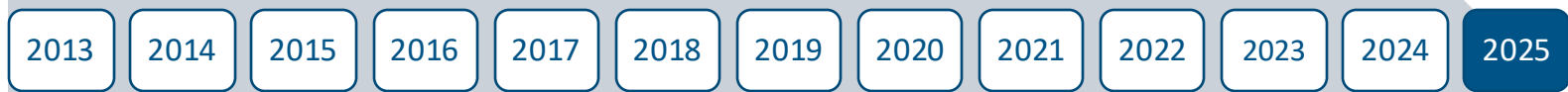
3 The ability to convert any line into a magnetic line

4 Replace model in drawings should display the open documents, just like with Replace component

5 Sort balloons so that the hint lines do not cross

Assemblies	Parts
1 New BOM Table	1 Pattern
2 Mate options	2 Cosmetic
3 Color settings	3 Automate
4 Additional Resources	4 Override
5 Minimize window only, do not close	5 Multi-block

1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10	10	10	10	10	10	10	10	10	10	10
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HISTORY OF THE TOP TEN LIST

2025

Sheet Metal

1 Tab and Slot: Option to give different depth/thickness to both tab and slot

2 Add ability for sheet metal edge flange end condition to use sketch point

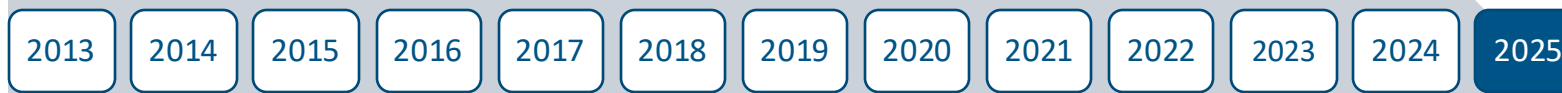
3 Convert radius into lofted bend

4 Create SM-FLAT-PATTERN when creating a sheet metal part

5 Sheet Metal Edge Flange: Leave focus in Edges selection box

Asm	Parts	Drawings
1 New E	1 Patter	1 Ad
2 Mate	2 Cosme	2 Imp
3 Color	3 Autom	3 The
4 Additi	4 Overri	4 Rep
5 Minimize window only, as not	5 Multi-body featu	5 Sor

1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10	10	10	10	10	10	10	10	10	10	10
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HISTORY OF THE TOP TEN LIST

2025

User Experience

1 *Escape button to Cancel all Ongoing Commands*

2 *Show reason for long term actions in status bar*

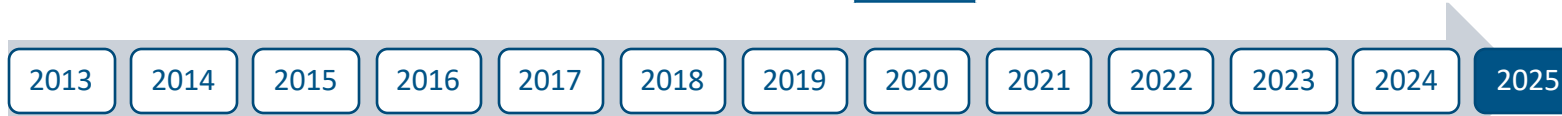
3 *FeatureManager: Stop scrolling the tree when opening a branch*

4 *Access to Material Database from outside a SolidWorks part document*

5 *Make Config. Table easier to read*

Asm	
1	New BOM Table Editor
Parts	
2	
3	1 Patterns feature
Drawings	
4	2
5	3 1
Sheet Metal	
1	2 1 Tab
2	3 2 Add
3	4 3 Con
4	5 4 Create SM LTR / RT LTR when creating a sheet metal part
5	5 Sheet Metal Edge Fillets

1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10	10	10	10	10	10	10	10	10	10	10
												11



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HISTORY OF THE TOP TEN LIST

2025

Sketch

1 Make it easier to change dimensioning direction for slots

2 Perpendicular constraint rather than Vertical Horizontal

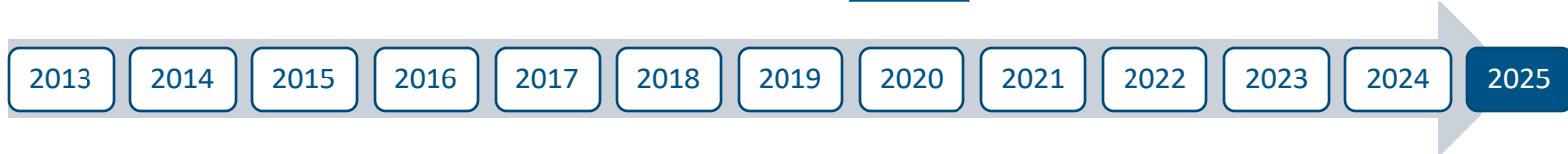
3 Link All Sketch Chamfer Dimensions

4 Mirror sketch with text mirrors the text direction, too

5 Improvement for Sketch Solver

- User Experience
 - Asm 1 Escape button
- Parts
 - 1 Patterns feature tree
- Drawings
 - 2 Add content
- Sheet Metal
 - 1 Tab and
 - 2 Add ab
 - 3 Conver
 - 4 Create SM-FLAT-PATTERN when creating a sheet metal part
 - 5 Sheet Metal Edg

1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10	10	10	10	10	10	10	10	10	10	10
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HISTORY OF THE TOP TEN LIST

2025

MBD

1 Ability to assign Note and Surface Finish, to a group of faces or a pattern

2 Add PMI list module in 3D PDF editor

3 Dynamically link to faces defined in compound plane

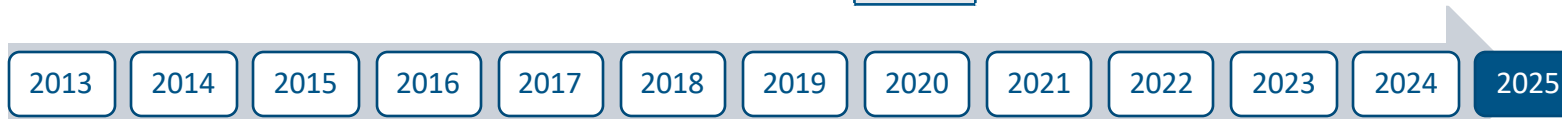
4 Ability to create DimXpert dimension for external thread features

5 Reassign DimXpert Features for a DimXpert dimensions

- 4 Create DIM-FLAT-PATTERN when creating a steel metal part
- 5 Sheet Metal Edge Fillets

1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10	10	10	10	10	10	10	10	10	10	10

11



- 2013
- 2014
- 2015
- 2016
- 2017
- 2018
- 2019
- 2020
- 2021
- 2022
- 2023
- 2024
- 2025

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HISTORY OF THE TOP TEN LIST

2025

Electrical

1 *Make Electrical an everything Schematic Tool*

2 *Electrical Schematic Interconnect*

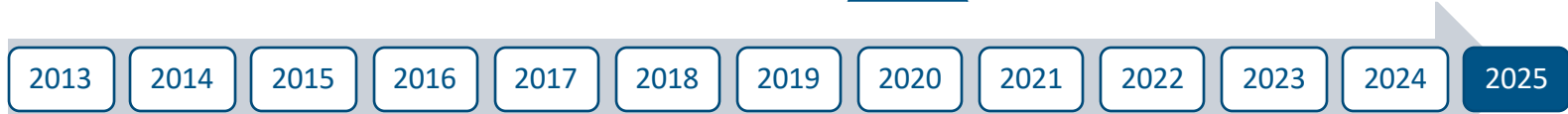
3 *Electrical Schematic Drawings vs 3D Flattened*

4 *Electrical Splines*

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Asm 1 E Sketch
 1 New F 1 M
 Parts
 1 Patterns 2 P
 Drawings
 2
 3 1 Sheet M
 4 3
 5 4 2 1 Tab and
 5 3 2 Add abi
 4 3 Convert
 5 4 Create Sm / E / F / A / E / V when creating a sheet metal part
 5 Sheet Metal Edge Elements for Flat Elements

1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10	10	10	10	10	10	10	10	10	10	10
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HISTORY OF THE TOP TEN LIST

2025

Asm	Parts	Drawings	Sheet Metal	UX	Sketch	MBD	Electrical
1 New BC	1 Patterns feat			1 Escal			
2 Mate op	2 Cosmetic Kn	1 Add center mar		2 Show	1 Make it easier to cha	1 Ability to ass	
3 Color se	3 Automate wo	2 Improve selecti	1 Tab and Slot: Option to give different dep	3 Feat	2 Perpendicular constr	2 Add PMI list	1 Make Electrical an everything Schematic Tool
4 Addition	4 Override Mas	3 The ability to col	2 Add ability for sheet metal edge flange e	4 Acce	3 Link All Sketch Chan	3 Dynamically	2 Electrical Schematic Interconnect
5 Minimiz	5 Multi-body Fe	4 Replace model i	3 Convert radius into lofted bend	5 Mak	4 Mirror sketch with tex	4 Ability to cre	3 Electrical Schematic Drawings vs 3D Flattened
		5 Sort balloons so	4 Create SM-FLAT-PATTERN when creating a sheet metal part		5 Improvement for Ske	5 Reassign Di	4 Electrical Splines
			5 Sheet Metal Edge Flange: Leave focus in Edges selection box				

1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10	10	10	10	10	10	10	10	10	10	10
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2013 2014 2015 2016 2017 2018 2019 2020 2021 2022 2023 2024 2025

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HISTORY OF THE TOP TEN LIST

2025



Houston

Asm					UX													
1	New BOM Table Editor				1	Escape button to Cancel all Ongoing Commands												
2	Material option "Use for positioning only" in the Quick Move pop				2	Show re status b	Sketch											
Parts					3	Feature when op	1	Make it easier to change dim	MBD									
3	Col 1	Patterns features: ability to create pattern l			4	Access a SolidV	2	Perf that	1	Ability to assign Note and Surface Finish, to a group of faces or a pattern	Electrical							
4	Adv 2	Cosm	Drawings		5	Make C	3	Link Dim	2									
5	Mir 3	Auto	1	Add center mark to detailed vi contain a	Sheet Metal					4	Mirr the	3						
			4	Over	2	Improve	1	Tab and Slot: Option to give d depth/thickness to both tab and slot	5	Imp	4							
			5	Multi	3	The abili	2	Add ability for sheet metal edge flange er condition to use sketch point			5							
					4	Replace documen	3	Convert radius into lofted bend										
					5	Sort ball	4	Create SM-FLAT-PATTERN when creating a sheet metal part										

1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4	4
5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5	5
6	6	6		6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6	6
7	7	7		7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7	7
8	8	8		8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8	8
9	9	9		9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9	9
10	10	10		10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10	10
																								11





CUSTOMER-DRIVEN ENHANCEMENTS

2026

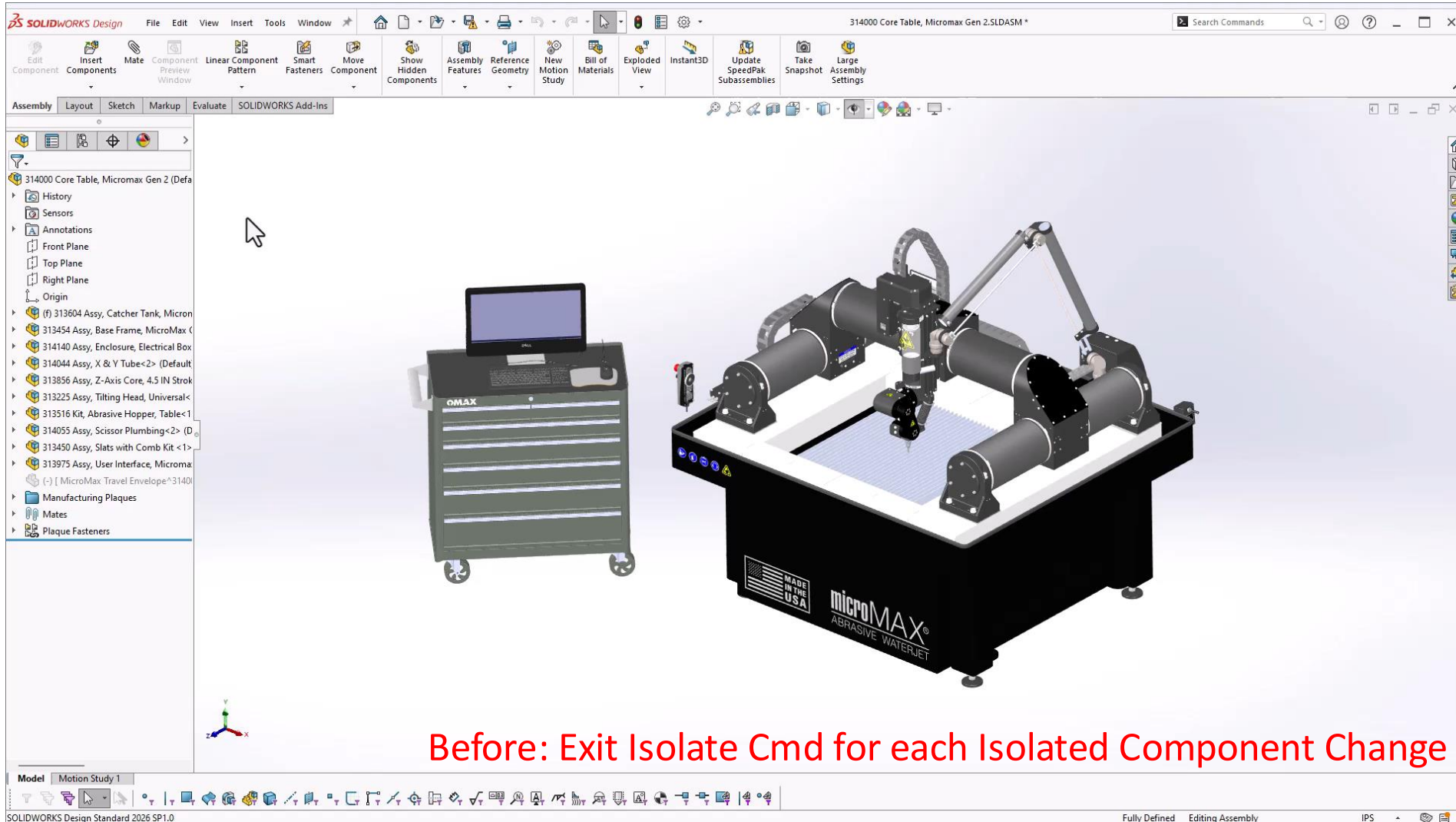


ASSEMBLIES

Customer-Driven Enhancements In SOLIDWORKS 2026

ABILITY TO ISOLATE COMPONENTS AGAIN WHILE IN ISOLATE MODE

Switch between different sets of isolated components in one click

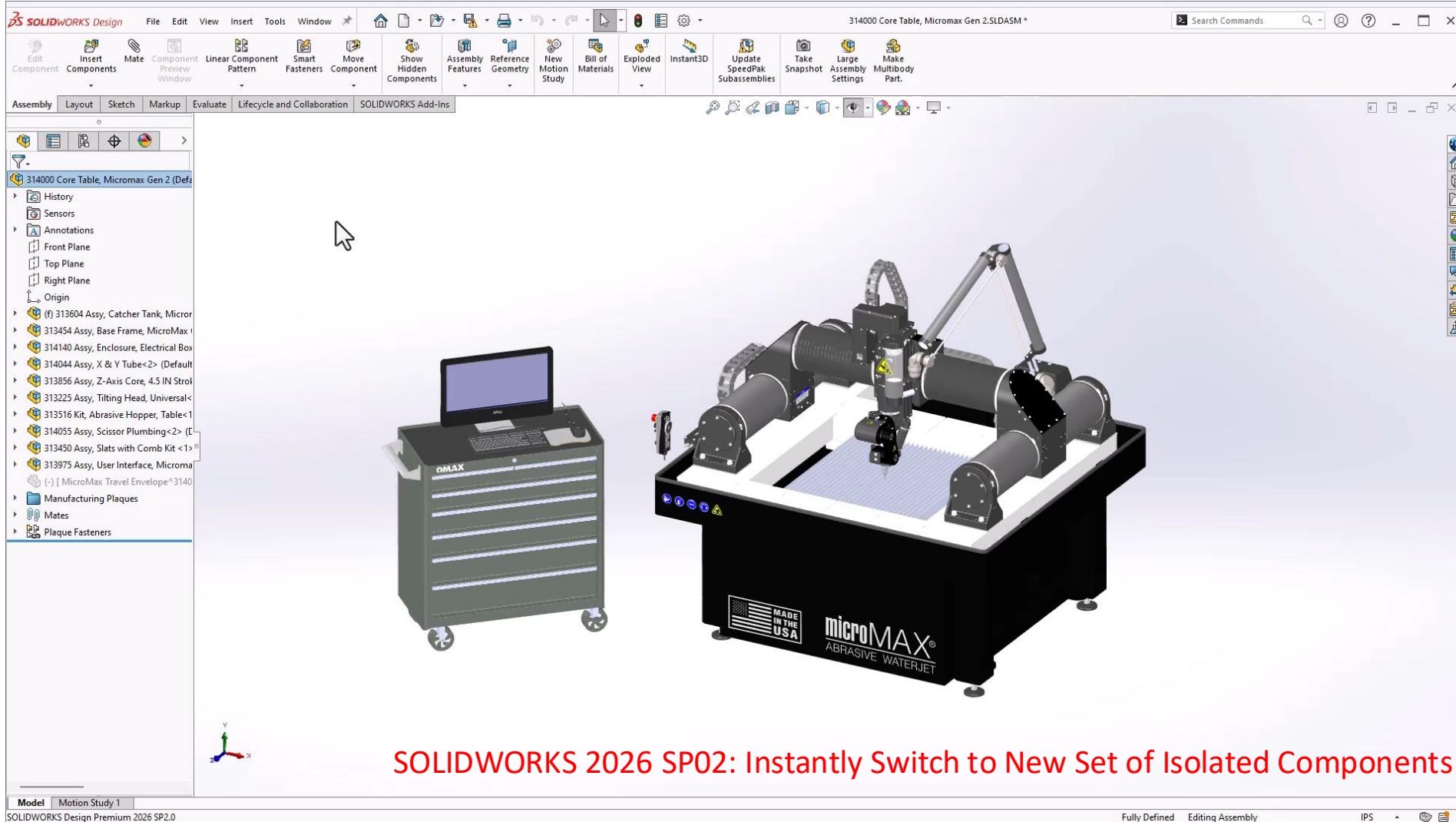


#20
Top Ten 2020

Before: Exit Isolate Cmd for each Isolated Component Change

ABILITY TO ISOLATE COMPONENTS AGAIN WHILE IN ISOLATE MODE

Switch between different sets of isolated components in one click



#20
Top Ten 2020

SOLIDWORKS 2026 SP02: Instantly Switch to New Set of Isolated Components

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LOAD ASSEMBLY BY ZONES USING SCRIPTS / RULES

Use the power of 3DEXPERIENCE Platform Filters for selective loading of large assemblies

#41
Top Ten 2021



💡 Turn Off "Fixed" Column in Simplified Design Tables

Gathering Support



Glenn Schroeder 2 year(s) ago

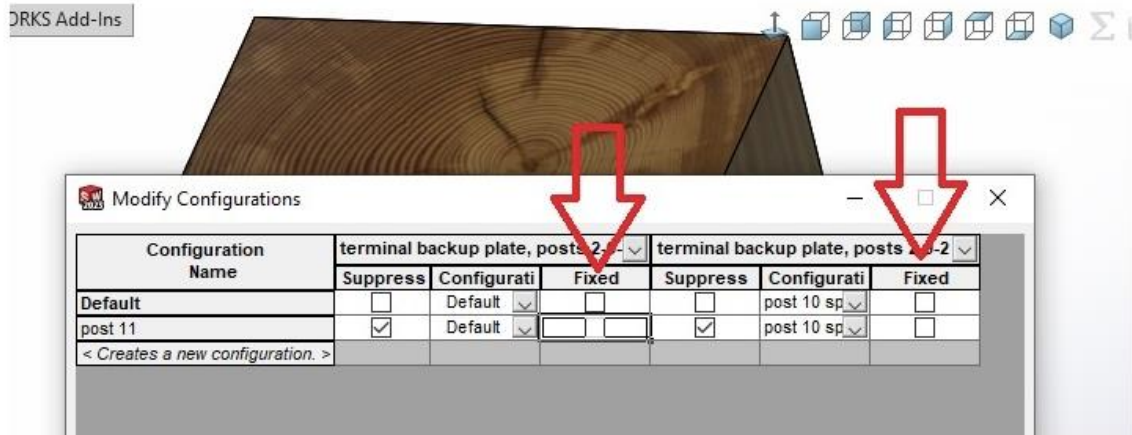
The Top Ten List - 3DEXPERIENCE World 2024

20



This column was added by default a year or so ago. It has caused me more work instead of less. There have even been times when this would cause a component that was Fixed to become Floating without any action on my part.

The chances of me ever wanting it there are very near zero. Please give an option to turn it off.

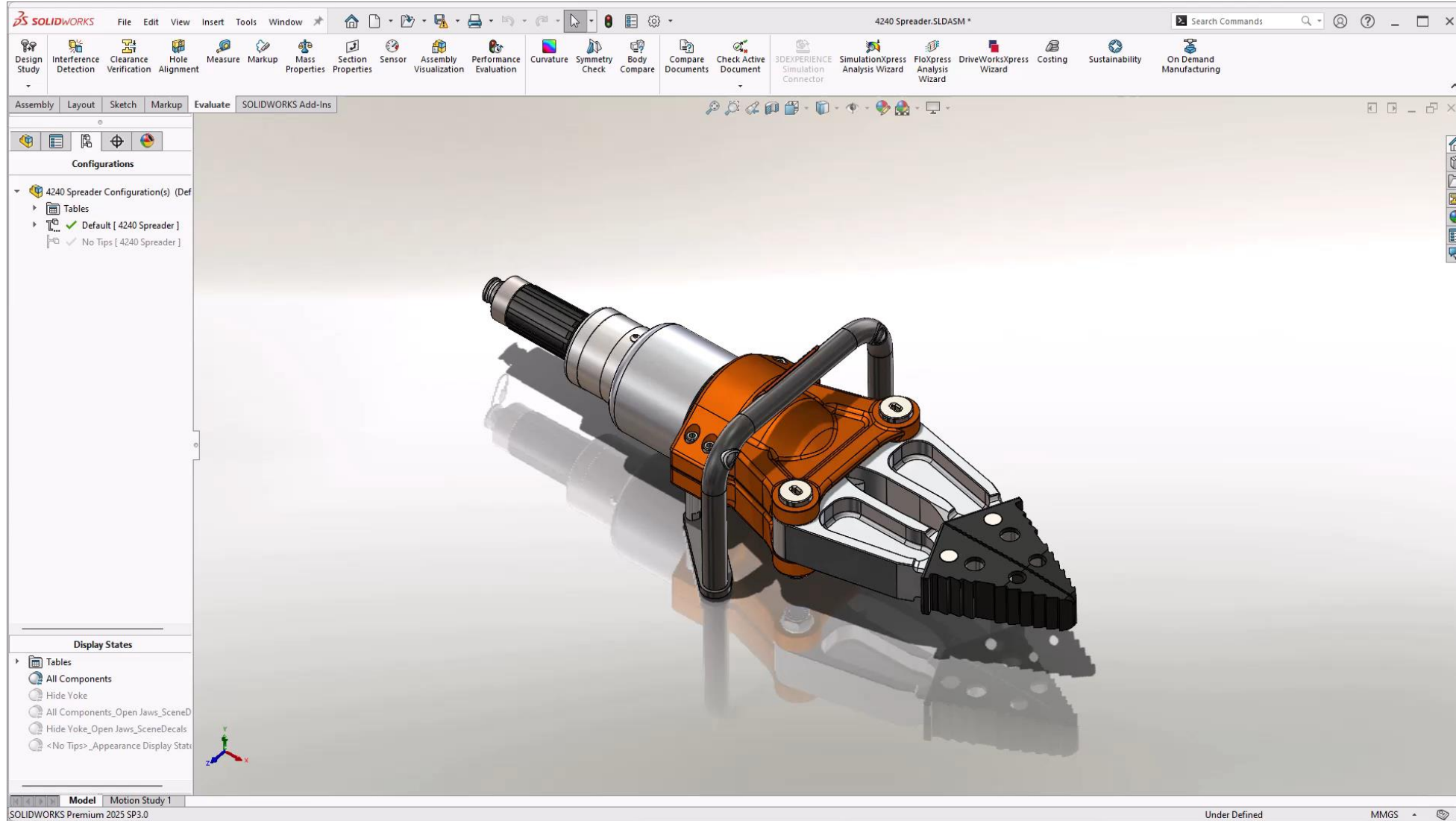


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TURN OFF “FIXED” COLUMN IN SIMPLIFIED DESIGN TABLES

Improve clarity of Configuration Tables

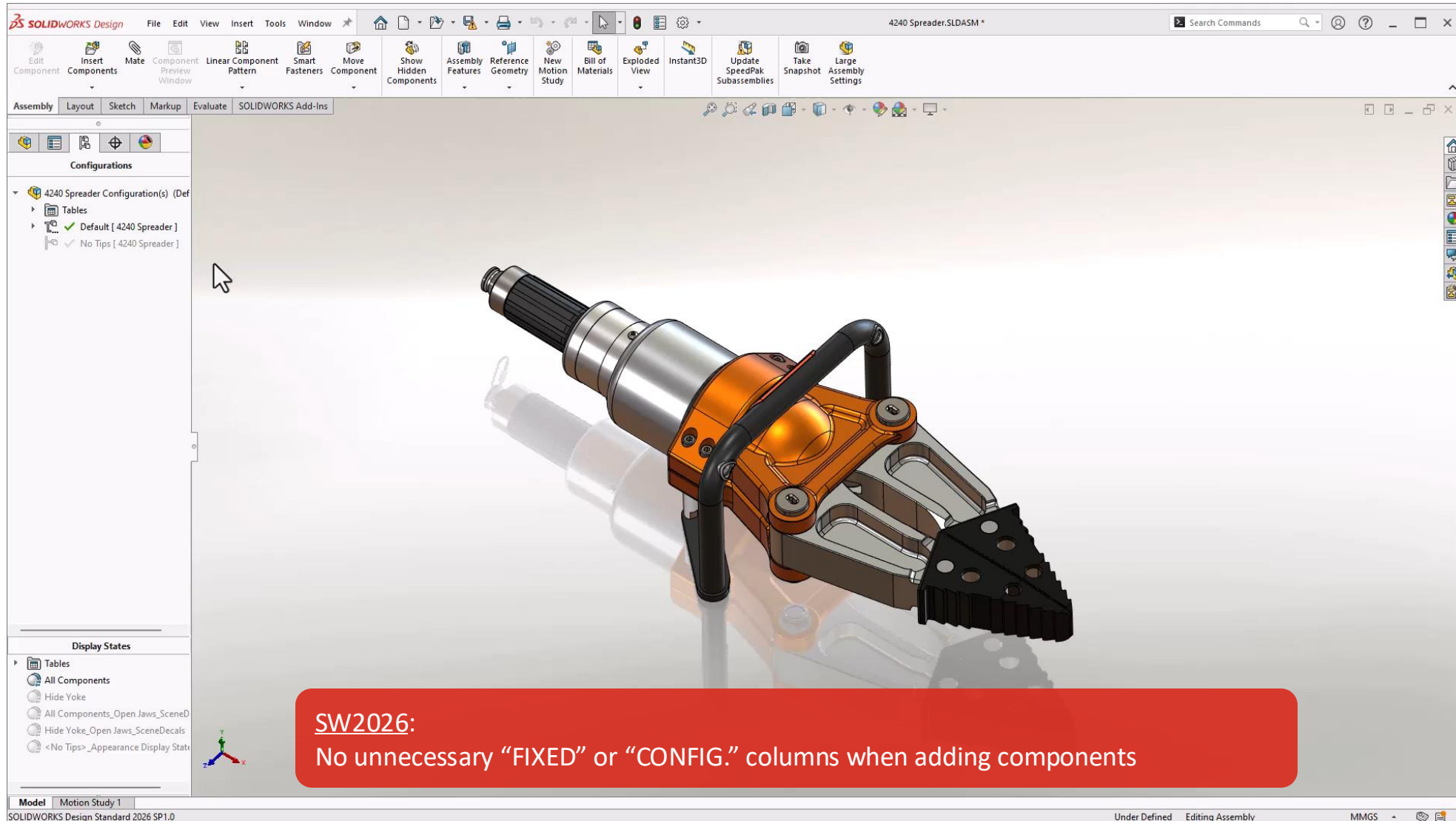
#188
Top Ten 2024



TURN OFF “FIXED” COLUMN IN SIMPLIFIED DESIGN TABLES

Improve clarity of Configuration Tables

#188
Top Ten 2024



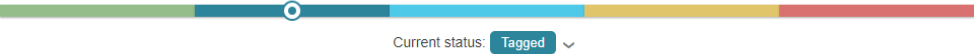
SW2026:
No unnecessary “FIXED” or “CONFIG.” columns when adding components

MAKE SOLIDWORKS GO MULTI-CORE (ONGOING EFFORT)

Background import of models without interrupting your workflows

Make SolidWorks go multi-core. Tagged

Kevin Chandler 4 year(s) ago
 The Top Ten List - 3DEXPERIENCE World 2022

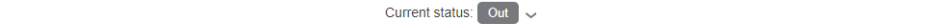


systems
 Make SolidWorks take advantage of multi-core architecture.

