WHAT'S NEW
SOLIDWORKS 2024

Contents

1

Welcome to SOLIDWORKS 2024

This chapter includes the following topics:

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 2024 contains new, user-driven enhancements focused on:

- **Working Smarter**. Reduce your workload in SOLIDWORKS with the ability to defeature models more efficiently, add part features to assemblies by first associatively inserting an assembly into a part, and include unit of measure as a custom property in your notes and tables.
- **Working Faster**. Work more efficiently in SOLIDWORKS with intelligent, instant creation of sketch dimensions, improvements to collinear dimensioning for chain dimensions in drawings, and access to new components in Toolbox.
- **Working Together**. SOLIDWORKS is better together with your friends! Empower others across product development disciplines with enhancements to SOLIDWORKS products including PDM, Simulation, Electrical, Visualize, MBD, Composer, and more. Best yet, SOLIDWORKS now includes access to the **3D**EXPERIENCE® platform.

> This document covers all enhancements that affect how you interact with the **3D**EXPERIENCE platform. This includes both of the platform-connected versions of SOLIDWORKS - SOLIDWORKS Connected and SOLIDWORKS with the 3DEXPERIENCE (Design with SOLIDWORKS) add-in. It also includes other apps that can connect to the platform such as DraftSight.

Top Enhancements

The top enhancements for SOLIDWORKS® 2024 provide improvements to existing products and innovative new functionality.

Parts and Features	• **Hole Wizard** on page 86 • **Making Multibody Parts from Assemblies** on page 87
Sheet Metal	• **Slot Propagation** on page 97 • **Stamp Tool** on page 99 • **Normal Cut in Tab and Slot** on page 101
Structure Systems and Weldments	• **Corner Management** on page 102
Assemblies	• **Defeature Rule Sets** on page 118 • **Repairing Missing References in Linear or Circular Component Patterns** on page 124
Drawings and Detailing	• **Overridden Dimensions** on page 131 • **Keeping Chain Dimensions Collinear** on page 130 • **Reattaching Dangling Dimensions** on page 132
SOLIDWORKS MBD	• **Hole Tables** on page 234 • **Repairing Dangling Dimensions** on page 234

Performance

SOLIDWORKS® 2024 improves the performance of specific tools and workflows.

Some of the highlights for performance and workflow improvements are:

SOLIDWORKS Fundamentals

- Graphics rebuild after exiting SOLIDWORKS options.

  SOLIDWORKS checks the changed options when you click **OK** to exit the Options dialog box. SOLIDWORKS only performs a graphics rebuild on the active document if the changed options require it. In earlier releases, SOLIDWORKS always performed a graphics rebuild on the active document.

- Silhouette edges.

  You can enable the GPU hardware to improve the display of silhouette edges in HLR, HLV, and wireframe views.

  In **Tools** > **Options** > **System Options** > **Performance**, select **Hardware accelerated silhouette edges**.

Sketching

Equal relations solve more efficiently which improves 3D sketch performance.

Sheet Metal

When rebuilding complex sheet metal parts with large numbers of sketched bends or jogs, rebuild time is improved by up to 50%.

Import/Export

The performance of importing `STEP`, `IGES`, and `IFC` assemblies as multibody parts is improved up to 30%.

SOLIDWORKS PDM

SOLIDWORKS PDM 2024 has improved the performance of file-based operations.

The following operations are approximately two times faster:

- Add files
- Change state
- Copy tree

  The copy tree to compressed archive operation is orders of magnitude faster.

SOLIDWORKS Electrical

- Archiving a project for remote users (VPN connection) is improved and is much faster.
- The automatic routing issue that caused the creation of loops while routing wires through splices is fixed. This allows cleaner and faster flattening of harnesses.

eDrawings

Performance improvements include:

- **Measure** tool. Up to 20 times faster when opening the Measure pane, entity selection, and changing units.
- **Markup** tool. Up to 10 times faster when creating markups.
- **Reset** tool. Up to 1.5 times faster when resetting a model.
- Faster rendering and printing with software OpenGL.
- Faster times for closing files.

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**.

2

Using SOLIDWORKS on the 3DEXPERIENCE Platform

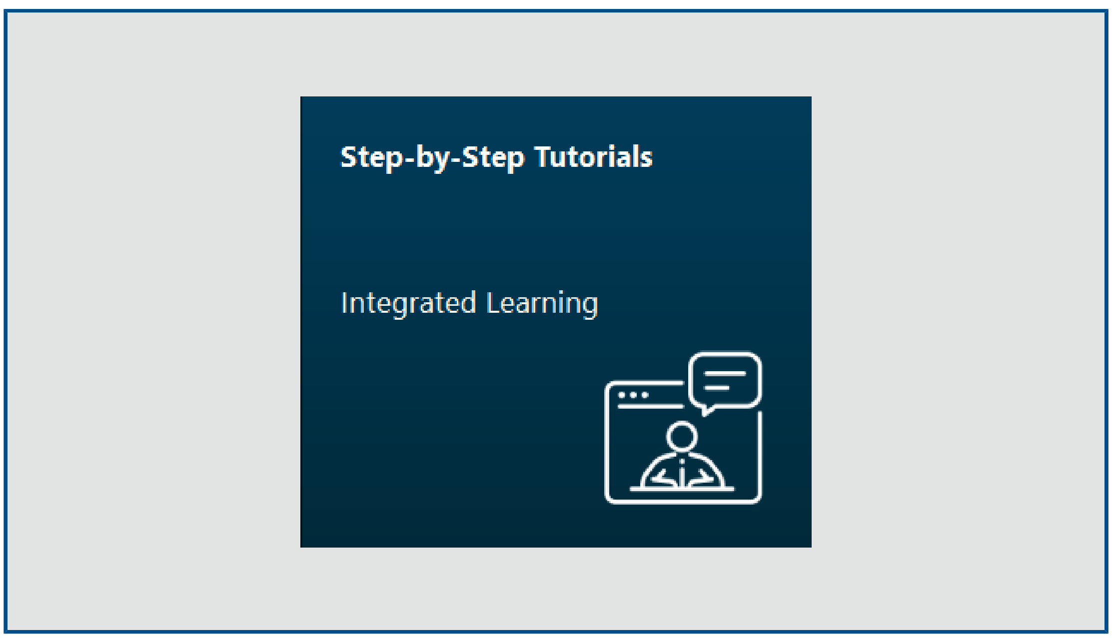
This chapter includes the following topics:

- **SP3-FD03**
- **SP2-FD02**
- **SP1-FD01**
- **SP0_GA**

This chapter covers all enhancements that affect how you use SOLIDWORKS with the 3DEXPERIENCE platform. Unless otherwise noted, the entries in this chapter are available in both SOLIDWORKS Connected (3DEXPERIENCE SOLIDWORKS roles) and in SOLIDWORKS with the 3DEXPERIENCE (Design with SOLIDWORKS) add-in (Collaborative Designer for SOLIDWORKS role).

SP3-FD03

SOLIDWORKS Connected Tutorials (2024 FD03, 2024 FD01)



You can access interactive SOLIDWORKS Connected tutorials that open in a resizeable viewer panel on the right side of your browser. Additional SOLIDWORKS Connected tutorials are available.

Benefits: You can access interactive tutorials directly in the app to help you learn SOLIDWORKS Connected. In previous releases, you had to use a browser to access these tutorials.

To access the tutorials, in the Welcome dialog box, click **Learn** > **Step-by-Step Tutorials**, or in the app, click **Help** > **Tutorials**.

The following tutorials are available:

Area	Tutorials
Basic Techniques	• Assembly Mates • Import/Export • Sheet Metal: Forming Tools • Surfaces
Advanced Techniques	• 3D Sketching • 3D Sketching with Planes • Advanced Design Techniques • Assembly Visualization • Equations • Mold Design • Multibody Parts • Sketch Blocks
Design Evaluation	• Animation • DimXpert
Productivity Tools	• Design Checker • Mouse Gestures • Smart Components • SOLIDWORKS Utilities

Several tutorials include downloadable models that you use to accomplish hands-on tasks to help support learning.

All of our existing SOLIDWORKS Connected tutorials remain available at **help.solidworks.com**.

SOLIDWORKS PDM Add-In for SOLIDWORKS Connected (2024 FD03)

In SOLIDWORKS Connected, the default data management system is the **3D**EXPERIENCE platform, but you can choose another system, such as the SOLIDWORKS PDM add-in.

Benefits: For dedicated PDM users, it is advisable to switch to the Data Management option, **SOLIDWORKS PDM or other separately installed data management**. This action deactivates **3D**EXPERIENCE integrations, which may cause conflicts or distractions for SOLIDWORKS PDM users.

To use a different data management system:

1. Click **Tools** > **Options** > **3DEXPERIENCE Integration** and select **SOLIDWORKS PDM or other data management installed separately**.
2. Click **OK**.

   This option requires a SOLIDWORKS restart.

Selecting another system removes the **3D**EXPERIENCE platform elements responsible for managing documents in collaborative spaces:

- MySession does not appear in the **3DEXPERIENCE Task Pane**.
- Lifecycle and Collaboration tools are not available in the CommandManager and menus.
- **Open** and **Save** operations cannot access the **3D**EXPERIENCE platform.
- The **3D**EXPERIENCE **Files on This PC** tab does not appear.

You can share files with **3D**Drive and **3D**EXPERIENCE Marketplace regardless of the data management system.

You can install SOLIDWORKS PDM separately, following the guidelines outlined in the *SOLIDWORKS® PDM and SOLIDWORKS Manage Installation Guide*. If SOLIDWORKS PDM is already installed, users can activate it through the Add-Ins dialog box from **Tools** > **Add-Ins** , whether or not they choose to modify the Data Management option.

Improved Licensing Support for SOLIDWORKS Flow Simulation and SOLIDWORKS Plastics Add-Ins (2024 FD03)

If you own licenses for SOLIDWORKS Flow Simulation and SOLIDWORKS Plastics, you can enable them to run in SOLIDWORKS Connected.

Benefits: The add-ins install automatically, making these tools readily available within SOLIDWORKS Connected.

When installing SOLIDWORKS Connected, optionally select SOLIDWORKS Flow Simulation or SOLIDWORKS Plastics and enter your serial number. In the case of a network license, you must specify the address (`port@server`) of your SolidNetWork (SNL) License server.

Once you install SOLIDWORKS Flow Simulation and SOLIDWORKS Plastics:

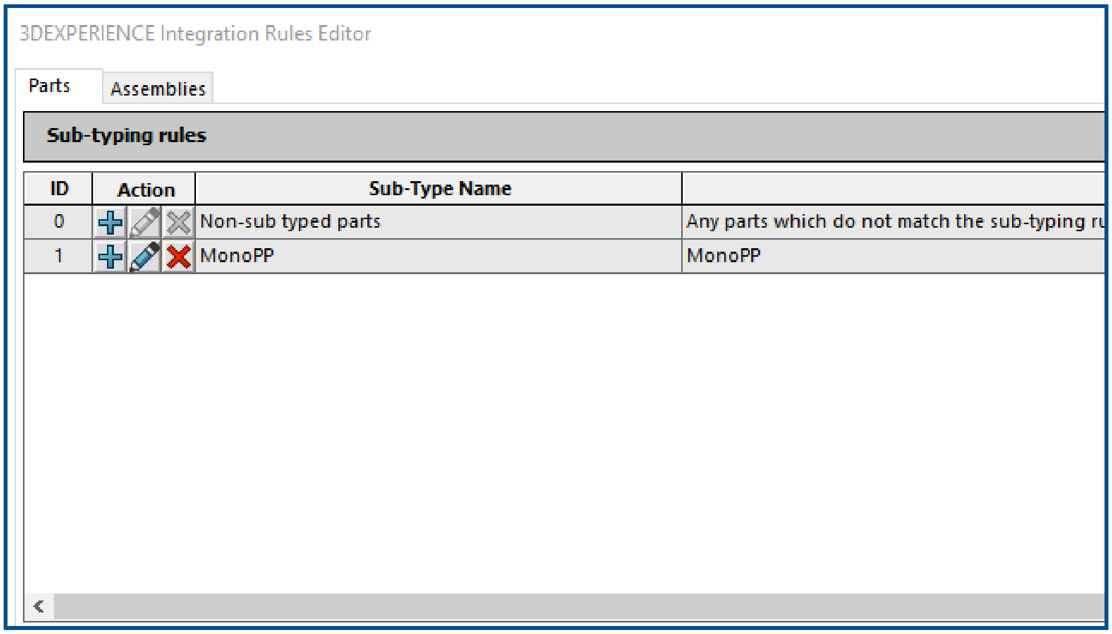
- You can activate or deactivate standalone versions from the **Help** menu in SOLIDWORKS Connected.
- SNL versions retrieve a license from the license server when you add them in.

File Preparation Assistant - Additional Checks (2024 FD03)

The File Preparation Assistant performs additional checks, including for files older than SOLIDWORKS 2021. This lets you find old files and save files in the latest version of SOLIDWORKS.

Benefits: More checks improve the success of saving your files to the **3D**EXPERIENCE platform.

Designate a Single Physical Product (2024 FD03)



When you use the **3D**EXPERIENCE Integration Rules Editor to designate a single physical product, you cannot add more physical products.

Benefits: You can define a single physical product in a consistent manner.

When you use the **Single physical product with representations** option in the **3D**EXPERIENCE Integration Rules Editor, the parts and assemblies within the scope of that rule should have the mono-physical product status, such as no CAD family in the ConfigurationManager.

In earlier releases, the model had a single physical product, however the model was not designated as a single physical product and you could add more physical products.

Refreshing PLM Information Only When Required (2024 FD03)

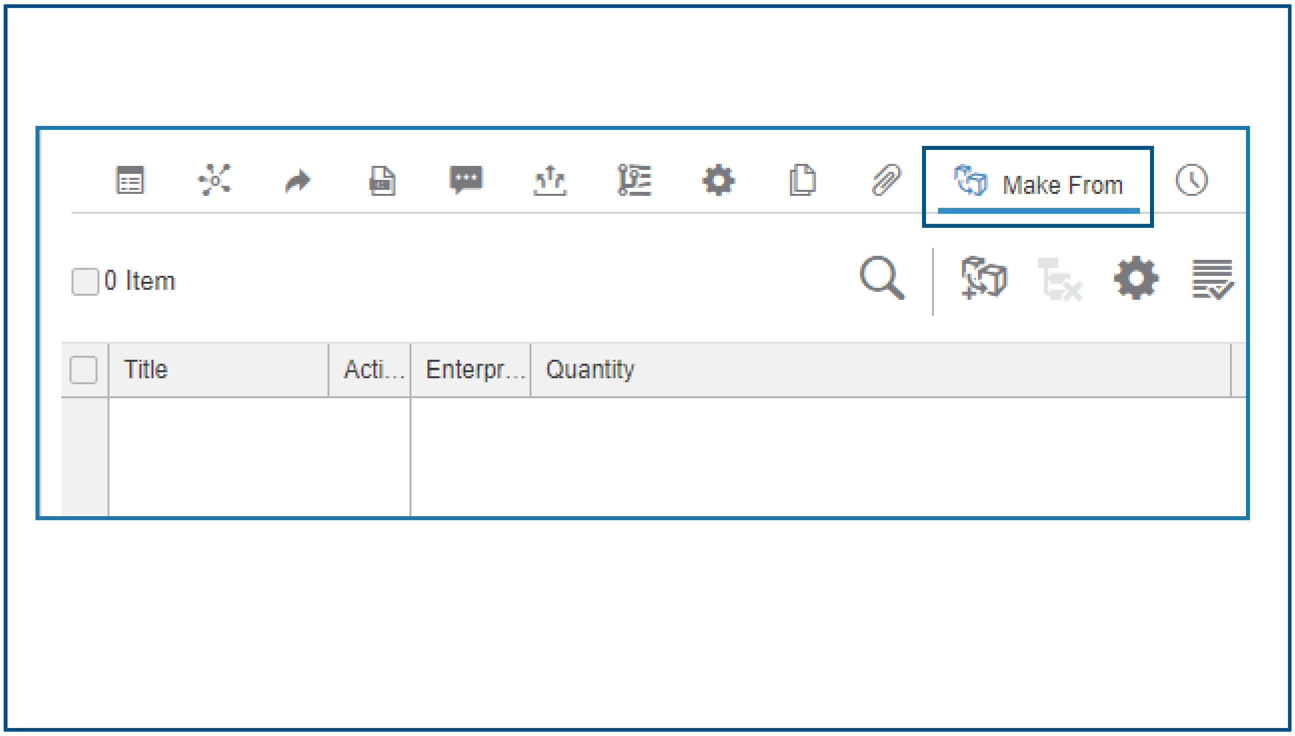
MySession content is refreshed only when required.

Benefits: This improves the performance of SOLIDWORKS as the time required to maintain the PLM information is saved.

With this change MySession content is refreshed only when any one of the following happens:

- Opening MySession from **View** > **Task Pane** option.
- Showing PLM information on the SOLIDWORKS feature manage tree.
- Accessing PLM commands from SOLIDWORKS.

Creating a Make From Relationship (2024 FD03)



You can use the **Make From** tab in the **Information** panel of an object to create a **Make From** relationship to a physical product or its subtypes.

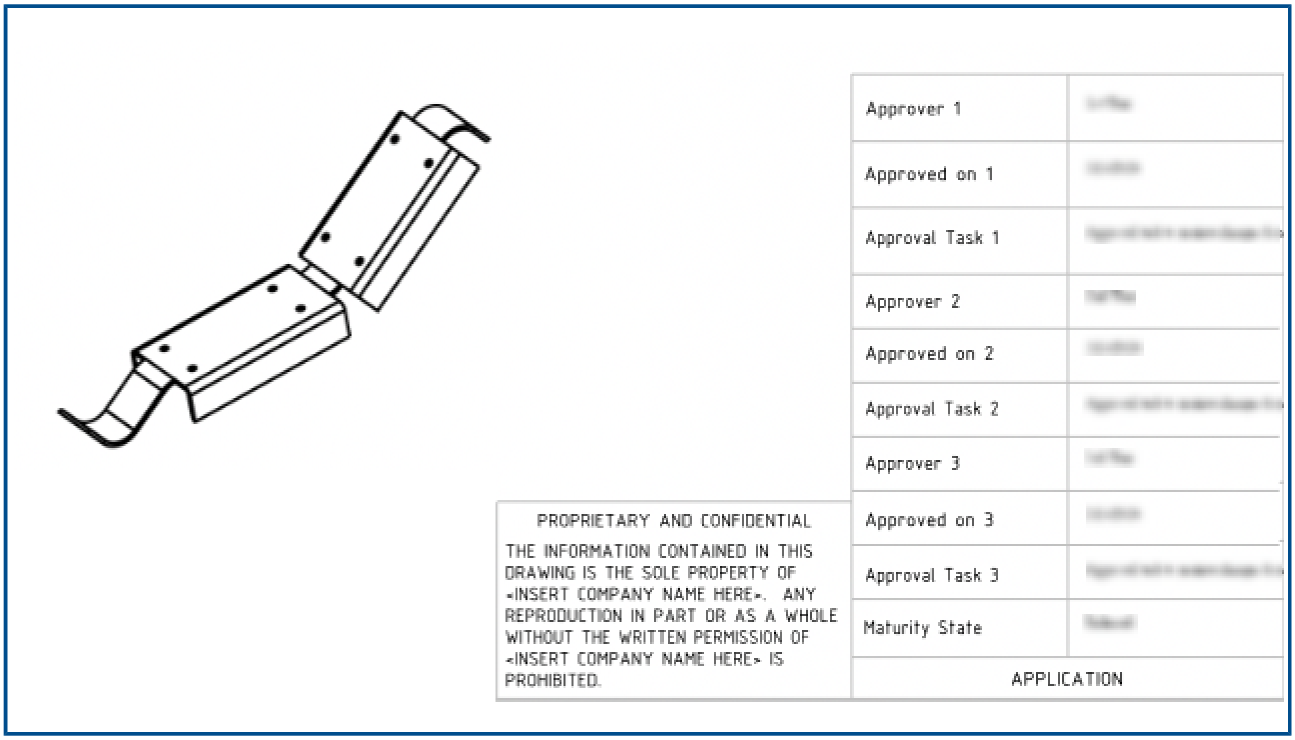
Benefits: You can review the materials assigned to a SOLIDWORKS product and if the materials are not assigned, assign them before releasing the document.

The **Make From** tab shows the name and quantity of objects needed to make the physical product. For an object when you select a 3D part, other physical products, raw materials, and their sub-types using the **Make From** option, a make from relationship is established between the two. This relationship is visible in the **Relations** tab of the **Information** panel.

To access **Make From**, from the **View** tab of action bar, click **Display Side Panel**.

The **Make From** tab displays the details of the object that is added as a material from which the object is made. Using the **Make From** command on this tab, you can link the objects.

Viewing Approval Details in Drawing Annotations (2024 FD03)



The extended attributes for a drawing in annotations are now expanded to display the approval details. You can now view the details of the approver through the annotations in **3DPlay** or **3DMarkup**.

Benefits: You can track the lifecycle of a drawing by viewing its properties in the preview.

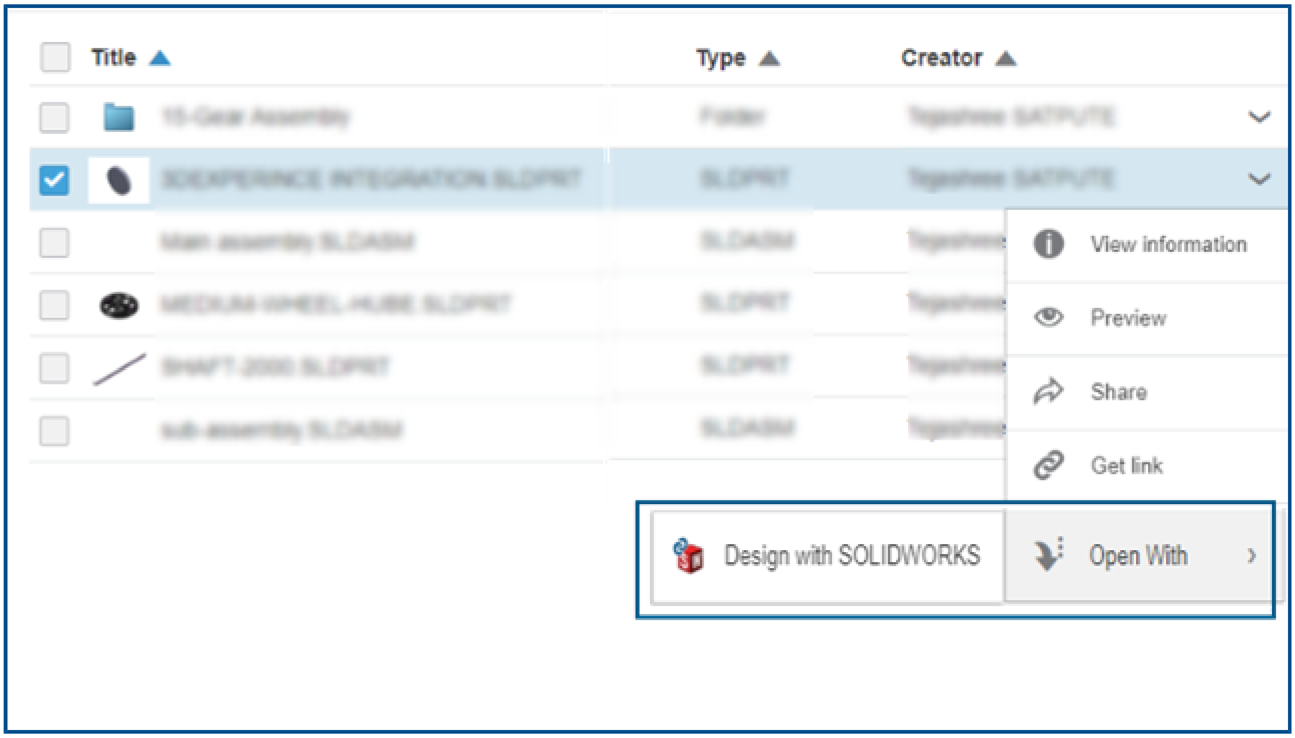
The drawing release process involves several approvers. If you view a drawing in **3DPlay** or **3DMarkup**, the information about the drawing release process (the list of approvers, the associated task, and the date of approval) are visible through the annotations.

The $PLMPRP properties are indexed corresponding to the approval order. The supported attributes are:

- `ea_releasedby.i`: represents the i^{th} (in time) approver of the drawing.
- `ea_releaseddate.i`: represents the date when the i^{th} (in time) approval is defined on the drawing.
- `ea_releasedtask.i`: represents the task title used when the i^{th} (in time) approval is defined on the drawing.

In the SOLIDWORKS properties dialog box, by default you can propose 3 approvers, but you can increase the number of approvers.

Installing Sync Client for 3DDrive (2024 FD03)

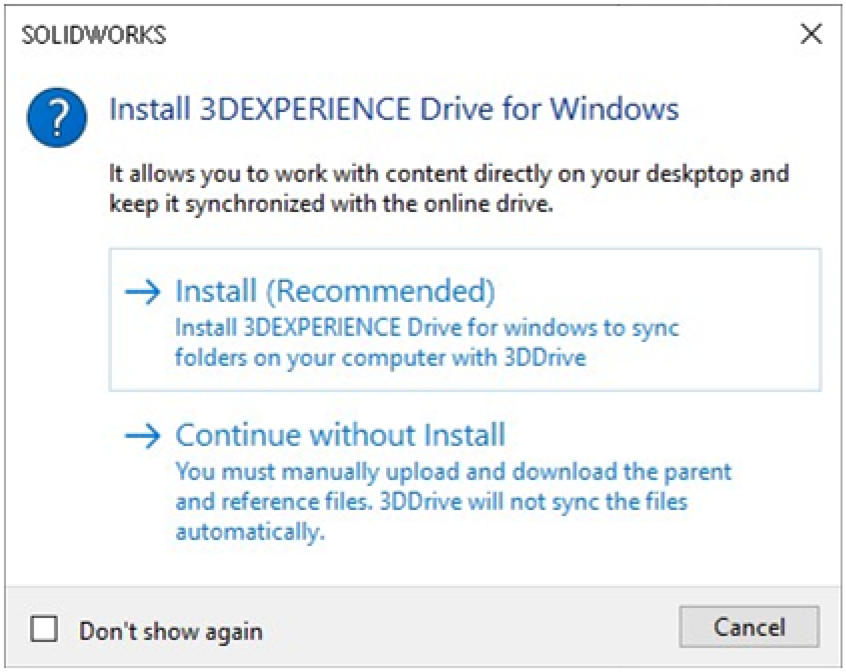


When you open a file from 3DDrive using the **Open With > Design with Solidworks** command, you can choose if you want to install the **3D**EXPERIENCE Drive for Windows.

Benefits: The app behaves differently depending on how you choose to install it. You can open the selected file in SOLIDWORKS even if the client is not installed on the machine.

A notification appears if you do not have **3D**EXPERIENCE Drive for Windows installed.

- If you choose **Install**, there is no change in the behavior of 3DDrive. You can work simultaneously with the files in SOLIDWORKS and keep it synced with 3DDrive.
- If you choose **Continue without Install**, the files will not be synced automatically. However, you can perform all the operations of upload, download, and drag a file from 3DDrive to SOLIDWORKS.



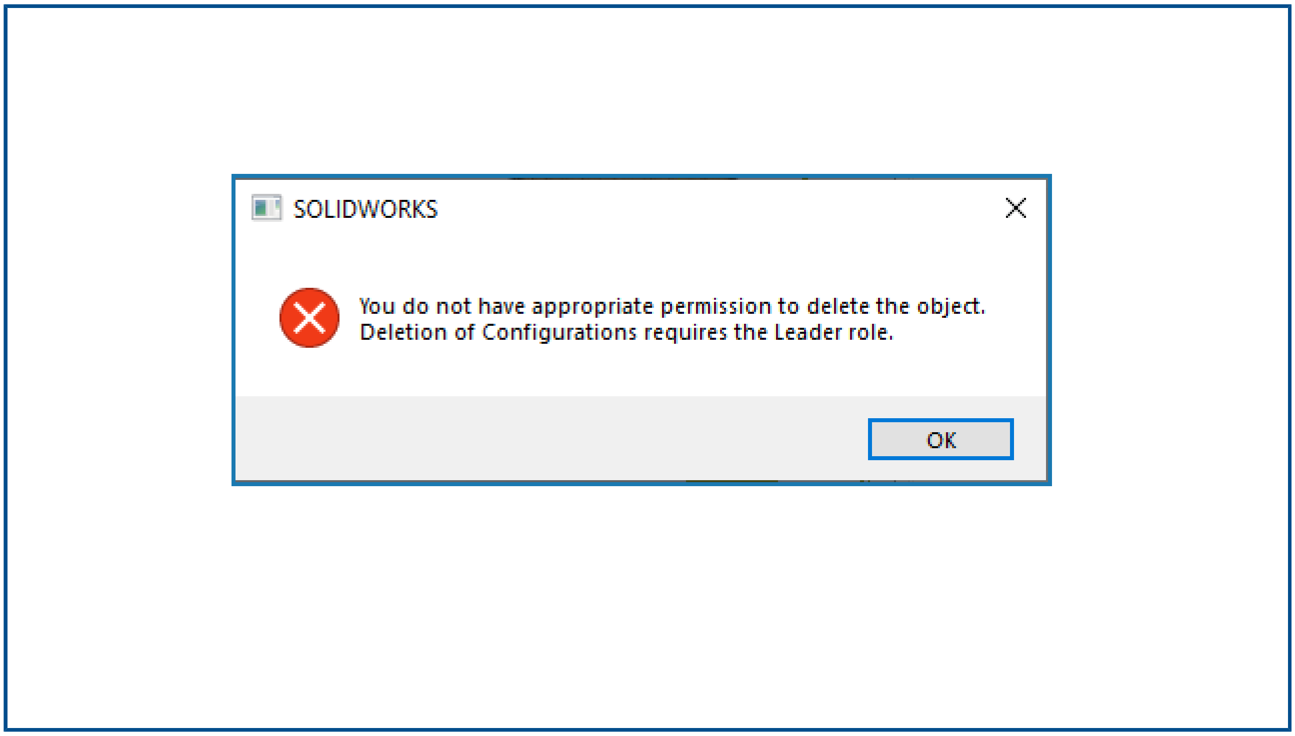
Accessing latest SOLIDWORKS Templates (2024 FD03)

When multiple revisions of the same template exist on the **3D**EXPERIENCE platform, only latest revision is downloaded.

