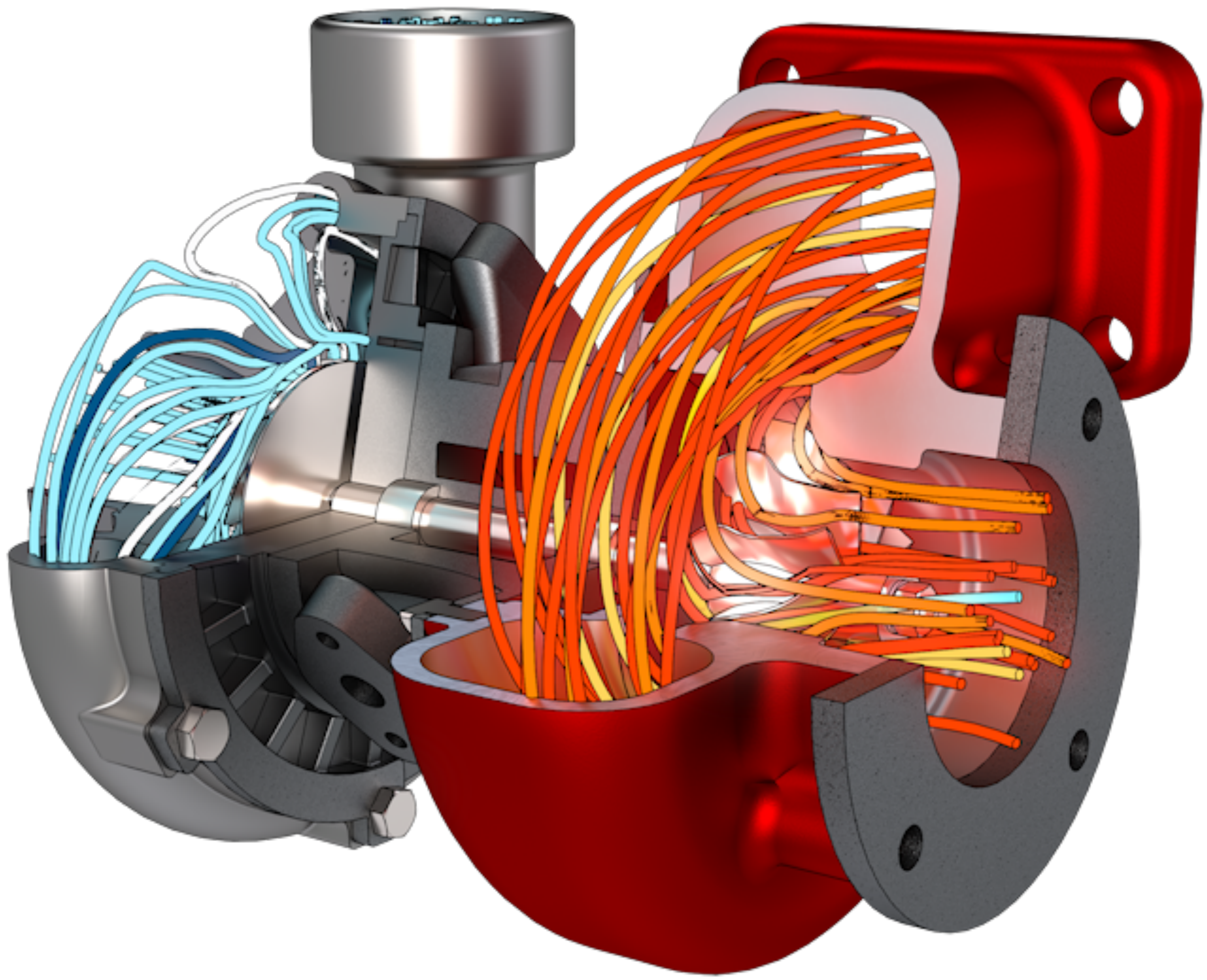
WHAT'S NEW

SOLIDWORKS 2022



Contents

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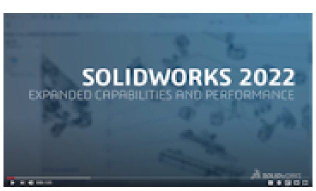
Welcome to SOLIDWORKS 2022

This chapter includes the following topics:

- **Top Enhancements**
- **Performance**
- **SPR Fixes**
- **For More Information**

Model courtesy of Rahul Gawde, SOLIDWORKS 2021 Beta Splash Screen Prize winner.

	**Video: What's New in SOLIDWORKS 2022**

At SOLIDWORKS, we know that you create great designs, and that your great designs get built. To streamline and accelerate your product development process from concept through manufactured products, SOLIDWORKS 2022 contains new, user-driven enhancements focused on:

- **Work Smarter**. Create better products in less steps with new workflows and feature enhancements in assembly and part design, drawing detailing, simulation, and product data management. Take advantage of new features in parts such as hybrid modeling and creating standardized threaded studs. Benefit from dozens of user interface enhancements such as the shortcut bar, configuration management, integrated messages, geometric tolerancing, and more.
- **Work Faster**. Get more done in less time with significant quality and performance improvements when working with large assemblies, importing `STEP`, `IFC`, and `DXF`/`DWG` files, detailing your drawings, and managing your product data. Optimize assembly performance automatically without worrying about modes and settings. Experience the freedom of our fastest graphics to date with improved display response and quality.
- **Work Together**. Improve innovation and decision making by connecting to the **3D**EXPERIENCE platform and leveraging its collaborative capabilities. Increase your competitive advantage by tapping into the power and breadth of the cloud-based **3D**EXPERIENCE Works portfolio. These expanded tools help your entire enterprise across the design and engineering, simulation, manufacturing, and governance domains.

Top Enhancements

The top enhancements for SOLIDWORKS® 2022 provide improvements to existing products and innovative new functionality.

Installation on page 14	Standard, Remote, and Compressed Administrative Images
Parts and Features on page 39	• Using Numeric Values to Define Coordinate Systems • Selection for Coordinate Systems • Draft Across Parting Lines • External Threaded Stud Wizard • Hole Wizard Slots • Hybrid Modeling • Mirroring About Two Planes
Sheet Metal on page 55	Etched Contours on Bends
Structure System and Weldments on page 57	Connection Element for a Structure System
Assemblies on page 69	• Open subassemblies in Large Design Review or resolved mode from an assembly opened in Large Design Review mode. • Use **Configuration Table** to modify configuration parameters for parts and assemblies with multiple configurations.
Detailing and Drawings on page 78	• Detailing Mode • Geometric Tolerance Symbols

	• Symmetric Linear Diameter Dimensions
Import/Export on page 89	Importing Selective IFC Entities from IFC Files

Performance

SOLIDWORKS® 2022 improves the performance of specific tools and workflows.

Some of the highlights for performance and workflow improvements are:

Import

Performance is improved for importing:

- A large `DXF` or `DWG` file into a part sketch.

  You can import a large `DXF` or `DWG` file into a part sketch with the **Explode Block** option turned off. You no longer need to explode the blocks to improve import performance.
- `STEP` files in SOLIDWORKS.

Assemblies

Improved performance when opening, saving, and closing assemblies.

Model Display

SOLIDWORKS 2022 offers improved performance for 3D textures and silhouette edges.

Functionality	Performance Improvement
3D textures	3D textures accelerate the process of refining the mesh to better match the detail in the textural appearance image. You can further refine the **Maximum Element Size** below its previous limit.
Silhouette edges	Performance is improved for rendering silhouette edges in dynamic mode. You can see the silhouette edges in **Shaded With Edges** mode.

Drawings

Performance is improved when you print large drawings on paper or to file. This applies to drawings where you:

- Have at least one drawing view as draft quality
- Specify these options in the Page Setup dialog box:
  - **High quality**

- **Color / Gray scale**

SOLIDWORKS Manage

SOLIDWORKS Manage 2022 offers improved performance for enhancing the user experience.

Functionality	Performance Improvement
Bill of material (BOM) display	When specifying the **Number of BOM levels to display** options at `1`, large BOMs display up to five times faster. For BOMs with **Link to 3rd Party** fields configured, the time required to calculate the values has decreased.
Projects	For projects with a high number of stages or Tasks, the Gantt chart display is faster than in previous releases.
Check out/Check in of SOLIDWORKS PDM files from SOLIDWORKS Manage	In previous releases, the **Check Out**/**Check In** operations refreshed the entire grid in the background. Now, only the individual line item that last changed refreshes, making performance faster.

Pack and Go

Pack and Go gathers references faster in 2022 than in previous releases. The time between initiating Pack and Go for a document and when the Pack and Go dialog box appears is reduced substantially.

SOLIDWORKS PDM

With SOLIDWORKS PDM 2022, you can experience improved performance of many file-based operations. You can perform the following actions faster for database servers with high latency:

- Open files
- Display the Save As dialog box
- Copy Tree
- Create a document in SOLIDWORKS

SOLIDWORKS PDM has improved performance for the following:

- Saving a data card with a large number of file extensions is faster by 15% to 60%.
- Checking in a drawing with a large SOLIDWORKS bill of materials (BOM) is significantly quicker.
- Displaying files in the Where Used tab with the **Show All** option and additional custom columns is many times faster for certain vaults.
- Displaying the Transition dialog box for dynamic notifications is quicker.

- Loading a Web2 preview is between 1.5 and 2 times faster for large models.

SOLIDWORKS Simulation

- Saving models that have simulation studies is faster, if at least one or more simulation studies are not modified.
- Function-based processing for the FFEPlus iterative and Intel Direct Sparse solvers is extended to simulation studies that include connectors and other features. The automatic solver selection is extended to Nonlinear, Frequency, and Buckling studies.

SOLIDWORKS Plastics

The performance of the Cool and Fill analysis modules is improved to accelerate the overall analysis time.

- For simulations where the Cool analysis takes up a large proportion of the overall solution time, the overall solution time is reduced by at least 20% compared to previous releases.
- The performance of the Fill and Pack analyses with the **Direct** solver option is optimized. The overall solution time is reduced by approximately 50% compared to previous releases. For relatively thick parts that are meshed with hexahedral elements, the **Direct** solver more accurately predicts the inertial effects.

SPR Fixes

We have fixed a large number of Software Performance Reports (SPRs) in SOLIDWORKS 2022 by development projects specifically selected to address quality and performance issues reported by customers.

See the full list of **fixed SPRs**.

SPR	Resolution
828867	Dimensions import properly when you replace a model in a drawing.
1195411	If you open a `.JT` file and select **Include PMI** when importing the file, the text on part faces imports properly.
1024876	Hole Wizard data imports properly for drawings with mirrored parts.
1083466	In an assembly, a driven part uses saved data to recreate geometry when data from the driving part is not available. This issue occurs when you switch configurations for the driven part and the data from the driving part is not available because it is in the inactive configuration.
190949	Includes the paper color when you save drawings as image and PDF file formats.

For More Information

Use the following resources to learn about SOLIDWORKS:

What's New in PDF and HTML	This guide is available in PDF and HTML formats. Click: • > **What's New** > **PDF** • > **What's New** > **HTML**
Interactive What's New	In SOLIDWORKS, appears next to new menu items and the titles of new or significantly changed PropertyManagers. Click to display the topic in this guide that describes the enhancement. To enable Interactive What's New, click > **What's New** > **Interactive**.
Online Help	Contains complete coverage of our products, including details about the user interface and examples.
SOLIDWORKS User Forum	Contains posts from the SOLIDWORKS user community on the **3D**EXPERIENCE platform (login required).
Release Notes	Provides information about late changes to our products, including changes to the *What's New* book, online help, and other documentation.
Legal Notices	SOLIDWORKS Legal Notices are available **online**.

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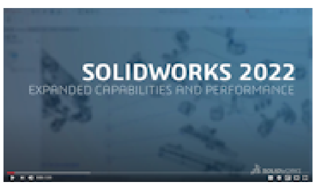
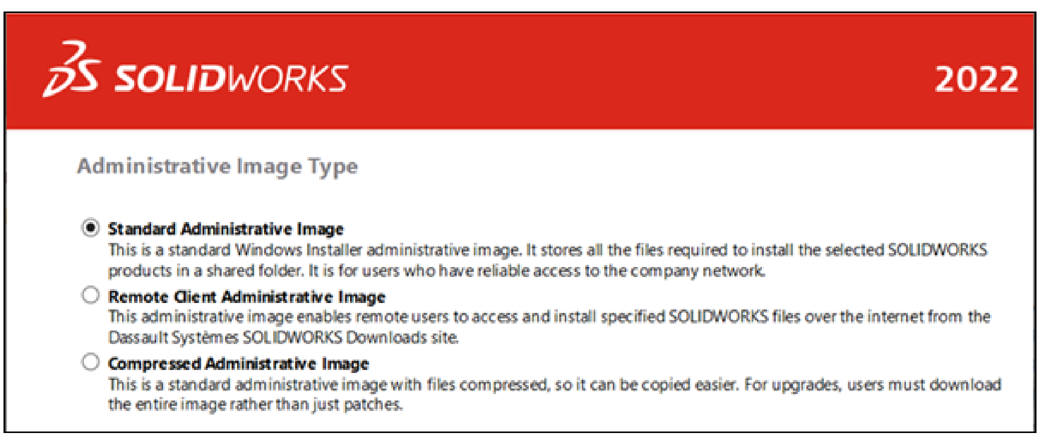
Installation

This chapter includes the following topics:

- **Standard, Remote, and Compressed Administrative Images**
- **Allow Subgroups in the Administrator Image Option Editor**
- **Improved SOLIDWORKS Electrical Client Installation**
- **Set Up a PDM Vault View When Creating an Administrative Image**

	**Video: What's New in SOLIDWORKS 2022 - Installation**

Standard, Remote, and Compressed Administrative Images



You can use the Administrative Image Option Editor to create several types of images for remote users and for those on your network.

Administrative Image Type	Description
Standard	This image is the default for all new administrative images. It is a standard Windows Installer Administrative image ideal for users who have reliable access to the company network. It stores all the files required to install the selected SOLIDWORKS products in a shared image folder. When users upgrade to a new service pack, they can download and apply patch files and do not need to download the entire image.
Remote Client	This image is designed for remote users who do not have reliable access to the company network. When installing this type of image, remote users inherit all the installation options you specify, such as the serial number and the products to install. Users download these images over the internet from the **Dassault Systèmes SOLIDWORKS Downloads** site. Accessing and installing these files from the internet is more reliable than downloading them from the company network and may reduce the load on the networks. Remote users install this image using the same techniques supported by the standard administrative image. You can still script installations using the `sldim.exe` and `startwinstall.exe` files. Installation files download to a path specified in the Administrative Image Option Editor before they install. Users must access the administrative image folder to start the installation and run any batch or settings files hosted there. Remote users must have enough available disk space to store all the files in the image, although users can delete the image after the installation completes.
Compressed	The compressed administrative image is for organizations that need to copy administrative images to multiple sites or client machines and want to minimize the impact on the network. This image is 60% smaller than a standard administrative image, but it installs in the same amount of time. It supports all install operations, such as modify and repair. Compressed administrative images also have short file paths. This avoids problems when images are stored in deep folder structures. When users upgrade to a new service pack, they must download the entire image, not only patches.

Selecting an Administrative Image Type

To select an administrative image type:

1. On the Welcome page of the SOLIDWORKS Installation Manager, click **Create an administrative Image to deploy to multiple computers**.
2. On the Summary page, next to the **Administrative Image Type**, click **Change**.
3. Select the type and click **Back** to return to the Summary page.
4. Click **Create Now**.

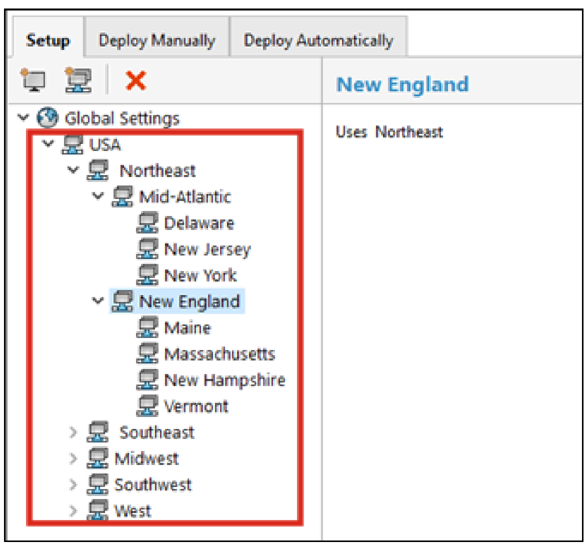
Specifying the Download Folder

In the Administrative Image Option Editor, you can specify the download folder for the remote client administrative image. This option is not available for any other administrative image type.

To specify the download folder:

1. In the Administrative Image Option Editor, under Setup, select a machine or group and click **Change** to edit options.
2. In the **Client Installation Options**, for **Where you want to download the required files**, click **Edit Path** and specify the file path.

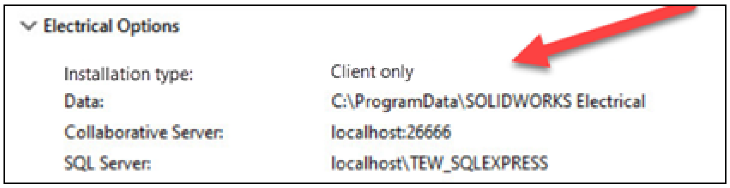
Allow Subgroups in the Administrator Image Option Editor

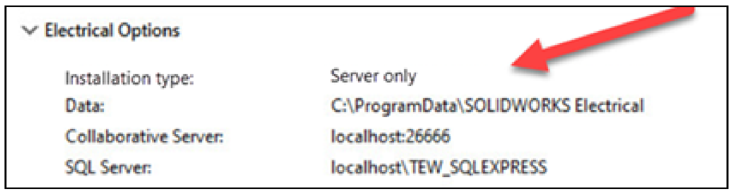


You can create subgroups in the Administrative Image Option Editor.

For example, if you have a group for all users in a country, you can create subgroups for users in different sites in the country, or with different roles. The parent group inherits the administrative image settings, but you can change those settings for subgroups. You can also move settings between groups and edit groups of machines.

Improved SOLIDWORKS Electrical Client Installation





The SOLIDWORKS Installation Manager makes it more apparent if you are installing the **Client only** or if you are including **Server** components for SOLIDWORKS Electrical.

This helps organizations that share SOLIDWORKS Electrical data and databases on a server instead of on individual machines. For this case, administrators install **Server** components once on a server and install the **Client only** on all machines running SOLIDWORKS Electrical.

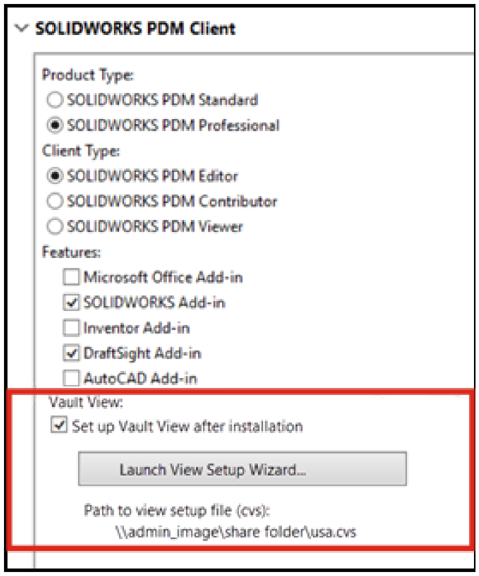
Updates include:

- The Summary page of the SOLIDWORKS Installation Manager identifies if you are installing a **Client Only** or if you are including **Server** components.
- The SQL Server options display only the options that are appropriate for a **Client only** installation.
- When the SOLIDWORKS Installation Manager cannot connect to the existing SQL Server, a warning displays, but the installation continues.

  > You can connect to the SQL Server later in SOLIDWORKS Electrical.

- The Client and Server options for installing SOLIDWORKS Electrical are also available in the Administrative Image Option Editor.

Set Up a PDM Vault View When Creating an Administrative Image



Administrators can set up a local vault view when creating an administrative image for SOLIDWORKS Enterprise PDM client users.

A local file vault view is the local working folder on a client workstation where files are cached and where a user edits the content of files that are checked out.

This feature lets administrators create a local vault view without having to create and run a separate script.

> To create the vault view on the client machine, you need a Conisio View Setup (`.cvs`) file that defines the parameters for configuring the vault view on the client machine. You can reference an existing `.cvs` file or create this file from the Administrative Image Option Editor.

Creating a PDM Vault View in the Administrative Image Option Editor

To create a PDM vault view in the Administrative Image Option Editor:

1. Under the SOLIDWORKS PDM Client options, click **Set up Vault View after installation**.
2. Do one of the following:
   - To reference an existing `.cvs` file, specify the path to this file.
   - To create a new `.cvs` file, click **Launch View Setup wizard**.

   > To open the View Setup wizard, you must run the Administrative Image Option Editor on a machine that has the SOLIDWORKS PDM client installed.

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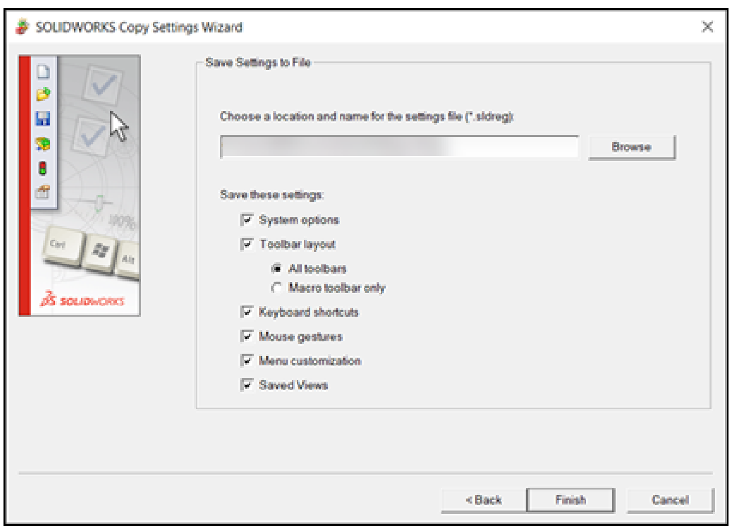
Administration

This chapter includes the following topics:

- **Copy Settings When Options Are Locked**
- **Pack and Go Includes Split and Save Bodies Features in Parts**
- **Controlling Display of Graphics Warnings in the Settings Administrator Tool**

Copy Settings When Options Are Locked

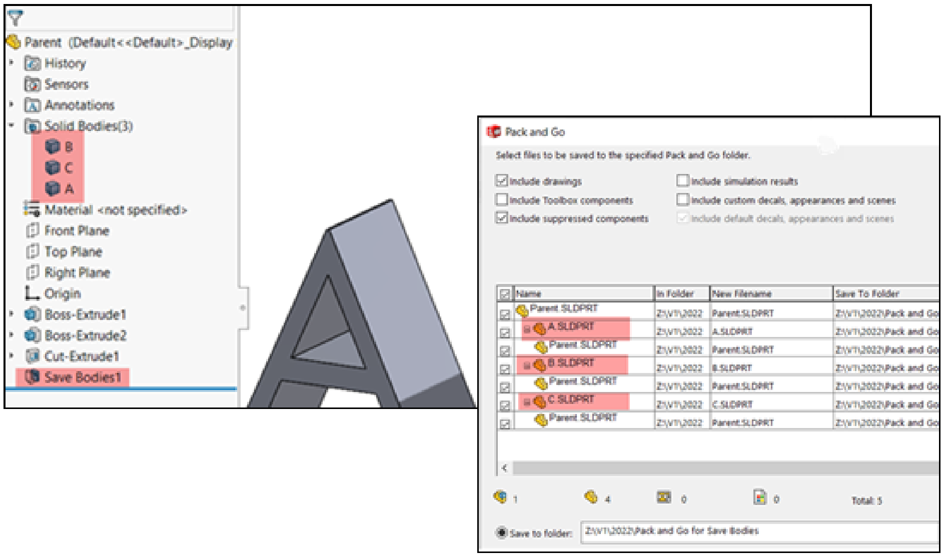


You can use the Copy Settings Wizard in SOLIDWORKS even if your administrator has locked options.

If an administrator defines options for you, the settings in your `swSettings.sldreg` file merge with the administrator settings. When settings conflict, the administrator settings override the individual settings.

To access the Copy Settings Wizard in SOLIDWORKS, click **Tools** > **Save/Restore Settings**.

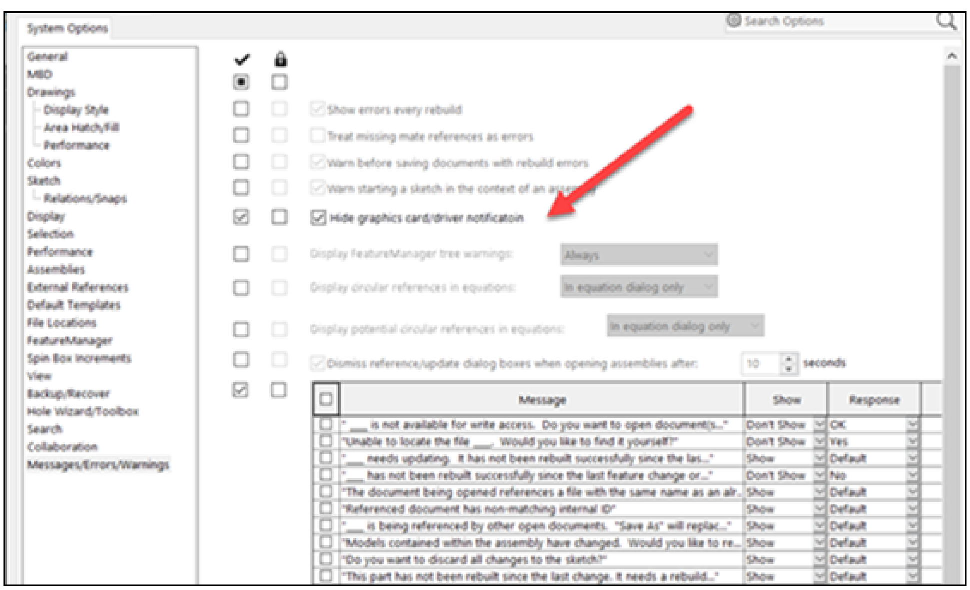
Pack and Go Includes Split and Save Bodies Features in Parts



You can include parts that you create with the **Split** and **Save Bodies** features.

When you select a parent part, Pack and Go lists the **Split** and **Save Bodies** features as references of the parent. Pack and Go also updates the names and file paths of the parent part and features when you rename and move the files.

Controlling Display of Graphics Warnings in the Settings Administrator Tool



Administrators can control whether or not users can see notifications in their Notifications area about graphics card and driver certification status.

You can turn **Hide graphics card/driver notification** on or off and optionally lock the setting. Click **Tools** > **Options** > **System Options** > **Messages/Errors/Warnings**.

> This option does not affect graphics check results, which are always written to the performance log.

4

SOLIDWORKS Fundamentals

This chapter includes the following topics:

- **Changes to System Options and Document Properties**
- **Application Programming Interface**
- **Quick Copy Setting**

Changes to System Options and Document Properties

The following options are new, changed, or removed in the software.

System Options

Option	Description	Access
Include Detailing Mode Data when saving	Removed from System Options.	**Drawings** > **Performance**
IFC	Imports `IFC` files into SOLIDWORKS. You can also specify `IFC` entities to import from the `IFC` files by defining filters in System Options.	**Import**
Include drawings paper color	Includes the paper color when you save drawings as image and **PDF** file formats.	• **Export** > **TIF/PSD/JPG/PNG**. Available when you select **Use specified color for drawings paper color (disable image in sheet background)** in **System Options** > **Colors**. • **Export** > **PDF**.

Option	Description	Access
Enable measure	Renamed from **Okay to measure this eDrawings file**.	**Export** > **EDRW/EPRT/EASM**
Allow STL export	Renamed from **Allow export to STL for parts & assemblies**.	**Export** > **EDRW/EPRT/EASM**
Save table features	Renamed from **Save table features to eDrawings file**.	**Export** > **EDRW/EPRT/EASM**
Save file properties	Saves custom properties from a SOLIDWORKS document in the resulting eDrawings® files when you **Save As** an eDrawings document or **Publish to eDrawings** in SOLIDWORKS. When selected, you can specify **Save file properties for each component in the assembly**. This option saves custom properties including configuration-specific properties for each component in the SOLIDWORKS assembly.	**Export** > **EDRW/EPRT/EASM**
Save shaded data	Renamed from **Save shaded data in drawings**.	**Export** > **EDRW/EPRT/EASM**
Save Motion Studies	Renamed from **Save Motion Studies to eDrawings file**.	**Export** > **EDRW/EPRT/EASM**
Dismissed Messages	Moved to a separate tab under **Messages / Errors / Warnings**.	**Messages / Errors / Warnings** > **Dismissed Messages**
Remind me when a document has not been saved for *n* minutes	Renamed from **Show reminder if document has not been saved for *n* minutes**. Moved from **Backup/Recover** to **Messages / Errors / Warnings**.	**Messages / Errors / Warnings**
Automatically dismiss notifications after *n* seconds	Renamed from **Automatically dismiss after *n* seconds**. Moved from **Backup/Recover** to **Messages / Errors / Warnings**	**Messages / Errors / Warnings**
When rebuild error occurs	Moved from **General** to **Messages / Errors / Warnings**.	**Messages / Errors / Warnings**

Option	Description	Access
Hide graphics card/driver notifications	Hides notifications about graphics card and driver certification status.	**Messages / Errors / Warnings**
Load components lightweight	Renamed from **Automatically load components lightweight**.	**Performance**

Document Properties

Option	Description	Access
Combine same length cut list items with different profiles (pre-2019 behavior)	Renamed from **Combine cutlist items in BOM regardless to profile when lengths are changed to be the same (legacy behavior)**.	**Tables** > **Bill of Materials**
Save model data	Saves all drawings with model data to use in Detailing mode.	**Performance**
Include standard views in View Palette	Creates standard views (such as front, back, top) when you add drawing views from the View Palette.	**Performance**

Application Programming Interface

See *SOLIDWORKS API Help: Release Notes* for late-breaking updates.

Support

There is API support for:

- SOLIDWORKS Inspection Add-In
- SOLIDWORKS PDM Professional Web
- Belt/chain assembly features
- Structure systems
- Graphics mesh and mesh BREP bodies

Redesign

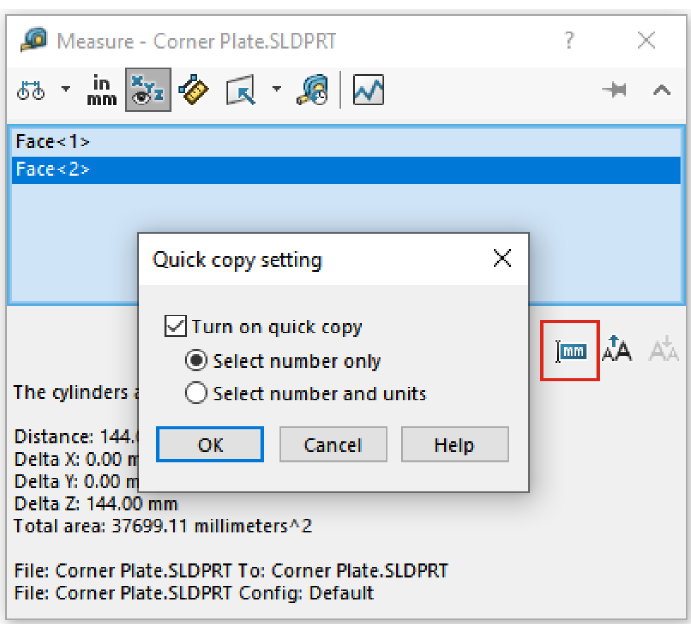
The following have been redesigned:

- Sheet metal base flange and corner relief features
- Messages and alerts for add-ins

Other Enhancements

- Replace a sketch entity with construction or contour geometry
- Get all decals applied to a component face in an assembly
- Get and set drawing sheet zone parameters
- Get angular ordinate dimension information from the current drawing sheet or view
- Restore default values of Hole Wizard hole or slot feature data
- Upgrade legacy custom properties
- Create a coordinate system based on position and orientation relative to the global coordinate system
- Add doubled distance dimensions and toggle between single and doubled distance dimensions
- Get and set whether to use the properties of the material applied when creating a new sheet metal feature
- For SOLIDWORKS Connected:
  - Add new Physical Products and Representations (configurations) to SOLIDWORKS Connected models
  - Convert between parent and derived configurations
  - Get and set whether a Representation configuration is shared
  - Add and replace assembly components from a **3D**EXPERIENCE® collaborative space

Quick Copy Setting



The **Quick Copy Setting** functionality in the **Measure** tool is re-enabled. You can use it to copy a value in the Measure dialog box.

To use Quick Copy Setting:

1. Click **Measure** (Tools toolbar).

2. In the Measure dialog box, click **Quick Copy Setting** .
3. In the Quick copy setting dialog box, specify options:
   - **Turn on quick copy**. Enables the **Quick Copy Setting** functionality in the Measure dialog box.
   - **Select number only**. Highlights the numerical value in the measurement text when you hover over it in the Measure dialog box.
   - **Select number and units**. Highlights the numerical value and units in the measurement text when you hover over it in the Measure dialog box.
4. Click **OK**.

   The next time you measure an entity, you can select **Click to copy** when you hover over a value in the Measure dialog box.

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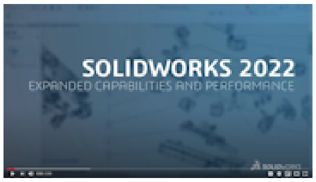
User Interface

This chapter includes the following topics:



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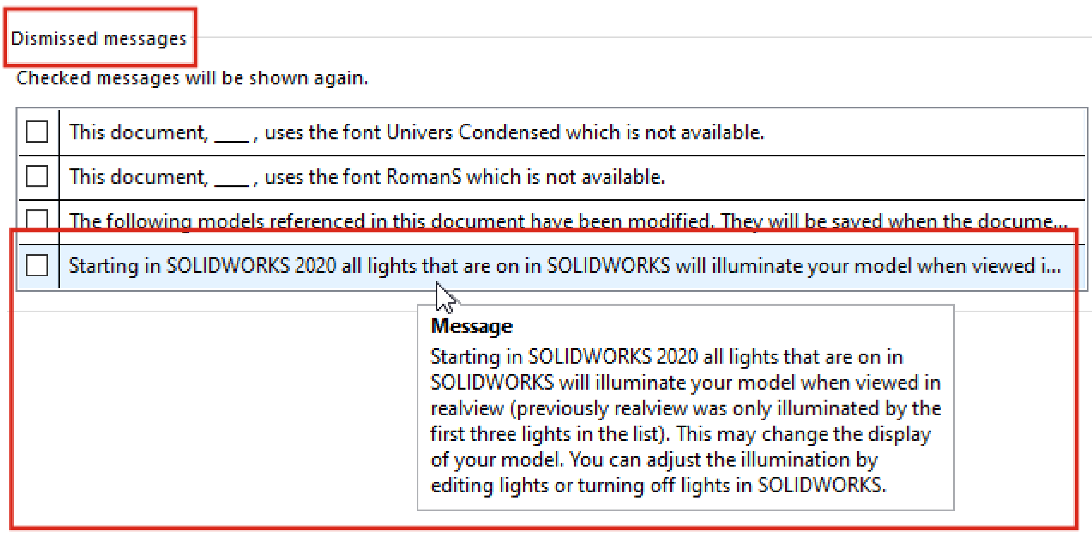
Dismissed Messages

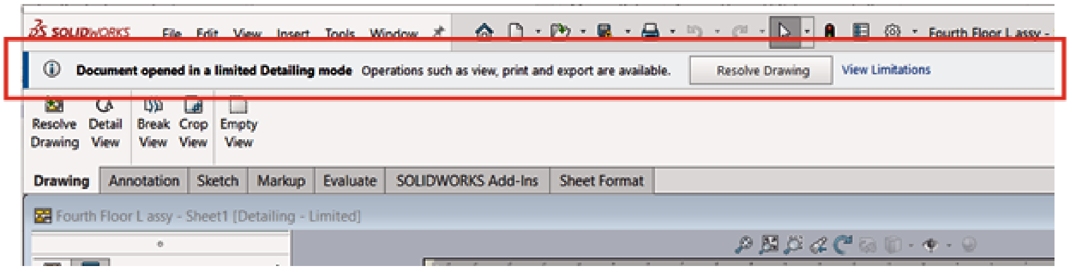


You can view the full information about dismissed messages on a separate tab in the System Options dialog box. Previously, dismissed messages were truncated and harder to find.

To view dismissed messages, click **Tools** > **Options** > **System Options** > **Messages/Errors/Warnings** > **Dismissed Messages**.

In the **Dismissed Messages** table, hover over a truncated message to view the full text.

Message Bar



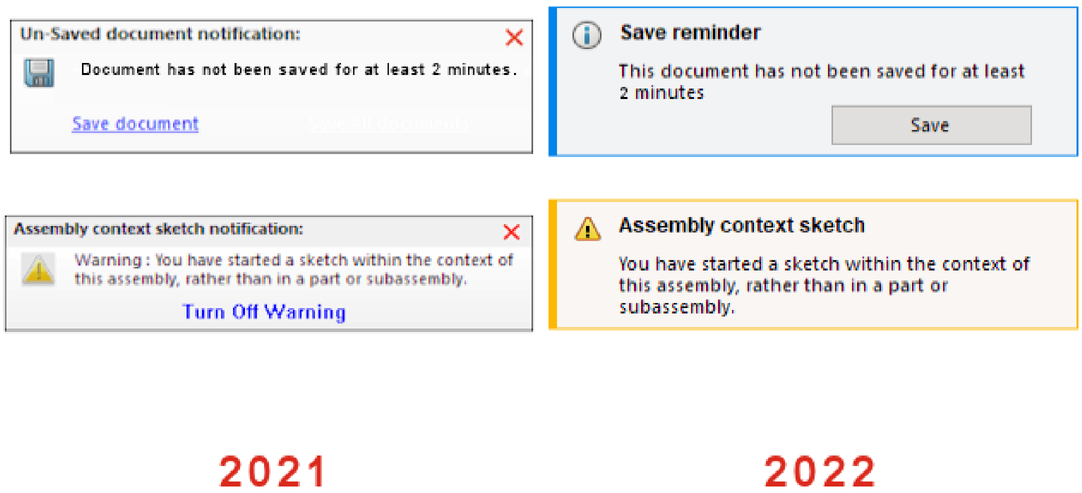
A Message Bar is a modeless bar at the top of the main SOLIDWORKS window that provides information specific to the active document.

There are four types of Message Bars:

- Information
- Acknowledgment
- Warning
- Error

A Message Bar can open in several situations. It is usually a warning about a document state, for example, that your changes will not be saved because you did not check out the document. A Message Bar never auto-dismisses - you must manually dismiss it.

Redesigned Notifications

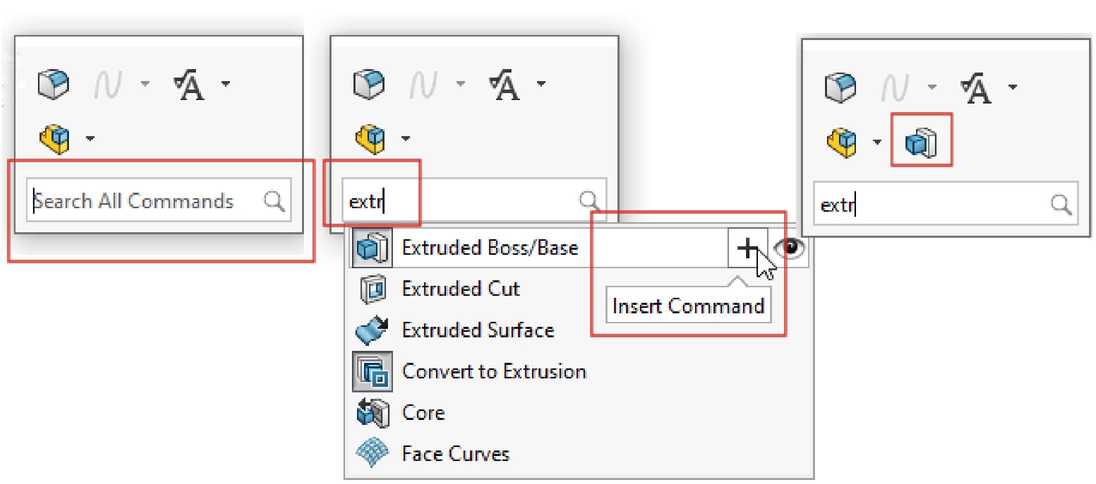


Notifications are redesigned so they are more intuitive and use a consistent layout. This includes informational, acknowledgement, warning, and error messages.

Notifications appear for 5 seconds by default, then disappear. Hover over them to keep them visible. In **Tools** > **Options** > **System Options** > **Messages/Errors/Warnings**, the layout is modified to support the redesign. For additional information about notifications, see **Controlling Display of Graphics Warnings in the Settings Administrator Tool** on page 20.

Option	Description
Notifications: • **Remind me when a document has not been saved for *X* minutes** • **Automatically dismiss notifications after *X* seconds**	Replaces options in **Tools** > **Options** > **System Options** > **Backup/Recover**.
Rebuild Errors > **When rebuild error occurs**	Moved from **Tools** > **Options** > **System Options** > **General**
System Notifications area	New

Shortcut Bar and Command Search



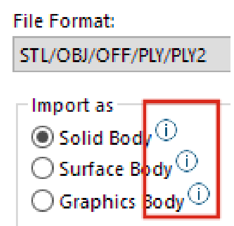
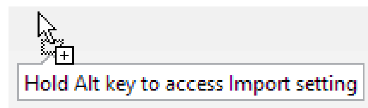
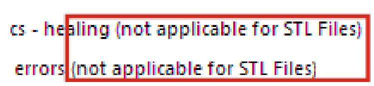
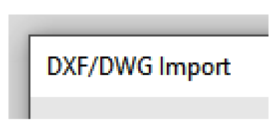
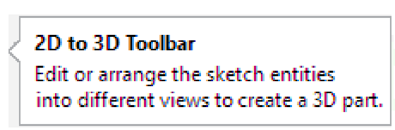
Search All Commands appears in the shortcut bar by default so you can quickly search for tools from the pointer location and add them to the shortcut bar. You can customize shortcuts directly from the shortcut bar.

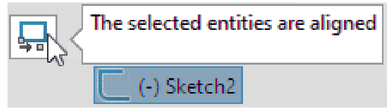
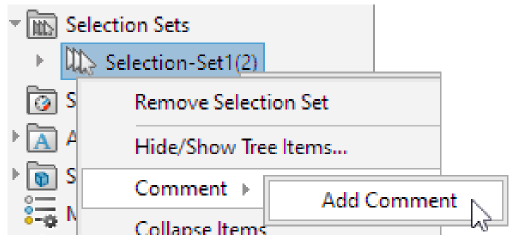
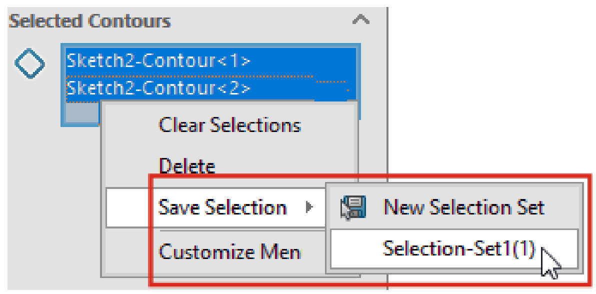
Press `S` to open the shortcut bar. Type in **Search All Commands**, select a tool, and click **Insert Command** + to add it to the shortcut bar.

To customize the shortcut bar, right-click it and click **Customize** to open the Customize dialog box to the Shortcut Bar tab. Then drag tools from the tab to the shortcut bar. To control the display of the Command Search, select or clear **Show Command Search in the shortcut bars**.