Related links +
 Like 22 Comment 8 Share

Multi Core Out

David Matula 2 year(s) ago
 The Top Ten List - 3DEXPERIENCE World 2024



We have computers that can be smoking fast. When will we get a Version of SolidWorks that will use the full capability of the computers. It would be nice to be able to spend more time modeling and drawing.

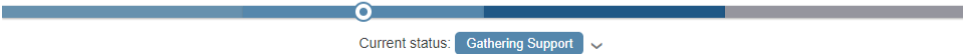
Frequency of Use	Time Savings
many times per hour	hours

user interface	SOLIDWORKS 2017
----------------	-----------------

Core SOLIDWORKS

Add multicore processing Gathering Support

William Alterf 3 year(s) ago
 TTop Ten List - 3DEXPERIENCE World 2023



systems
 I have a brand new, insanely-expensive I9 laptop. Processing never exceeds 10% CPU usage, yet I sit waiting for it for LONG periods of time because my part is so complicated. When I check the CPU monitor, I find that only ONE of the 20 processors is doing anything and it's pegged at 100%. Intel and AMD have both been selling consumer-level multi-core processors since 2005, isn't it time that Solidworks could use two processors at the same time?

Related links +
 Like 9 Comment 8 Share

Cores one Or Multi Gathering Support

David Matula 1 year(s) ago
 The Top Ten List - 3DEXPERIENCE World 2025



Some processes of SolidWorks only use one Core on the computer with multiple cores. This can cause some files to open more slower than others. Remove this limitation and allow the program to use all the processing power that there is on the computer to open and rebuild files faster. Some of the addins can use multi core processes why not the rest of the program.

Frequency of Use	Time Savings
many times per hour	hours

systems	SOLIDWORKS 2024
---------	-----------------

Core SOLIDWORKS

#19
 Top Ten 2022

#131
 Top Ten 2023

#39
 Top Ten 2024

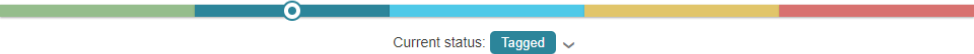
#227
 Top Ten 2025

MAKE SOLIDWORKS GO MULTI-CORE (ONGOING EFFORT)

Background import of models without interrupting your workflows

Make SolidWorks go multi-core. Tagged

Kevin Chandler 4 year(s) ago
 The Top Ten List - 3DEXPERIENCE World 2022

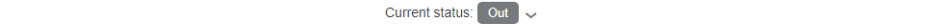


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Related links +
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David Matula 2 year(s) ago
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many times per hour	hours

user interface	SOLIDWORKS 2017
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Related links +
 Like 9 Comment 8 Share

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David Matula 1 year(s) ago
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Frequency of Use	Time Savings
many times per hour	hours

systems	SOLIDWORKS 2024
---------	-----------------

Core SOLIDWORKS

#19
 Top Ten 2022

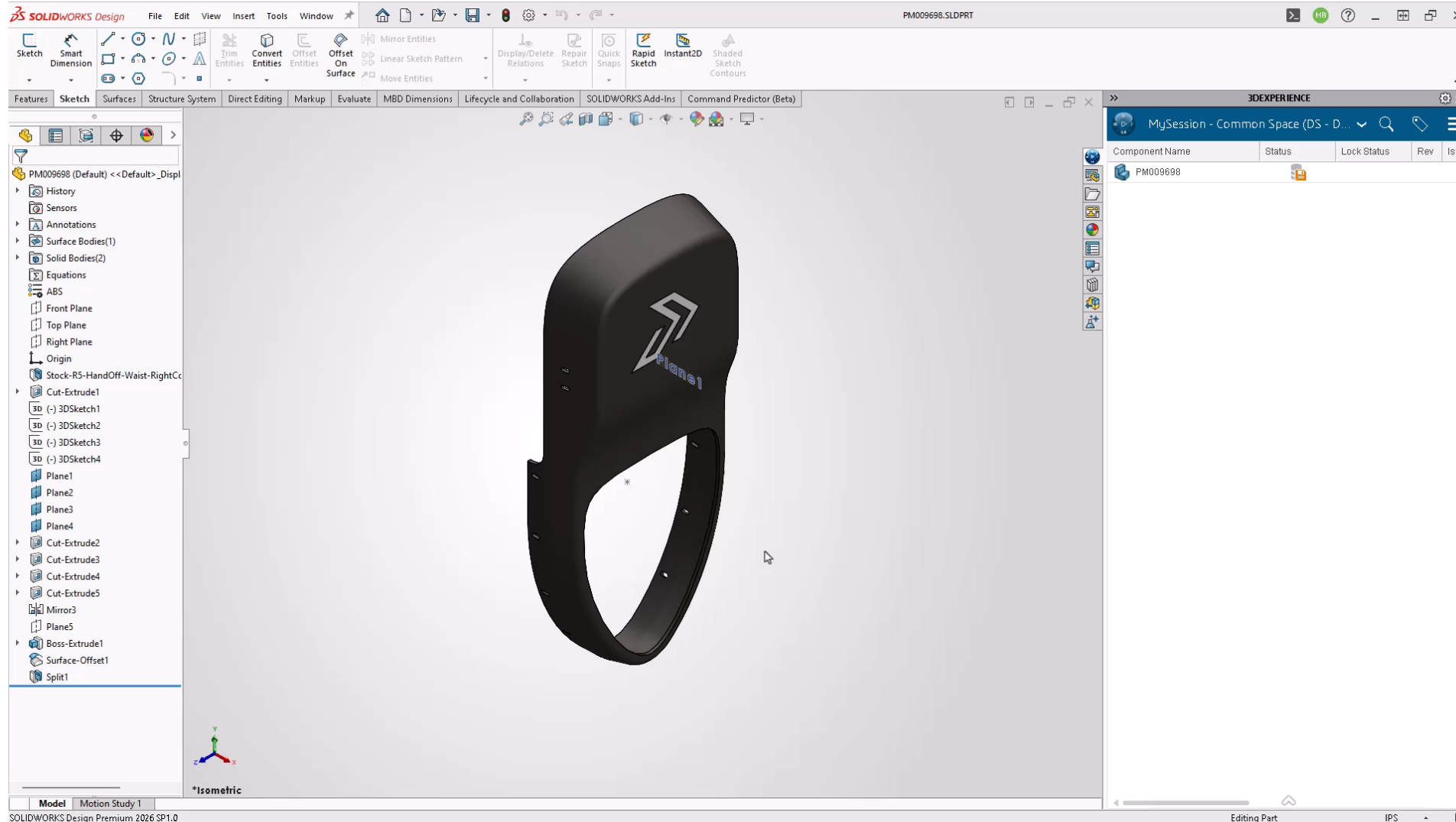
#131
 Top Ten 2023

#39
 Top Ten 2024

#227
 Top Ten 2025

MAKE SOLIDWORKS GO MULTI-CORE (ONGOING EFFORT)

Background import of models without interrupting your workflows



#19
Top Ten 2022

#131
Top Ten 2023

#39
Top Ten 2024

#227
Top Ten 2025

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PARTS

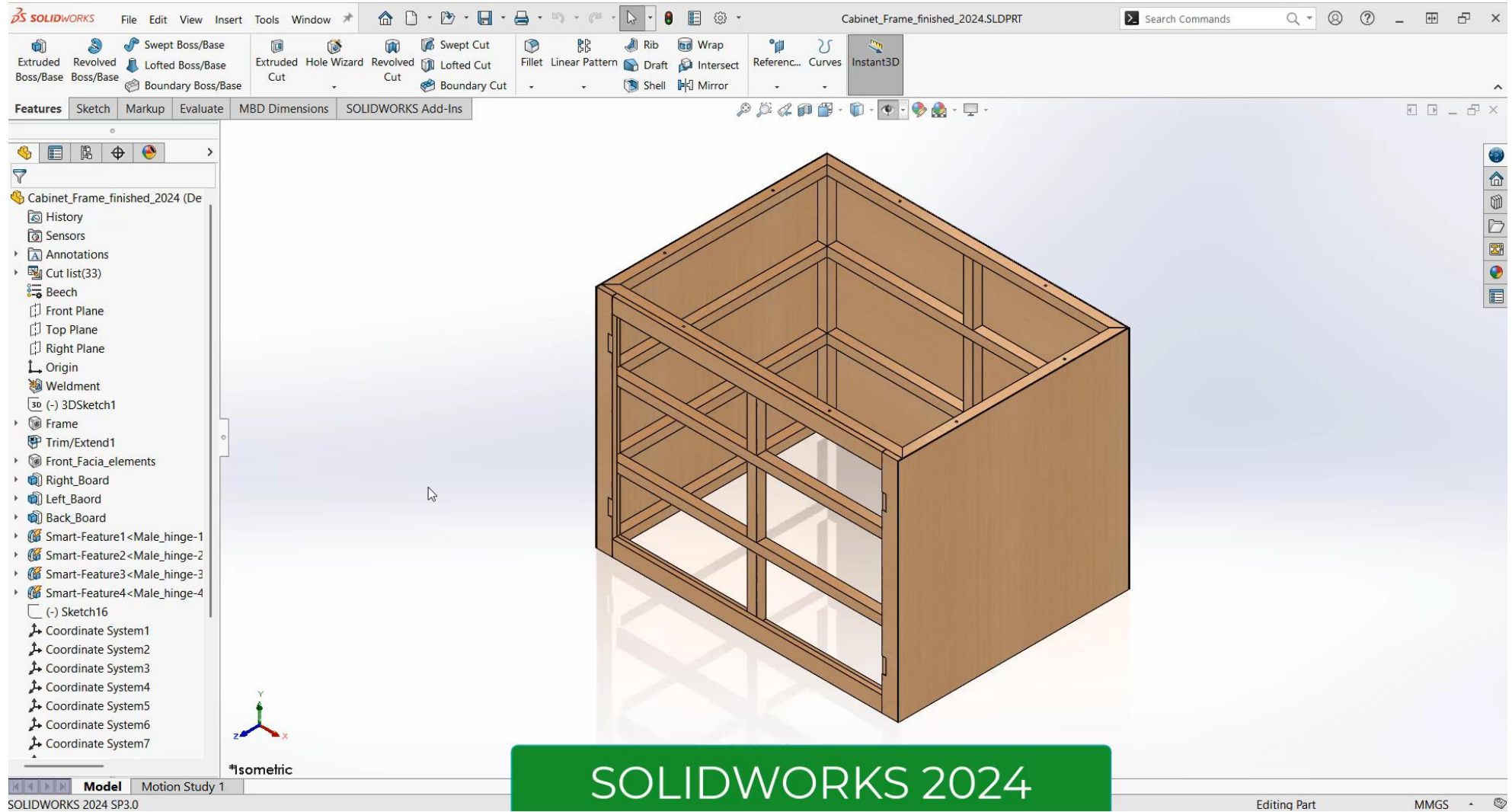
Customer-Driven Enhancements In SOLIDWORKS 2026

APPLY FILLETS TO MULTI-BODY PARTS IN ONE GO

#229
Top Ten 2025

#75
Top Ten 2023

#31
Top Ten 2022

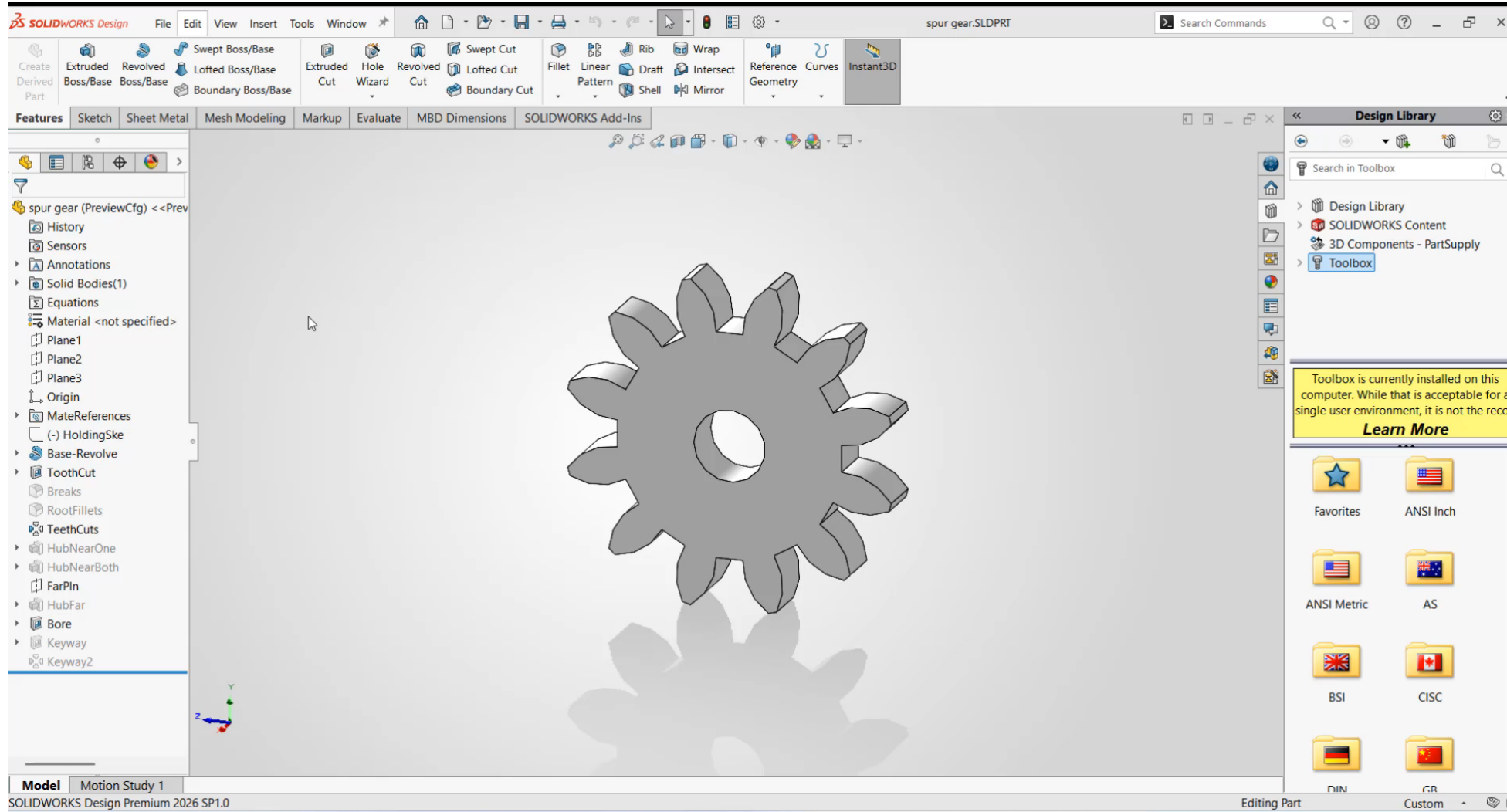


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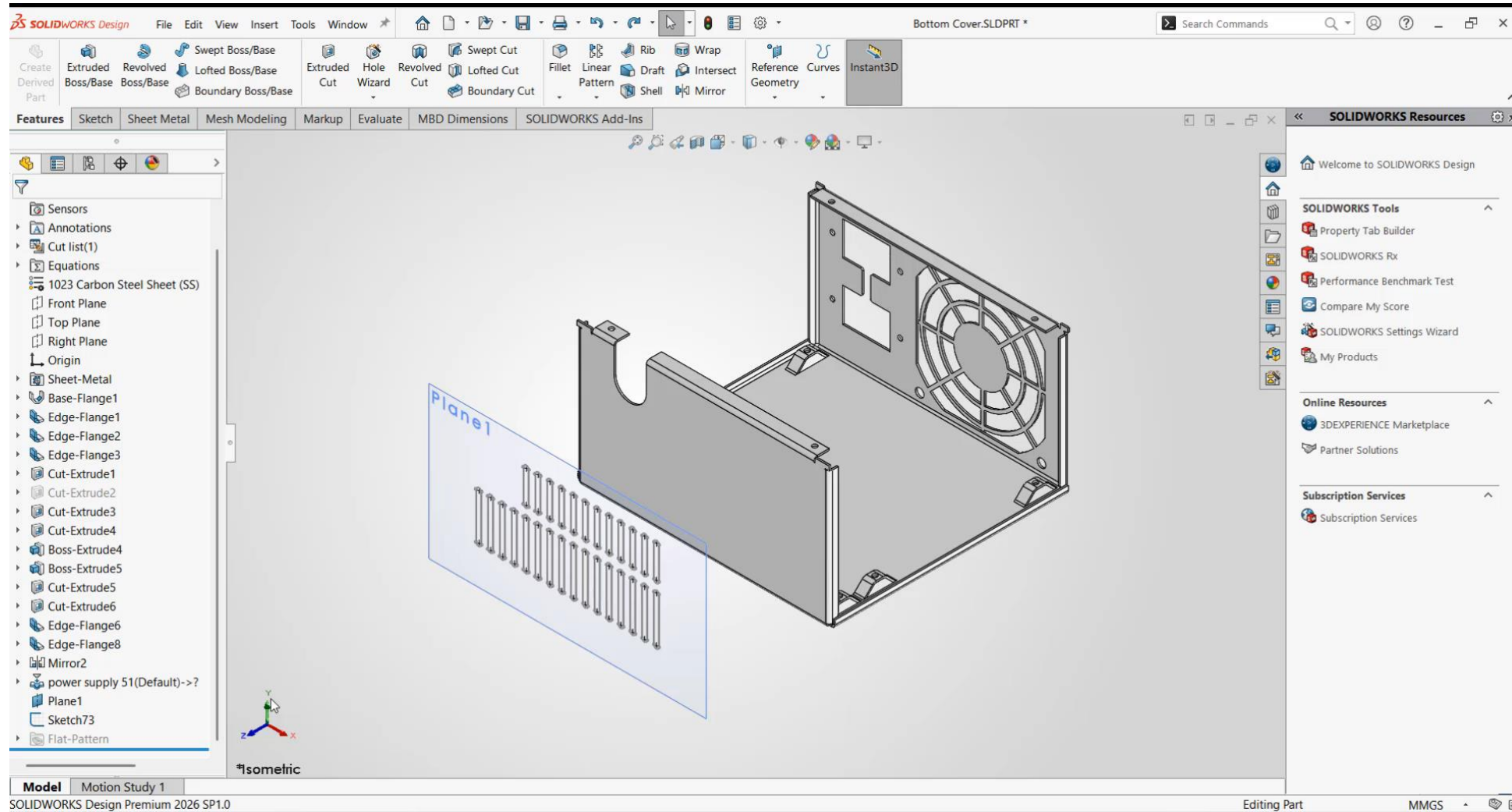
DELETE ASSOCIATED EQUATIONS WHEN DELETING THE SKETCH/FEATURE

More control over how the deletion of sketches or features impacts related equations

#44
Top Ten 2024



CUT-EXTRUDE DIRECTION ALWAYS TOWARDS THE MATERIAL SIDE

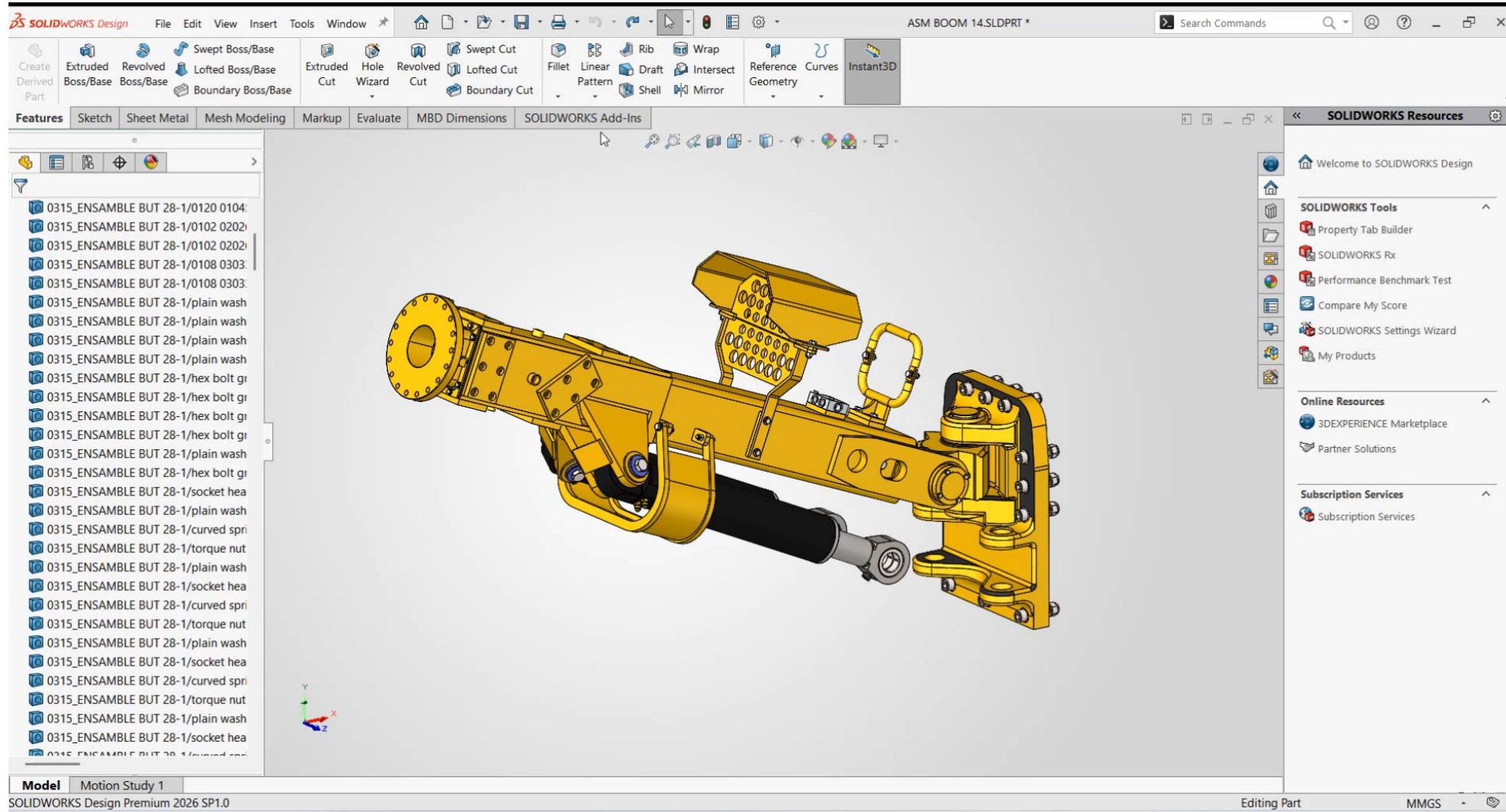


#57
Top Ten 2022

SELECTING BODIES AND FEATURES IN MULTIBODY PARTS

Lets you select discrete bodies and features of imported multibody parts by Size or Volume:

#87
Top Ten 2019



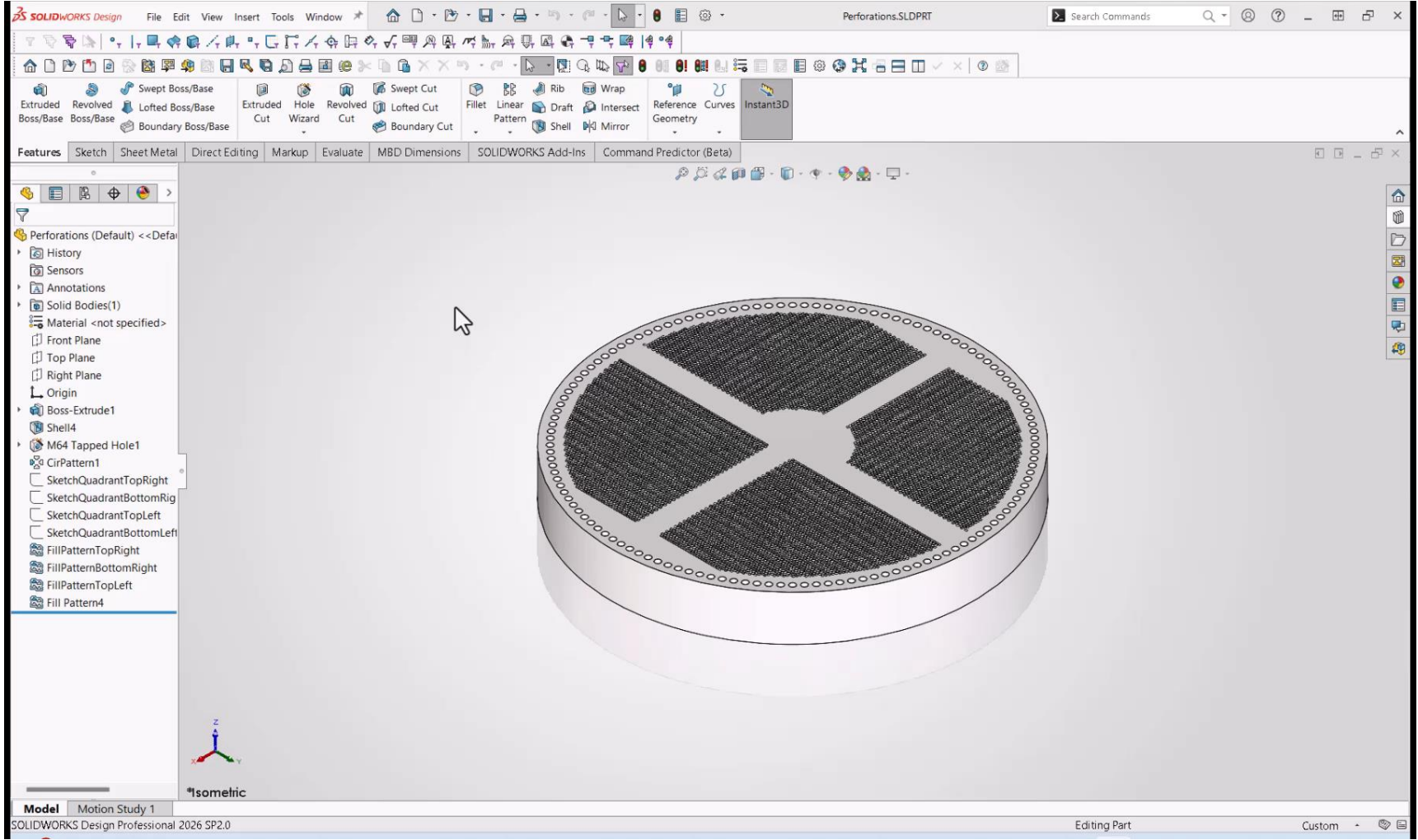
EXITING PART PROCESSES WITH THE ESCAPE KEY

Esc Key will always stop the current process and revert to the state before it

#1
Top Ten 2021

#69
Top Ten 2023

#1
Top Ten 2024



Combine







Intersect



Shell

2025 (SP02)



-  Linear Pattern
-  Circular Pattern
-  Fillet
-  Chamfer

2026



DRAWINGS

Customer-Driven Enhancements In SOLIDWORKS 2026

BREAK DIMENSION AROUND ANNOTATIONS

Break dimension lines around intersecting and overlapping dimension text and annotations.

The screenshot displays the SolidWorks CAD interface for a drawing titled "LSD Main Shaft Assembly - Sheet1". The software's ribbon and toolbars are visible at the top. The main drawing area shows a technical drawing of a main shaft assembly with various views: a front view, a side view, and a detail view of a gear. Dimension lines are used to specify lengths and diameters. A key feature is the use of break dimension lines (indicated by orange lines) to allow dimension lines to pass over dimension text and annotations without being obscured. For example, a dimension line for a length of 354.46 mm is shown with a break, passing over the dimension value and other text.

Annotations include a note: "Ensure 'Grease' is applied before assembling the gears onto the shaft".

The Bill of Materials (BOM) table is visible in the upper right corner of the drawing area:

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	6000M	Mainshaft	1
2	60846M	Gear	1
3	62941M	Gear	1
4	61234M	Gear	1
5	60303C	Gear	1
6	62212M	Gear	1
7	61005M	5th Main Gear	1
8	Spacer	Sprocket 33344-94	1
9	Guard Seal		1
10	12035B	Seal Main Gear	1
11	34091-85	Inner race	1
12	HC250	Bearing Main Gear	2
13	8876A	Bearing	3
14	11047	Snap Ring	3
15	Thrust Washer	6003B	4
16	Clutch Release Rod		1
17	AL2035	Shim	1
18	F4945A235		1

The drawing also includes a detail view of a gear with dimensions: $\phi 35.09$, $\phi 85.09$, and $\phi 77.79$. A note indicates a chamfer: $3/16 \times 60^\circ$. The drawing is set on a grid with columns 1-12 and rows A-H. The status bar at the bottom shows "LSD Main Shaft Assembly", "160.08mm 302.94mm 0mm Under Defined Editing Sheet1 1 : 1.5 MMGS".