Benefits: You always have access to the latest SOLIDWORKS templates stored in the **3D**EXPERIENCE platform.

If there are multiple templates with same filename, a single random template is downloaded. Also, if no modifications are done since the last download, the templates are not downloaded again locally.

Deleting Virtual Components (2024 FD03)

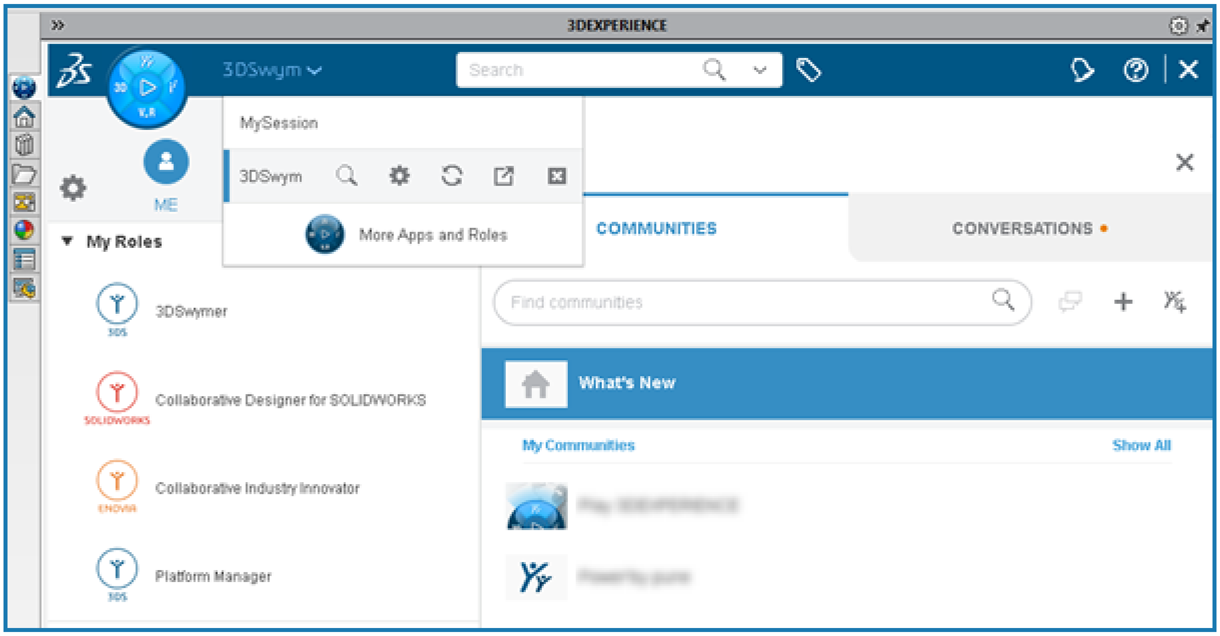


You can now delete a virtual part or a virtual assembly even if you are an Author.

Benefits: Deleting the virtual components in not dependent on the roles.

Now even if you delete the virtual components the save process does not get blocked. However the save process is blocked if you delete a configuration. To delete a configuration you must have the Leader role.

Opening 3DSwym from SOLIDWORKS (2024 FD03)

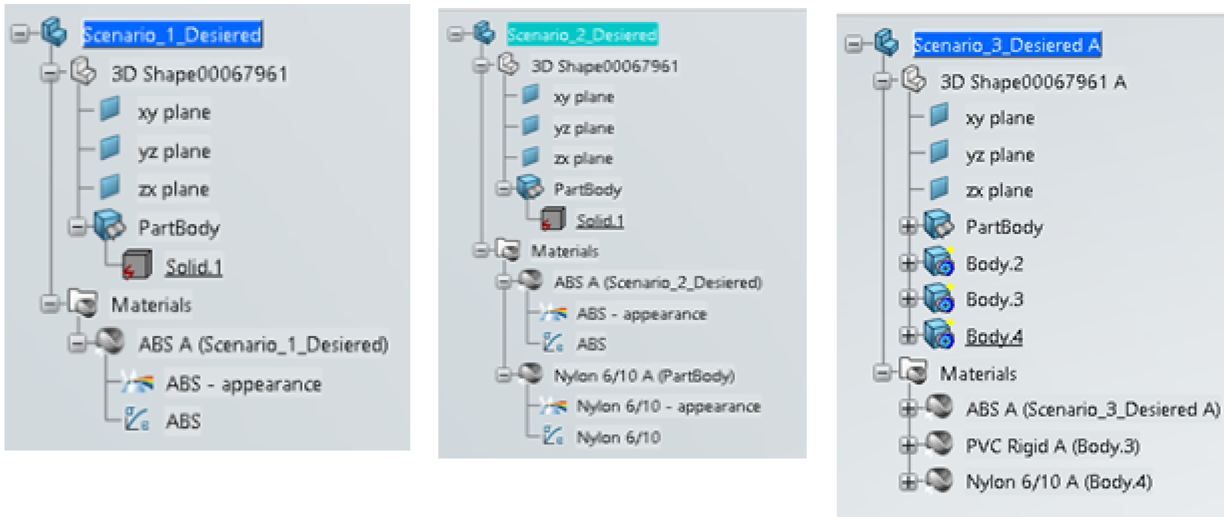


You can now open the 3DSwym app and notifications from the SOLIDWORKS task pane.

Benefits: You can access more **3D**EXPERIENCE platform functionality without leaving the SOLIDWORKS environment. The **3D**EXPERIENCE platform apps do not open in a separate web browser and thus saves the reloading time.

3DSwym helps you to collaborate and access communities and conversations. Once you open 3DSwymand then open any other app, you can reopen it again from the top bar by clicking . The notifications from the apps like Collaborative Tasks or 3DSwym **Conversations** open within the SOLIDWORKS task pane.

Applying Material to SOLIDWORKS Objects (2024 FD03)



Material applied at part level

Material applied at part and body level

Material applied at part level and selected bodies

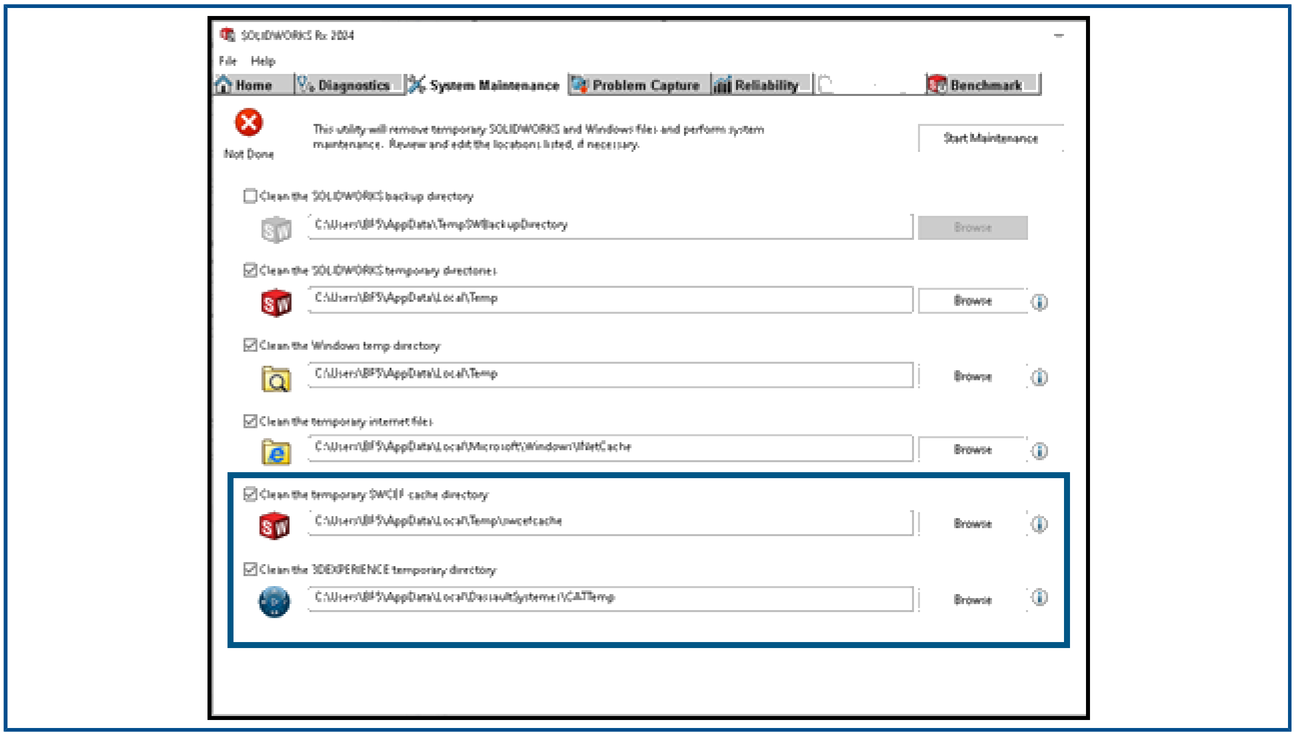
When you apply material to a part or body in SOLIDWORKS, the same material assignment and tree order structure is replicated in the **3D**EXPERIENCE platform.

Benefits: You can maintain the same design structure for structures that involve multibody parts.

In the earlier releases, when material was applied at part level or body level, the material definition was lost while saving it to the **3D**EXPERIENCE platform. Now when you apply material to a SOLIDWORKS part and save it to the **3D**EXPERIENCE platform, material exposition is managed in any of the following ways:

- Material applied at the part level is applied at the **3DPart** level in the **3D**EXPERIENCE platform.
- Material applied at the body level is applied at the body level in the **3D**EXPERIENCE platform.
- Material applied at the part and body level is applied at the **3DPart** and body level in the **3D**EXPERIENCE platform. For multibody structure, if material is applied at part level and some bodies, the material definition was applied to the bodies that did not have material definition. But now the bodies that do not have any material definition, do not display any material definition.

Updates to System Maintenance Tab in SOLIDWORKS RX (2024 FD03)



Two new tasks are available in the System Maintenance tab.

Benefits: These tasks simplify the diagnosis of technical issues.

- **Clean the temporary swcef cache directory**
- **Clean the 3DEXPERIENCE temporary directory**

> The **Clean the 3DEXPERIENCE temporary directory** task is only available when the Collaborative Designer for SOLIDWORKS app or **3D**EXPERIENCE SOLIDWORKS is installed.

When you work with support representatives, they may ask you to run these tasks to clean temporary files as a troubleshooting or corrective step. Content in these directories is recreated as necessary during normal SOLIDWORKS use.

These new tasks replace the following tasks:

- **Clean the temporary files in SOLIDWORKS data folders**
- **Run checkdisk to check for disk errors**
- **Run Windows Defragmenter**

SP2-FD02

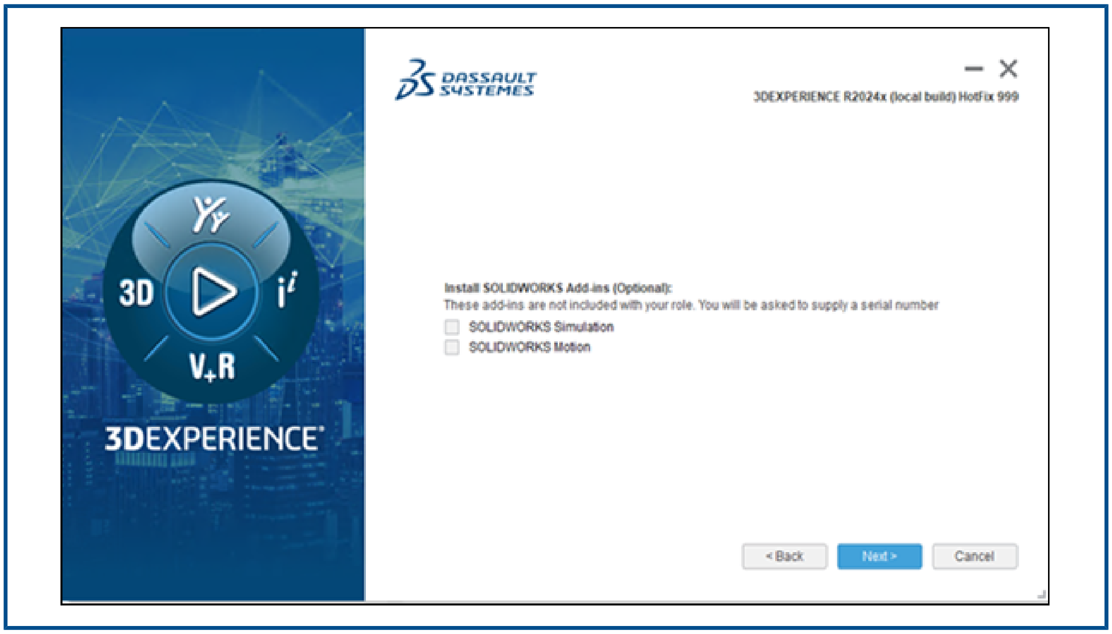
Support for the Turkish Language (2024 FD02)

SOLIDWORKS Connected supports Turkish menus and the user interface.

Benefits: This enhancement increases usability for Turkish users.

If you install SOLIDWORKS Connected 2024x HF2 on a Turkish version of Windows, you can use it with Turkish menus and interface. The **3D**EXPERIENCE Task Pane in SOLIDWORKS Connected does not support Turkish until a future release of the **3D**EXPERIENCE platform.

Improved Licensing Support for SOLIDWORKS Simulation and SOLIDWORKS Motion Add-ins (2024 FD02)



If you own licenses for SOLIDWORKS Simulation and SOLIDWORKS Motion, you can enable them to run in SOLIDWORKS Connected. During the installation of SOLIDWORKS Connected, you can select SOLIDWORKS Simulation or SOLIDWORKS Motion when prompted.

Benefits: The add-ins install automatically. There is no need to run the `addswxlicenses.exe` tool.

In the installation wizard, enter your serial number. For network licenses, you must provide an address, such as `port@server`, of your SolidNetWork License server.

After installing SOLIDWORKS Simulation and SOLIDWORKS Motion:

- You can activate or deactivate standalone versions through the **Help** menu in SOLIDWORKS Connected.
- The SolidNetWork License server retrieves licenses when you add them.

Notification of Updated Status When Opening Files (2024 FD02)

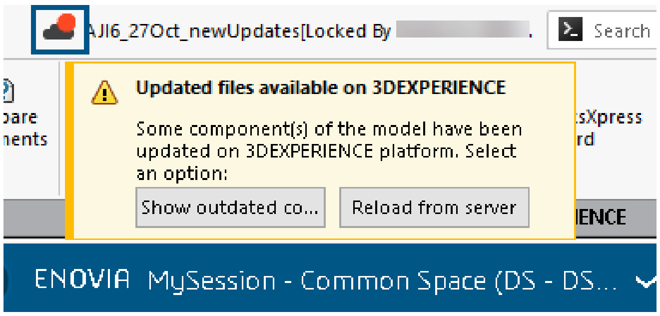
When the system opens **3D**EXPERIENCE files from your computer, the message bar notifies you about the new updates to the files on the platform.

Benefits: The notifications help ensure that you are always working with the latest version of your files.

Save Status

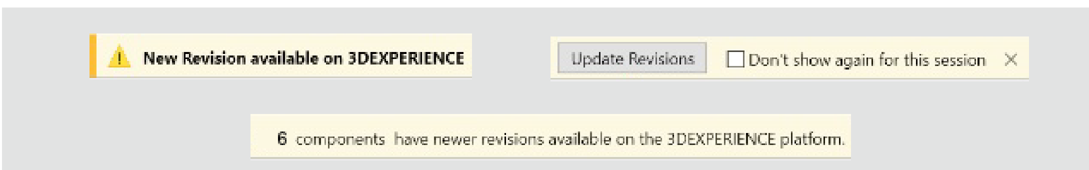
When the system opens **3D**EXPERIENCE files from your computer, the message bar notifies you about the new updates to the files on the platform.

When you refresh MySession, if any files have newer updates available on the platform, an orange dot on the cloud icon and a tooltip alert you in the title bar. You can select to show the outdated components or reload them from the server.



Revision Status

When the system opens individual or multiple **3D**EXPERIENCE assembly files from your computer, and where one or more components of the assembly have newer revisions on the platform, message bars notify you about the new revisions available on the platform.

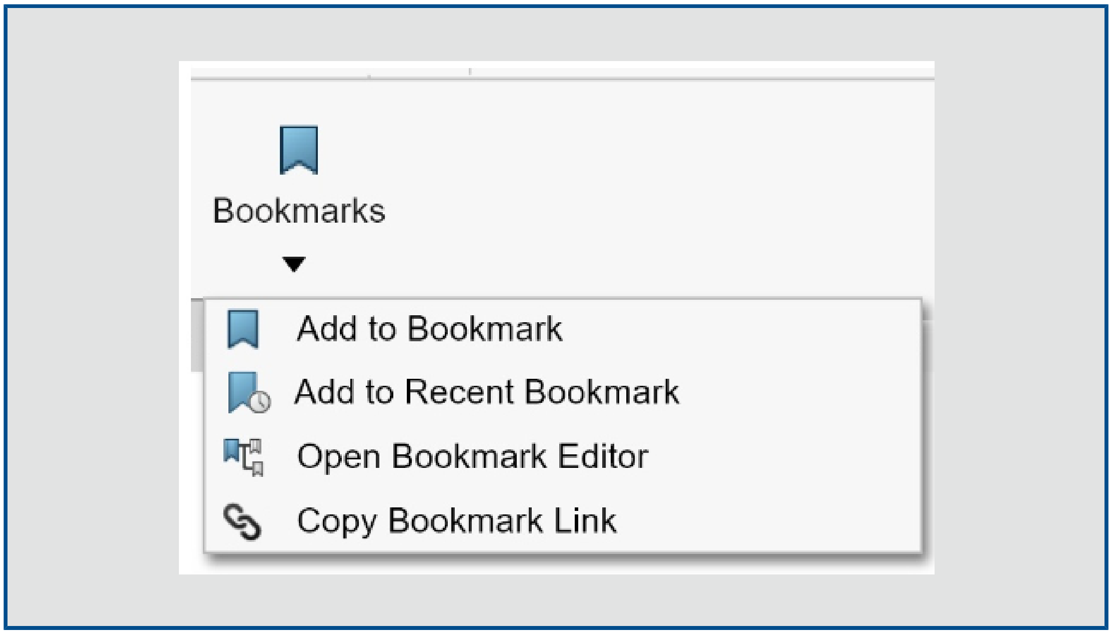


For files with revisions, you can update the revisions in the Update Revisions dialog box.

> To see this functionality, in MySession, on the action bar, click **Tools** > **Options** > **Open** and select **Refresh MySession after opening files**. Some scenarios might require a manual refresh of MySession.

In earlier releases, if you work with assemblies with a large number of components, you may have missed the visual status indicators in MySession.

Bookmarks (2024 FD02)



There are multiple enhancements to bookmarks.

Benefits: Improved organization, new tools and tooltips, and usability improvements help you work more efficiently.

Reorganized Commands

All bookmark commands are organized to appear on the Lifecycle and Collaboration CommandManager tab under the **Bookmark** tool.

- **Add to Bookmark**
- **Add to Recent Bookmark** (new)
- **Open Bookmark Editor**
- **Copy Bookmark Link** (new)

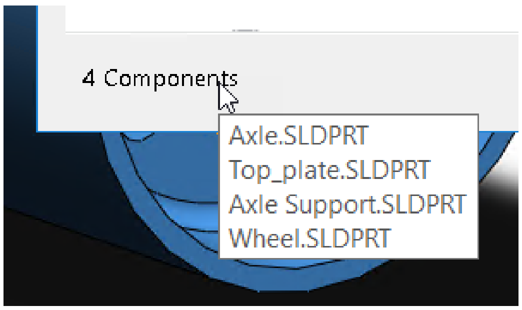
New Tools

The **Add to Recent Bookmark** tool adds a file or selected objects to a **Recent Bookmark**. You can add a bookmark to the 30 most-recent bookmarks. Select the object, click **Add to Recent Bookmark**, and select the recent bookmark to which to add the objects.

The **Copy Bookmark Link** tool creates a link to bookmarked objects that you can share with others. Select components and click **Copy Bookmark Link** to open the **Bookmark List**. Select a bookmark and click **Copy Link**. The system notifies you of the copy. You can then share that link with others in 3DSwym, email, or other methods of communication.

Tooltips

When you use the **Add to Bookmark**  command, in the Select a Bookmark dialog box that appears, tooltips list the full names of all the selected components that you are bookmarking. In earlier releases, the full names were truncated. In addition, if you add multiple files to a bookmark, for example from an assembly FeatureManager design tree, the number of components appears at the bottom of the Select a Bookmark dialog box. Hover over that text to reveal the full names of the components.

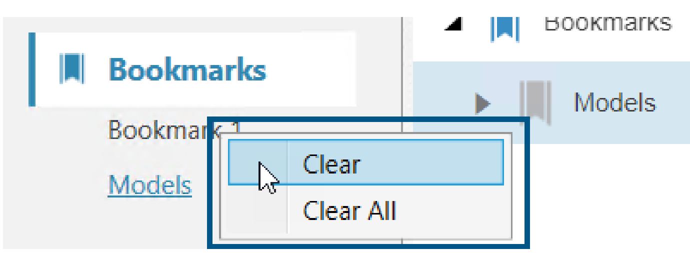


Usability

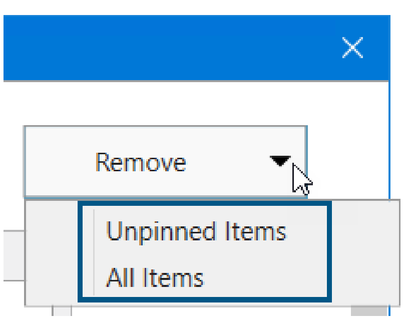
When you click **Open Bookmark Editor** and have already bookmarked files, the editor navigates to the bookmarked location of the file. If the file has not been bookmarked, the editor navigates to the last interacted bookmark location. In earlier releases, the Bookmark Editor opened with no predetermined location.

In the Open from 3DEXPERIENCE dialog box:

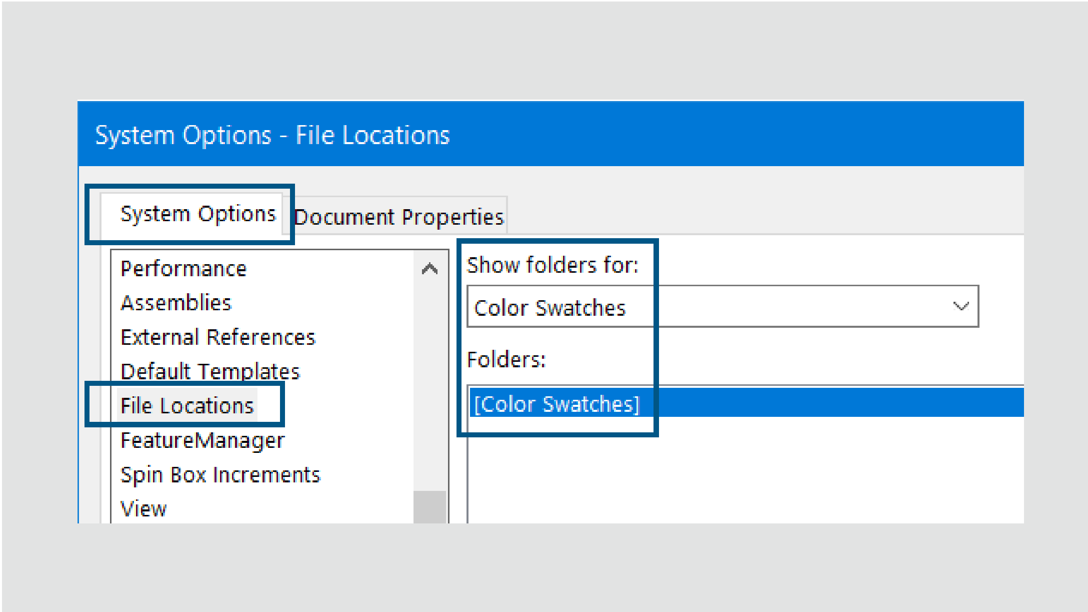
- On the Recent tab, under the list of recently visited bookmarks, you can right-click a bookmark and click **Clear** to clear that recent bookmark, or click **Clear All** to clear all recent bookmarks.



- On the Recent tab, in the upper-right corner, you can click **Remove** and select to remove **Unpinned items** or **All Items** from the tiled list of recent items.



Bookmark Support for File Locations



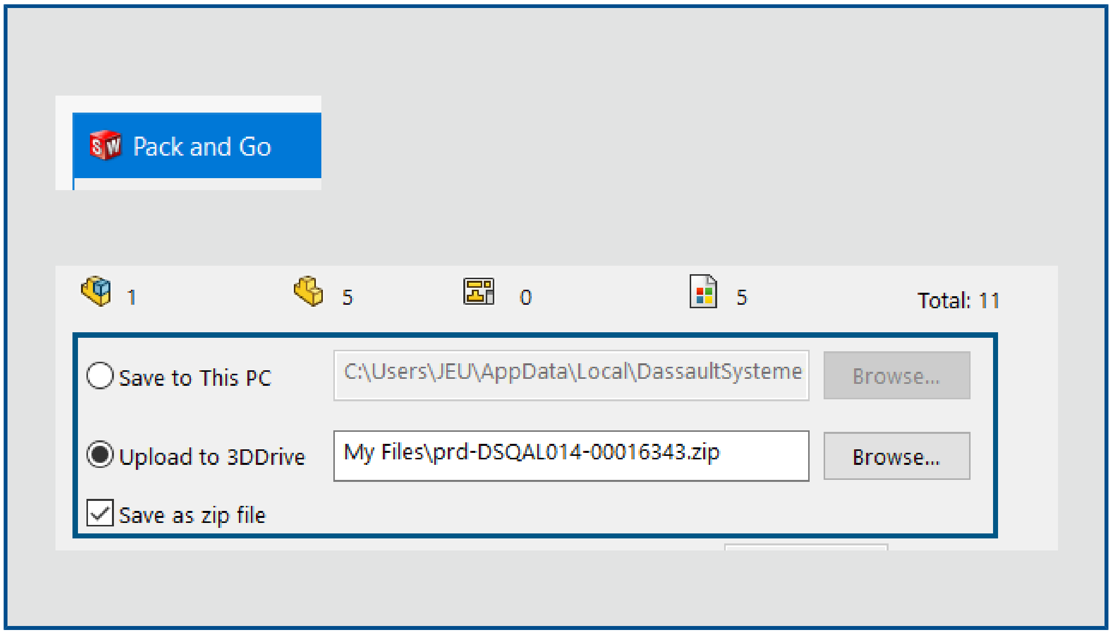
The number of **File Locations** that support bookmarks is enhanced. **3D**EXPERIENCE users can save content for practically all **File Locations** to bookmarks, with a few exceptions.

All **File Locations** support bookmarks except for the following:

- **Document Templates**
- **Referenced Documents**
- **Materials Databases**
- **Search Paths**
- **Default Save Folder**
- **Inspection Default Export Folder**

For more information, see **Adding Bookmarks for SOLIDWORKS File Locations**.

Sharing Pack and Go Files to 3DDrive (2024 FD02)



3DEXPERIENCE users can share Pack and Go files to 3DDrive from the Pack and Go dialog box or the Share dialog box.

Benefits: You can easily share Pack and Go files with others by 3DDrive.

To share files to 3DDrive from Pack and Go:

1. In SOLIDWORKS, open the files to share.
2. Click **File** > **Pack and Go**.
3. In the dialog box, click **Upload to 3DDrive** and click **Browse** to open the Select Folder dialog box.
4. Select the 3DDrive folder where you want to share the files and click **OK**.

   The Pack and Go dialog box reappears.

5. Click **Save** to upload the files to the selected 3DDrive folder.

To share Pack and Go assemblies to 3DDrive from the Share dialog box:

1. In SOLIDWORKS, open the assembly file.
2. Click **File** > **Share**.
3. In the Share dialog box, click **Share file**.
4. For **File type**, select **SOLIDWORKS Assembly (*.sldasm, *.zip)**.
5. Click **Continue** to open the Pack and Go dialog box. The **Upload to 3DDrive** option is selected by default.
6. Next to **Upload to 3DDrive,** click **Browse** to open the Select Folder dialog box.
7. Select the 3DDrive folder where you want to share the files and click **OK**.

   The Pack and Go dialog box reappears.