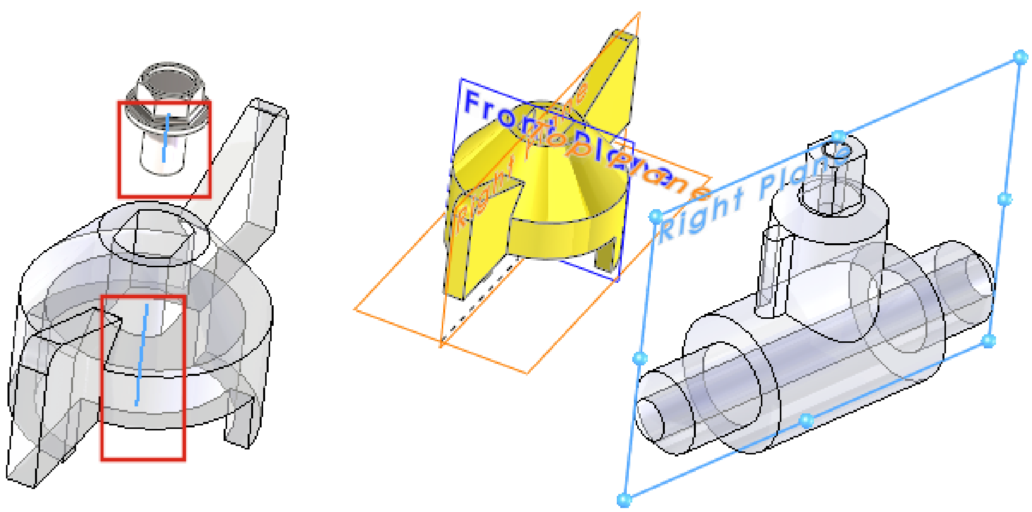
General Usability

The user interface is enhanced to improve productivity.

Enhancement	User Interface
In **Tools** > **Options** > **System Options** > **Import**, tooltips provide more information. Hover over ⓘ to display the information.	
When you drag a non-SOLIDWORKS file into a part file, a persistent tooltip prompts you to hold the **Alt** key to access the Import Options dialog box for the file type. Drop the file to open the dialog box.	
When you import mesh models with the **File Format** of **STL/OBJ/OFF/PLY/PLY2**, the appropriate CommandManager tab opens. If you import mesh models as solid bodies or surface bodies, the Data Migration CommandManager tab appears. If you import mesh files as graphics bodies, the Mesh Modeling CommandManager tab appears. If the tabs do not automatically appear, a notification prompts you to use the appropriate tab to edit the imported model.	
In **Tools** > **Options** > **System Options** > **Import**, text alerts you that the options **Automatically perform import diagnostics - healing** and **Perform full entity check and repair errors** do not apply to STL files.	
The DXF/DWG Import dialog box displays the options in a more visible location and has an improved user interface.	
When you import a drawing into a part file, a tooltip points out the 2D to 3D toolbar. The tooltip disappears when you click in the graphics area.	

Enhancement	User Interface
On the Align toolbar, a tooltip indicates when selected entities are aligned. Previously, there was no indication. This helps you align 2D files that you import and use as references to create 3D geometry.	
You can add comments to selection sets and they appear in the **Comments** folder in the FeatureManager® design tree.	
In group boxes of PropertyManagers, you can select entities, right-click, and save them to selection sets.	

Reference Geometry



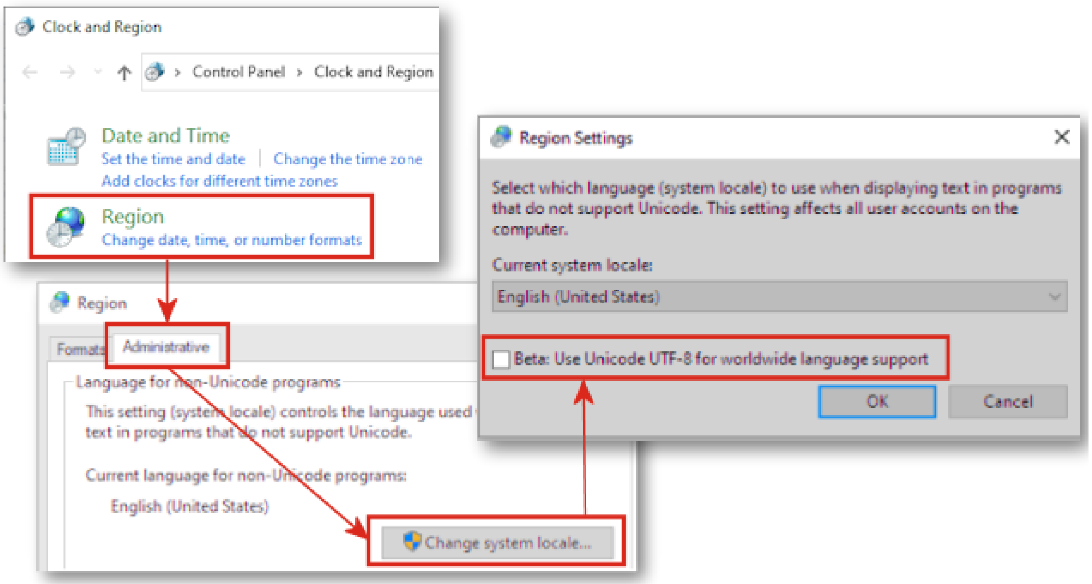
Hover for axes Press Q for planes

Reference geometry has improved usability that helps you select axes and planes directly in the graphics area. This is useful for commands such as **Mate**, **Measure**, or patterns.

Hover over temporary axes of cylindrical faces and surfaces to display the axes. Hover over faces, then press **Q** to display reference planes. Select multiple reference geometry entities by pressing the **Shift** or **Ctrl** keys. After you select reference geometry, SOLIDWORKS automatically dismisses all unneeded reference geometry.

You can right-click a component in the graphics area, click **Reference Geometry Display**, then show primary planes, reference planes, reference axes, or the coordinate system. Previously, these options were available only from the FeatureManager design tree.

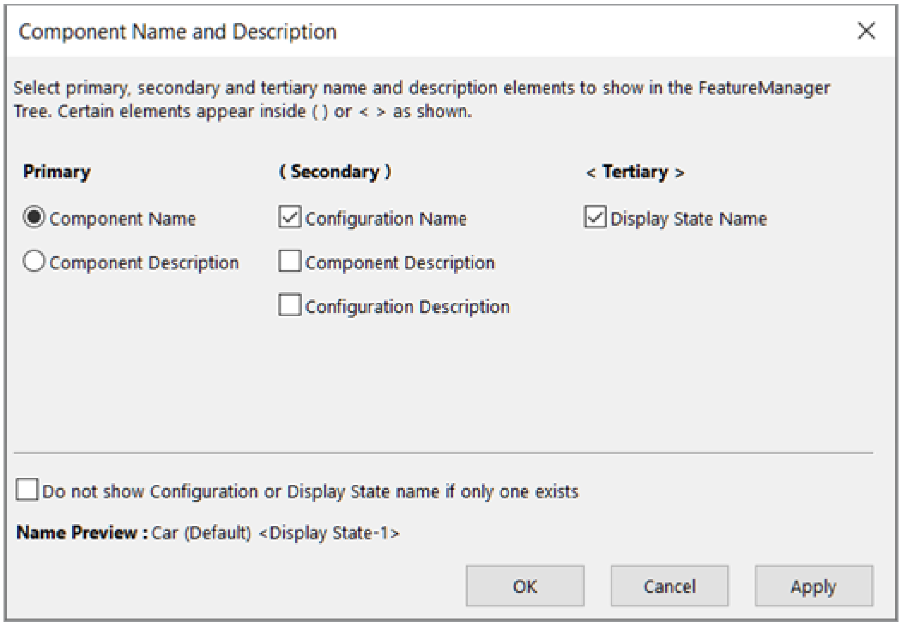
Junk Characters in Beta Unicode View Resolved



In Windows 10 Version 1803 or later, if you selected the **Beta: Use Unicode UTF-8 for worldwide language support** option, many user interface elements in SOLIDWORKS displayed garbled text. Most of these issues have been fixed in SOLIDWORKS® 2022.

For example, in the **Tools** > **Options** dialog box, text in several drop-down lists displayed incorrect characters. This was a problem seen in almost all languages but it affected Asian languages more severely.

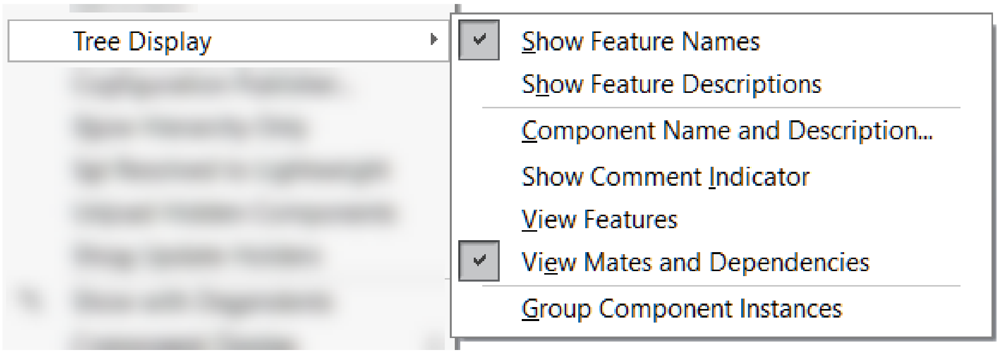
Component Name and Description



You can use the Component Name and Description dialog box to specify display options for the FeatureManager design tree.

The dialog box contains these options:

Primary	• **Component Name** • **Component Description**
(Secondary)	• **Configuration Name** • **Component Description** • **Configuration Description**
< Tertiary >	• **Display State Name**
Do not show Configuration or Display State name if only one exists	Suppresses the configuration and display state names when there is only one configuration.
Name Preview	Displays the component name based on the selected options.



To access these options, right-click the part or assembly name in the FeatureManager design tree and click **Tree Display** > **Component Name and Description**.

Under **Tree Display**, **Component Name and Description** replaces:

- **Do not show Configuration/Display State Names if only one exists**
- **Show Component Names**
- **Show Component Description**
- **Show Component Configuration Names**
- **Show Component Configuration Descriptions**
- **Show Display State Names**

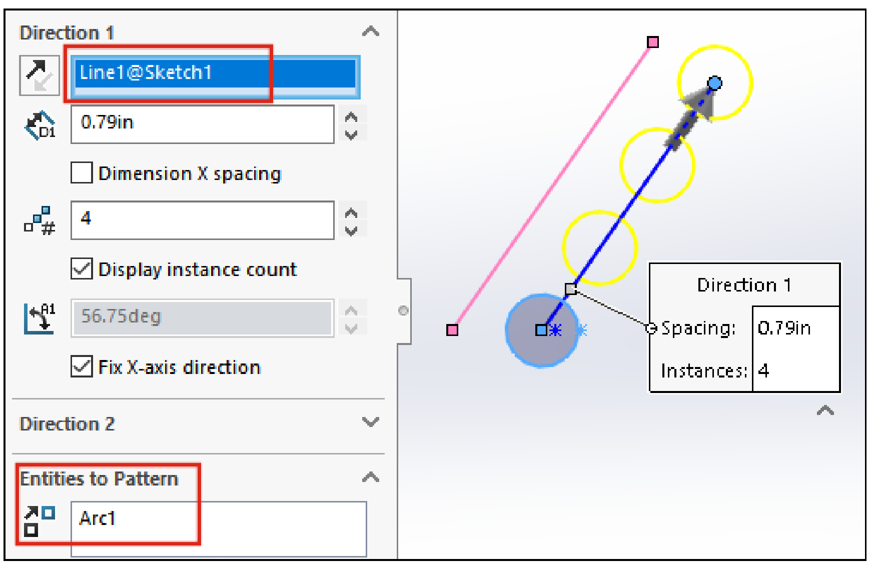
6

Sketching

This chapter includes the following topics:

- **Linear Sketch Entity as Direction Reference**
- **Pattern and Copy Text in Sketches**

Linear Sketch Entity as Direction Reference



For direction reference in a linear sketch pattern, you can select a line from the same sketch that contains the entities to pattern. Previously, the selected line became a part of the entities to pattern rather than the direction reference.

- You cannot select a line from a block for direction reference.
- You cannot select and pattern the line that you use for a direction reference.

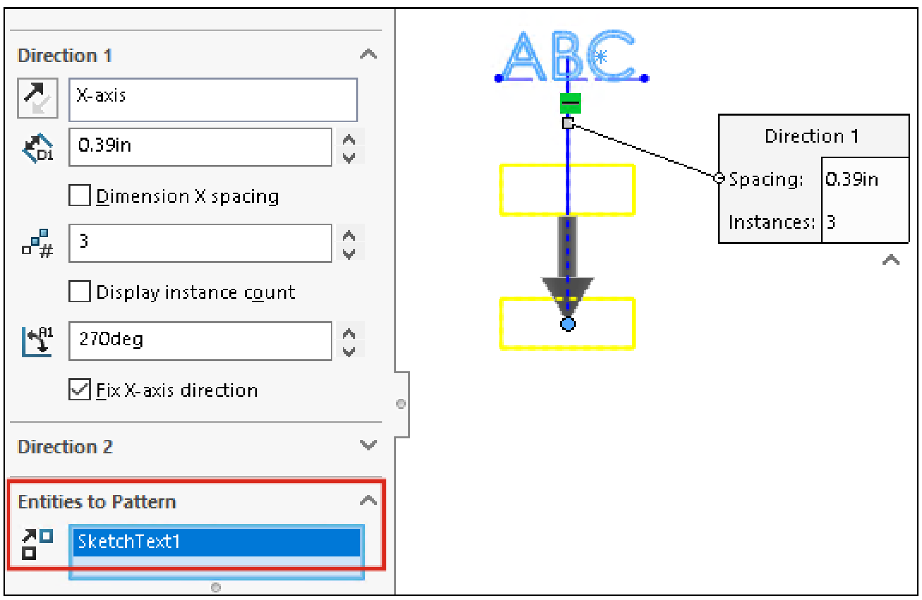
Selecting Linear Sketch Entities for Direction References

To select linear sketch entities for direction references:

1. Select a plane or a face to sketch on.

2. Sketch a line for direction reference.
3. Sketch the entities to pattern.
4. Click **Linear Sketch Pattern** (Sketch toolbar) or **Tools** > **Sketch Tools** > **Linear Pattern**.
5. In the PropertyManager, under **Entities to Pattern**, select the sketch entities to pattern .
6. Select the line to define **Direction 1**.
7. Define **Direction 2**.
8. Click .

Pattern and Copy Text in Sketches



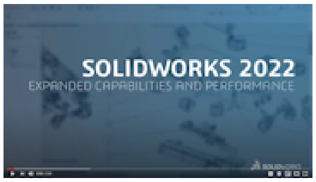
In a linear sketch pattern, you can select text as an entity to pattern. You can use **Copy Entities** to copy text.

7

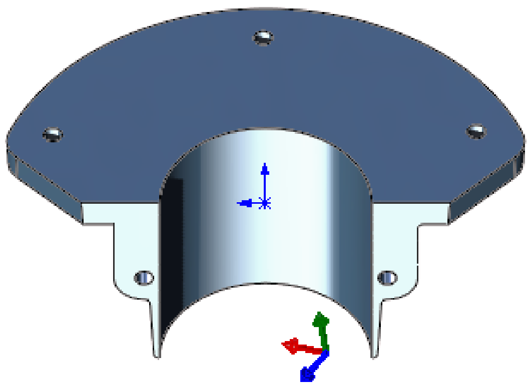
Parts and Features

This chapter includes the following topics:

- **Coordinate Systems**
- **Cosmetic Threads**
- **Draft Across Parting Lines**
- **External Threaded Stud Wizard**
- **Hole Wizard Slots**
- **Hybrid Modeling**
- **Mirroring About Two Planes**
- **Rotating a Section View About a Hole or Axis**
- **Thickness Analysis Resolution**
- **Redo Support for Parts**

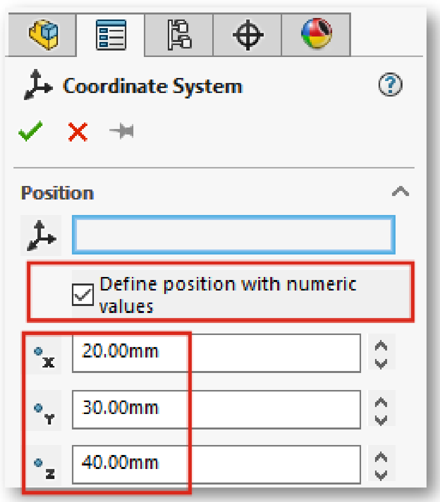
	**Video: What's New in SOLIDWORKS 2022 - Parts**

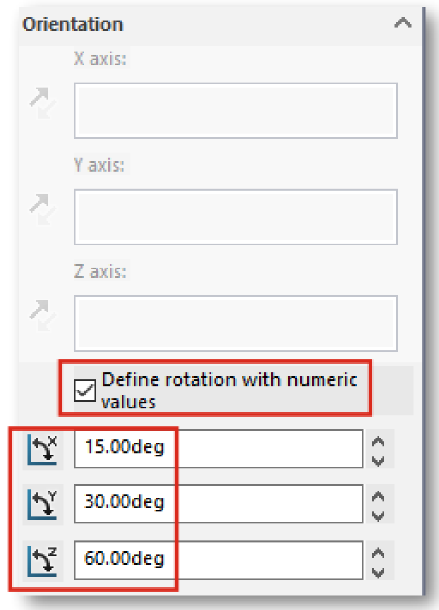
Coordinate Systems



Coordinate systems have improved in how you define and select them.

Using Numeric Values to Define Coordinate Systems

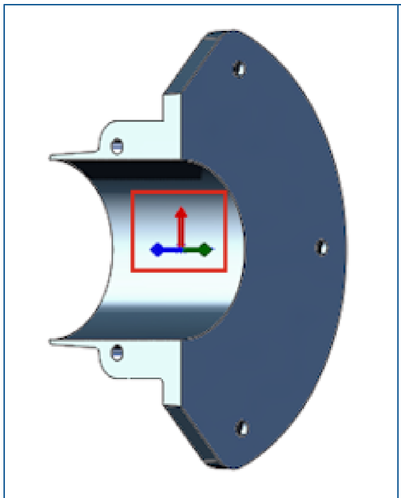
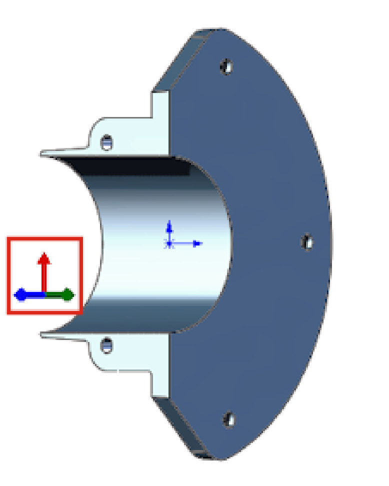
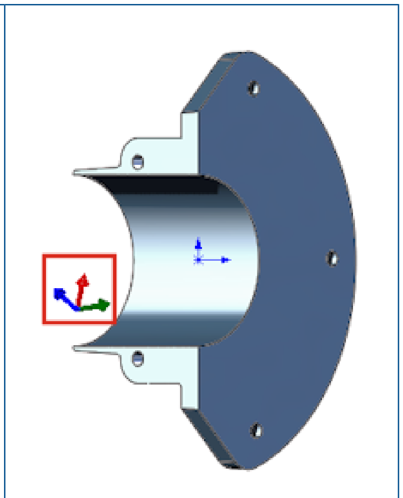




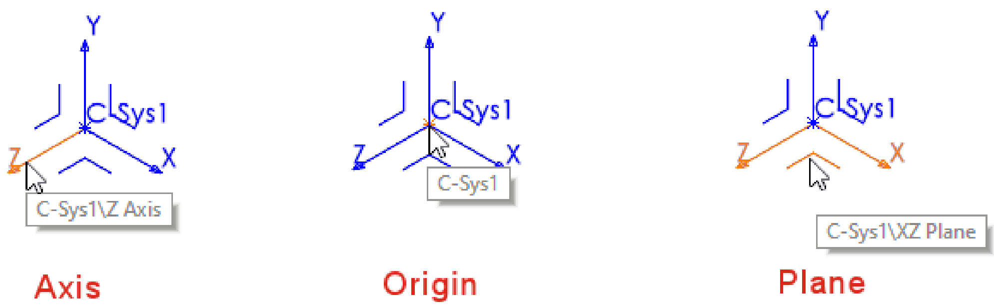
In parts and assemblies, you can define coordinate systems by entering absolute numeric values for position and orientation.

In the Coordinate System PropertyManager, under **Position**, select **Define position with numeric values** and enter numeric values for the X, Y, and Z coordinates (X, Y, and Z). The values define the position relative to the local origin, not the global origin (0, 0, 0).

To rotate the coordinate system, under **Orientation**, select **Define rotation with numeric values** and enter numeric values for at least one axis. The axes always rotate in the sequence X, then Y, then Z.

		
Original	Position defined	Position and rotation defined

Selection for Coordinate Systems

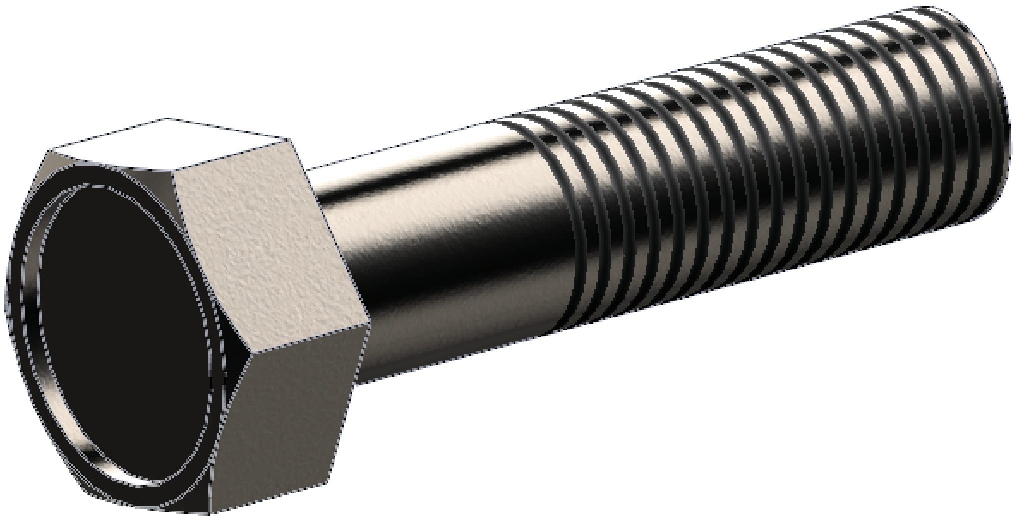


You can reference all parts of coordinate systems, such as planes, axes, and origin, in downstream features. For example, you can sketch on the **XY** plane of a coordinate system or use the **Z** axis of a coordinate system as the axis for a revolve feature.

In a part or assembly, click **Insert** > **Reference Geometry** > **Coordinate System** and create the coordinate system. Hover over a plane, axis, or the origin to highlight each entity. This functionality is useful for actions such as sketching, mating, and more.

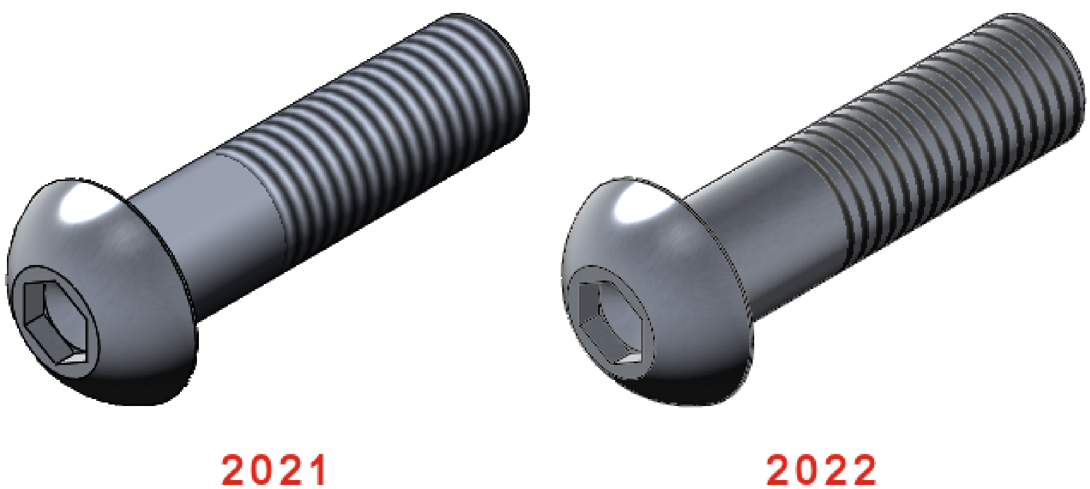
> To see this functionality, show coordinate systems. Click **View** > **Hide/Show** > **Coordinate Systems**.

Cosmetic Threads



Cosmetic threads have improved in appearance and ease of use.

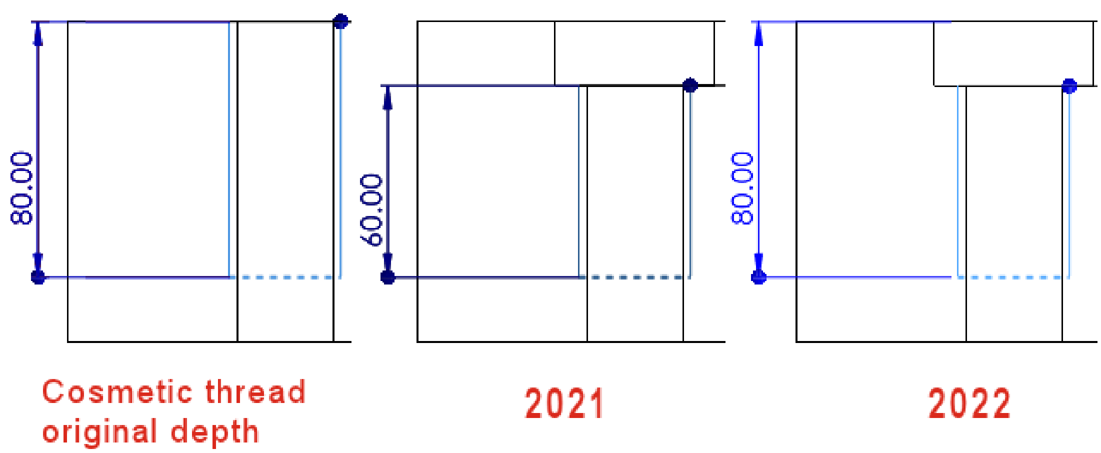
Appearances and Textures



When you apply a cosmetic thread to a part, any underlying appearances or textures on the part appear between the cosmetic threads.

The cosmetic threads that you apply in SOLIDWORKS are also supported in SOLIDWORKS Visualize. See **Cosmetic Threads** on page 139.

Depth and Feature Ownership

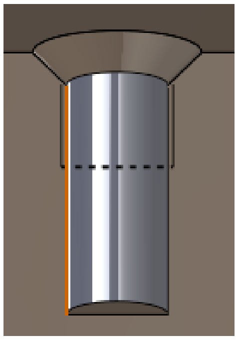


Cosmetic thread behavior is modified to improve usability.

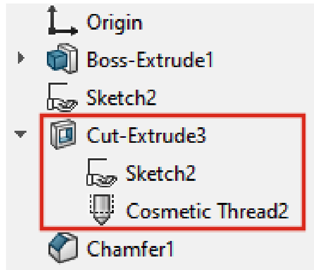
> This functionality applies to new parts created in SOLIDWORKS 2022 and later. To apply this functionality to legacy parts and upgrade them, right-click the part node in the FeatureManager® design tree and select **Upgrade cosmetic thread features**.
>
> For part templates created in SOLIDWORKS 2022 and later, you can retain the legacy functionality for **Depth** and feature ownership. In part templates, before you add cosmetic threads, click **Tools** > **Options** > **Document Properties** > **Drafting Standard** > **Annotations** and clear **Apply new cosmetic thread behavior to new parts**. This option is selected by default for new part templates and cleared for legacy part templates. This option is enabled for new part templates only; it is disabled for part documents.
>
> If you use the **Insert** > **Mirror Part** command, the mirrored part inherits the cosmetic thread behavior from the base part. For example, if the base part is created in SOLIDWORKS 2021, the mirrored part inherits the legacy behavior for cosmetic threads from the base part.

SOLIDWORKS measures **Depth** from the original location of an edge regardless of changes made by downstream features that relocate that edge. In the image above, the original thread depth is 80 from the edge of the cut extrude. If you add a second cut extrude that relocates that edge, the cosmetic thread retains the original thread depth of 80 mm.

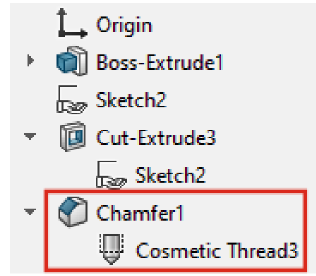
Cosmetic threads are owned by the latest feature in the FeatureManager design tree whose face shares the common edge selected for attaching the thread. In the image below, you create the cut extrude, add a chamfer, then add a cosmetic thread from the chamfer edge. In SOLIDWORKS® 2022, the chamfer feature owns the cosmetic thread because its face shares the common edge with the cosmetic thread.



Cut extrude + chamfer + cosmetic thread

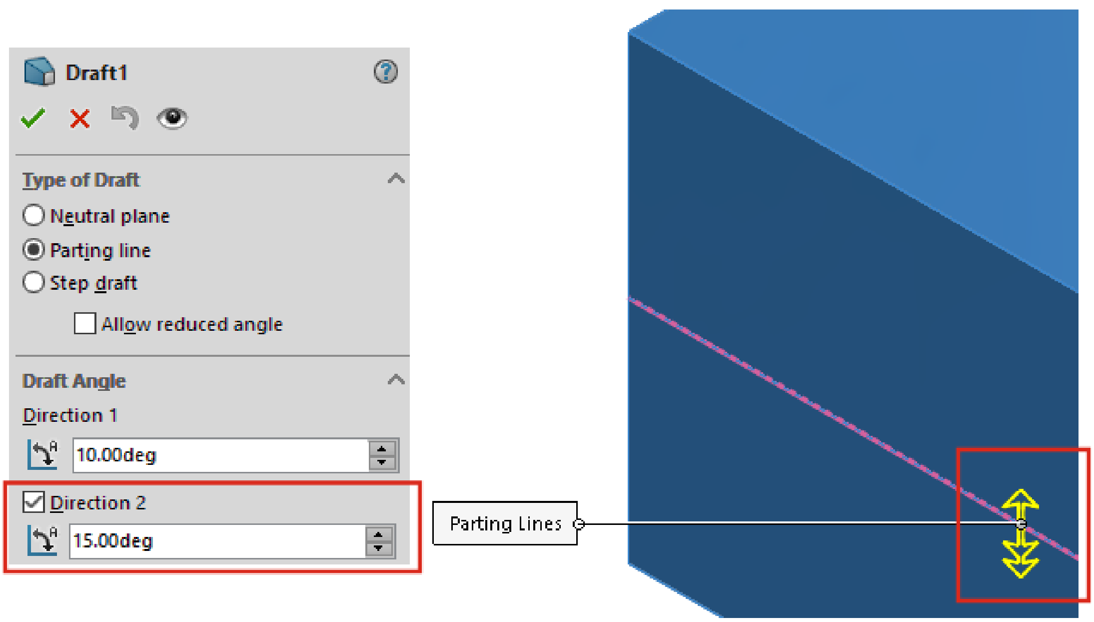


2021



2022

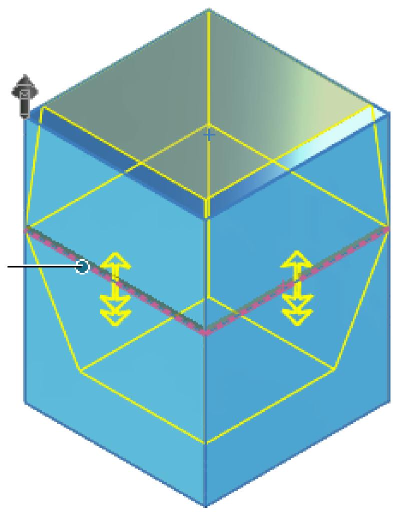
Draft Across Parting Lines



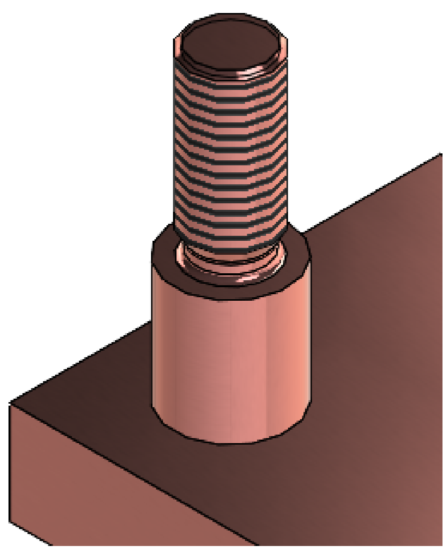
You can create draft on both sides of parting lines at the same time from the PropertyManager. Previously, this process required multiple drafts.

In the Draft PropertyManager, under **Draft Angle**, select **Direction 2** and specify the draft angles. To use the same draft angle in both directions, select **Symmetrical Draft**.

Under **Parting Lines**, select the geometry. Select **Show preview** to display a detailed preview of the draft.



External Threaded Stud Wizard



You can use the Stud Wizard to create external threaded stud features. This tool works similar to the Hole Wizard. You define the stud parameters then position the studs on the model. You can also apply thread parameters to existing circular studs.

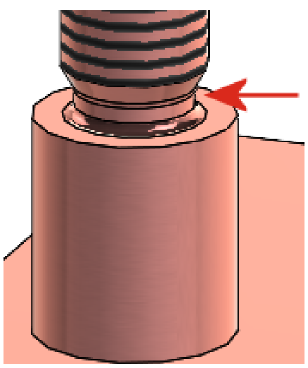
To create stud features, click **Stud Wizard** (Features toolbar) or **Insert** > **Features** > **Stud Wizard** . To view the threads, click **Tools** > **Options** > **Document Properties** > **Detailing** and under **Display Filter**, select **Cosmetic threads** or **Shaded cosmetic threads**.

Creating External Threaded Studs

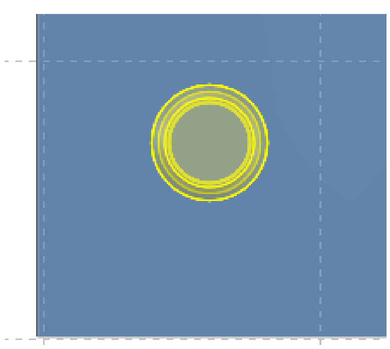
In this example, you create a new stud on a planar surface. You can also create a stud on an existing cylindrical stud that you can then modify.

To create external threaded studs:

1. In a part, click **Stud Wizard** (Features toolbar) or **Insert** > **Features** > **Stud Wizard**.
2. In the PropertyManager, click **Creates Stud on a Surface**.
3. On the **Stud** tab:
   a) Under **Shaft Length**, specify a value.
   b) Under **Standard**, select a standard, the **Type** of thread, and the thread **Size**. The **Major Diameter** value is based on the **Size**.
   c) Under **Thread**, for **End Condition**, select **Blind** and specify the **Thread Depth**.
   d) To add an external **Thread class**, select the check box and a class. You can access this information in drawings.
   e) To add an **Undercut**, select the check box and specify the **Undercut diameter**, **Undercut depth**, and **Undercut radius**.

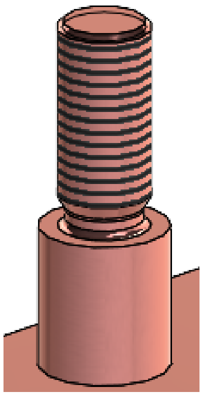


4. On the **Position** tab, select the face to position the stud.



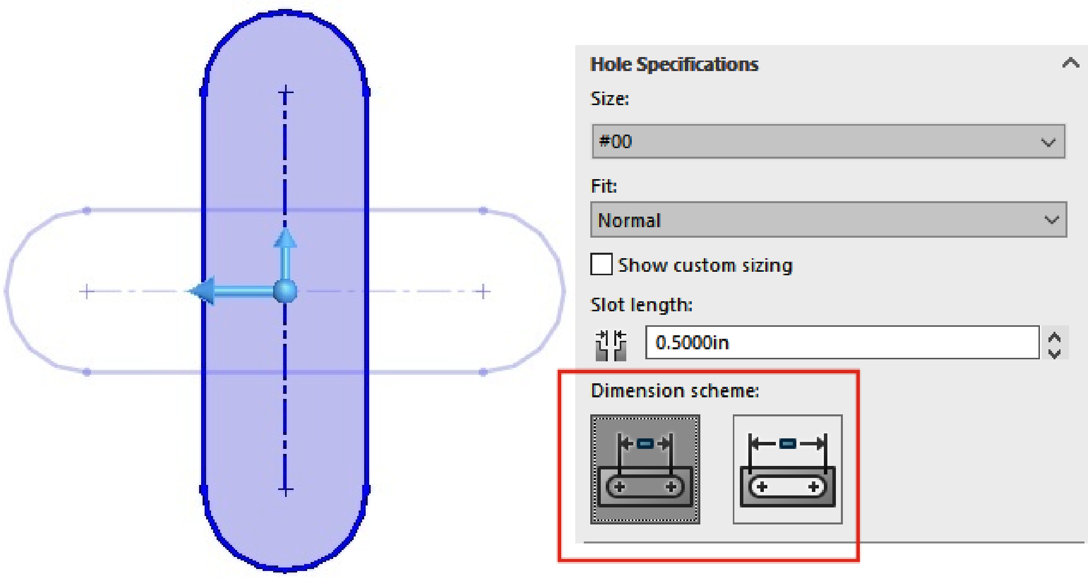
You can position only one stud per feature.

5. Click ✔.



> To view the threads, click **Tools** > **Options** > **Document Properties** > **Detailing** and under **Display Filter**, select **Cosmetic threads** or **Shaded cosmetic threads**.

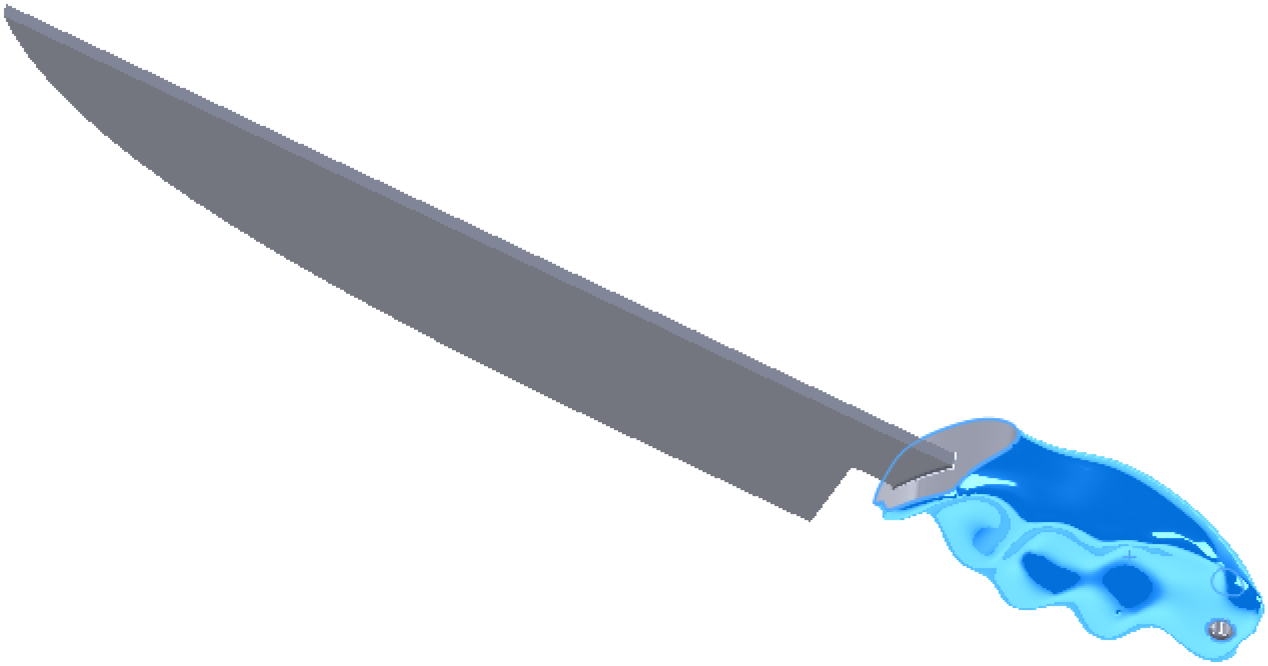
Hole Wizard Slots



When you create and position Hole Wizard slots, press **Tab** to rotate the orientation 90° clockwise. You can dimension the slot length using the arc centers.

Previously, you could only drag slots to reposition them and dimension them only with end-to-end dimensions.

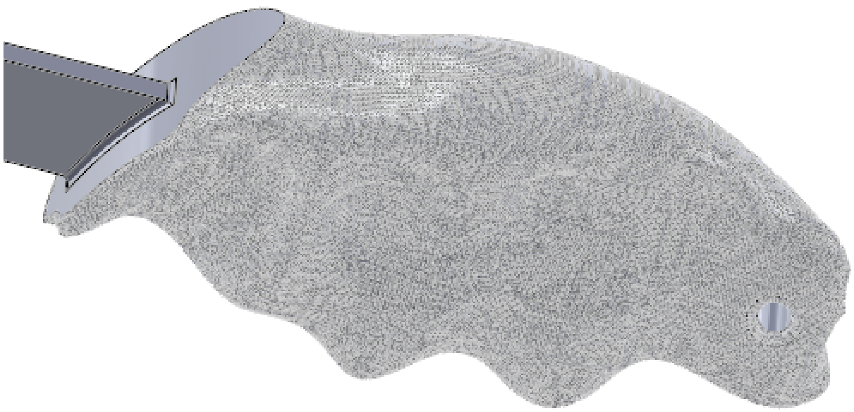
Hybrid Modeling



You can create a hybrid solid or surface body that includes mesh BREP geometry and standard SOLIDWORKS BREP geometry. Previously, you could not combine mesh BREP and standard SOLIDWORKS BREP geometries in a single body.

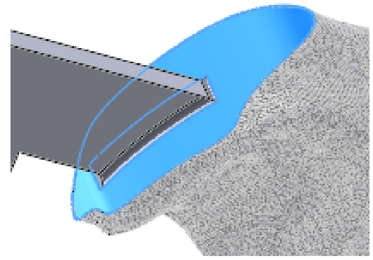
In this example, do the following:

1. Mold the custom grip in a piece of clay and scan it to create an `.stl` file.

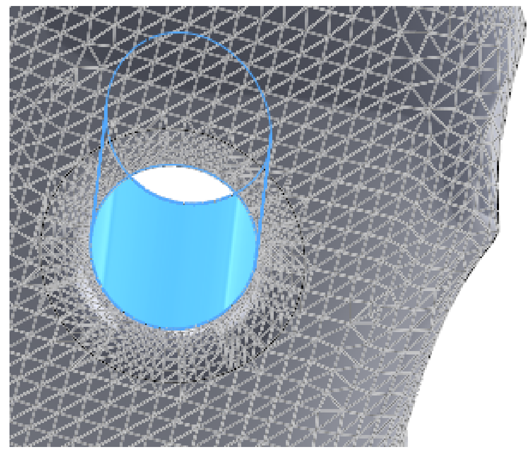


2. Import the `.stl` file as a graphics body and convert it to mesh BREP.
3. Add standard SOLIDWORKS BREP features to the part.