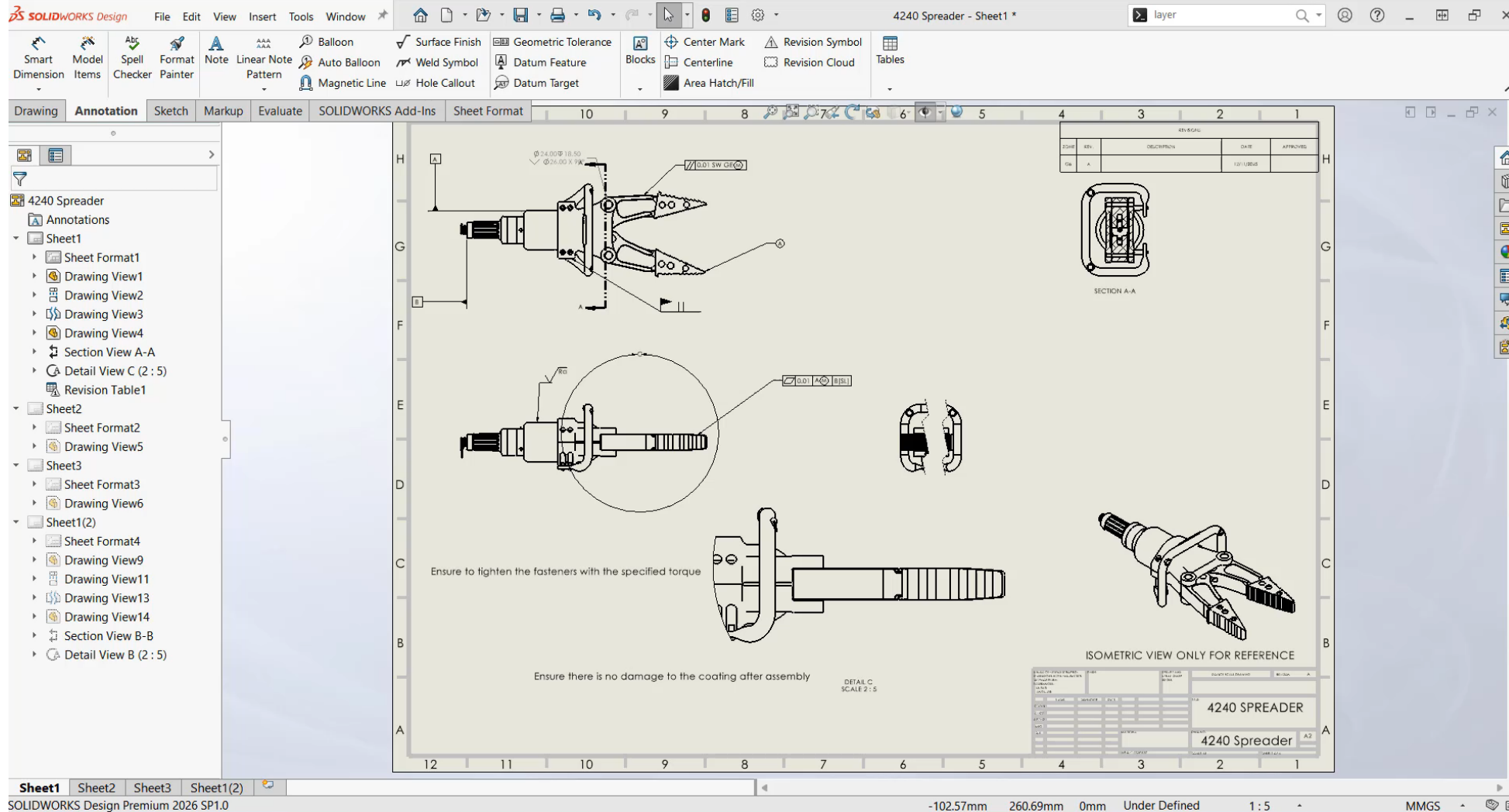
#47
Top Ten 2023

SW EI
Community
150 Likes

ALIGN ANNOTATIONS WITH MAGNETIC LINES

Use Magnetic Lines to align more than just balloons.

#149
Top Ten 2025





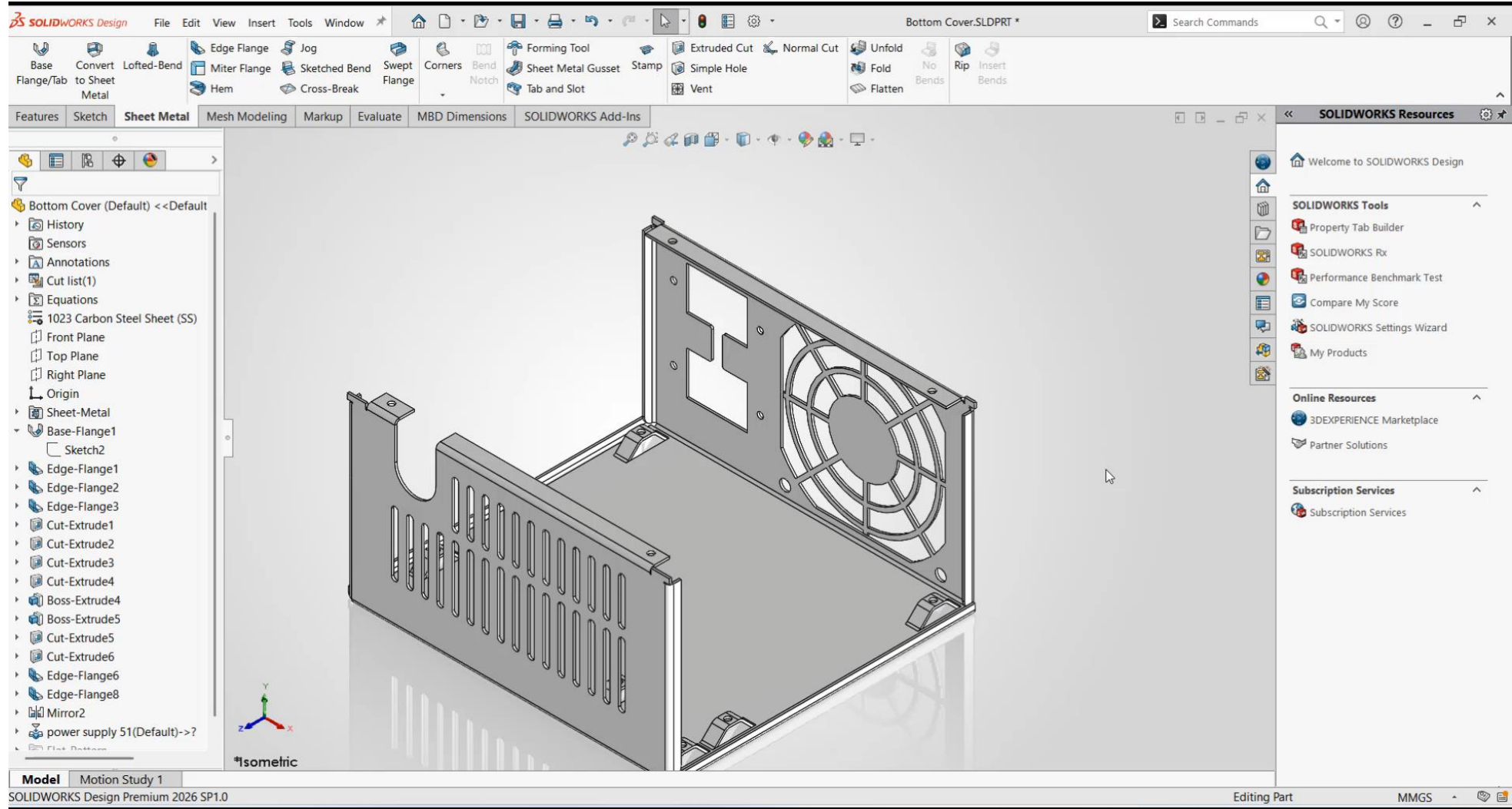
SHEET METAL

Customer-Driven Enhancements In SOLIDWORKS 2026

ADDITIONAL START CONDITIONS FOR SHEET METAL BASE FLANGE (2025 SP2.0)

Control the 'Start Conditions' for the base flange - From: Surface/Face/Plane, Vertex and Offset

#38
Top Ten 2021

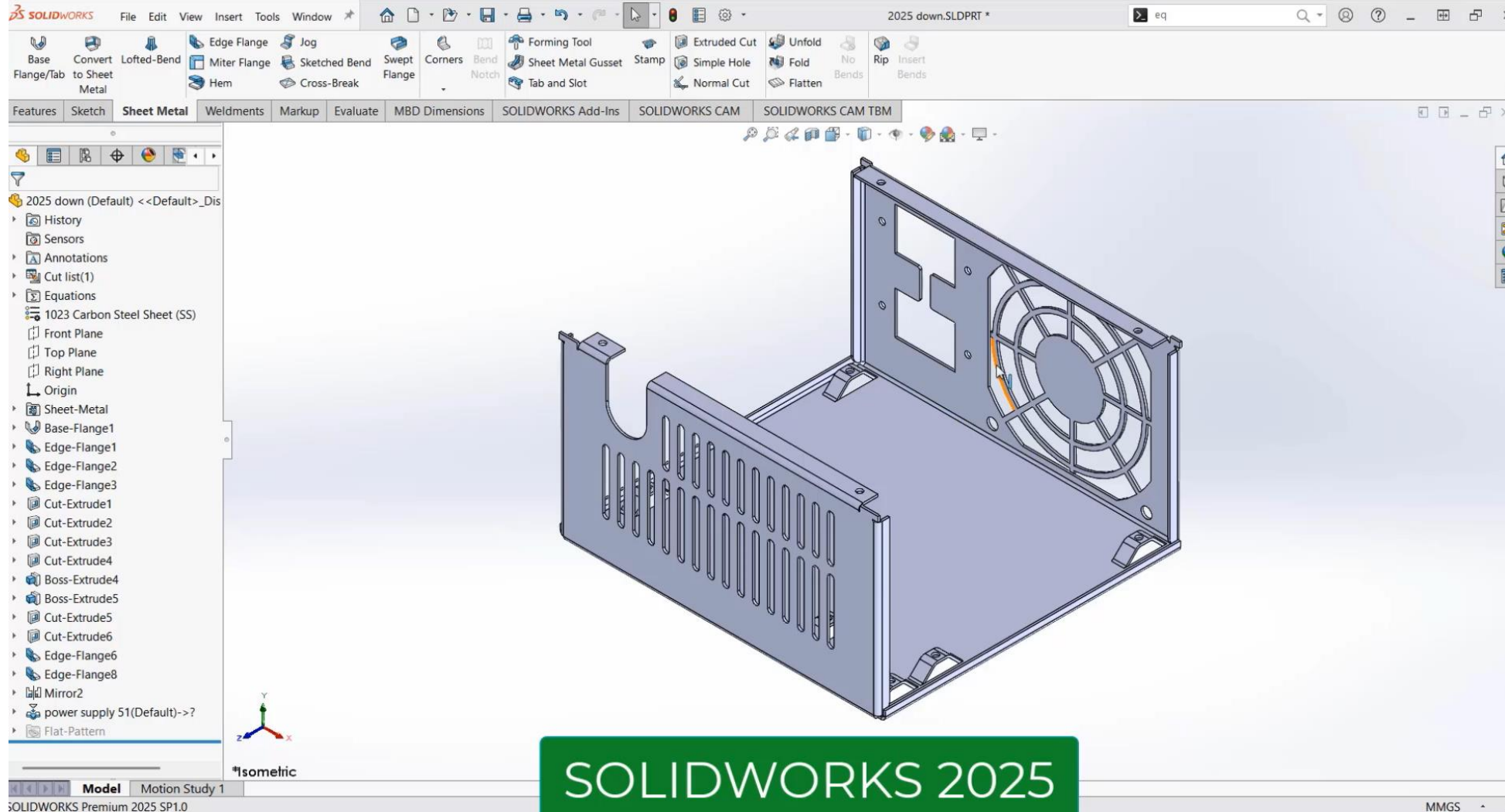


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BREAK EDGE FOR INTERIOR CORNERS

Select a sheet metal face to add break corners to inner corners in one go

#38
Top Ten 2022



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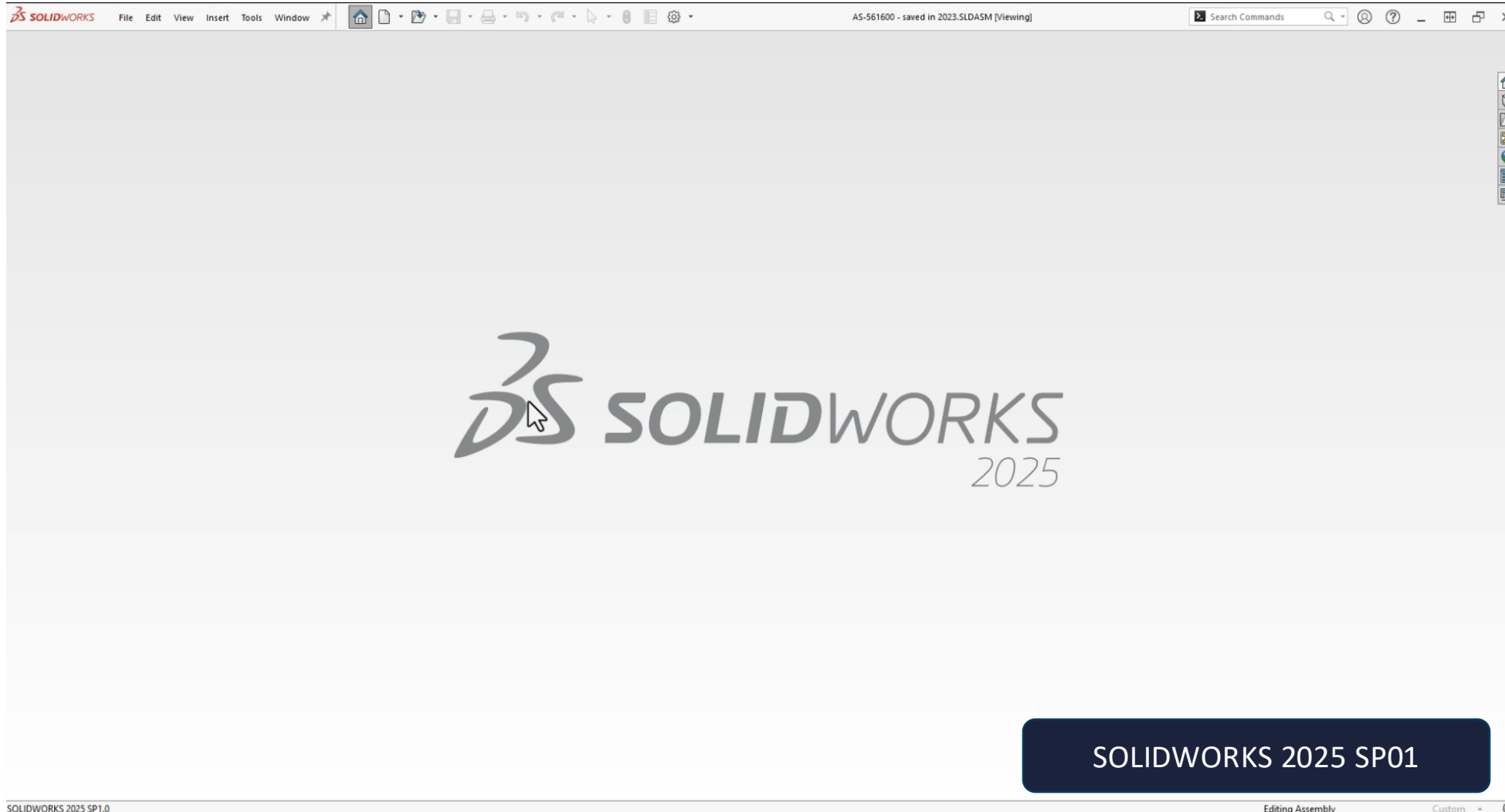


USER EXPERIENCE

Customer-Driven Enhancements In SOLIDWORKS 2026

DISMISSED MESSAGES

Show the full text of Dismissed Messages in System Options

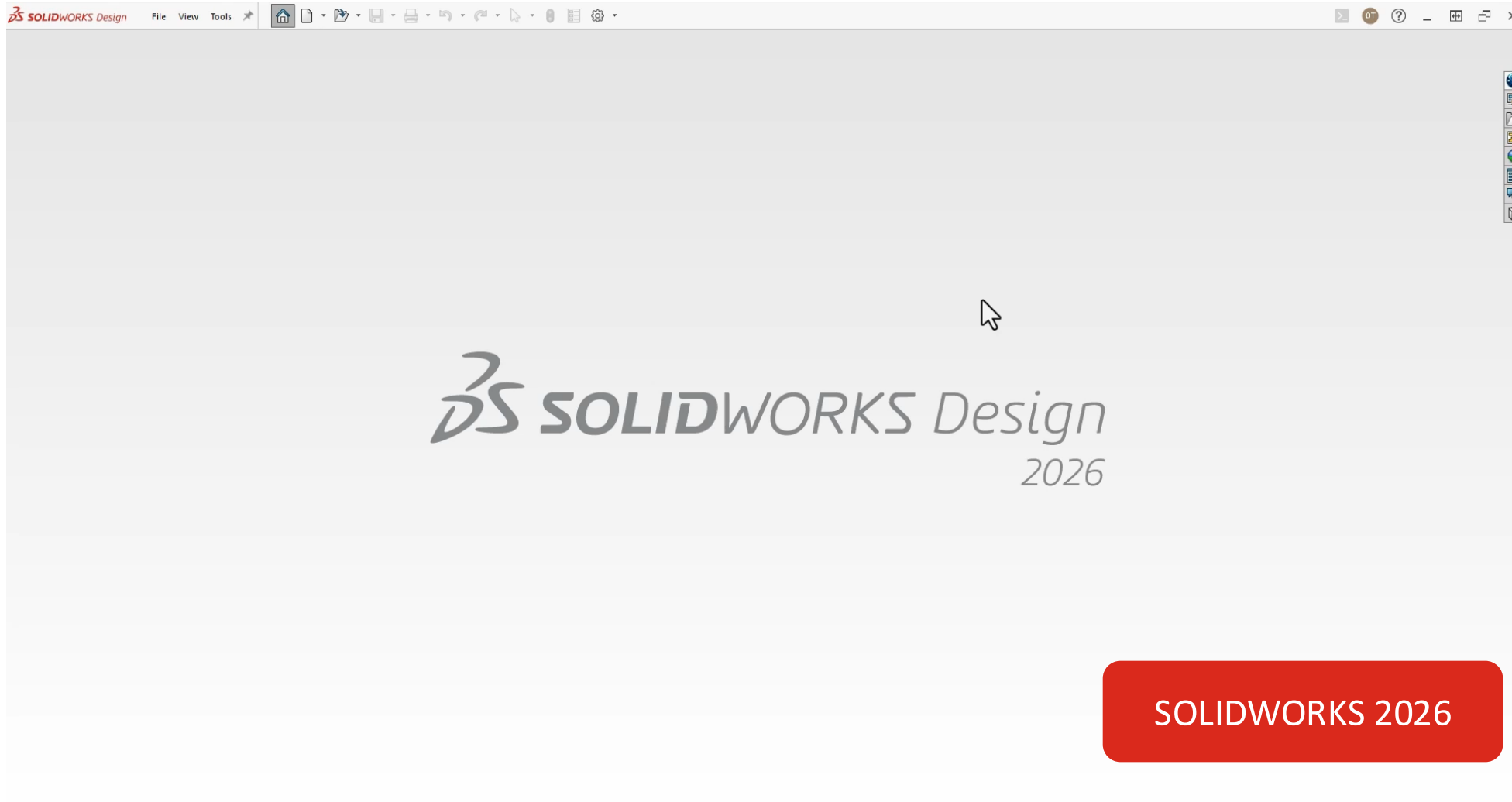


#77
Top Ten 2020

#136
Top Ten 2024

DISMISSED MESSAGES

Indicate when there are Dismissed Messages



#77
Top Ten 2020

#136
Top Ten 2024


SOLIDWORKS 2026

SELECTION FILTER – CURRENT BEHAVIOR

💡 Add Selection Filters (F5) for Features and Part Components

Gathering Support

#13
Top Ten 2024

 Alin Vargatu 2 year(s) ago
The Top Ten List - 3DEXPERIENCE World 2024

91

Current status: Gathering Support

Have you noticed the two most important types of entities missing in the options above? There is no filter for Features or for Components.

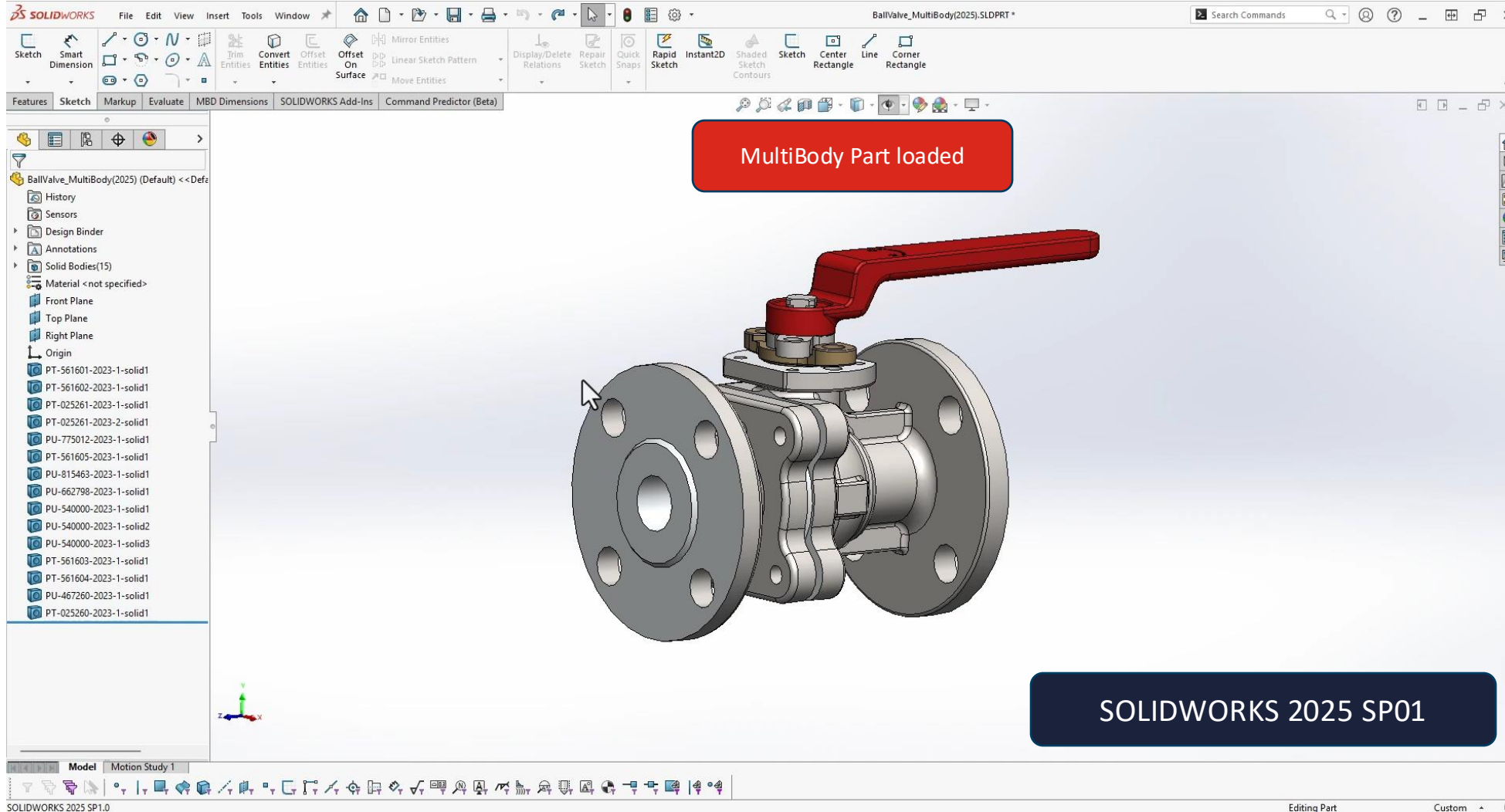
Why is this a huge problem for users? Here are two examples:

- A complex multibody part resulting from importing a STEP file. You want to select in the graphics area for the imported features to keep, then invert the selection and delete the rest.
- You cannot do that without using crazy workarounds. When you select in the graphics area, you think you selected features (you see them selected in the Feature Manager Tree) but you actually selected faces. If you invert the selection, you see this:



You thought you selected features? Think again!