8. Click **Save** to upload the files to the selected 3DDrive folder.

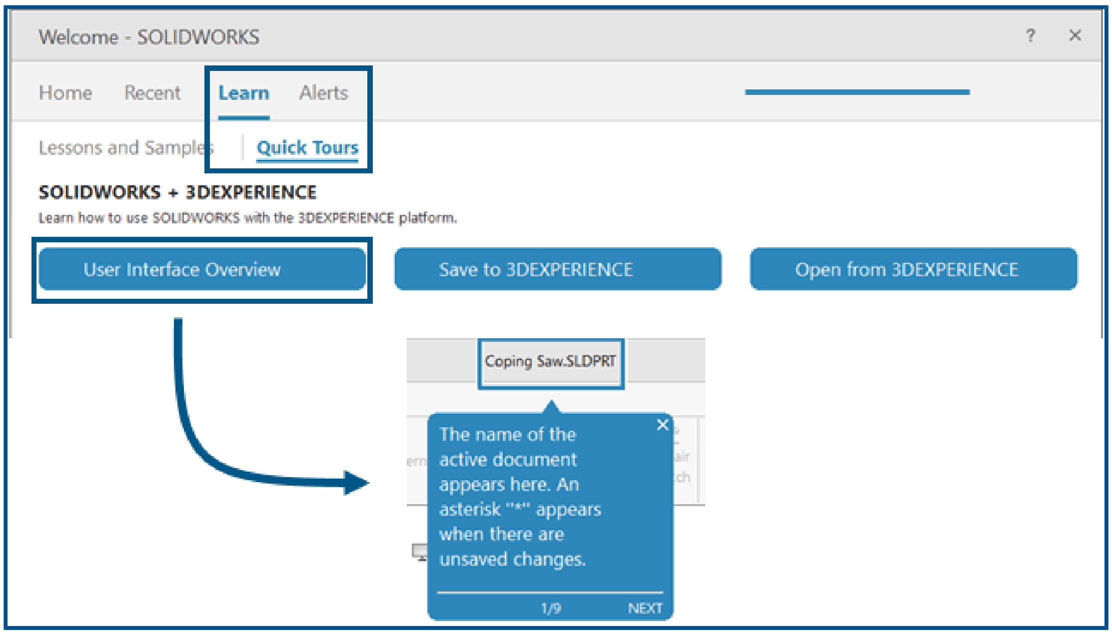
Pack and Go Dialog Box Changes

2023 Option Name	2024 Option Name
Save to folder	**Save to this PC**
Save to Zip File	**Upload to 3DDrive**
None	**Save as zip file**

The **Save as zip file** option packages the files into a zip file. The path to the zipped package appears in **Save to this PC** or **Upload to 3DDrive**, depending on your selection.

> If you run Pack and Go from File Explorer as a stand-alone tool, the **Upload to 3DDrive** option is not available.

Quick Tours (2024 FD02)



3DEXPERIENCE users can follow compact, integrated learning modules called Quick Tours. Each Quick Tour has a sequence of steps shown as interactive popups that point to elements in the user interface.

Benefits: You can interactively learn the **3D**EXPERIENCE apps to help you quickly understand basic functionality and concepts.

Available Quick Tours:

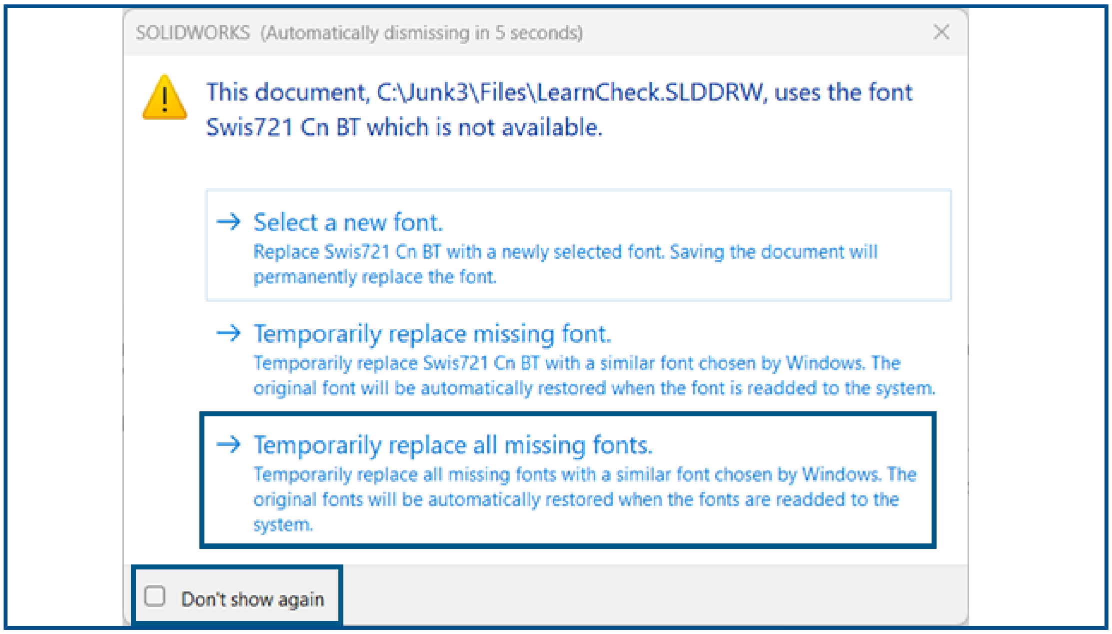
- User Interface Overview
- Save to **3D**EXPERIENCE
- Open from **3D**EXPERIENCE

To access Quick Tours, in the Welcome dialog box, on the Learn tab, click **Quick Tours**.

To start a Quick Tour, click the named button, for example **User Interface Overview**. To progress through the steps, click **Next** inside the popup step. The popups include the step numbers so you can gauge your progress.

To exit a Quick Tour, in a step, click **X**. A message confirms you are exiting the Quick Tour. You can restart the Quick Tour from the Learn tab.

Managing Missing Fonts (2024 FD02)



When you open a document that is missing fonts, you can permanently turn off all missing font warnings for that document and all other documents you open in the future that are missing fonts.

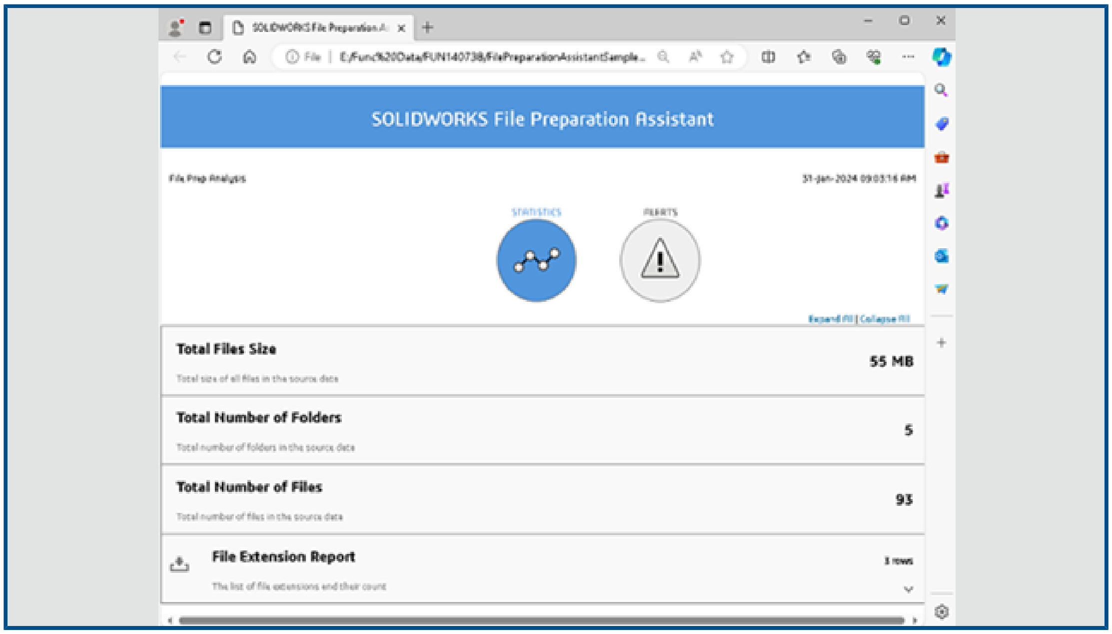
Benefits: You have fewer interruptions to your design work because fewer missing font dialog boxes appear.

In the missing fonts dialog box, first select **Don't show again** and then select **Temporarily replace all missing fonts**.

The missing fonts dialog box automatically dismisses itself after a configurable time that you specify in **Tools** > **Options** > **System Options** > **Messages/Errors/Warnings** > **Assemblies** > **Automatically dismiss reference and update messages after *n* seconds**. If the dialog box automatically dismisses itself, the document uses the **Temporary replace all missing fonts** option.

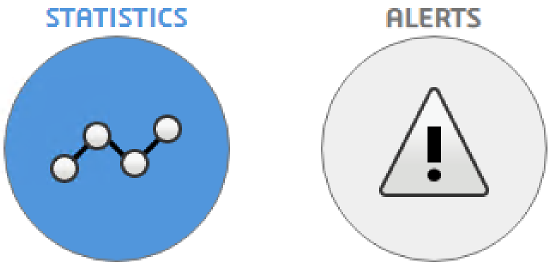
In earlier releases, in the missing fonts dialog box, you had only the first two options to select a new font or temporarily replace a missing font.

Saving File Preparation Assistant Results to HTML (2024 FD02)

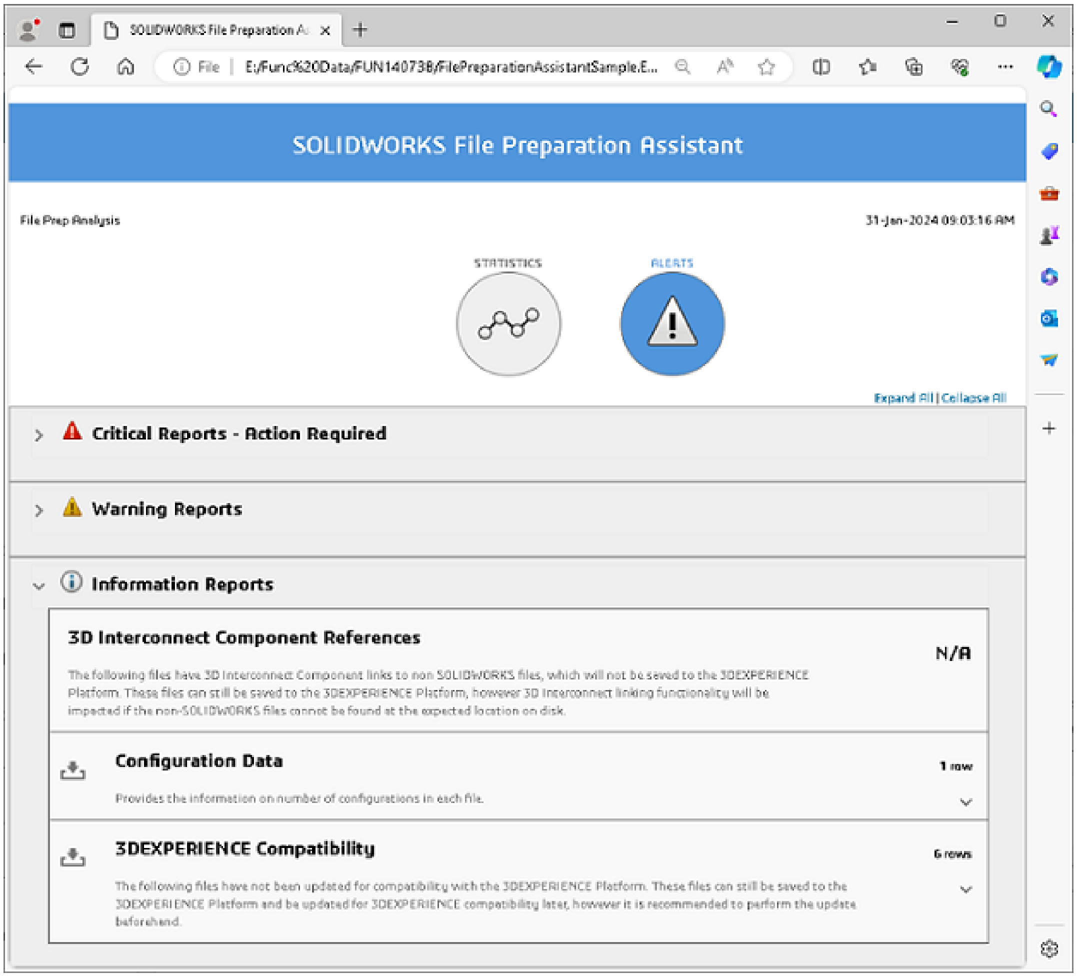


For **3D**EXPERIENCE users, the File Preparation Assistant automatically saves the results to an HTML file that is saved in the default location used for the log files. This HTML file replaces the previously output a CSV file.

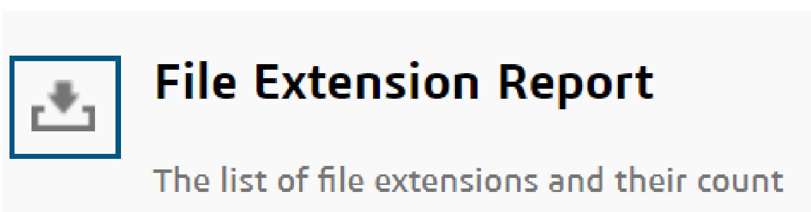
Benefits: You can study the File Preparation Assistant results in a more-user-friendly HTML file.



To display the required data, click **Statistics** as shown earlier or **Alerts** as shown below.

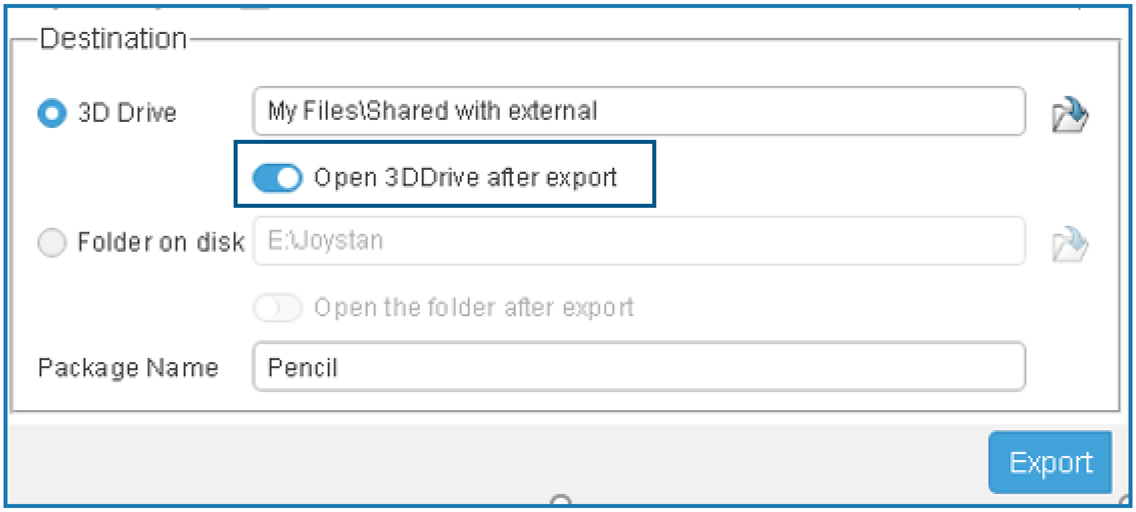


To download individual reports as CSV files from the HTML analysis, click next to the report.



You can review this HTML output to evaluate potential issues that might impact uploading the file to the **3D**EXPERIENCE platform.

Accessing 3DDrive in Export as Package (2024 FD02)

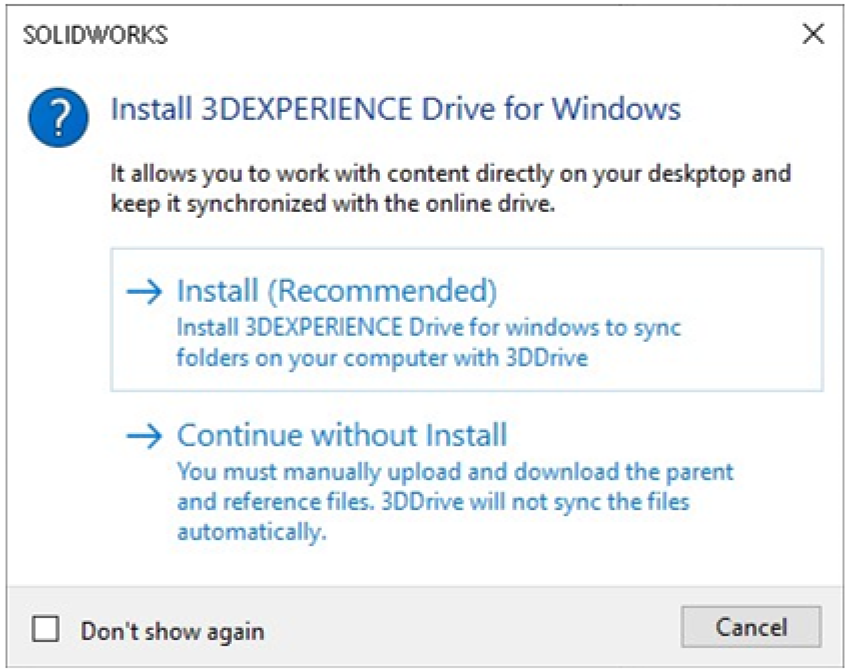


You can use the **Open 3DDrive after export** option as part of your workflow for exporting a package.

Benefits: 3DDrive opens in the task pane without explicitly opening in a web browser. This improves the experience as you do not need to switch windows.

In earlier releases, you had to upload the package to 3DDrive and then open 3DDrive manually to share the package. With the **Open 3DDrive after export** option, 3DDrive opens in the task pane and highlights the uploaded package. This helps you to quickly identify the uploaded package and perform different actions like share, preview, add to favorites, move to.

Installing Sync Client for 3DDrive (2024 FD02)



You can now choose if you want to install the **3D**EXPERIENCE Drive for Windows. In earlier releases opening 3DDrive or performing any actions in the files located in 3DDrive required mandatory installation of **3D**EXPERIENCE Drive for Windows.

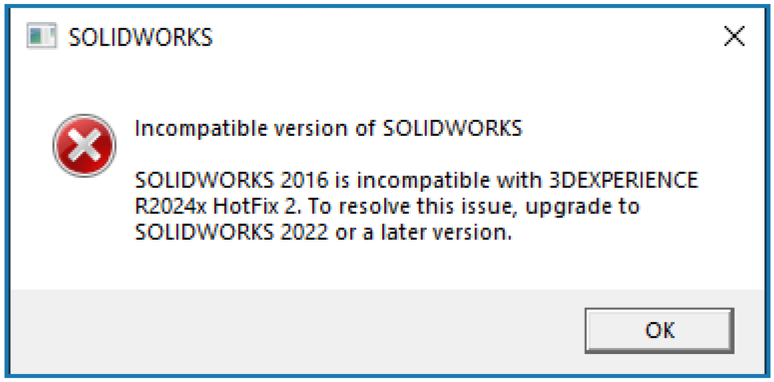
Benefits: As per the preference for installation of 3DDrive, the usability of the app changes.

While uploading or downloading files, a dialog box displays the options to install the **3D**EXPERIENCE Drive or continue without installing **3D**EXPERIENCE Drive.

If you choose **Install**, there is no change in the behavior of 3DDrive. You can work simultaneously with the files in SOLIDWORKS and keep it synced with 3DDrive.

If you choose **Continue without Install**, the files will not be synced automatically. However, you can perform all the operations of upload, download, and drag a file from 3DDrive to SOLIDWORKS. Also, when you drag multiple files from 3DDrive to SOLIDWORKS all the selected files open in SOLIDWORKS. But if you drag an assembly structure in SOLIDWORKS, only the assembly is downloaded and opened in SOLIDWORKS. The reference files are not downloaded.

Informing Users about Unsupported SOLIDWORKS Version (2024 FD02)



Error Message for incompatible version of SOLIDWORKS and 3DEXPERIENCE Platform

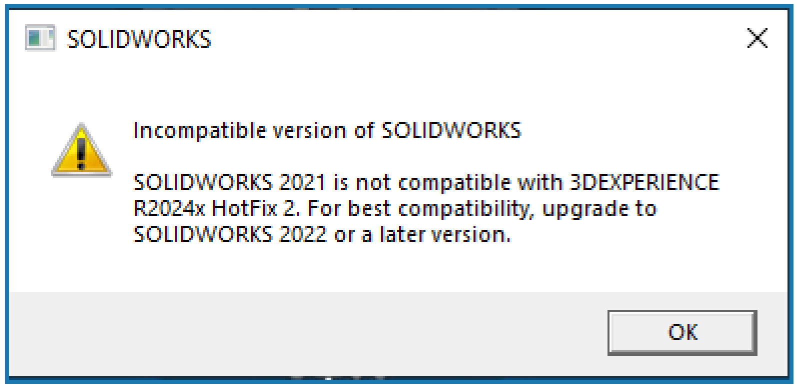
An appropriate message appears if the installed SOLIDWORKS version is not compatible with the current version of the **3D**EXPERIENCE Platform.

Benefits: You are informed to install the supported version so that you can continue working in compatible environments.

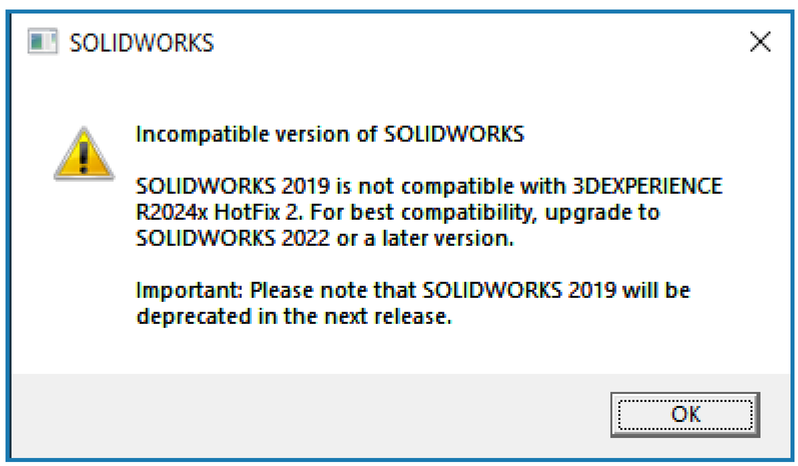
Depending on the installed SOLIDWORKS version and its compatibility with the **3D**EXPERIENCE Platform, you can either continue using SOLIDWORKS or get blocked.

For a given **3D**EXPERIENCE Platform release X, one of the following situations might occur:

- The last 3 SOLIDWORKS versions are supported: X, X-1, and X-2.
- A warning is displayed if the SOLIDWORKS version is X-3. Here, the message suggests you upgrade to a higher version that is compatible with the **3D**EXPERIENCE Platform. You can continue using SOLIDWORKS, but the version will be deprecated in the subsequent releases.
- An error message is displayed if the SOLIDWORKS version is X-4. In this case, you can proceed only when you install a higher version.

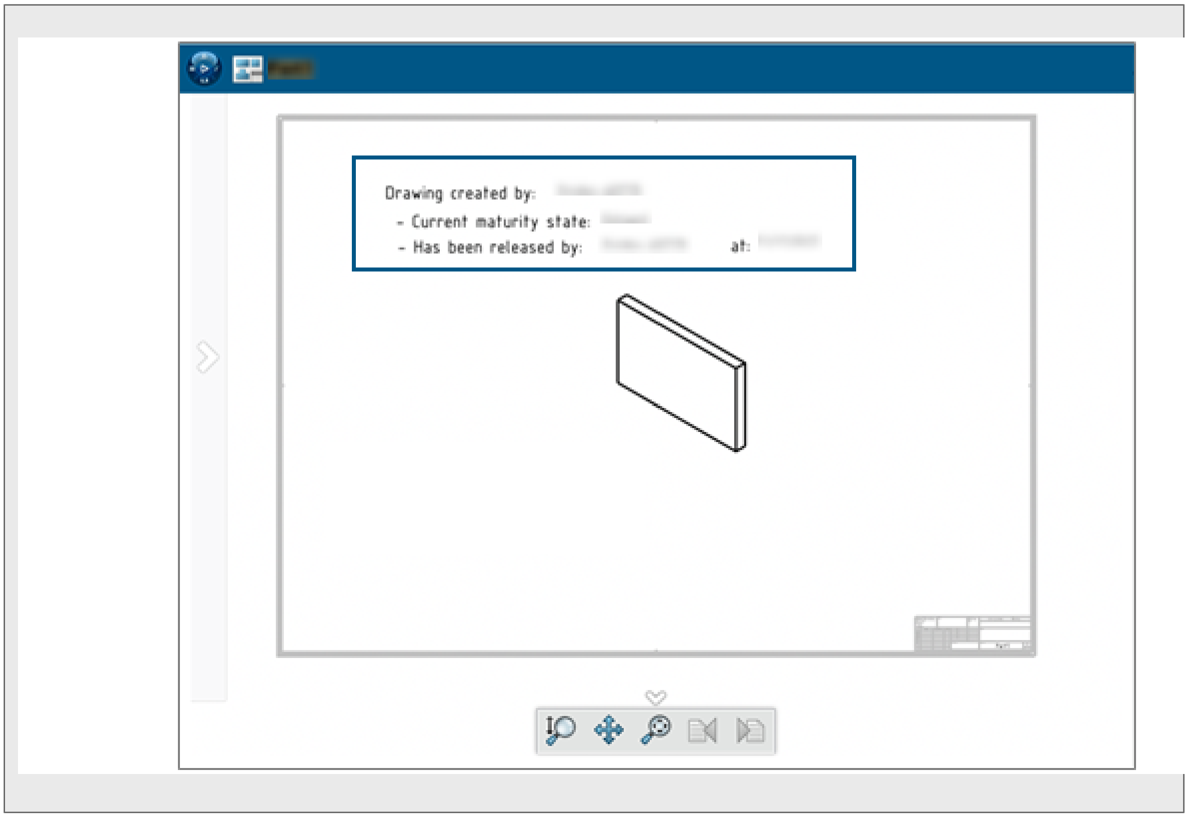


Warning message for incompatible version of SOLIDWORKS and 3DEXPERIENCE Platform



Warning message to inform about the deprecated version of SOLIDWORKS

Viewing the Drawing Annotations (2024 FD02)



You can now view the annotations for the extended attributes of a drawing in **3DPlay** or **3DMarkup**.

Benefits: You can track the lifecycle of a drawing by viewing its properties in the preview.

In earlier releases, when you changed the maturity state of a drawing to **Released** you were able to view its properties only through the **Properties** page. Now, if you view the drawing in **3DPlay** or **3DMarkup** along with the PLM properties, the extended properties are also visible.

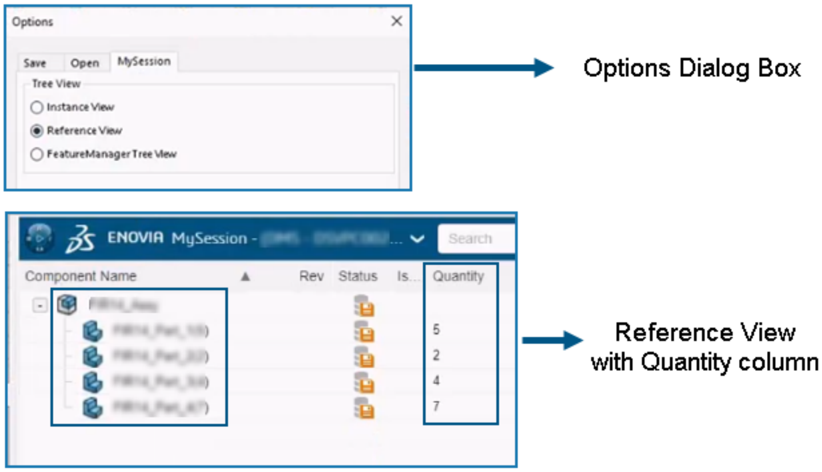
The supported extended attributes are:

- `$PLMPRP.ea_releaseddate.1`
- `$PLMPRP.ea_releasedby.1`
- `$PLMPRP.ea_createdby`

.

The annotations for the extended attributes are visible only if the drawing is released using the Change Maturity command in the Collaborative Lifecycle app.