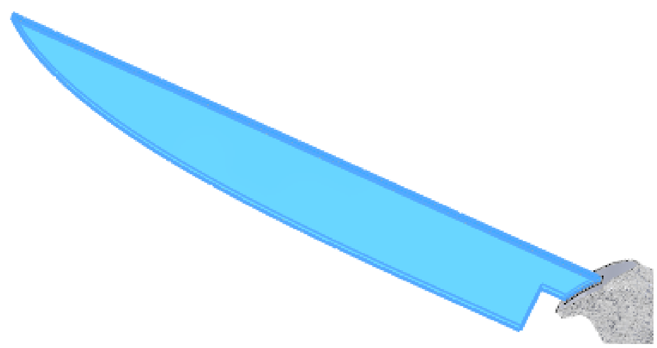
Surface extrude and surface cut



Cut extrude and fillet

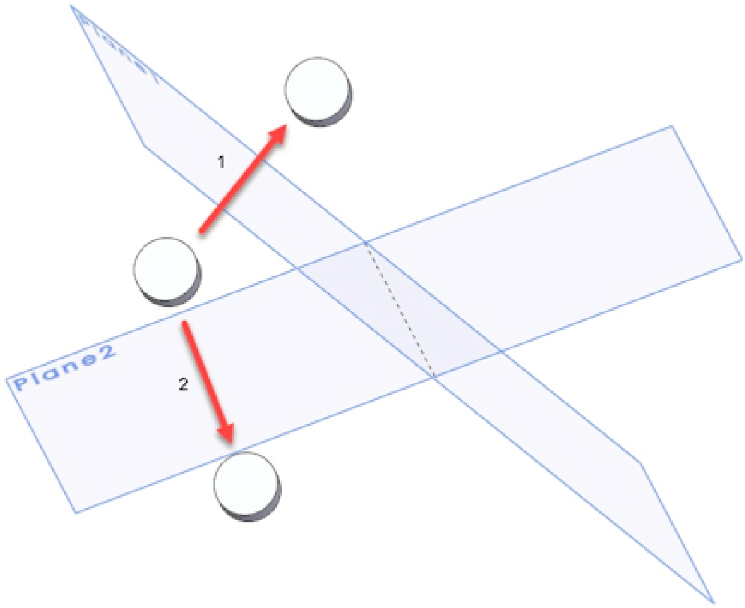


Boss extrude



Because the blade is standard SOLIDWORKS BREP geometry, you could refine it with additional features. For example, you can sharpen the blade, add a serrated edge, or add back blade features.

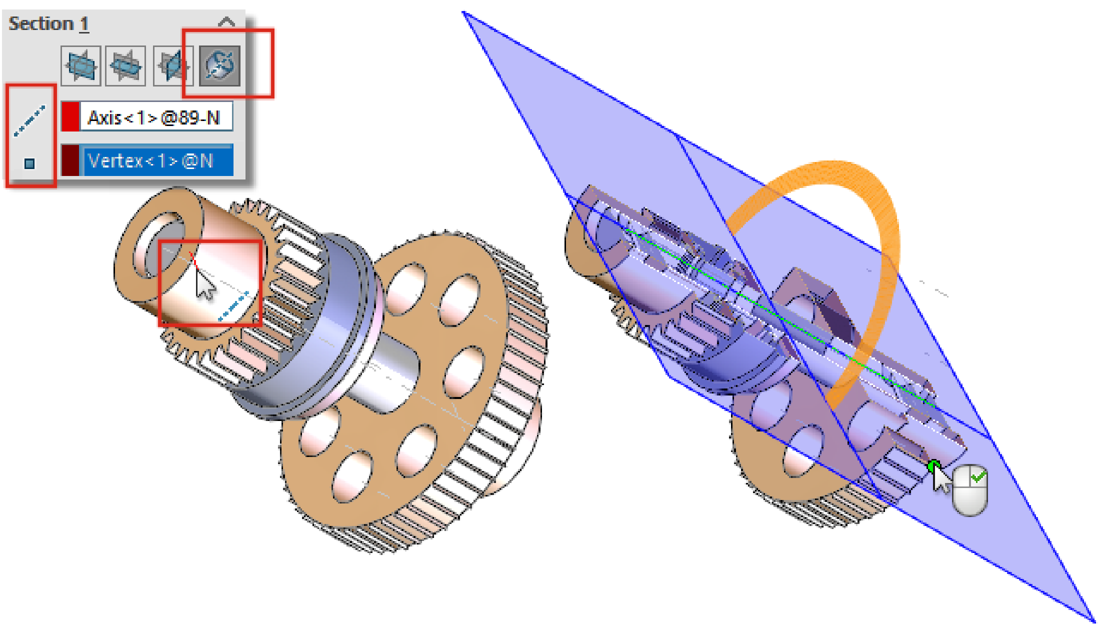
Mirroring About Two Planes



You can mirror about two planes at once. Previously, you needed to create multiple features to achieve this.

In the Mirror PropertyManager, select a second plane in **Secondary Mirror Face/Plane**.

Rotating a Section View About a Hole or Axis

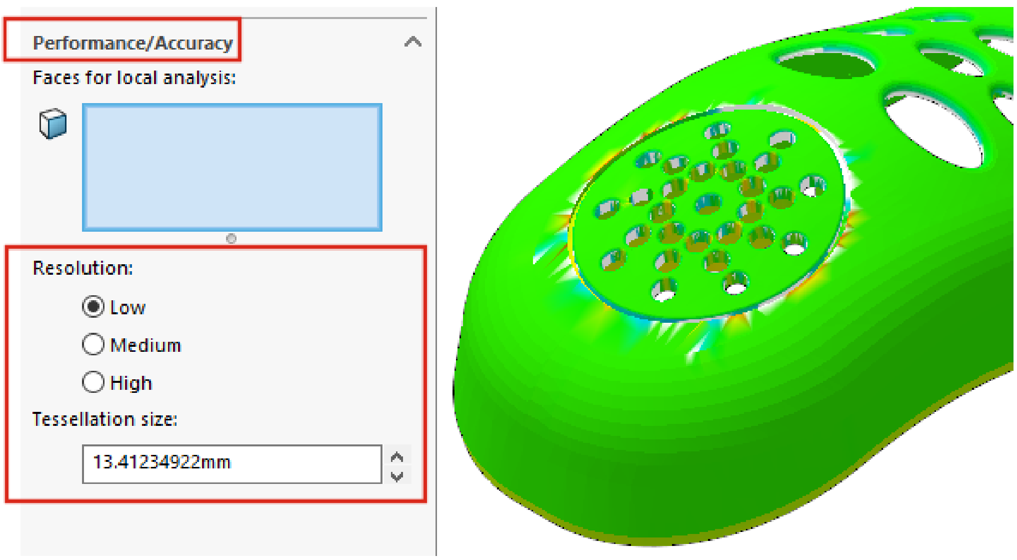


In parts and assemblies, you can rotate a section view plane about an axis, temporary axis, hole, or cylindrical face.

In the Section View PropertyManager, under **Section**, click **Section About Axis**.

Select an axis, hole, or cylindrical face plus a point or vertex to create the section view plane.

Thickness Analysis Resolution

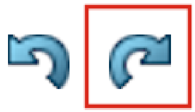


To optimize the results of a thickness analysis, you can specify the resolution, regardless of the model size. Previously, the resolution used depended on the model size.

Enable the SOLIDWORKS Utilities add-in. Click **Tools** > **Thickness Analysis**. In the PropertyManager, under **Performance/Accuracy**, for **Resolution**, select **Low**, **Medium**, or **High**. Under **Tesselation size**, the value updates to reflect the suggested values. To customize the resolution, enter a custom value. Consider using custom values for models that have a large bounding box or to define a specific resolution.

The custom **Tesselation size** value cannot exceed the suggested value for the **Low** resolution.

Redo Support for Parts

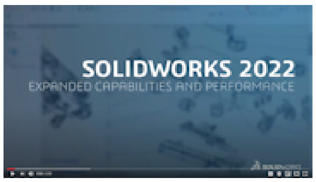


Redo support is extended to more commands and actions.

- Inserting and editing of features:
  - **Hole Wizard**
  - **Simple Hole**
  - **Linear Pattern**
- Commands and actions:
  - **Instant2D**
  - Reorder features
  - Rollback

8

Model Display



Video: What's New in SOLIDWORKS 2022 - Graphics

Model Display Performance Improvements

SOLIDWORKS 2022® offers improved performance for 3D textures and silhouette edges.

Functionality	Performance Improvement
3D textures	3D textures accelerate the process of refining the mesh to better match the detail in the textural appearance image. You can further refine the **Maximum Element Size** below its previous limit.
Silhouette edges	Performance is improved for rendering silhouette edges in dynamic mode. You can see the silhouette edges in **Shaded With Edges** mode.

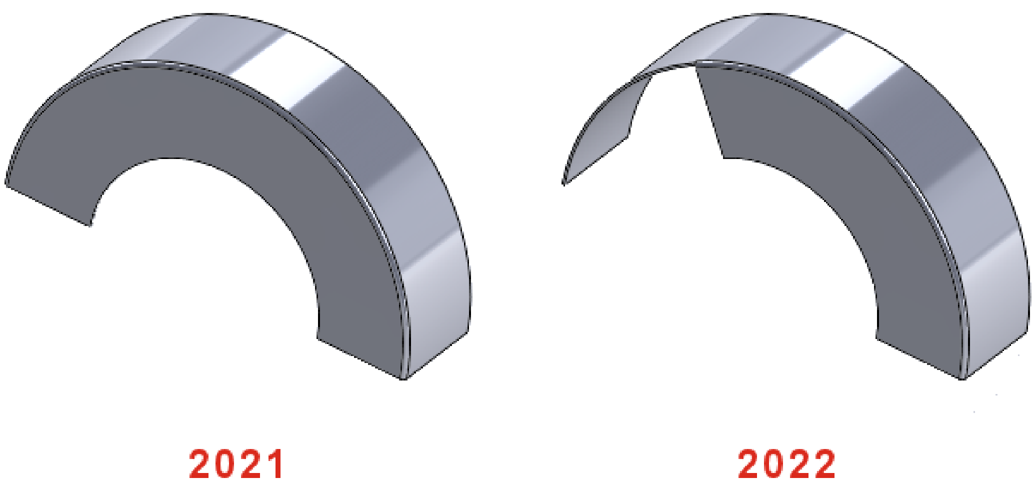
9

Sheet Metal

This chapter includes the following topics:

- **Edge Flanges**
- **Etched Contours on Bends**

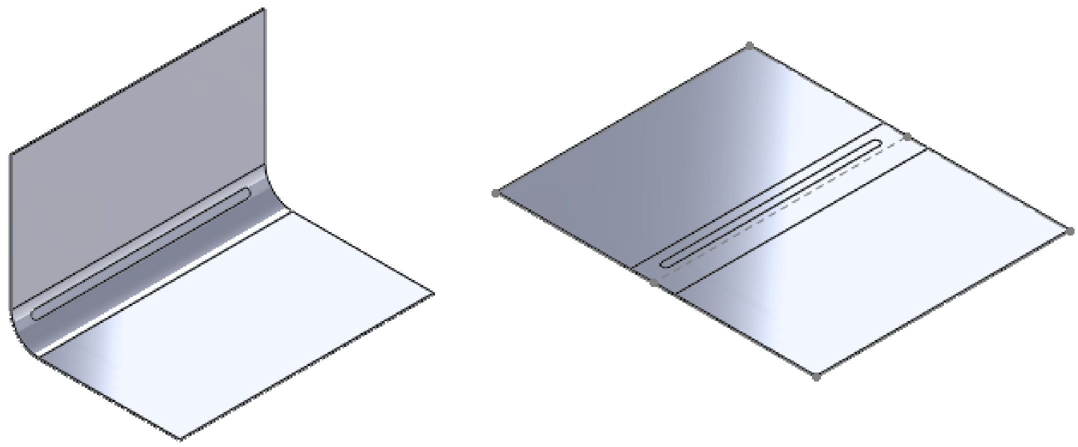
Edge Flanges



If you create an edge flange on a nonlinear edge, you can edit the sketch for the nonlinear edge flange.

In a sheet metal part, select a nonlinear edge and click **Edge Flange**  (Sheet Metal toolbar). In the PropertyManager, click **Edit Flange Profile** and edit the sketch.

Etched Contours on Bends

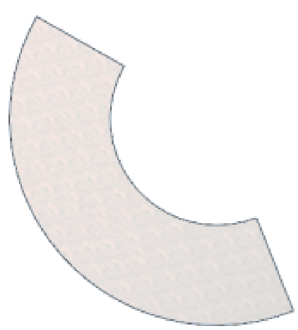
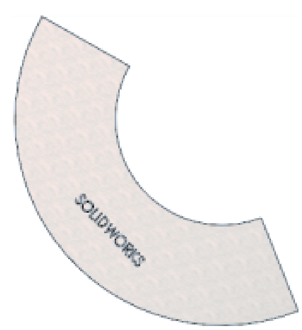


In sheet metal parts with inscribed text or split line features on the bend face, you can specify an option to keep the text or split line intact when you flatten, unfold, or fold the part.

For conical bends, the text or split line only appears in the flattened state when the conical face is an analytical face. To determine whether the face is analytical, create an axis by selecting the conical face. If the conical face is analytical, you can create the axis.

To use etched contours on bends:

1. In the FeatureManager design tree of a folded sheet metal part:
   a. Expand **Flat-Pattern**.
   b. Right-click **Flat-Pattern*n*** and click **Edit Feature**.
2. In the PropertyManager, under **Parameters**, select **Merge faces** and **Retain Scribed Faces** and click.

		
Folded part	Flattened - 2021	Flattened - 2022

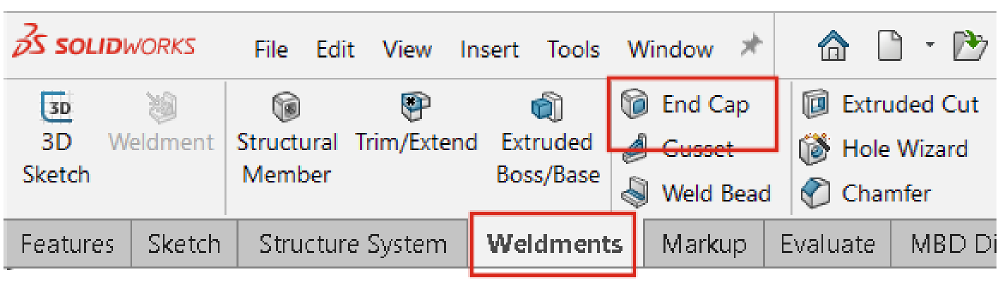
10

Structure System and Weldments

This chapter includes the following topics:

- **End Cap Support**
- **Custom Properties Architecture**
- **Complex Corner PropertyManager**
- **Secondary Members**
- **Connection Element for Structure Systems**
- **Properties Dialog Box**

End Cap Support

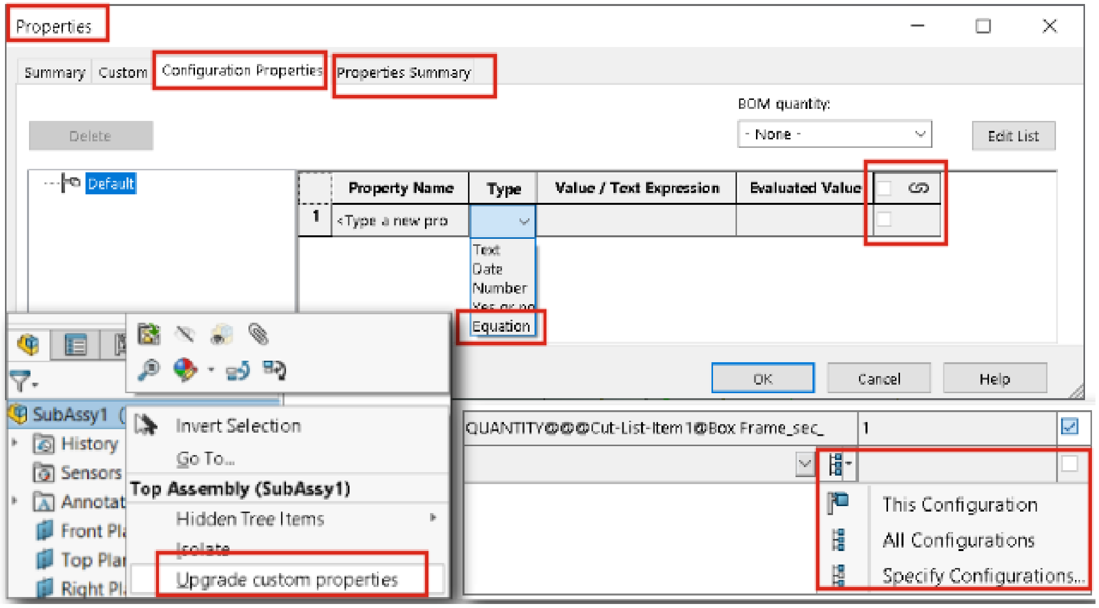


You can add end caps to structure system bodies with closed profiles like tubes, squares, and rectangular tubes.

Previously, you could add an end cap to weldments only.

Click **End Cap** (Weldments toolbar) or **Weldments** > **End Cap**.

Custom Properties Architecture



You can upgrade custom properties of files created in SOLIDWORKS 2017 or earlier.

When you run **Upgrade custom property** on legacy files, their custom properties are upgraded to the new architecture. After the upgrade, the following features are available to the legacy files:

- **Linked** column in the Custom Properties dialog box
- Configuration-specific cut list and custom properties
- Equation in **File Properties**
- Enhanced Configuration Specific Properties dialog box

When you upgrade the custom properties of a part, you may have to fix issues with assemblies and drawings where the part is used. For example, the assemblies and drawings that include derived parts, annotations, and BOMs.

You can upgrade a file manually or in batches using the API (`IModelDocExtension::UpgradeLegacyCustomProperties`).

For information, see *SOLIDWORKS API Help*. You can also contact SOLIDWORKS Technical Support.

Upgrading Custom Properties

You can upgrade the custom properties of parts, assemblies, and drawings.

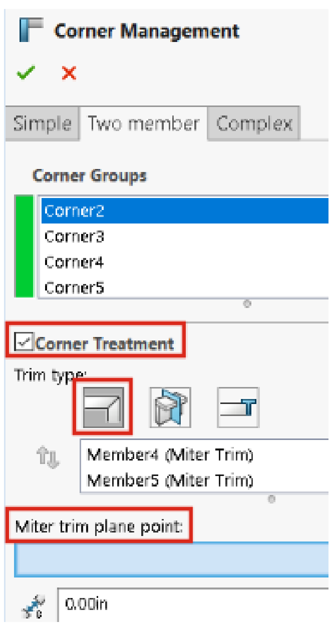
To upgrade custom properties:

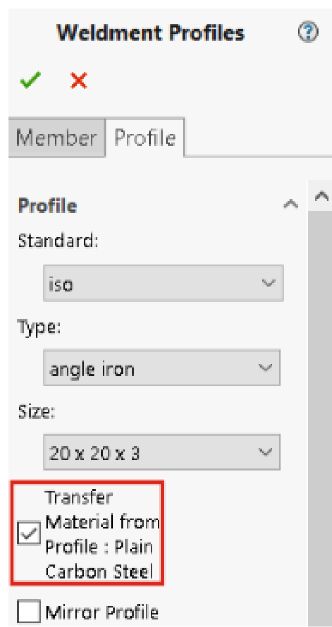
1. In the FeatureManager design tree, right-click the top item and select **Upgrade custom properties**.

   After you upgrade, **Upgrade custom properties**, is no longer available for that model.

   When you upgrade the custom properties of a drawing, the custom properties of its model views do not upgrade.
2. Optional: Right-click an assembly that includes parts or subassemblies and click **Upgrade custom properties**, then select **Upgrade top level assembly** or **Upgrade all the components**.

Complex Corner PropertyManager

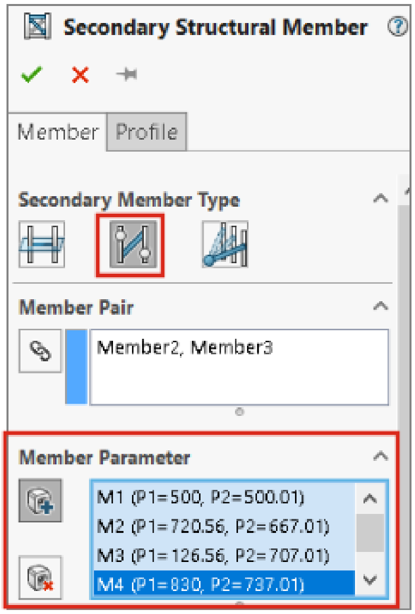


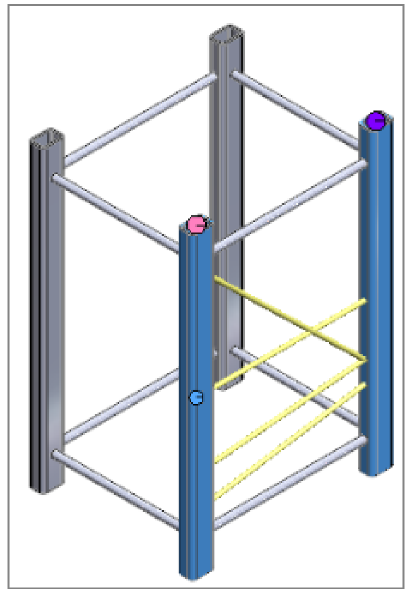


Improvements in the Complex Corner PropertyManager provide a clear workflow to use the Corner Management functionality.

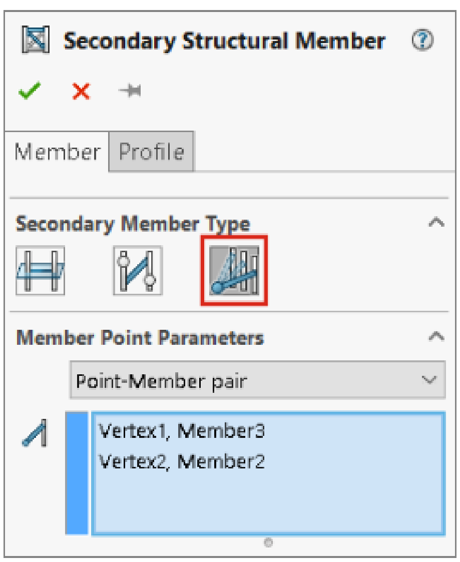
- On the Two member tab, under **Corner Management**, the **Miter trim plane point** lets you select a point from the graphics area to create the miter trim through the selected point.
- Options related to Corner Treatment are displayed only if you select the **Corner Treatment** check box.
- Icons for planar trim options are smaller than icons for trim type to depict the hierarchy.
- **Transfer Material from Profile** transfers the material of the selected library profile if the profile has a material assigned to it.

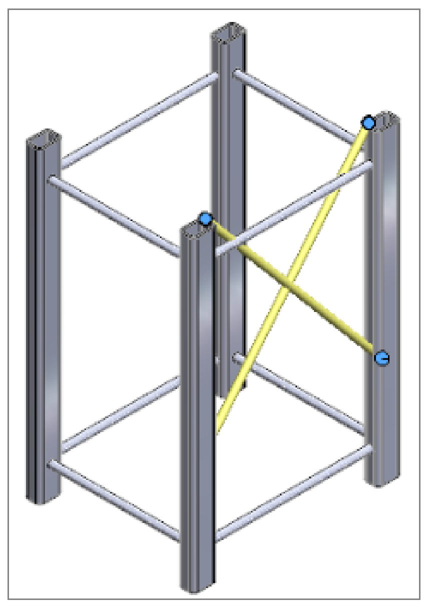
Secondary Members





You can create multiple secondary members using **Between Points Member** for the selected pair and create secondary members with the **Up to Members** method.





Creating Multiple Secondary Members Using Between Point Member

You can create multiple secondary members for a pair of members.

To create multiple secondary members using Between Points Member:

1. In the Secondary Member PropertyManager, click **Between Points Member**.
2. Under **Member Pair**, select the pair of members.
3. Under **Member Parameter**, click to add a secondary member.

   Click to delete the selected secondary member.
4. Optional: Adjust the offset using **Offset from First Member** and **Offset from Second Member** and flip it for the member that is selected under **Member Parameter**.

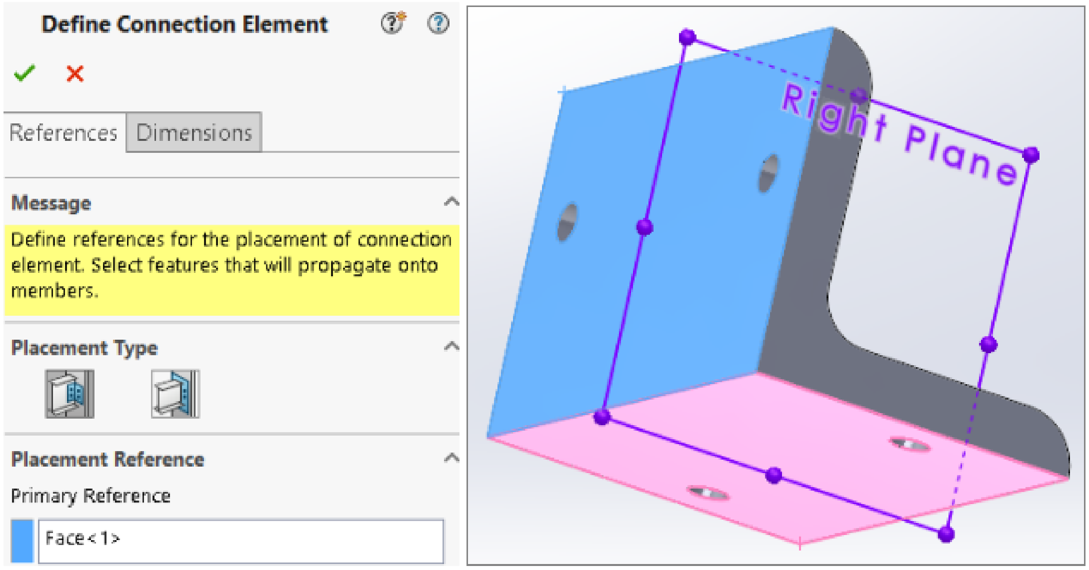
Creating Secondary Members with the Up to Members Method

You can create secondary members with **Up to Members** methods, for example, with the **Point Member pair** or **From Point** options.

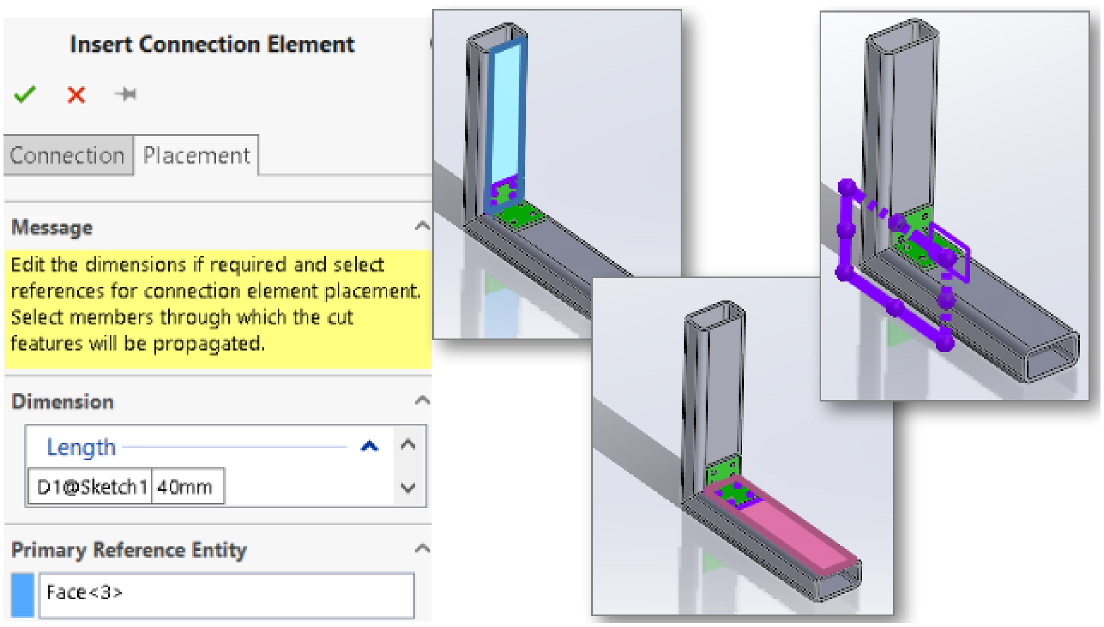
To create secondary members with the Up to Members Method:

1. In the Secondary Member PropertyManager, on the Member tab, for **Secondary Member Type**, click **Up to Members**.
2. For **Member Point Parameters**, select an option:
   - **Point Member pair**. When you select a point and member, creates a member between them.
   - **From Point**. When you select a point and multiple members, creates multiple members.
3. Optional: Adjust the offset or flip the members.

Connection Element for Structure Systems



SOLIDWORKS supports connection elements for structure systems. You can define the connection element and insert it on a structure system part.



Defining and Inserting Connection Elements

To define and insert connection elements:

1. Click **Define Connection Element** (Structure System toolbar) or **Insert** > **Structure System** > **Define Connection**.

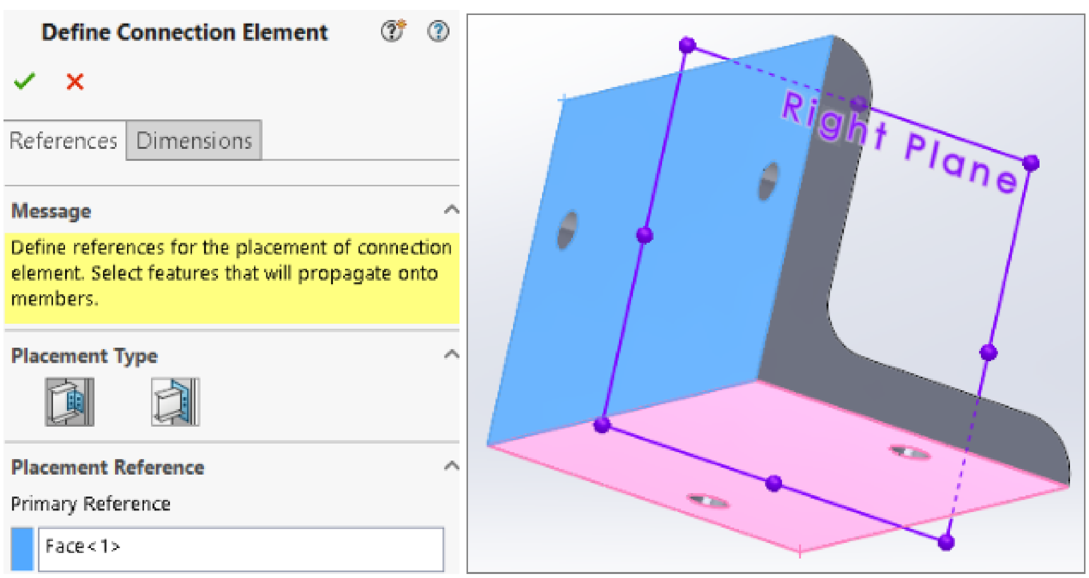
2. In the PropertyManager, on the References tab, specify options to define the connection element.
3. Click .
4. Save the connection element.

> The default location to save the connection element is `install_dir\data\Structure System - Connection Elements`. If you cannot save the connection element to the installation directory, save it locally. You can add the file location for **Structure System-Connection Elements** in **Tools** > **Options** > **System Options** > **File Locations**.

5. Click **Insert Connection Element** (Structure System toolbar) or **Insert** > **Structure System** > **Insert Connection**.
6. In the PropertyManager, on the Connection tab, specify options to insert the connection element.
7. On the Placement tab, specify options.
8. Click .

Connection Definition PropertyManager - References Tab

The References tab in the Connection Definition PropertyManager lets you create a connection element for a structure system from a part.



Placement Type

	Generic Connection	Places the connection element based on the face selection.

	End Connection	Places the connection element at the end.

Generic Connection

If you selected **Generic Connection** for the **Placement Type**, you can select the faces or planes to place the connection element.

Placement Reference	Select **Primary Reference** and **Secondary Reference**.
Primary Reference	Specifies the first face or plane.
Secondary Reference	Specifies the second face or plane.
Mate Type	Select **Coincident**, **Concentric**, or **Parallel** based on the **Secondary Reference**.
Tertiary Reference	Specifies the third face or plane.
Mate Type	Select **Coincident**, **Concentric**, or **Parallel** based on the **Tertiary Reference**.

Feature Propagation

You can select features such as extruded cuts, holes, advanced holes, patterns, and mirrors to propagate to the target part. However, if you created a hole based on the extruded boss feature, you cannot propagate the hole.

End Connection

If you selected **End Connection** for the **Placement Type**, you can select a face on which you want to insert the connection element.

Primary Reference	Specifies a face. You cannot select reference planes as inputs.

Connection Definition PropertyManager - Dimensions Tab

You can select the dimensions of the connection element and then modify them when you insert the connection element.

Click **Define Connection Element** (Structure System toolbar) or **Insert** > **Structure System** > **Define Connection**, and select the Dimensions tab.

Dimension Group

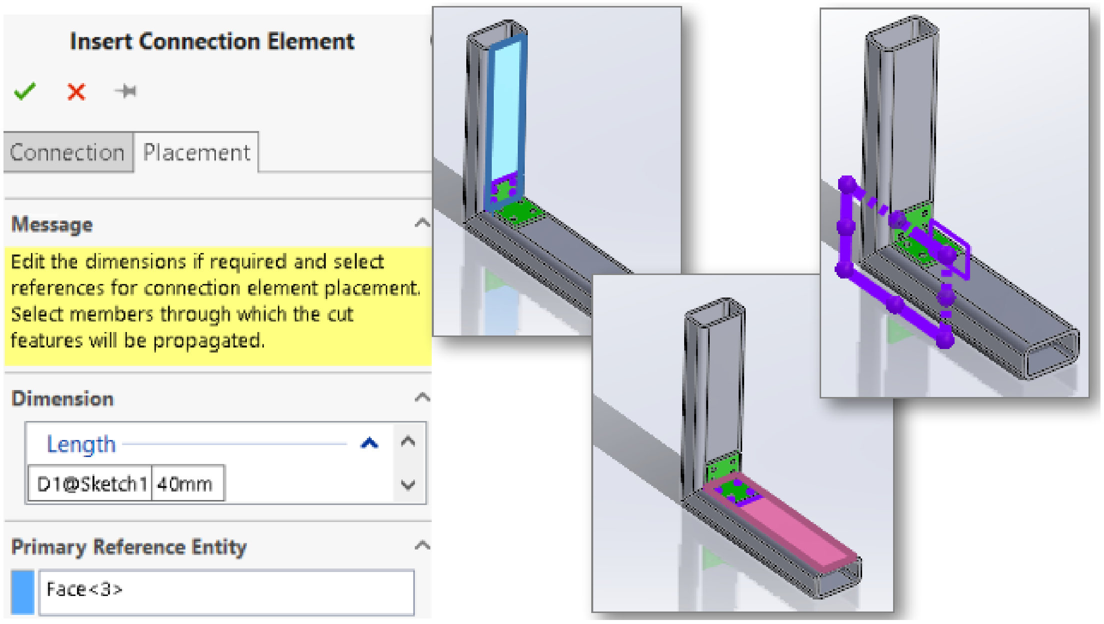
You can create different dimension groups and select dimensions in the graphics area.

New Dimension Group	Creates a new dimension group.

When you insert the connection element, you can edit the selected dimensions.

Insert Connection PropertyManager

This PropertyManager lets you insert the connection element on a structure system part.



To open this PropertyManager:

Click **Insert Connection Element** (Structure System toolbar) or **Insert** > **Structure System** > **Insert Connection**.

Connection Tab

This tab displays the default values for **Standard**, **Type**, and **Size**. You can select the values based on the connection element you saved.

Placement Tab

This tab displays dimension, placement reference, and cut scope.

Dimension

Change the dimension of the feature that you selected in the **Dimension Group** when you defined the connection element.

Placement Reference

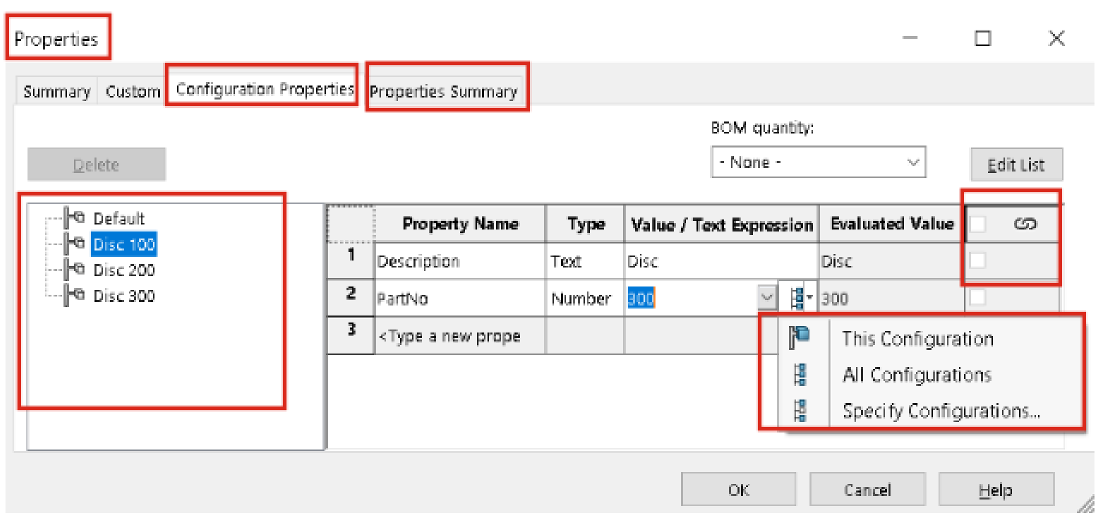
Primary Reference Entity	Specifies a face in the graphics area to coincide with the primary reference of the connection element.
Secondary Reference Entity	Specifies a face in the graphics area to coincide with the secondary reference of the connection element.
Tertiary Reference Entity	Specifies a face in the graphics area to coincide with the tertiary reference of the connection element.

Cut Scope

In the graphics area, select the members on which to propagate the cut features. Select one of the following options:

Up to Next	Propagates the cut feature up to the adjacent face
Through All	Propagates the cut features throughout the member

Properties Dialog Box



The enhanced Properties dialog box provides flexibility to add or edit custom properties.

Click **File Properties** (Standard toolbar) or **File** > **Properties**.

The Summary Information dialog box is renamed to the Properties dialog box. The tabs included in this dialog box are Summary, Custom, Configuration Properties, and Properties Summary.

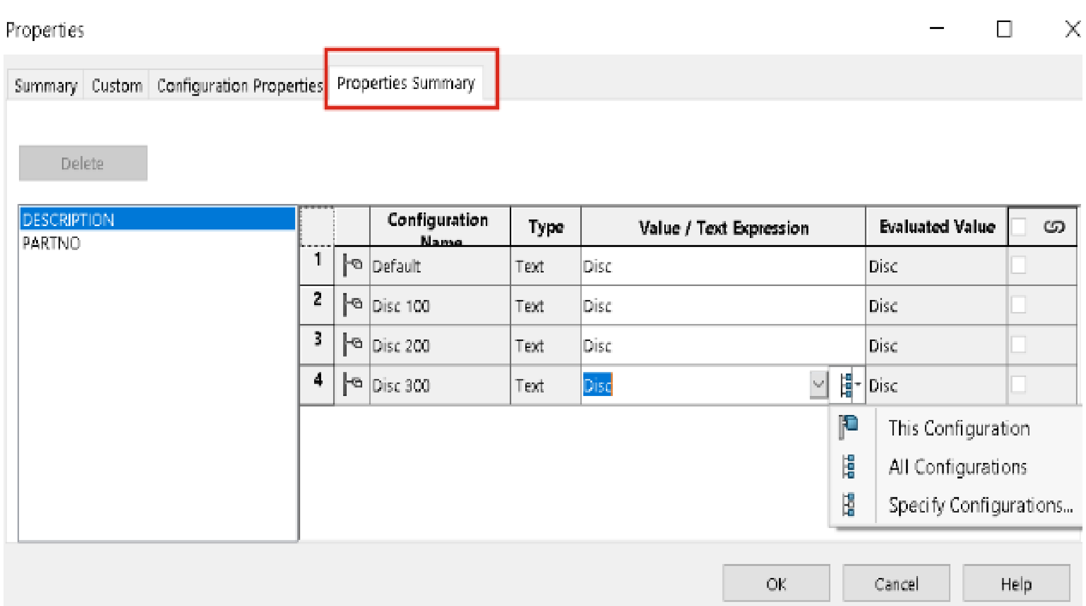
Configuration Properties Tab

The Configuration Properties tab contains two sections.

The first section lists configurations and derived configurations. You can select only one configuration at a time and enter the properties.

The second section displays the configurations and the values of the selected property. In the **Value/Text Expression** column, you can select **This Configuration**, **All Configuration**, or **Specify Configuration** to apply the property to the selected configuration, all configurations, or specific configurations.

Properties Summary Tab



The Properties Summary tab contains two sections. The first section lists the properties of all configurations. The second section lists all configurations. You can add or modify the property values.

Adding or Modifying a Property

You can add or modify configuration properties.

To add or modify a property:

1. Click **Properties** (Standard toolbar) or **File** > **Properties**.

2. In the dialog box, on the Configuration Properties tab, click a blank row of the table and select a property name.
3. In **Type**, select a property type.
4. For **Value/Text Expression**, enter the value and select one of the following:

This Configuration	Applies the property to the selected configuration.
All Configurations	Applies the property to all configurations.
Specify Configurations	Applies the property to specific configurations.

5. Optional: On the Properties Summary tab, select the property and the edit the value.
6. Click **OK**.

Properties Dialog Box in Large Design Review Mode

The Properties dialog box in Large Design Review (LDR) mode displays only the Configuration Properties tab in view-only mode.

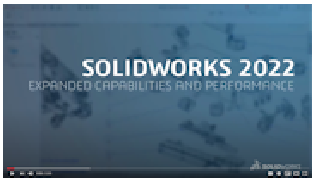
The Configuration Properties tab contains two sections, and you cannot add or edit the properties.

- The first section displays only the last saved active configuration with representation (if any).
- The second section displays all properties of the configuration.

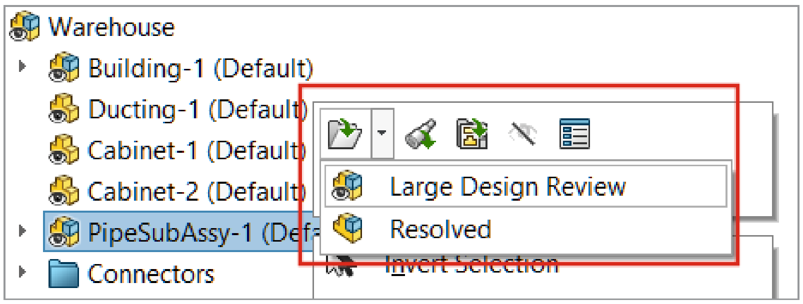
11

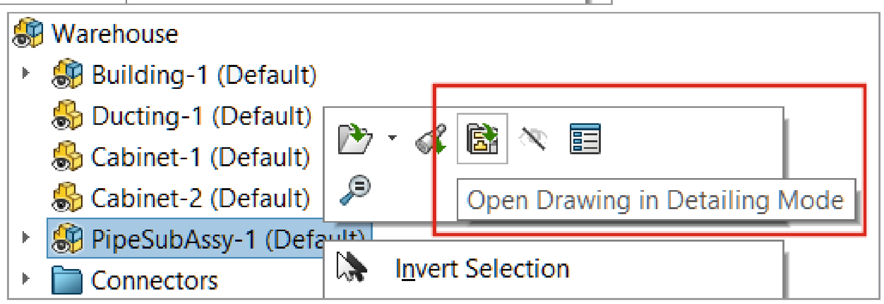
Assemblies

This chapter includes the following topics:

	**Video: What's New in SOLIDWORKS 2022 - Assemblies**

Opening Subassemblies in a Different Mode





You can open a subassembly in Large Design Review or resolved mode from an assembly opened in Large Design Review mode. You can also open a drawing in Detailing mode.

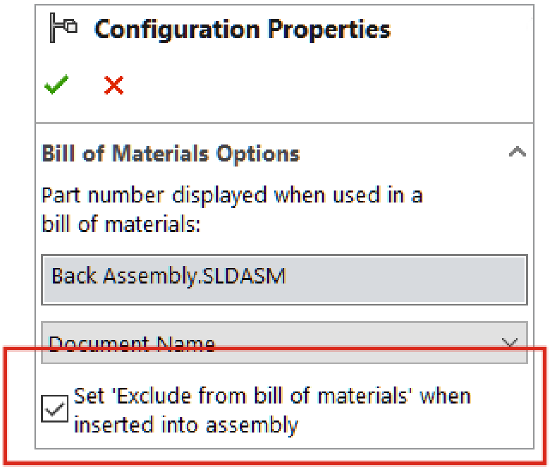
To open subassemblies in a different mode:

1. Open an assembly that contains a subassembly in Large Design Review mode.
2. Right-click a subassembly and click the down arrow for **Open**.
3. Click **Large Design Review** or **Resolved**.