SELECTION FILTER – SW2025 BEHAVIOR

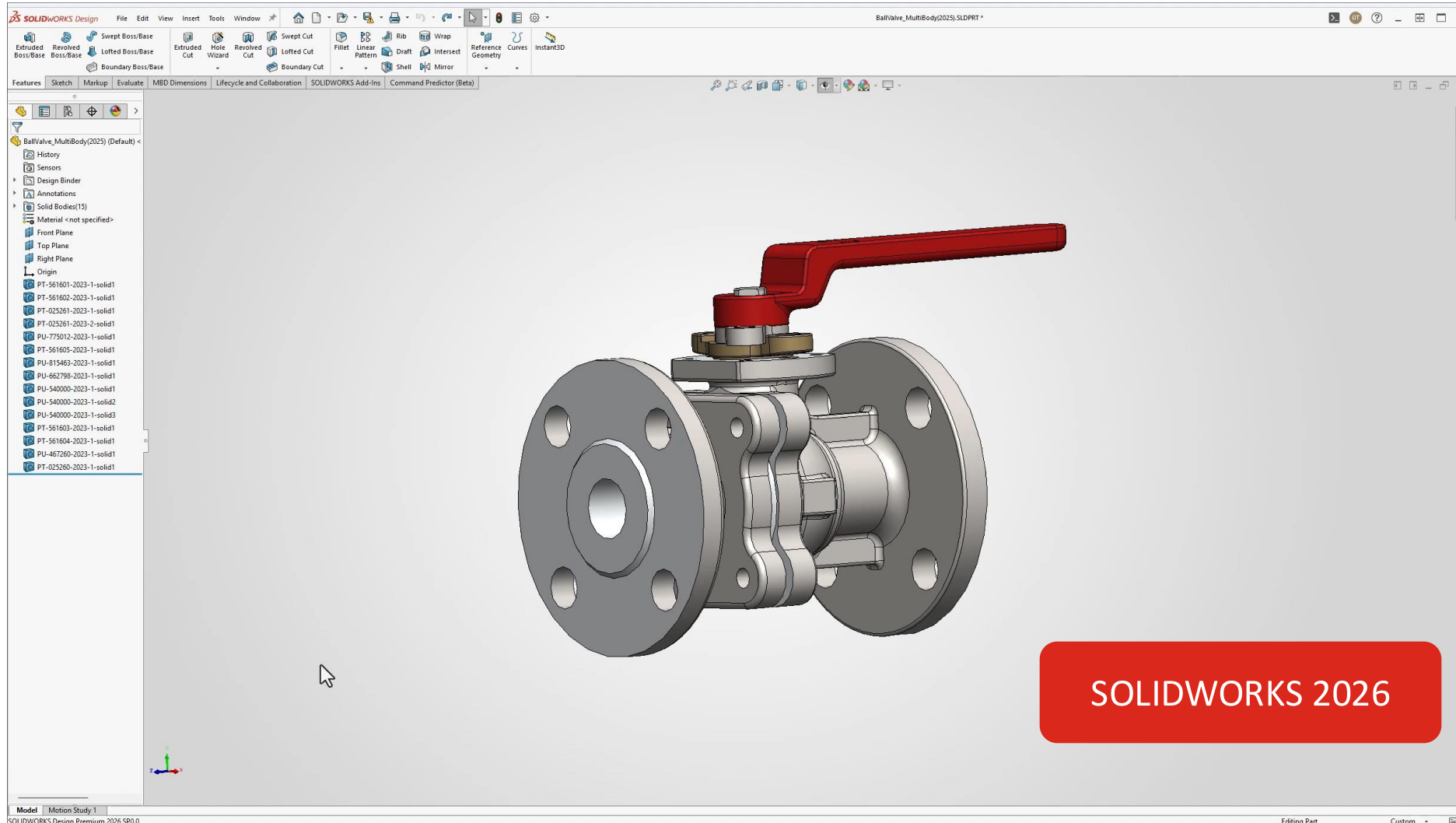


#13
Top Ten 2024

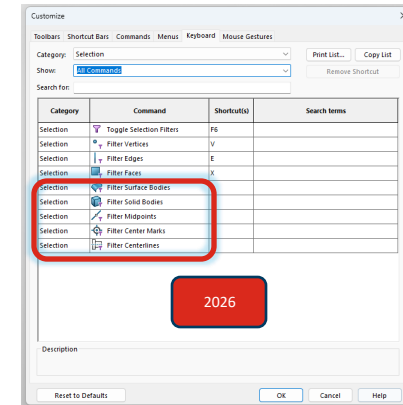
SOLIDWORKS 2025 SP01

SELECTION FILTER ENHANCEMENTS

#13
Top Ten 2024

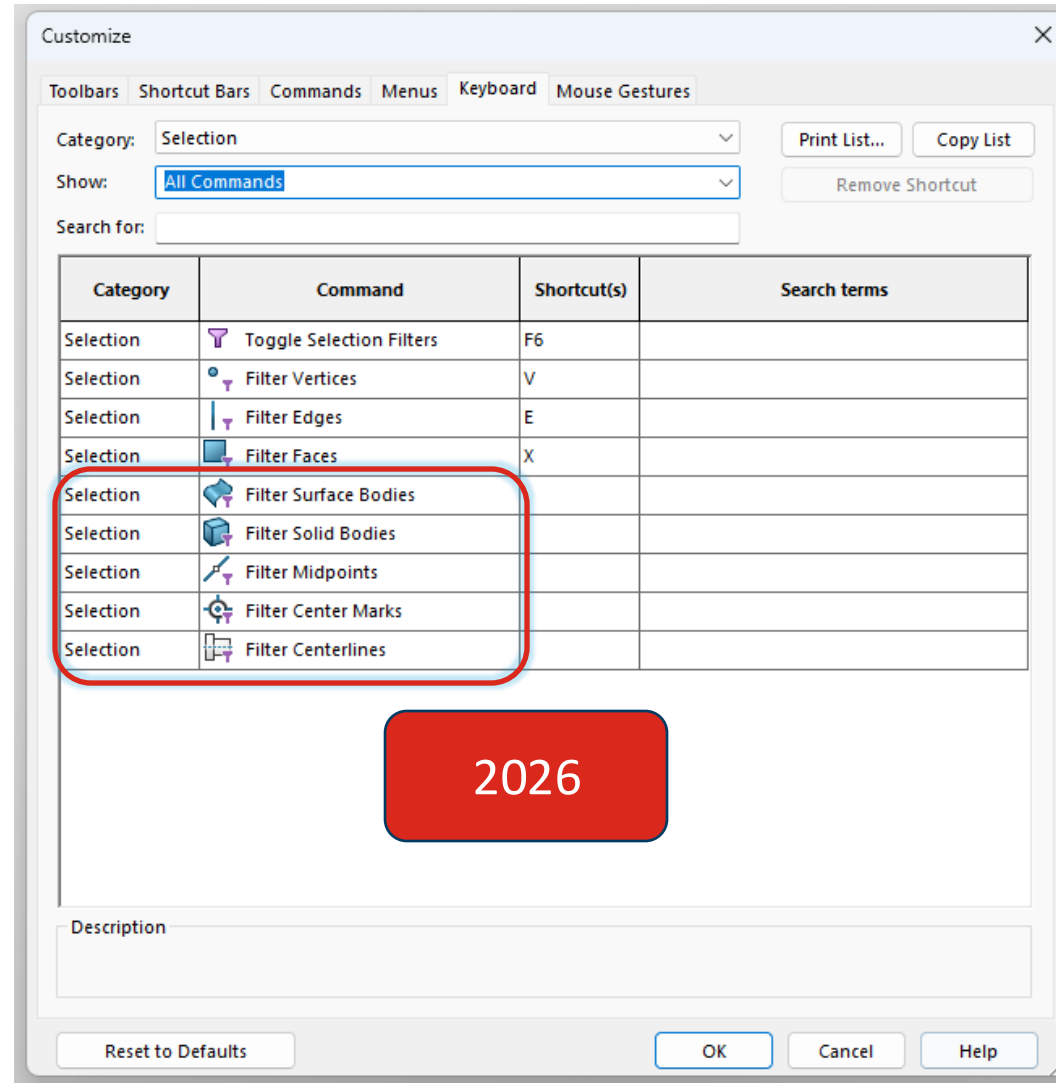


SOLIDWORKS 2026



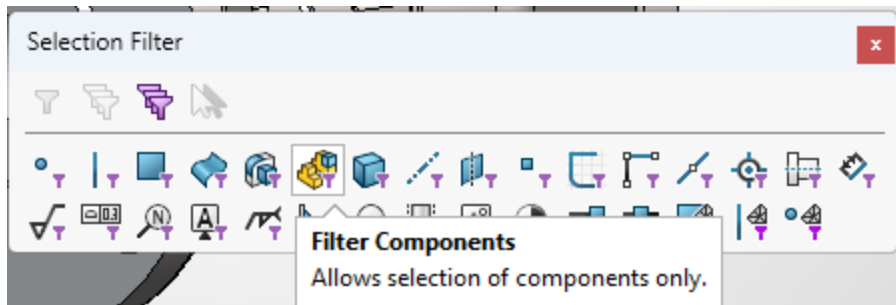
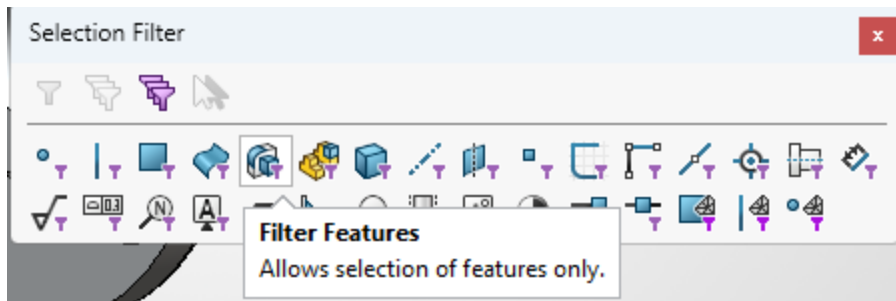
SELECTION FILTER ENHANCEMENTS

#13
Top Ten 2024



SELECTION FILTER ENHANCEMENTS

#13
Top Ten 2024



Customize

Toolbars | Shortcut Bars | Commands | Menus | Keyboard | Mouse Gestures

Category: Selection

Show: All Commands

Search for:

Category	Command	Shortcut(s)	Search terms
Selection	Toggle Selection Filters	F6	
Selection	Filter Vertices	V	
Selection	Filter Edges	E	
Selection	Filter Faces	X	
Selection	Filter Surface Bodies		
Selection	Filter Solid Bodies		
Selection	Filter Midpoints		
Selection	Filter Center Marks		
Selection	Filter Centerlines		

2026

Reset to Defaults | OK | Cancel | Help

MOVE THE MODIFY BOX



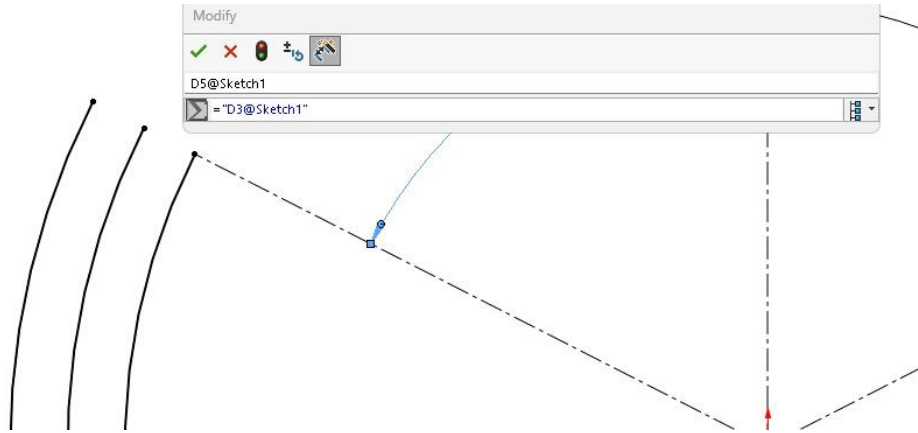
William Alterf 2 year(s) ago

26

[The Top Ten List - 3DEXPERIENCE World 2024](#)

Current status: **Gathering Support** ▾

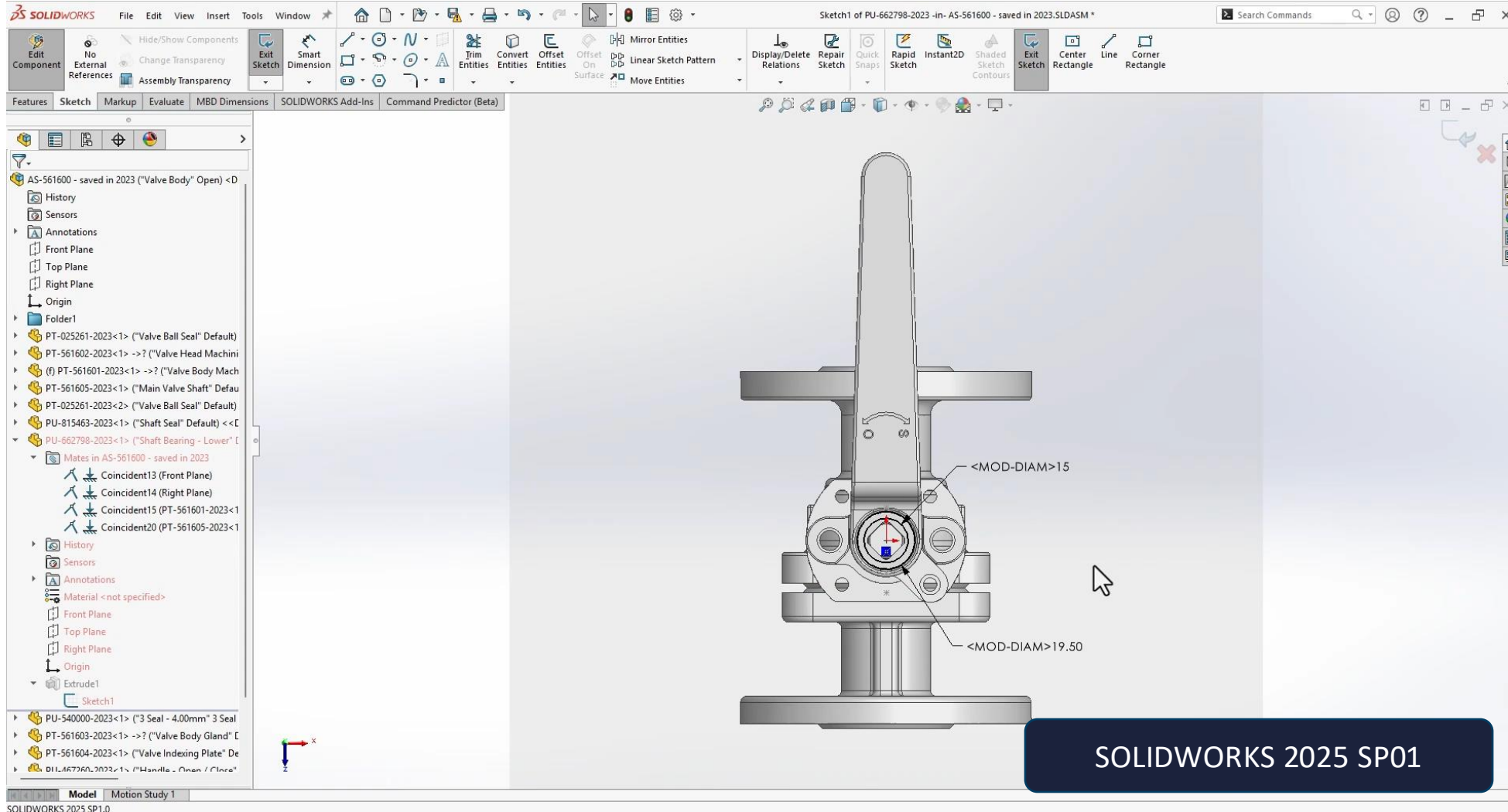
When creating or editing sketch dimensions, the Modify box always appears directly on top of the dimension being edited. It would be far less obtrusive to have the box appear slightly higher or lower than the dimension. This would save the user from having to move the box all the time to allow selecting the dimension when creating formulas.



#82
Top Ten 2024

MOVE THE MODIFY BOX

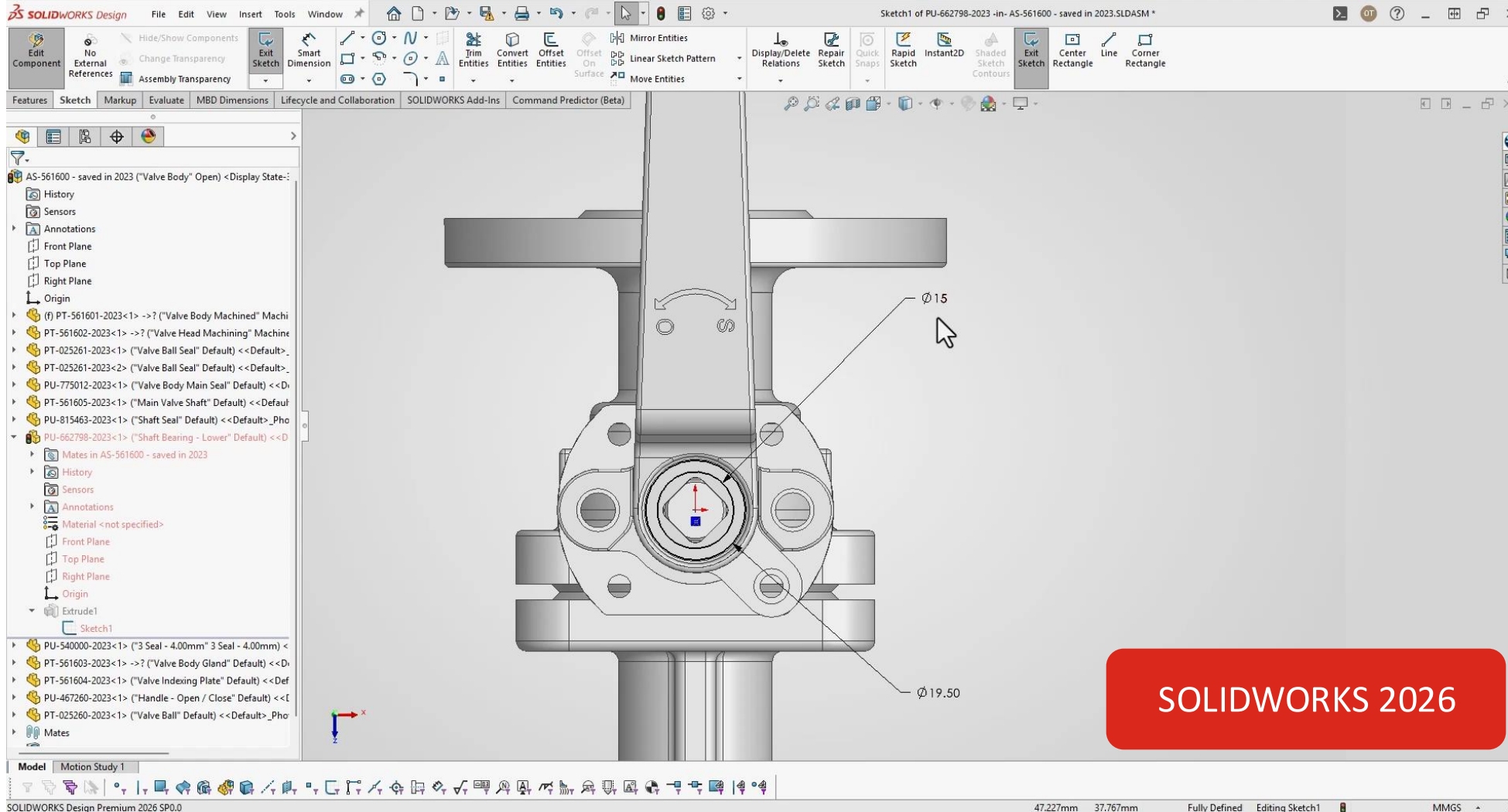
Modify Dimension dialog box avoids overlapping the dimension being edited.



#82
Top Ten 2024

MOVE THE MODIFY BOX

Modify Dimension dialog box avoids overlapping the dimension being edited.



#82
Top Ten 2024

DRAG PARTS BETWEEN ORIGIN AND FOLDER IN FMT



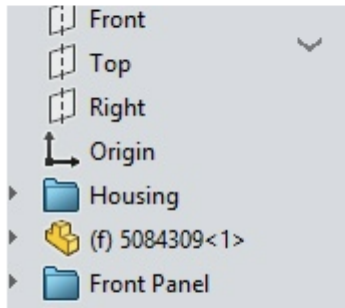
Sarah GISLER 3 year(s) ago

[3DEXPERIENCE World Top Ideas](#)

48

#68
Top Ten 2020

In an assembly Feature Manager Tree, if a Folder is the first object below the Origin, it is not possible to drag a Part / Sub-Assembly to a position in the tree where it is between the Origin and the Folder.



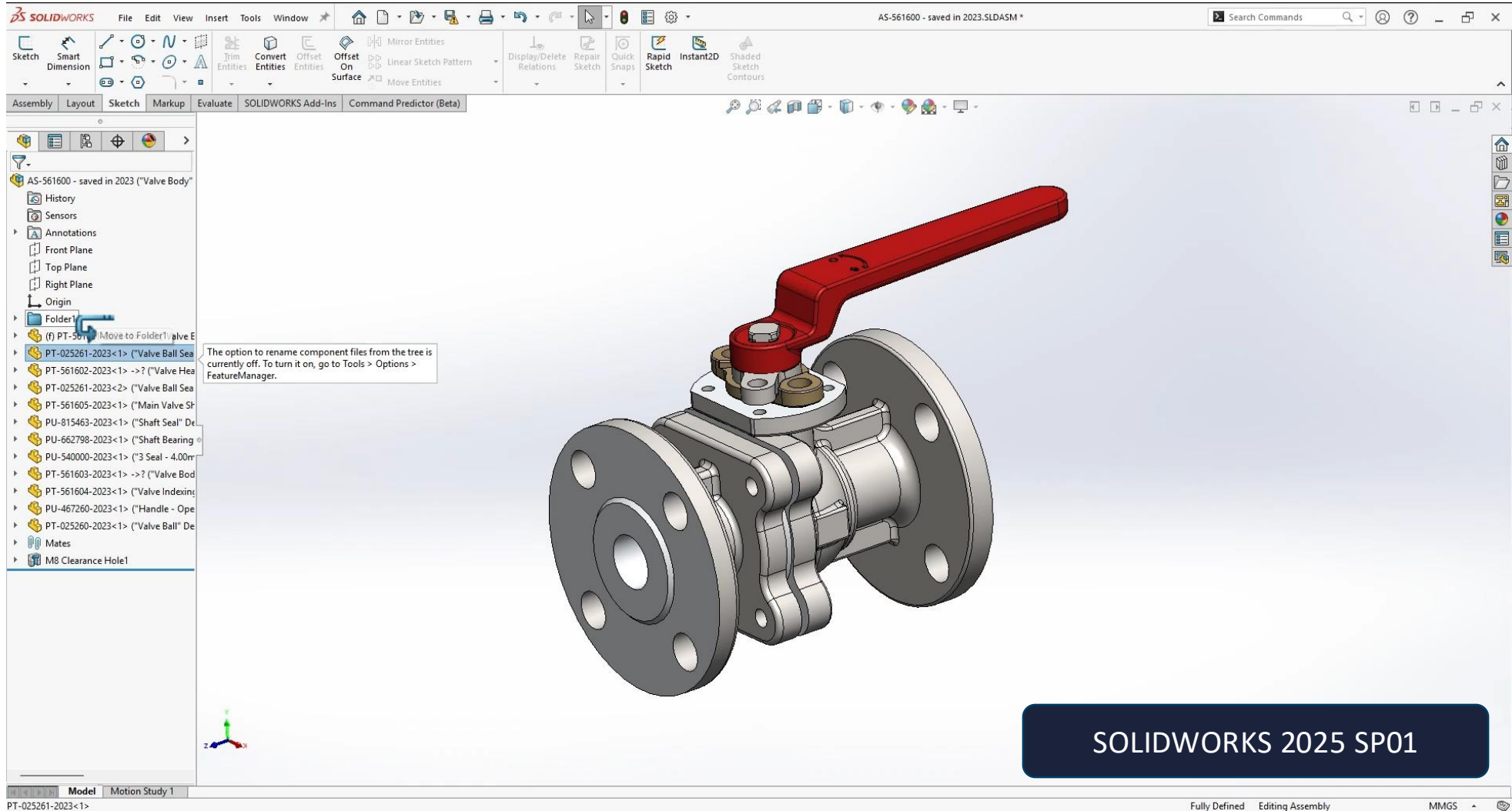
Either

1. The Part must be put in to a new Folder, which can then be dragged between the existing folder and origin, then delete the folder.
2. The Part must be dragged below the Folder, then the Folder can be dragged below the Part.
3. Delete the existing folder to expose the Parts inside it, then drag the required part to the top of the tree, then recreate the folder.

This seems to be an arbitrary limitation on rearranging the tree that could be removed.

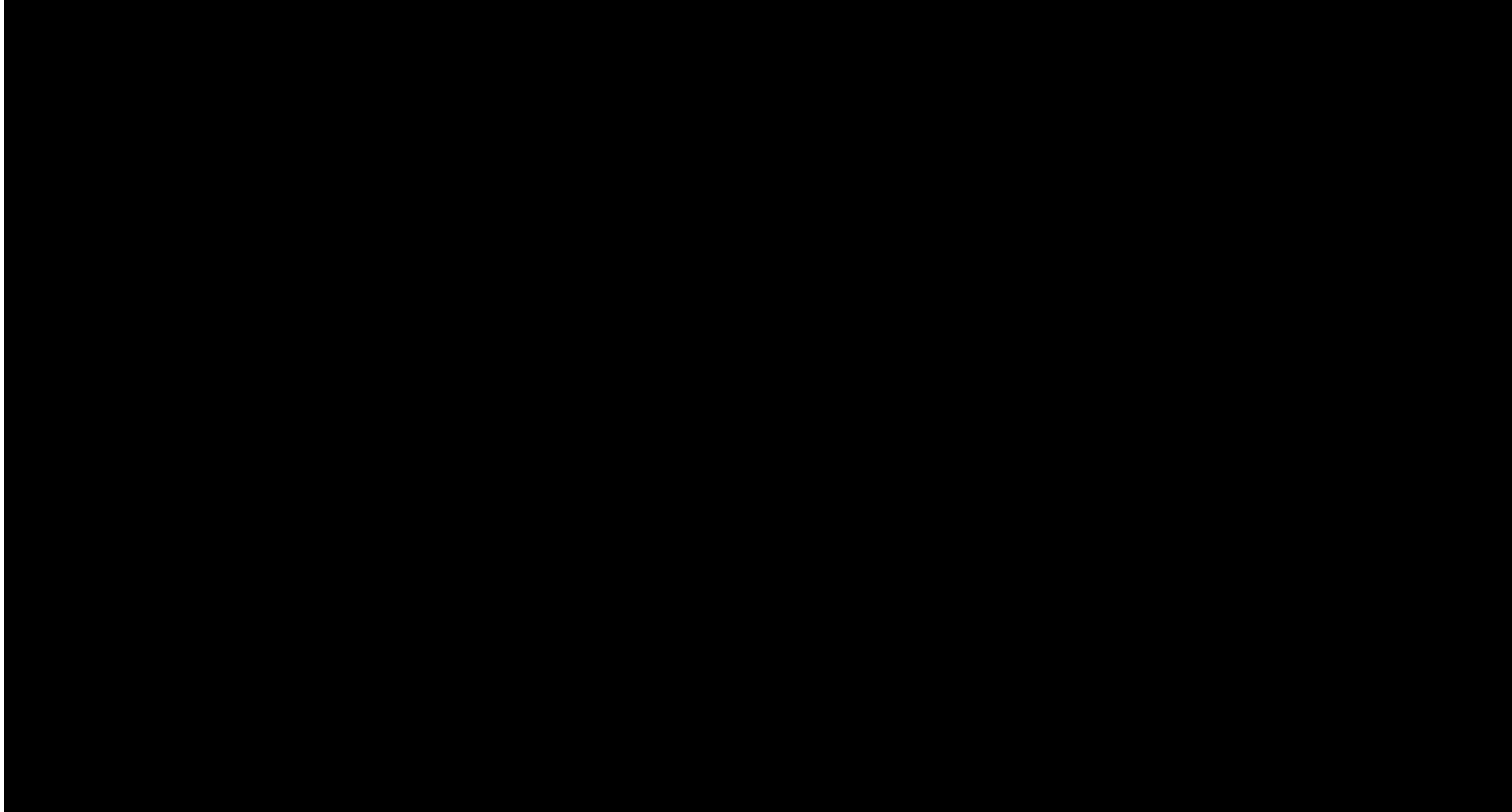
DRAG PARTS BETWEEN ORIGIN AND FOLDER IN FMT

#68
Top Ten 2020



SOLIDWORKS 2025 SP01

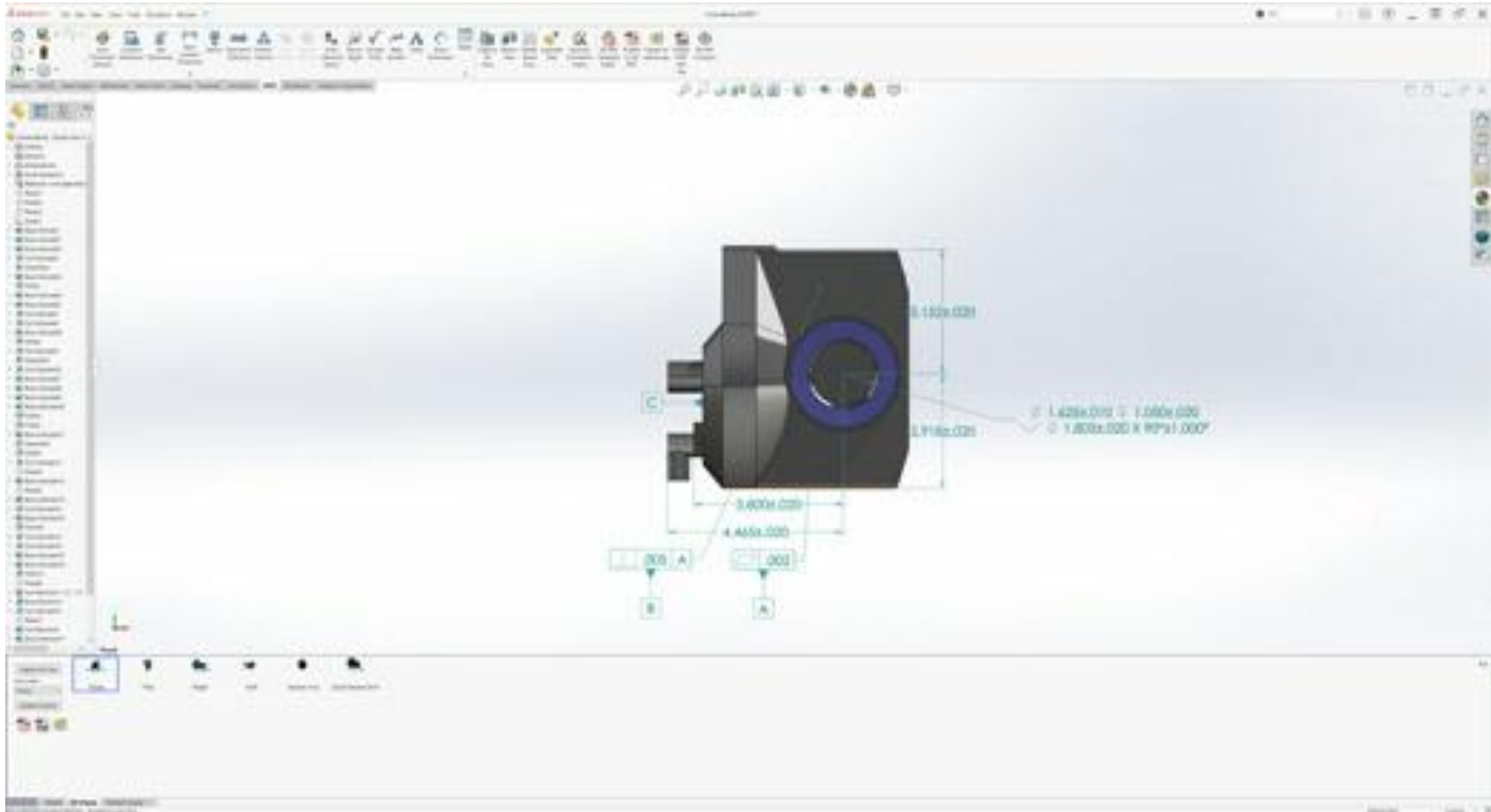
DRAG PARTS BETWEEN ORIGIN AND FOLDER IN FMT



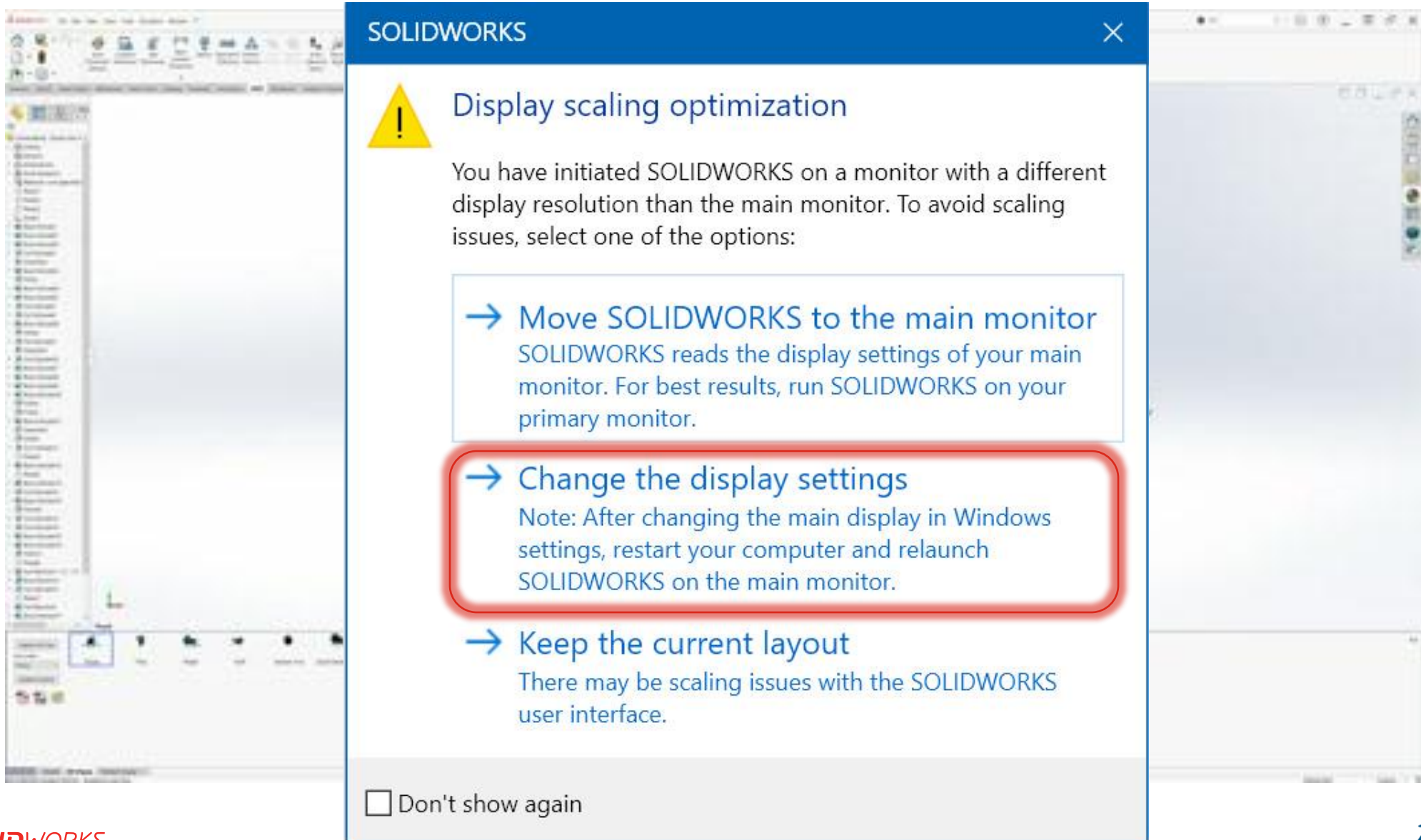
#68
Top Ten 2020

SOLIDWORKS 2026

MULTIPLE MONITORS (SCALING ISSUES)




MULTIPLE MONITORS (SCALING ISSUES)



The image shows a SOLIDWORKS error dialog box titled "SOLIDWORKS" with a yellow warning icon. The message reads: "Display scaling optimization. You have initiated SOLIDWORKS on a monitor with a different display resolution than the main monitor. To avoid scaling issues, select one of the options:". Three options are listed with blue arrows: "Move SOLIDWORKS to the main monitor", "Change the display settings", and "Keep the current layout". The "Change the display settings" option is highlighted with a red rounded rectangle. A "Don't show again" checkbox is at the bottom left.