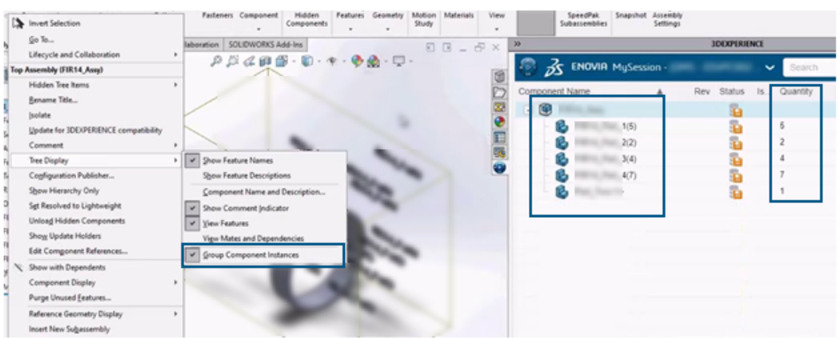
Selecting the Tree View for Objects in MySession (2024 FD02)



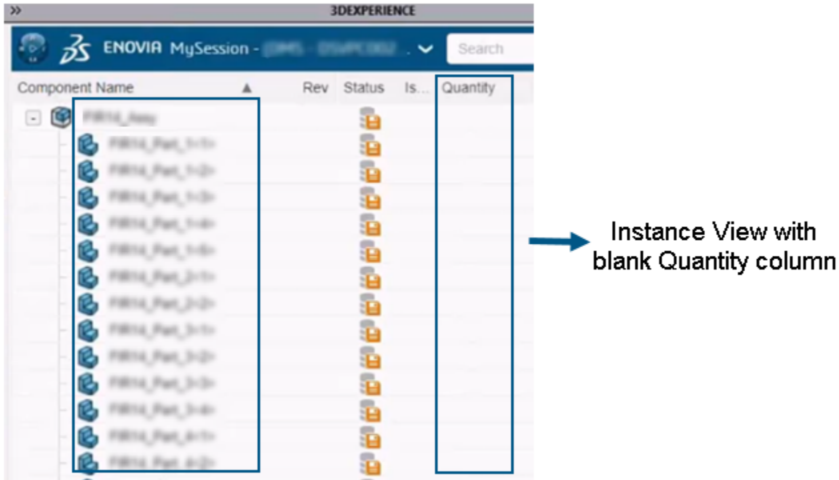
You can choose the way the objects and their associated instances appear in **MySession**.

Benefits: You can view the unique references and the number of the references used in a particular product structure. These enhancements help you to review and evaluate the product design and quickly analyse the Bill of Material.

In the **Options** dialog box, a new tab **MySession** is added. In this tab, you can choose a type of tree view that appears in **MySession**.



FeatureManager Tree View with Quantity column



The types of tree view are: **Instance View**, **Reference View**, and **FeatureManager Tree View**. Based on the view selected, the objects and their associated instances appear in **MySession**. Also a **Quantity** column is added in **MySession** that displays the number of associated instances.

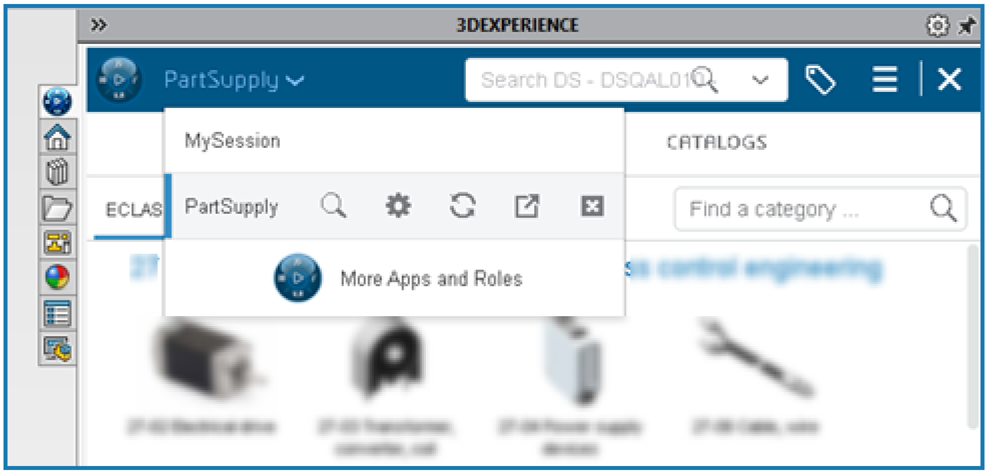
On-Premise: Using the Derived Format Converter for Generating Output (2024 FD02)

You can now generate output for SOLIDWORKS files asynchronously only by using the **Derived Format Converter**.

Benefits: This improves the quality of the output and also the efficiency of the save process.

Earlier the CGR and UDL output formats were not supported for save process through **Batch Save to 3DEXPERIENCE** command or asynchronous save. To overcome this situation, install the **Derived Format Converter**.

Viewing PartSupply Components SOLIDWORKS (2024 FD02)



The **PartSupply** app now opens in the SOLIDWORKS task pane.

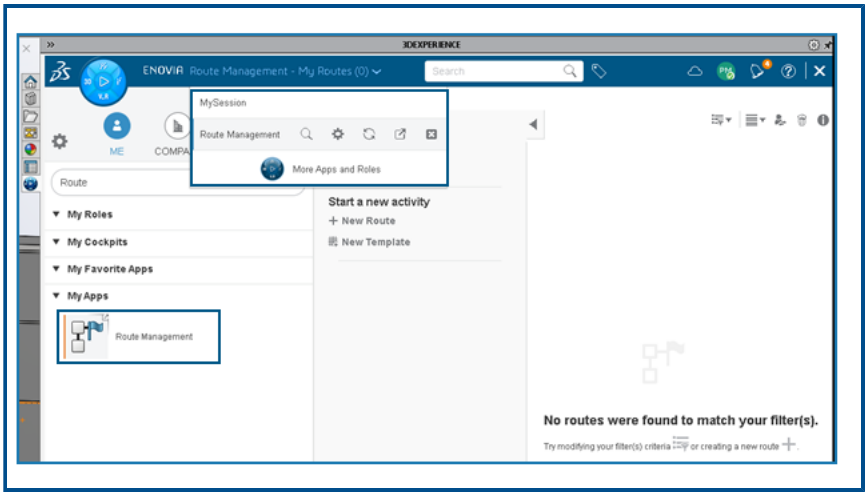
Benefits: This improves the user experience of accessing the app and saves the reloading time.

When you open **PartSupply** in any of the following ways, it opens in the SOLIDWORKS task pane.

- **Design Library**
- **Insert Components**
- **Compass** > **As a Business Model**
- **Compass** > **Part Supply Optimised Components**

Also **PartSupply** is added to the list of apps and you can switch between different apps easily from the top bar by clicking .

Opening Route Management in SOLIDWORKS (2024 FD02)

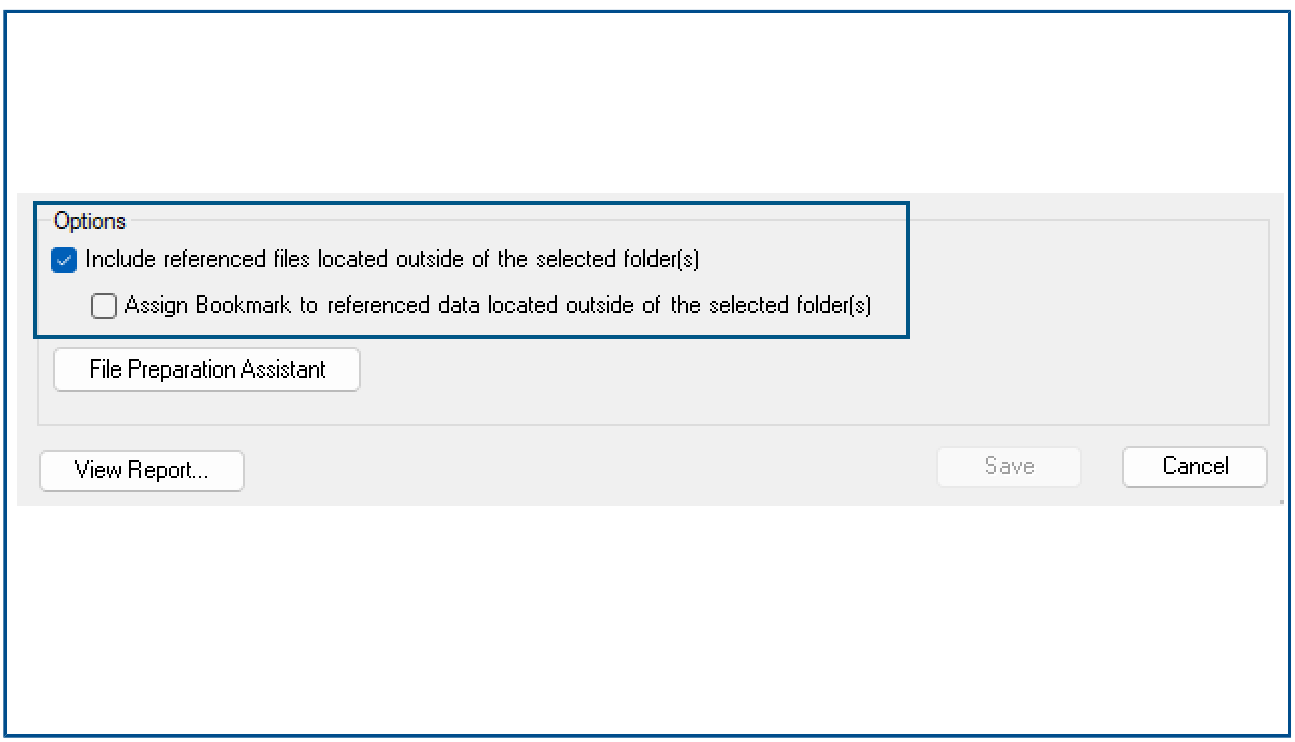


You can now open the **Route Management** app in the SOLIDWORKS task pane.

Benefits: This enhances the experience of using the different **3D**EXPERIENCE platform apps without opening them in a web browser and thus saves the reloading time.

Route Management helps to create, access, and manage routes and route templates. The app is added to the list of apps and you can switch between different apps easily from the top bar by clicking . You can also open the notifications received from this app within the SOLIDWORKS task pane.

Managing Bookmark Reference in Batch Save (2024 FD02)



An option **Assign Bookmark to referenced data located outside selected folder** is added to the **Batch Save to 3DEXPERIENCE** dialog box.

Benefits: You get the flexibility to attach the referenced files to the bookmarks.

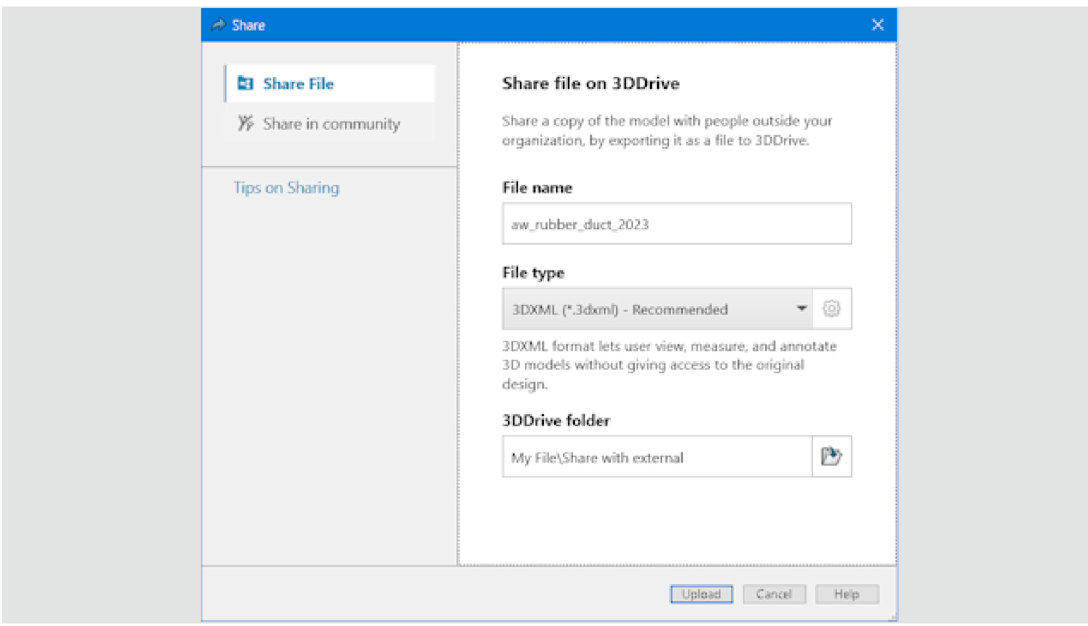
While saving with **Batch Save to 3DEXPERIENCE**, if in a folder there are files with references in other folder, and the **Include referenced files located outside the selected folder** and **Assign Bookmark to referenced data located outside selected folder** options are selected, the references get added to the selected bookmark.

SP1-FD01

Sharing Files (2024 FD01)

The various methods of sharing files are unified into a single **Share** tool on the Lifecycle and Collaboration toolbar.

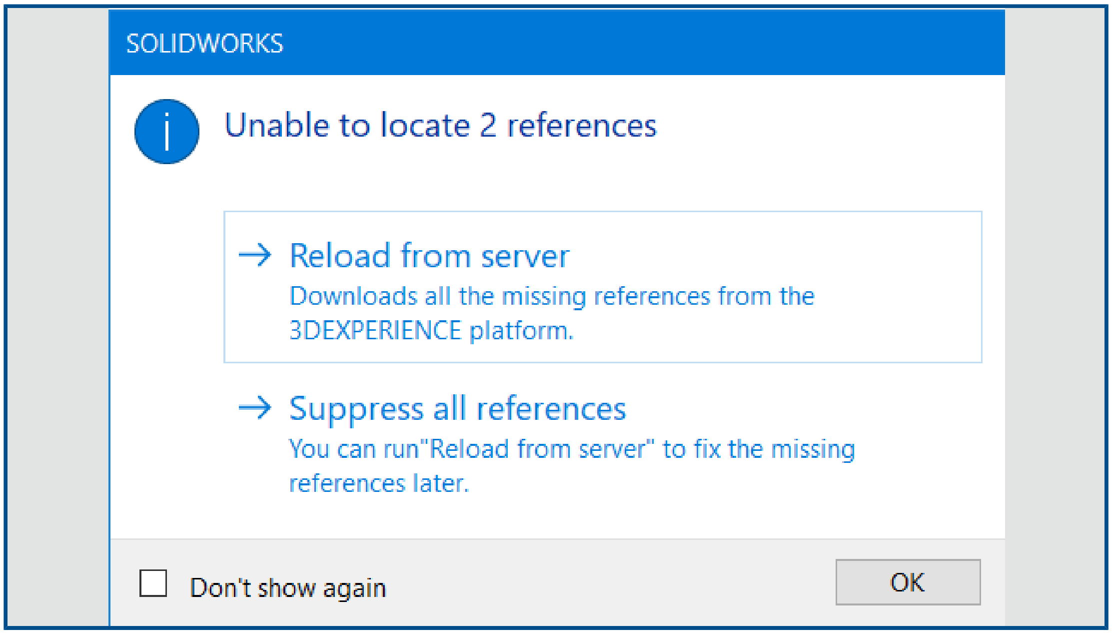
Benefits: You have a consistent method that simplifies and accelerates sharing files.



To access this tool, you can also click **File** > **Share**. The **Share** tool lets you share files using one dialog box. You can:

- Share by 3DDrive
- Share by 3DSwym communities and conversations

Automatically Fix Missing References (2024 FD01)



If you open a **3D**EXPERIENCE file from your computer and some of the references are missing on your machine, you can use the Unable to Locate References dialog box to fix the missing references.

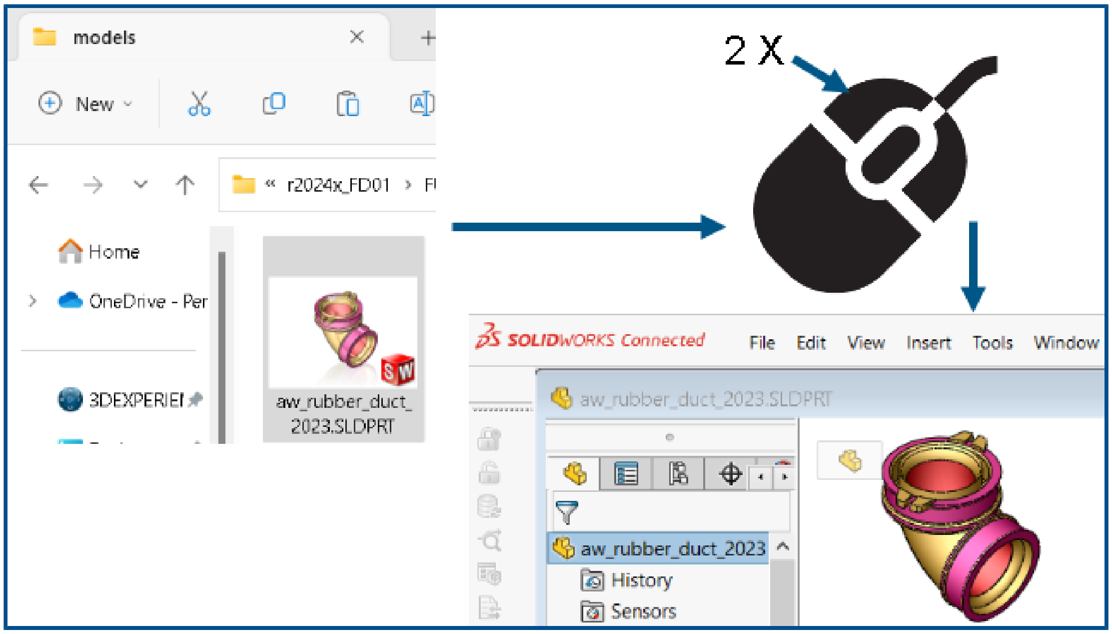
In the dialog box, you can select **Reload from server** to download all the missing references from the platform or **Suppress all references** to fix the missing references later.

Benefits: You can more easily fix broken references to files. In previous releases, you had to individually find and download all missing references from the **3D**EXPERIENCE platform.

Missing references typically happen if the file is already saved to your local cache and some of the references were deleted from the local cache.

If you are not connected to the **3D**EXPERIENCE platform, the existing dialog box appears and is unchanged. You can select **Browse for file**, **Suppress this component**, or **Suppress all missing components**.

Double-Clicking SOLIDWORKS Files to Open SOLIDWORKS Connected (2024 FD01)

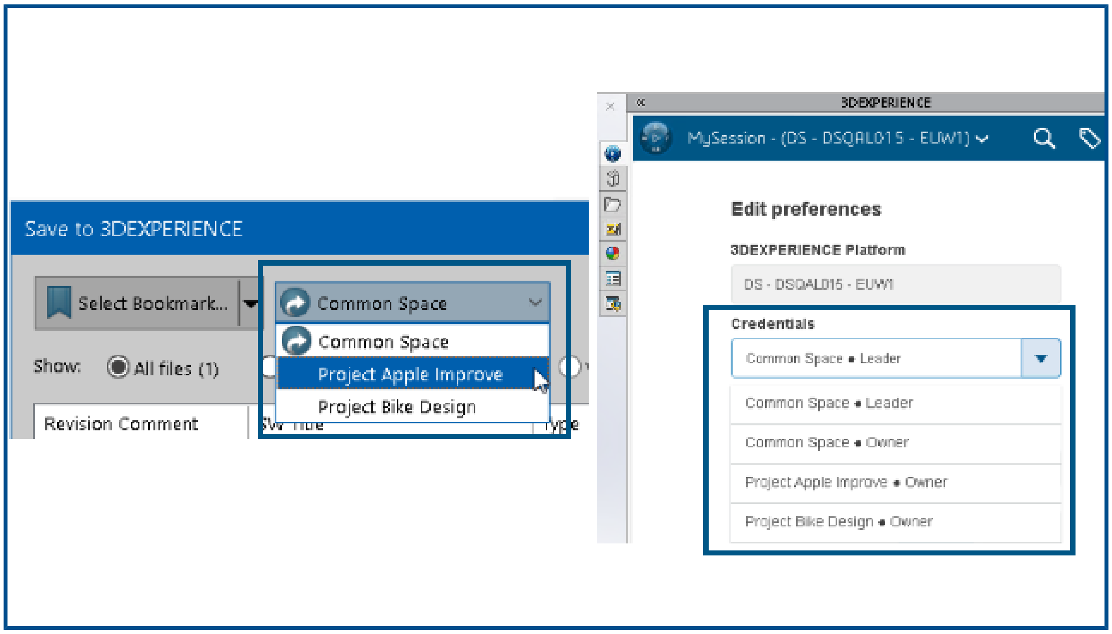


From File Explorer, you can double-click or right-click **> Open** a SOLIDWORKS file to start SOLIDWORKS Connected and open the file. In previous releases, you could open SOLIDWORKS Connected only from the Compass in a browser or from a desktop shortcut.

Benefits: You can more quickly and conveniently open the SOLIDWORKS Connected app to view files.

- If you are required to log in, SOLIDWORKS Connected prompts you for your username and password when you double-click a file.
- If you have installed both SOLIDWORKS Connected and SOLIDWORKS, the software prompts you to choose the app to open.
- If SOLIDWORKS Connected cannot find the last-used tenant, the software prompts you to open the app from the Compass or a desktop shortcut.

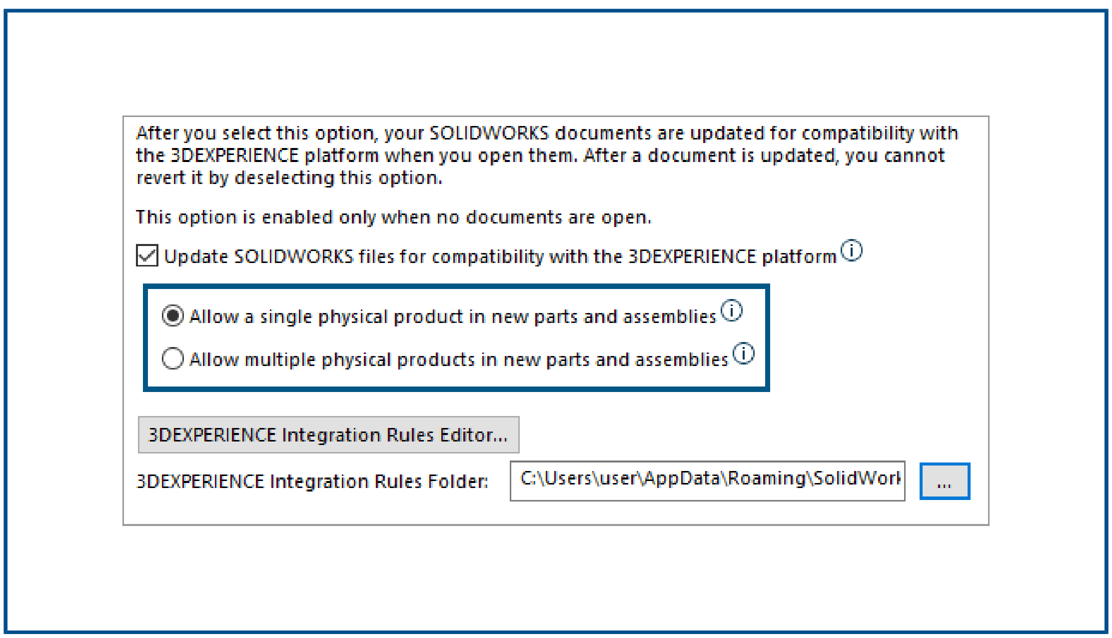
Collaborative Space Selection Menu (2024 FD01)



The collaborative space selection menu now appears in only two locations: The Save to 3DEXPERIENCE dialog box and in **MySession** > **Edit preferences**. The menu is removed from all other locations where it was previously located.

Benefits: The collaborative space selection workflow is clearer and more understandable.

Specifying a New Part or Assembly as a Single Physical Product (2024 FD01)



You can designate a new part or assembly as a single physical product.

When you select **Update SOLIDWORKS files for compatiblity with the 3DEXPERIENCE platform**, these options are available:

Allow a single physical product in new parts and assemblies	Uses representations to show different configurations of a model. Select this option if you do not use unique part numbers for your configurations.
Allow multiple physical products in new parts and assemblies	Uses physical products to show different configurations of a model. Select this option if you use unique part numbers for your configurations.

To specify a new part or assembly as a single physical object:

1. Click **Tools** > **Options** > **System Options** > **3DEXPERIENCE Integration**.
2. Select **Update SOLIDWORKS files for compatibility with the 3DEXPERIENCE platform**.
3. Select an option:
   - **Allow a single physical product in new parts and assemblies**
   - **Allow multiple physical products in new parts and assemblies**
4. Create a new part.
5. Save the part to the **3D**EXPERIENCE platform.

Selecting Recently-Accessed Bookmarks (2024 FD01)

You can select from recently-accessed bookmarks in the Save to **3D**EXPERIENCE dialog box.

Benefits: You can quickly select the bookmarks that you have used recently as part of the Save workflow.

In the **Save to 3D**EXPERIENCE dialog box, the **Select from Recent** option in the **Select Bookmark** list lists the 10 most recently accessed bookmarks. Each time a bookmark is chosen from the **Select Bookmark** dialog box, the recent list is updated.

Managing Deleted Configurations (2024 FD01)

If a structure has physical products that are deleted locally, the save process is blocked and an appropriate warning is displayed in the **Status** column of the **Save** dialog box.

Benefits: You can troubleshoot more easily when the save process fails.

If you continue to save a structure that contains deleted physical products, the Relations app opens, allowing you to change the reference relationships and remove the dependencies.

Editing the Properties of an Object (2024 FD01)

You can edit the properties of an object from the **Action Bar** > **View** > **Display Side Panel** > **Properties**. In the **Properties** tab of **Display Side Panel**, click **Edit** to edit the attributes of the object.

Benefits: In earlier releases the properties of an object from **Display Side Panel** were not editable.

Once the attributes are edited, the changes that impact SOLIDWORKS files are propgated to the **Properties** dialog box.

Selecting an Appropriate Collaborative Space (2024 FD01)

If multiple organizations belong to a common collaborative space, the collaborative space list in the **Save** dialog box and the **Destination** column in the **Batch Save to 3D**EXPERIENCE dialog box display the name of the collaborative space and the name of the organization.

Benefits: You can easily select a collaborative space that has write access before the save operation starts.

The save operation is blocked if you have read access to the selected collaborative space. An error message in the **Status** column indicates whether you have write access to the selected collaborative space.

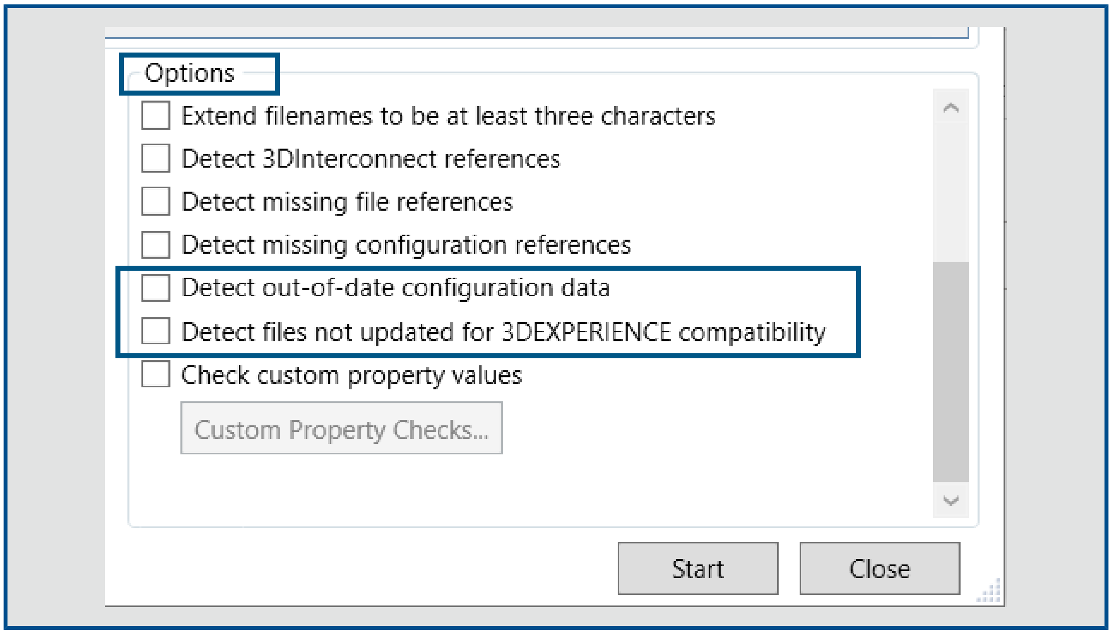
Connecting to the 3DEXPERIENCE Platform from SOLIDWORKS (2024 FD01)

A **Welcome** dialog box appears when you connect to the **3D**EXPERIENCE platform for the first time. Also, a notification is displayed when a connection is established with the **3D**EXPERIENCE platform.

Benefits: The intuitive messages inform you if the connection to the **3D**EXPERIENCE platform is successful or not.

The **Welcome** dialog box provides a way to open documents, view folders, and access SOLIDWORKS resources. You can view the user name and the profile picture of the logged user in the upper-right corner of the **Welcome** dialog box and SOLIDWORKS window.

File Preparation Assistant - Additional Checks (2024 FD01)



The File Preparation Assistant dialog box contains two additional options to check for out-of-date configuration data and incompatible files. The software also silently performs two other checks for file names and the number of configurations.

Benefits: More checks improve the success of saving your files to the **3D**EXPERIENCE platform.