To open a drawing in Detailing mode:

1. Open an assembly that contains a drawing in Large Design Review mode.
2. Right-click the assembly and click **Open Drawing in Detailing Mode**.

Excluding a Component from a Bill of Materials

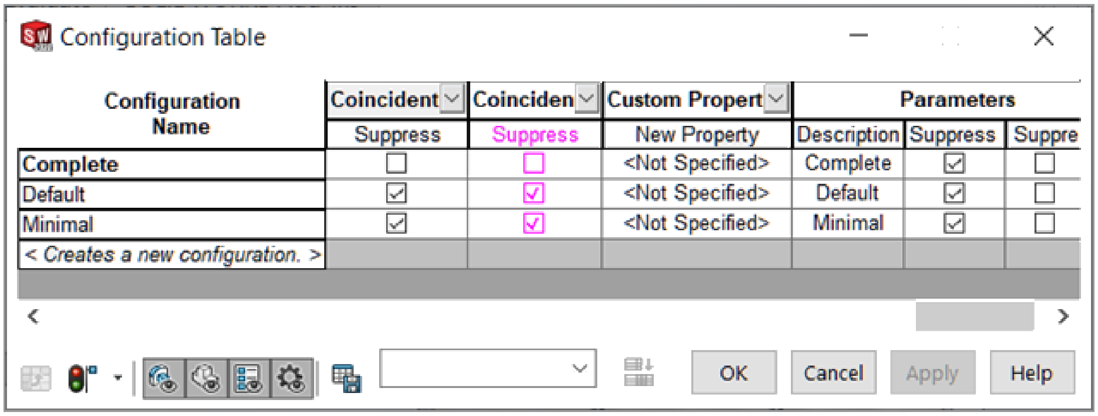


You can exclude a component, configuration, or model from the bill of materials. When you select this option, the exclusion occurs the next time you insert the component, configuration, or model.

To exclude a component from a bill of materials:

1. Open a component and click the ConfigurationManager tab.
2. Right-click a configuration and click **Properties**.
3. In the PropertyManager, under **Bill of Materials Options**, select **Set 'Exclude from bill of materials' when inserted into assembly**.

Configuration Table



You can use **Configuration Table** to modify configuration parameters for parts and assemblies. SOLIDWORKS creates this table automatically when the part or assembly has multiple configurations. The configuration table extends the functionality provided in the Modify Configurations dialog box.

The configuration table contains the following sections:

Sketches and Features	Feature dimension values and suppression states.
Components	Component suppression states and referenced configuration. Assemblies only.
Custom Properties	Configuration-specific custom properties.
Parameters	Parameters applied to the configuration.

The Configuration Table dialog box includes options from the Modify Configuration dialog box and these options:

	Hide/Show Sketches and Features	Controls the visibility of columns that contain suppression states, and sketch and feature dimensions.
	Hide/Show Components	Controls the visibility of columns that contain component configurations, fix/float states, and suppression states.

	Hide/Show Configuration Parameters	Controls the visibility of columns that contain configuration parameters.
	Block model edits	Prevents updates to the model. To select this option, right-click a column header. When you block edits, the column displays in a different color. To change this color, click **Tools** > **Options** > **System Options** > **Colors**. Modify the **Dimensions, Controlled by Design Table** color setting. In the graphics area, dimensions that are blocked for editing appear in the same color.

You can modify the following parameters:

Suppress new features	Parts only.
Suppress new features and mates	Assemblies only.
Suppress new components	
Fix / Float	Assemblies only.
Child components in BOM	Configures the setting for **Child component display when used as a subassembly** in the Configuration Properties PropertyManager. Select **Show**, **Hide**, or **Promote**.
Exclude from BOM on Insert	Configures the setting for **Set 'Exclude from bill of materials' when inserted into assembly** in the Configuration Properties PropertyManager.

Design Table is renamed to **Excel Design Table**.

To enable configuration tables:

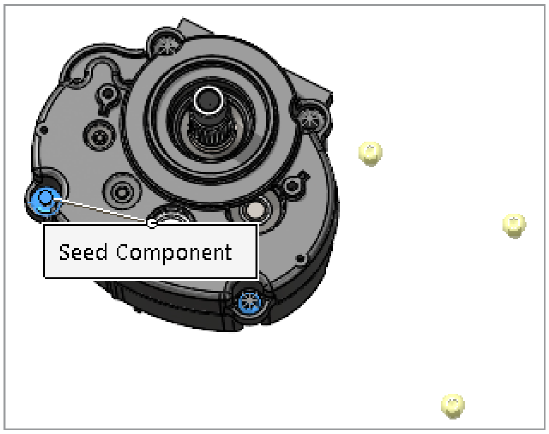
1. Click **Options** > **Tools** > **System Options** > **General**.
2. Select **Create configuration tables on open**.

   Using this option can affect performance.

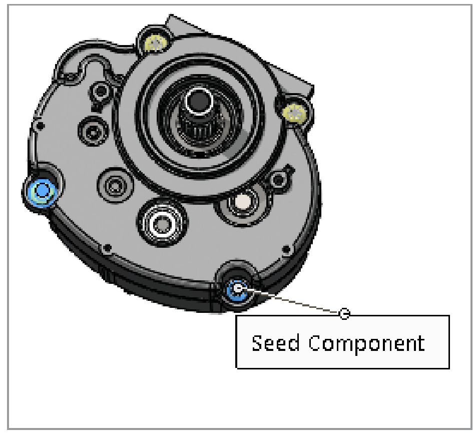
To access a configuration table:

1. Open an assembly that has multiple configurations and click the ConfigurationManager tab.
2. Expand **Tables**.
3. Right-click **Configuration Table** and click **Show Table**.

Default Seed Position for Patterns



2021



2022

SOLIDWORKS selection for the default seed position is improved when you create Pattern Driven Component Patterns.

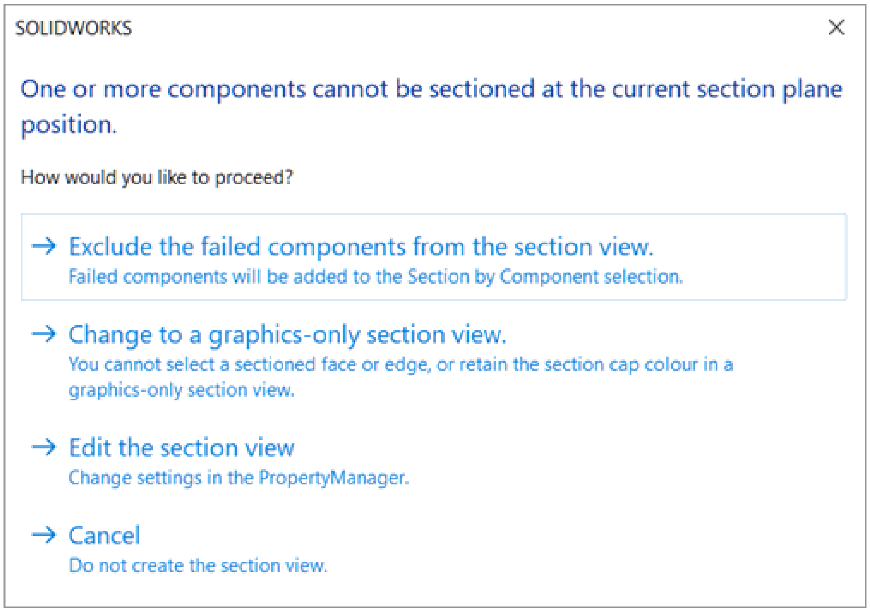
To view the default seed position for patterns:

1. Open a model and click **Insert** > **Component Pattern** > **Pattern Driven Component Pattern**.
2. In the PropertyManager, select components for **Components to Pattern** and **Driving Feature or Component**.

   The default seed position appears as the **Seed Component** in the graphics area.

   When you select a driving feature, **Seed Feature** appears in the graphics area.

Excluding Failed Components in a Section Views



When SOLIDWORKS cannot calculate a section view in an assembly, you can exclude the components causing the failure, switch to a graphics-only section view, or edit settings in the PropertyManager.

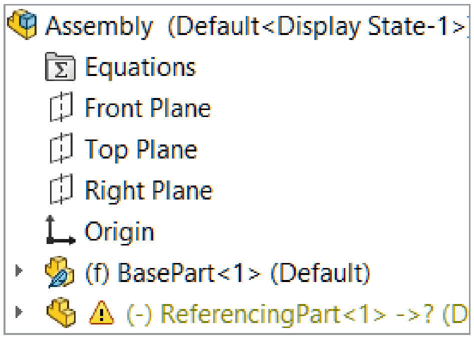
To exclude failed components in a section view:

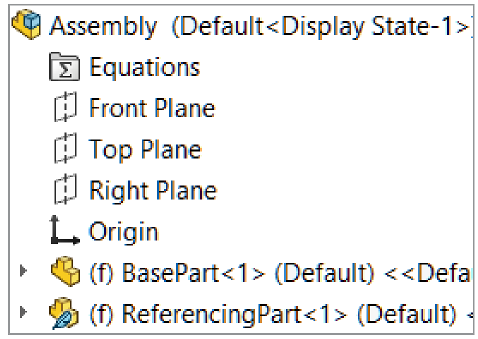
1. In an assembly, click **View** > **Display** > **Section View**.
2. Specify options in the PropertyManager and click .
3. When you receive the message that a component cannot be sectioned, select **Exclude the failed components from the section view**.

   In the PropertyManager, the failed components move to **Section by Component** and **Exclude selected** is selected.
4. Optional: To view the list of excluded components, open the PropertyManager and review **Section by Component**.

If you switch to a graphics-only section view, **Graphics-only section** is selected in the PropertyManager.

Resolving Equations in Lightweight Mode





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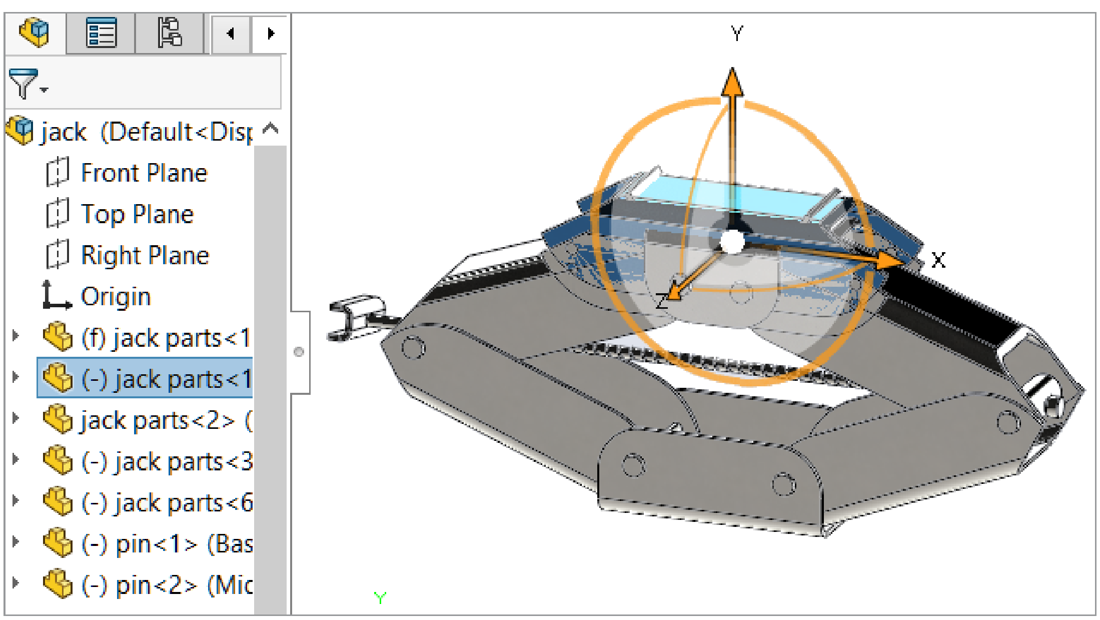
When you open an assembly in lightweight mode, components referenced by equations in the top-level assembly display as resolved.

When you resolve a part or subassembly that has equations and the equations reference another lightweight component, the referenced component is resolved.

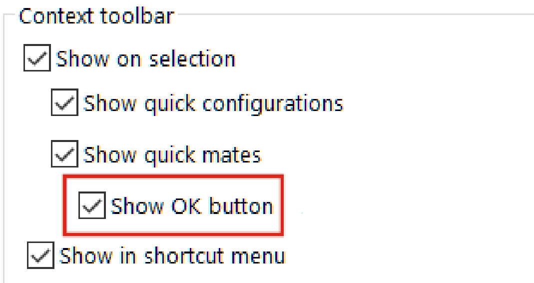
Move with Triad

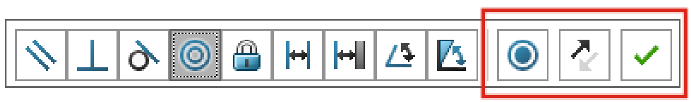


The triad appears automatically when you select one or more components.

When the triad is available, **Ctrl** + drag to copy the selected components along the specified direction.

Quick Mates Context Toolbar





By default, **Add/Finish Mate** is hidden in the Quick Mates context toolbar for the following mates: coincident, concentric, parallel, symmetric, and tangent.

Lock Rotation and **Flip Mate Alignment** appear in the Quick Mates context toolbar.

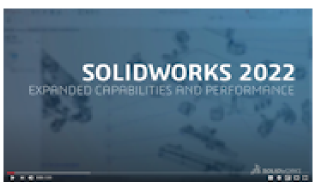
To show **Add/Finish Mate** in the toolbar, open a document and click **Tools** > **Customize**. On the Toolbars tab, under **Context toolbar**, select **Show quick mates** and **Show OK button**.

Show flip alignment and lock rotation after quick mates is removed from the context toolbar.

12

Detailing and Drawings

This chapter includes the following topics:

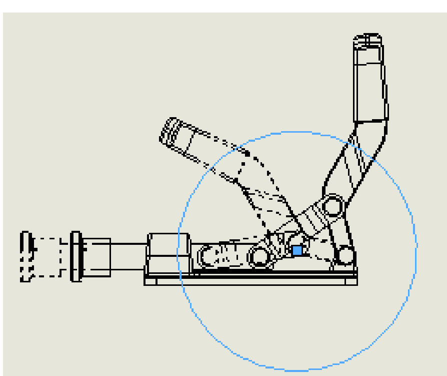
	**Video: What's New in SOLIDWORKS 2022 - Drawings**

Cropping an Alternate Position View

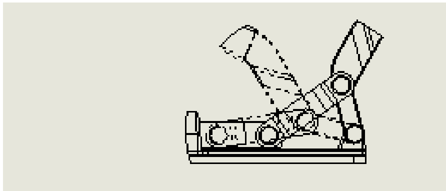
You can create a cropped view of an alternate position view.

Previously, the **Crop View** tool did not crop the images of the alternate position view.

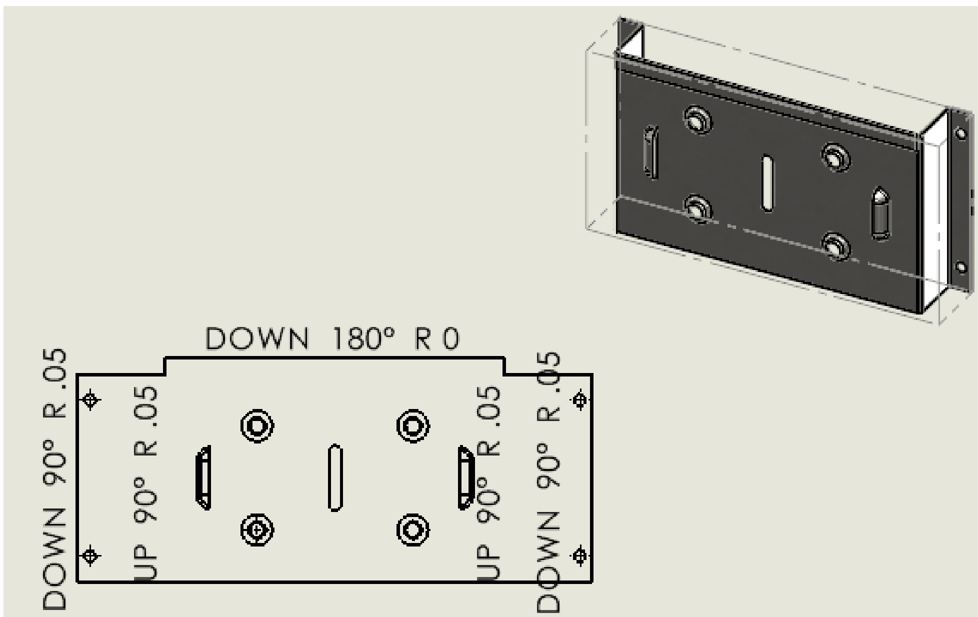
1. In an alternate position view, sketch a closed profile such as a circle.



2. Click **Crop View** (Drawing toolbar) or **Insert** > **Drawing View** > **Crop**.



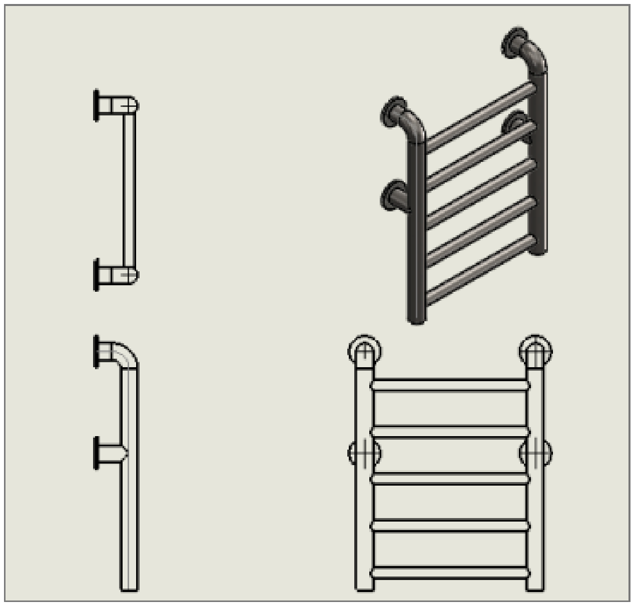
Predefined Views



When you create a predefined view in a drawing, you can specify the model orientation as trimetric, dimetric, or flat pattern. This is helpful if you need drawing templates that contain specific predefined views.

Click **Predefined View** or **Insert** > **Drawing View** > **Predefined**. In the PropertyManager, under **Orientation**, select **Flat pattern**, **Trimetric**, or **Dimetric**.

Detailing Mode



Detailing mode is available for all drawings (except detached drawings), regardless of the SOLIDWORKS version in which you saved the drawing or whether you saved the drawing in Detailing mode.

In Detailing mode, you can:

- Use limited Detailing mode if you saved drawings in previous versions of SOLIDWORKS or in SOLIDWORKS® 2022 without model data. If you open a drawing in limited Detailing mode, the window title displays `file name - sheet name [Detailing - Limited]`. (Limited Detailing mode is an automatic mode - you cannot specifically select it.)
- Create hole tables.
- Drag standard views (such as front, top, back) from the View Palette to the drawing.

In **Tools** > **Options** > **Document Properties** > **Performance**, there are two new options:

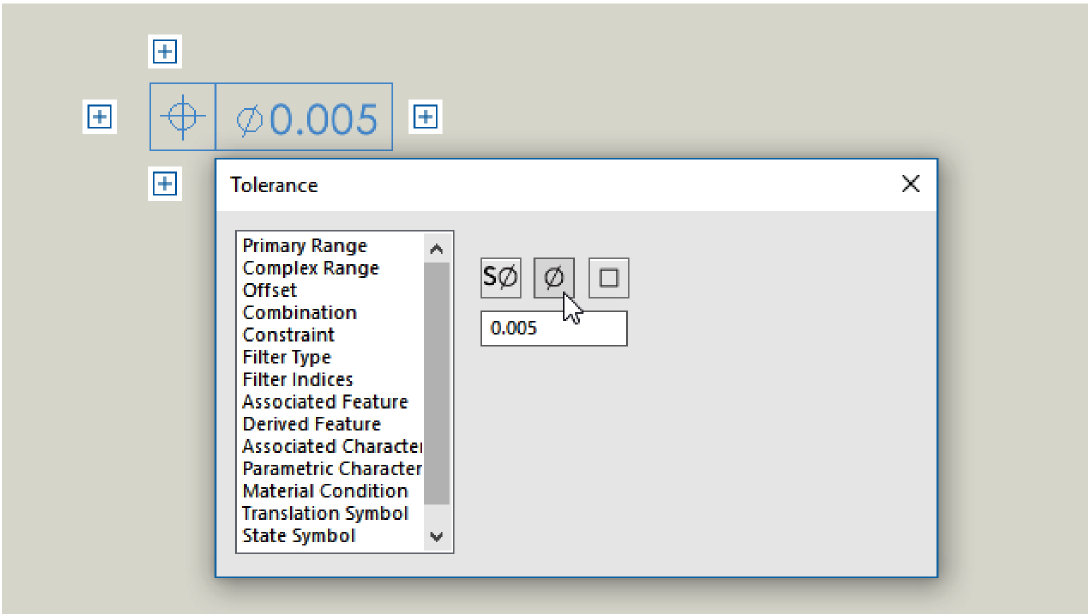
- **Save model data**. Saves all drawings with model data to use in Detailing mode.
- **Include standard views in View Palette**. Lets you create standard views when you add drawing views from the View Palette.

You cannot change these options while in Detailing mode, and they only apply when you save resolved drawings.

In **Tools** > **Options** > **System Options** > **Drawings** > **Performance**, the option, **Include Detailing Mode Data when saving**, has been removed.

Detailing mode has replaced Quick view mode. Quick view mode has been removed from SOLIDWORKS 2022.

Geometric Tolerance Symbols



The user interface for geometric tolerancing improves your workflow. You work directly in the graphics area, and the interface guides you as you build feature control frames cell-by-cell.

With cell-specific context menus and on-screen handles, you can build complex feature control frames while keeping focus on the frame itself rather than a separate dialog box.

To enter content into the active cell, you enter values and select items from the context menu. You can randomly enter and edit all the content in the frame. Various cell types are available, including

- **Datum/Datum Group**
- **Indicator**
- **Text Box**

To add another cell, click a handle and select the cell type. Depending on the location of the handle, other options are available, for example **New Frame** and **Text Box**.

Creating Geometric Tolerance Symbols

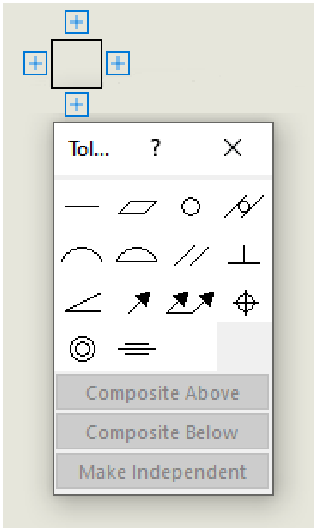
When creating geometric tolerance symbols, you can use the handles surrounding the feature control frame to build the symbol.

To create geometric tolerance symbols:

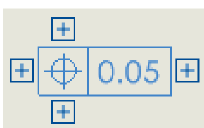
1. In a part or drawing, click **Geometric Tolerance** or **Insert** > **Annotations** > **Geometric Tolerance**.

2. In the graphics area, click to place the symbol.

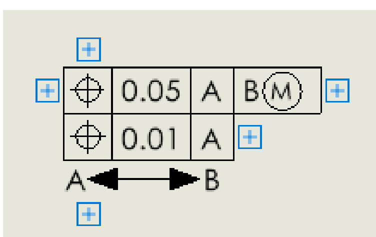
   A feature control frame appears with handles and a Tolerance dialog box surrounding it.

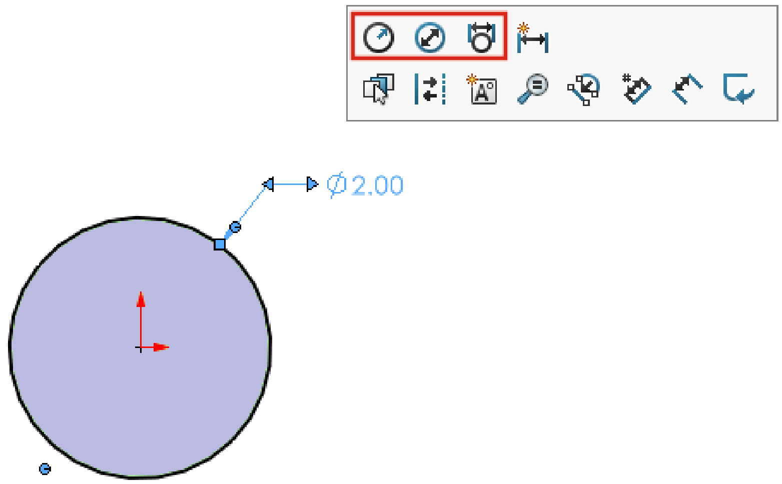
3. In the dialog box, specify options to add content to the feature control frame.

4. Click any handle surrounding the feature control frame to add more content.

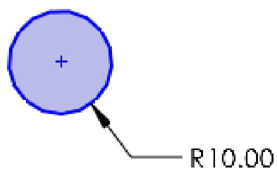
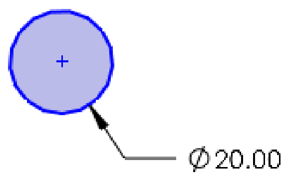
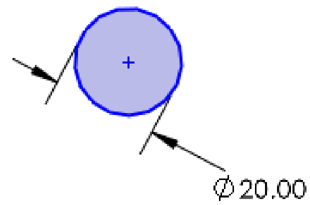
   

Switching Between Radius and Diameter Dimensions

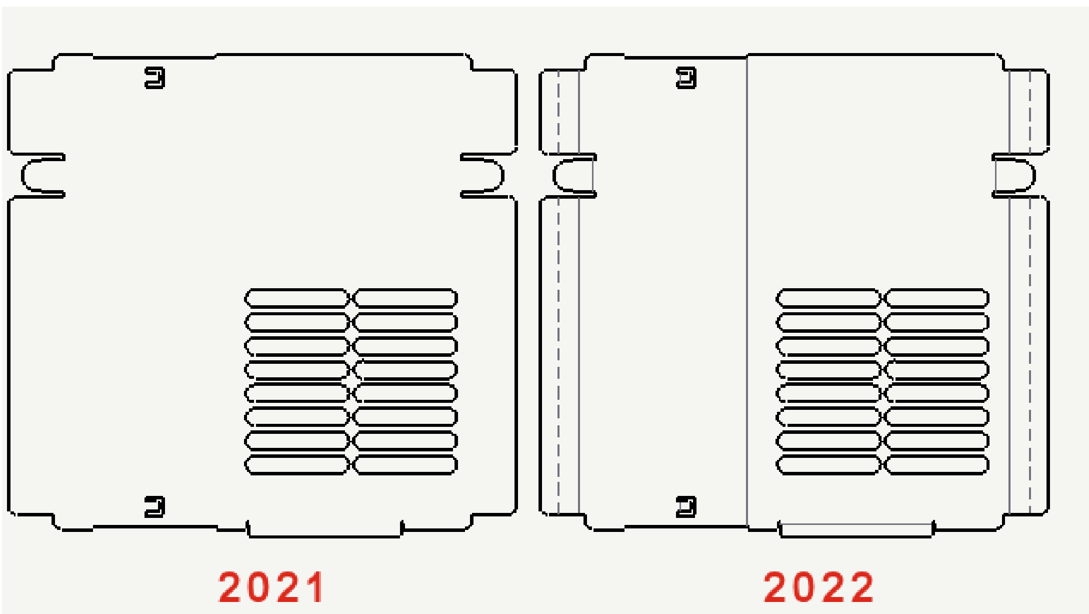


In drawings, parts, and assemblies, for dimensions of arcs and circles, you can use the context toolbar to switch the dimension to display as a radius, diameter, or linear diameter.

Click an existing circular or radial dimension to access the context toolbar.

		
Display as Radius	**Display as Diameter**	**Display as Linear Diameter**

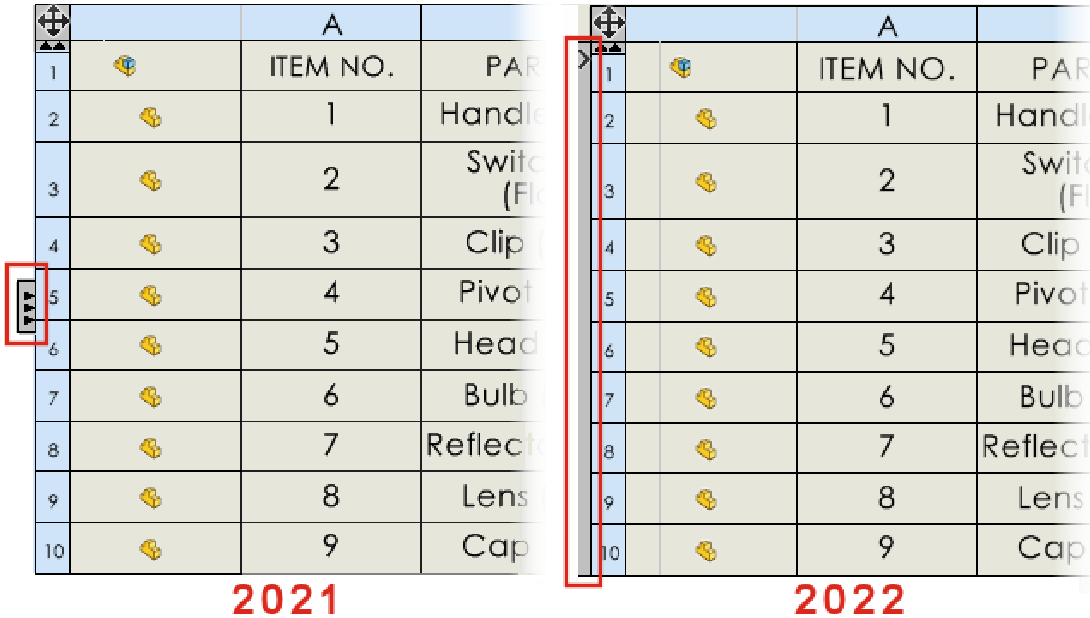
Bend Lines in Drawing Views



In flat pattern drawing views of sheet metal parts, bend lines are independent of sketches. You can hide sketches while keeping bend lines visible. Previously, if you hid sketches in a flat pattern view, the bend lines would also be hidden.

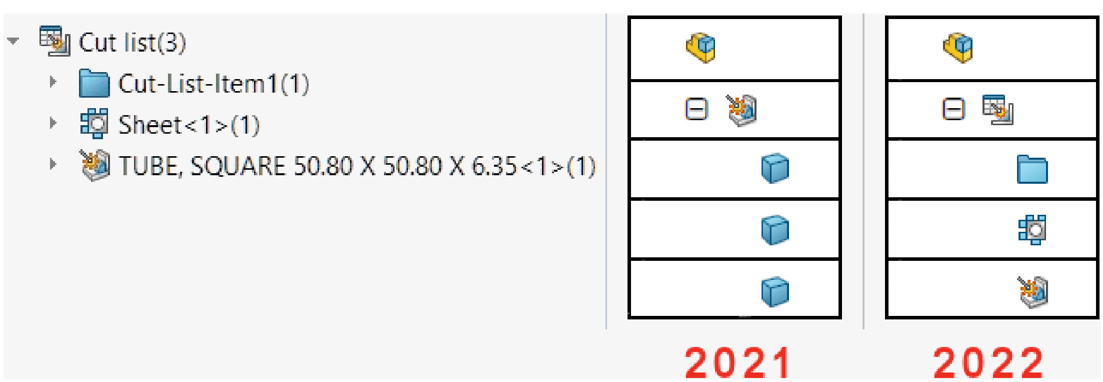
Click **View** > **Hide/Show** > **Bend Lines** or **Sketches**.

Bill of Materials



The user interface for bill of materials (BOMs) tables has been updated for ease of use. When you expand or collapse a BOM, the selectable area spans the entire side of the BOM table.

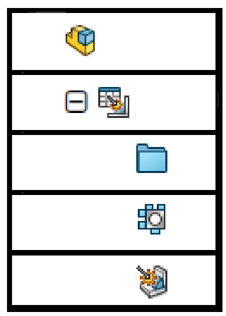
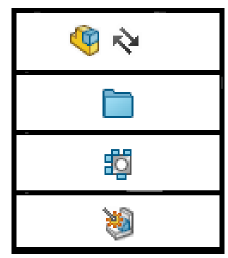
Cut List Support in BOM Tables



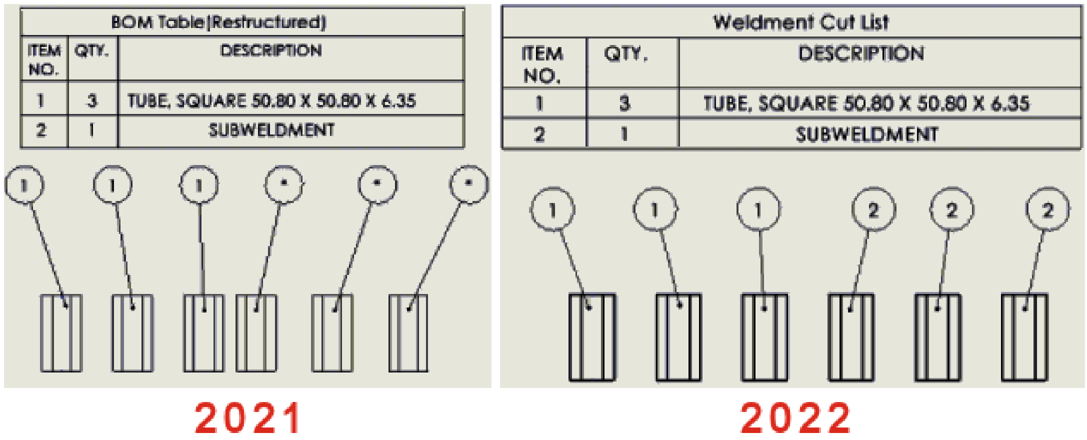
The user interface and functionality for detailed cut lists in Bill of Materials (BOM) tables has been updated for ease of use.

Updates to weldment cut lists tables:

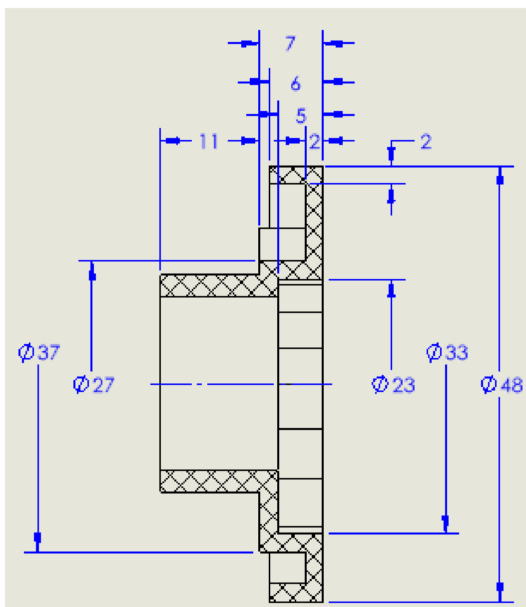
- Table icons match those used in the FeatureManager® design tree.
- In **Tools** > **Options** > **Document Properties** > **Bill of Materials**, the option, **Combine cutlist items in BOM regardless to profile when lengths are changed to be the same (legacy behavior)** is renamed to **Combine same length cut list items with different profiles (pre-2019 behavior)**.
- Detailed cut lists are available for all BOM types. Previously, detailed cut lists were only available for indented BOMs. You can specify options in the **Detailed Cut List** section of the Bill of Materials PropertyManager.
- You can dissolve components in indented BOMs automatically or manually. In the Bill of Materials PropertyManager, select **Dissolve part level rows** to dissolve all weldment part level rows automatically.

		
Detailed Cut List cleared, **Dissolve part level rows** cleared	**Detailed Cut List** selected, **Dissolve part level rows** cleared	**Detailed Cut List** selected, **Dissolve part level rows** selected

- In the Configuration Properties PropertyManager, under **Bill of Materials Options**, you can specify options for **Cut list item display when component is used in an assembly (Indented BOM type with detailed cut list only)**. This option shows, hides, or promotes cut list items of indented BOMs. **Promote** dissolves the part level row and shows the cut list items in the indented BOM when you select **Detailed Cut List**. In the FeatureManager design tree, you can right-click any component and click **Restore in BOM** to move the components back to **Show Cutlist**.
- When you use balloons for subweldment cut list items in BOMs, the balloon properties correspond to the linked BOM. Before SOLIDWORKS 2022, balloons applied to bodies that belonged to subweldments appeared with an asterisk. Now the balloons have a corresponding item number.



Symmetric Linear Diameter Dimensions



You can create symmetric linear diameter dimensions for diameter dimensions that need only one side of a leader displayed. This is helpful for drawings with turned components and detailed section views.

To create symmetric linear diameter dimensions:

1. Open a drawing with a turned component or a section view.

   The symmetric linear diameter dimension is available for any drawing view, but it is best suited for drawings views that need only one side of a leader.

2. Click **Symmetric Linear Diameter Dimension** (Dimensions/Relations toolbar) or **Tools** > **Dimensions** > **Symmetric Linear Diameter Dimension**.
3. In the PropertyManager, specify options:

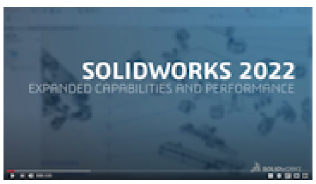
- **Single**. Lets you manually dimension entities. Select **Rapid dimensioning** to dimension entities with the rapid dimension functionality.
- **Multiple**. Lets you create multiple dimensions based on a specified axis. The dimensions are automatically arranged.

4. Dimension the drawing view.

13

Import/Export

This chapter includes the following topics:

	**Video: What's New in SOLIDWORKS 2022 - Import**

Import Performance Improvements

SOLIDWORKS 2022® offers improved performance for importing certain file types.

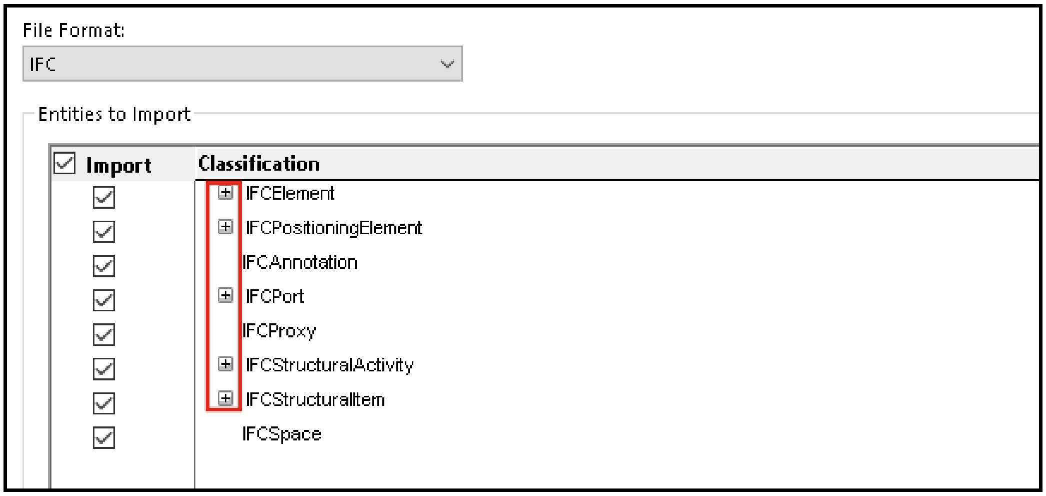
Performance is improved for importing:

- A large `DXF` or `DWG` file into a part sketch.

  You can import a large `DXF` or `DWG` file into a part sketch with the **Explode Block** option turned off. You no longer need to explode the blocks to improve import performance.

- `STEP` files in SOLIDWORKS.

Importing Selective IFC Entities from IFC Files



You can select which `IFC` entities to import from the `IFC` files by defining filters in System Options.

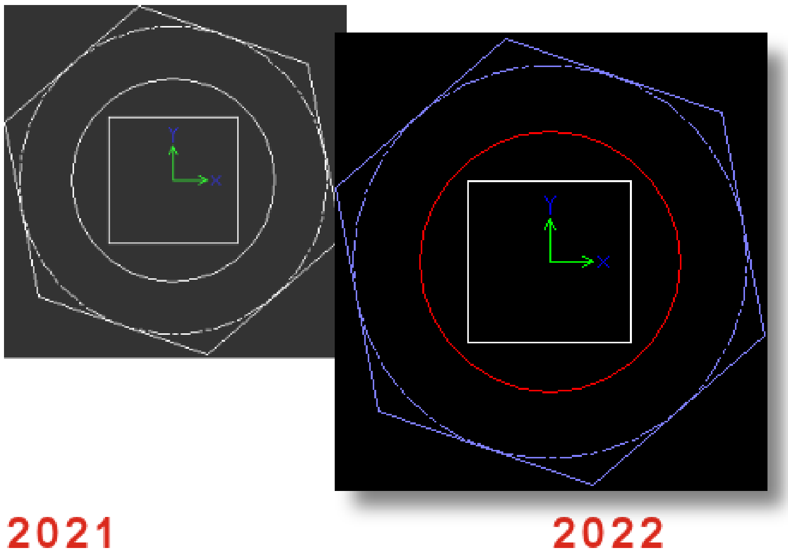
The file format, IFC, is added to **Tools** > **Options** > **System Options** > **Import** > **File Format**.

To import selective `IFC` entities from `IFC` files:

1. In an `IFC` file, click **Tools** > **Options** > **System Options** > **Import**.
2. Under **File Format**, select **IFC**.
3. Under **Entities to import**, expand the `IFC` entities under **Classification**.
4. Under **Import**, select the `IFC` entities to import.
5. Click **OK**.

After you open the `IFC` file, only the `IFC` entities that you specified in the System Options dialog box are imported.

Colors in Exported Sketches



When you save a part or drawing as a `DWG` or `DXF` file, sketch entities appear in the assigned sketch color in the exported file. The colors are also supported for sketches in flat patterns of sheet metal parts if you specify **Flat pattern colors** in **Tools** > **Options** > **Document Properties** > **Sheet Metal MBD**.

Opening Non-Native Assemblies with Reference Files Located in Different Folders

You can open non-native assemblies whose reference component files are stored in folders that are not in the same folder tree as the assembly files.

In **Tools** > **Options** > **System Options** > **File Locations**, in **Show folders for**, select **Referenced Documents**. SOLIDWORKS searches the specified folders that contain the referenced files in addition to the main folder tree to resolve assembly components.

Supported file formats are Autodesk® Inventor, Pro/E, Solid Edge®, and UG-NX.

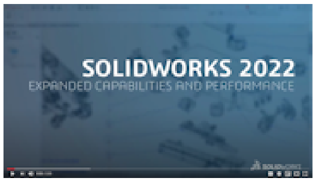
14

SOLIDWORKS PDM

This chapter includes the following topics:

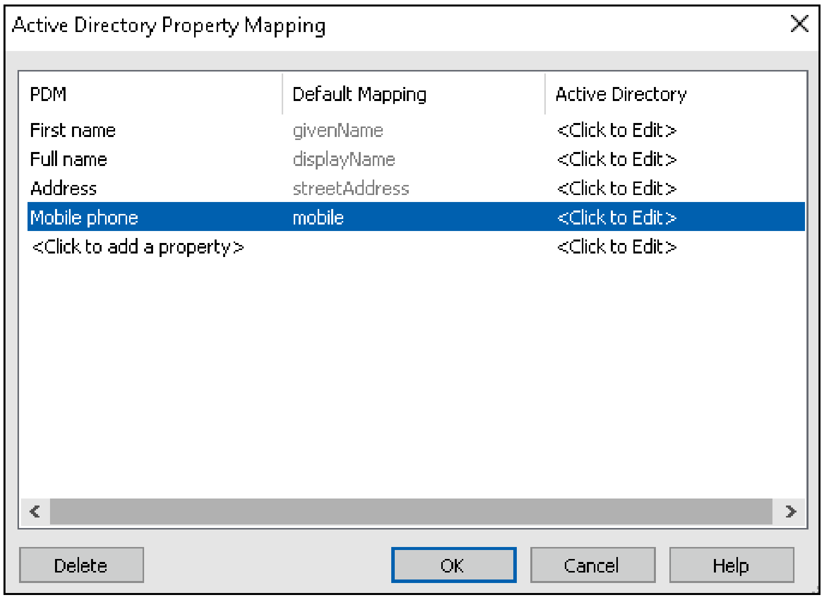
SOLIDWORKS® PDM is offered in two versions. SOLIDWORKS PDM Standard is included with SOLIDWORKS Professional and SOLIDWORKS Premium, and is available as a separately purchased license for non-SOLIDWORKS users. It offers standard data management capabilities for a small number of users.