SOLIDWORKS

 **Display scaling optimization**

You have initiated SOLIDWORKS on a monitor with a different display resolution than the main monitor. To avoid scaling issues, select one of the options:

- **Move SOLIDWORKS to the main monitor**
SOLIDWORKS reads the display settings of your main monitor. For best results, run SOLIDWORKS on your primary monitor.
- **Change the display settings**
Note: After changing the main display in Windows settings, restart your computer and relaunch SOLIDWORKS on the main monitor.
- **Keep the current layout**
There may be scaling issues with the SOLIDWORKS user interface.

Don't show again

MULTIPLE MONITORS (SCALING ISSUES)

SOLIDWORKS

! Display scaling optimization

You have initiated SOLIDWORKS on a monitor with a different display resolution than the main monitor. To avoid scaling issues, select one of the options:

- Move SOLIDWORKS to the main monitor
SOLIDWORKS reads the display settings of your main monitor. For best results, run SOLIDWORKS on your primary monitor.
- Change the display settings
Note: After changing the main display in Windows settings, restart your computer and relaunch SOLIDWORKS on the main monitor.
- Keep the current layout
There may be scaling issues with the SOLIDWORKS user interface.

Don't show again



System > Display

Select a display to change the settings for it. Drag displays to rearrange them.

Identify Extend these displays

Multiple displays
Choose the presentation mode for your displays

- Make this my main display
- Remember window locations based on monitor connection
- Minimize windows when a monitor is disconnected
- Ease cursor movement between displays

TILE WINDOWS ACROSS MULTIPLE DISPLAYS

#17
Top Ten 2020

#32
Top Ten 2022



Dominic STAVIS 3 year(s) ago

3DEXPERIENCE World Top Ideas

Enable Tile Vertically to work if SW is maximised across multiple displays (currently they only tile in the first display).





SKETCH

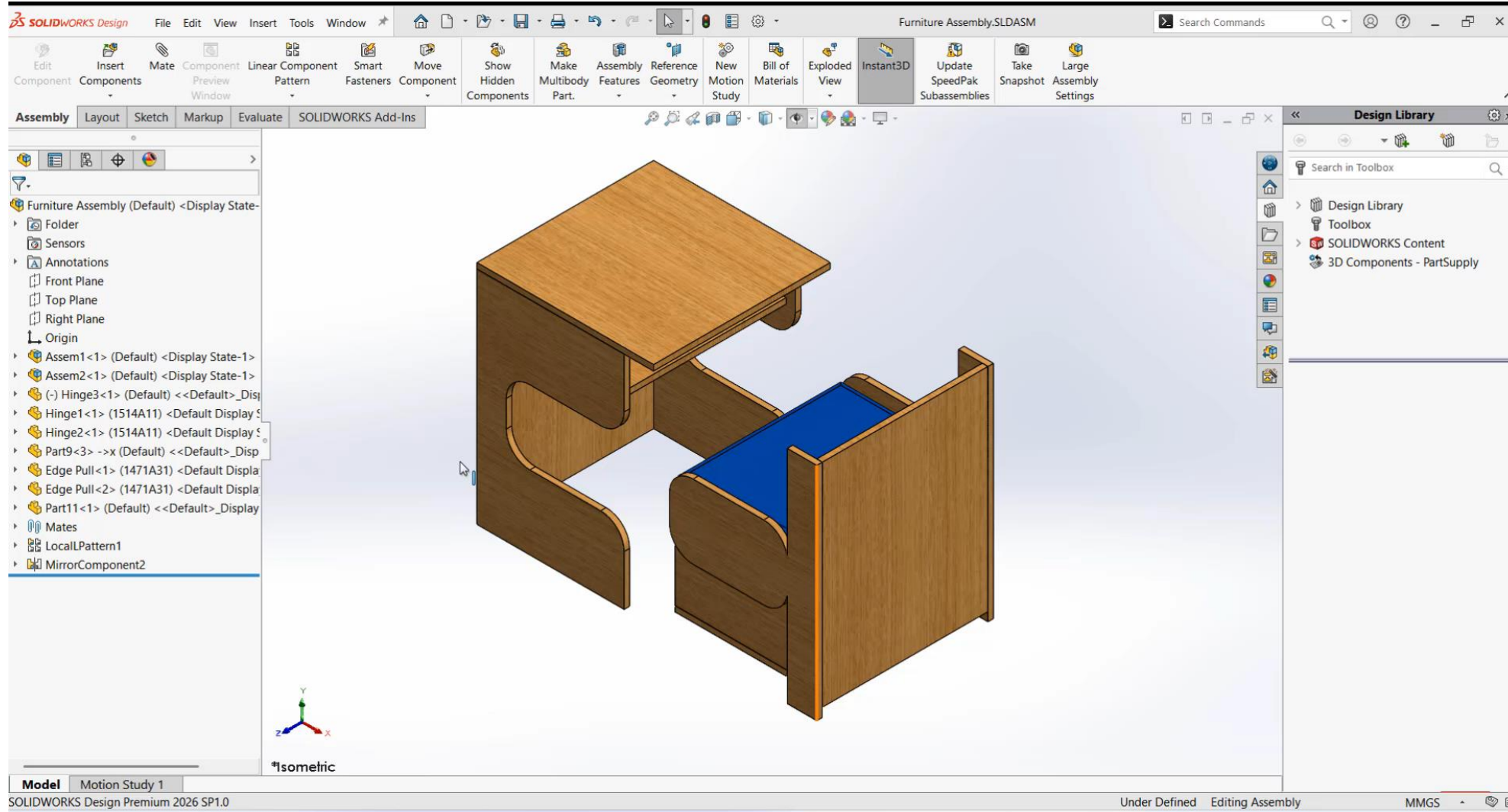
Customer-Driven Enhancements In SOLIDWORKS 2026

INVERT FILLET/ARCS IN SKETCHES (2025 SP1)

Quickly flip the tangency of the selected arc with respect to the lines attached.

#197
Top Ten 2023

#63
Top Ten 2024

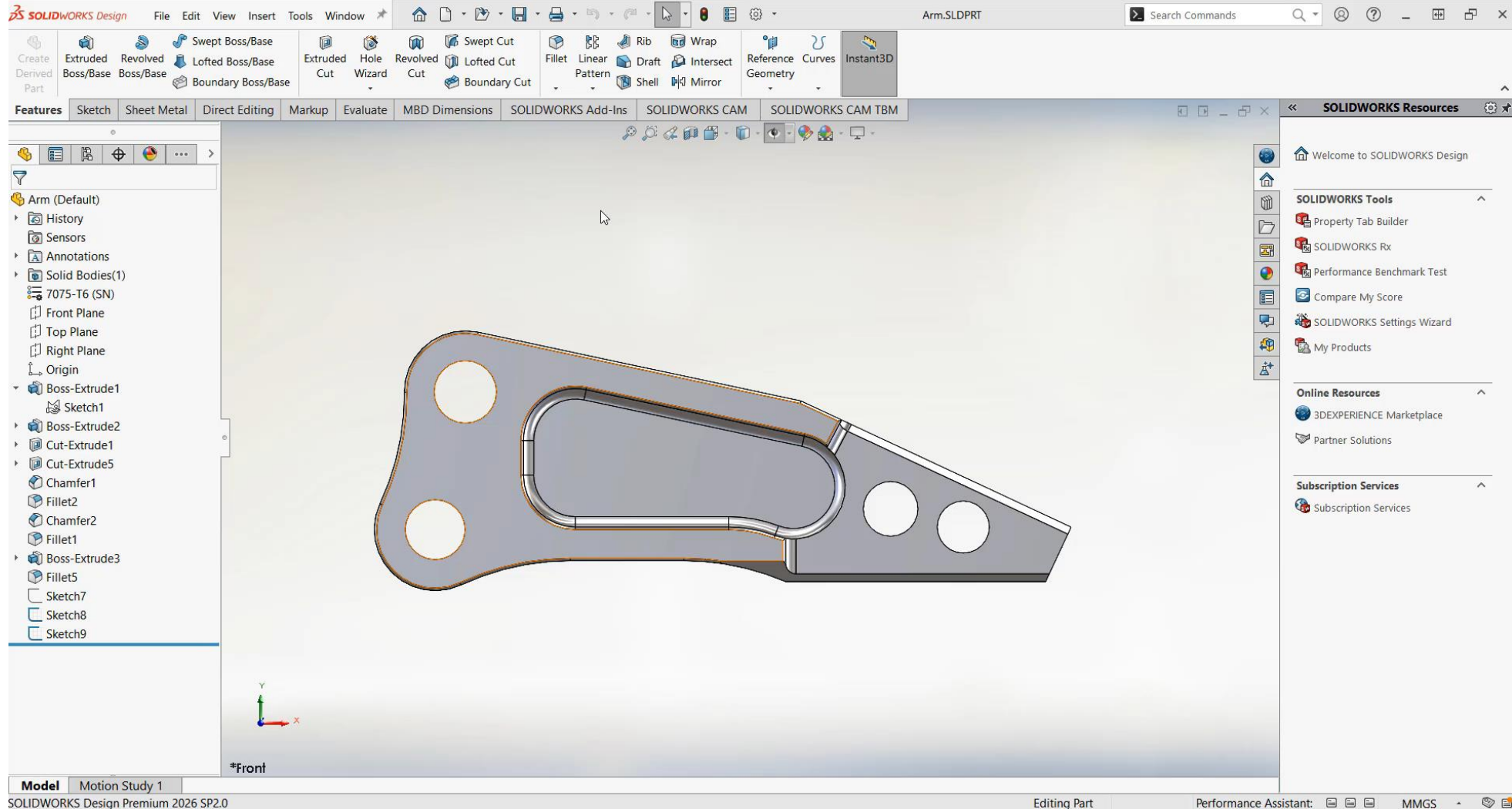


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QUICKLY CREATE AN EQUAL RELATION RECTANGLE I.E. SQUARE (2025 SP2.0)

Create a square on the fly while sketching a rectangle using a key-board shortcut.

#22
Top Ten 2020





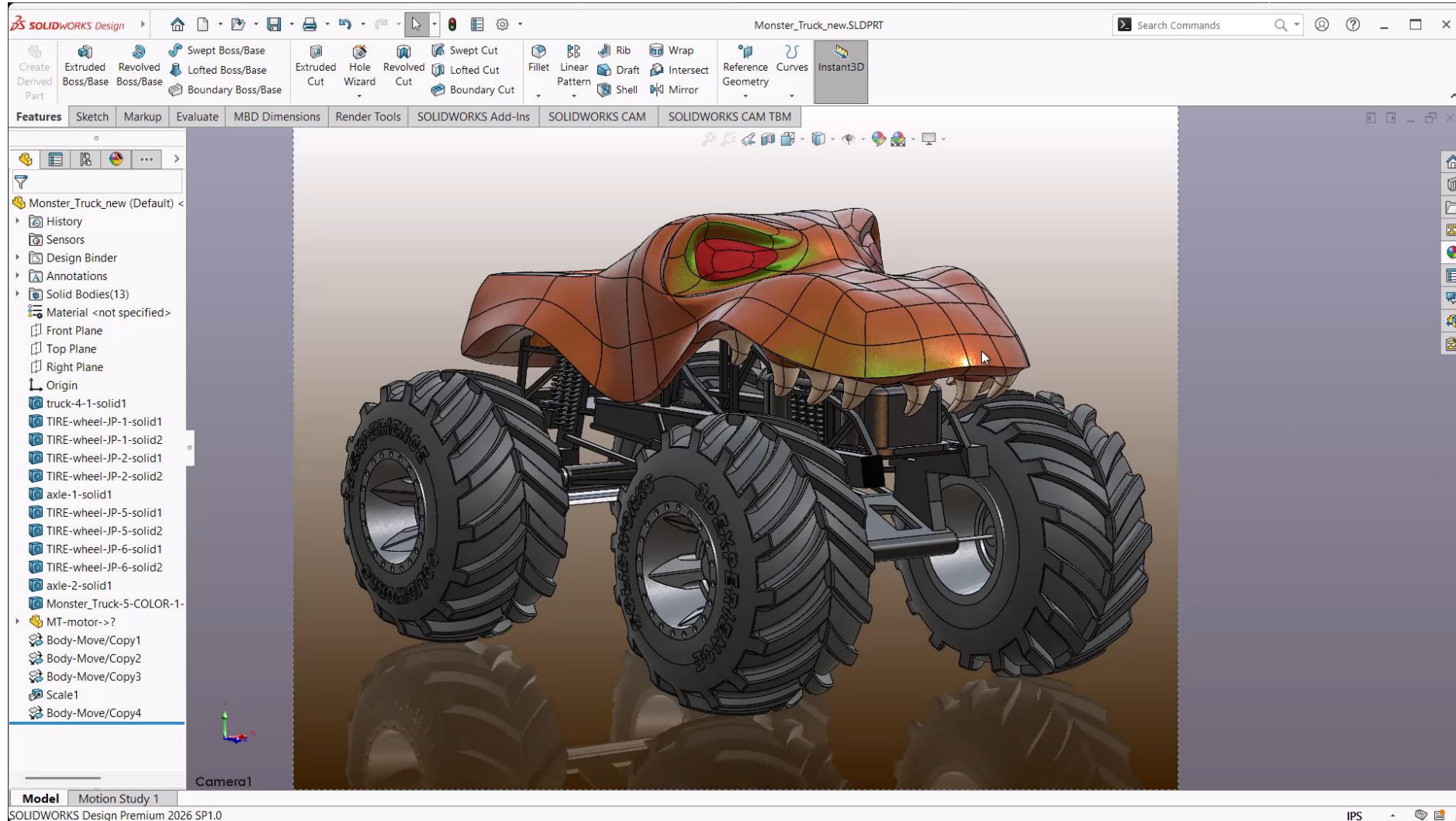
VISUALIZE

Customer-Driven Enhancements In SOLIDWORKS 2026

ADD BASIC QUICK RENDER IN SOLIDWORKS

Produce photorealistic renders from SOLIDWORKS models with Visualize

#38
Top Ten 2022



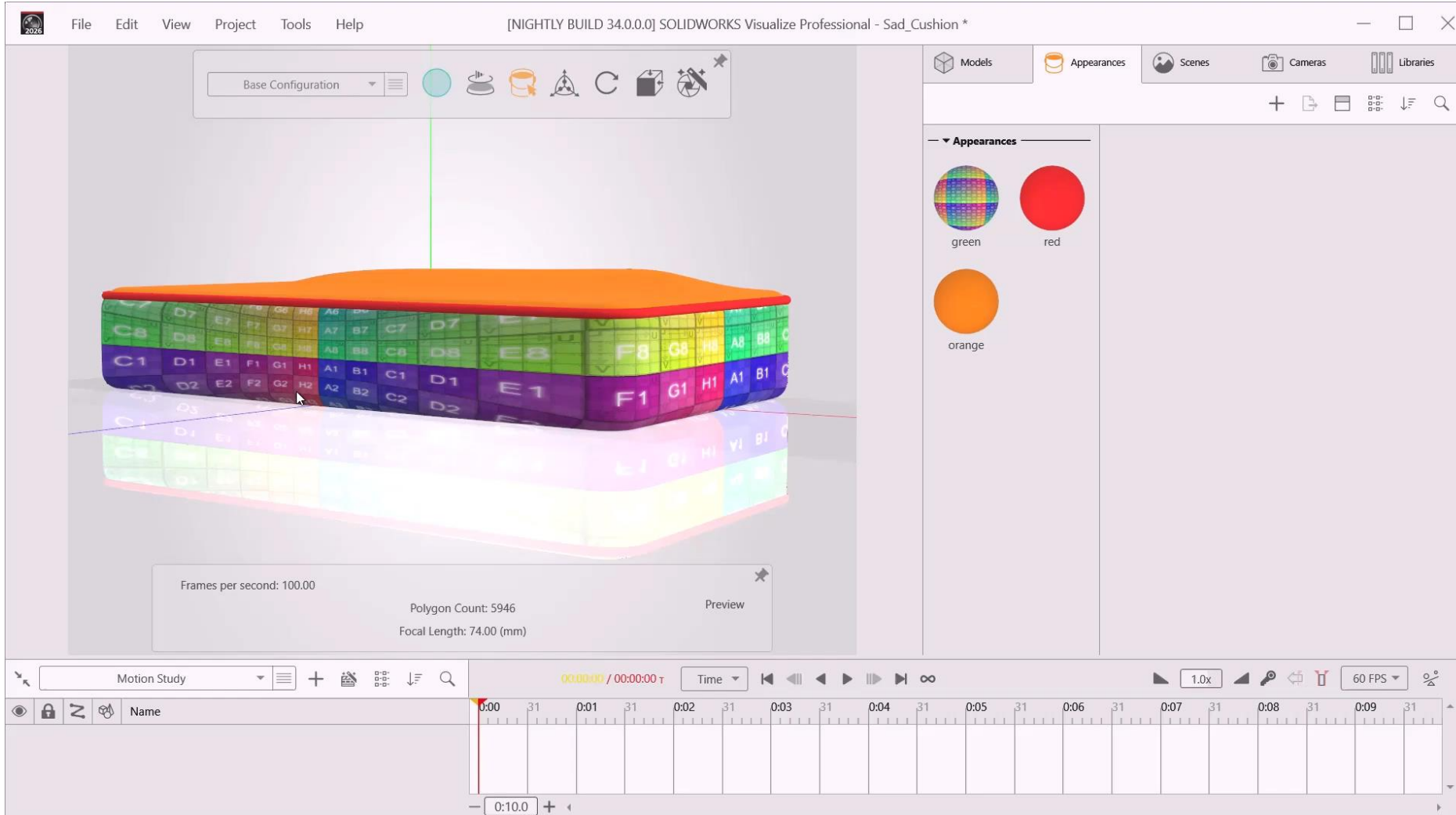
SOLIDWORKS Design Premium 2026 SP1.0

IPS

UV MAPPING FOR SOLIDWORKS VISUALIZE

Precision UV Editor is here – take full control of textures with the new UV Editor.

#9
Top Ten 2021





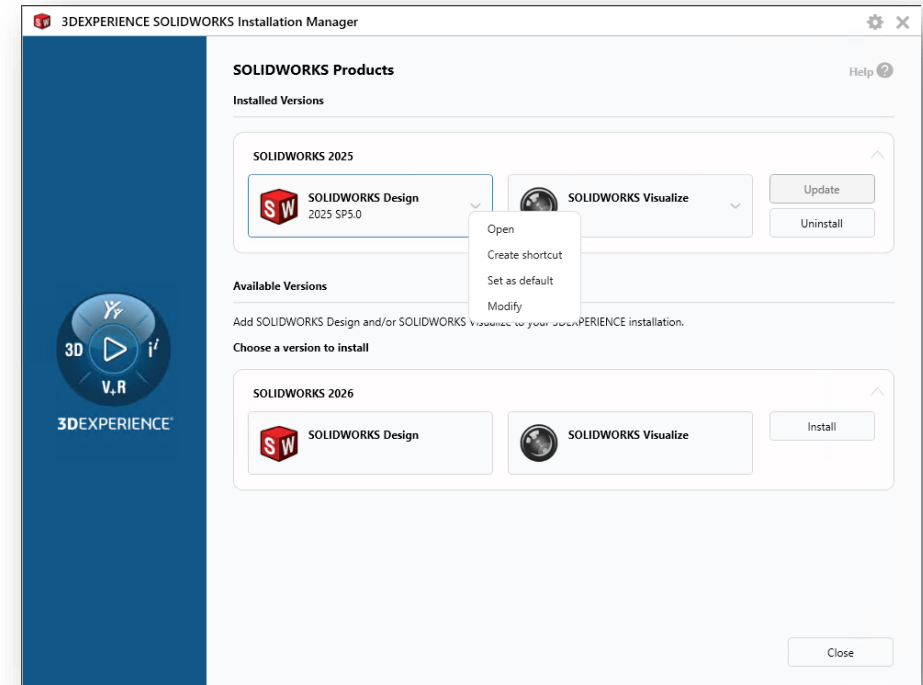
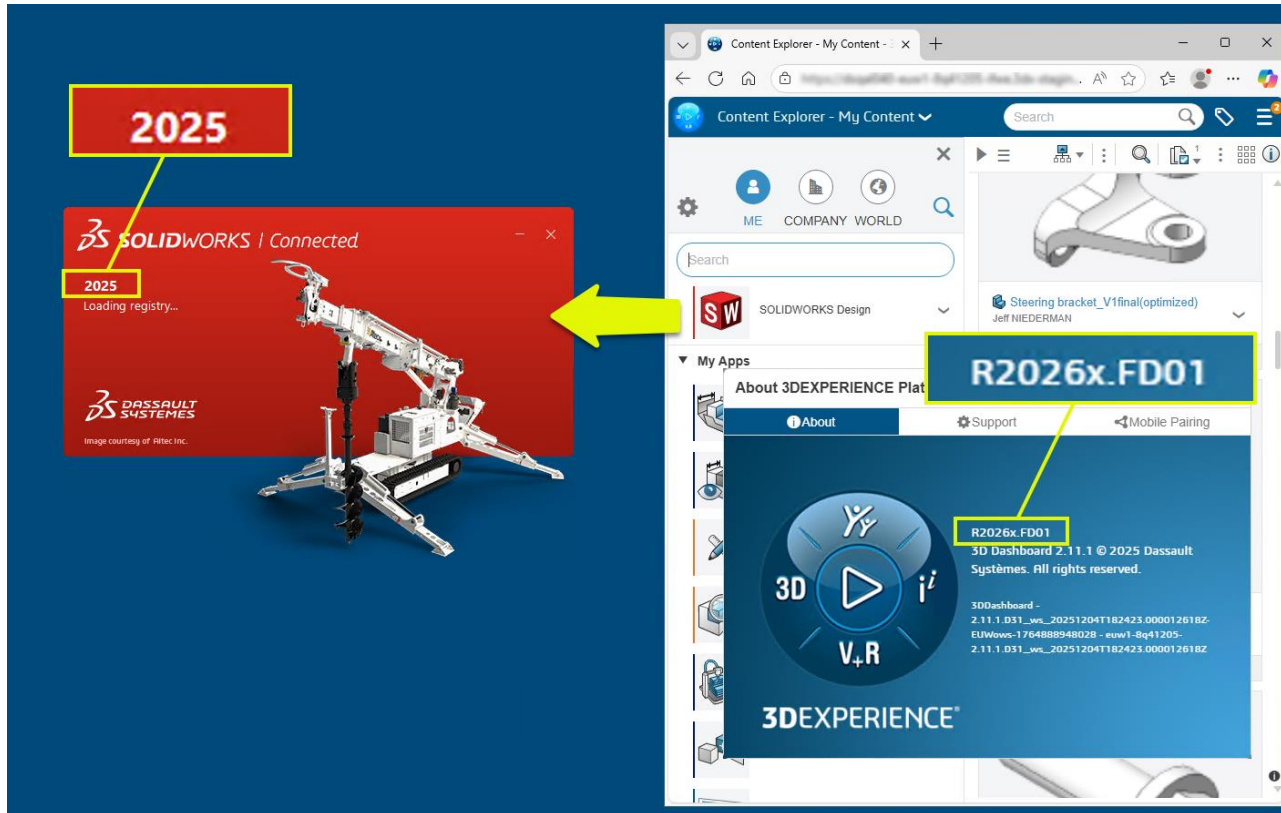
INSTALLATION