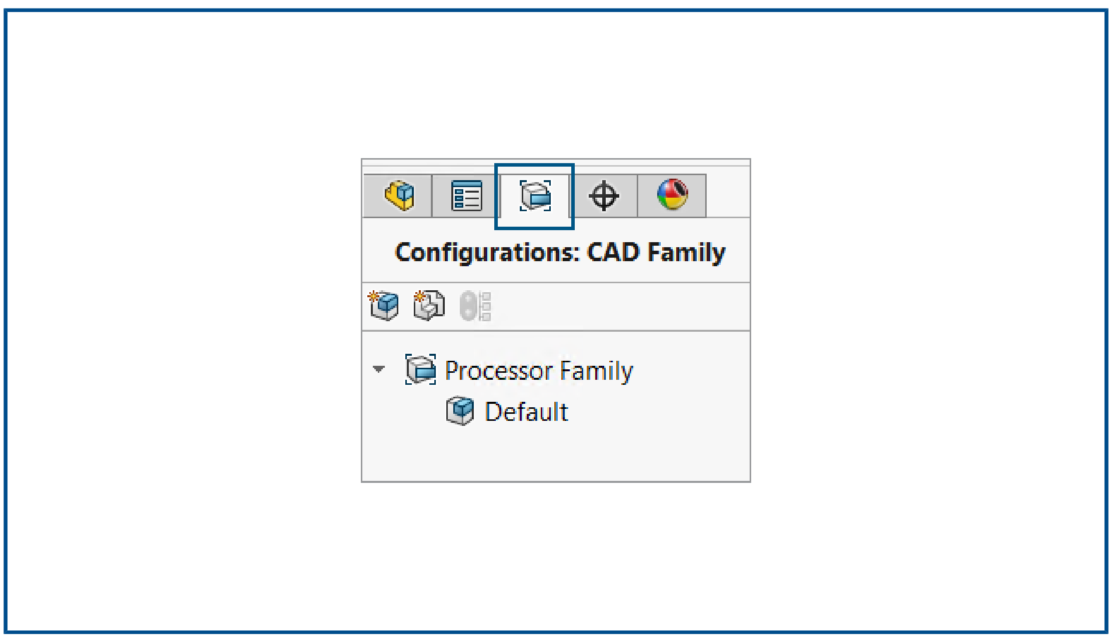
Additional Check	Description
Detect out-of-date configuration data	Lists information about outdated configurations. This could happen if you delete a configuration and do not rebuild the model. Rebuild the documents before saving them to the **3D**EXPERIENCE platform.
Detect files not updated for 3DEXPERIENCE compatibility	Runs the compatibility check on the selected files, which verifies if the files have been updated to the new **3D**EXPERIENCE Configuration Manager. To automatically update files for **3D**EXPERIENCE compatibility, click **Tools** > **Options** > **System Options** > **3DEXPERIENCE Integration** and select **Update SOLIDWORKS files for compatibility with the 3DEXPERIENCE platform**. For more information, see *SOLIDWORKS Help: 3DEXPERIENCE Integration Options*.

Additional Check	Description
	To manually update files for **3D**EXPERIENCE compatibility, with a model open in the FeatureManager design tree, right-click the top item and select **Update for 3DEXPERIENCE Compatibility**. For more information, see *SOLIDWORKS Help: Updating Models for 3DEXPERIENCE Compatibility*.

The File Preparation Assistant automatically performs two additional silent checks.

Additional Silent Check	Description
Updates file extension	Updates files that have old file format extensions (`.prt`, `.asm`, `.drw`) to the current file extensions (`.SLDPRT`, `.SLDASM`, `.SLDDRW`).
Number of configurations	Counts the number of configurations and displays that information in the log file.

CAD Family Tab (2024 FD01)



Models updated to the **3D**EXPERIENCE platform can use only the CAD Family tab for configuration views.

Previously, updated models appeared in the CAD Family tab and the ConfigurationManager tab when you selected **Both CAD Family and Configurations**.

In **Tools** > **Options** > **System Options** > **FeatureManager**, the **Only CAD Family View** and the **Both CAD Family and Configurations** options have been removed.

Updating the Server Information in the 3DEXPERIENCE Files on This PC Tab (2024 FD01)

The current server information for the files on the **3D**EXPERIENCE Files on This PC tab might become outdated. To overcome this, the **Refresh** command is replaced with two options: **Refresh View** and **Refresh from Server** .

Benefits: You can synchronize cache files with the **3D**EXPERIENCE platform. While the refresh operation is in progress, you can continue using SOLIDWORKS.

Refresh from Server is also available on the shortcut menu.

While the refresh operation is continuing, a progress message informs you about the estimated time for the operation and the number of files in the queue to be refreshed.

When the operation finishes, a notification message gives details about the number of files refreshed from the **3D**EXPERIENCE platform.

The **3D**EXPERIENCE Files on This PC tab includes the **Last Refreshed** column, which displays the time when the files were last synchronized with the **3D**EXPERIENCE plaform.

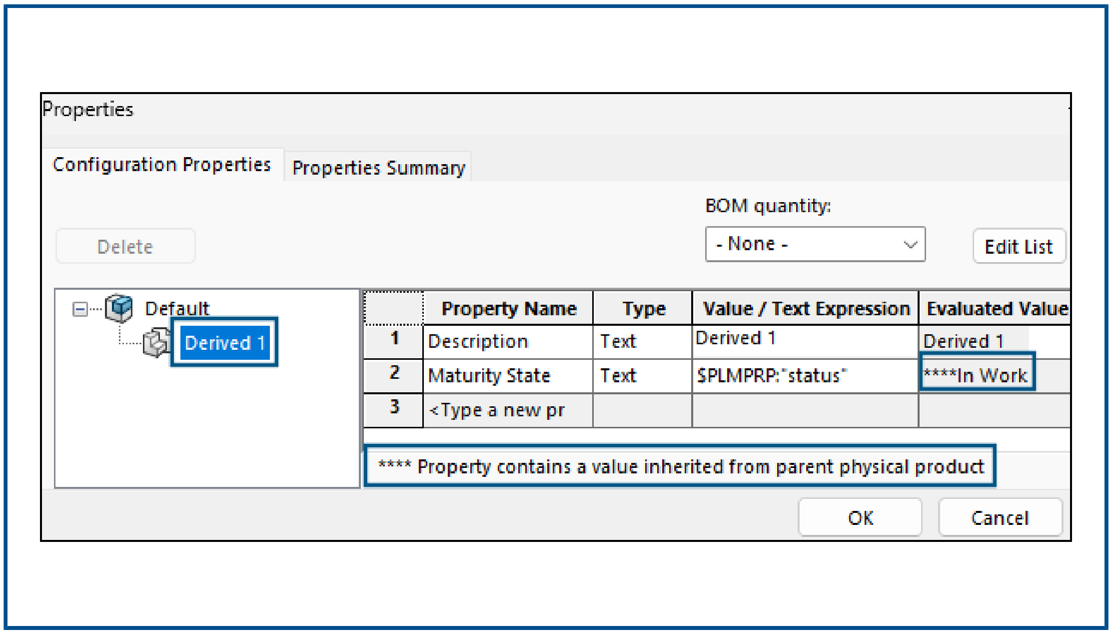
Selecting the Position of Work Under (2024 FD01)

When **MySession** is loading, you can hide or display the **Work Under** and also select its position.

Benefits: You can control the visibility and position of the **Work Under**, so that it reduces the probability of wrong operations.

On the **Preference** page, you can select the **Display Work Under** option to decide its visibility. Using the **Work Under Postion** option, you can choose the position where the **Work Under** is displayed.

Linking PLM Custom Properties of Representations to Physical Products (2024 SP1)



The software links the PLM attributes of custom properties of representations to the parent physical products.

The software adds ******** as a prefix to **Evaluated Value** and displays a footnote if the:

- Configuration is a representation
- Custom property has at least one PLM attribute that it inherits from the parent physical product

Previously, for a PLM property, the software did not display a value for a representation of a parent physical product.

Click **Tools** > **Options** > **3DEXPERIENCE Integration** and select **Update SOLIDWORKS files for compatibility with the 3DEXPERIENCE platform**.

In the Properties dialog box, when you select a representation, the evaluated value appears for the PLM property that you select.

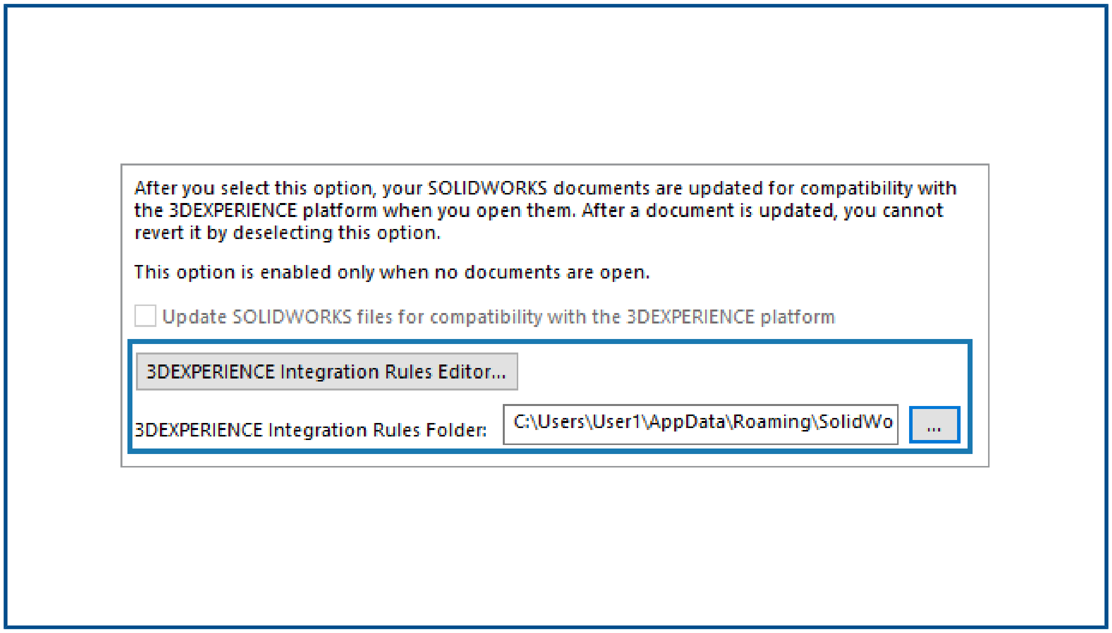
Support for the 3DEXPERIENCE (Design with SOLIDWORKS) Add-In in Routing (2024 SP1)

With the **3D**EXPERIENCE (Design with SOLIDWORKS) add-in, you can use routing components or assemblies from the **3D**EXPERIENCE platform.

For more information, see **Using the 3DEXPERIENCE Add-In with Routing (2024 SP1)** on page 275.

SP0_GA

Defining Rules for Updating Models to the 3DEXPERIENCE Platform



You can use the 3DEXPERIENCE Integration Rules Editor to specify if a configuration is mapped as a physical product or a representation when you update a model to the **3D**EXPERIENCE platform.

When creating a sub-type rule, you specify document level criteria like filename, custom properties, and weldments and sheet metal file types. You can use these rules to group parts and assemblies.

For each sub-type rule, you define a configuration mapping rule to specify if the configuration is a physical product or a representation.

To save a part configuration that is referenced by an assembly as a physical product, you must create a sub-type rule. Previously, the part configuration was always saved as a physical product.

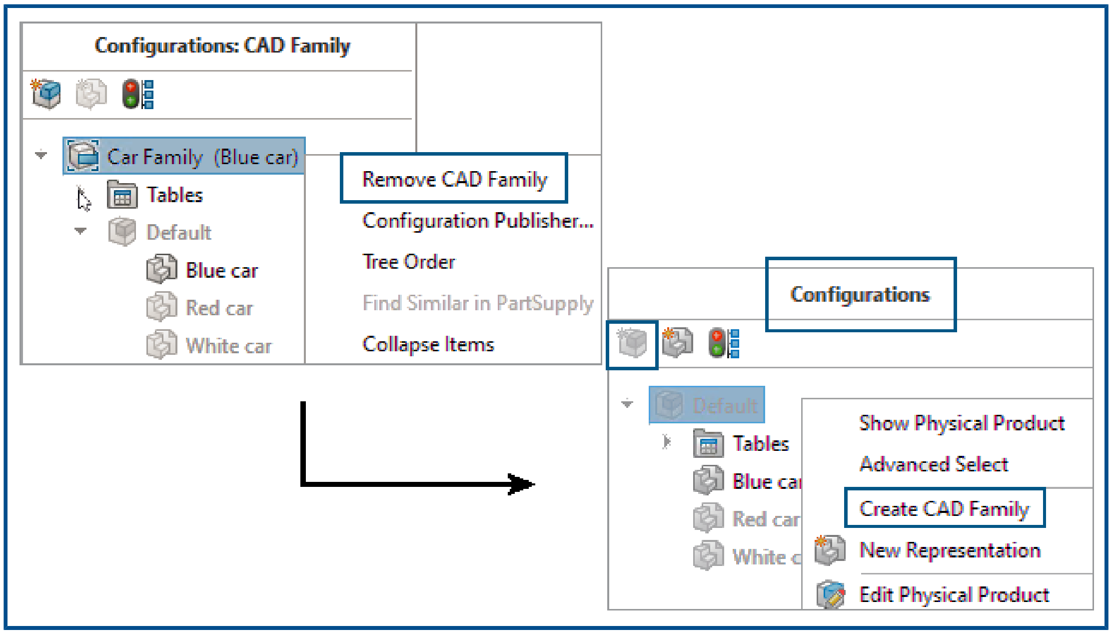
You can save the rules in the 3DEXPERIENCE Integration Rules Folder.

New configurations are not created when you update a model.

To open the 3DEXPERIENCE Integration Rules Editor:

1. Open a model and click **Tools** > **Options** > **3DEXPERIENCE Integration**.
2. Click **3DEXPERIENCE Intergration Rules Editor**.

Creating a Single Physical Product



In the Design with SOLIDWORKS app, you can use **Remove CAD Family** to designate a part or an assembly as a single physical product.

When you remove the CAD Family, the following changes occur:

- The part or assembly becomes a physical product.
- If the physical product is the active configuration, SOLIDWORKS uses the physical product as the single physical product. If the representation is the active configuration, SOLIDWORKS uses the parent physical product of the representation as the single physical product.
- Other configurations change to representations of the single physical product.
- **Inserts new physical product** is disabled.
- The ConfigurationManager title changes from Configurations: <CAD Family> to Configurations.

When you have a single physical product, you can change the configuration used for the physical product. Right-click a representation and click **Convert to Physical Product**.

You can add a CAD Family object to a single physical product. Right-click the physical product and click **Create CAD Family**.

You cannot use **Convert to Physical Product** on the following configurations:

- Speedpak configurations
- Exploded views
- Model Break views
- Defeature configurations
- Child configurations that required a parent configuration

To create a single physical product:

1. Open a model that has multiple physical products.

2. Right-click the CAD Family and click **Remove CAD Family**.

3

Installation

This chapter includes the following topics:

Installation Access Starting with SP0 for SOLIDWORKS Student and Education Editions

Users with Student and Education licenses can install SOLIDWORKS version 2024 starting with SP0. Previously, these users could not access SOLIDWORKS until SP2.

Render Installation Manager with Microsoft Edge WebView 2

The SOLIDWORKS Installation Manager uses Microsoft Edge WebView2 to render the Installation Manager pages. WebView2 installs if not found on your machine.

Previously, the Installation Manager pages were rendered with Microsoft Internet Explorer.

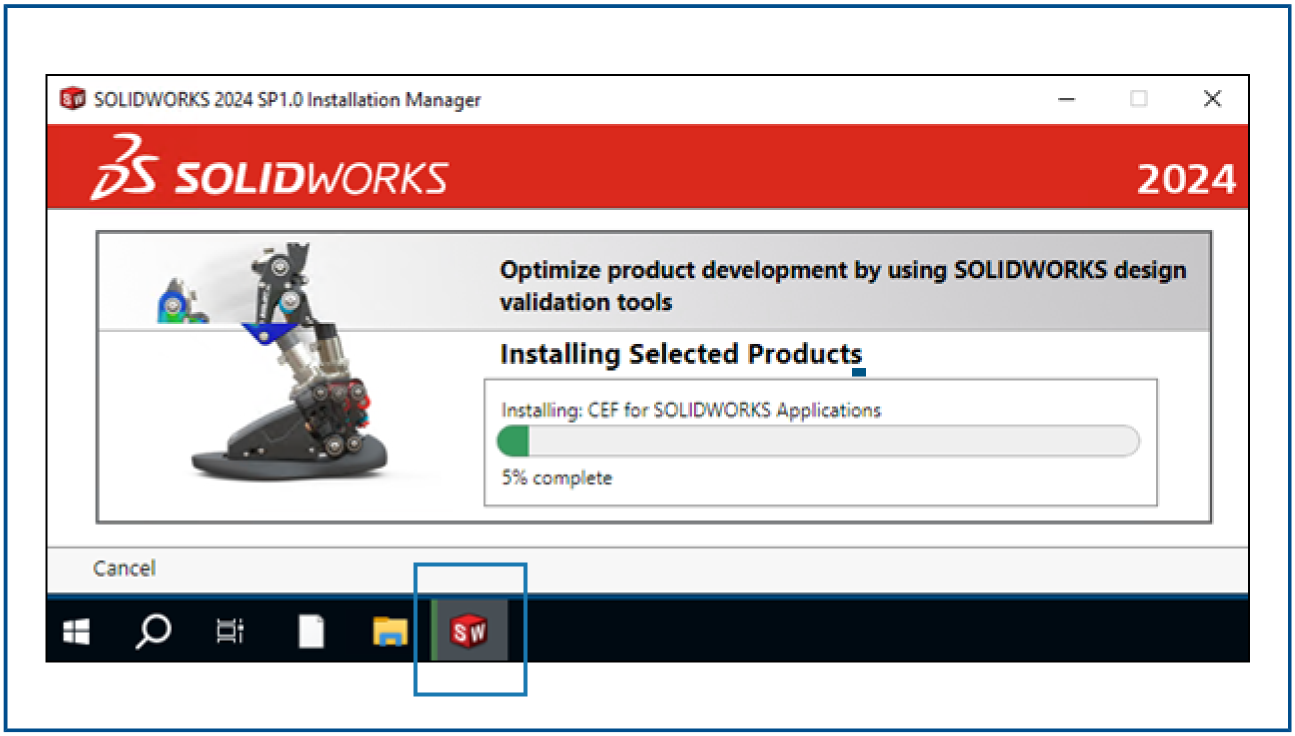
Inactivity Timeout for SOLIDWORKS Simulation, SOLIDWORKS Flow Simulation, and SOLIDWORKS Plastics

When you run SOLIDWORKS Simulation, Plastics, or Flow Simulation studies, the network licenses remain active and do not time out. SOLIDWORKS holds the licenses during the calculation process, which is considered an activity.

Inactivity periods, defined by a TIMEOUT option, only take effect after the studies finish calculating.

Previously, licenses could time out while studies were still running. In situations with limited licenses, another user in the network could take your licenses, leaving you without licenses to resume an analysis after completing a study.

Show Install Progress in Windows Taskbar



When you open the SOLIDWORKS Installation Manager (SLDIM) and select installation options, the progress bar shown in the SLDIM reflects in the Windows taskbar.

These operations include:

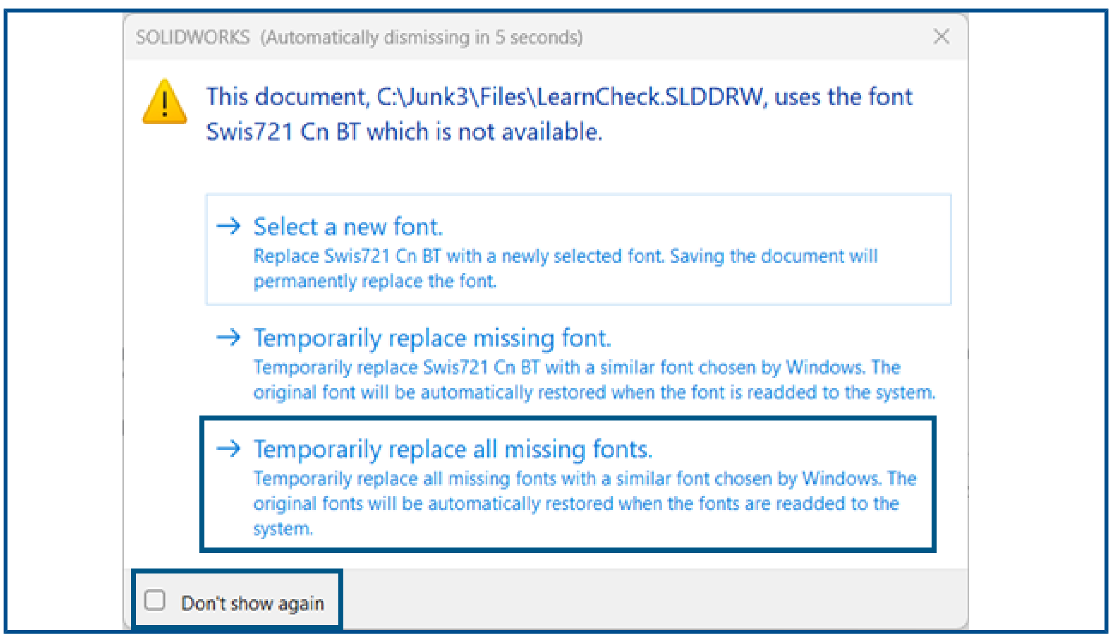
- Download Progress
- Install Progress
- Modify Progress
- Repair Progress
- Uninstall Progress
- Create Admin Image Progress
- Installations from Admin Image where the progress bar displays

4

SOLIDWORKS Fundamentals

This chapter includes the following topics:

Managing Missing Fonts (2024 FD02)



When you open a document that is missing fonts, you can permanently turn off all missing font warnings for that document and all other documents you open in the future that are missing fonts.

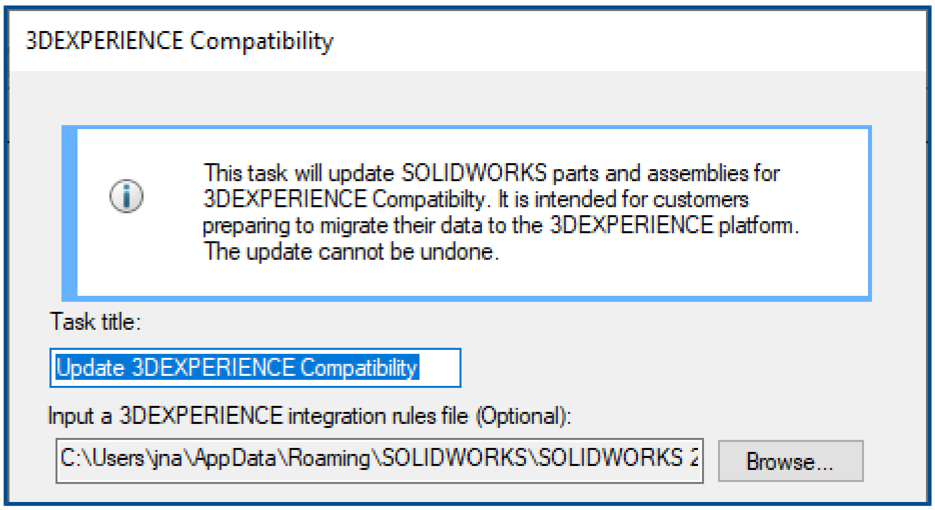
Benefits: You have fewer interruptions to your design work because fewer missing font dialog boxes appear.

In the missing fonts dialog box, first select **Don't show again** and then select **Temporarily replace all missing fonts**.

The missing fonts dialog box automatically dismisses itself after a configurable time that you specify in **Tools** > **Options** > **System Options** > **Messages/Errors/Warnings** > **Assemblies** > **Automatically dismiss reference and update messages after *n* seconds**. If the dialog box automatically dismisses itself, the document uses the **Temporary replace all missing fonts** option.

In earlier releases, in the missing fonts dialog box, you had only the first two options to select a new font or temporarily replace a missing font.

3DEXPERIENCE Compatibility Updates in the SOLIDWORKS Task Scheduler (2024 SP1)



You can schedule a task to update SOLIDWORKS parts and assemblies for **3D**EXPERIENCE compatibility. The update modifies custom properties and configuration behavior to align with the **3D**EXPERIENCE requirements.

You can also apply **3D**EXPERIENCE integration rules to the task. The rules map parts and assemblies to physical products and representations in the platform. For details about using **3D**EXPERIENCE integration rules, see *SOLIDWORKS Help: 3DEXPERIENCE Integration Options*.

> This task is exclusively intended for customers who are preparing to save their models to the **3D**EXPERIENCE platform. Once the update is applied, you cannot revert the changes.

To create a 3DEXPERIENCE compatibility update task in the SOLIDWORKS Task Scheduler:

1. In SOLIDWORKS, go to **Tools** > **SOLIDWORKS Applications** > **SOLIDWORKS Task Scheduler**.
2. Click **3DEXPERIENCE Compatibility** on the sidebar.
3. Specify the following:
   - Title
   - **Optional 3D**EXPERIENCE integration rules file
4. Add the files or folders that you want to update.

5. Schedule the task, specify the backup location and advanced options.
6. Click **Finish**.

Changes to System Options and Document Properties

The following options have been added, changed, or removed in the software.

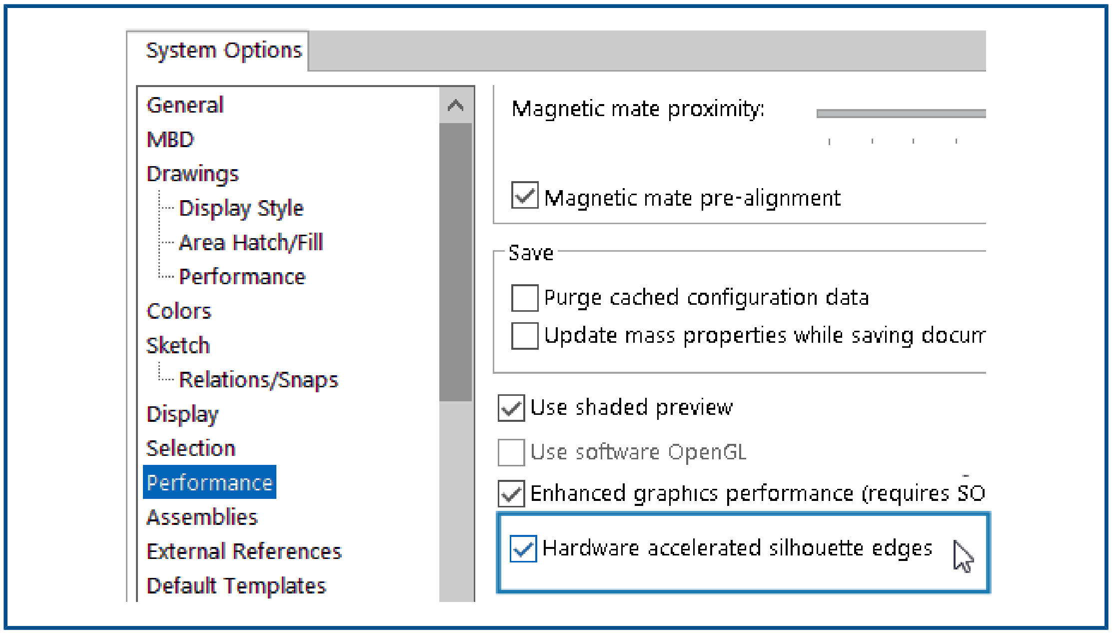
System Options

Option	Description	Access
Opposite hand mirror components	Defines default values for **Add Prefix** and **Add Suffix** when creating opposite-hand components.	**Assemblies**
Prefix for virtual components created from external files	Defines a default prefix for virtual components that are created from external files.	**Assemblies**
Display DimXpert dimensions on top of model	Controls the visibility of dimensions.	**Display**
Display SpeedPak graphics circle	Changed to a slider that allows the user to increase or decrease the transparency of the graphics circle.	**Display**
Drawings, Overridden dimensions	Specifies a color for overridden dimensions.	**Colors**
Hardware accelerated silhouette edges	Enables the GPU hardware to improve the display of silhouette edges in HLR, HLV, and wireframe view modes.	**Performance**
Preview sketch dimension when selected	Turns on sketch dimension previews.	**Sketch**
Always open a drawing in detailing mode	Opens a drawing by default in Detailing mode.	**Drawings** > **Performance**
Defeature Rule Sets	Under **Show folders for**, specifies a location for defeature rule sets, `*.slddrs`, and related log files.	**File Locations**
Only CAD Family View and **Both CAD Family and Configurations**	Removed from system options.	**FeatureManager**

Document Properties

Option	Description	Access
Decimal Separator	Specifies a value for the Decimal Separator. Options are **Comma** or **Period**.	**Annotations** > **Geometric Tolerances**
Highlight associated elements of dimension selection	Highlights the associated elements of a dimension.	**Detailing**
Offset text automatically when space is limited	Places dimension text that cannot fit within the extension lines outside of the extension lines on an extended dimension line.	**Dimensions** > **Linear**
When arrowhead overlaps substitute arrowhead termination automatically with:	Specifies arrowhead replacements when arrowheads overlap. Options are **Points** or **Oblique Strokes**.	**Dimensions** > **Linear**
Hole	(Available for parts only). Specifies the options for hole tables in the active document.	**Drafting Standard** > **Tables**
Highlight overridden dimensions in a different color	Displays the color of overridden dimensions.	**Dimensions**

Accelerate the Display of Silhouette Edges



You can enable the GPU hardware to improve the display of silhouette edges in HLR, HLV, and wireframe views.