SOLIDWORKS PDM Professional is a full-featured data management solution for a small and large number of users, and is available as a separately purchased license.



Video: What's New in SOLIDWORKS 2022 - SOLIDWORKS PDM

Integration with Microsoft Windows Active Directory



With improved integration with Microsoft® Windows Active Directory, you have more options to manage users and groups that use Windows login.

The User Properties dialog box has additional fields. You can override the default property mappings or map properties such as **User data** that are not mapped to any Active Directory attribute. In the Administration tool, right-click **Users and Groups Management** and click **Active Directory Property Mapping**.

- You must have **Can administrate users** administrative permission.
- You can map a user property to only one Active Directory attribute.

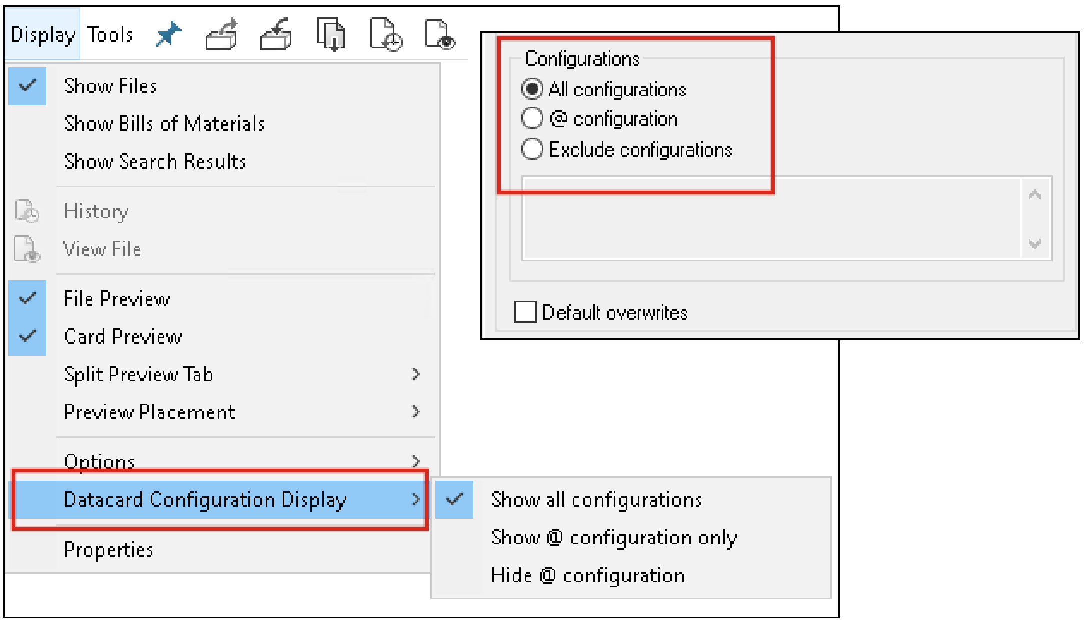
You can also:

- Import the user's profile picture from the Active Directory.
- View the profile picture in the SOLIDWORKS PDM menu bar if you have set it in the Active Directory. Hover over **Logged in as** to view the image in larger size.
- Edit the presence note directly from the user pop-up window.
- Identify Windows users and groups through an indicator in the icons .
- Validate Windows groups. In the Administration tool, under **Users and Groups Management**, right-click **Groups** and click **Validate Groups from Active Directory**.

If the group does not exist in the Active Directory, the icon displays with a red plus indicator . Right-click the group name and click **Information** to view details.

- Validate SOLIDWORKS PDM and Windows user logins. Right-click the user name and click **Validate Logins**. If SOLIDWORKS PDM user information is invalid, the user icon displays with a red lock indicator . Right-click the user name and click **Information** to view details.

Configuration Handling



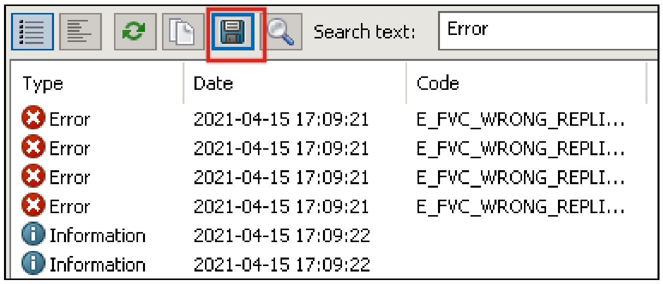
SOLIDWORKS PDM 2022 offers more control over configurations.

- For many card controls, you can assign a default value to automatically fill in the control variable when a user saves a new file or item or adds it to the vault. Previously, you could either apply default values to all configurations or use **Exclude configurations** to specify a list of configurations to exclude. You also have the ability to apply default values only for **@ configuration**.
- You can configure the display of configurations in data cards. In SOLIDWORKS PDM File Explorer, click **Display**, select **Datacard Configuration Display**, and select one of the following:
  - **Show all configurations**
  - **Show @ configuration only**
  - **Hide @ configuration**

  For SOLIDWORKS PDM 2021 or earlier client versions, if you selected **Hide @ configuration**, it remains selected after you upgrade to SOLIDWORKS PDM 2022.

  You can add **Datacard Configuration Display** to shortcut menus and SOLIDWORKS PDM menus located above the right pane. In the Administration tool, right-click a user or group and click **Settings**. Use the **Menus** page to add the **Datacard Configuration Display** command to the menus.

Exporting Archive Server and User Logs



You can export a summary of archive server and user logs in multiple file formats.

You can export the logs in:

- **Comma Separated Value Files (*.csv)**
- **JSON Files (*.json)**
- **Text Files (*.txt)**
- **All Files (*.*)**

Exporting Archive Server Logs

To export archive server logs:

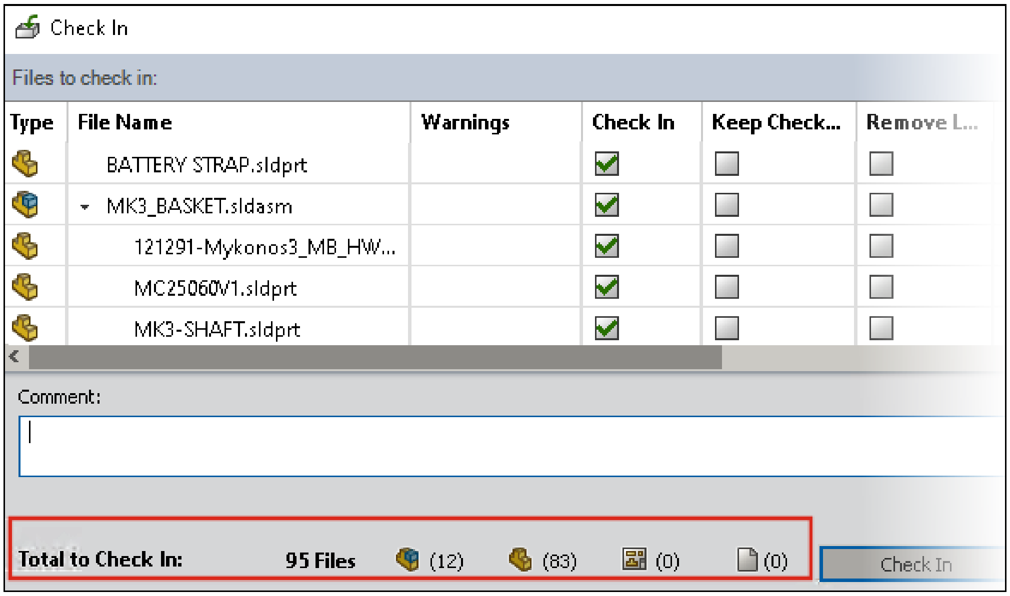
1. In the Administration tool, right-click archive server name and click **Show the Archive Server Log**.
2. In the Log File dialog box, click **Save As**.
3. In the Save As dialog box, enter a name in **File name**, select **Save as type**, and click **Save**.

Exporting User Logs

To export user logs:

1. In the Administration tool, under **Local Settings**, double-click **Log File**.
2. In the Log File dialog box, click **Save As**.
3. In the Save As dialog box, enter a file name in **File name**, select **Save as type**, and click **Save**.

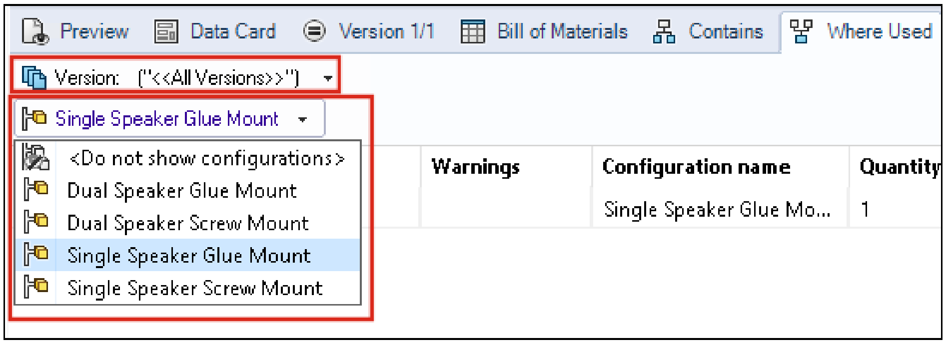
SOLIDWORKS PDM User Interface Enhancements



The SOLIDWORKS PDM user interface has improvements that enhance usability and readability.

- When you perform a file operation, you can see the number and type of files affected by that operation. This feature is available in the following dialog boxes:
  - Check In
  - Check Out
  - Undo Check Out
  - Change State
  - Get
  - Rollback
- You can resize columns of the variables area in the SOLIDWORKS PDM task pane add-in. This improves the readability of variables and values.
- The SOLIDWORKS PDM task pane add-in follows the color theme of SOLIDWORKS. When you hover over any row in the file list of the add-in, it is highlighted.

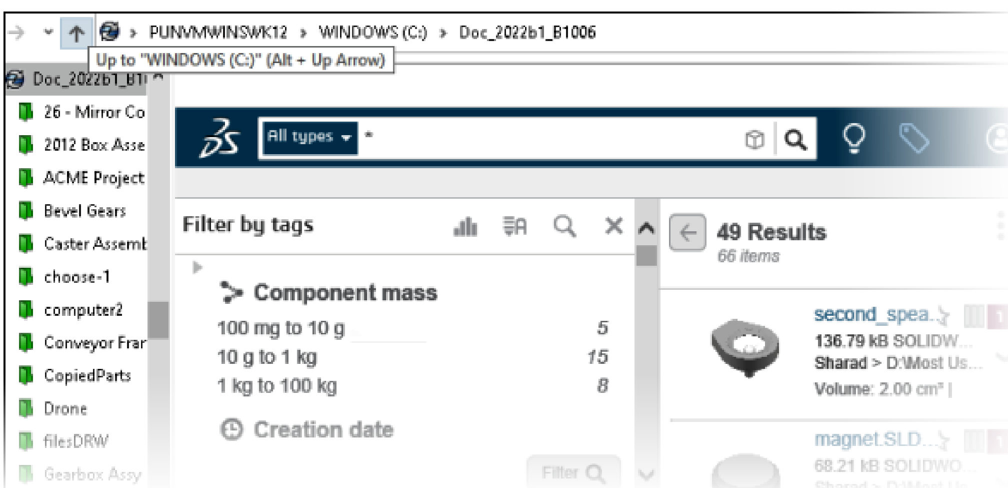
Viewing Configurations for All Versions in the Where Used Tab



You can view configurations of part or assembly files when you select **All Versions** in the Where Used tab.

The list contains all configurations across all versions of the selected file to which you have access permission. From the list, select a **Configuration** to view its references.

Using EXALEAD OnePart Search in SOLIDWORKS PDM



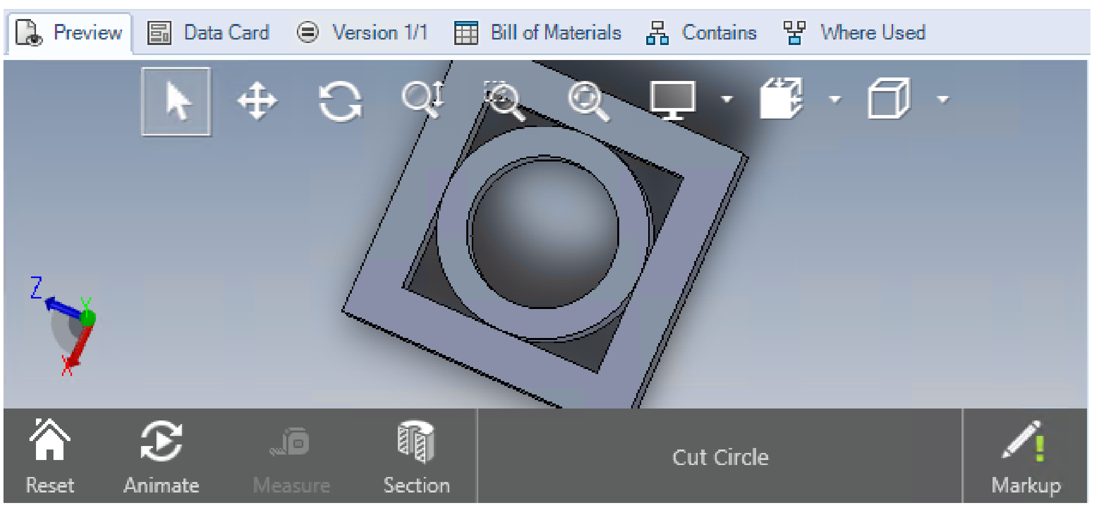
EXALEAD OnePart Search is now integrated in SOLIDWORKS PDM.

You must configure EXALEAD® OnePart in the Administration tool. Right-click **EXALEAD OnePart** and click **Open**. In the EXALEAD OnePart dialog box, enter **URL**, **Connection name**, and select **Protocol**.

- This functionality is available only in SOLIDWORKS PDM Professional.
- This requires an EXALEAD OnePart license. The SOLIDWORKS PDM Installation package does not include this license and you need to obtain it separately. Contact your SOLIDWORKS Value Added Reseller for more information.
- You must have **File vault management** administrative permission.

To perform a search in the SOLIDWORKS PDM File Explorer, click on the SOLIDWORKS PDM menu bar, and select **EXALEAD OnePart Search**. The EXALEAD OnePart Search user interface displays in an embedded browser control in the vault view.

SOLIDWORKS eDrawings Viewer in the Preview Tab



All of the view functions in eDrawings® Viewer are available in the Preview tab of SOLIDWORKS PDM File Explorer.

Option	Description
View SOLIDWORKS eDrawings toolbar	Click **Display** > **Options** > **Show full UI in SOLIDWORKS preview**.
View markup associated to a file	Displays a markup if the selected file contains a markup.

Option	Description
View eDrawings preview for SOLIDWORKS files	Click **Display** > **Options** > **Show bitmap for SOLIDWORKS files**. On the Preview tab, click **Load Preview**. Previously, you had to click the thumbnail preview image to access the eDrawings preview.

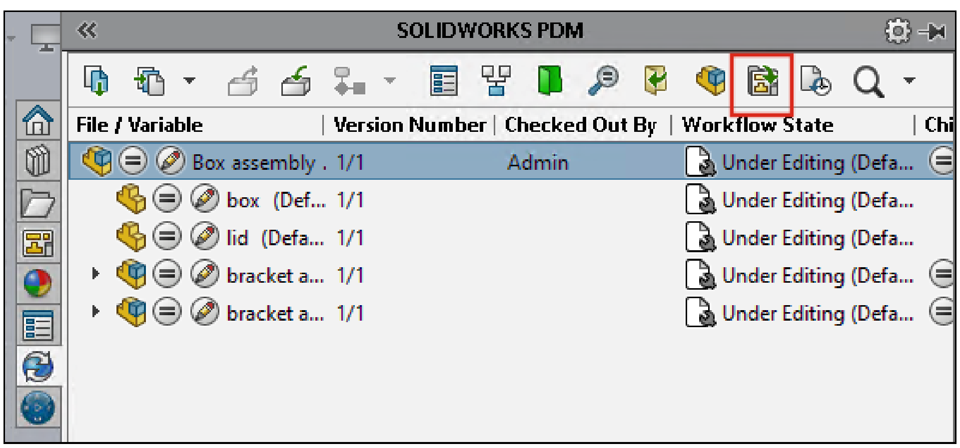
Support for Neutral CAD File Formats in eDrawings Web Preview

In SOLIDWORKS PDM Web2, you can open and view neutral file formats for eDrawings WebGL.

Support is available for the following file formats:

- `STEP (.step, .stp)`
- `IGES (.iges, .igs)`
- `Parasolid (.X_B, .X_T, .XMT, .XMT_TXT)`
- `JT (.JT)`
- `Acis Sat (.SAT, .SAB)`

Opening a Drawing from the SOLIDWORKS PDM Add-In



In the SOLIDWORKS PDM Add-in, you can open a drawing of a SOLIDWORKS part or assembly file.

To open a drawing, click  on the SOLIDWORKS PDM task pane or right-click the part or assembly and click **Open Drawing**.

The drawings may not be locally cached, can have a different name from the part or assembly file, and can exist in a different folder.

SOLIDWORKS PDM Performance Improvements

With SOLIDWORKS PDM 2022, you can experience improved performance of many file-based operations.

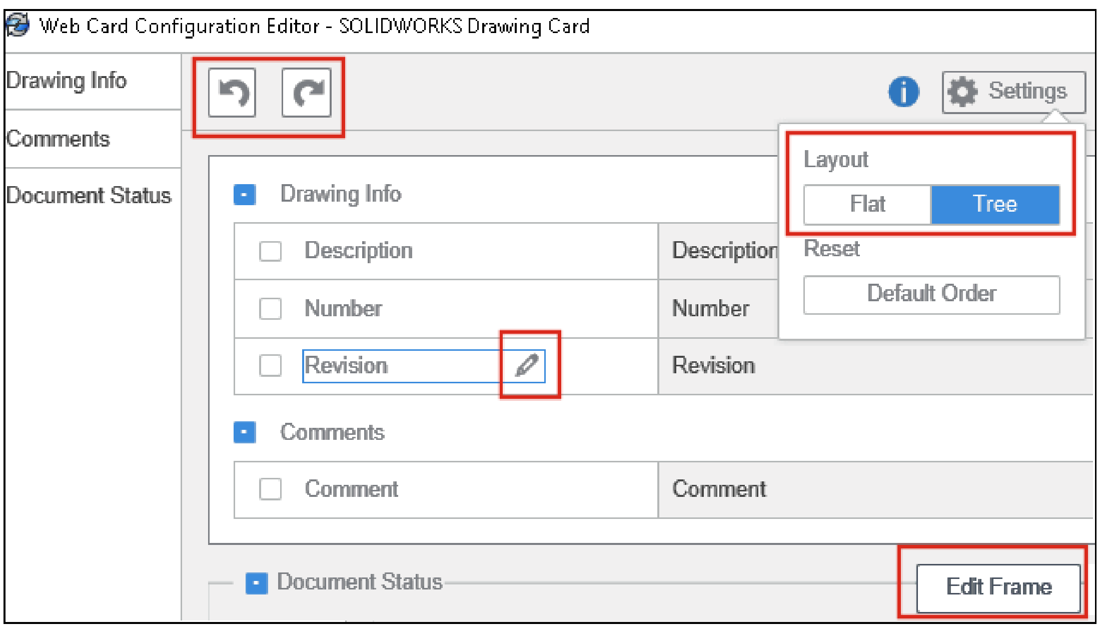
You can perform the following actions faster for database servers with high latency:

- Open files
- Display the Save As dialog box
- Copy Tree
- Create a document in SOLIDWORKS

SOLIDWORKS PDM has improved performance for the following:

- Saving a data card with a large number of file extensions is faster by 15% to 60%.
- Checking in a drawing with a large SOLIDWORKS bill of materials (BOM) is significantly quicker.
- Displaying files in the Where Used tab with the **Show All** option and additional custom columns is many times faster for certain vaults.
- Displaying the Transition dialog box for dynamic notifications is quicker.
- Loading a Web2 preview is between 1.5 and 2 times faster for large models.

Web2 Data Cards



SOLIDWORKS PDM offers more ways to configure the layout of the data card for Web2 and improve the readability of your data.

Available in SOLIDWORKS PDM Professional only.

In the Administration tool, in the Card Editor window, click **Configure card for web** on the Card Properties pane.

You can control the visibility of data card control for Web2. In the Card Editor window, in the right pane, under **Flags**, select **Show in web card**. This lists the data card control in the web configuration editor and displays it in the data card for Web2.

- Available only for file cards.
- Available only for the following data card controls:
  - **Checkbox**
  - **Combobox Dropdown**
  - **Combobox Droplist**
  - **Combobox Simple**
  - **Edit**
  - **List**
  - **Date Field**
  - **Radio Button**

In the web configuration editor of data cards, you can:

- Click **Settings** and specify the layout of the data card controls as **Flat** or **Tree**.
- Edit labels and change the order of controls.

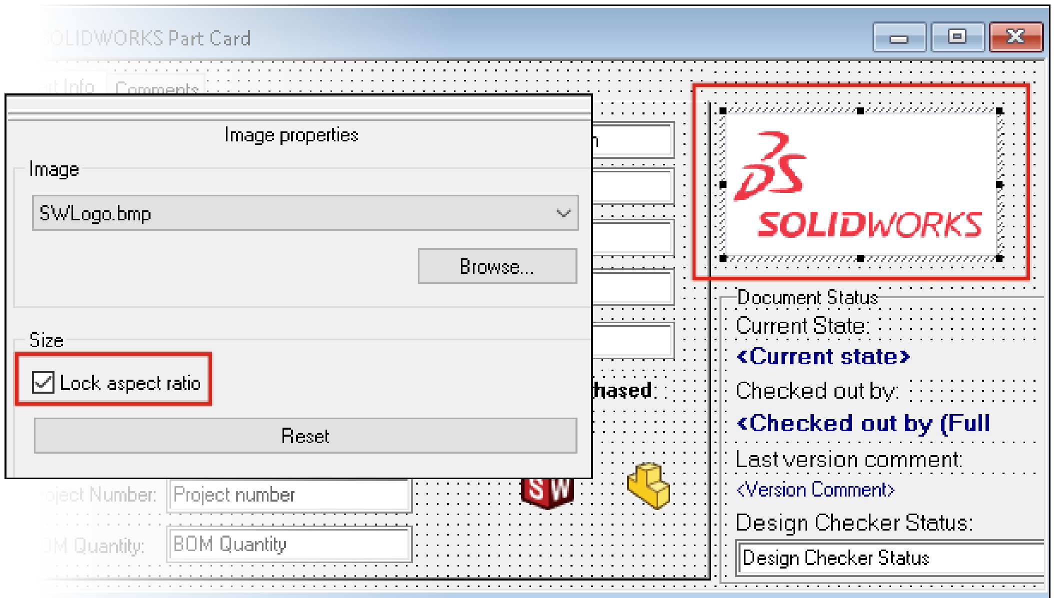
  You cannot add or remove controls from tabs. To do so, use the main card editor.

- Add controls in a frame in the **Tree** layout and define their order of appearance.

The user interface of Web2 data card has the following improvements:

- Navigation control for **Tree** layout
- Date picker to specify DATE type controls
- Multiline box to enter descriptions
- Radio button and check box controls

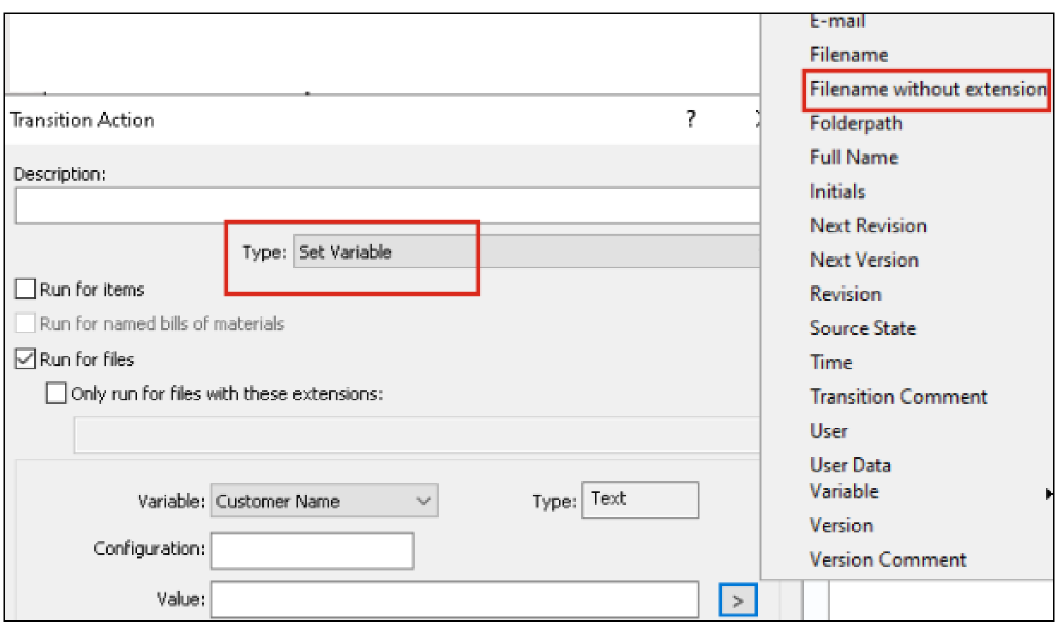
Resizing an Image in a Data Card



In the Administration tool, you can resize an image on a data card by dragging the control handles.

In the Card Editor, in the **Image properties** pane, select **Lock aspect ratio** to maintain the image aspect ratio. Click **Reset** to restore the image to its original size.

Other SOLIDWORKS PDM Enhancements



SOLIDWORKS PDM 2022 has new APIs and other improvements.

- You can save a card that has more than 2000 characters in an alias in an input formula.
- In the Transition Action dialog box, when you specify **Type** as **Set Variable**, you can define the value of the selected variable as **Filename without extension**.

SOLIDWORKS PDM APIs are available to:

- Define **Next counter value** for serial numbers in the Administration tool.
- Add or modify some of the user settings.
- Edit the name of a named BOM.
- Select files to destroy from the deleted files.

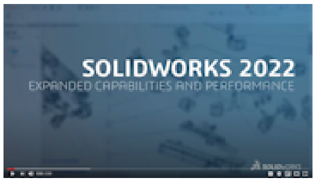
15

SOLIDWORKS Manage

This chapter includes the following topics:

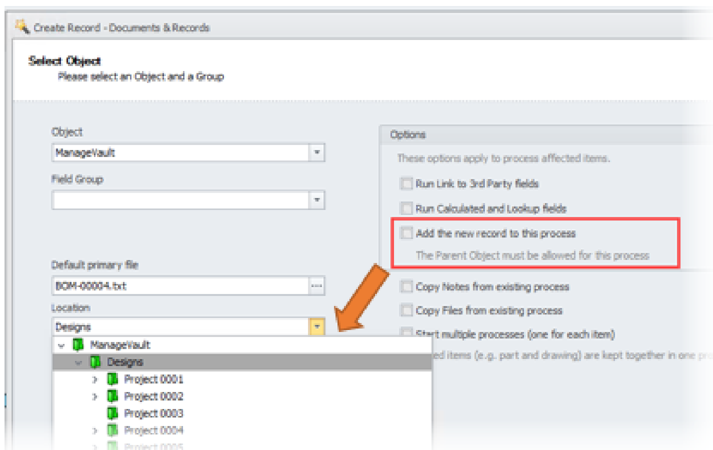
SOLIDWORKS® Manage is an advanced data management system that extends the global file management and application integrations enabled by SOLIDWORKS PDM Professional.

SOLIDWORKS Manage is the key element in providing Distributed Data Management.



Video: What's New in SOLIDWORKS 2022 - SOLIDWORKS Manage

Create Record Process Output

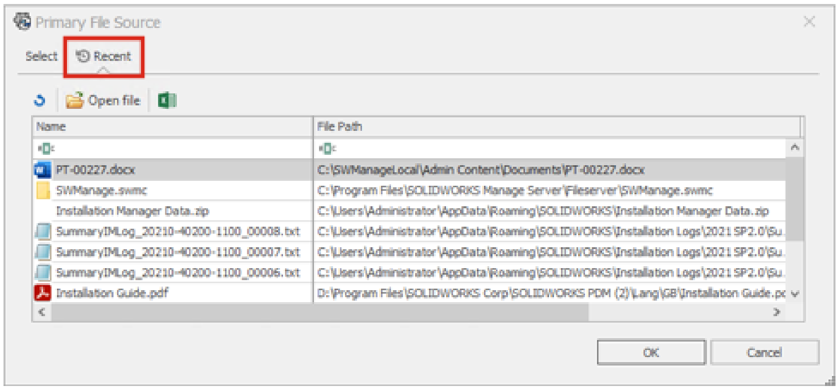


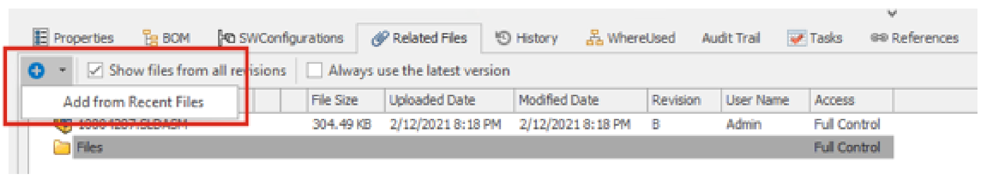
The Create Record process outputs have improved with new record attachments and subfolder locations.

You can:

- Attach a new record created by a Create Record process output as an affected item in the process.
- Specify a subfolder in which the new record is created.

Recent Files

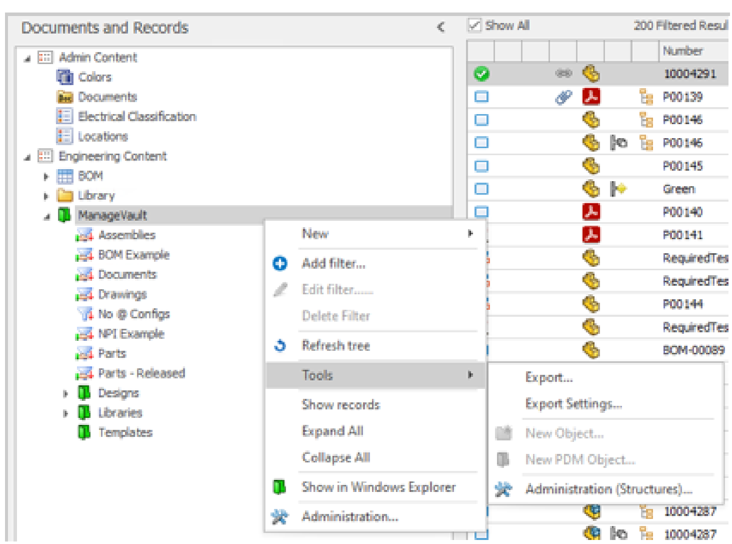




When you add new file data, you can choose from a list of recently accessed files in Windows. This is a convenient way to add data by not having to browse to recent locations.

Click the Recent tab in the Primary File Source dialog box when creating a new document record or click **Add** > **Add from Recent Files** when adding related files.

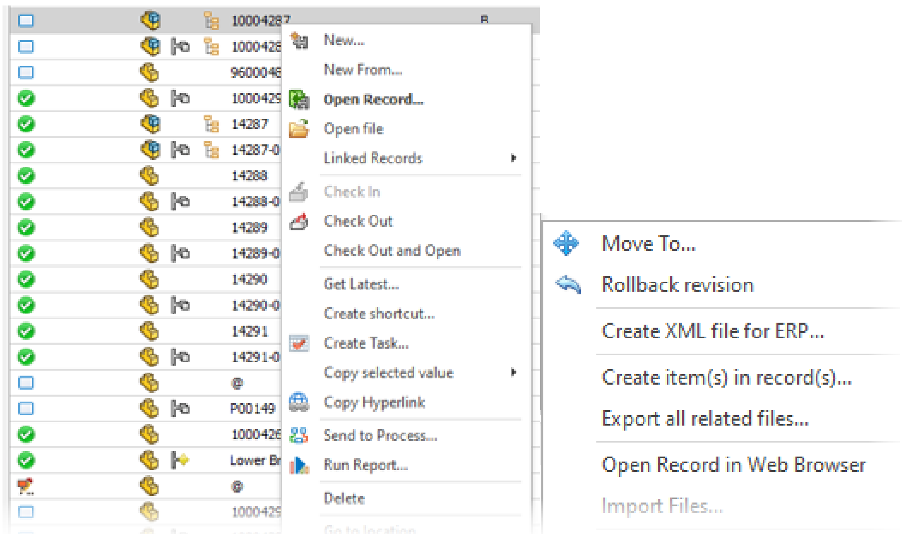
Object Structure Editor



For ease of use when administrators work with object structures, you can access the Administration (Structures) dialog box directly from the user interface.

In the Module Objects area, right-click and click **Tools** > **Administration (Structures)**.

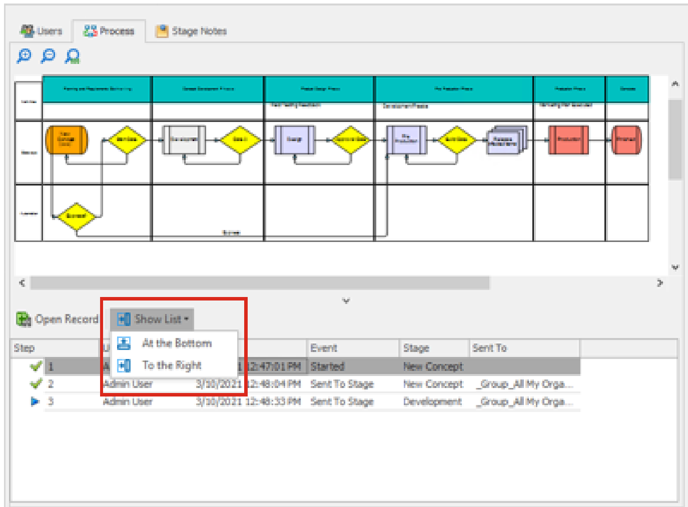
Record Hyperlinks



You can copy a hyperlink to a record that you can copy into a document or email, to provide quick access through the Plenary Web client. Right-click and click **Tools** > **Copy Hyperlink**.

You can also open the record in the web client directly from the desktop client. Right-click and click **Tools** > **Open Record in Web Browser**.

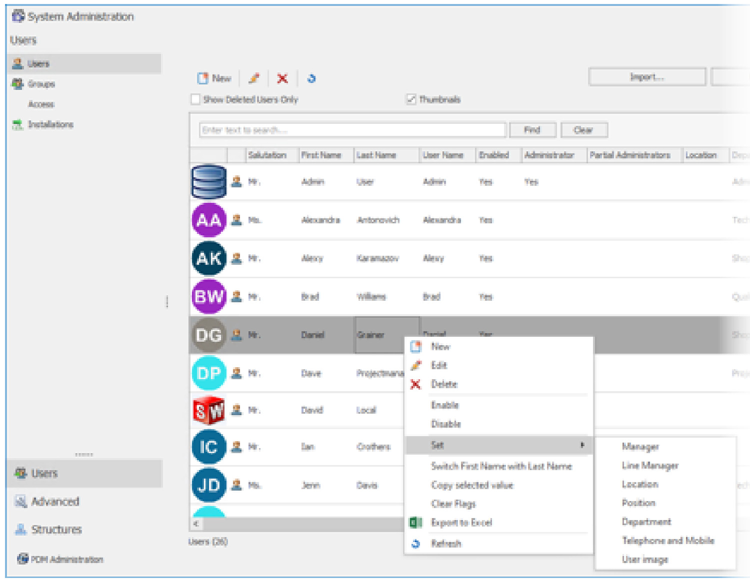
User Interface



The SOLIDWORKS Manage user interface has many improvements that create consistency and a better user experience.

Functionality	Improvement
Bill of Materials (BOM) layout	The controls in the BOM interface are modernized and consistent.
BOM flyout panel	The flyout panel in the BOM tab is reorganized. You can collapse different areas, providing a better view of information.
SOLIDWORKS Add-in	You can reorder columns in the Open and Structure tabs of the SOLIDWORKS Manage Add-in.
Where Used As Field tab	The object grouping displayed in the Where Used As Field tab (usually displayed as the Referenced tab) shows the associated icon, module name, and number of results for each module type.
Process tab	You can specify the position of the history lines in the Process tab to display at the bottom or on the right side. This improves screen usage depending on the layout of the process diagram.
Project Properties card	The project Properties card displays a yellow information bar for consistency with other record types. You can collapse the system property and thumbnail image area to provide more space.

Avatar Images and Icons



To make users more recognizable in SOLIDWORKS Manage, you can add default avatar icons for users. Avatar images are displayed on the Users tab of the Administration tool.

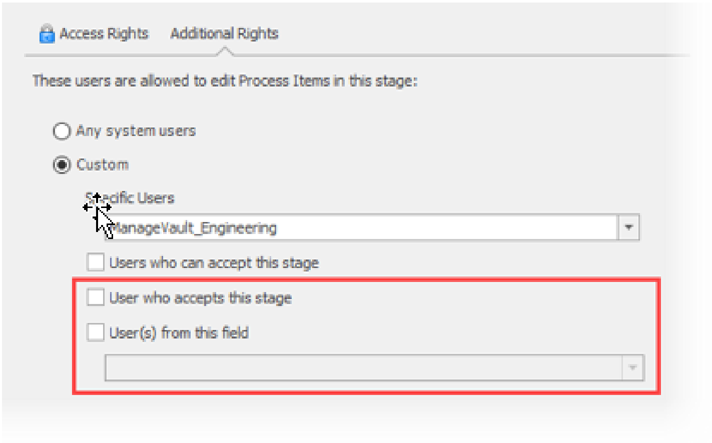
To specify avatar images and icons:

1. In the Administration tool, click the Users tab.
2. Right-click and click **Set** > **User image**.
3. In the dialog box, specify options:
   - **Create new image for all selected users**
   - **Create new image for selected users without an image**
4. Click **Apply**.

Plenary Web Client

You can log in to the Plenary Web Client with a SOLIDWORKS PDM username and password. Previously you had to use a SOLIDWORKS Manage password, then separately log in to a SOLIDWORKS PDM object.

Check Out Rights for Affected Items

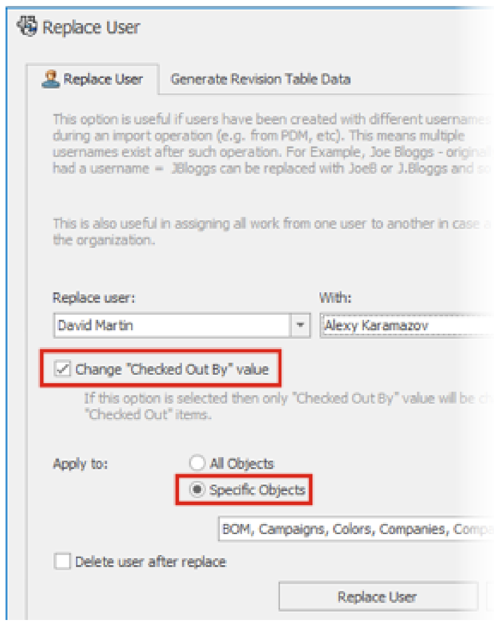


Process administrators have additional controls over who can check out affected items while they are in a process.

Options include:

- **User who accepts this stage**. Limits check out rights to a single user who accepts the stage.
- **User(s) from this field**. Allows administrators to specify check out rights to users for a specified object type field defined for the process.

Replace User

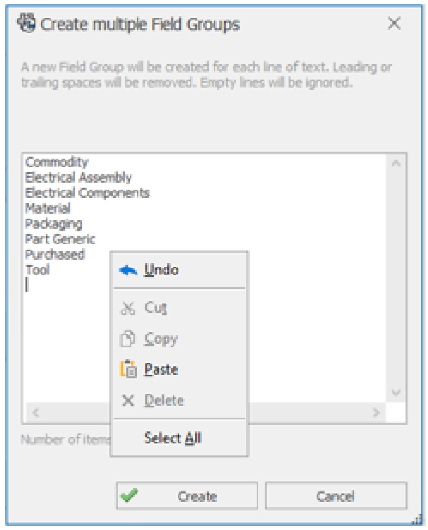


You can select specific objects to replace users and specify the **Change "Checked Out By" value** for someone who is replacing the selected user.

Replacing users is helpful when importing data where there are duplicate usernames. It is also useful when a user leaves the company and you need to reassign their work to another user.

You cannot replace users for SOLIDWORKS PDM objects.

Creating and Deleting Multiple Field Groups

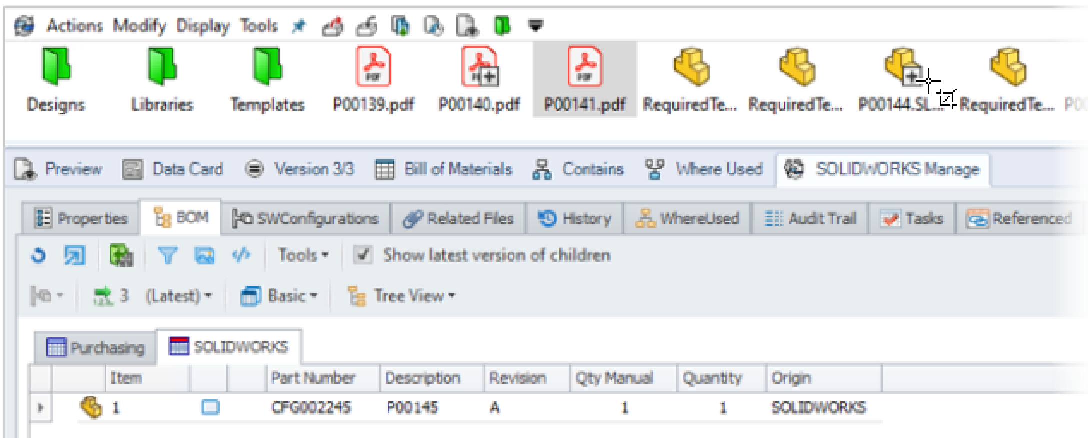


You can create multiple field groups at once, by entering multiple lines or copying and pasting from a text-based file (such as `.xlsx`, `.txt`, or `.csv`). You can **Shift +** or **Ctrl + select** multiple field groups to delete them.

To create multiple field groups:

1. In the Field Groups dialog box, click the New tab and click **Create multiple Field Groups**.
2. In the Create multiple Field Groups dialog box, copy and paste field groups from a text-based file.
3. Click .

SOLIDWORKS PDM User-Defined References



There is expanded support for displaying SOLIDWORKS PDM user-defined references (**Paste as Reference**).

You can include references between:

- SOLIDWORKS part files and other SOLIDWORKS part files
- Non-SOLIDWORKS files (such as Microsoft® Word documents) and SOLIDWORKS part files

SOLIDWORKS Manage Performance Improvements

SOLIDWORKS Manage 2022 offers improved performance for enhancing the user experience.

Functionality	Performance Improvement
Bill of material (BOM) display	When specifying the **Number of BOM levels to display** options at `1`, large BOMs display up to five times faster. For BOMs with **Link to 3rd Party** fields configured, the time required to calculate the values has decreased.
Projects	For projects with a high number of stages or Tasks, the Gantt chart display is faster than in previous releases.
Check out/check in of SOLIDWORKS PDM files from SOLIDWORKS Manage	In previous releases, the **Check Out**/**Check In** operations refreshed the entire grid in the background. Now, only the individual line item that last changed refreshes, making performance faster.