Customer-Driven Enhancements In SOLIDWORKS 2026

SELECTABLE SOLIDWORKS DESIGN VERSION

Allow users to select which SOLIDWORKS Design version to run from the platform

#276
Top Ten 2023



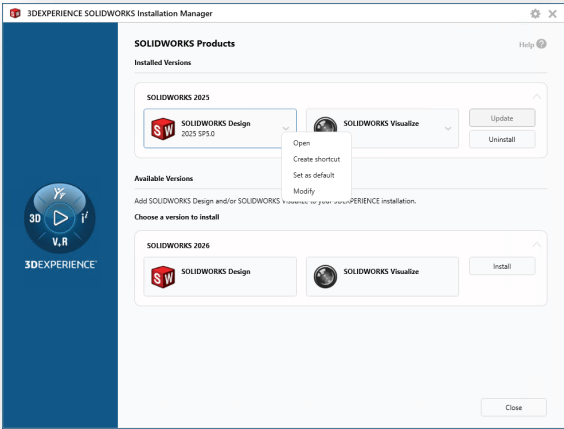
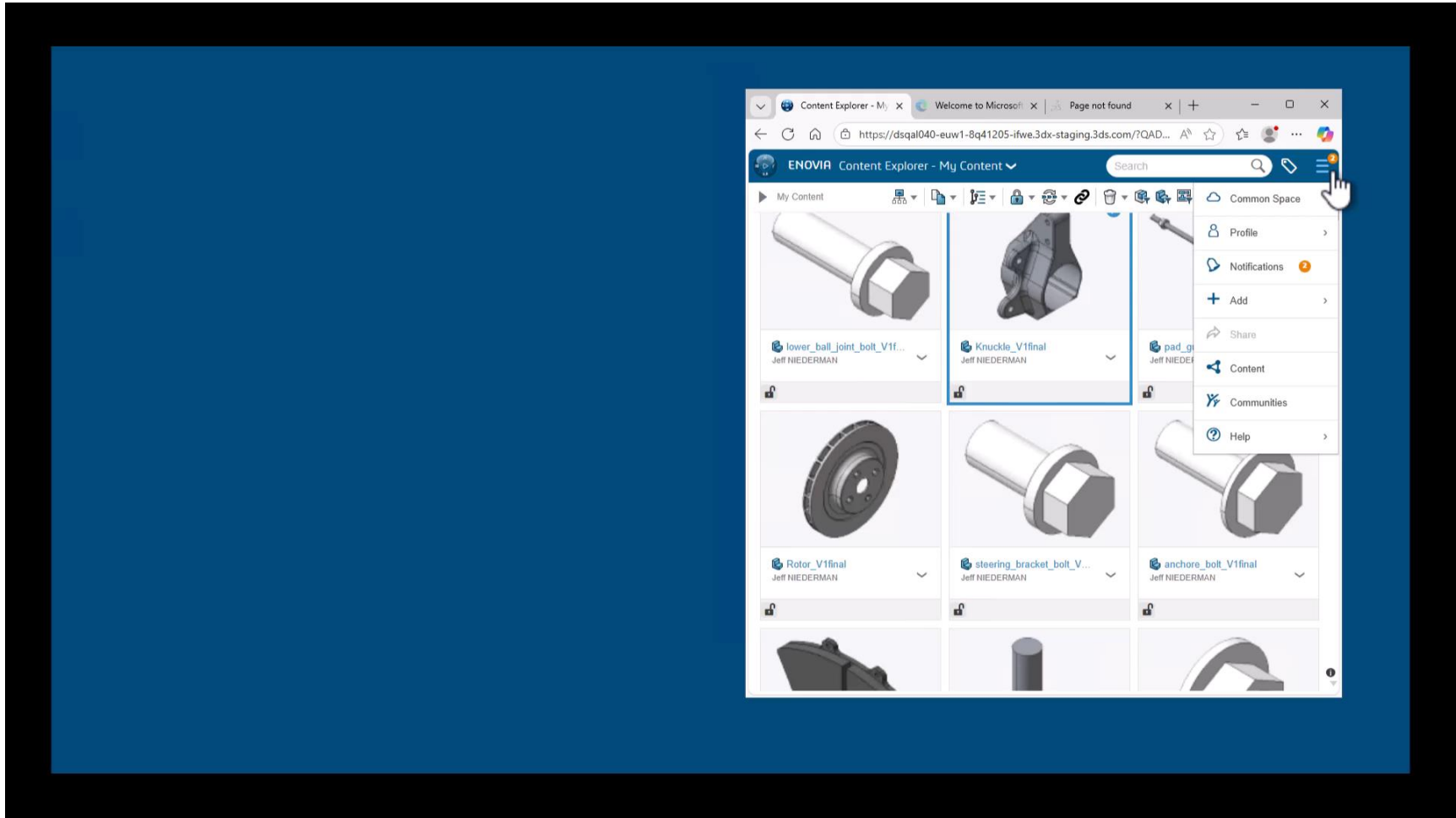
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SELECTABLE SOLIDWORKS DESIGN VERSION

Allow users to select which SOLIDWORKS Design version to run from the platform

#276
Top Ten 2023

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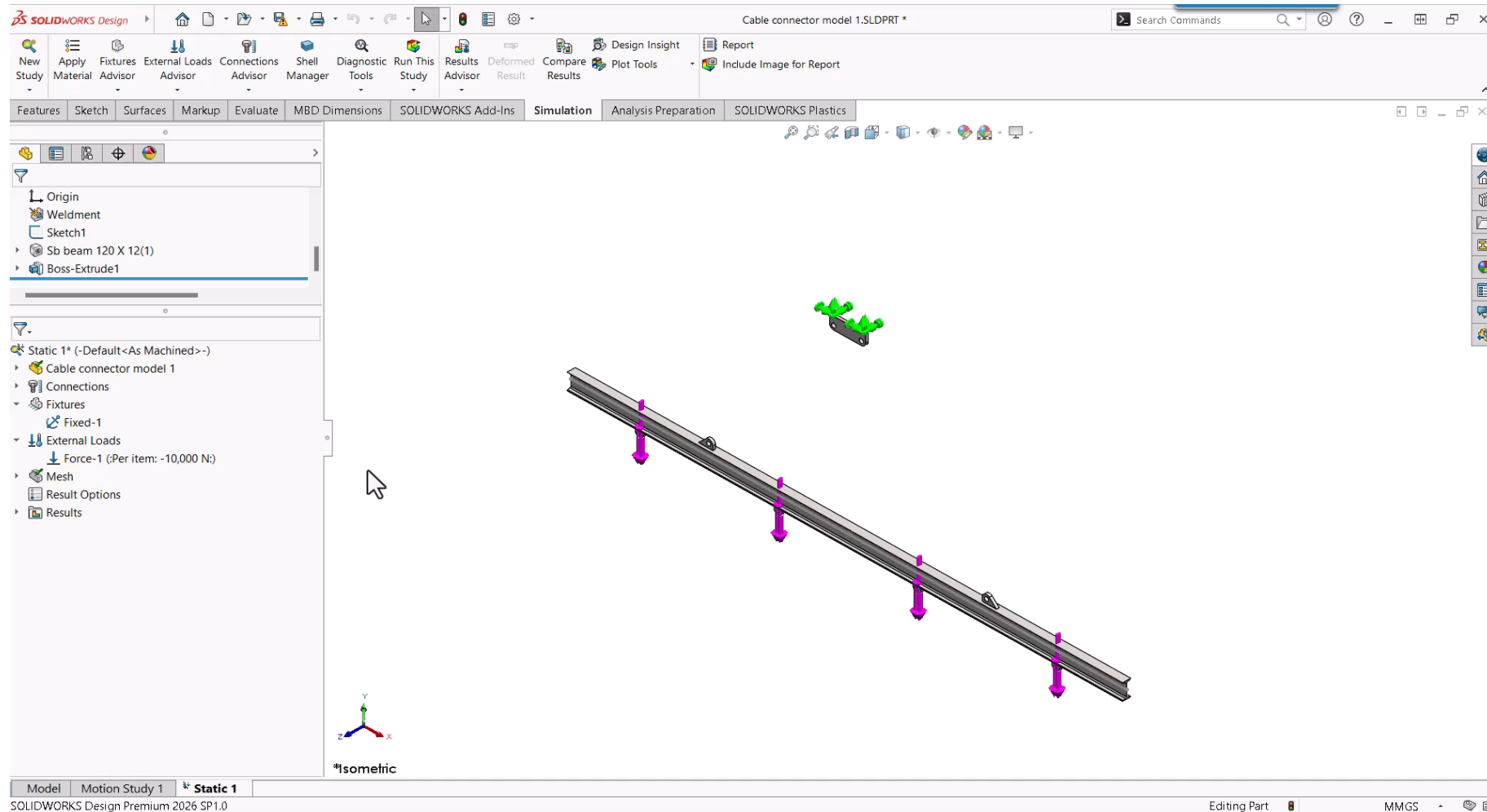


SIMULATION

Customer-Driven Enhancements In SOLIDWORKS 2026

CABLE CONNECTOR

You can use a cable connector to precisely represent cable elements, which are widely used in cable-stayed bridges, cranes, and other load-bearing structural systems.



278

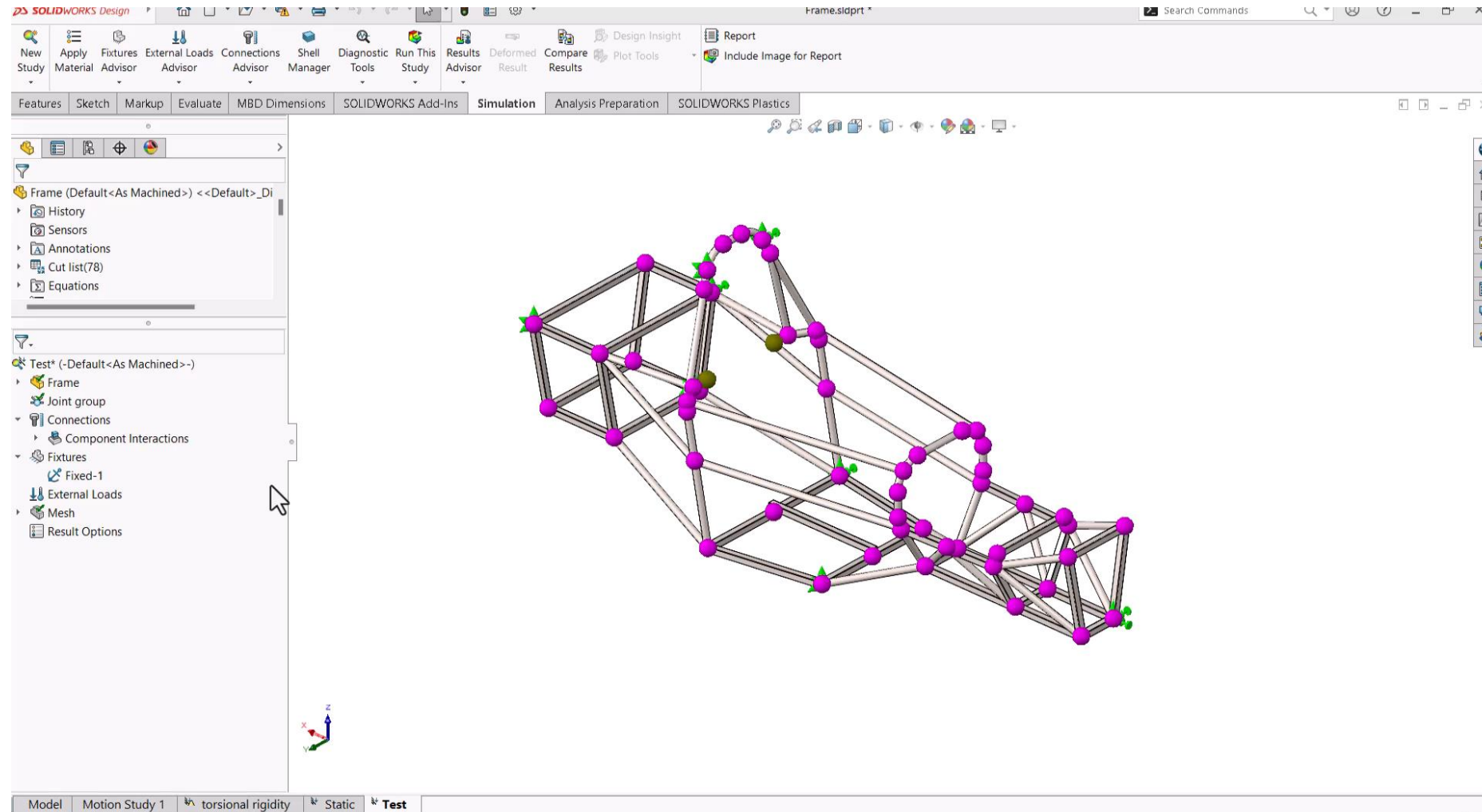
Top Ten 2017

#436

Top Ten 2018

TOTAL FORCE ON BEAMS

Use the Total option in the Force/Torque PropertyManager to apply a force that is automatically distributed across multiple beam bodies in proportion to their lengths.



#93
Top Ten 2017

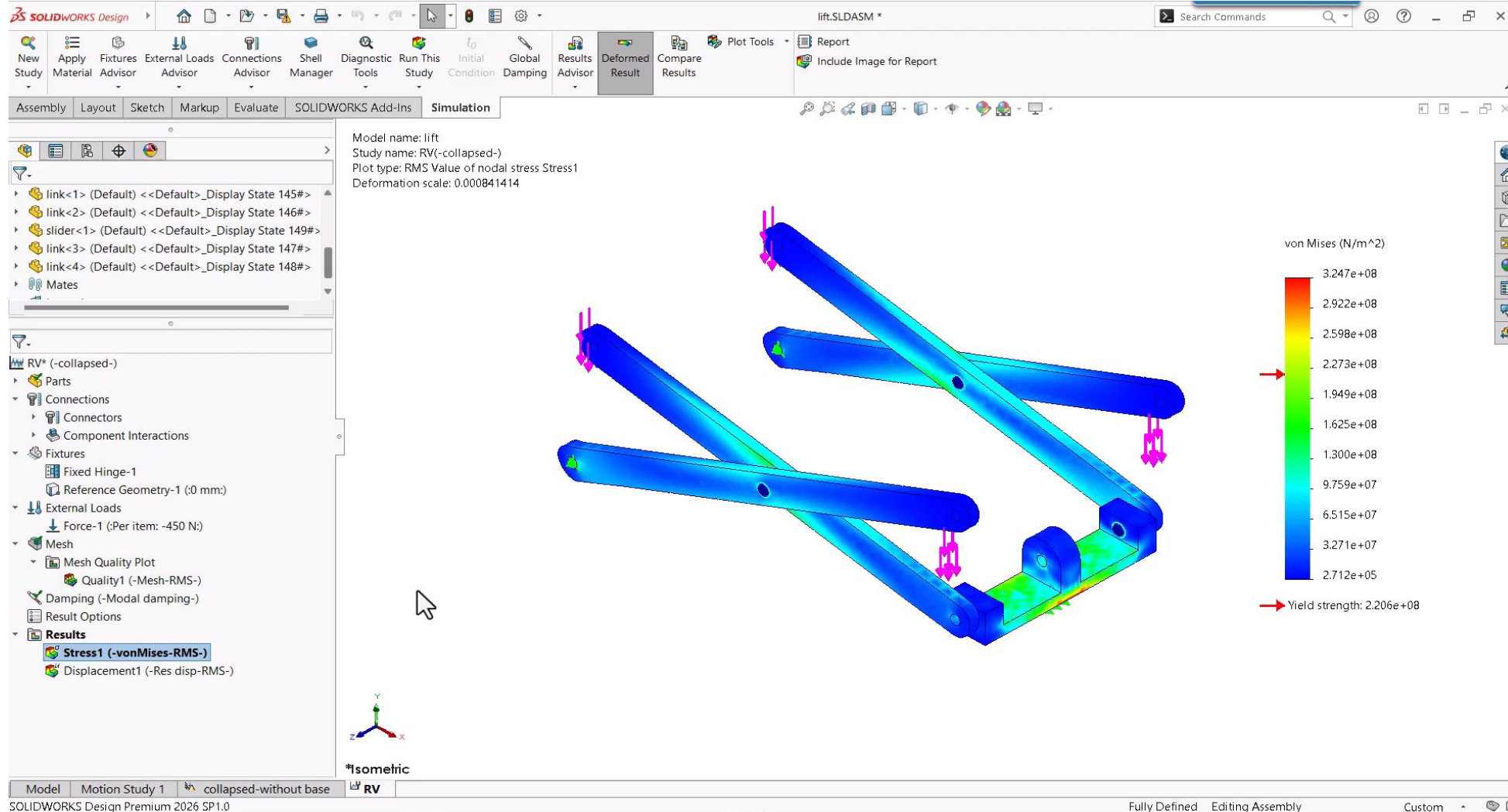
#495
Top Ten 2018

#322
Top Ten 2019

PIN CONNECTOR FORCES IN RANDOM VIBRATION STUDIES

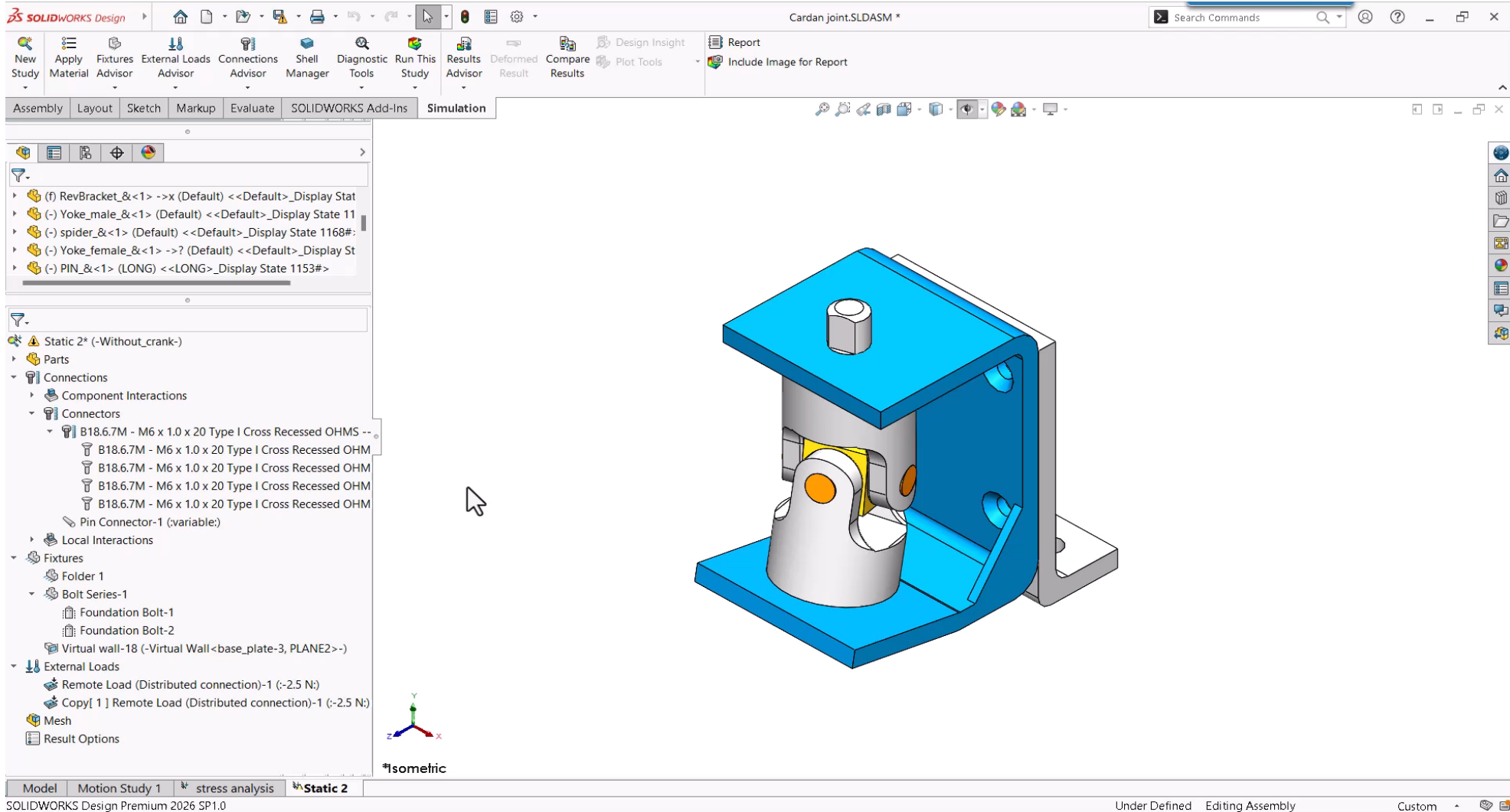
Extract pin connector forces, including shear force, axial force, bending moment, and torque, in linear dynamic random vibration studies

#197
Top Ten 2014



UNDEFINED MATERIAL BEHAVIOR

All components without assigned materials are precisely identified to assist users in locating them and assigning the required material.



#250
Top Ten 2024

#181
Top Ten 2023

TTL IDEAS DELIVERED IN SOLIDWORKS 2026 -PDM



- Streamlined Workflow Administration
- Granular User Access
- Enhanced CAD File Preview Option
- Improved Database Upgrades
- Improved File Version Upgrades (FVUT)

Top Ten 2015

Top Ten 2014

Top Ten 2017

Top Ten 2016

Top Ten 2015

Top Ten 2012



THE TOP TEN LIST ENHANCEMENT IDEAS 2026

(Drumroll, please!)



ASSEMBLIES

Top Ten Enhancement Ideas 2026

#5 ASSEMBLIES

3DEXPERIENCE World 2026

💡 Assembly - Allow assembly-level-features to be associated to a smart component

Gathering Support



Adrien MOINE 5 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

13



Current status: Gathering Support

Since it's better to use a repetition function than multiplying a component and its mates, it would be really nice to be able to associate this type of feature, performed at the assembly level, to a smart component.

For a guide rail, for example, the current workaround consists in associating, when defining the smart component, a drilling function with only a single tapped hole, the pattern of this tapped hole and a single screw, thus directing the designer towards the realization of the screw pattern by a pattern-driven pattern feature.

A loss of productivity, as you will agree.

It will be much more simple and effective to be able to associate the pattern-driven pattern feature to the smart component in order to retrieve it in the working assembly when deploying the smart features.

assembly

#4 ASSEMBLIES

3DEXPERIENCE World 2026

💡 Maintain Dimension and Annotation References for Named Entities

Gathering Support

DM David Mandl 4 month(s) ago - Moderated
The Top Ten List - 3DEXPERIENCE World 2026

43

Current status: Gathering Support

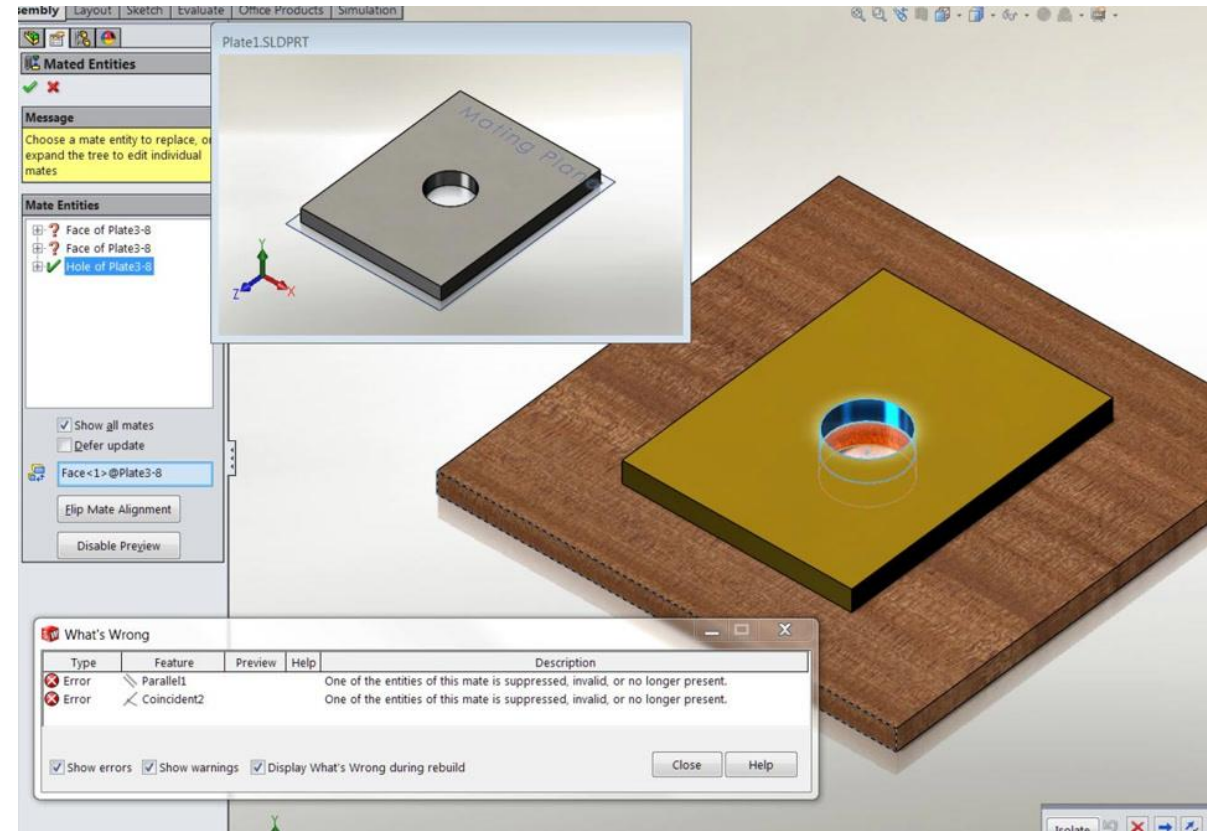
SOLIDWORKS has long had a technique available to give Names to surfaces and edges in your model. When you have a [Named Entity](#) in your model, it is a [very effective technique for preventing dangling references in your assembly mates](#) without having to rely on Datum Planes or Persistent Identifiers.

One of the only downsides with this technique is that it only works to maintain references in sketch relations and mates. This technique does not work to preserve attachment references for dimensions or annotations (BOM balloons, reference dims in drawings, annotation arrow connections, weld symbols, et al). If you replace a component in an assembly, all of those references that are used in your drawings (or model-based annotations) will go dangling.

Allowing annotations and dimensions to function the same as sketch/mate relations would make replacement components operation MUCH more efficiently, which would be a **considerable** value add for any company that does a significant amount of product configuration (such as companies with Engineer-to-Order products).

This request has long been documented in the Knowledge Base as [SPR1027130](#), and has had [a post in the Enhancement Ideas forum](#). It's a pet topic of mine, and I would LOVE to see it get more traction.

assembly



#3 ASSEMBLIES

3DEXPERIENCE World 2026

💡 Explode PropertyManager enhancement

Gathering Support



Adrien MOINE 5 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

👁 13



Current status: **Gathering Support** ▾

Enhance the Explode PropertyManager so that user get:

- Ability to ungroup several/all components in a single operation
- Ability to name groups

and make it follow feature tree display settings regarding displaying name/description of components.

#2 ASSEMBLIES

3DEXPERIENCE World 2026

💡 Assembly - Make independent: possibility to choose Virtual or External when saving component instances as a new file

Gathering Support



Adrien MOINE 5 month(s) ago - Edited

The Top Ten List - 3DEXPERIENCE World 2026

👁 12

Current status: Gathering Support

Add a new "assemblies option" in System options about saving component Instances as new files and let user choose between:

- Virtual (Save component instances as New Virtual Files)
- External (Save component instances as New External Files)
- Prompt (Ask if you want to save component instances as new virtual or external files)

assembly

#1 ASSEMBLIES

3DEXPERIENCE World 2026

💡 Assembly - Extra assembly instance data "Reference component" entry fields

Gathering Support



Adrien MOINE 5 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

16

Current status: Gathering Support

Currently, there is only one "reference component" entry field for instance data in an assembly. Having more of these data entry fields would be a big improvement. The reference component data field in the component properties can be used to input the specific data of each instance in an assembly.

Ideally, there would be as many instance data fields as needed.

The more instance data we can integrate into SolidWorks files, the less dependent we are on other sources of data. The principle of "one unique source of truth" is very important. When creating assemblies in SolidWorks, users want to link instance data directly to objects in the assembly, rather than doing so in another application (Excel) or data management system (PDM/PLM), as this approach lacks the virtual contact of the 3D assembly. This working method causes data desynchronization and a higher margin of error.

There should be the possibility to add a description for each instance of a component reference (e.g., position number, comments, etc.). This feature should be compatible with the SOLIDWORKS PDM environment (see "Look for variable in reference-specific values").

They could be shown and managed as pictured below:

#1 ASSEMBLIES (CONT.)

3DEXPERIENCE World 2026

💡 Assembly - Extra assembly instance data "Reference component" entry fields

Gathering Support



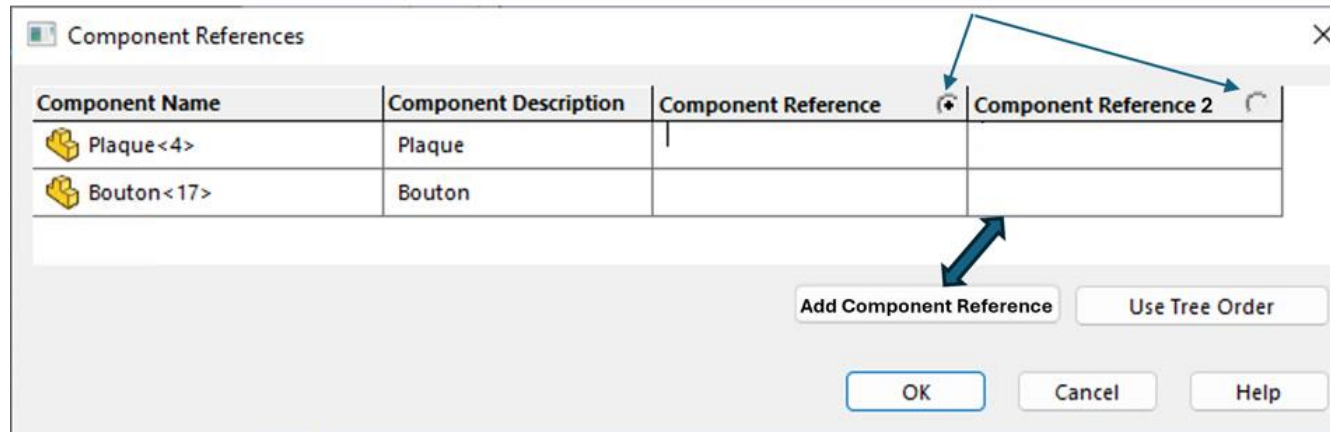
Adrien MOINE 5 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

16

Current status: Gathering Support

Option to select which Component Reference to display in the FeatureManager Tree



ASSEMBLIES

3DEXPERIENCE World 2026

1. *Extra Asm. instance data "Reference component" entry fields*
2. *Make independent: possibility to choose Virtual or External when saving component instances as new file*
3. *Explode PropertyManager enhancement*
4. *Maintain Dimension and Annotation References for Named Entities*
5. *Allow asm-level features to be associated to a smart component*



PARTS

Top Ten Enhancement Ideas 2026

#5 PARTS

3DEXPERIENCE World 2026

💡 Material Properties: Ability to retrieve Material Properties the same as model custom properties (\$PRPMAT)

Gathering Support



Adrien MOINE 5 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

👁️ 19



Current status: Gathering Support

Ability to retrieve material properties in the same way as custom model properties (e.g. \$PRPMAT) so that they can be used in BOMs and drawings (annotations and title block)

Ideally, this functionality would take into account materials assigned to bodies (e.g. \$PRPWLDMAT).