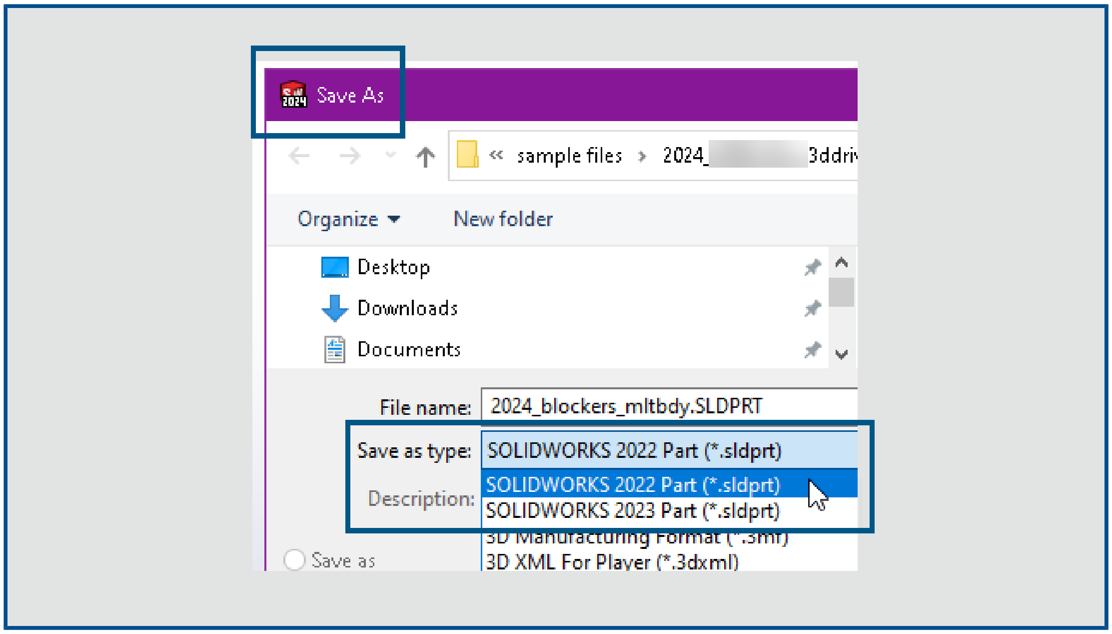
In **Tools** > **Options** > **System Options** > **Performance**, select **Hardware accelerated silhouette edges**.

Application Programming Interface

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

- Access the configuration-specific custom PropertyManagers of cut lists and assembly components
- Retrieve errors that occurred during the last call to IFeatureManager::CreateFeature
- Use the option, **Exclude parent surface**, to exclude the parent surface from the **Surface-Untrim** feature result
- Insert bills of materials (BOMs) in parts, assemblies, and drawings with detailed cut lists and specify whether to dissolve components in indented BOMs
- Get and set whether to display dual unit values in dimension range lengths of geometric tolerance symbols
- Get and set the decimal separator type for geometric tolerance symbols
- Get the diameter of a model's spherical bounding box

Saving SOLIDWORKS Documents as Previous Versions



Beginning with SOLIDWORKS 2024, you can save SOLIDWORKS parts, assemblies, and drawings created or saved in the latest version of SOLIDWORKS as fully functional documents in a previous version of SOLIDWORKS. You can save documents back to the previous two releases. Pack and Go also supports this functionality.

You can save SOLIDWORKS 2024 files as SOLIDWORKS 2023 or SOLIDWORKS 2022 versions. This previous release compatibility lets you share files with others who use one of the two previous versions of SOLIDWORKS. You cannot extend the previous release compatibility beyond those two releases.

SOLIDWORKS users must have an active subscription license to access this functionality. **3D**EXPERIENCE users are active subscribers by default.

Workflow

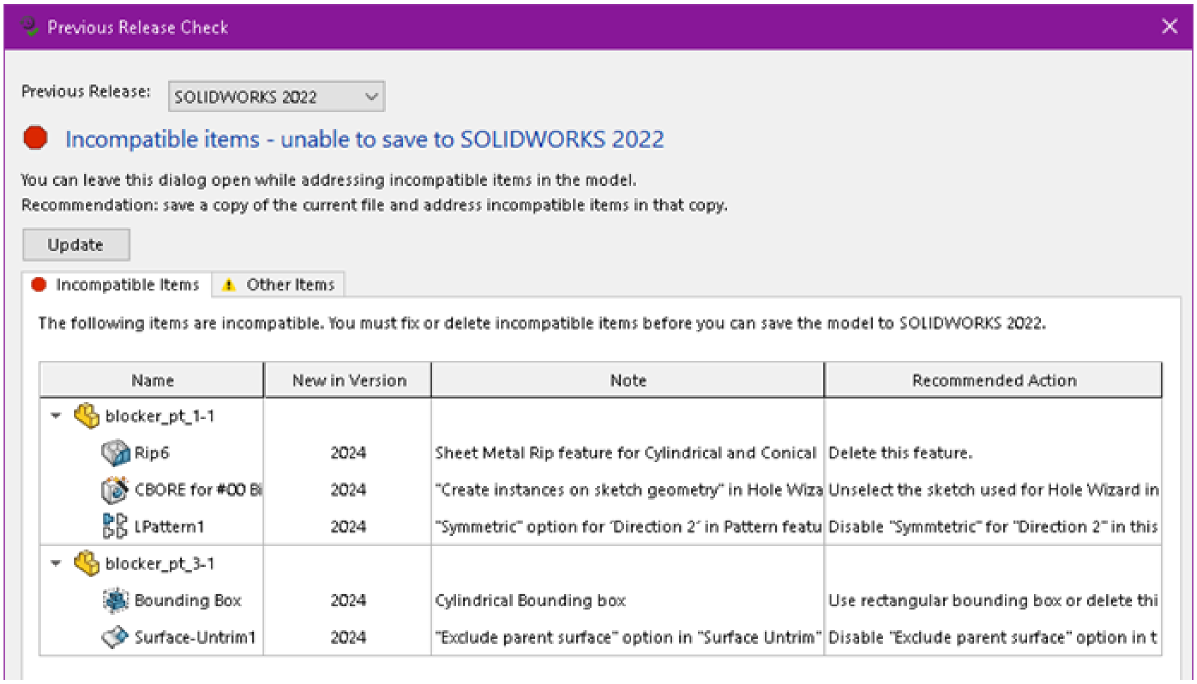
You must manually address incompatible items in this process. Incompatible items, as described in the table below, are items that do not exist or are not supported in the selected previous release.

Recommendation: Addressing incompatible items might significantly change a model. Save a copy of the current model and address incompatible items in that copy before saving it as a previous version.

To save a SOLIDWORKS document as a previous version:

1. Open or save a SOLIDWORKS document in the latest version of SOLIDWORKS.
2. Click **File** > **Save As**.
3. In the dialog box, for **Save as type**, select the previous version to which to save the document and click **Save**.

If the document contains Incompatible Items or Other Items as described below, the Previous Release Check dialog box appears. Otherwise, the software saves the document as the previous version.



> To open this dialog box at any time, click **Tools** > **Evaluate** > **Previous Release Check**.

Tab	**Description**
Incompatible Items	Lists items that you must manually address before you can save the file as a previous version of SOLIDWORKS. If you remove or edit the incompatible items, it might change the mass properties, size, shape, or rebuild behavior of the model. In some instances, you must delete the incompatible item. In other instances, changing a feature option might address the incompatible item. The list of incompatible items is in the order that they first appear in the FeatureManager design tree.
Other Items	Lists items that the software will automatically remove in the save process. These are items that do not impact the rebuild, mass properties, or topology of the document, such as display items like annotations or information on drawings.

> If the document contains only Other Items and no Incompatible Items, on the Other Items tab, click **Proceed With Save** to save the document to the previous version.

After you address all the Incompatible Items, a message confirms that the document is fully compatible with the selected previous release.

4. Repeat the save process to save the file as the previous version.

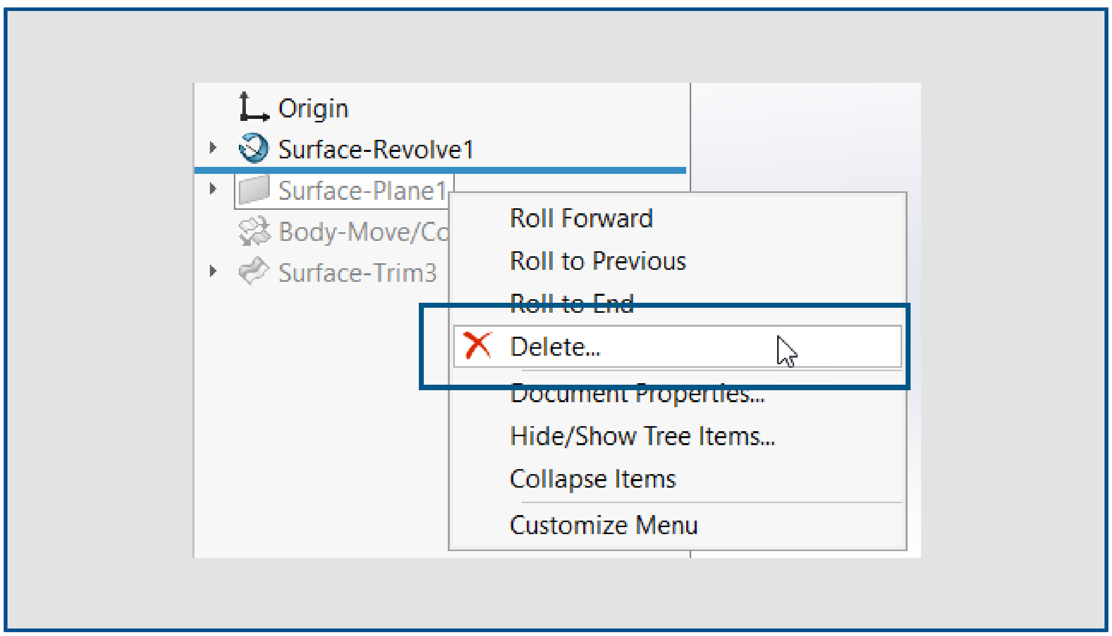
5

User Interface

This chapter includes the following topics:

- **Deleting Rolled-Back Features (2024 SP2)**
- **Usability**
- **Hide and Show**
- **Icon Updates for Open, Save, and Properties Commands**

Deleting Rolled-Back Features (2024 SP2)

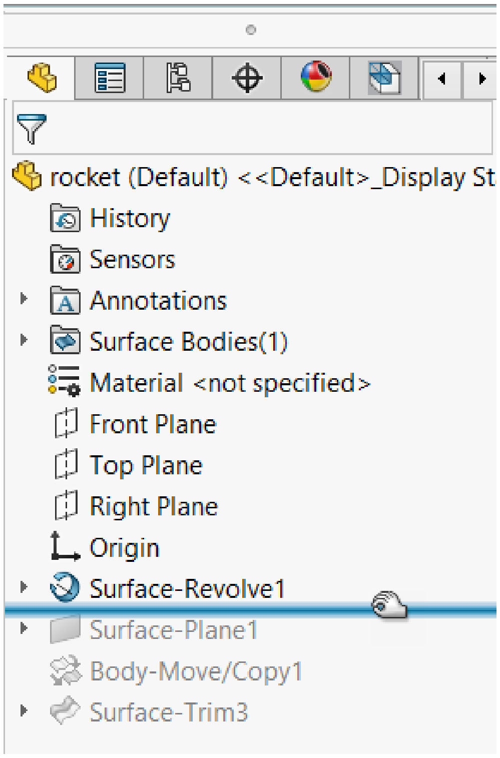


You can delete features that are in a rolled-back state from models.

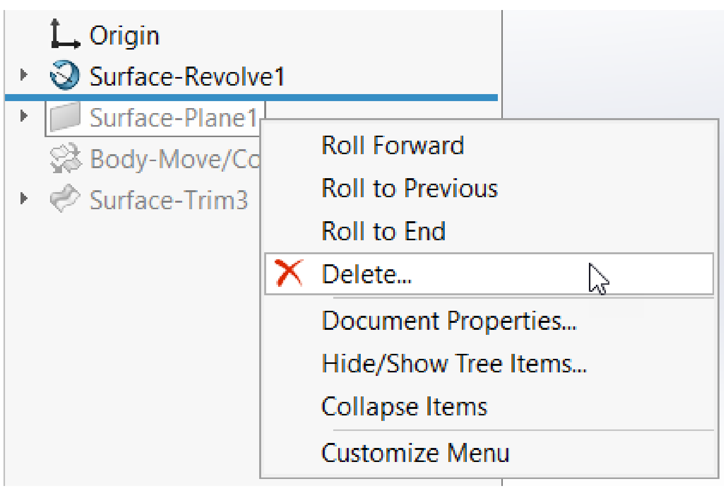
Benefits: You can delete rolled-back features that might have blocked you from completing your design.

To delete rolled-back features:

1. In the FeatureManager design tree of your model, drag the rollback bar to roll back some features.

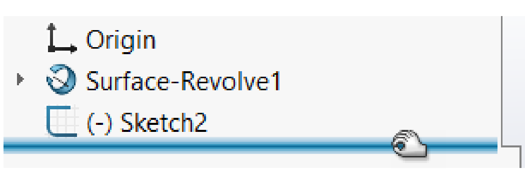


2. Right-click a rolled-back feature (below the rollback bar) to delete and click **Delete** ✗.



3. In the Confirm Delete dialog box, verify that you accept the deletion and click **Yes**.

   The feature and dependent items that you agreed to delete are deleted from the model. You can now drag the rollback bar to the bottom of the FeatureManager design tree to exit the rolled-back state.



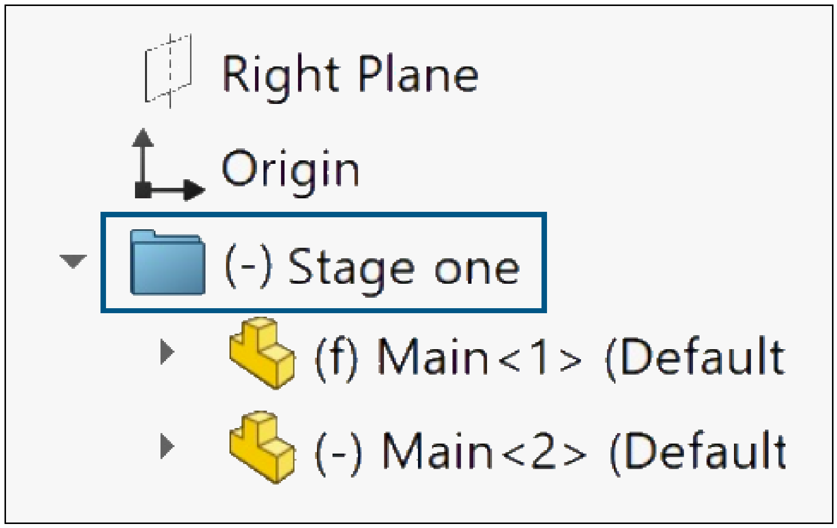
Usability

Usability (2024 SP2)

The user interface is enhanced to improve productivity.

The following items appear with SOLIDWORKS 2024 SP2.

Issues indicator for folders in the FeatureManager design tree

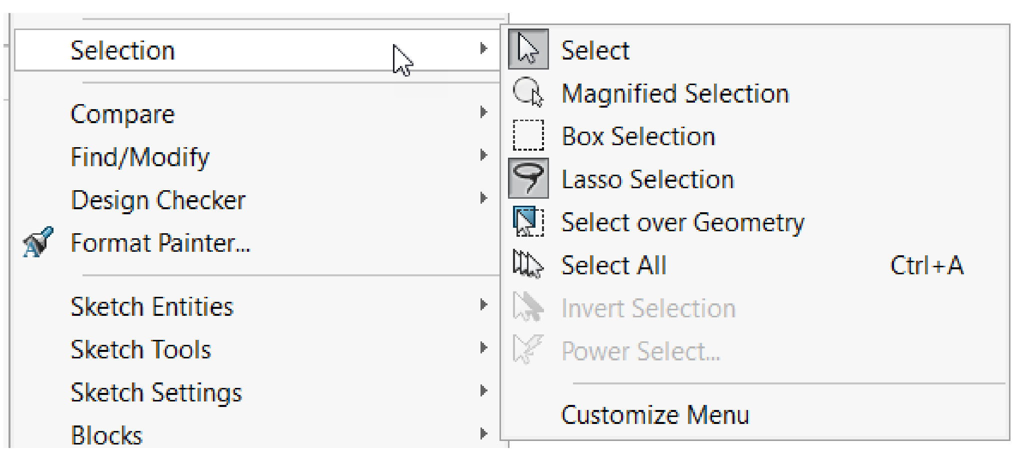


A prefix **(-)** appears next to the folder name to indicate that the folder has components with some issues.

In parts, the prefix indicates that some features have underdefined sketches or missing references. In assemblies, the prefix indicates that some components are underconstrained.

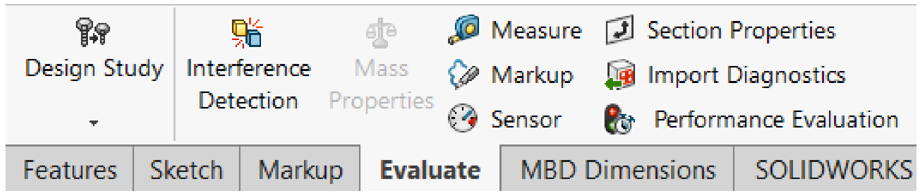
The prefix also appears if subfolders contain features or components that have these issues.

Tools > Selection submenu



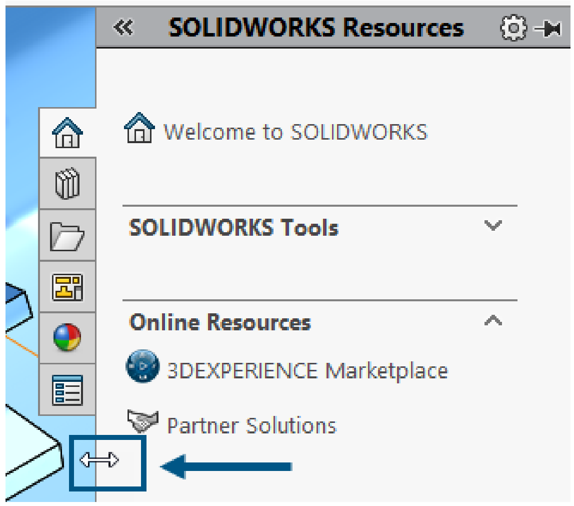
Under **Tools**, the **Selection** submenu contains all the selection commands that were previously listed directly under **Tools**. This gives you quicker access to the entire **Tools** menu.

Restructured CommandManager tab - Evaluate



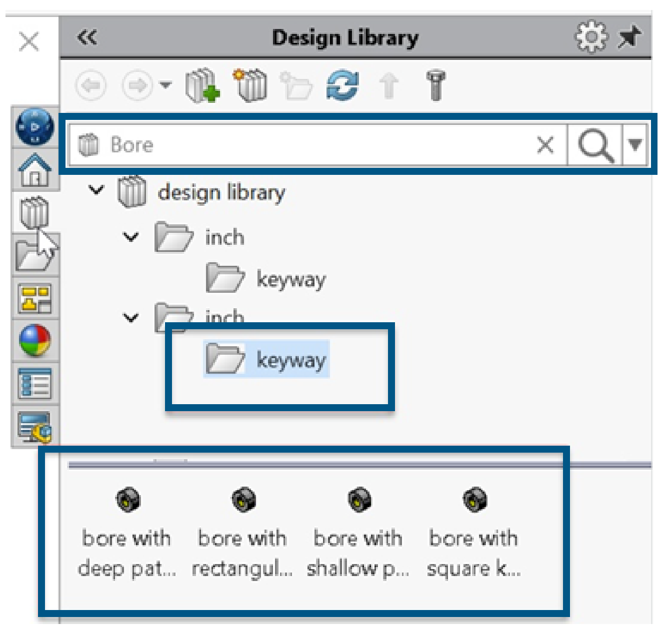
The Evaluate CommandManager tab for parts and assemblies is reorganized to provide quicker access to commands. The tab is unchanged for drawings.

Larger dragger and splitter lines

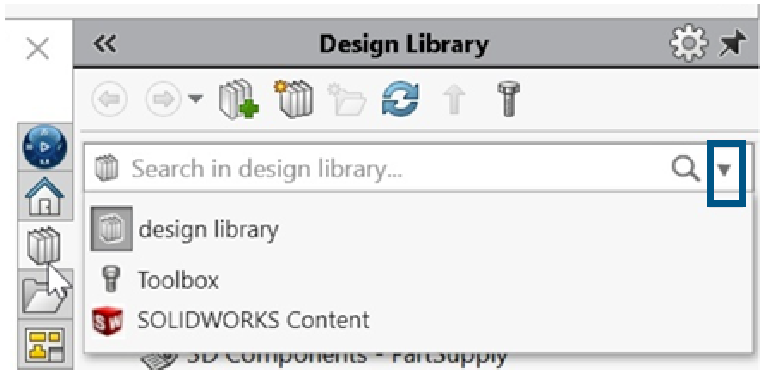


The drag zone for lines that you use to drag or split sections of the user interface are consistently sized. For example, the drag line in the Task Pane and the vertical adjuster line in Motion Studies are double the size of previous versions. This improves selection and dragging.

Searching the Design Library



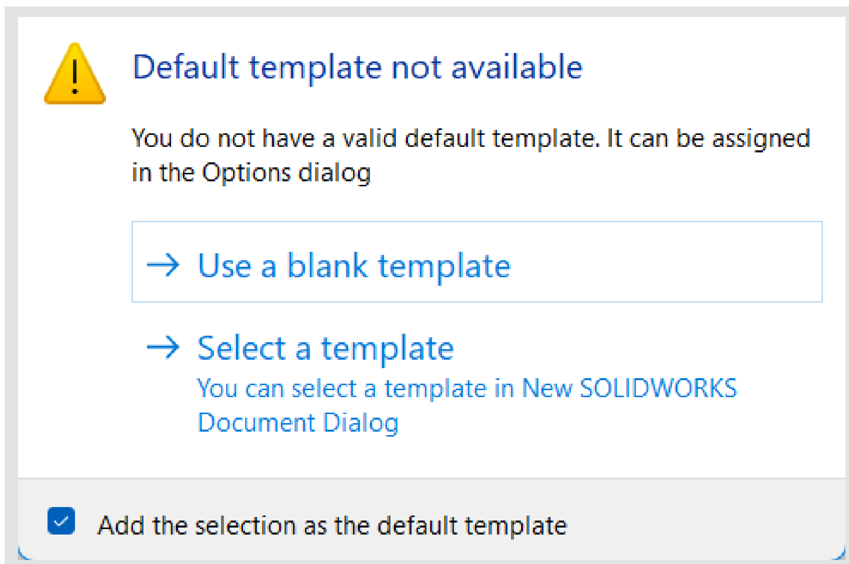
You can use the Search bar to search the Design Library or within a specific library. To limit the search to a specific library, click the down arrow and select a library.



In earlier releases, there was no search functionality for the Design Library.

If you select **Toolbox** but did not configure it, a prompt appears directing you to add in Toolbox.

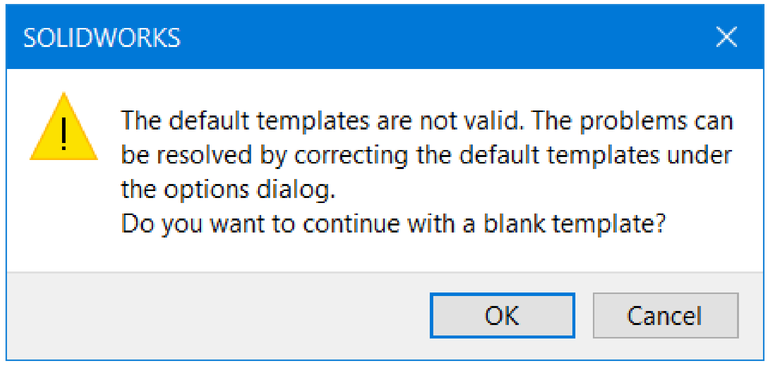
Dialog box for default templates



When there are issues with your default template not being available for parts, assemblies, or drawings, the Default template not available dialog box appears with these options:

- **Use a blank template**. Creates a default template.
- **Select a template**. Opens the New SOLIDWORKS Document dialog box where you can select a template to use.
- **Add the selection as the default template** check box. Applies the selected template to all files that you are opening. When you select this option, the **Default template not available** dialog box no longer appears for files that you open in the future that have issues with their default templates. Those files use the default templates that you have specify here.

In earlier releases, you received this alert.



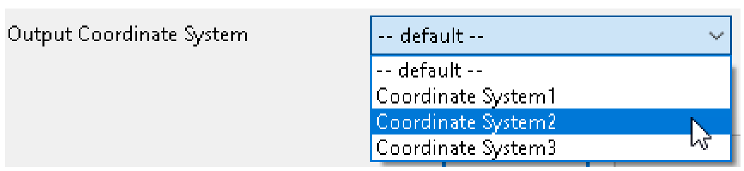
It appeared when you upgraded your version of SOLIDWORKS and had issues with the default templates, such as incorrect paths. Also, when **3D**EXPERIENCE users downloaded files from the platform, such as in an assembly, as the components downloaded, this alert appeared for each component with no option to apply your selected template to all the subsequent components.

Usability (2024 SP0)

The user interface is enhanced to improve productivity.

The following items appear with SOLIDWORKS 2024 SP0.

Coordinate System to Save

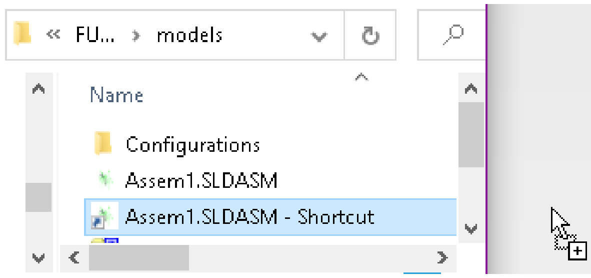


In the Save As dialog box, you can choose which coordinate system to save with a file. In the dialog box, in **Output Coordinate System**, specify the coordinate system to save. When you open the file, the new coordinate system is the origin.

This functionality does not apply to parts or assemblies. It applies to the following file types:

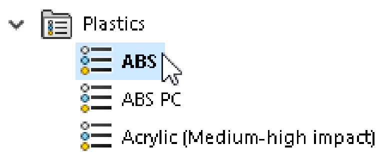
- `3D Manufacturing Format (*.3mf)`
- `ACIS (*.sat)`
- `Additive Manufacturing File (*.amf)`
- `IFC 2x3 (*.ifc)`
- `IFC 4 (*.ifc)`
- `IGES (*.igs)`
- `Parasolid (*.x_t;*.x_b)`
- `STEP AP203 (*.step;*.stp)`
- `STEP AP214 (*.step;*.stp)`
- `STL (*.stl)`
- `VDAFS (*.vda)`
- `VRML (*.wrl)`

Opening SOLIDWORKS Files from Shortcuts



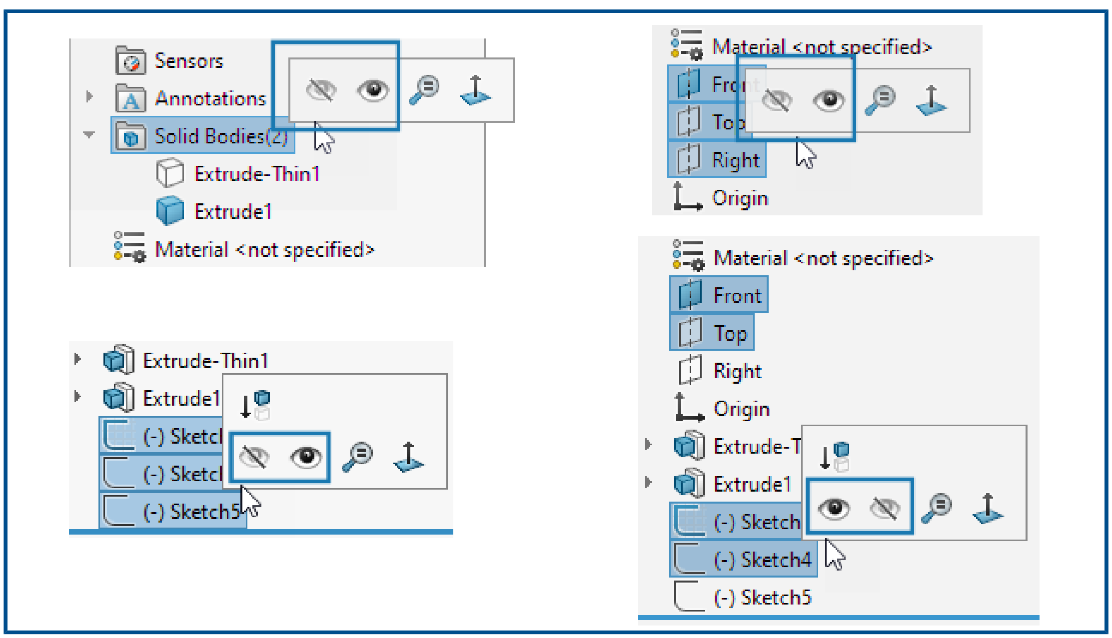
You can drop a shortcut to a SOLIDWORKS file directly from a local drive into SOLIDWORKS to open the file.

Selecting Materials



In the Material dialog box, you can double-click a material to automatically apply the material to the model and close the dialog box. You can still click **Apply** to review the material properties before applying the material.

Hide and Show



When you multiselect bodies, planes, or sketches that have a combination of shown and hidden states in the FeatureManager® design tree, the context toolbar shows both the **Hide** and **Show** tools. You can click **Hide** or **Show** to change the visibility state of all the selected entities.