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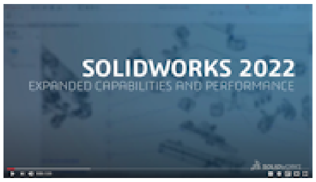
SOLIDWORKS Simulation

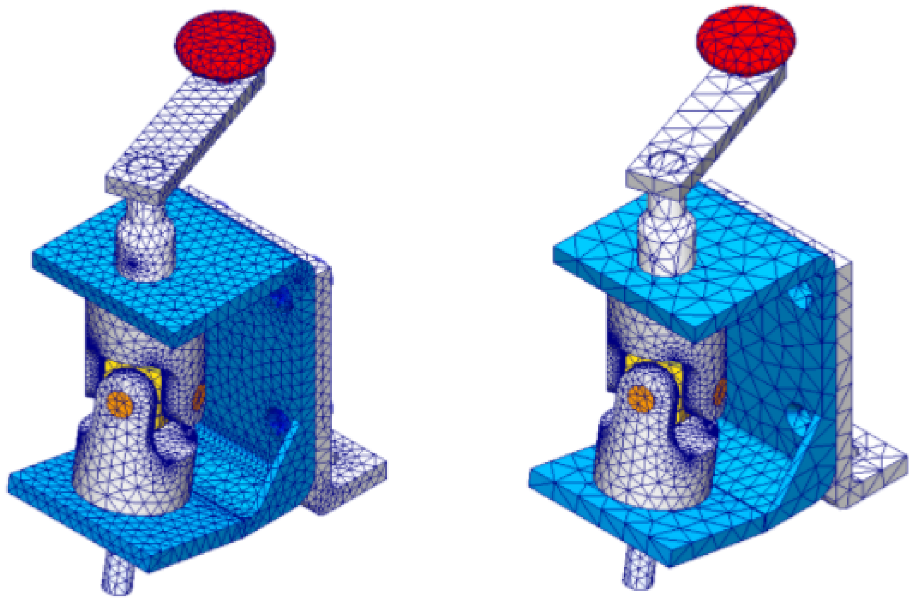
This chapter includes the following topics:

- **Blended Curvature-Based Mesher**
- **Bonding and Contact Architecture**
- **Linkage Rod Connector**
- **Simulation Solvers**
- **Simulation Performance**

SOLIDWORKS® Simulation Standard, SOLIDWORKS Simulation Professional, and SOLIDWORKS Simulation Premium are separately purchased products that you can use with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium.

	**Video: What's New in SOLIDWORKS 2022 - SOLIDWORKS Simulation**

Blended Curvature-Based Mesher



Using the Blended Curvature-based mesher, you can apply mesh control that has a larger element size than the global mesh size.

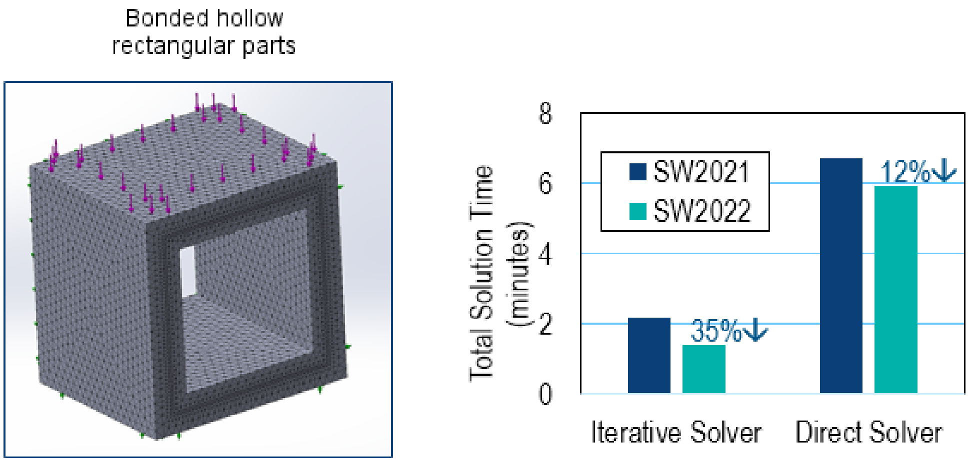
Create a coarser mesh for bodies that may be not critical for the simulation to reduce the analysis time. Previously, you could only apply a mesh control to refine the mesh of selected bodies and geometric entities.

After creating a mesh, you can access the Mesh summary, which reports the size assignments of individual bodies and geometric entities. In a Simulation study tree, right-click **Mesh** and click **Summary**.

The Blended Curvature-based mesher is the default mesher for new simulation studies.

> The option **Calculate Minimum Element Size** (available for the Blended curvature-based mesher) is removed from the Mesh PropertyManager.

Bonding and Contact Architecture

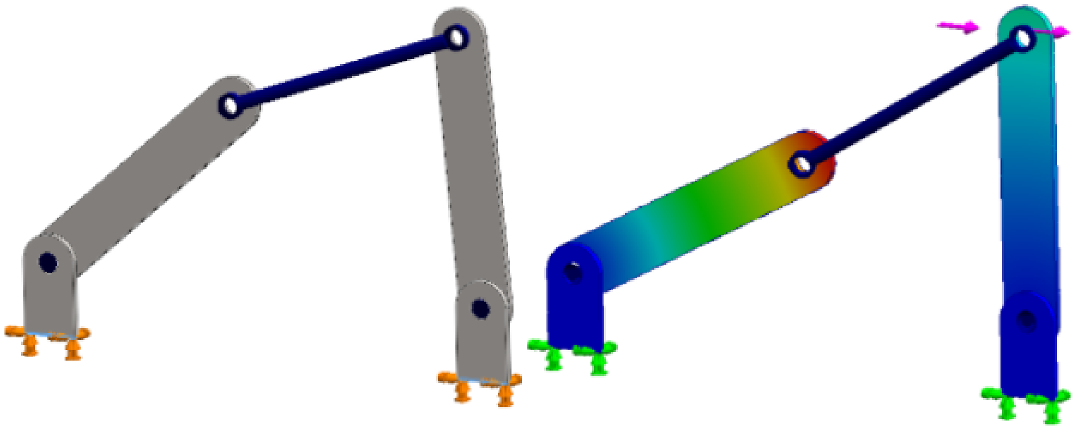


Several enhancements to the bonding and contact architecture improve the overall performance and accuracy of the simulations.

- Removal of duplicate degrees of freedom in bonding and contact constraint equations
- Reduced bonding and contact constraint equations
- Measurement of bonding and contact strains in units of distance rather than volume
- Improved calculation for the constraint area
- Optimization of unitless parameter associated to bonding and contact penalty stiffness
- Elimination of unnecessary function calls for small tasks within the contact search code

Performance improvements based on these enhancements are more noticeable for simulations that run with the **FFEPlus** iterative solver and have a large percentage of nodes participating in bonded and contact interactions.

Linkage Rod Connector



You can specify a **Linkage Rod** connector between cylindrical faces, circular edges (for shells), or vertices to model the behavior of connecting rods.

The **Linkage Rod** connector is available with SOLIDWORKS Simulation Professional and SOLIDWORKS Simulation Premium.

To open the Linkage Rod PropertyManager:

In the Simulation study tree, right-click **Connections** and click **Linkage Rod**.

The following table describes the key options of the Linkage Rod PropertyManager.

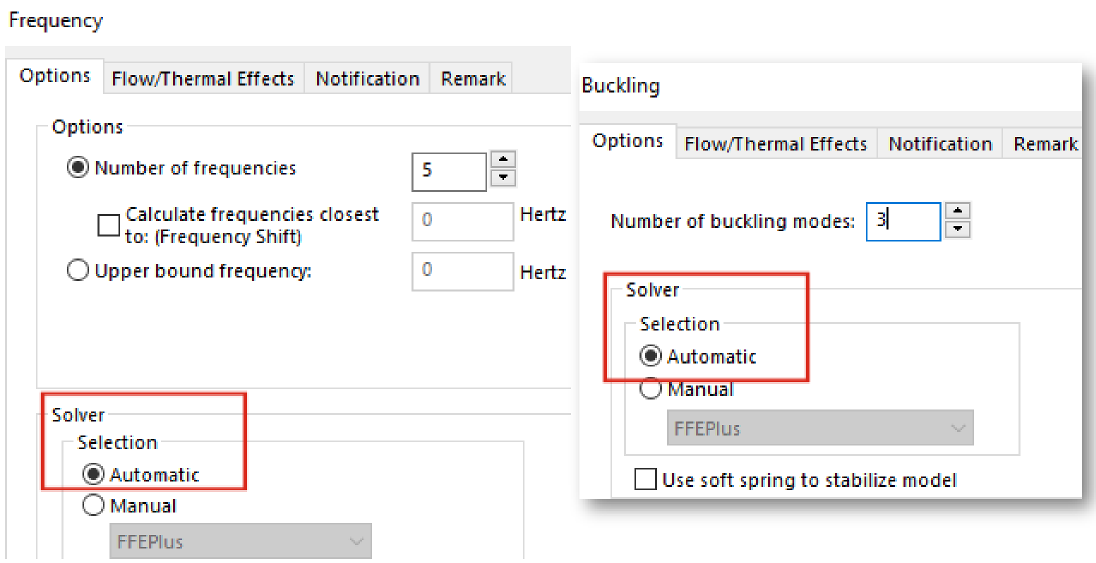
	Concentric cylindrical faces or edges (for shells)	Specifies cylindrical faces or shell edges to attach a linkage rod connector. You select two geometric entities to position the end joints of the connector.
	Vertex	Specifies two vertices to attach a linkage rod connector. You can also select a vertex for one end joint of the connector and a cylindrical face or shell edge for the other end joint.
	Rigid joint	Specifies the connector's end joint to a rigid joint. A rigid joint prevents any rotations or deformations. A linkage rod connector with rigid joints can transfer all moments from one part to another.

	Pivot joint	Specifies the connector's end joint to a pivot joint. A pivot joint allows only one rotation about the axis normal to the connector's axis.
	Spherical joint	Specifies the connector's end joint to a spherical joint. A spherical joint acts like a ball and socket joint where the ball can rotate inside the socket, but it cannot dislocate from the socket.
	Offset	Specifies an offset distance to position the end joints of the connector. You can only select cylindrical faces or circular edges to define an offset distance.
	Section Parameters	Specifies the cross section geometry of the connector: • **Solid circular** • **Hollow circular** • **Solid rectangular** • **Hollow rectangular**
	Material	Applies a material to the connector from the SOLIDWORKS material library or applies a custom material.

You can list a linkage rod connector's forces such as shear force, axial force, bending moments, and torque after you run a simulation. Right-click **Results** and click **List Connector Force**.

> The linkage rod connector is not available for nonlinear and thermal studies.

Simulation Solvers



Function-based processing for the FFEPlus iterative and Intel Direct Sparse solvers is extended to simulation studies that include connectors and other features. The automatic solver selection is extended to Nonlinear, Frequency, and Buckling studies.

- **FFEPlus iterative** and **Intel Direct Sparse** solvers

  The transfer of stiffness data to solve the systems of equations is optimized because file-based processing is replaced with function-based processing. Performance is improved for simulations that contain:

  - Connectors: spring, bearing, bolt, and rigid
  - Cyclic symmetry, remote load with rigid connection, and beams acting as stiffeners.

- **Automatic Solver Selection**

  The algorithm that selects the best equation solver is improved to include Nonlinear, Frequency, and Buckling studies. The selection of the best equation solver (Intel Direct Sparse or FFEPlus Iterative) depends on the number of equations, load cases, mesh type, geometric features, contact and connector features, and available system memory.

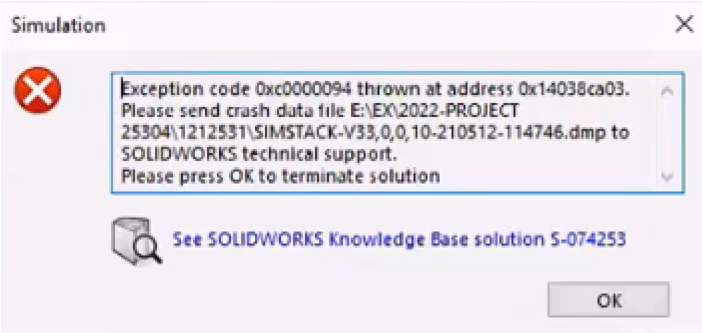
  For Frequency studies, in addition to the previously mentioned parameters, the algorithm also factors in the number of frequencies. For Buckling studies, it factors in the number of modes.

- **Intel Direct Sparse** solver for Linear Dynamic studies

  You can use the **Intel Direct Sparse** solver for Linear Dynamic studies with **Selected Base Excitation** for frequency and response calculations.

- If you experience a solver failure while running a simulation, SOLIDWORKS Simulation prompts you to send the file that records information related to the solver failure to the technical support team.

The development team can extract information from the modules that caused the solver failure based on the data from the `SIMSTACK-*.dmp` file without using any additional information. The benefit of this enhancement is that you do not need to share confidential model data to troubleshoot a simulation solver failure.



Simulation Performance

It takes less time to save SOLIDWORKS models that have simulation studies.

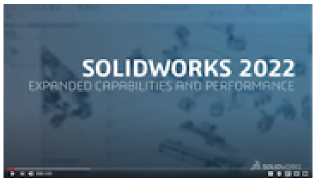
Saving models that have simulation studies is faster, if at least one or more simulation studies are not modified.

17

SOLIDWORKS Visualize

This chapter includes the following topics:

SOLIDWORKS® Visualize is a separately purchased product that you can use with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium, or as a completely separate application.

	**Video: What's New in SOLIDWORKS 2022 - SOLIDWORKS Visualize**

Match Camera Perspective to Backplate

Match Camera off

Match Camera on

With the **Match Camera** tool, you can manipulate a camera by aligning vanishing lines to a backplate image. This allows more precise positioning of a model in front of a backplate.

A common challenge when placing a model in front of a background image (backplate) is that you have to manually adjust the extrinsic (position and orientation) and intrinsic (perspective and focal length) camera parameters so the composition looks plausible. In SOLIDWORKS Visualize Professional, the **Match Camera** tool helps you find the perfect camera parameters for real world photographs.

Click **Tools** > **Match Camera**.

Using the Match Camera Tool

To use the Match Camera tool:

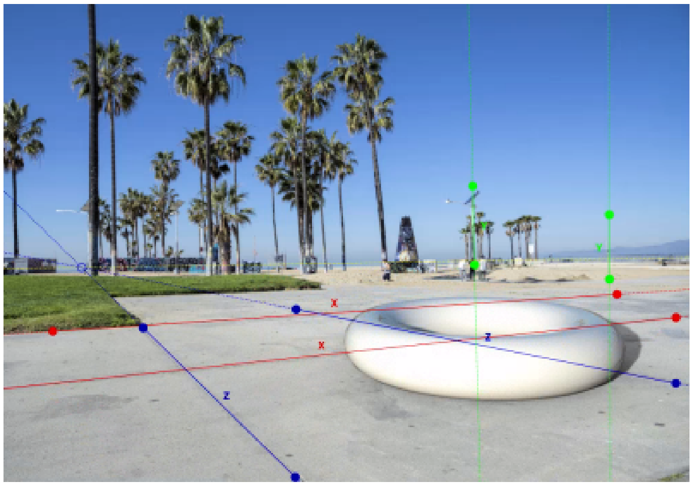
1. In SOLIDWORKS Visualize Professional, open a project that has a backplate.

2. Click **Tools** > **Match Camera**.

   Solid and dashed lines appear over the backplate image.

3. In the Match Camera dialog box, specify options.

4. In the 3D Viewport, align the lines and points with the backplate's vanishing lines.



Line/Point Type	Description
Red solid line and solid points	Vanishing line in the X direction (horizontal). Drag the line or its points to match the backplate image.
Red dashed lines and hollow points	Extension of the vanishing line in the X direction beyond its endpoints. The hollow red point is the vanishing point on the horizon line (which may be outside of the 3D Viewport).
Blue solid line and solid points	Vanishing line in Z direction (horizontal).Drag the line or its points to match the backplate image.
Blue dashed line and hollow points.	Extension of the vanishing line in Z direction beyond its endpoints. The hollow blue point is the vanishing point on the horizon line (which may be outside of the 3D Viewport).
Green solid line and solid points	Vanishing line in Y direction (vertical). Drag the line or its points to match the backplate image. This is optional and only displayed when you specify **Vanishing Points** as **Three**. The vertical vanishing line does not result in a vanishing point on the horizon line.
Yellow dashed line	Horizon line that connects the two (horizontal) vanishing points. Match the horizon line to the actual horizon in the backplate image. It validates the result.

5. Click **OK**.

Match Camera Dialog Box

To access this dialog box:

1. In SOLIDWORKS Visualize Professional, open a project that has a backplate.
2. Click **Tools** > **Match Camera**.

Vanishing Points	Specifies **Two** dimensions (horizontal) or **Three** dimensions (horizontal and vertical axes). **Two** is recommended.
Reference Axes	Flips the reference axes. Depending on the order and direction of the axes, the **Camera Match** tool may result in a camera that is oriented upside down.
Lock Rotation	Locks the camera longitude so that the object remains focused from the same angle.
Display Options	• **Horizon**. Shows the horizon line (yellow dashed line). • **Line Extensions**. Shows the dashed extensions of the vanishing lines (red, blue, green lines). • **Vanishing Points**. Shows the vanishing points (hollow red and blue points) on the **Horizon**.
Reset	Resets the vanishing lines and points to their default locations.
Status	Indicates whether the match camera solution is mathematically solvable. If **Undefined**, the camera properties remain unchanged.

Shadow Catcher Property

Shadow catcher off

Shadow catcher on

You can turn any part object geometry into a shadow catcher object. Shadow catchers show the background through them and display shadows cast from lighting in the scene. For example, you can create a wall and specify it as a shadow catcher.

The shadow catcher object allows control over the shadow intensity, reflection and reflection roughness.

Shadow catcher is only available:

- In SOLIDWORKS Visualize Professional or SOLIDWORKS Connected.
- When you specify **Rendering Selection** as **Accurate**.

Using a Shadow Catcher

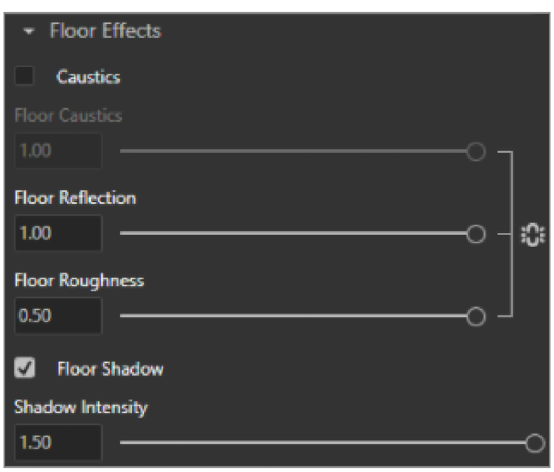
To use a shadow catcher:

1. In the Palette, on the Models tab, select a part in the Model tree to use as a shadow catcher.

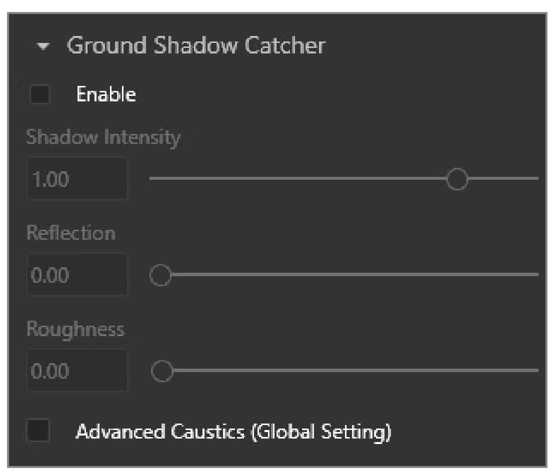
2. On the General subtab, under **Shadow Catcher**, specify:
   - **Enable**. Turns on the shadow catcher functionality.
   - **Shadow Intensity**. Lightens or darkens the shadow.
   - **Reflection**. Specifies the amount of reflectivity on the shadow catcher object.
   - **Roughness**. Specifies how blurry the reflections appear for **Reflections** greater than `0`.
   - **Advanced Caustics**. Improves the quality of caustic reflections in some scenes. This applies globally in the scene (not per shadow catcher object). An example is the shifting patterns of light and shade cast on the floor of a swimming pool on a sunny day.

	
A plane is at the back wall of the stage in the backplate image	When you specify the plane as a shadow catcher, you see the backplate image with the shadow cast more naturally

Scenes Tab



2021



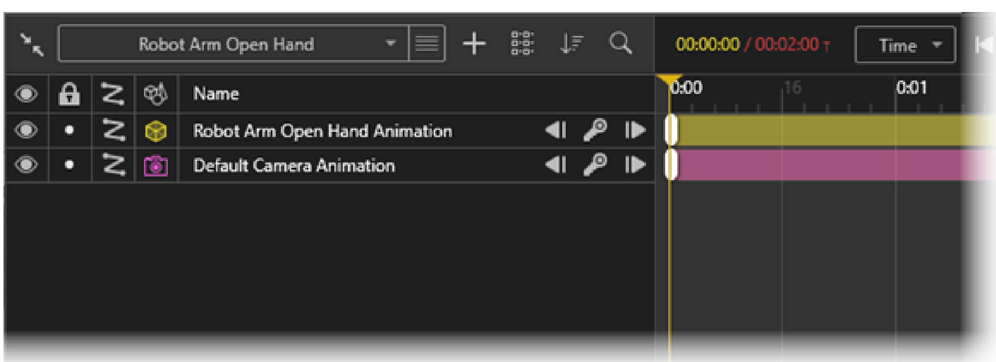
2022

Floor Effects parameters on the Scenes tab have been renamed for consistency with the shadow catcher functionality.

The following updates are available in the Palette, on the Scenes tab, on the Advanced subtab.

2021 User Interface	2022 User Interface
Floor Effects	**Ground Shadow Catcher**
Caustics	**Advanced Caustics**
Floor Caustics	Removed
Floor Reflection	**Reflection**
Floor Roughness	**Roughness**
Floor Shadow	**Enable**

Animations



Animations are improved in areas such as motion studies, organization, keyframes, and cameras.

Improvements include:

- Support for multiple motion studies per model set. You can import multiple SOLIDWORKS motion studies for SOLIDWORKS parts and assemblies. This is helpful because you:
  - Do not have to reanimate the part or assembly in SOLIDWORKS Visualize.

- Can use tools such as **Mate Controller** and flexible subassemblies.

- You can organize animated items in each animation with grouping functionality. You can also use tools to sort and filter the animation timeline.
- Suppress or unsuppress individual keyframes in animations and copy keyframe properties from one keyframe to another.
- Use multiple cameras in animation sequences to switch among multiple cameras during animation playback.

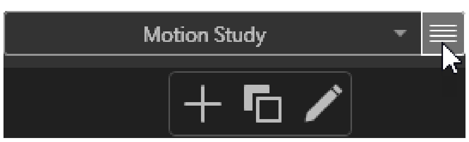
Animation List User Interface

The user interface for SOLIDWORKS Visualize animation list has additional functionality and is updated for ease of use.

Tools

Updates to animation tools:

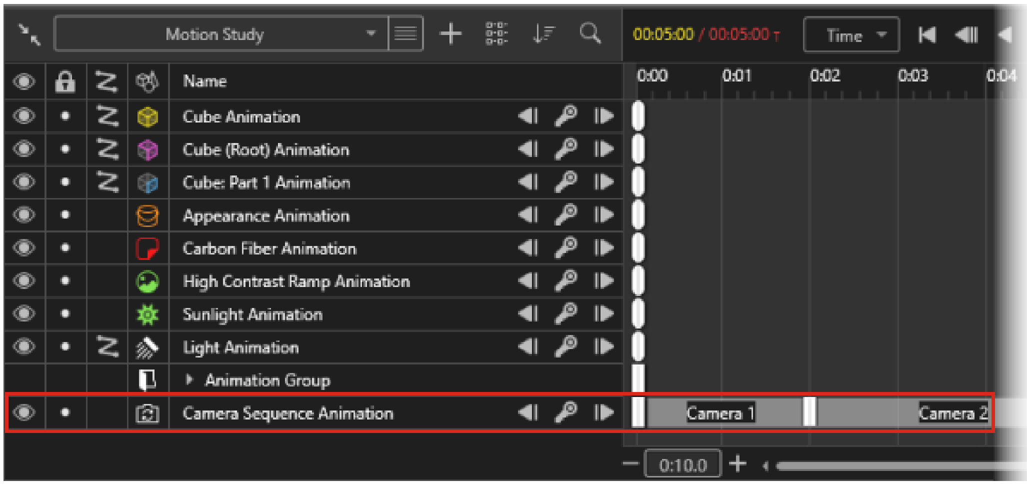
- Basic animation tools. Located at the top center of the animation timeline.
- Advanced animation tools. Located at the upper right of the animation timeline.
- Motion study tools. Above the animation list, click the **Motion Study** list to create a new motion study or change to a different motion study.



On the menu  next to the **Motion Study** list, you can:

- **Adds a new item** . Creates a new motion study. Subsequent animation editing applies to the new motion study only.
- **Duplicates the current item** . Copies the active motion study.
- **Edit the name of the current item** . Renames the active motion study.

- Camera switch animations. You can create a **Camera Sequence** layer to switch among multiple cameras during animation playback.



Animation list

Updates to the animation list:

1. **Add** . Lets you:
   - **Create New Group**. Creates a new group into which you can drag animation tracks.
   - **Add to New Group**. Adds selected animation tracks into a group.
   - **Create New Camera Sequence Animation**. Creates a camera switch so you can assign or switch cameras in the animation timeline. The camera automatically switches during animation playback.
2. **Change View** . Increases (**Large List**) or decreases (**Small List**) the animation timeline icons and font.
3. **Change Sort Mode** . Sorts the animations based on **Name**, **Type**, and in an ascending or descending order. You can also filter object types in the animation list by **Decals**, **Environment**, and so on.
4. **Search** . Narrows the animation list when you enter search criteria.
5. **Animated Object Types** . Displays a column with an icon representing the animated object type, such as **Model** , **Group** , and **Part** .
6. Animation track colors. Assigns colors to the animation tracks depending on the object type. You can change the colors in Animation Properties.
7. Selection. If you select an animation track, the animated object highlights in the viewport and Model tab, and vice versa.

Keyframes

Updates to keyframes:

- **Suppress Keyframe**/**Unsuppress Keyframe**. Suppresses or unsuppresses animation keys. On the timeline, right-click an animation key and click **Suppress Keyframe** or **Unsuppress Keyframe**. When suppressed, the animation keys are ignored during playback..
- **Transition**, **Tension**, and **Motion Ease**.
  - In the Keyframe Properties dialog box, you can specify numerical values for **Tension** and **Motion Ease**. You can specify their default keyframe properties in **Tools** > **Options** > **User Interface**.
  - You can copy/paste keyframe values across other animations. Right-click a keyframe and click **Copy Settings**. Right-click another keyframe and click **Paste Settings**. In the Paste Settings dialog box, you can select **Transition**, **Tension**, and **Motion Ease** settings to paste.

Render Output Viewer

You can use the render Output Viewer to manage render jobs and their respective output on a per project basis. You can recall (or re-execute) any previous rendering done in a project, directly from the render Output Viewer.

Previously, rendered jobs were not stored with the project and thumbnails were not available for viewing. This made it difficult to share renderings among users. With the render Output Viewer:

- Referenced rendered output remains when you save a project. For example, if you send the project to another user, that user can see the rendered output content in the Output Viewer.
- Render jobs can include an individual image or multiple rendered images. Content includes:

- Single image renders
- 360 camera renders
- Rendered layers (such as **Albedo**, **Alpha**, and **Depth**)
- Configurations
- All camera renders

- You can scroll through thumbnails of rendered output content and multiselect rendered content to publish to 3DSpace or 3DSwym®.

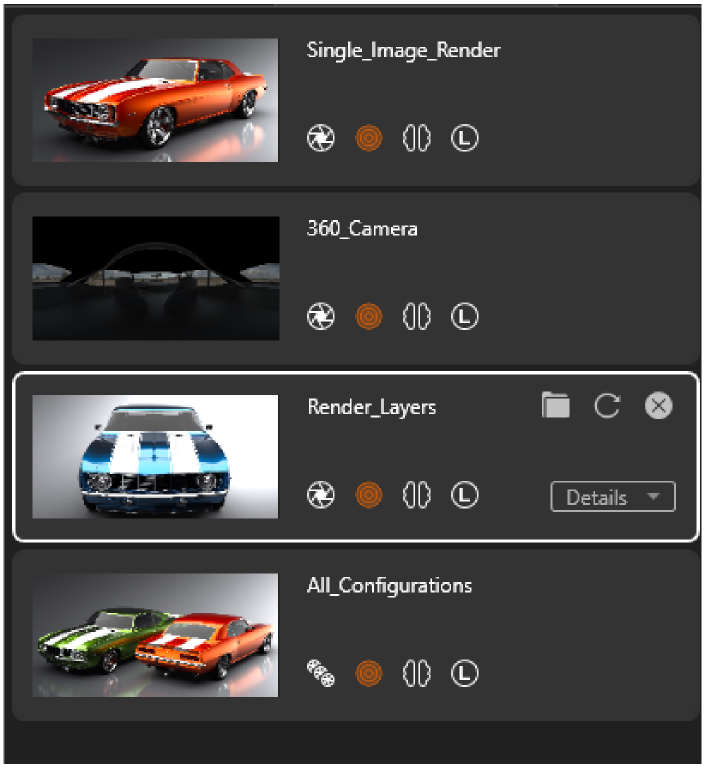
The Render tab has been renamed to the Output Viewer tab. Click **Output Tools** (Main toolbar). In the Output Tools dialog box, click **Start Render** or click **View** > **Show Output Viewer**.

User Interface

The Output Viewer tab (formerly the Render tab), is redesigned for a better user experience.

Areas of the Output Viewer tab include:

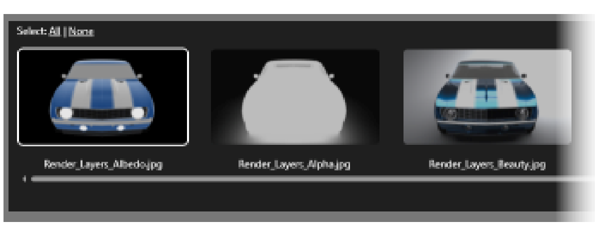
<table>
<tr><th>Area</th><th>Description</th></tr>
<tr><td>1. Render job palette


</td><td>Displays a list of render job submissions. You can:

- Monitor render progress
- Pause or save renders
- Initiate rerenderings
- Review the **Output Tools** options for the renders
- Right-click a render job and click **Remove Job**
- Right-click in the render job palette and:
  - Delete unlinked render jobs
  - Sort render jobs
- Right-click the scroll bar and scroll to various areas</td></tr>
<tr><td>2. Render viewport

</td><td>Displays output content being rendered. It also acts as a content viewer where you can see completed render jobs. In the upper right corner of the render viewport, you can:

- **Publish to 3DSpace** .
- **Publish to 3DSwym** . Publishes rendered content to 3DSwym.</td></tr>
<tr><td>3. Image thumbnails


</td><td>Lets you select renderings to open or view in the render viewport.
When you click a thumbnail, the image opens in the render viewport. If you double-click a thumbnail, the image opens in an external viewer.</td></tr>
</table>

Patterns

The Pattern tool is an evolution of the traditional **Formation** functionality.

You can base a pattern on a single model that you instance multiple times or on several different models.

Click **Project** > **Models** > **New Pattern**.

Creating Patterns

To create patterns:

1. Click **Project** > **Models** > **New Pattern**.
2. In the Palette, on the Models tab, on the General subtab, under **Formation**, specify options.

Formation Settings

The **Formation** settings apply when you pattern a model.

To access this dialog box, click **Project** > **Models** > **New Pattern**.

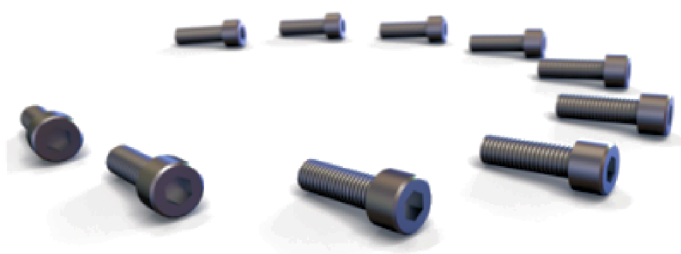
Formation	Specifies the model to pattern. The selected model is hidden in the 3D Viewport. You can drag different models into the pattern to form a customized pattern. In this case, **Formation** is not available.
Type	Specifies the pattern type: **Vee**, **Circle**, **Grid**, **Scatter**.

Vee

Vee formations use an angle to specify the opening angle of the vee.

Number of Objects	Specifies the number of objects in the pattern.
Angle	Specifies the opening angle of the vee.
Distance XYZ	Specifies a vector whose length defines the distance between instances and whose direction affects the pattern orientation.
Rotation XYZ	Specifies the rotation of the instances in the pattern in Euler angles (degrees).
Scale XYZ	Specifies the scale of the instances in the pattern in X,Y, and Z dimensions.
Relative	Accumulates the distance, rotation, or scale over the sequence of instances. When cleared, the distance, rotation, or scale is absolute (constant).
Scale All	Specifies an overall scale multiplier to the X, Y, and Z dimensions of the scale for all instances.

Circle



You can use the **Circle** formation to arrange instances in a circle or arc. The **Circle** is the only formation where several parameters are linked such that changing one affects the others. For example, if you increase the **Number of Objects**, the **Distance XYZ** value decreases so the **Radius** stays the same.

Number of Objects	Specifies the number of objects in the pattern.
Radius	Specifies the radius of the circular pattern.
Angle	Specifies a circle or an arc by specifying a value less than 360°.
Distance XYZ	Specifies a vector whose length defines the distance between instances and whose direction affects the pattern orientation.
Rotation XYZ	Specifies the rotation of the instances in the pattern in Euler angles (degrees).
Scale XYZ	Specifies the scale of the instances in the pattern in X,Y, and Z dimensions.
Relative	Accumulates the rotation or scale over the sequence of instances. When cleared, the rotation or scale is absolute (constant).
Scale All	Specifies an overall scale multiplier to the X, Y, and Z dimensions of the scale for all instances.

Grid

Depending on the value of **Number of Objects X**, **Y**, and **Z**, the **Grid** formation can be a line (a value greater than 1 in a single dimension and a value of 1 in the other two dimensions), a plane (a value greater than 1 in two dimensions and a value of 1 in the third dimension), or a cube (a value greater than one in all three dimensions). The total number of instances is equal to the product of the **Number of Objects X**, **Y**, and **Z**.

The total number of objects is the product of the values of **Number of Objects X**, **Number of Objects Y**, and **Number of Objects Z**.

Number of Objects X	Specifies the number of objects in the X dimension.

Number of Objects Y	Specifies the number of objects in the Y dimension.
Number of Objects Z	Specifies the number of objects in the Z dimension.
Distance XYZ	Specifies a vector whose length defines the distance between instances and whose direction affects the pattern orientation.
Rotation XYZ	Specifies the rotation of the instances in the pattern in Euler angles (degrees).
Scale XYZ	Specifies the scale of the instances in the pattern in X,Y, and Z dimensions.
Relative	Accumulates the distance, rotation, or scale over the sequence of instances. When cleared, the distance, rotation, or scale is absolute (constant).
Scale All	Specifies an overall scale multiplier to the X, Y, and Z dimensions of the scale for all instances.

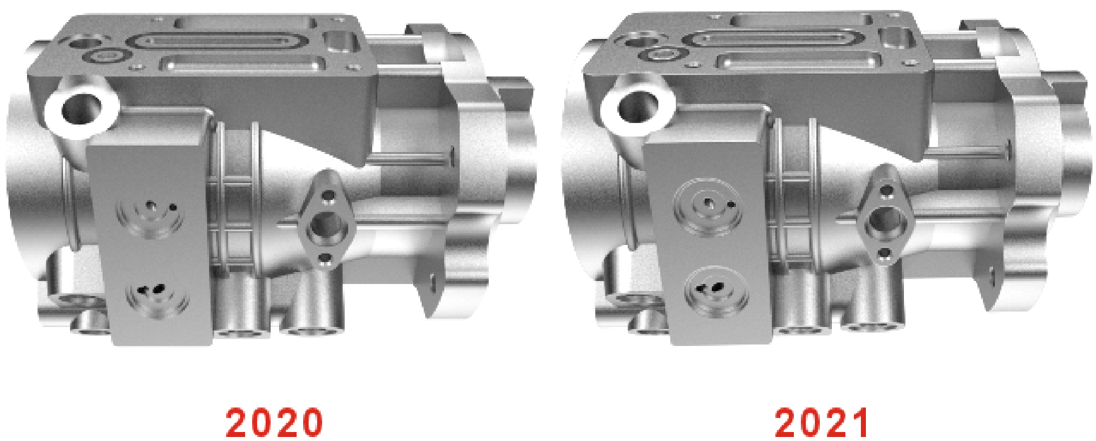
Scatter

The **Scatter** formation allows completely random arrangements of objects within a specific range. Equal values for **Minimum** and **Maximum** create a deterministic transformation value for that degree of freedom.

Number of Objects	Specifies the number of objects in the pattern.
Position Range XYZ	Specifies the **Minimum** and **Maximum** limits for the random computation of translations (position) of each instance.
Rotation Range XYZ	Specifies the **Minimum** and **Maximum** limits for the Euler angles X, Y, Z for the random computation of the rotations of each instance.

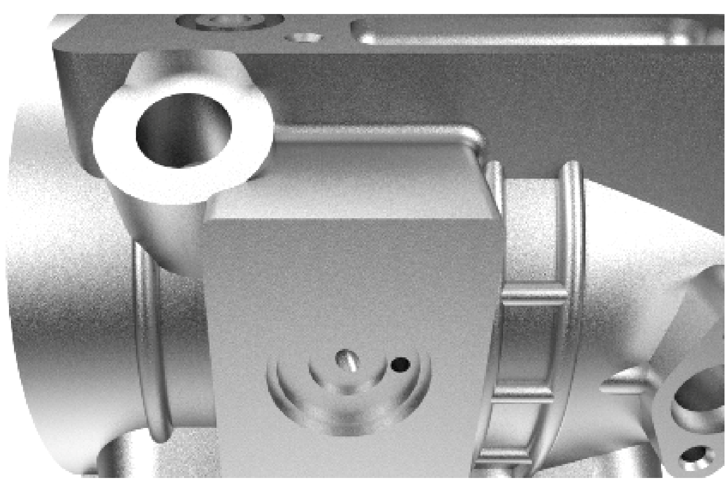
Scale Range XYZ	Specifies the **Minimum** and **Maximum** limits for the random computation of scale of each instance. **Uniform** specifies a random scale in the X, Y, and Z dimension.
Scale All	Specifies an overall scale multiplier to the X, Y, and Z dimensions of the minimum and maximum scales for all instances.

Corner Radius

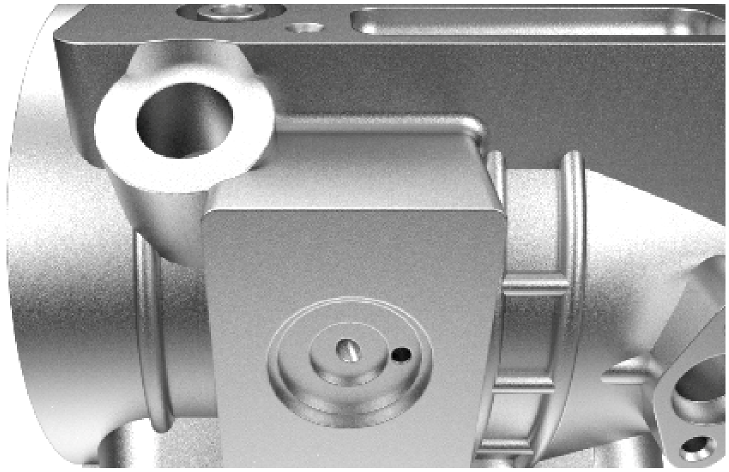


The **Corner Radius** functionality is enhanced to smooth edges in parts.

Previously, the effect of **Corner Radius** was visible only if the appearance attached to the geometry did not use a bump or normal map.

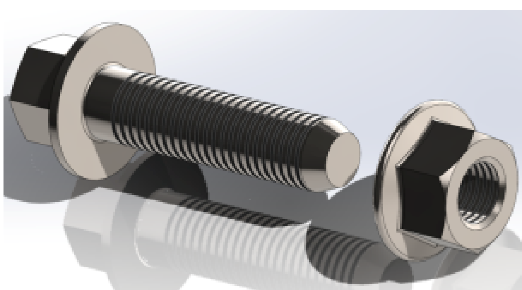
2021: All hard edges are sharp because the appearance attached to the part uses a bump or normal map.	

2022: The **Corner Radius** is visible even when the appearance on the part uses a bump or normal map.

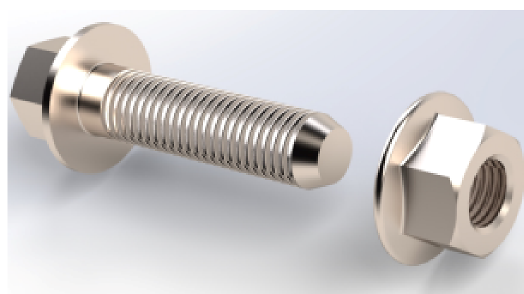


In the Palette, on the Models tab, on the Advanced subtab, specify **Corner Radius (mm)**.

Cosmetic Threads



SOLIDWORKS



SOLIDWORKS Visualize

For more realistic-looking models, SOLIDWORKS Visualize automatically applies a normal map to models imported with cosmetic threads.

18

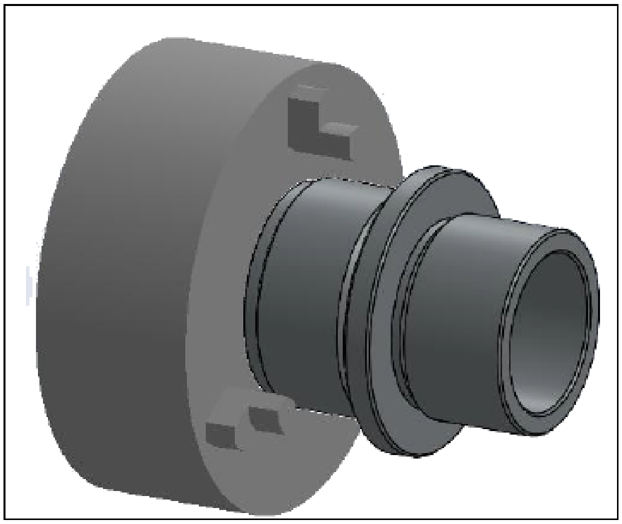
SOLIDWORKS CAM

This chapter includes the following topics:

SOLIDWORKS CAM is offered in two versions. SOLIDWORKS CAM Standard is included with any SOLIDWORKS license that has SOLIDWORKS Subscription Service.

SOLIDWORKS CAM Professional is available as a separately purchased product that you can use with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium.

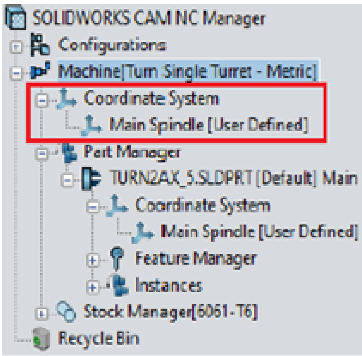
Assembly Support for Turn



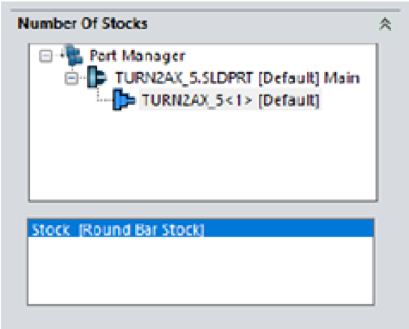
Turn mode supports assemblies that contain a single Turn part model.