It would be highly advantageous for managing material impacts (costs, supplier, etc.).

part

#4 PARTS

3DEXPERIENCE World 2026

💡 "Insert Part" & "Save bodies" features homogenization

Gathering Support



Adrien MOINE 5 month(s) ago - Edited

The Top Ten List - 3DEXPERIENCE World 2026

👁 19



Current status: Gathering Support ▾

Description:

Currently, when editing Insert part feature in the FM Design Tree, the source file path is not available for change like it is with the Stock-Part feature (Save bodies feature used in source file).

On the other hand, select items to transfer (axes, plans, cosmetic threads, hole wizard datas) or not selectable when using Save bodies or editing the Stock-Part feature.

Suggested improvement:

It would be very very advantageous to streamline these two features by allowing the selection of items to transfer in Save Bodies /Stock-Part features:

#3 PARTS

3DEXPERIENCE World 2026

💡 Show dims when double-clicking on Feature from breadcrumbs

Gathering Support



Adrien MOINE 4 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

👁 20



Current status: Gathering Support ▾

With Instant3D disabled, when you want to view the sketch and its dimensions, you need to double-click on its feature in the FM tree. Double-clicking on the feature from Breadcrumbs should have the same behavior.

part

#2 PARTS

3DEXPERIENCE World 2026

💡 Ability to change type of pattern within its feature

Gathering Support



Anne Bouman 2 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

40

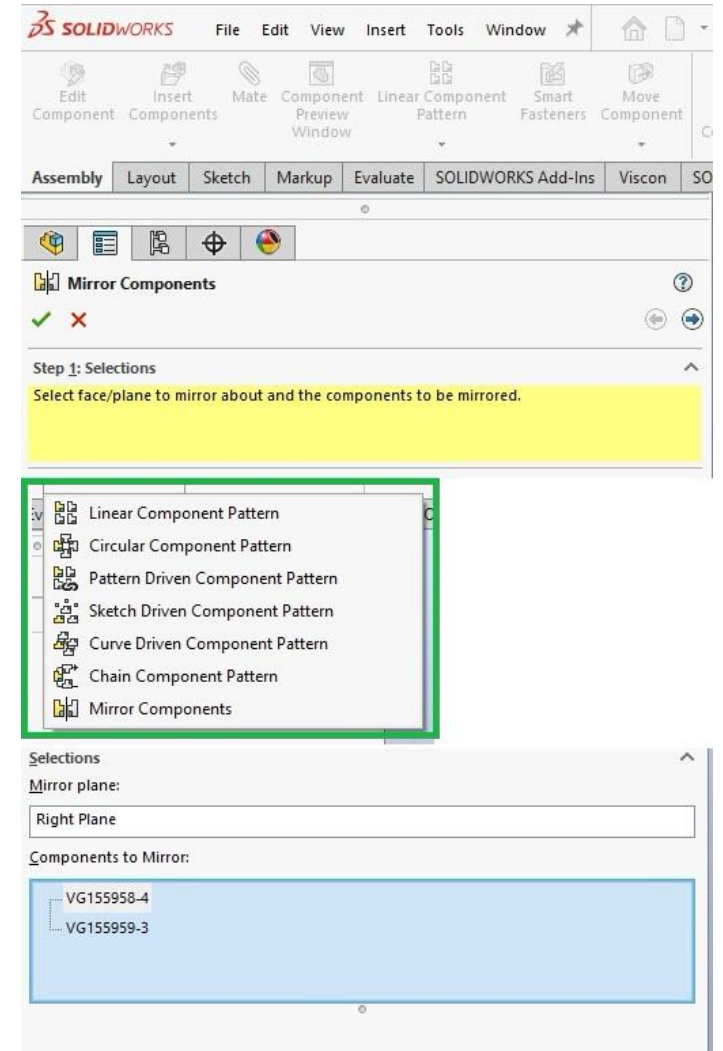
Current status: Gathering Support

Would like to change the pattern type without the need to recreate it.

Current use: start a mirror pattern, find out you actually need circular, so build it again via circular pattern.

Wish use: start a general 'Pattern' feature (not existing yet), select 'Mirror' via a dropdown menu within Pattern feature. In case you needed another pattern type, just select your choice in the dropdown menu within Pattern feature.

This would be useful for both Assemblies and Parts.




#1 PARTS

3DEXPERIENCE World 2026

🔦 SPR 807836 – Fillet all Concave Corners Connected to Start Loop

Gathering Support

 **Dave Laban** 3 month(s) ago - Moderated
The Top Ten List - 3DEXPERIENCE World 2026

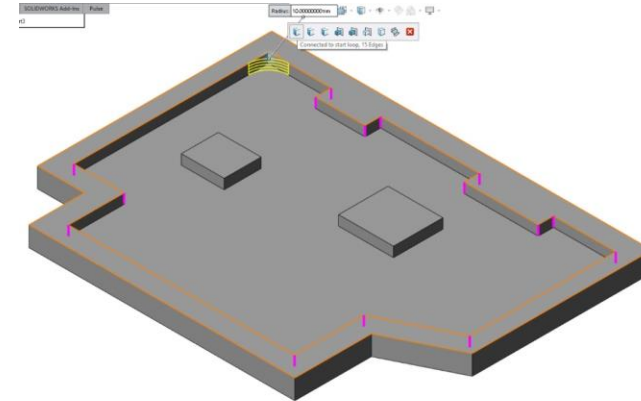
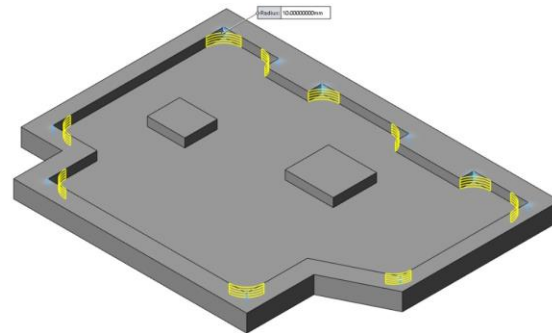
120

Current status: Gathering Support

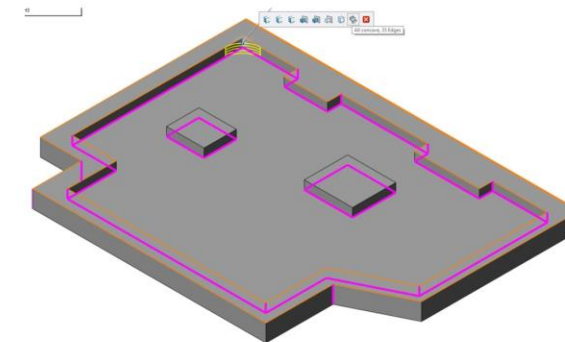
When selecting edges for fillets, numerous tools are available to speed up the selection process, e.g. select all connected to start loop, or all concave corners. It would be great to add another filter that combines these two particular filters in to “all concave corners connected to start loop” to select all the edges that would be affected by a single machining operation, e.g. a complex pocket with a flat end mill – it would be rare to need to fillet the convex corners, while all the concave ones *must* be filleted. The existing “all concave corners” filter selects all of the corners in the model, which is likely more than the pocket under consideration.

Example: “Connected to Start Loop” picks up all convex corners (unnecessary for machining pockets).

Being able to combine “connected to start loop” and “all concave” would give the following selection, replicating what would be machined:



Example: “All concave” assesses the whole model.



PARTS

3DEXPERIENCE World 2026

1. *[SPR] Fillet all Concave Corners Connected to Start Loop*
2. *Ability to change type of pattern within its feature*
3. *Show dims. when double-clicking on Feature from breadcrumbs*
4. *"Insert PRT" and "Save Bodies" features homogenization*
5. *Ability to retrieve Material properties the same as model custom properties*



DRAWINGS

Top Ten Enhancement Ideas 2026

#5 DRAWINGS

3DEXPERIENCE World 2026

Allow Dimension Lines to Break on Other Annotation Types

Gathering Support



Glenn Schroeder 1 month(s) ago - Moderated

 The Top Ten List - 3DEXPERIENCE World 2026

 33



Current status: **Gathering Support** 

It is sometimes difficult to avoid having a leader for a Note, Weld Symbol, Balloon, etc crossing the extension line of a Dimension. It would be very helpful (and make the drawing look better) to be able to break those dimension lines on other annotation types than just dimensions.

#4 DRAWINGS

3DEXPERIENCE World 2026

💡 Allow Re-naming Models from Drawing

Gathering Support



Glenn Schroeder 5 month(s) ago - Moderated

👁 62

🏆 The Top Ten List - 3DEXPERIENCE World 2026



Current status: Gathering Support ▾

We can currently re-name Parts and sub-assemblies from the tree in an Assembly. I would like the same option in Drawings.

#3 DRAWINGS

3DEXPERIENCE World 2026

💡 Drawing - Balloon qty: option to display pattern qty

Gathering Support



Adrien MOINE 5 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

👁️ 21



Current status: **Gathering Support** ▾

When adding quantity to a balloon attached to a component that is part of a pattern (seed component or pattern component) in a drawing, it would be very useful to be able to choose whether to display either the pattern quantity or the total quantity of the component in the assembly.

#2 DRAWINGS

3DEXPERIENCE World 2026

💡 Display States in Drawings

Gathering Support



Jamie Hill 4 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

👁 107



Current status: **Gathering Support** ▾

Display states used to isolate specific bodies or parts in drawings simply do not work. The Drawing View does not resize to just the isolated components, and you are still able to select geometry that has been hidden in the Display State - making dimensioning impossible.

Display States have many advantages to configurations, but they are in a lot of cases un-usable in drawings when it comes to detailing isolated bodies or parts due to these oversights. It would be a game changer to have these minor issues fixed!

drawing

#1 DRAWINGS

3DEXPERIENCE World 2026

💡 Add attachments (eg. STEP, DXF, STL) when exporting a drawing PDF

Gathering Support



Tracey Lem 1 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

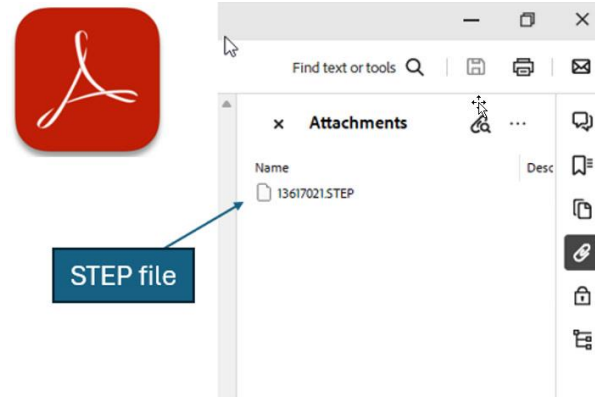
85

Current status: Gathering Support

It would be valuable to have an ability to add attachments when exporting a drawing as PDF, by using Save-As to PDF or using a PDM task to create a PDF. This would be most useful for STEP files or any other neutral format like .dxf, .stl, .x_t, .igs.

For MBD, there is already an ability to include attachments when exporting a 3D PDF. It allows STEP files to be included on the PDF automatically, and it ensures that the STEP file matches the drawing. It saves back-and-forth for requesting a STEP file and save people's time of manually exporting STEP files.

This ability will be useful for 2D drawings as well.



DRAWINGS

3DEXPERIENCE World 2026

1. *Add attachments (STEP, DXF, STL) when exporting a drawing PDF*
2. *Display States in Drawings*
3. *Balloon qty: option to display pattern qty*
4. *Allow Renaming Models from Drawing*
5. *Allow dimension lines to break on other annotation types*



SHEET METAL

Top Ten Enhancement Ideas 2026

#5 SHEET METAL

3DEXPERIENCE World 2026

💡 Sheet Metal: Have Break Corner spinner follow System Options setting.

Gathering Support



Kevin Chandler 1 month(s) ago - Moderated

🚩 The Top Ten List - 3DEXPERIENCE World 2026

👁 9



Current status: Gathering Support

As of SW2025, the spinner control in Sheet Metal's Break Corner PM doesn't increment in the amount set in System Options>Spin Box Increments. Please correct this.

📦 sheet metal 📍 Core SOLIDWORKS

#4 SHEET METAL

3DEXPERIENCE World 2026

💡 SHEET METAL: SW-Sheet Metal Gauge should use the name of your gauge table selection without any substitutions.

Gathering Support



David Mandl 5 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

👁 26

Current status: Gathering Support ▾

SW 2023 introduced a new sheet metal cutlist property called "SW-Sheet Metal Gauge". What it did was provide a text-based property to translate your Gauge Table selection to a description.

However, the name of your Gauge Table selection goes through some text edits before it's reported out as the "SW-Sheet Metal Gauge" property. You can test this out with the default gauge tables or with your own custom gauge tables, but we've found several things happening:

- The word "aluminum" is shortened
- The word "gauge" is removed entirely
- No matter what, "GA." is appended to the end of the text string.

The "GA." addition in particular has caused my company to not directly use this property at all. If we choose 14 Gauge material or 1/4" plate material, the description ends up as either 14GA. or 1/4"GA., which is too similar for comfort. My company in particular also has a variety of types of stock that we use gauge tables for (gauge stock, plate stock, embossed tread plate, etc.). We have to use a redundant custom property with a VERY long and cumbersome equation to replace it.

The single thing that could make this property immediately more useful would be if there were *zero* changes to the name of the Gauge Table selection. Let us decide what appears in that property by customizing the selections in our Gauge Table.

#3 SHEET METAL

3DEXPERIENCE World 2026

💡 SHEET METAL: Drive SW-Material via Gauge Tables

Gathering Support



David Mandl 5 month(s) ago - Moderated

🏆 The Top Ten List - 3DEXPERIENCE World 2026

👁 18



Current status: Gathering Support ▾

This is a relatively straightforward enhancement: allow for a Gauge Table to include a column that would drive the Material of the model (preferably at the Part level for the Sheet Metal folder, otherwise at the cutlist/body level).

📦 [sheet metal](#)

#2 SHEET METAL

3DEXPERIENCE World 2026

🔍 Fix Sheet Metal DXF exporting once and fall all.

Gathering Support



Adrian Crisci 2 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

47

Current status: Gathering Support

Why can't you guys fix the toggles ? so we do not have to pick and choose EVERY single time.

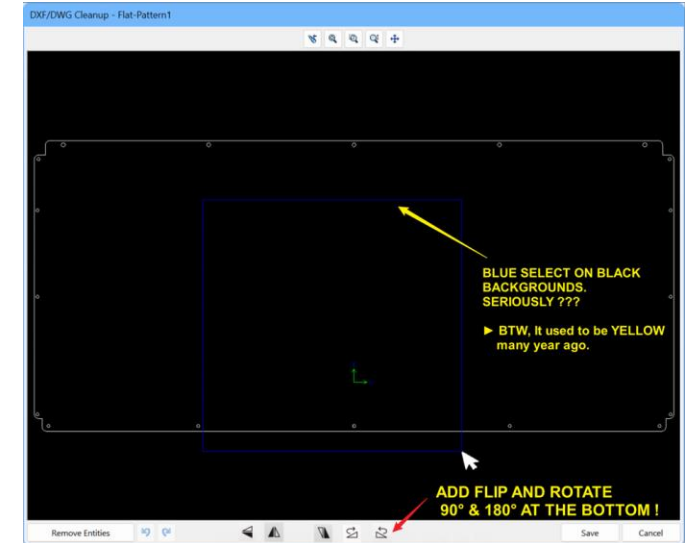
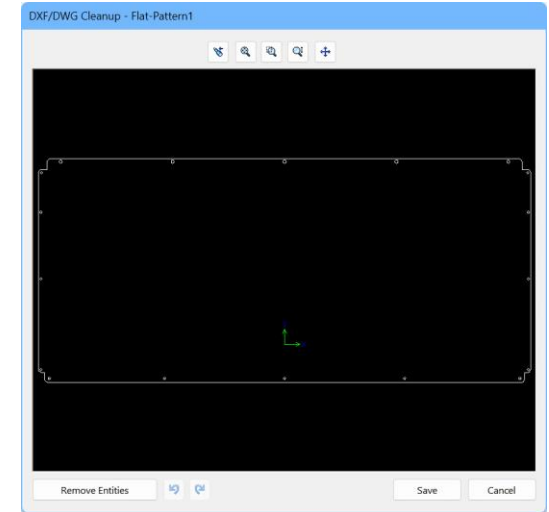
Also remove the abomination of output alignment. It never works. You always have a 90% change of having the part flipped in the wrong direction... specially if your sheet metal (like stainless) has a protective film... the film must always face up. You as the engineer know which way the part should face in the punch or laser unit. The flat part in the cleanup screen is always flipped the wrong way... and you have to flip around several times with X & Y until it comes out the the right orientation. It's annoying !

Remove the output alignment in lieu of buttons in the clean up window (last picture - see below).

Why is the cleanup window always very small ? Why can't it remember the size you previously used (just like the open menu) ?

When you have large parts with tons of features... zooming in and out does not cut it. Remember the cleanup window size !

ADD buttons on the cleanup window at the bottom (or top) to flip or rotate the DXF by 90° left / right or 180°. This will fixt the part from always showing up flipped or rotated the wrong way by simply allowing you to flip the part the right way without much fuzz.



#1 SHEET METAL

3DEXPERIENCE World 2026

We want more flexible sheet metal functionality.

Gathering Support



Myungjin Kim 3 month(s) ago - Moderated

 The Top Ten List - 3DEXPERIENCE World 2026

 172



Current status: **Gathering Support** 

One of our customers is a company that manufactures connectors.

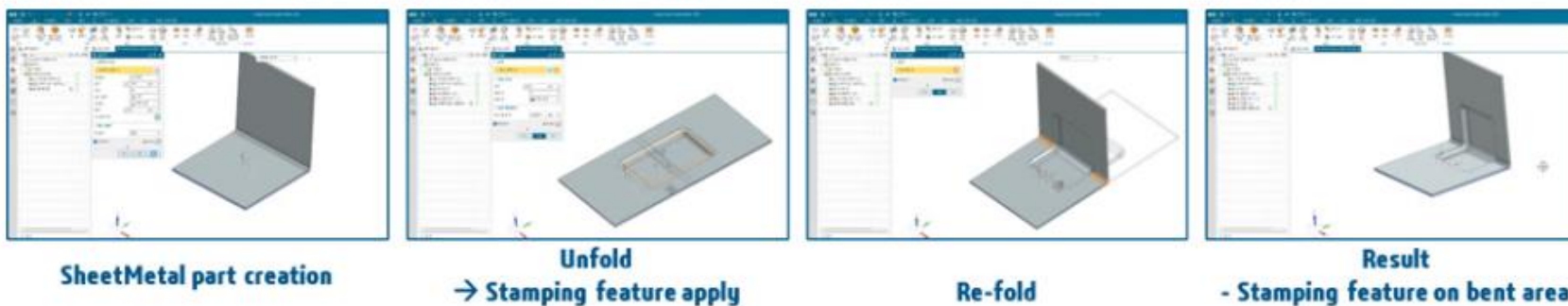
This company produces various types of connectors, and the internal pins used in these products are created using sheet metal features.

However, some operations that were possible in their previous CAD tools cannot currently be performed in SOLIDWORKS. Therefore, we would like to request an improvement in this area.

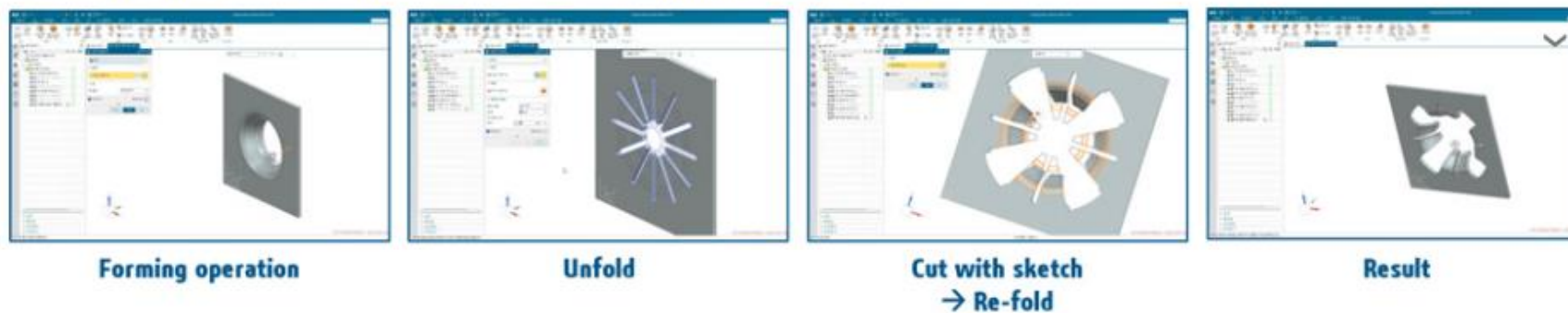
#1 SHEET METAL (CONT.)

3DEXPERIENCE World 2026

1. **Unfold a bent area, apply a stamping feature, and then rebend it.**



2. **Perform multiple bends along an arc, unfold for cutting, and then rebend.**



Based on our review, operation (1) is supported in **NX**, **SOLID Edge**, and **Inventor**, while operation (2) is supported in **NX** and **SOLID Edge**.

We would greatly appreciate it if SOLIDWORKS could introduce more flexible sheet metal capabilities to support these workflows as well.

SHEET METAL

3DEXPERIENCE World 2026

1. *More flexible sheet metal functionality*
2. *Fix Sheet Metal DXF exporting once and for all*
3. *Drive SW-Material via Gauge Tables*
4. *SW-Sheet Metal Gauge should use the name of your gauge table selection without any substitutions*
5. *Have Break Corner spinner follow System Options setting*



UX

Top Ten Enhancement Ideas 2026

#5 UX

3DEXPERIENCE World 2026

💡 Undo/Redo - Un-Limited/Un-Restricted

Gathering Support



Rick Becker 5 month(s) ago - Moderated

🏷️ The Top Ten List - 3DEXPERIENCE World 2026

👁️ 27



Current status: Gathering Support ▾

Would like to have a true Undo and Redo function similar to many other CAD systems/programs.

Within a given session (session defined as the period between opening and closing (not just saving) a file) have the ability to step back, one-by-one, each and every command (Undo), and, have the ability to step forward, one-by-one, each and every command (Redo).

📦 user interface

#4 UX

3DEXPERIENCE World 2026

💡 UI - Features naming settings

Gathering Support



Adrien MOINE 5 month(s) ago - Moderated

[The Top Ten List - 3DEXPERIENCE World 2026](#)

👁 21



Current status: **Gathering Support** ▾

Add system options to define the name of the main features based on their parameter.

Taking cosmetic threads for example: when not part of a Tapped Hole made via the Hole Wizard, "M4_1" is much more meaningful than "Hole Thread1"

 user interface

#3 UX

3DEXPERIENCE World 2026

💡 Display states: individualize the "link display states to configurations" option

Gathering Support



Adrien MOINE 5 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

👁 25



Current status: Gathering Support ▾

Individualize the "link display states to configurations" option, so as to make only one or more existing display states available for all configurations.

Currently, this option controls the link of all display states. In short, it's all or nothing.

[user interface](#)

#2 UX

3DEXPERIENCE World 2026

💡 Add a command to reach the related pattern seed component from a selection

Gathering Support



Adrien MOINE 1 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

37

Current status: Gathering Support

Description:

In an assembly with component patterns, when you want to select the seed component in the graphic area, we rarely succeed (in other words, your selection is often a component resulting from the pattern feature).

Furthermore, you sometimes cannot identify that the component you have selected is a result of a pattern because the pattern feature includes so many components that it is not visible in the feature manager.

Suggested improvement:

- Enhance the Breadcrumbs tool to display the pattern feature and the seed component of the patterned one selected.

OR/AND

- Ad an option in contextual menu (RMB) "Go to Seed Component"



#1 UX

3DEXPERIENCE World 2026

💡 Faster, Modern BOM Editor in a Separate Window in SW assemblies and drawings

Gathering Support



Peter Virag 3 month(s) ago - Moderated

🏆 The Top Ten List - 3DEXPERIENCE World 2026

👁 60

Current status: Gathering Support ▾

Working with large Bills of Materials (BOMs) in SOLIDWORKS can be slow and cumbersome. While the existing BOM functionality is good, its performance drops significantly with large assemblies.

It would be a huge improvement to have a **fast, modern BOM editor** that runs in a **separate window**, designed for high performance and responsiveness with large datasets.

Currently, even when the BOM is shown in a separate window, it remains slow and clunky. A redesigned BOM editor — optimized for speed, modern UI, and smoother interaction — would greatly enhance productivity for users managing complex assemblies.

📦 user interface

USER EXPERIENCE

3DEXPERIENCE World 2026

1. *Faster, Modern BOM editor in separate window in SW ASM and DRW*
2. *Add a cmd to reach the related pattern seed component from a selection*
3. *Individualize the "link display states to configs" option*
4. *Features naming settings*
5. *Unlimited/unrestricted Undo/Redo*



SKETCH

Top Ten Enhancement Ideas 2026

#5 SKETCH

3DEXPERIENCE World 2026

💡 Convert Entities: Option to convert geometry when selecting a complete construction geometry sketch

Gathering Support



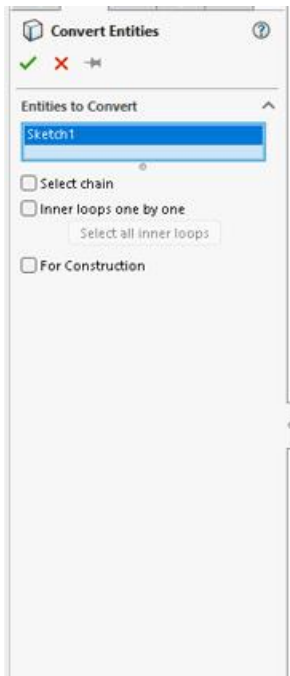
Deepak Gupta 3 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

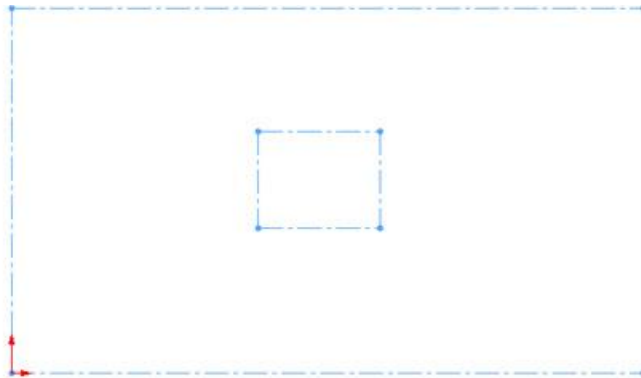
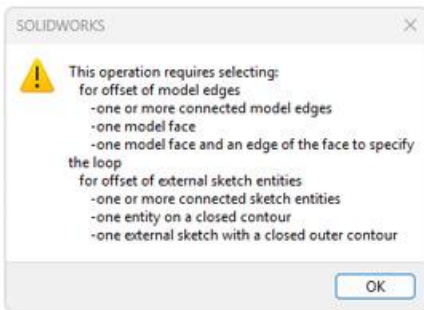
👁 17



Current status: **Gathering Support** ▾



Currently, converting an entire construction sketch using the Convert Entities feature is not supported; it only works when individual sketch segments are selected. Removing this limitation and allowing Convert Entities to operate on selected construction geometry sketches would improve usability.