The **Hide** and **Show** tools also appear when you multiselect a combination of hidden and shown planes and sketches. The **Show Hidden Bodies** tool is added to the **Tools** >

Customize > **Commands** > **Features** tab so you can add it to toolbars and the CommandManager. You can use the **Search** tool or the **S** key to find **Show Hidden Bodies** and **Show Hidden Components**.

Icon Updates for Open, Save, and Properties Commands

Tool icons are updated for Open, Save, and Properties commands for SOLIDWORKS and SOLIDWORKS **3D**EXPERIENCE apps.

Tool	2023	2024	Change
Open			Arrow color
Open Drawing			Arrow color
Save			Removed label lines and modernized
Save As			Removed label lines and moved pencil
Save All			Removed label lines and modernized
Save to 3DEXPERIENCE (**3D**EXPERIENCE users only)			New icon with cloud
Save to This PC (**3D**EXPERIENCE users only)			Removed label lines and modernized
Older Version File			Removed label lines and modernized
PLM Properties (**3D**EXPERIENCE users only)			New icon to distinguish it from standard Properties icon

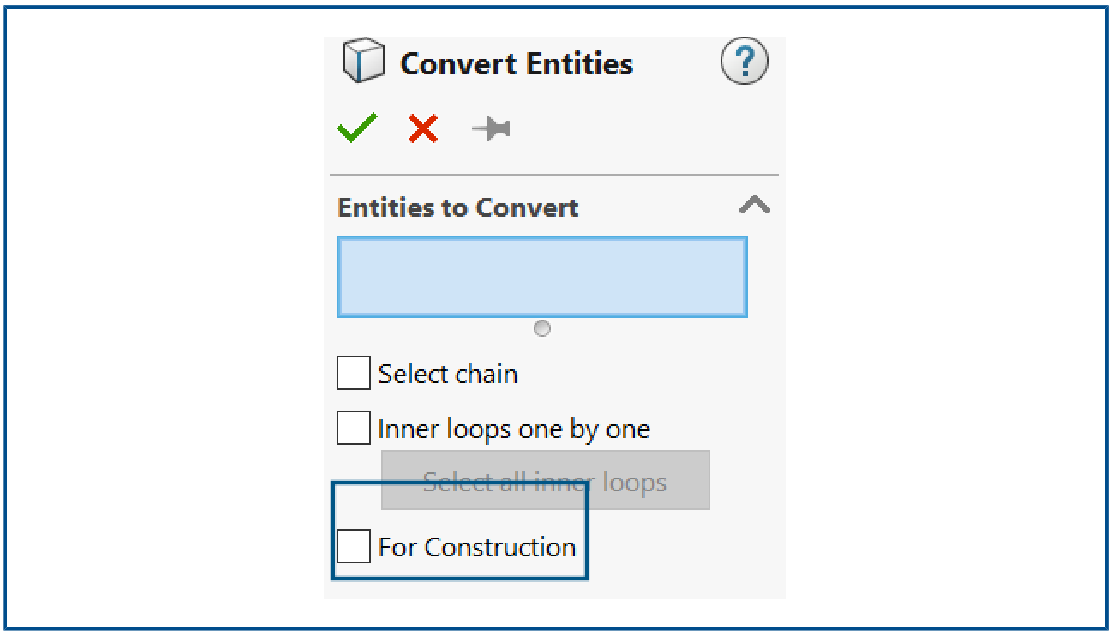
6

Sketching

This chapter includes the following topics:

- **Convert Entities as Construction Geometry (2024 SP1)**
- **Sketch Blocks**
- **Sketch Dimension Previews**

Convert Entities as Construction Geometry (2024 SP1)

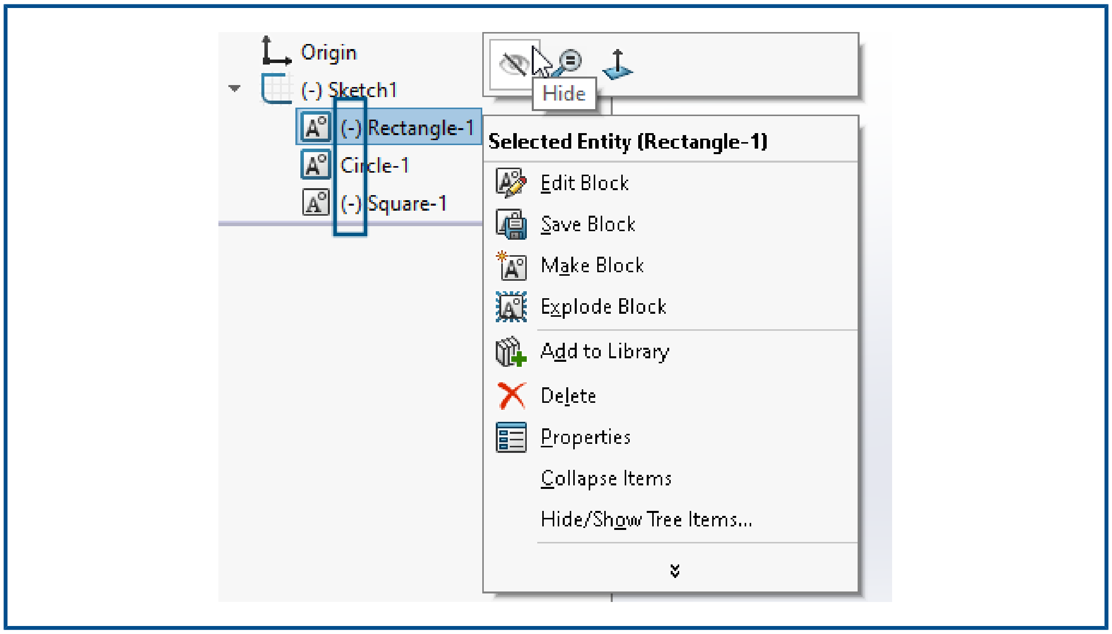


In the Convert Entities PropertyManager, you can convert selected sketch entities into construction geometry.

To convert the entities into construction geometry in a sketch,

1. Click **Convert Entities**
2. Select the sketch entities to convert
3. Select **For construction**.

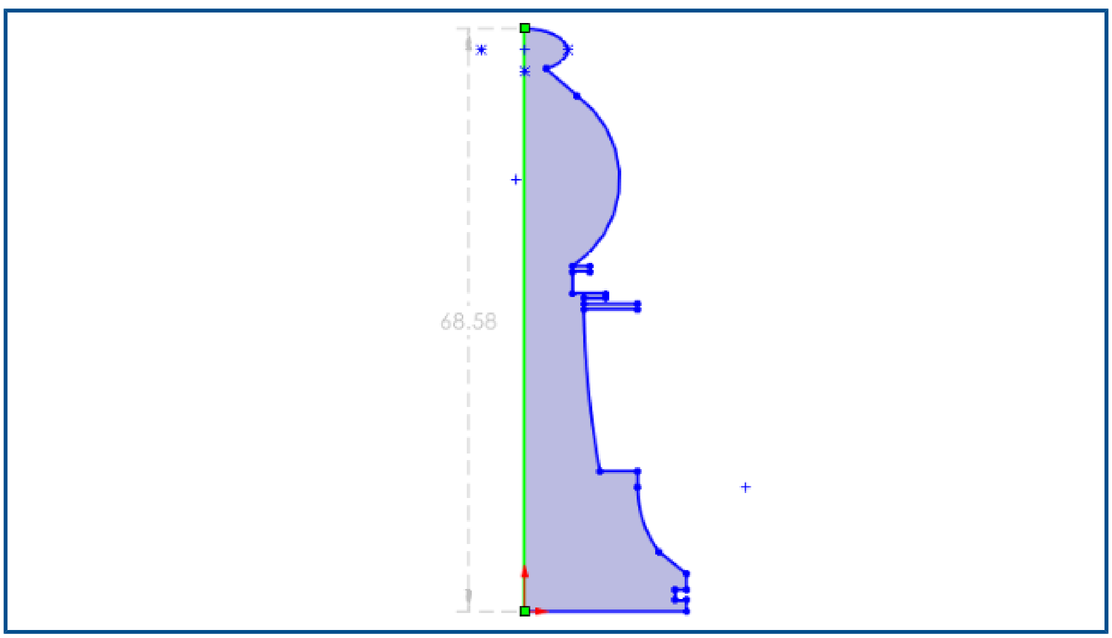
Sketch Blocks



In the FeatureManager® design tree, you can hide and show individual blocks in sketches. You can also see whether a block is under defined (-), over defined (+), or fully defined.

To hide and show individual blocks in sketches, right-click the sketch block in the FeatureManager design tree and click **Hide** or **Show**.

Sketch Dimension Previews



You can preview sketch dimensions when you select a sketch entity.

You can select the dimension to edit it. When you click anywhere else in the graphics area, the preview dimension disappears.

To turn on sketch dimension previews, click **Tools** > **Options** > **System Options** > **Sketch** and select **Preview sketch dimension when selected**.

To change the dimension preview color, click **Tools** > **Options** > **System Options** > **Colors**. Under **Color scheme settings**, edit the color for **Dimensions, Preview**.

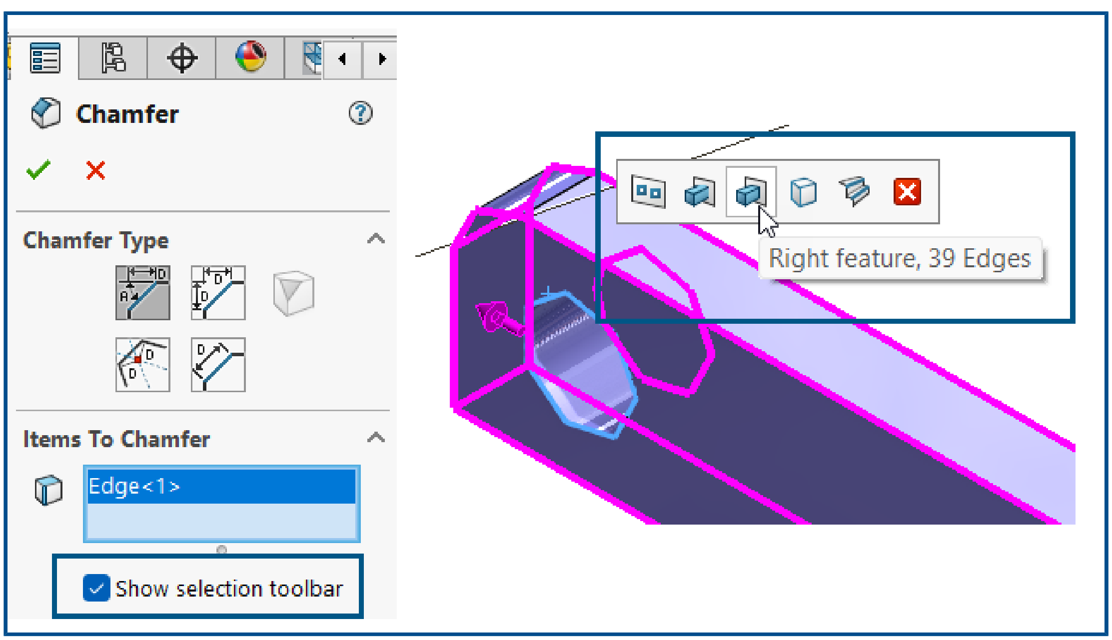
> Sketch dimension previews are not supported for path lengths.

7

Parts and Features

This chapter includes the following topics:

Selection Accelerator Toolbar for Chamfers (2024 SP2)



A selection accelerator toolbar is available for chamfers so you can quickly select edges to chamfer.

Benefits: You spend less time on details and have more time for design.

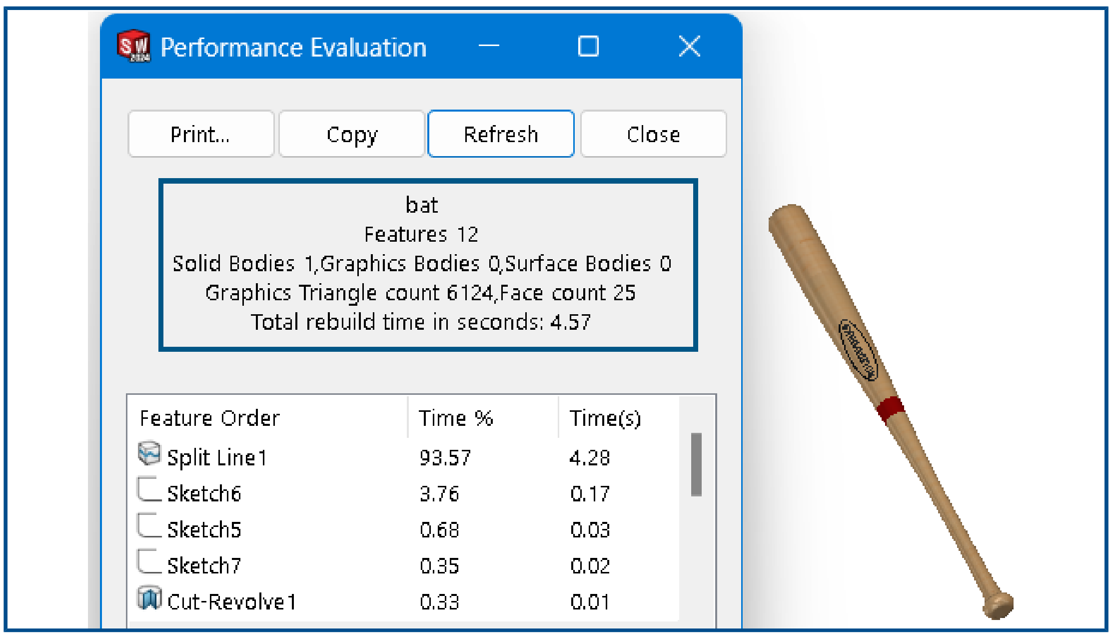
To use the selection accelerator toolbar:

1. In the Chamfer PropertyManager, click **Show selection toolbar** to activate the toolbar.
2. For **Items to Chamfer**, select an edge to display the selection toolbar in the graphics area.
3. Hover over the available selections on the toolbar to display the selected edges on the model in the graphics area. To select those edges, click the item on the toolbar.

The selection accelerator toolbar is available for these types of chamfers:

- **Angle Distance**
- **Distance Distance**
- **Offset Face**

Graphics Triangle and Face Count (2024 SP1)

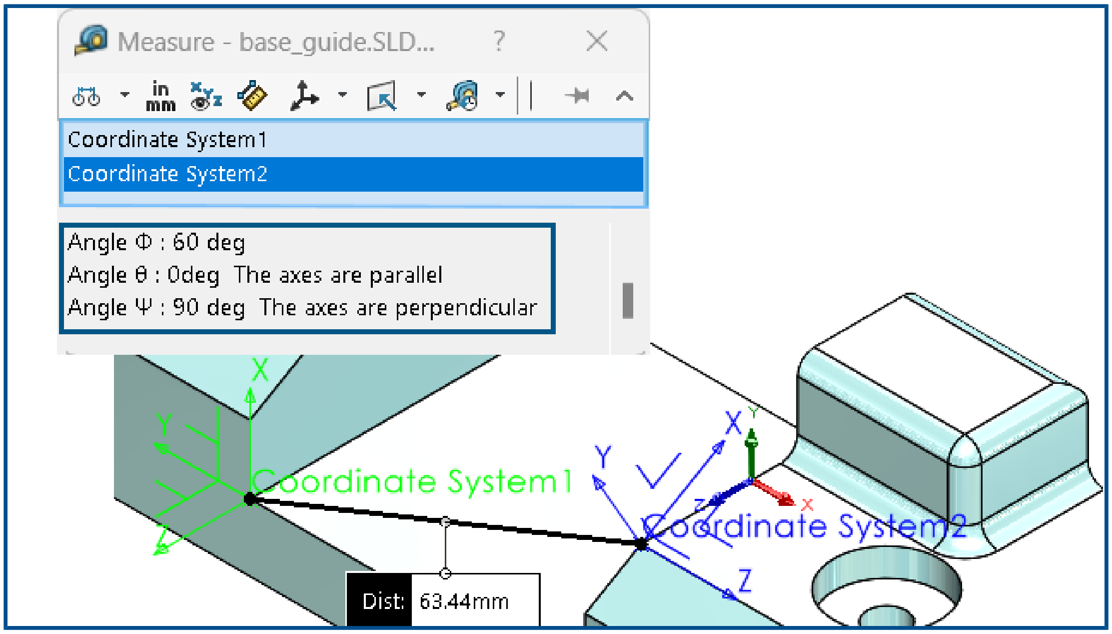


For parts, the Performance Evaluation dialog box displays the total number of graphics triangles and faces of all bodies combined plus other useful information.

The dialog box also displays the number of solid, graphics, and surface bodies, and the total rebuild time in seconds. To access this information, with a part open, click **Performance Evaluation** (Evaluate toolbar) or **Tools** > **Evaluate** > **Performance Evaluation**.

This information helps you determine the complexity of the model's geometry and the potential impact on performance.

Measuring the Angular Rotation between Coordinate Systems (2024 SP1)



You can measure the angular rotation between two coordinate systems.

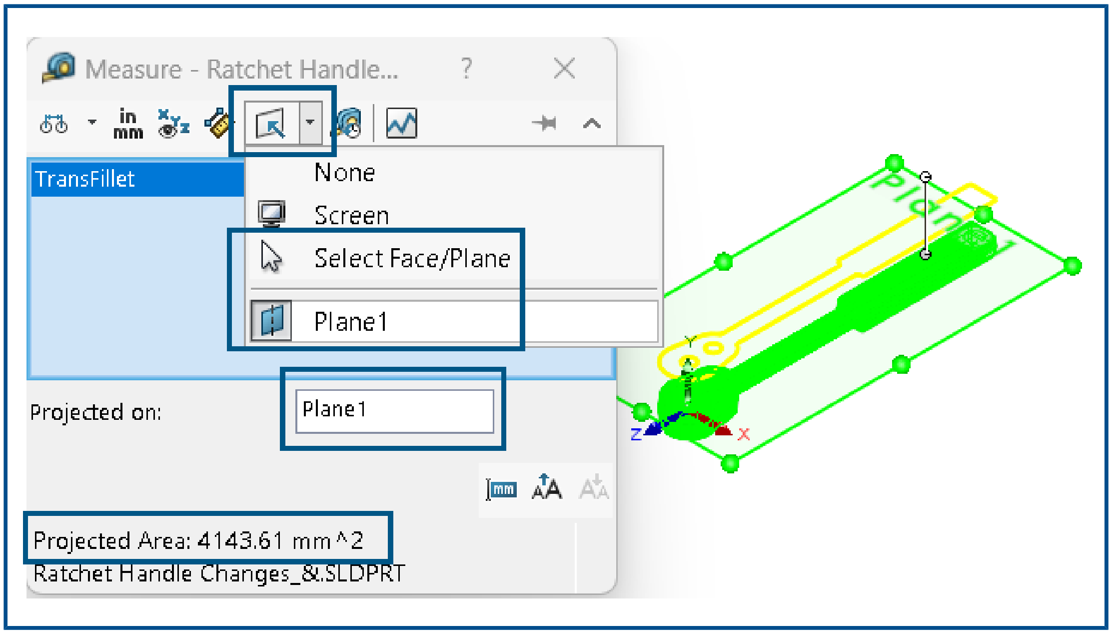
Click **Measure** (Tools toolbar) or **Tools** > **Evaluate** > **Measure**. In the graphics area, select the two coordinate systems. The results appear in the output section as roll (Phi Φ - X-axes), pitch (Theta Θ - Y-axes), and yaw (Psi Ψ - Z-axes).

Scroll to the bottom of the Measure dialog box to see the results.

The software calculates the angle of rotation based on the Tait-Bryan (XYZ method) rotation theory.

All angles appear with positive values. Parallel angles appear as zero or 360 degrees while perpendicular angles appear as 90 or 270 degrees. Text also appears to indicate parallel or perpendicular angles.

Measuring the Projected Surface Area of Bodies (2024 SP1)



You can measure the projected surface area of bodies, faces, and components. The selections must be solid or surface bodies. In previous releases, you had to create a sketch and use silhouette entities to calculate this value.

The projected surface area is useful in designing molds for plastic parts. Combined with the pull direction, the projected surface area helps you calculate the cost of the part and the machine tonnage.

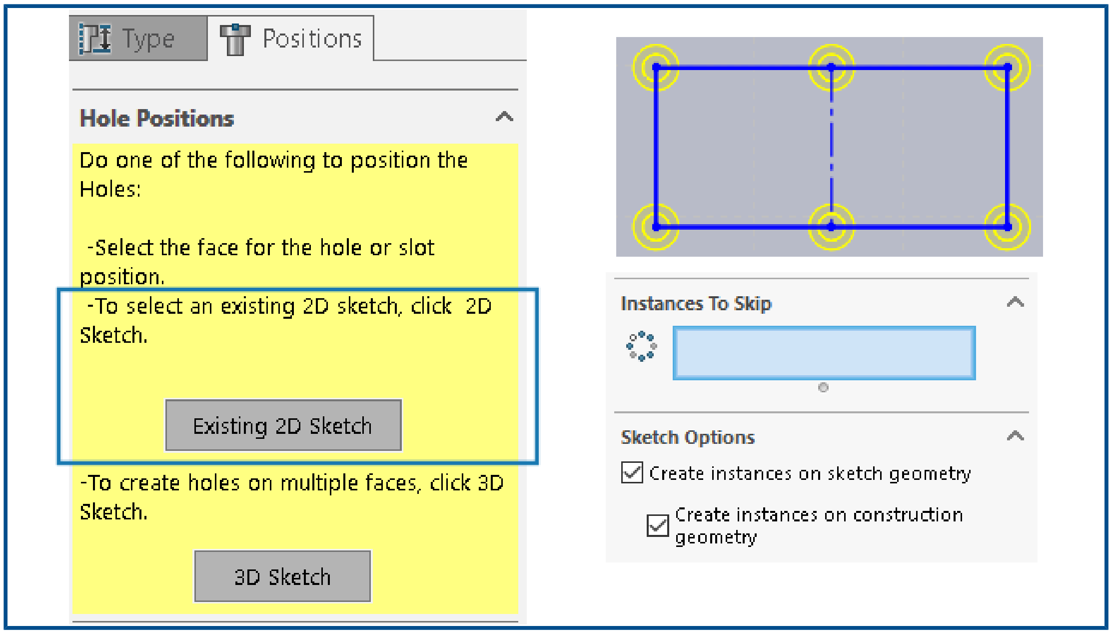
To measure the projected surface area of a model:

1. Click **Measure** (Tools toolbar) or **Tools** > **Evaluate** > **Measure**.
2. Select solid or surface bodies, faces, or components of the model.
3. In the dialog box, in **Projected On**, click **Select Face/Plane**, and select the planar face onto which to project the bodies, faces, or components.

   The software projects a silhouette of the selections onto the selected planar face and calculates the projected area.

   In the dialog box, **Projected Area** shows the value for the projected surface area of the bodies, faces, and components.

Hole Wizard



Sketching with the Hole Wizard is enhanced when you use the Positions tab of the PropertyManager.

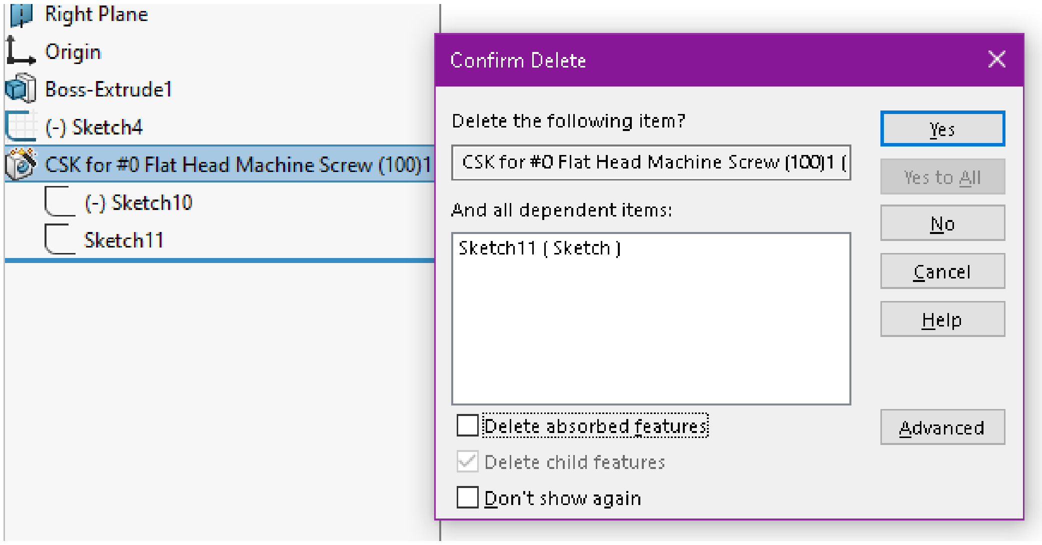
Under **Hole Positions**, you can click **Existing 2D Sketch** and select an existing 2D sketch to position and automatically create the holes at all endpoints, vertices, and points of the sketch geometry. You can select sketch entities like lines, rectangles, slots, and splines. **Sketch Options** specify the geometry used to automatically create the instances.

Under **Sketch Options**, there are two options:

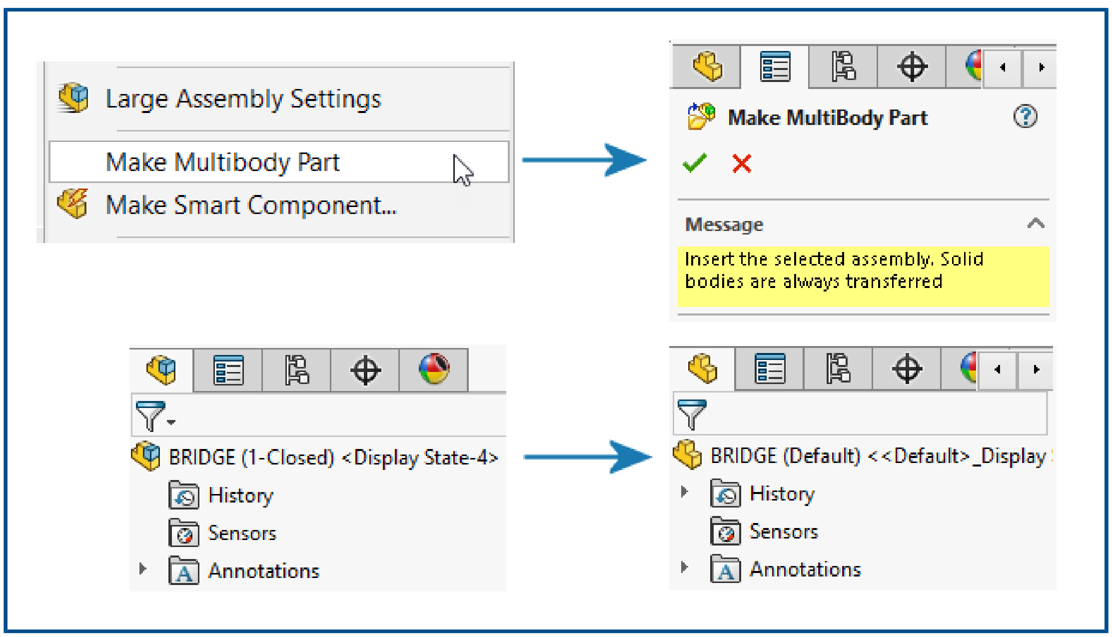
- **Create instances on sketch geometry** (Enabled by default). Positions holes at all endpoints, vertices, and points of the sketch geometry.
- **Create instances on construction geometry**. Positions holes at all endpoints, vertices, and points of the construction geometry.

You can skip hole instances. Under **Instances to Skip**, select hole instances to skip in the graphics area.

When you delete a Hole Wizard feature, you can retain the hole position sketch. In the Confirm Delete dialog box, clear the **Delete absorbed features** option to delete only the hole profile sketch and keep the hole position sketch. To delete the hole position sketch, select **Delete absorbed features**.



Making Multibody Parts from Assemblies

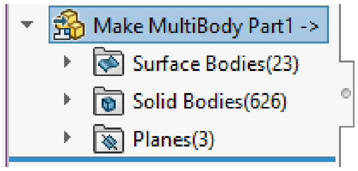


The **Make Multibody Part** tool converts an entire assembly into a separate, single multibody part that is linked to the parent assembly.

The multibody part reflects all the assembly features that you create in the parent assembly. Features that you create on the multibody part will not be reflected in the parent assembly. You can perform post-assembly operations on the multibody part, such as material removal, and these appear in downstream platform applications.