You must define the following:

- In the **Machine** tab of the Machine dialog box, tthe **Main Spindle Coordinate** system for machining the parts in the assembly

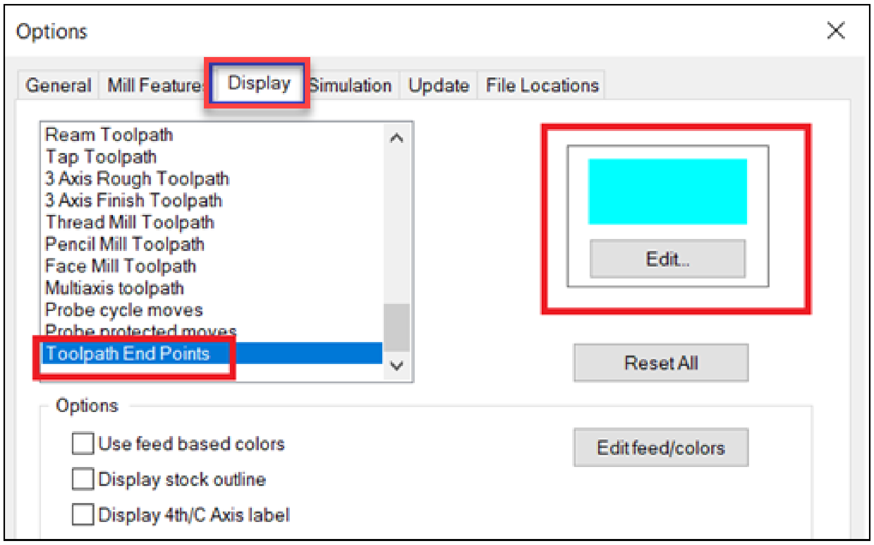


- Under the **Stock Manager** node, the stock for the individual parts that will be listed as subnodes



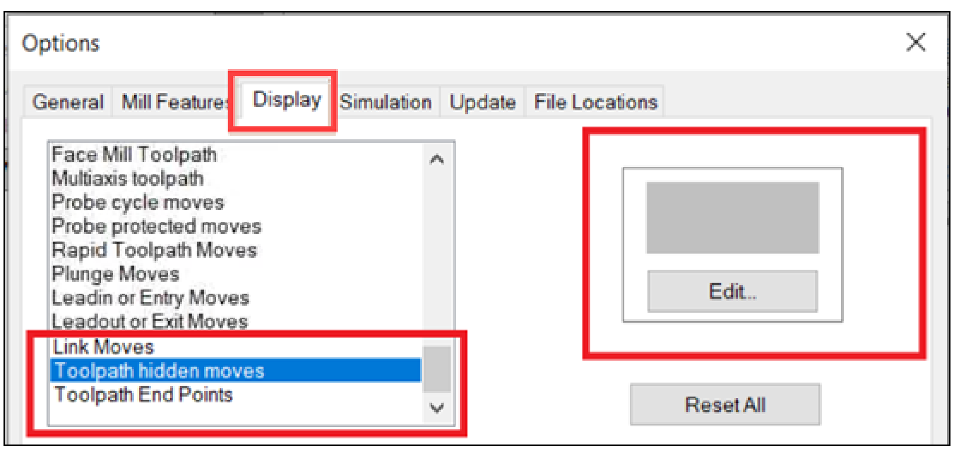
- In the Manage Parts dialog box, the turn part, the spindle designation, origin of the coordinate system and turn feature section plane
- In the Spindle work coordinate dialog box, the programmable offsets for the main spindle of the machine

Customize Color Settings for Toolpath End Points



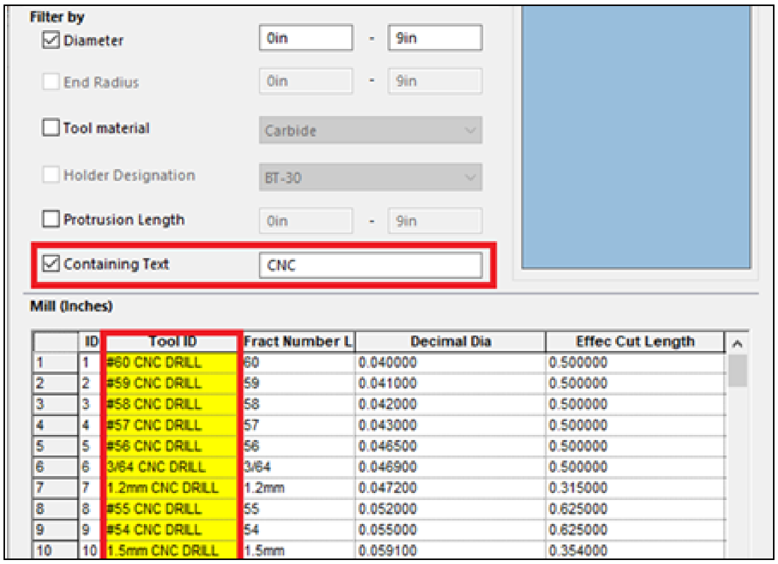
From the Display tab in the Options dialog box, you can apply color settings for toolpath end points.

Display Color for Hidden Toolpath Moves



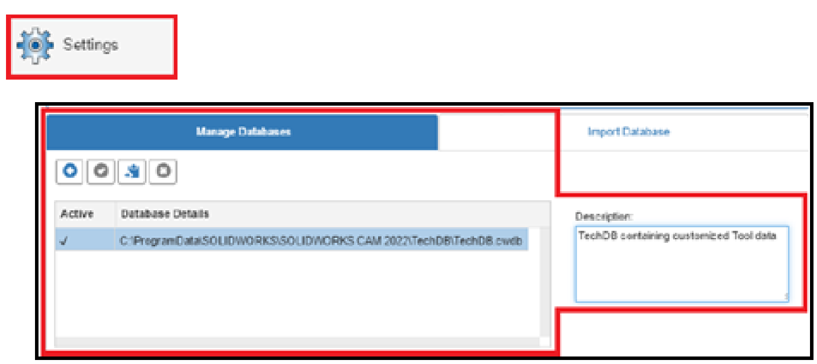
In the Display tab of the Options dialog box, you can assign a color to display **Toolpath hidden moves** in the graphics area.

Filter for Mill and Turn Tools and Assemblies with Text



The Tool Select Filter dialog box lets you enter text to filter for mill and turn tools and assemblies.

Manage Multiple Technology Databases

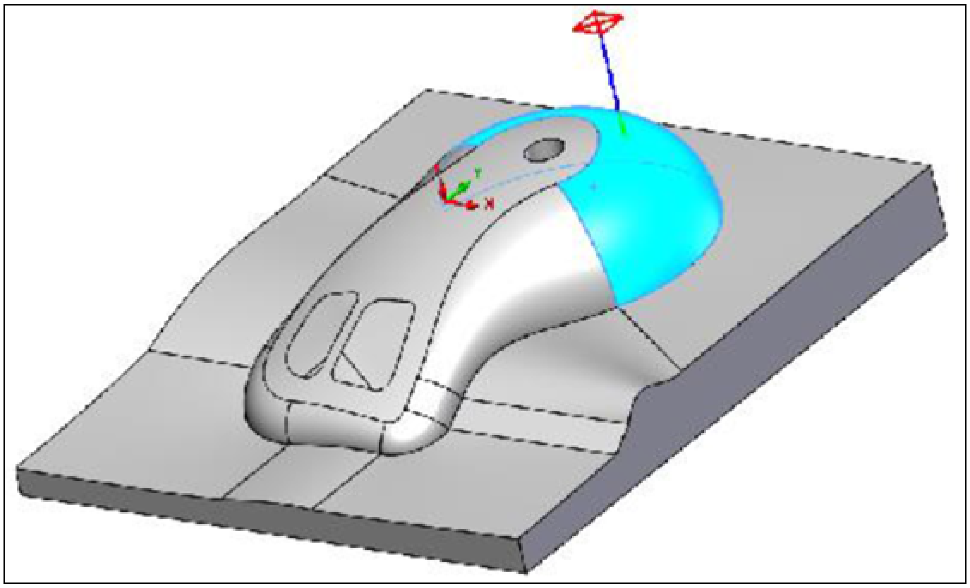


You can manage multiple technology databases in the Manage Database tab.

In **Settings**, the Link Database tab has been renamed Manage Database.

The Manage Database tab lets you specify multiple technology database source files, but you can only assign one database as the active database to the application.

Support for Nonplanar Surfaces for Z Axis Probing



You can select nonplanar surfaces in the direction of the Z axis for probe cycles.

The **Z Probe Cycle** is available in the Probe tab of the Operation Parameters dialog box. The default touch off point for toolpath generation is the top-most point of the selected face in the Z direction.

Revised CNC Finish Parameters for Clarity

Previous Labels	Renamed Labels
Off	None
On	Yes
With compensation	With compensation (Toolpath is offset by tool radius)
Without compensation	Without compensation (Tool center is on feature geometry)
Gouge check	Limited look ahead
Sharp corner	Internal sharp corners
Add tool radius to leadin/leadout	Add tool radius to leadin/leadout

The **NC** tab in the Operation Parameters dialog box and the interface of the Technology Database have updated and rearranged labels to improve the readability of CNC finish parameters.

Supported Platforms for SOLIDWORKS CAM

SOLIDWORKS CAM supports the 64-bit version of SOLIDWORKS 2022 and SOLIDWORKS 2021 running on the 64-bit of Windows 10.

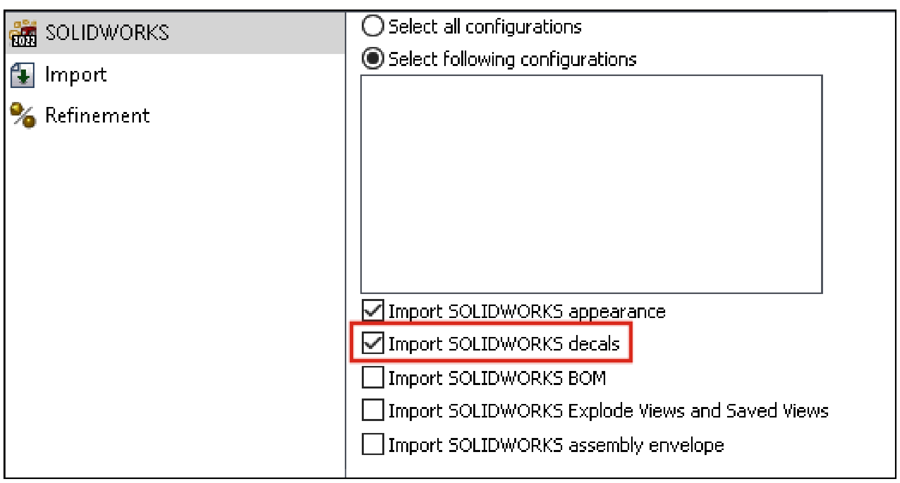
19

SOLIDWORKS Composer

This chapter includes the following topics:

SOLIDWORKS® Composer™ software streamlines the creation of 2D and 3D graphical contents for product communication and technical illustrations.

Importing Decals from SOLIDWORKS Files



You can import decals from SOLIDWORKS files to Composer files.

To import decals:

- In SOLIDWORKS Composer, click **File** > **Open** and select a SOLIDWORKS file. Under **SOLIDWORKS**, select **Import SOLIDWORKS decals** in the dialog box.
- In SOLIDWORKS Composer, click **File** > **Properties** > **Default Document Properties** > **Advanced Properties**. Under **INPUT - IMPORT**, select **IOSWImportDecals**. See *Managing (Default) Document Properties* > *Input* in *Composer Help*.
- In SOLIDWORKS Composer Sync, at the bottom of the window, click **More Properties** > **Advanced Properties**. In the dialog box, under **INPUT - IMPORT**, select

Batch.IOSWImportDecals to enable batch import of decals. See *Sync > Managing Default Document Properties > Advanced Properties* in *Composer Help*.

Support for Higher Version of Import Formats

SOLIDWORKS Composer and SOLIDWORKS Composer Sync support higher versions of the following import formats:

- ACIS™ up to R2021 1.0
- Pro/E® Creo 1.0 to 7.0
- SOLIDWORKS 2006 to 2022

See *Importing and Opening Files > About Supported Import Formats or Sync > About Import Formats and File Types > About Supported Import Formats* in *Composer Help*.

20

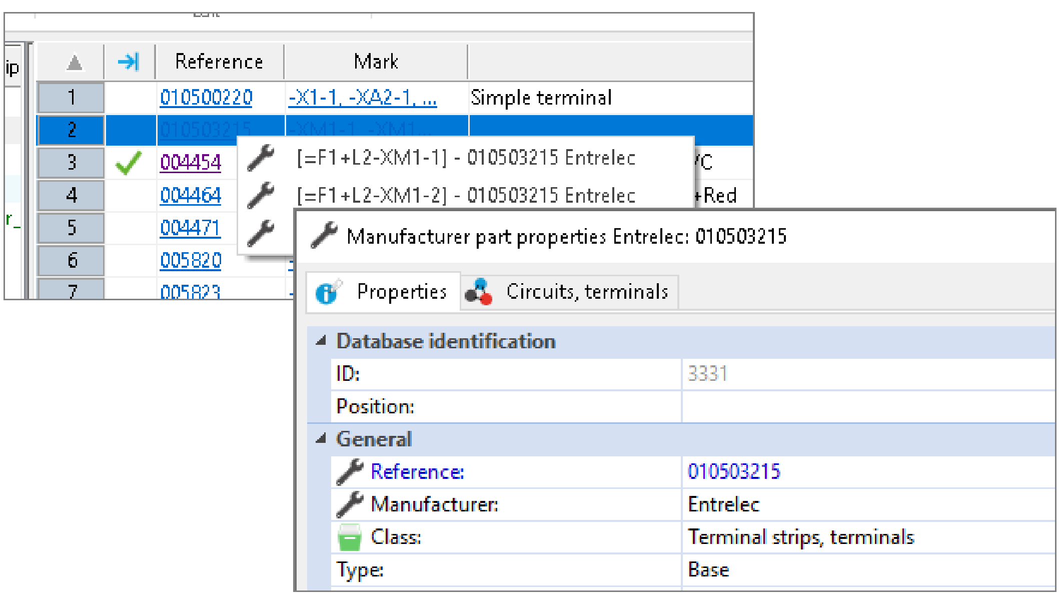
SOLIDWORKS Electrical

This chapter includes the following topics:

- **Links in BOMs**
- **Add Data Files in the Export PDF**
- **Testing the Query in the Expert Mode**
- **Displaying ERP Data in the Manufacturer Parts Manager**
- **Including Data Sheets in Exported PDFs**
- **Displaying Break Condition in Report Manager**
- **User Interface Redesign**
- **Attribute in Origin - Destination Arrows**
- **Displaying All the Wire Numbers on the Middle of the Line**
- **Electrical Content Portal Integration**
- **Connection Point Creation Enhancements**

SOLIDWORKS® Electrical is a separately purchased product.

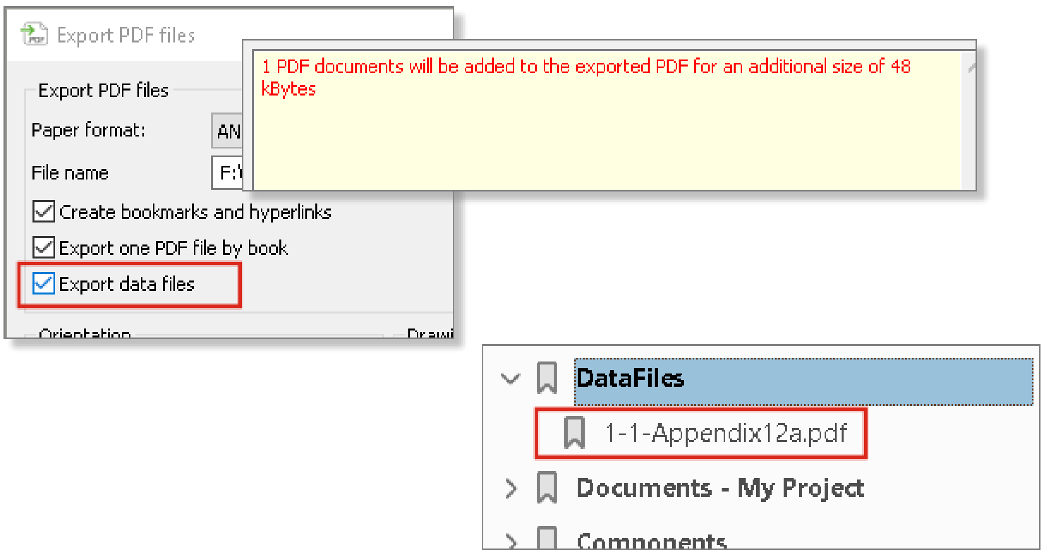
Links in BOMs



In BOMs, you can manage links in a merged cell, for example, when several components use the same manufacturer part.

In a BOM or other reports, the report viewer displays links. These links display information like the properties, the location in the drawings of the selected element, or open the drawing and zoom the component. By clicking the right mouse button on the link, you can select the component you want to display the information.

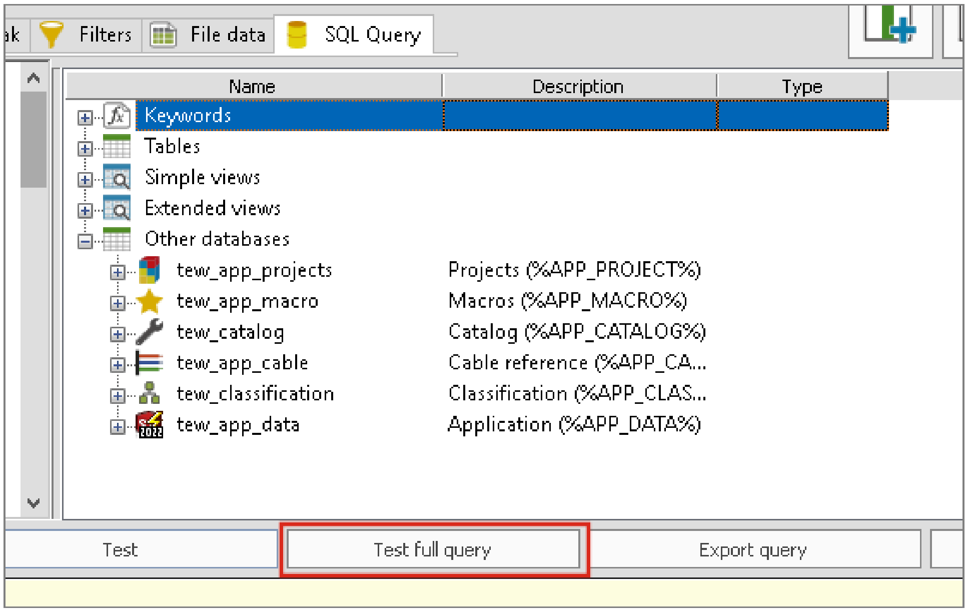
Add Data Files in the Export PDF



If you attach PDF files to the project as data files, you can export them in the exported PDF file.

When you select **Export data files**, a message displays the PDF file size. The PDF file gathers all the datafiles in a new section.

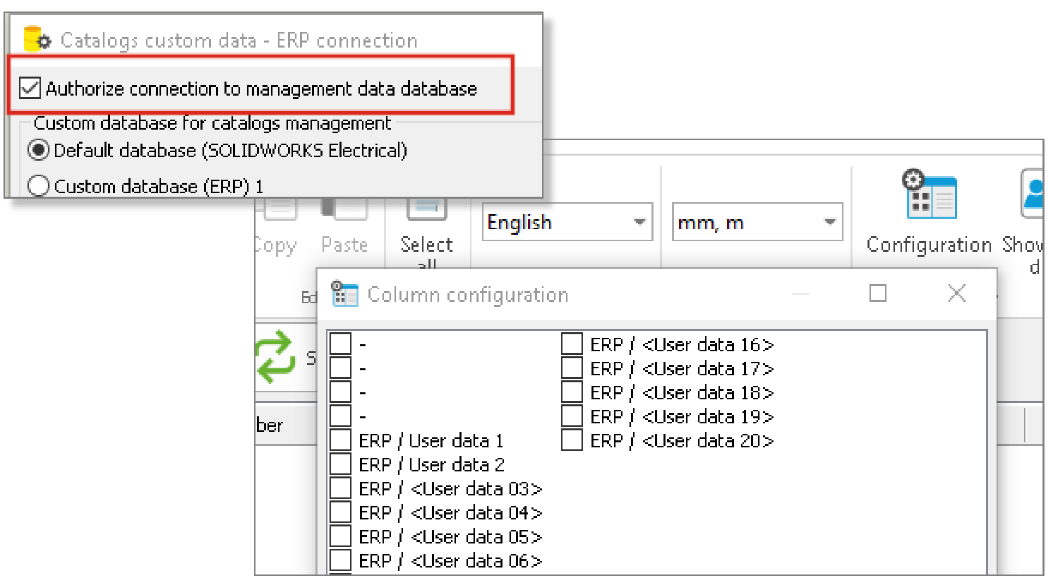
Testing the Query in the Expert Mode



When you edit report configurations in **Expert** mode, **Test fully query** includes the sort conditions and filters.

In report configurations, the **Expert** mode option is on the SQL Query tab.

Displaying ERP Data in the Manufacturer Parts Manager

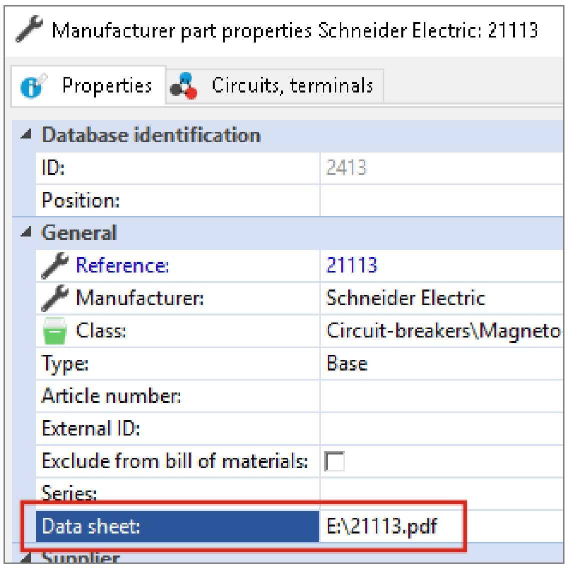


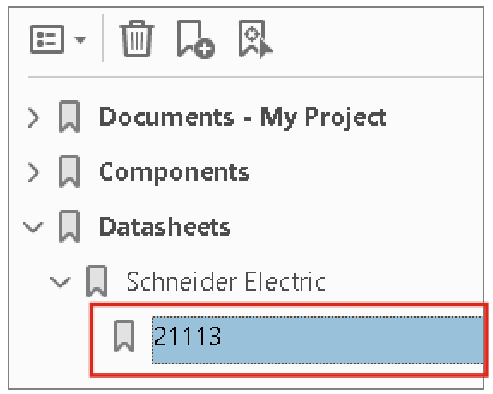
If you connect the ERP data, you can display it in the Manufacturer Parts Manager.

To display the ERP data in the Manufacturer Parts Manager, open the Column configuration and select the ERP data you want to show.

If you do not connect the ERP Data, you can only select the **User data**.

Including Data Sheets in Exported PDFs

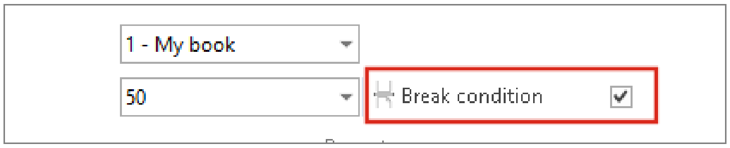


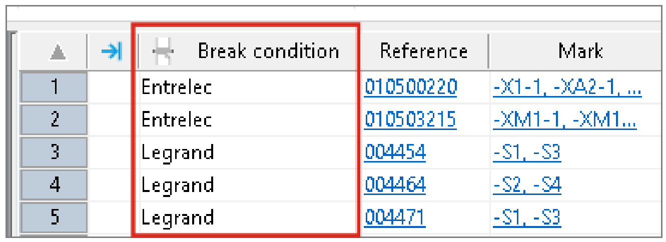


You can include the data sheets associated with the manufacturer parts in exported PDF files.

A data sheet is a file or a link associated to a manufacturer part. When you export the project drawings in a PDF file, the data sheets appear in the Data sheets section of the PDF. Click **Export PDF file** and select **Create bookmarks and hyperlinks** to create the **Data sheets** section.

Displaying Break Condition in Report Manager

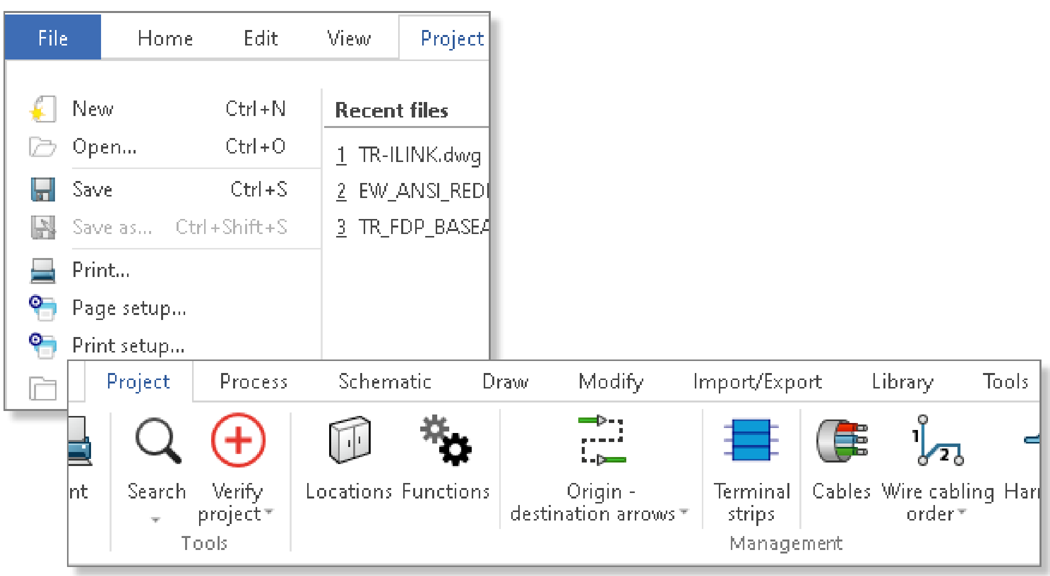




In the Report Manager, you can display a **Break condition** column that shows the field in which you have created the break condition.

To display this column, select **Break Condition**. This column is not printable.

User Interface Redesign

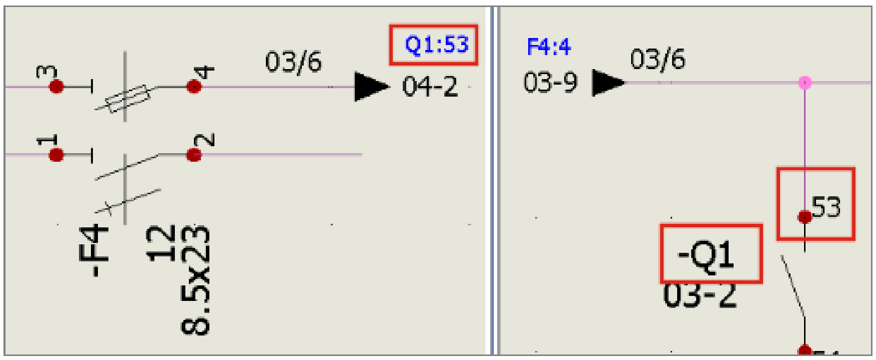


The user interface is enhanced to be more ergonomic and user friendly.

- New icons

- You can **Expand** or **Collapse** the ribbon menu
- The commands to manage external files are in a **File** menu
- You can select the theme color of the interface
- The **Drawing style** commands move from the **Tools** menu to the **Modify** menu
- The **Search** and **Verify project** commands move from the **Tools** menu to the **Project** menu

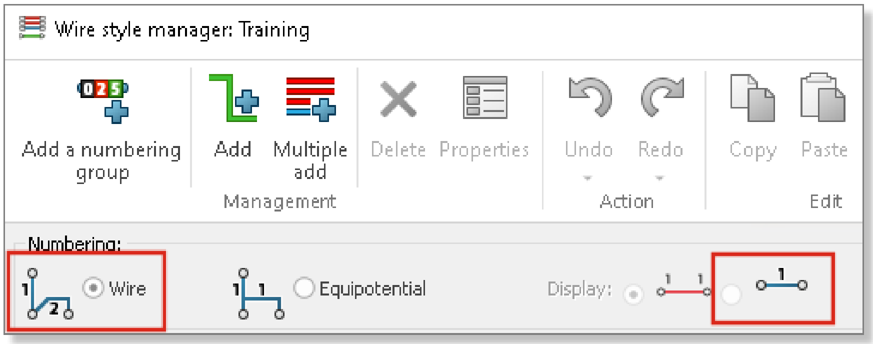
Attribute in Origin - Destination Arrows



In the **Origin - Destination arrows** feature, the attribute **#P_CONNECTED_0** lets you display the mark of the components connected through the arrows.

From the **Symbols manager**, edit the symbol of the type **Origin - Destination arrows** to add the attribute **#P_CONNECTED_0**.

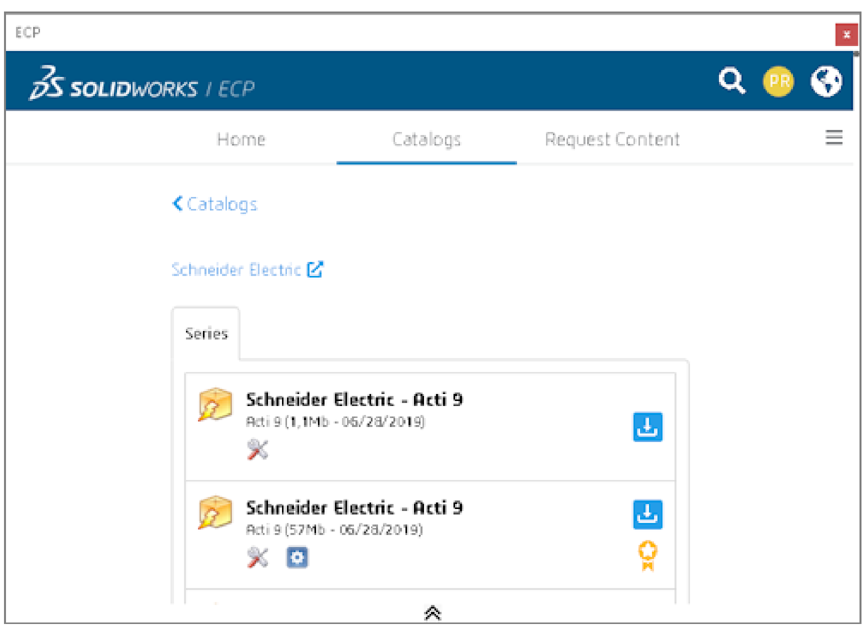
Displaying All the Wire Numbers on the Middle of the Line



When you number the wires, you can select a specific mode allowing you to display the wire number on the middle of the line segment.

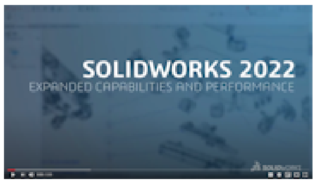
When you connect an equipotential to three or more components, this **Display** mode lets show all the wire numbers on the middle of the line segment. You can select this mode in the Wire styles manager.

Electrical Content Portal Integration



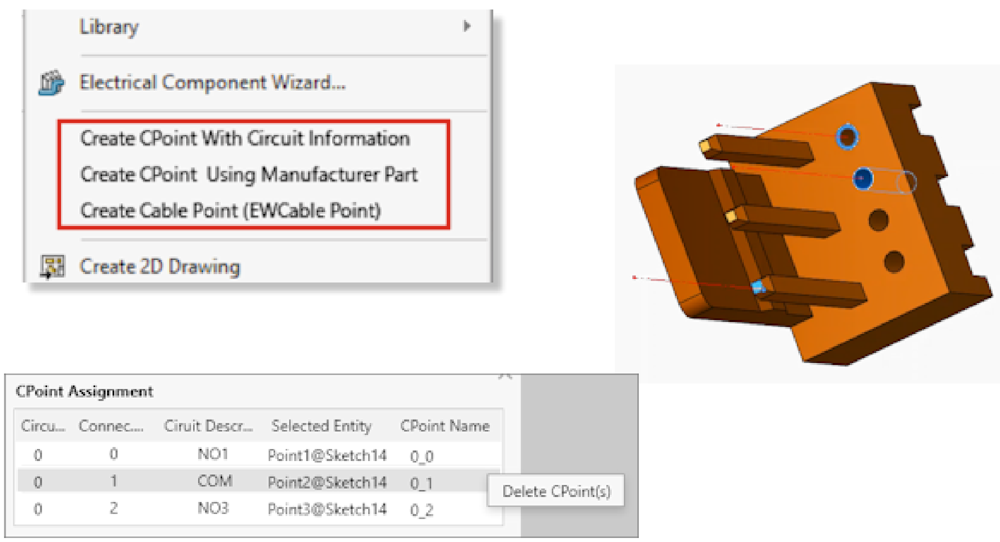
Electrical Content Portal (ECP) is where you can download content, such as manufacturer parts, cable references, and files for 2D layouts.

A dockable panel displays the ECP interface. It lets you download content and automatically unarchive it into the respective libraries.



Video: What's New in SOLIDWORKS 2022 - SOLIDWORKS Electrical

Connection Point Creation Enhancements



Connection points (CPoints) let you connect wires or cables with connectors.

- Commands to create the CPoints are available in the SOLIDWORKS Electrical menu.
- You can assign Cpoints by selecting an edge or a cylindrical surface.
- To delete an assignment in the **CPoint Assignment** table, right-click the row and select **Delete CPoint(s)**.

21

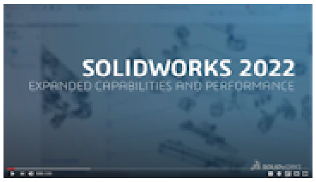
SOLIDWORKS Inspection

This chapter includes the following topics:

- **SOLIDWORKS Inspection Add-in**
- **SOLIDWORKS Inspection Standalone**

SOLIDWORKS Inspection is a separately purchased product that you can use with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium, or as a completely separate application (see *SOLIDWORKS Inspection Standalone*).

SOLIDWORKS Inspection Add-in



Video: What's New in SOLIDWORKS 2022 - SOLIDWORKS Inspection

Application Programming Interface

```
'General settings
Debug.Print InspectionPrjData.DocumentName
InspectionPrjData.DocumentName = "changed"
Debug.Print InspectionPrjData.DocumentName

Debug.Print InspectionPrjData.Basic
InspectionPrjData.Basic = True
Debug.Print InspectionPrjData.Basic

'Extraction settings
Debug.Print InspectionPrjData.SecondaryUnits
InspectionPrjData.SecondaryUnits = True
Debug.Print InspectionPrjData.SecondaryUnits

If InspectionPrj Is Nothing Or Not err = swiErrorCode_e.swiSuccess Then
MsgBox ("Project not created")
End If

'Balloon settings
Set BalloonSetting = INSPECTIONMgr.GetBalloonSettings()
```

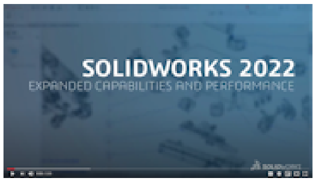
SOLIDWORKS Inspection functionality is available through the Application Programming Interface (API).

You can use the API to:

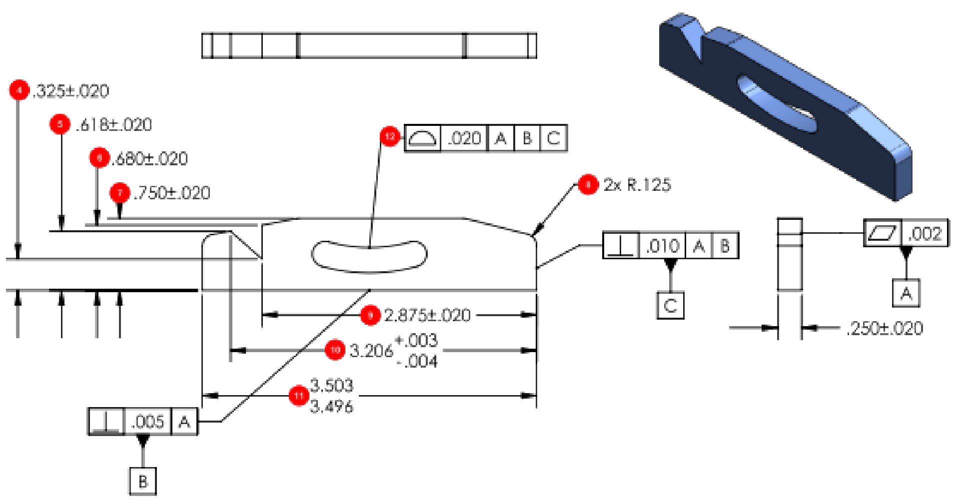
- Automatically open SOLIDWORKS files
- Create inspection projects
- Export First Article Inspection (FAI) reports
- Balloon drawings
- Invoke any feature available in the user interface

SOLIDWORKS Inspection Standalone

SOLIDWORKS® Inspection is a separately purchased product that you can use with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium, or as a completely separate application (see *SOLIDWORKS Inspection Standalone*).

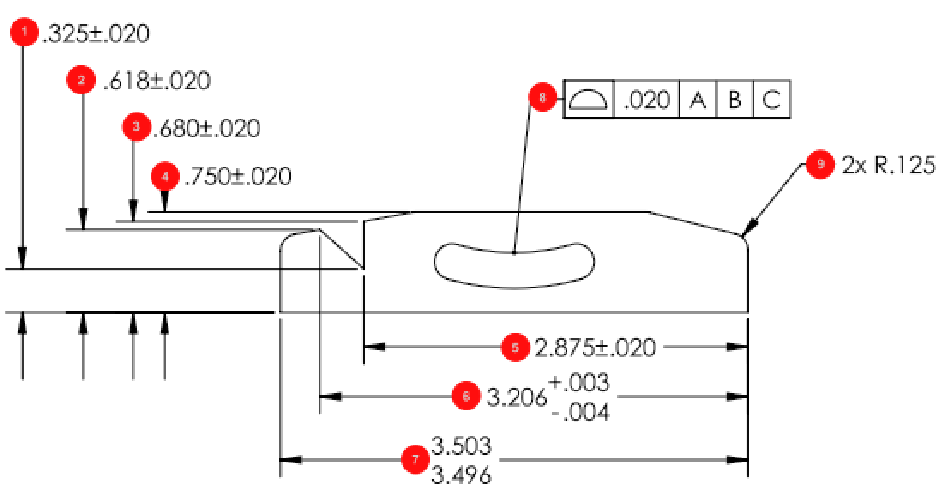
	**Video: What's New in SOLIDWORKS 2022 - SOLIDWORKS Inspection**

Supported File Types



SOLIDWORKS Inspection Standalone supports SOLIDWORKS (`.SLDPRT`, `.SLDASM`, `.SLDDRW`) and NX™/Unigraphics® (`.prt`) files.

Smart Extract



The **Smart Extract** tool has improved character recognition and parsing. This improves accuracy when extracting information from `pdf` files.

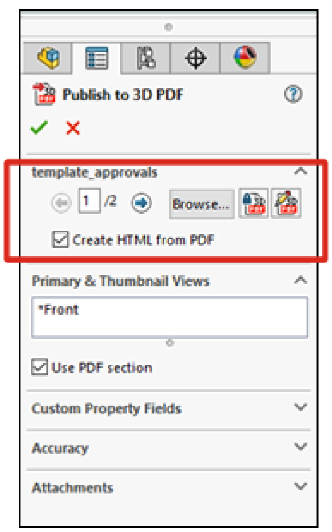
22

SOLIDWORKS MBD

This chapter includes the following topics:

SOLIDWORKS® MBD is a separately purchased product that you can use with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium.

Creating HTML Output from the 3D PDF

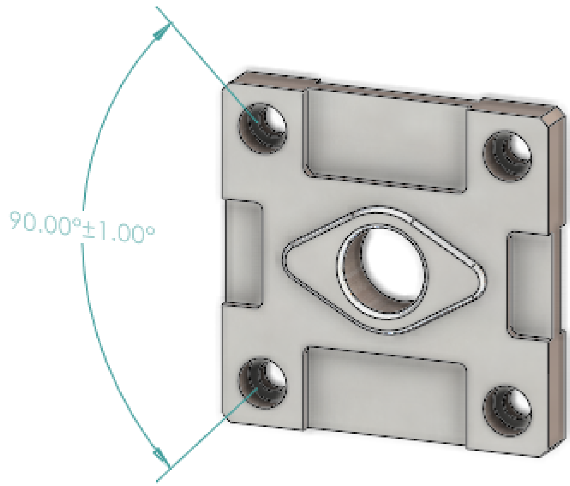


When you publish a 3D PDF of a model, you can create an `.html` file in addition to the 3D PDF file.

When you are ready to publish:

1. Click **Publish to 3D PDF** (MBD toolbar).
2. In the Template Selection dialog box, select a template and click **OK**.
3. In the Publish to 3D PDF PropertyManager, under the template name, select **Create HTML from PDF** and click .

DimXpert Angle Dimension Tool

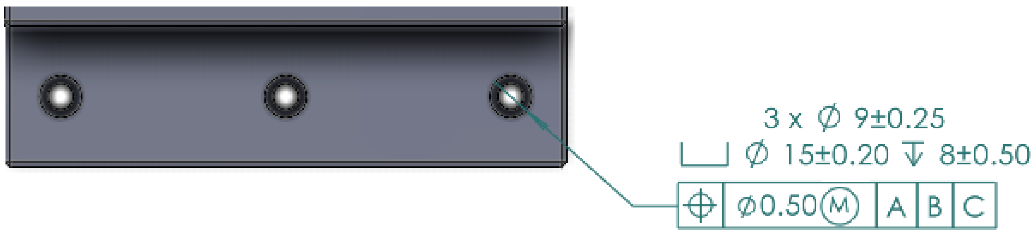


You can manually create DimXpert angle dimensions with the **Angle Dimension** tool, similar to creating reference angle dimensions with **Smart Dimension** tool.

To create a DimXpert angle dimension, you can select two or three DimXpert features, where the first two selections define the origin and the third selection establishes the toleranced feature. Previously, you could only use the **Auto Dimension Scheme** tool to create DimXpert angle dimensions, which did not allow specific input for the dimensioned angle.

Click **Angle Dimension** (MBD Dimension toolbar) or **Tools** > **MBD Dimension** > **Angle Dimension**.

Geometric Tolerancing for DimXpert



Geometric tolerancing for DimXpert includes several improvements in addition to the new user interface described in **Geometric Tolerance Symbols** on page 81 in the *Detailing and Drawings* chapter.

In **Tools** > **Options** > **Document Properties** > **DimXpert**, you can specify the **Base DimXpert standard**. If you select the ANSI/ASME Y14.5 or ISO standard, you can also select a release date for the standard.

Standard	Release Date Options
ANSI/ASME Y14.5	• 1994 • 2009 • 2018
ISO 1101	• 1983 • 2004 • 2012 • 2017

When you first click **Geometric Tolerance** (MBD or MBD Dimension toolbar), you must then select an existing DimXpert feature or define a new DimXpert feature.

When you select a feature, the software evaluates the feature and tolerance specifications for the drafting standard you selected. In the dialog box, options are available based on the standard.