#4 SKETCH

3DEXPERIENCE World 2026

💡 Add button option: Text Centered Vertically (Sketch)

Gathering Support



Nicholas Roche 2 month(s) ago

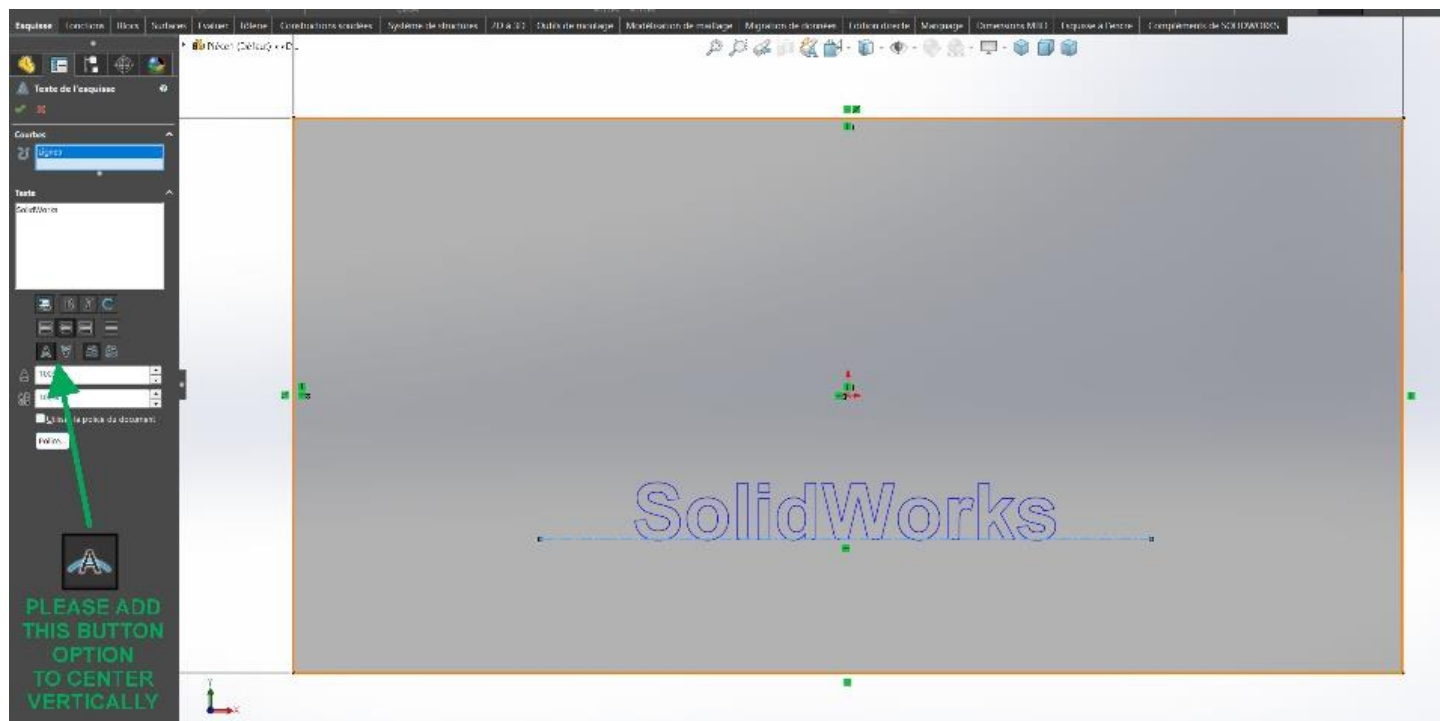
Forwarded from Atomic Design

13

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Current status: Gathering Support

When you need to add text to a surface, there is no BUTTON option to center the text vertically to the associated construction line.



#3 SKETCH

3DEXPERIENCE World 2026

💡 Sketch Convert Entities: Option to keep construction geometry to construction geometry when using convert entities.

Gathering Support



Deepak Gupta 5 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

👁 19



Current status: Gathering Support ▾

The Convert Entities tool allows us to convert existing sketches or sketch segments into new ones. However, the centerlines or construction geometry do not retain their construction status by default after using the converted entities tool.

Currently, users must manually convert these entities back to construction geometry (if required) after using the tool, which can be time-consuming. It would be more efficient if there were an option to automatically retain construction geometry as construction geometry during the conversion process. To make it simple, and user driven, a checkbox can be added in the Convert Entities Property Manager, allowing users to retain construction geometry with a single click.

#2 SKETCH

3DEXPERIENCE World 2026

3D sketches: sketch plane

Gathering Support



Jamie Hill 4 month(s) ago - Moderated

 20

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Current status: **Gathering Support** 

3D Sketch Planes should have the same robust functionality as Feature Planes. They should be editable (able to redefine references), should need to be fully defined (else, present a warning), be defined using equations, and ideally be configurable.

#1 SKETCH

3DEXPERIENCE World 2026

💡 Sketch blocks: possibility to apply color

Gathering Support



Adrien MOINE 1 month(s) ago - Moderated

👁 28

🏷 The Top Ten List - 3DEXPERIENCE World 2026

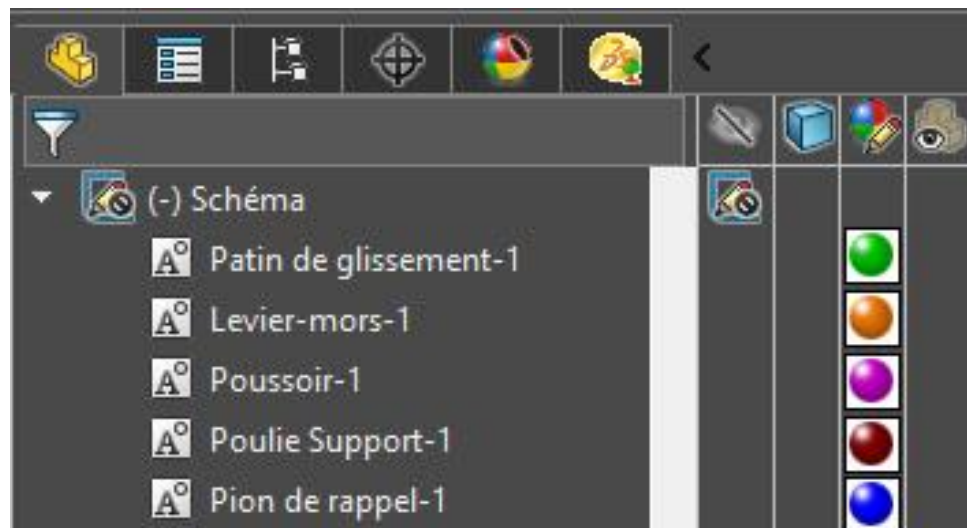


Current status: Gathering Support ▾

In the FeatureManager design tree, add the possibility to apply color to individual blocks in sketches.

Use case:

When designing a product from a master model, you must use a single layout sketch with multiple sketch blocks, otherwise it becomes much more difficult to represent a mechanism. Adding a different color for each sketch block will allow you to distinguish between them.



SKETCH

3DEXPERIENCE World 2026

1. *Sketch Blocks: ability to add color*
2. *3D Sketches: Sketch Plane*
3. *Sketch convert entities: Option to keep construction geometry when using convert entities*
4. *Add button option: Text Centered Vertically*
5. *Convert Entities: Option to convert geometry when selecting a complete construction geometry sketch*



MODEL-BASED DEFINITION

Top Ten Enhancement Ideas 2026

#5 MBD

3DEXPERIENCE World 2026

🔍 MBD: Comply with SAE J1231 - Ability to add Runout tolerances, max diameter on torus features

Gathering Support



Tracey Lem 1 month(s) ago - Moderated

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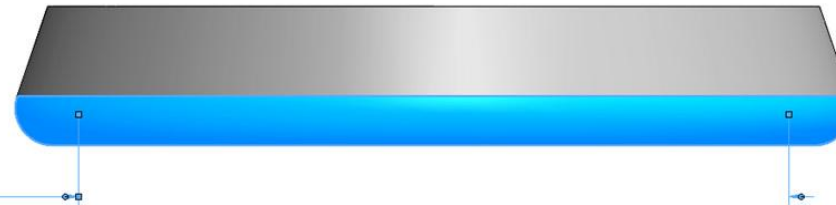
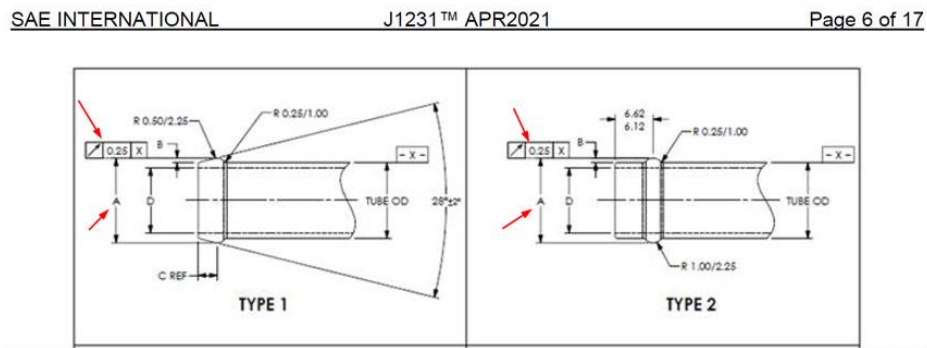
89

Current status: Gathering Support

Ability to add Runout and a max diameter on Torus features. This would allow it to be compliant with the SAE J1231 standard for barb designs.

Currently, you can only add Line Profile, Surface Profile and Circularity GD&T tolerances to torus features in MBD. When a diameter is added, it will be taken from the center of the torus and there is no option to customize between center/min/max.

Ideally, there would be controls similar to a distance between two cylindrical features (3rd image).



$\phi 164.53 \pm 0.20$

No option to add Max/OD diameter, the diameter is calculated to the center of the torus radius.

Dimension

Value Leaders Other

Leader/Dimension Line Style

- Use document display
- Style: [Dropdown]
- Thickness: 0.18mm

Extension Line Style

- Same as leader style
- Use document display
- Style: [Dropdown]

Arc Condition

First arc condition:

- Center
- Min
- Max

Second arc condition:

- Center
- Min
- Max

It would be ideal to customize torus dimensions similar to the distance between two cylindrical features.

#4 MBD

3DEXPERIENCE World 2026

💡 Add attachments (eg. STEP, DXF, STL) when exporting a drawing PDF

Gathering Support



Tracey Lem 1 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

87

Current status: Gathering Support

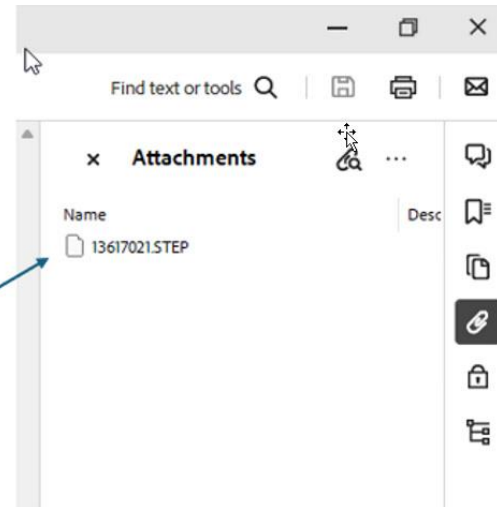
It would be valuable to have an ability to add attachments when exporting a drawing as PDF, by using Save-As to PDF or using a PDM task to create a PDF. This would be most useful for STEP files or any other neutral format like .dxf, .stl, .x_t, .igs.

For MBD, there is already an ability to include attachments when exporting a 3D PDF. It allows STEP files to be included on the PDF automatically, and it ensures that the STEP file matches the drawing. It saves back-and-forth for requesting a STEP file and save people's time of manually exporting STEP files.

This ability will be useful for 2D drawings as well.



STEP file



#3 MBD

3DEXPERIENCE World 2026

💡 Intelligent Assembly Driven MBD

Gathering Support



Grant Mattis 1 month(s) ago - Moderated

✂ The Top Ten List - 3DEXPERIENCE World 2026

👁 0



Current status: Gathering Support ▾

The industry is at a turning point. Model-Based Definition (MBD) must evolve beyond parametric annotations, into assembly-driven workflows. This enhancement marks a fundamental shift for widespread adoption, and reveals a clear path to tolerance-based design with GD&T at its core.

SolidWorks MBD and DimXpert often feel like recreating 2D drawings in 3D. They help define tolerances but lack assembly-level

#2 MBD

3DEXPERIENCE World 2026

💡 Extend the DimXpert "Edit Feature" to edit collections, Intersect Circles, Surface of Revolution

Gathering Support



Jane Liu 3 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

👁 123



Current status: **Gathering Support** ▾

Currently, for DimXpert features, the "Edit Feature" function is only available for Plane, Cone, Cylinder, Simple Hole, Boss, Manual Pattern, Counterbore hole, Width.

Following features cannot be edited: Constructed Features (Eg. Intersect Circle); Collection; Lined Pattern; Surface of Revolution.



Once those features become dangling, there is no way to fix but to delete and recreate. It would be nice to expand the "Edit Feature" function to include those features, which will make the MBD creation much more efficient.

#1 MBD

3DEXPERIENCE World 2026

💡 MBD: Improve ability to manage DimXpert Tree and delete unused features

Gathering Support



Tracey Lem 1 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

161

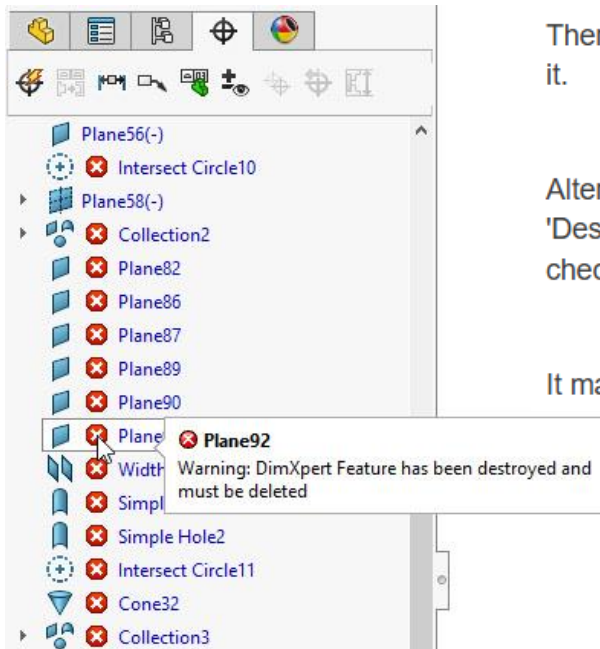


Current status: Gathering Support

There should be a function to remove all 'Destroyed' features with one-click, and all features that no longer have a PMI associated with it.

Alternatively, 'Destroyed' DimXpert features should be accessible through API so it can be deleted with a macro or script. Currently, 'Destroyed' DimXpert features do not have a valid API object so it cannot be selected or deleted with a script. Users currently need to check and delete features 1-by-1.

It makes the DimXpert Tree harder to navigate, and it's unintuitive for new MBD users to understand.



MBD

3DEXPERIENCE World 2026

1. *Improve ability to manage DimXpert Tree and delete unused features*
2. *Extend the DimXpert "Edit Feature" to edit collections, Intersect Circles, Surface of Revolution*
3. *Intelligent Asm-driven MBD*
4. *Add attachments (STEP, DXF, STL) when exporting a drawing PDF*
5. *Comply with SAE J1231 - Ability to add Runout tolerance, max diameter on torus features*



ELECTRICAL

Top Ten Enhancement Ideas 2026

#5 ELECTRICAL

3DEXPERIENCE World 2026

💡 Assembly - Allow assembly-level-features to be associated to a smart component

Gathering Support



Adrien MOINE 5 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

👁 14



Current status: Gathering Support

Since it's better to use a repetition function than multiplying a component and its mates, it would be really nice to be able to associate this type of feature, performed at the assembly level, to a smart component.

For a guide rail, for example, the current workaround consists in associating, when defining the smart component, a drilling function with only a single tapped hole, the pattern of this tapped hole and a single screw, thus directing the designer towards the realization of the screw pattern by a pattern-driven pattern feature.

A loss of productivity, as you will agree.

It will be much more simple and effective to be able to associate the pattern-driven pattern feature to the smart component in order to retrieve it in the working assembly when deploying the smart features.

#4 ELECTRICAL

3DEXPERIENCE World 2026

💡 SW PDM Paste as reference, show in BOM: ability to choose the configuration in which the reference is shown in the BOM.

Gathering Support



Adrien MOINE 4 month(s) ago - Moderated

🚩 The Top Ten List - 3DEXPERIENCE World 2026

👁 17

Current status: Gathering Support

When creating a custom reference by pasting a file as a reference to a SolidWorks part or assembly file, we currently have the possibility to check the option "Show in BOM" and set a quantity to display in BOM. That's great, but...

If the parent file got multiple configurations, I may not want to show the reference in BOM for all configurations ! That is why ability to choose the configuration(s) in which the reference is shown in the BOM would be very beneficial.

As examples, I can mention warning signs labels (picture or pdf) and technical documentation (word document or pdf) that could be different regarding the configuration of a machine.

#3 ELECTRICAL

3DEXPERIENCE World 2026

💡 SWE/ESX: Maintain wire connections when moving symbols

Gathering Support



Artur Wojcik 2 month(s) ago - Moderated

The Top Ten List - 3DEXPERIENCE World 2026

👁 52

Current status: Gathering Support ▾

When a user wants to move a symbol on a schematic while keeping the wire connections intact, this isn't possible by simply grabbing the symbol and moving it, because the wires do not follow the symbol. In short, the connection between the wire and the symbol is lost.

Currently, we use a workaround with the "Stretch" tool, but the user has to perform several steps, as shown in my video below. The number of steps depends on the situation. This feature would be very useful for all users.

#2 ELECTRICAL

3DEXPERIENCE World 2026

SOLIDWORKS API Documentation - Improve the UI

Gathering Support

Ian Farquharson 1 month(s) ago - Moderated
The Top Ten List - 3DEXPERIENCE World 2026

165

Current status: Gathering Support

It's great that we have an online, searchable resource for the available API in SOLIDWORKS tools, but it's lacking in functionality that would make it great.

Here are a few ideas that would make it more user friendly:

2. The interface used to grey out the years that it wasn't available in, now you guess wrong and then it navigates back to the API Documentation start page

1. Make the interfaces ordered logically, not alpha-numerically
 - a. The IEdmVault* interface has a list of 5->22, but it's ordered 10->22, 5->9. This makes it difficult when trying to figure out what interface introduced a method and using the "next" button which would result in IEdmVault9 -> IEdmVersion5 instead of IEdmVault9 -> IEdmVault10.

- > IEdmVault10
- > IEdmVault11
- > IEdmVault12
- > IEdmVault13
- > IEdmVault14
- > IEdmVault15
- > IEdmVault16
- > IEdmVault17
- > IEdmVault18
- > IEdmVault19
- > IEdmVault20
- > IEdmVault21
- > IEdmVault22
- > IEdmVault5
- > IEdmVault6
- > IEdmVault7
- > IEdmVault8
- ✓ IEdmVault9
 - Members
- > IEdmVersion5

SOLIDWORKS API Help 2024

Search Help 2024

2024

\ ... \ ... \ ... \ IEdmVault22 \ Members

IEdmVault22 Interface Members

The following tables list the members exposed by IEdmVault22.

Public Methods

Name	Description
ADRunSync	Synchronizes the vault's archive servers with the Active Directory.
ClearLogs	Deletes the Archive Server Get logs.
GetColumnSets	Gets the column sets assigned to the currently logged-in user of this vault.
GetCurrentColumnSet	Gets the current column set for the currently logged-in user of this vault.
SetColumnSetID	Sets the specified column set ID in the user's registry.

Top

See Also

- IEdmVault22 Interface
- EPDM.Interop.epdm Namespace

#2 ELECTRICAL (CONT.)

3DEXPERIENCE World 2026

- Update the CSS to dock the navigation arrows for browsing interfaces at the bottom so that a user doesn't have to move their mouse up and down when the contents of the interface or members change

< IEdmVault9

IEdmVersion5 >

- Fix the display in the syntax field

.NET Syntax

```
Public Interface IModelDoc2
```

```
Dim instance As IModelDoc2
```

```
public interface IModelDoc2
```

```
public interface class IModelDoc2
```

Visual Basic (Declaration)

Visual Basic (Usage)

C#

C++/CLI

#2 ELECTRICAL (CONT.)

3DEXPERIENCE World 2026

5. Add an access diagram to the PDM API (and any others that are lacking)

6. SOLIDWORKS Electrical API documentation has partial dark mode support, but is so bad I had to add it here. I would prefer that there is only a light mode as it's useable as-is unless there is a plan to drastically improve it

a. Light mode:

b. Dark mode:

Class Name	Description
IEwAddIn	Interface that allow applications to create add-ins
IEwAddInInternalX	Interface that allow applications to create add-ins
IEwAPIX	Use this interface to initialize this API
IEwApplicationSettingsX	Use this interface to manage the application settings
IEwApplicationX	Use this interface to manage the application
IEwArchiveEnvironmentX	Use this interface to manage the archive environment with options
IEwCableCoreX	Use this interface to manage a cable core inside a cable reference
IEwCableReferenceManagerX	Use this interface to manage EwCableReferenceX objects
IEwCableReferenceX	Use this interface to manage cable reference
IEwCaptionBarControlX	Use this interface to manage the control on captionbar
IEwCaptionBarX	Use this interface to manage the caption Bar
IEwCircuitTypeManagerX	Use this interface to manage EwCircuitTypeX objects
IEwCircuitTypeX	Use this interface to manage a circuit type
IEwClassificationManagerX	Use this interface to manage EwClassificationX objects
IEwClassificationX	Use this interface to manage an IEwClassificationX object

Here are the classes, structs, unions and interfaces with brief descriptions:

- IEwAddIn**
- IEwAddInInternalX**
- IEwAPIX**
- IEwApplicationSettingsX**
- IEwApplicationX**
- IEwArchiveEnvironmentX**
- IEwCableCoreX**
- IEwCableReferenceManagerX**
- IEwCableReferenceX**
- IEwCaptionBarControlX**
- IEwCaptionBarX**
- IEwCircuitTypeManagerX**
- IEwCircuitTypeX**
- IEwClassificationManagerX**
- IEwClassificationX**
- IEwClassX**
- IEwCommandContextX**
- IEwCommandManagerX**
- IEwCommandX**
- IEwDialogAlignPartsParametersSelectorX**

#2 ELECTRICAL (CONT.)

3DEXPERIENCE World 2026

7. Add SPR's (or BR's) to methods that are marked obsolete so that developers know what methods need to be updated vs. *should* be updated to the latest method.

a. Why should someone avoid IModelDoc2::Select and use IModelDocExtension::SelectByID2? Etc...

8. Icons are missing for some interfaces (or don't load sometimes for larger ones?)

BONUS:

Create a repository of code samples of the current examples by product








Overall I am thankful for having the documentation that is available in the state it is in, but I think there is room for improvement.

 API  Core SOLIDWORKS

IModelDoc2 Interface Members

The following tables list the members exposed by IModelDoc2.








Public Properties

Name	Description
 Property ActiveView	Gets the current active model view in read-only mode. NOTE: This property is a get-only property. Set is not implemented.
 Property ConfigurationManager	Gets the IConfigurationManager object, which allows access to a configuration in a model.
 Property CustomInfo	Obsolete. Superseded by IModelDocExtension::CustomPropertyManager.
 Property CustomInfo2	Obsolete. Superseded by IModelDocExtension::CustomPropertyManager.
 Property Extension	Gets the IModelDocExtension object, which also allows access to the model document.
 Property FeatureManager	Gets the IFeatureManager object, which allows access to the FeatureManager design tree.
 Property FeatureManagerSplitterPosition	Splits the FeatureManager design tree and gets or sets the location of the split bar in the FeatureManager design tree panel.

IModelDoc2 Interface Members

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Public Properties

Name	Description
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 CustomInfo	Obsolete. Superseded by IModelDocExtension::CustomPropertyManager.
 CustomInfo2	Obsolete. Superseded by IModelDocExtension::CustomPropertyManager.
 Extension	Gets the IModelDocExtension object, which also allows access to the model document.
 FeatureManager	Gets the IFeatureManager object, which allows access to the FeatureManager design tree.
 FeatureManagerSplitterPosition	Splits the FeatureManager design tree and gets or sets the location of the split bar in the FeatureManager design tree panel.

#1 ELECTRICAL

3DEXPERIENCE World 2026

💡 SWE/ESX: Ability to create manufacturing documentation for harnesses without using Electrical 3D and SOLIDWORKS CAD

Gathering Support

Artur Wojcik 2 month(s) ago - Moderated
The Top Ten List - 3DEXPERIENCE World 2026

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Current status: Gathering Support

In the current version of SWE/ESX, it is possible to create electrical harnesses within schematic diagrams. However, generating full manufacturing documentation for these harnesses requires the Electrical 3D add-in and SOLIDWORKS CAD. As a result, users must go through the entire 3D workflow, even if their needs are limited to working only in the 2D schematic environment.

I propose adding a new functionality — or a new dedicated add-in — that would allow users to create complete harness manufacturing documentation directly within the schematic environment, without the need for Electrical 3D or SOLIDWORKS CAD. This functionality would enable users to manually create and define harnesses in 2D, including:

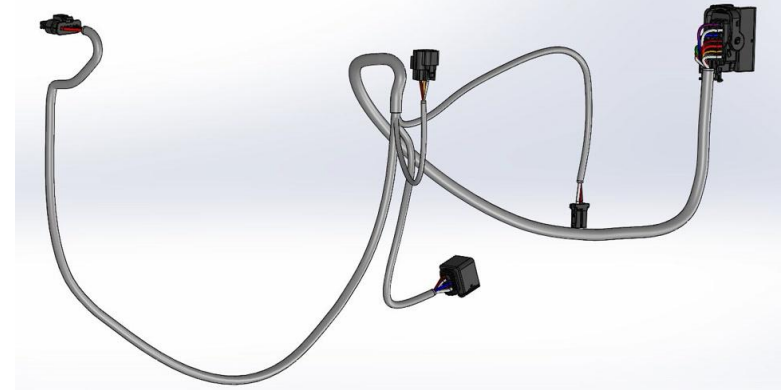
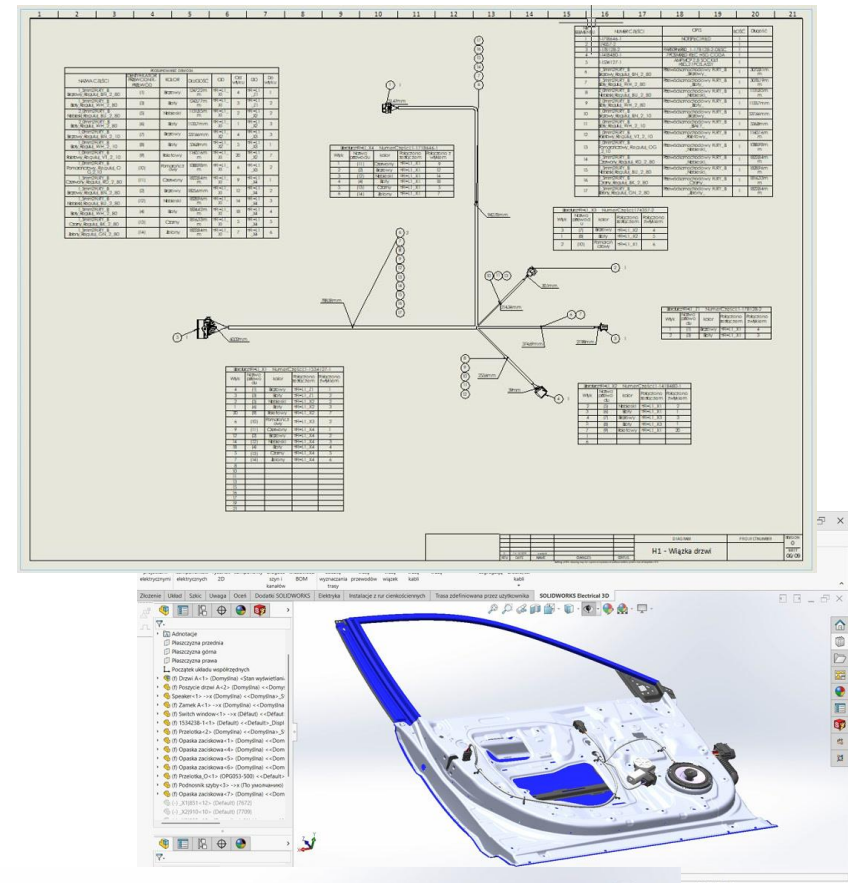
- manually specifying the length of each harness segment,
- adding labels, coverings, and other harness accessories.

Such an enhancement would greatly benefit users who do not require a 3D model and only need to quickly prepare production documentation directly from 2D schematics.

Sometimes, under time pressure, a customer needs to quickly create manufacturing documentation without having to prepare a 3D project or 3D libraries. This functionality would allow manual creation of documentation entirely from the schematic side.

This would be especially useful for companies that do not work in a 3D environment or produce harnesses based on simple, repeatable schematic layouts.

Perhaps a good idea would be to introduce a new package that includes the ability to create manufacturing documentation directly on the schematic.



ELECTRICAL

3DEXPERIENCE World 2026

1. *SWE/ESX: Ability to create manufacturing documentation for harnesses without using Electrical 3D and SOLIDWORKS CAD*
2. *SOLIDWORKS API Documentation - Improve UI*
3. *SWE/ESX: Maintain wire connections when moving symbols*
4. *SW PDM Paste as reference, show in BOM: ability to choose config in which reference is shown in BOM*
5. *Allow asm-level features to be associated to a smart component*

YOUR VOICE MATTERS!

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SURVEY IN THE MOBILE APP

The logo for 3DEXPERIENCE WORLD is positioned on the right side of the slide. It features the text "3DEXPERIENCE" in a smaller, white, sans-serif font above the word "WORLD" in a much larger, bold, white, sans-serif font. The background of the logo area is a dark blue circle containing a network of glowing blue nodes and lines, and a circular graphic in the top right corner with a blue and red split and concentric circles.

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