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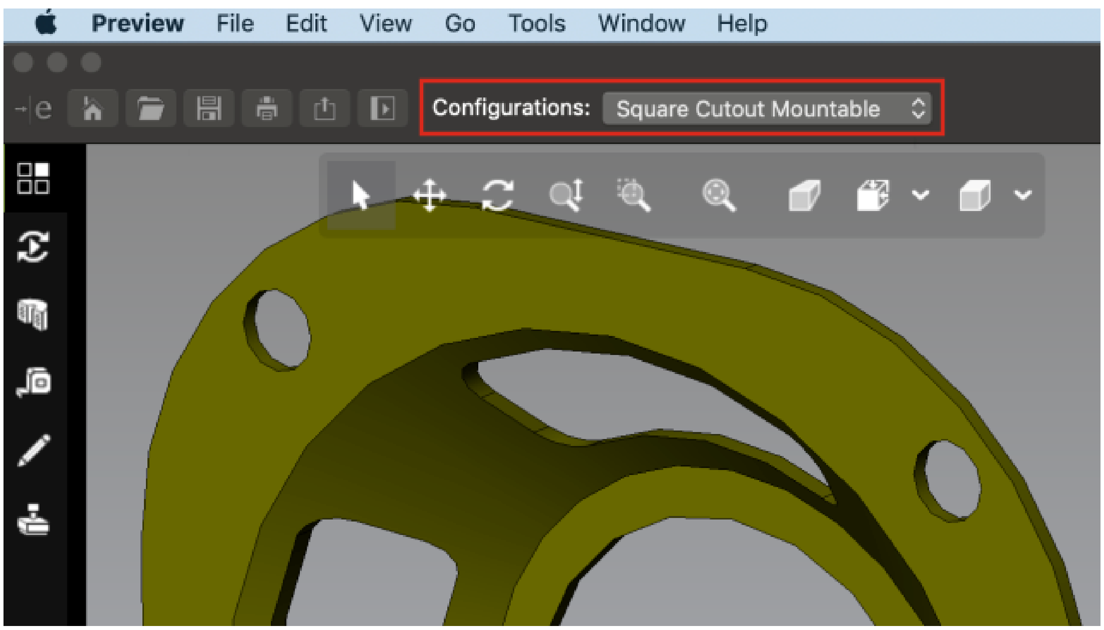
eDrawings

This chapter includes the following topics:

- **Configuration Support**
- **eDrawings Options in SOLIDWORKS**
- **File List**
- **Custom Properties Options**
- **User Interface**
- **Components Pane**

eDrawings® Professional is available in SOLIDWORKS® Professional and SOLIDWORKS Premium.

Configuration Support



You can specify configurations of SOLIDWORKS parts and assemblies to be available when you open them in eDrawings® for Mac®.

In SOLIDWORKS, in the ConfigurationManager, right-click one or more configurations and click **Add Display Data Mark**.

eDrawings Options in SOLIDWORKS

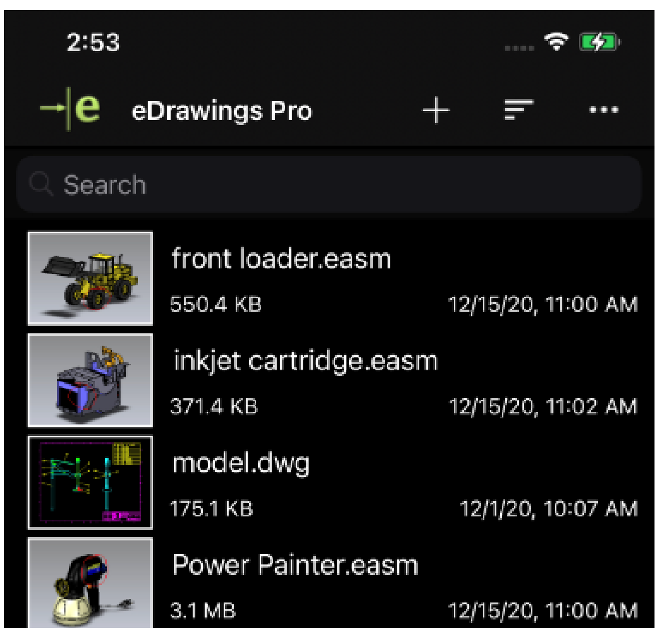
eDrawings related options in SOLIDWORKS have been renamed and reorganized for ease of use.

In SOLIDWORKS, click **Tools** > **Options** > **System Options** > **Export**. In **File Format**, select **EDRW/EPRT/EASM**.

The following options have been renamed, but the functionality remains the same.

New name	Previous name
Enable measure	**Okay to measure this eDrawings file**
Allow STL export	**Allow export to STL for parts & assemblies**
Save table features	**Save table features to eDrawings file**
Save shaded data	**Save shaded data in drawings**
Save motion studies	**Save Motion Studies to eDrawings file**

File List



The file list in eDrawings for iOS™ is enhanced.

In the list, you can:

- See detailed file information

- Sort on file parameters, such as filename, date, extension, and size by tapping
- Share or delete files by touching and holding a file name

Custom Properties Options

You can save custom properties from a SOLIDWORKS document in resulting eDrawings files when you **Save As** an eDrawings document or **Publish to eDrawings** in SOLIDWORKS.

For lightweight assemblies, only custom properties for the top-level assembly are available in the resulting eDrawings files.

Configuration-specific properties are not available for assemblies opened in lightweight mode.

To specify custom properties options:

1. In SOLIDWORKS, click **Tools** > **Options** > **System Options** > **Export**.
2. In the dialog box, in **File Format**, select **EDRW/EPRT/EASM**.
3. Select options:
   - **Save file properties**. Saves custom properties from a SOLIDWORKS document in resulting eDrawings files when you **Save As** an eDrawings document or **Publish to eDrawings** in SOLIDWORKS.
   - **Save file properties for each component in the assembly**. (Available if you select **Save file properties.**) Saves custom properties including configuration-specific properties for each component in the SOLIDWORKS assembly.

     When you open the file in eDrawings, the **Properties** tool is available if the file has custom properties.

Exporting Custom Properties

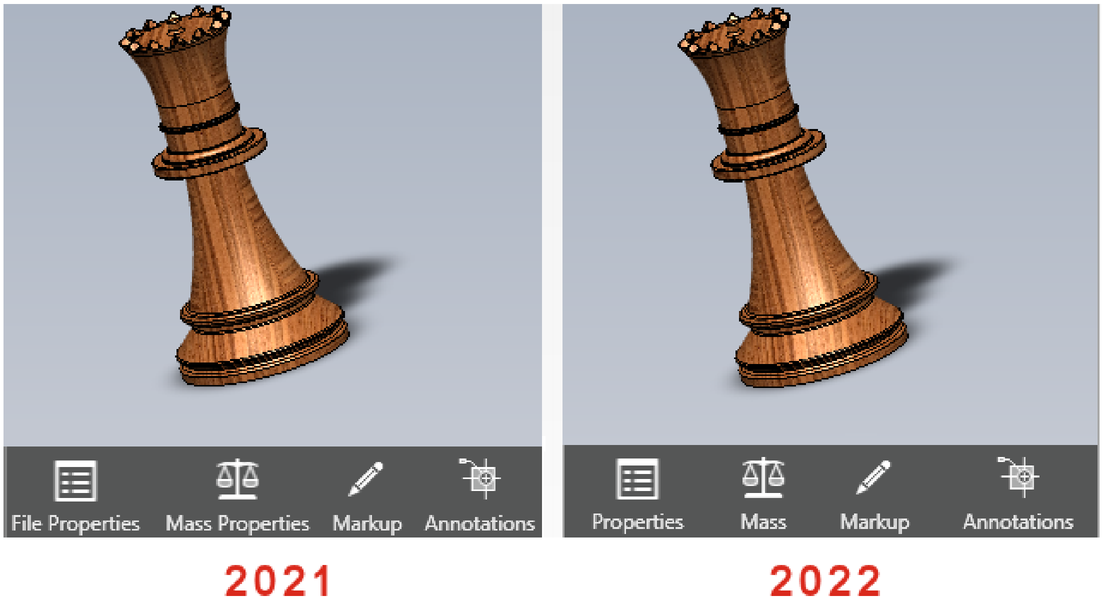
When you save a SOLIDWORKS file as an eDrawings file from within eDrawings, you can include custom properties in the eDrawings file.

To export custom properties:

1. In eDrawings, open a SOLIDWORKS file.
2. Click **File** > **Save As**.
3. In the dialog box, select **Include file properties** and click **Save**.

   Custom and configuration-specific properties in the SOLIDWORKS file are saved in the eDrawings file.

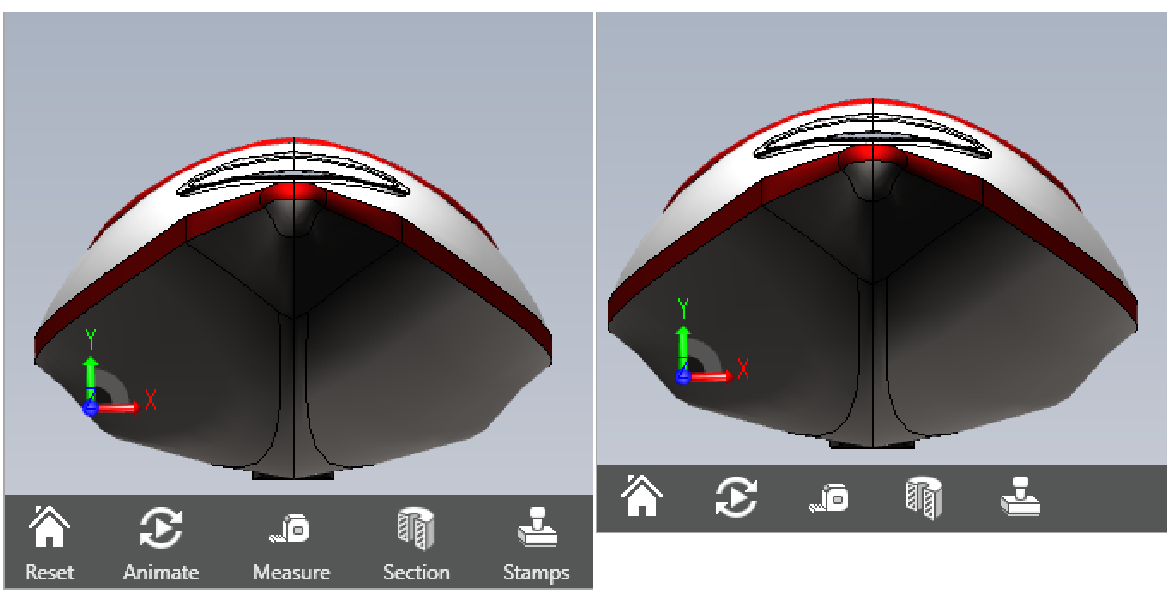
User Interface



The eDrawings user interface is updated for ease of use.

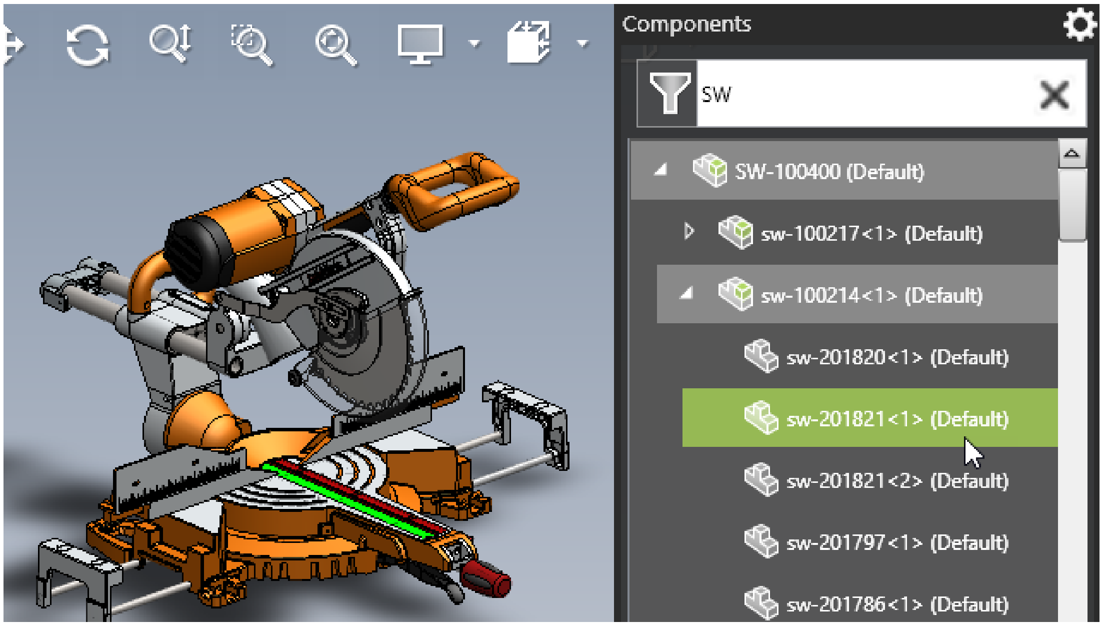
The tools, **File Properties** and **Mass Properties**, have changed to **Properties** and **Mass**, respectively.

To reduce the height of the tools at the bottom of the eDrawings window, you can remove the labels. Right-click a tool and click **Show labels**.



Show labels selected	**Show labels** cleared

Components Pane



When you work with assemblies in eDrawings, the Components pane is enhanced for ease of use.

Enhancement	Description
Options	The following are options in the Components pane: • **Highlight parents on hover**. Highlights the parent row when you hover over a child row. • **Show Component Descriptions**. This option has moved from the Options dialog box.
Clear filtered text	Clears previously filtered content.
Top-level assembly display	Displays the top-level assembly node in the components tree.
Pane height	Displays the Components pane at full height to reduce scrolling.
Expand All/Collapse All	Expands or collapses all items below the selected assembly node.

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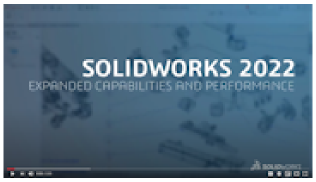
SOLIDWORKS Flow Simulation

This chapter includes the following topics:

SOLIDWORKS® Flow Simulation is a separately purchased product that you can use with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium.

	**Video: What's New in SOLIDWORKS 2022 - Flow Simulation**

Scene Plot

Scene plot stores all displayed plots, model orientation, zoom, and part visibility. Switching between scenes displays the plots saved in the scene and retains the corresponding model display, zoom, and orientation.

Compare: Results Summary

The Compare and Parametric Studies include a Results Summary.

Compare: Merged Plots

With the **Compare** tool, you can merge plots from different projects to see critical results in one image.

For example, you can merge contour plots by maximum value to show the maximum temperature for all design cases in one image.

Compare: Difference Plot

With the **Compare** tool, you can create a plot image displaying the difference of one particular case to a reference case.

Heat Source

The specific power (W/m3) can be dependent on local (calculated in the cell) temperature in formula or table dependency.

Range Function

The **RANGE** function gets goal values at any time during calculation. This lets you model complex behavior of the transient system, for example, power derating based on a temperature sensor.

Remove Missing Entities

You can automatically remove reference geometry (faces, edges, and points) of missing or suppressed bodies from the selection.

Check Geometry

You can create solid and fluid bodies for **Improve Geometry Handling** mode.

Goals

You can show equation goals based on defined goals after calculation.

Flux Plot

You can display a Flux Plot in the Transient Explorer.

Surface Parameters

Crop region is accounted for when evaluating surface parameters.

Probes

Probes are copied to projects together with plots that define the probes.

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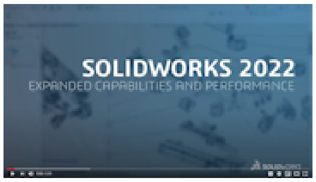
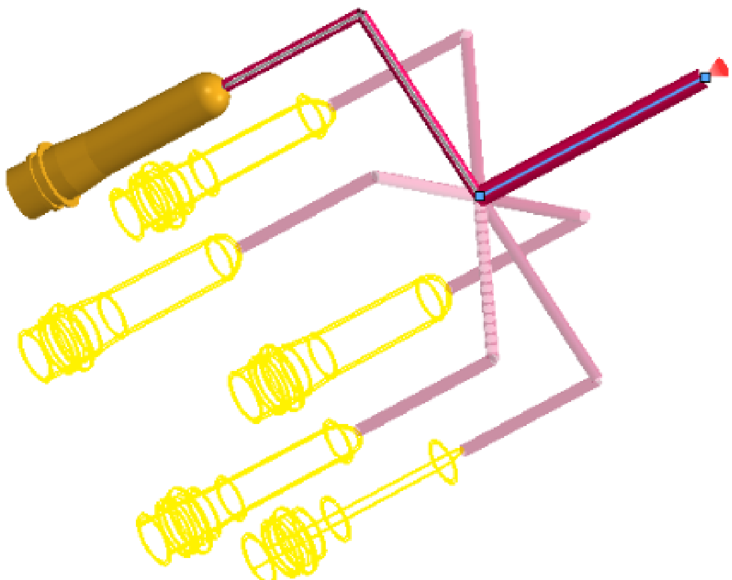
SOLIDWORKS Plastics

This chapter includes the following topics:

SOLIDWORKS® Plastics Standard, SOLIDWORKS Plastics Professional, and SOLIDWORKS Plastics Premium are separately purchased products that you can use with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium.

	**Video: What's New in SOLIDWORKS 2022 - SOLIDWORKS Plastics**

Cavity and Runner Layouts



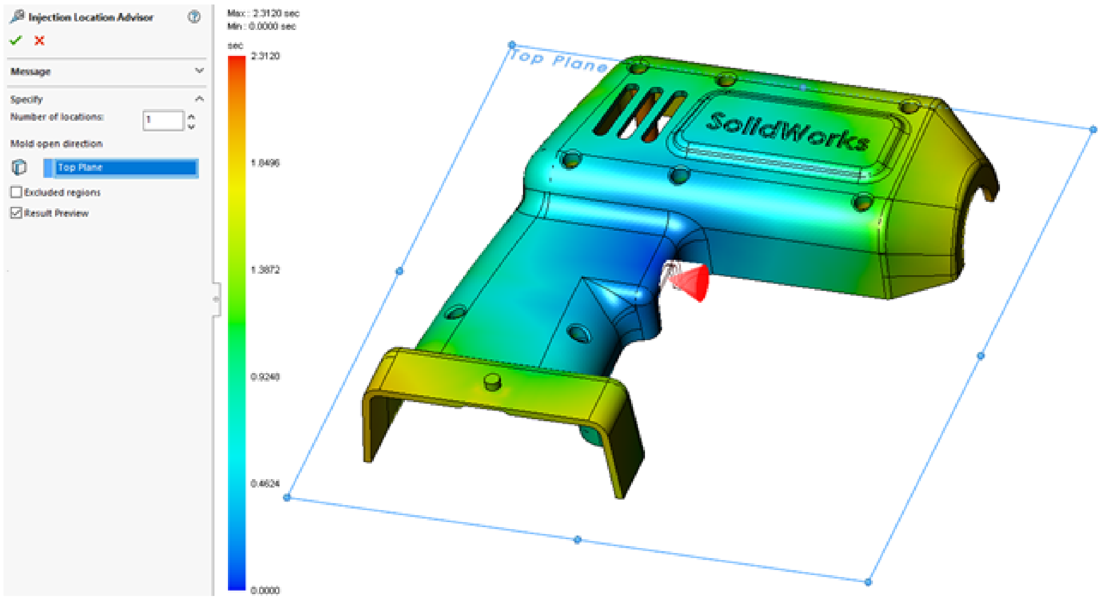
You can create dedicated boundary conditions for cyclic and symmetric cavity and runner layouts. You can also preview the cavity and runner layouts during the modeling stage to confirm their design specifications.

To open the Symmetry or the Cyclic PropertyManager:

In the PlasticsManager tree, right-click **Boundary Conditions** and click **Symmetry** or **Cyclic**.

You can use solid bodies and sketch-based runners to create the layouts. For cavity layouts with symmetry conditions, you can also visualize results for the whole layout, even though the simulation runs only for the symmetric part.

Injection Location Advisor



The **Injection Location Advisor** assesses a part's geometry to identify up to four suitable injection locations by considering the specified material, process conditions, and mold opening direction.

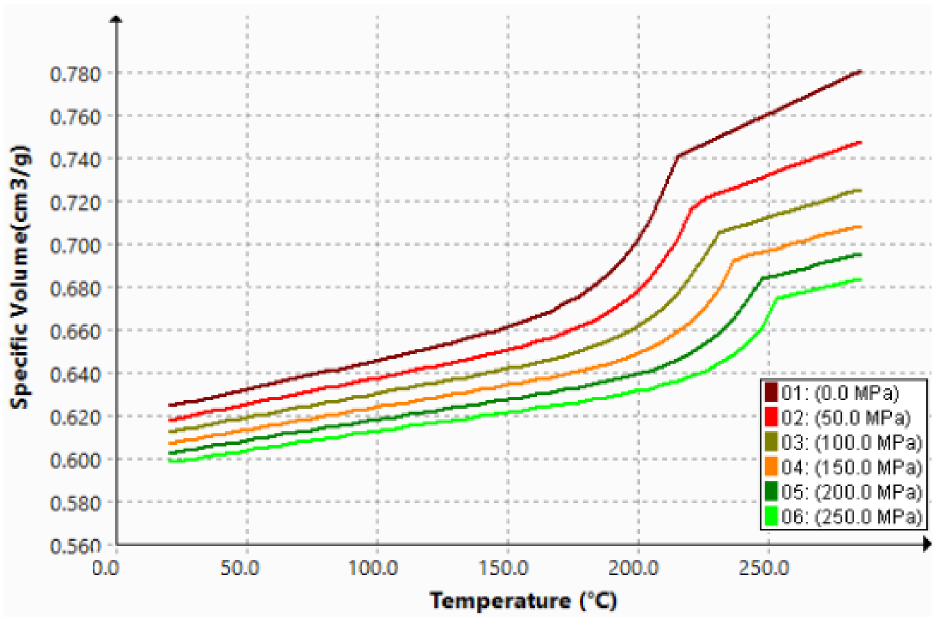
To open the Injection Location Advisor PropertyManager:

In the PlasticsManager tree, right-click **Boundary Conditions** and click **Injection Location Advisor**.

The following table describes the options of the Injection Location Advisor PropertyManager.

Number of locations	Specifies the number of the suitable injection locations (maximum of four).
Mold open direction	Specifies the plane for the mold open direction. The default is the **Front Plane**, which corresponds to the positive Z-axis.
Excluded regions	Defines regions of a model that the **Injection Location Advisor** excludes from consideration.
Result Preview	Shows the recommended injection locations and a preview of the fill plot. You can predict how the plastic material fills the mold based on the recommended injection locations.

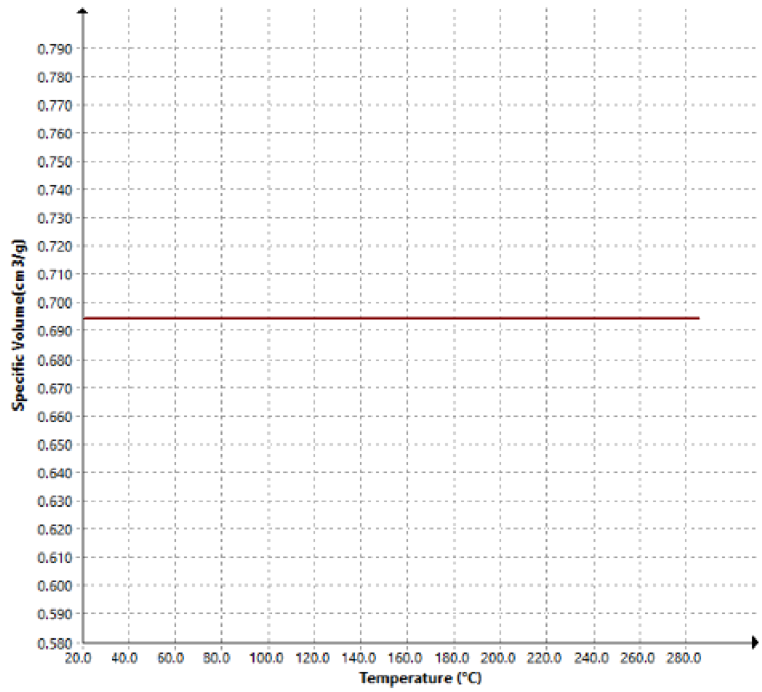
Plastics Materials Database



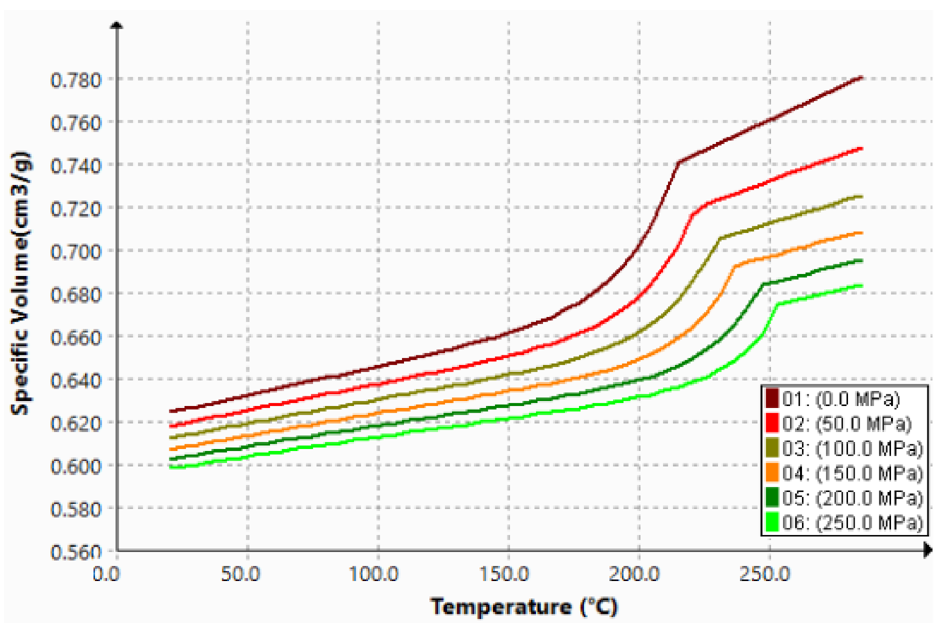
The Plastics material database is updated according to the latest data from the material manufacturers.

New Materials	Modified Materials	Removed Materials
Added 112 new material grades from these material manufacturers: • SABIC Specialties: 49 • Polyplastics: 40 • Solvay Specialty Polymers: 16 • RadiciGroup: 6 • LANXESS: 1	• Renamed 441 grades according to the latest data from the SABIC website • Consolidated seven different SABIC manufacturer categories under the single category SABIC Specialties • Updated 1167 grades from constant density to generic Pressure Volume Temperature (PVT) data to improve the accuracy of Fill, Pack, and Warp simulations	Removed 76 materials from the database because they were duplicates or obsolete per manufacturer: • SABIC Specialties: 29 • LANXESS GmbH: 17 • Polyplastics: 4 • BASF: 4 • ICI: 4 • ARKEMA: 3 • DuPont Engineering Plastics: 2 • DuPont Engineering Polymers: 2 • Rhodia Engineering Plastics: 2 • Rhone-Poulenc: 2 • KUO FU: 2 • CWH, Chemwerk Huls: 2 • DSM Engineering Plastics: 2

New Materials	Modified Materials	Removed Materials
		• UBE: 1

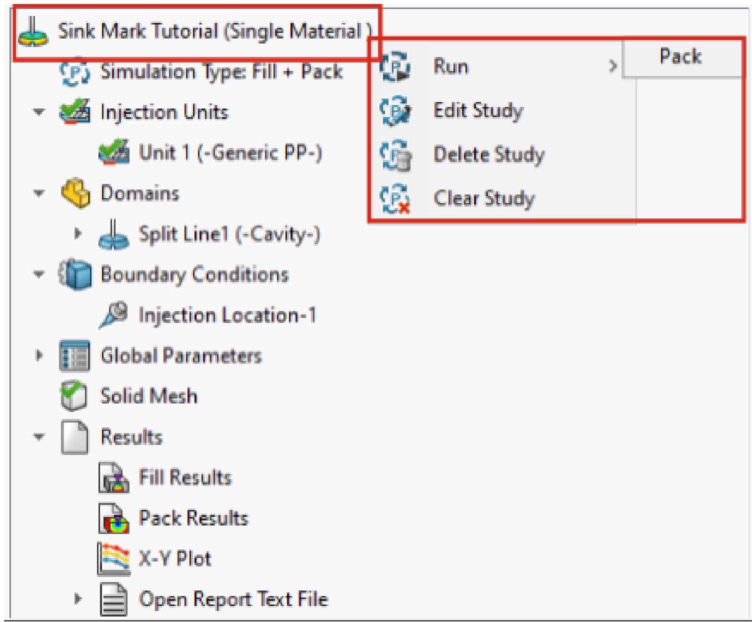


Material grade with constant Pressure Volume Temperature (PVT) data (2021)



Same material grade updated with generic PVT data (2022)

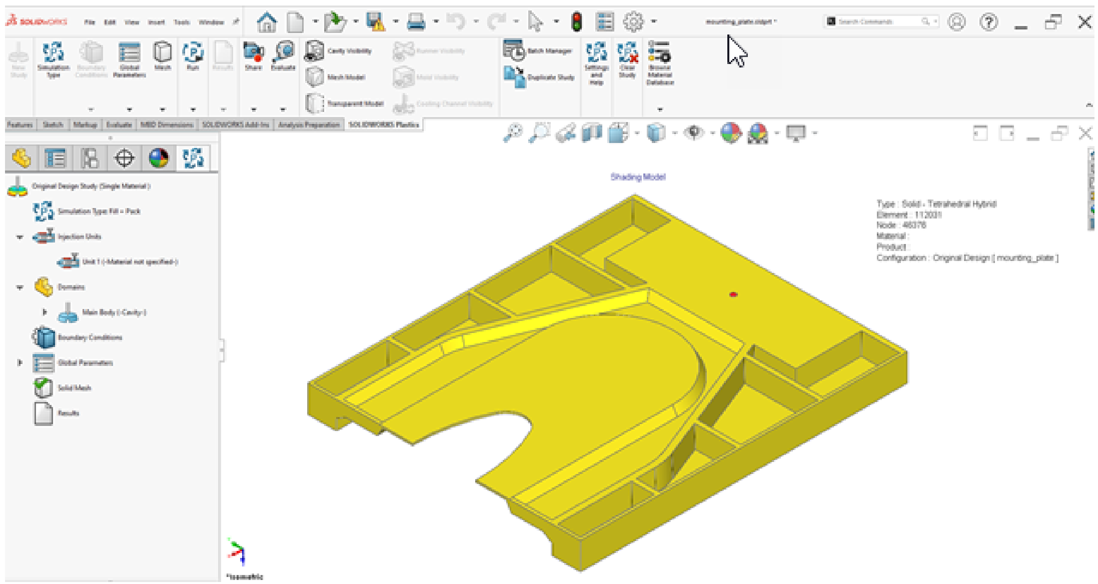
PlasticsManager Tree



Usability is improved for the PlasticsManager tree.

- The PlasticsManager tree displays the simulation type under the study node to provide more clarity on the active analysis module (Fill, Pack, Cool, and Warp).
- Modeling features and boundary conditions are filtered based on the simulation type to provide a simulation-centric user experience.
- You can run a simulation without explicitly creating a mesh if you have defined an injection unit and injection location. SOLIDWORKS Plastics automatically generates a mesh before the start of the simulation.
- You can delete the mesh from the **Solid Mesh** (or **Shell Mesh**) node of the PlasticsManager tree. Right-click **Solid Mesh**, and click **Delete Mesh**.
- The **Results** node becomes visible after you create a mesh.
- You can right-click the top study node and click **Run** to run a simulation.
- You can right-click **Results** to access these features:
  - **Summary and Report**
  - **Clipping Plane Settings**
  - **Isosurface Manager**
  - **Path Line**
  - **Export**
  - **Remove All Results**
- You can delete the results for a specific analysis module. For example, right-click **Fill Results** and click **Remove Results**. In previous releases, only **Remove All Results** was available.
- The term Fill replaces the term Flow in all user interface instances to match conventional industry terminology.

Scaling for High-Resolution Displays

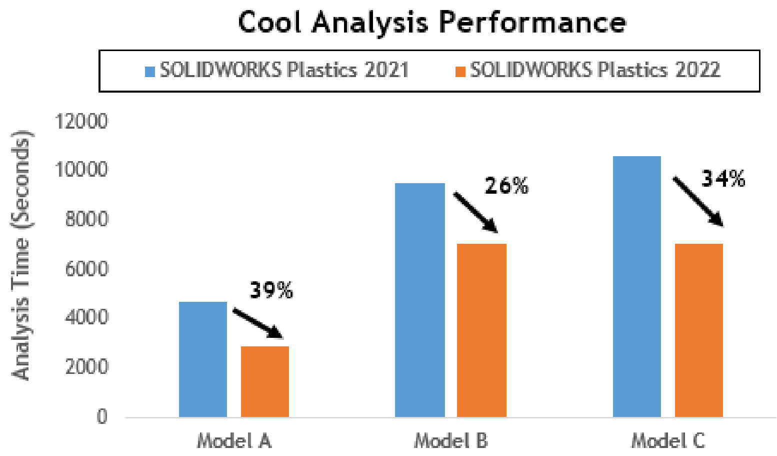


SOLIDWORKS Plastics supports monitors with 4K and higher resolution displays.

The user interface icons scale to an appropriate size so that their appearance does not degrade or become blurry on high-resolution and high-pixel density displays. The user interface of the PlasticsManager tree, dialog boxes, and PropertyManagers responds to the Microsoft Windows® display scaling setting.

Icons with text scale to a size appropriate for the text.

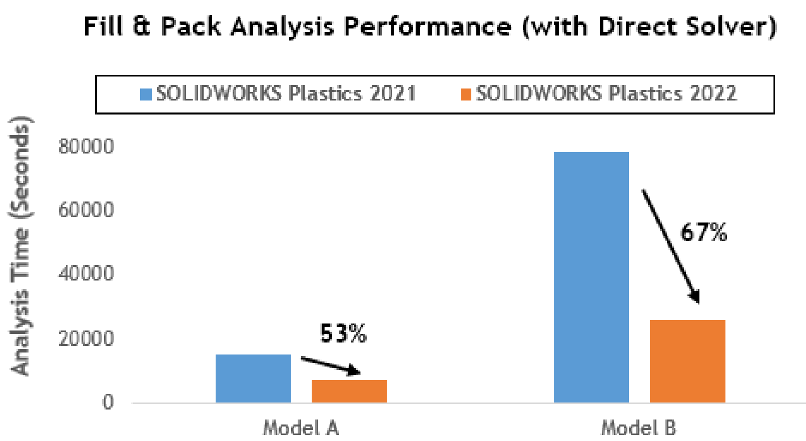
SOLIDWORKS Plastics Solvers



The performance of the Cool and Fill analysis modules is improved to accelerate the overall analysis time.

- For simulations where the Cool analysis takes up a large proportion of the overall solution time, the overall solution time is reduced by at least 20% compared to previous releases. The image shows the performance gains of a Cool analysis for three models that have a varying number of elements.
- The performance of the Fill and Pack analyses with the **Direct** solver option is optimized. The overall solution time is reduced by approximately 50% compared to previous releases. For relatively thick parts that are meshed with hexahedral elements, the **Direct** solver more accurately predicts the inertial effects.

  The following image shows the performance gains for the Fill and Pack analyses of two models that have a varying number of elements.



To access the **Direct** solver, from the PlasticsManager, select **Global Parameters** > **Fill-Pack** > **Solver Settings** > **Volume of Fluids (VoF) Algorithm** > **Direct**.

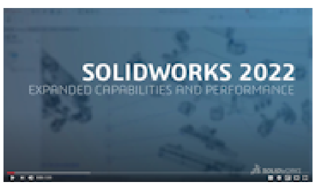
26

Routing

This chapter includes the following topics:

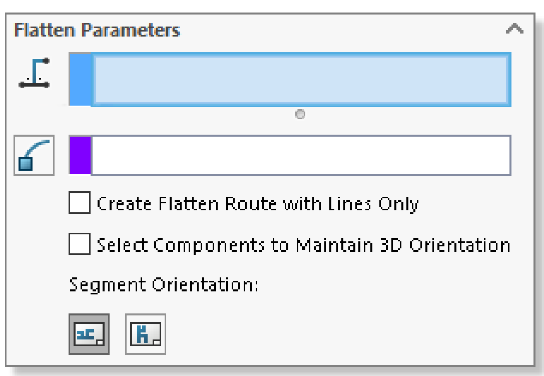
- **Flatten Route Improvements**
- **External Connectors in the Flattened Routes**
- **Backshells for Connectors**
- **Backshells and Flatten Routes**
- **Replacing a Connector in a Routing Assembly**

Routing is available in SOLIDWORKS® Premium.



Video: What's New in SOLIDWORKS 2022 - Routing

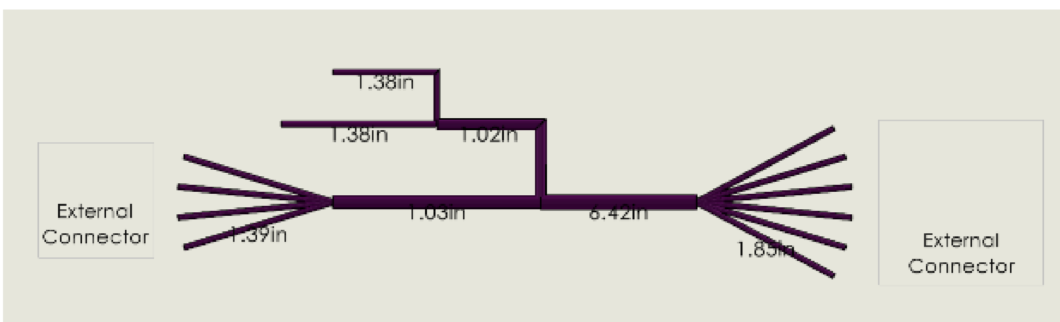
Flatten Route Improvements



The Flatten Route Property Manager lets you manage the creation of the flattened route from a route assembly.

- The tool to start the Flatten Route Property Manager is in the SOLIDWORKS Electrical menu.
- In **Horizontal Route Segment Selection**, you can select several continuous route segments to be appeared as horizontal in the flatten configuration.
- **Create Flatten with Lines Only** allows you to convert the splines to lines.
- The Flatten Route Property Manager automatically detects the connected segments when you select the first sketch segment and displays them in a selected dialog box.
- Select **Components to Maintain 3D orientation for splices with loops** to have loops with splices maintain their 3D orientation wherever you place them.
- When the Edit Flatten Route Property Manager is open, left-click on a route segment in the graphics area to access modification tools.

External Connectors in the Flattened Routes

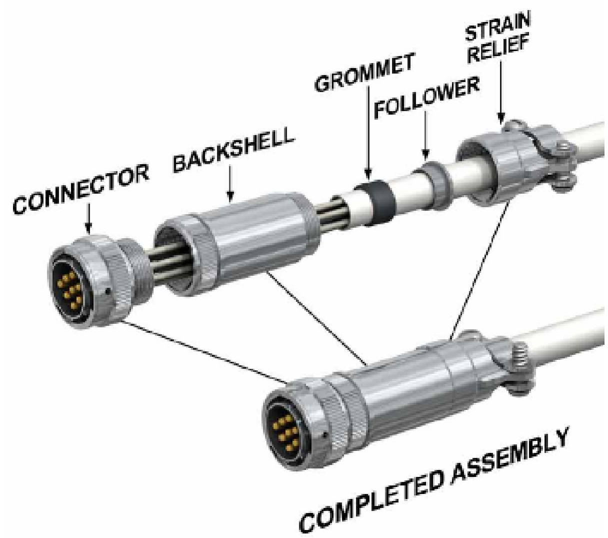


In the flattened routes, you can select connectors that are not in the harness but connected to the cables.

To select the connectors, in the Connector Tables Property Manager, click **Select All Connectors**, then remove those that you want to exclude.

The Flattened Route drawing displays the **External** connectors. In the **Circuit Summary** tables, the **From** and **To** columns display the **External** connector tags.

Backshells for Connectors

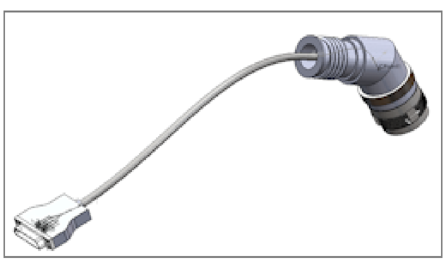


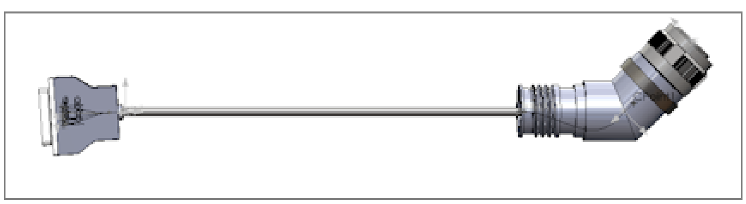
Backshells protect connectors and connected cables from electrical interference or physical damage due to environmental conditions. Backshells can include a clamping device.

To use a backshell, you must:

- Add an axis to the backshell to align the backshell with the route.
- Add a mate between the backshell and the connector.

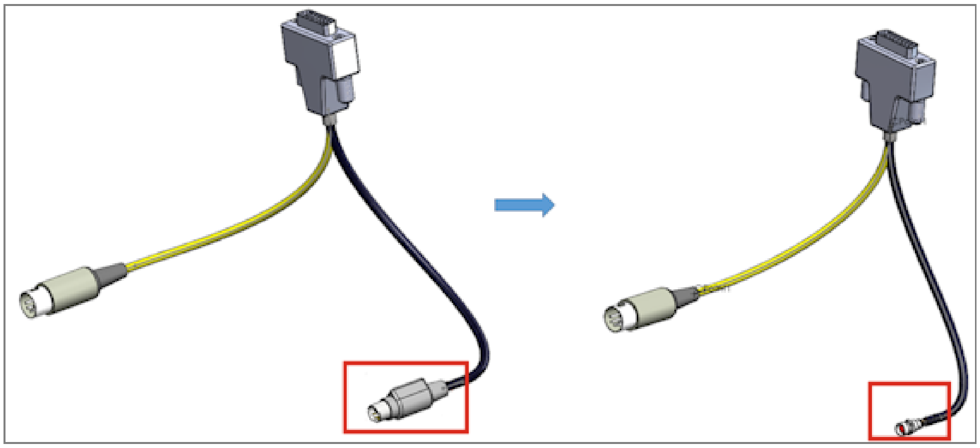
Backshells and Flatten Routes





When you flatten the Route Assembly having Backshell, the location and the orientation of the Backshell are displayed properly in the Flatten Route.

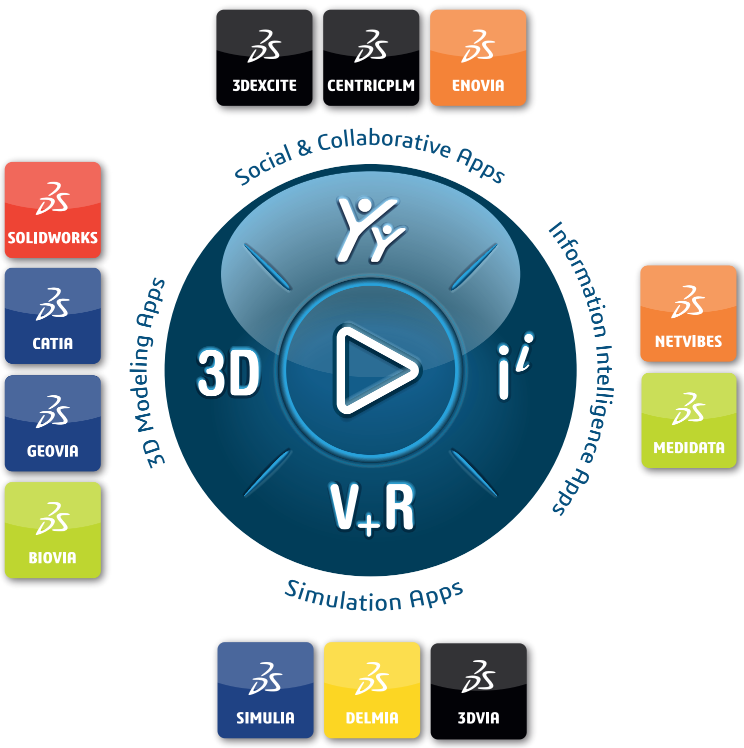
Replacing a Connector in a Routing Assembly



In a routing assembly, when you replace a connector in a **From/To** list, you can keep the original connections and the electrical data.

In the Import Electrical Data PropertyManager, select **Replace component** to replace the existing connector by the new one in the **From/To** list. Select **Delete existing segment and create new guidelines** if the new connector does not have the same connections.

Select **Update data** to add new data or to change the data of an existing component without resetting any unchanged components.



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