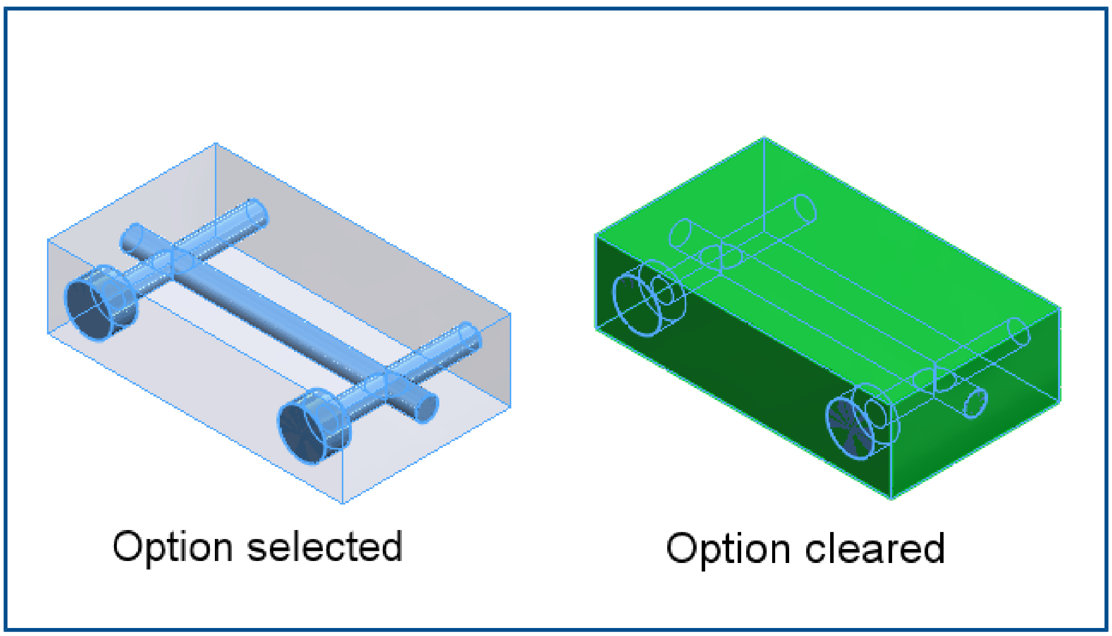
To create a multibody part, in an assembly, click **Tools** > **Make Multibody Part**.

The **Make Multibody Part** feature appears in the FeatureManager® design tree. Solid bodies are transferred by default. You can decide which other assembly entities to transfer such as surface bodies, reference geometry, and materials. Under the **Make Multibody Part** feature, the tool groups the entities into folders that show the number of instances.



All the bodies in the multibody part inherit their names from the assembly. They also match the position of the parts relative to the origin in the parent assembly. You can choose the configuration to create the multibody part.

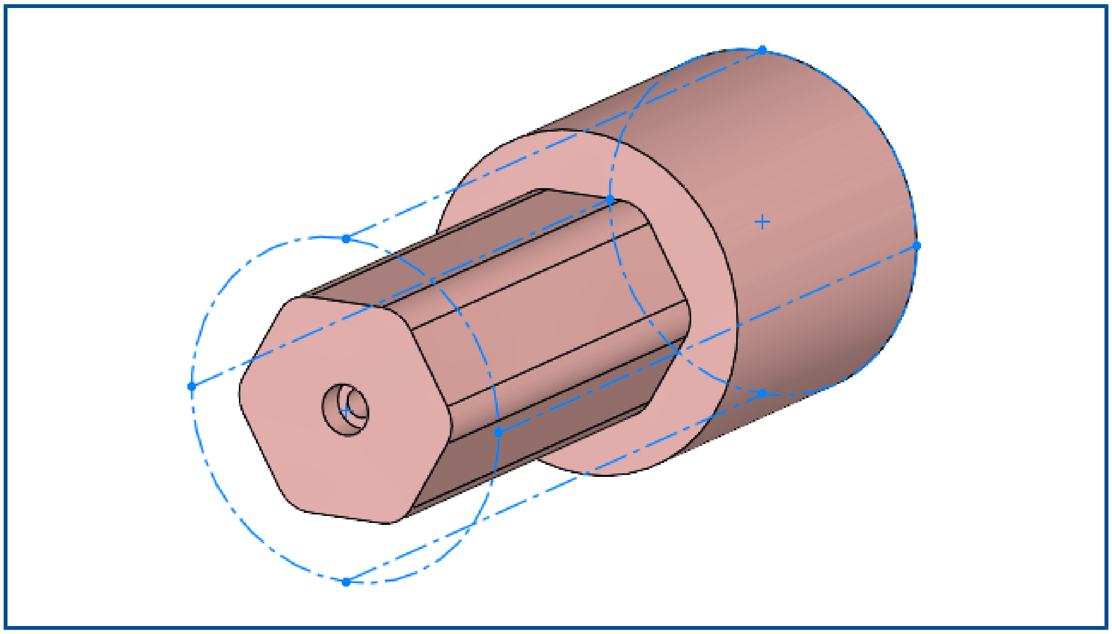
Body Transparency for Combine Features



In the Combine PropertyManager, for the **Subtract** operation, you can make the main body transparent. This helps you select smaller bodies that are completely immersed inside the main body.

Click **Insert** > **Features** > **Combine**. In the PropertyManager, under **Operation Type**, select **Subtract** and under **Main Body**, select **Make main body transparent**.

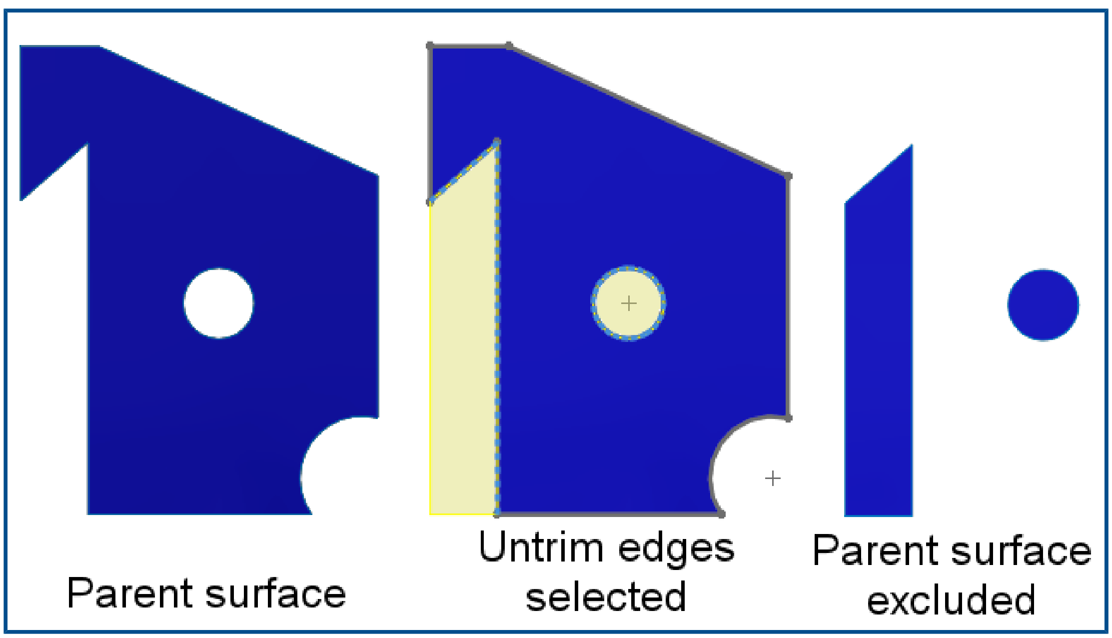
Cylindrical Bounding Boxes



You can create cylindrical bounding boxes that are useful for bodies with cylindrical geometry such as rotational, circular, or turned parts. SOLIDWORKS® captures the bounding box parameters and records them in the Custom Properties dialog box.

Click **Insert** > **Reference Geometry** > **Bounding Box**. In the PropertyManager, under **Type of Bounding Box**, select **Cylindrical**. SOLIDWORKS generates the smallest cylindrical bounding box that fits the model.

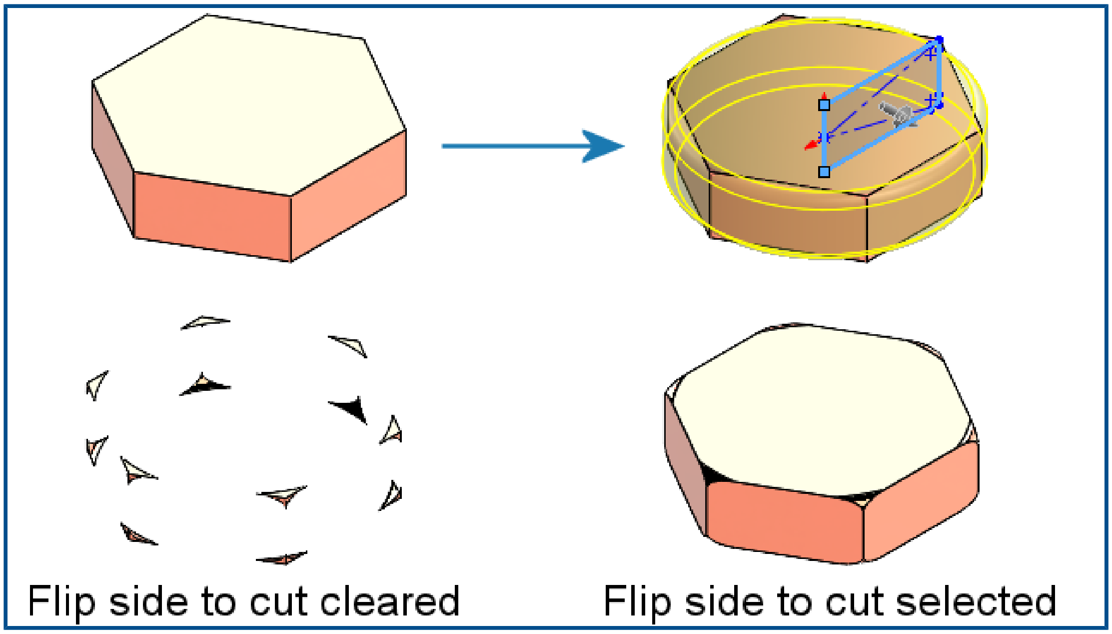
Excluding Parent Surfaces in Untrim Features



You can exclude the parent surface from the results of **Surface-Untrim** features. In the Untrim Surface PropertyManager, under **Options**, select **Exclude parent surface** to exclude the parent surface from the **Surface-Untrim** feature results.

To view the **Surface-Untrim** feature, hide the parent surface. This option simplifies the control of the untrimmed surfaces. In earlier releases, you had to use multiple tools to achieve the required results.

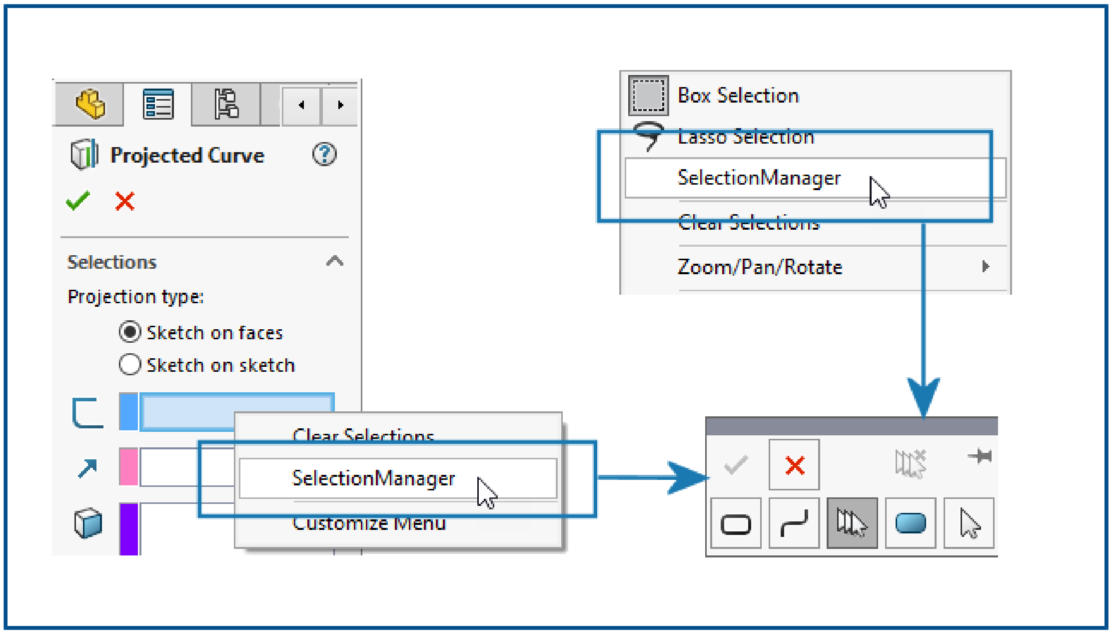
Flip Side to Cut for Cut Revolves



You can flip the side to cut for cut-revolve features, similar to cut-extrude features. This retains the inner portion of a sketch and discards the region outside the sketch.

In the Cut-Revolve PropertyManager, under **Direction 1**, select **Flip side to cut**. In earlier releases, this option did not exist and required extra steps to achieve the required results.

SelectionManager for Projected Curves



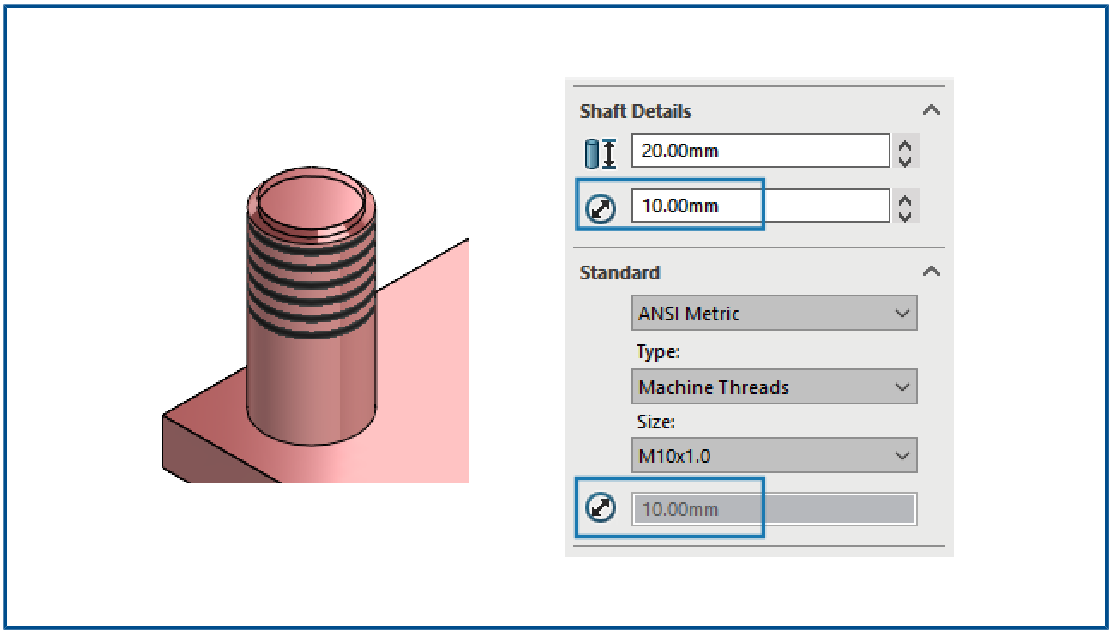
In the Projected Curve PropertyManager or if you right-click in the graphics area, you can use the SelectionManager to select portions of sketches to create projected curves.

To access the Projected Curve PropertyManager, click **Insert** > **Curve** > **Projected**.

> With the SelectionManager, you can select only one continuous group of entities. You cannot select multiple disconnected entities.

In earlier releases, the SelectionManager was not available and you could project only the entire sketch.

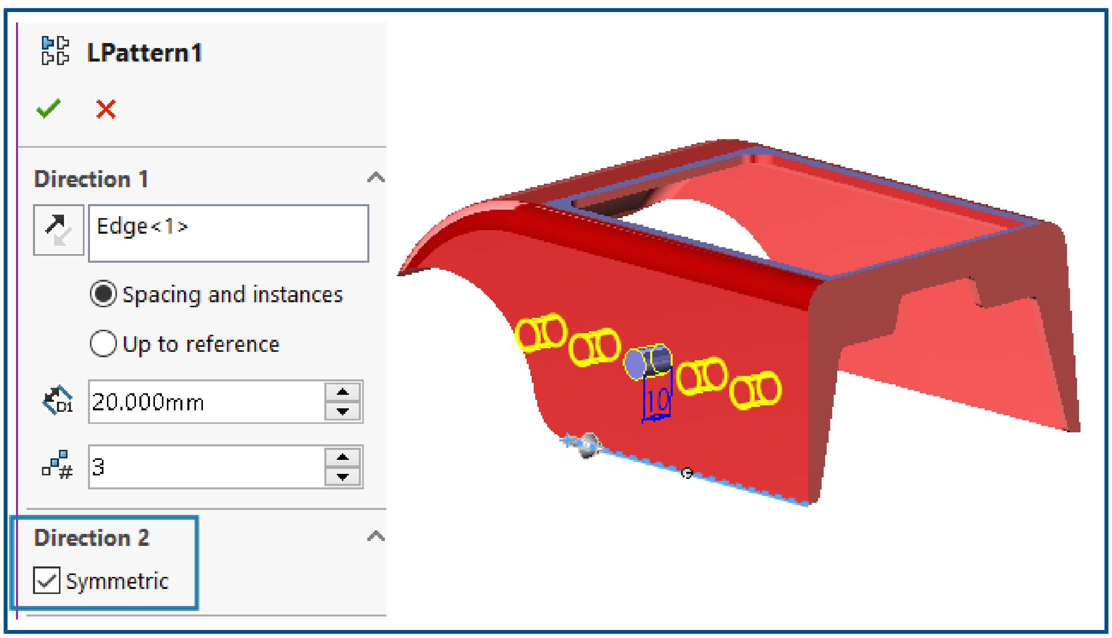
Stud Wizard



You can apply a **Stud Wizard** feature to a shaft that has the same diameter as the thread. You can modify the size of **Stud Wizard** features created in previous versions of SOLIDWORKS so the thread diameter matches the shaft diameter.

The software supports this functionality for studs created on a cylindrical body or surface. In earlier releases, the thread diameter had to be smaller than the shaft diameter.

Symmetrical Linear Patterns



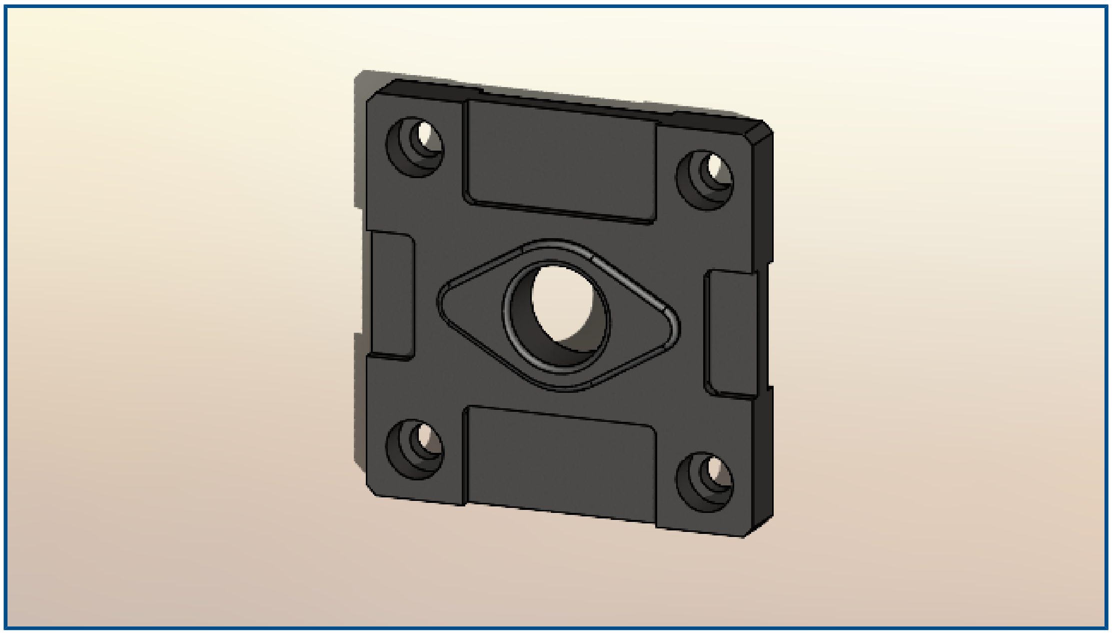
You can create symmetrical linear patterns from a seed feature. The linear pattern uses the parameters from **Direction 1** to create a symmetrical linear pattern in **Direction 2**.

In the Linear Pattern PropertyManager, under **Direction 2**, click **Symmetric** to create a symmetrical linear pattern using the **Direction 1** parameters.

8

Model Display

Materials for 3DEXPERIENCE Models (2024 SP2)



The software maps SOLIDWORKS physical materials applied to bodies and parts in SOLIDWORKS models to bodies and parts of models on the **3D**EXPERIENCE platform. In previous releases, mapping was not supported.

For information about prerequisites for SOLIDWORKS physical materials, see **https://help.3ds.com/HelpDS.aspx?P=11&F=SwsUserMap/sws-t-materialmgmt.htm** Managing Materials in 3DEXPERIENCE.

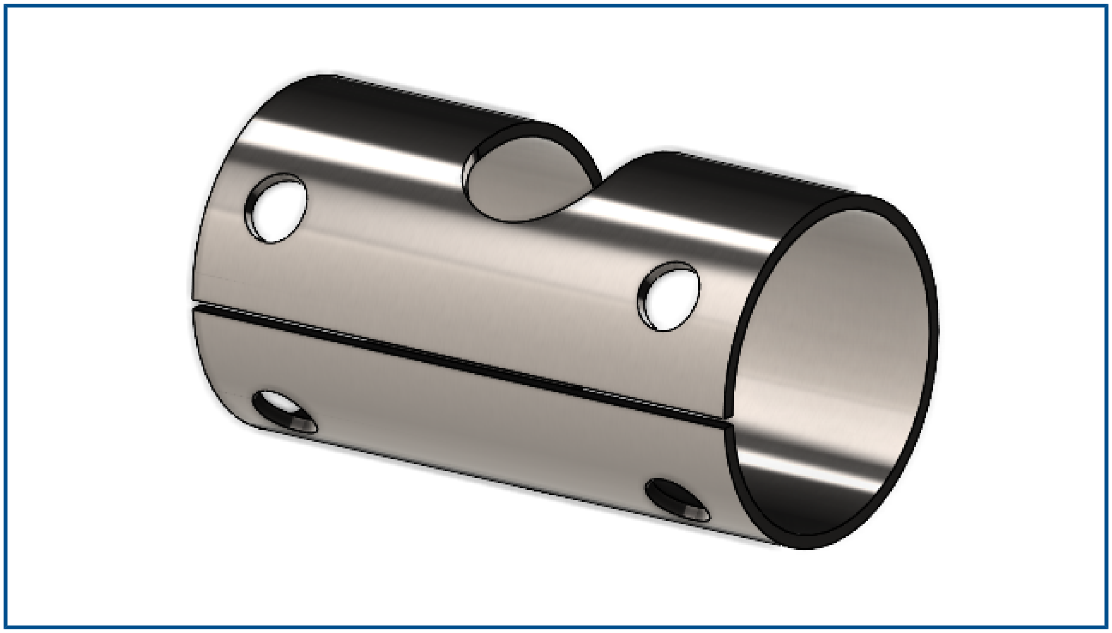
9

Sheet Metal

This chapter includes the following topics:

- **Rip Tool**
- **Slot Propagation**
- **Stamp Tool**
- **Normal Cut in Tab and Slot**

Rip Tool



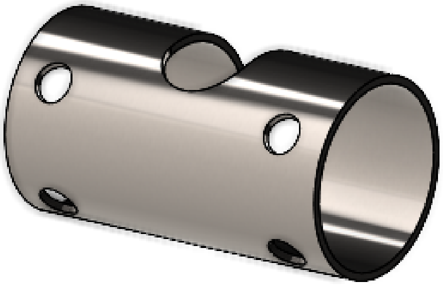
You can use the **Rip** tool to create rips in hollow or thin-walled cylindrical and conical bodies. By selecting an edge on a cylindrical or conical face, you can flatten the part as sheet metal.

In earlier releases, if you had a cylindrical or conical part, you had to create an intentional gap in the base sketch to convert the part to sheet metal.

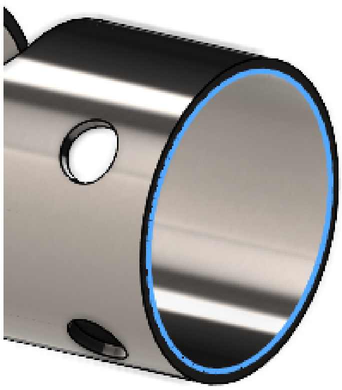
SOLIDWORKS supports straight cuts only, not slanted cuts.

To use the rip tool in a cylindrical part:

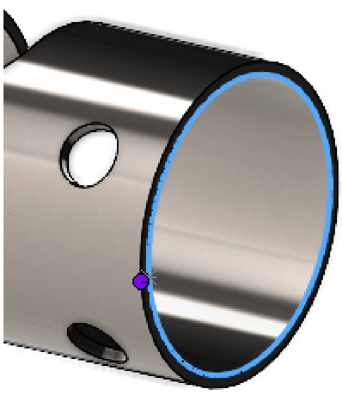
1. In a hollow or thin-walled cylindrical or conical part, click **Rip** (Sheet Metal toolbar).



2. In the graphics area, select:
   a. An edge.

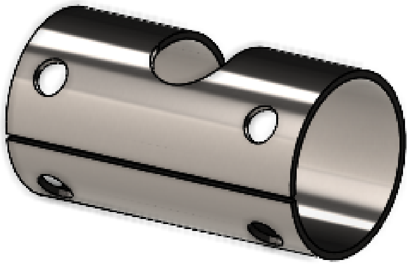


   b. A reference point on the model.



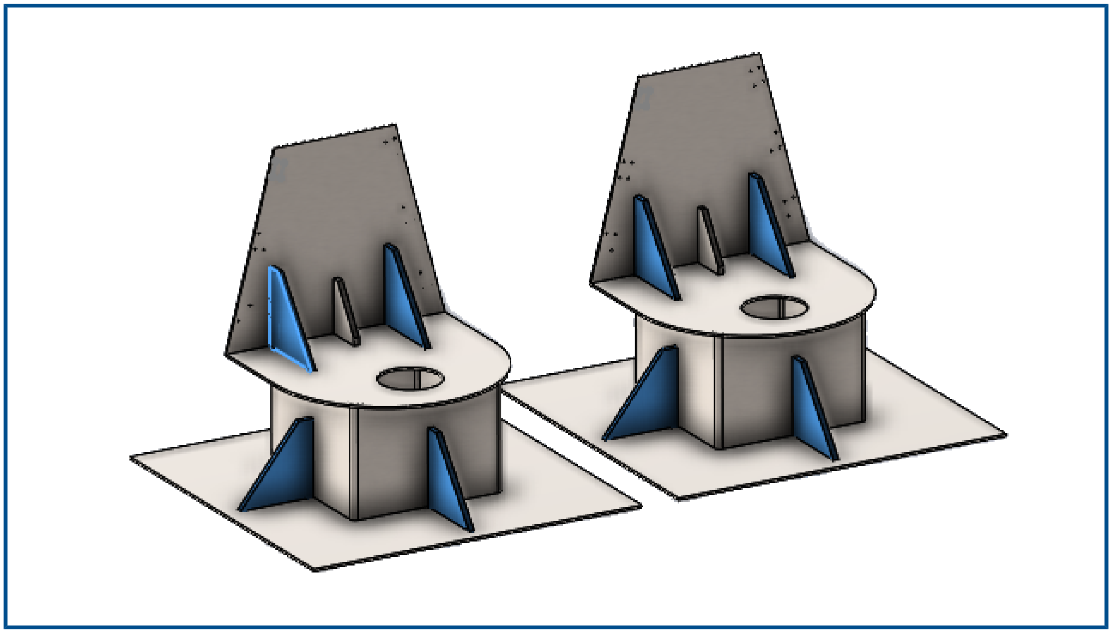
The reference point can be on the model or anywhere in the graphics area. If you select a reference point that is not on the model, the software projects the point onto the model.

3. Specify options in the PropertyManager and click ✔.



With the rip completed, you can convert the part to sheet metal using the **Insert Bends** tool.

Slot Propagation



When creating a tab and slot feature in an assembly component, you can propagate the slots to other instances of the same component in the assembly.

If an assembly has a component with a tab previously created with the **Tab and Slot** tool, you can propagate slots for that tab to other instances of the component in the assembly as well.

For example, if you have an assembly with multiple instances of a part with a tab, you can propagate slots for the corresponding instances.

Slots propagate only when the tab component intersects with the slot component.

If you pattern or mirror a component with a tab, you can select **Propagate Slots** in the PropertyManager to apply slots to intersecting components in the assembly.

To use slot propagation for assemblies when creating tab and slot features:

1. In an assembly, click **Tab and Slot** (Sheet metal toolbar).

2. In the graphics area, select an edge for the tabs and a corresponding face for the slots.
3. Specify options in the PropertyManager.

   If SOLIDWORKS detects multiple instances of the component in the assembly, you can specify options under **Propagate Slots**:

   - **Only selected**. Propagates slots to the selected component only.
   - **All instances in same parent assembly**. Propagates slots to all instances of the selected component that are in the same parent assembly.
   - **All instances**. Propagates slots to all instances of the selected component.

4. Click ✓.

To use slot propagation for assemblies with existing tab and slot features:

1. In an assembly with a component that has a tab and slot, right-click the component and click **Propagate Slots**.
2. In the Slot Propagation PropertyManager, under **Instances for slot propagation**, specify an option:

   - **Only selected**. Propagates slots to the selected component only.
   - **All instances in same parent assembly**. Propagates slots to all instances of the selected component that are in the same parent assembly.
   - **All instances**. Propagates slots to all instances of the selected component.

3. Click ✓.

Slot Propagation PropertyManager

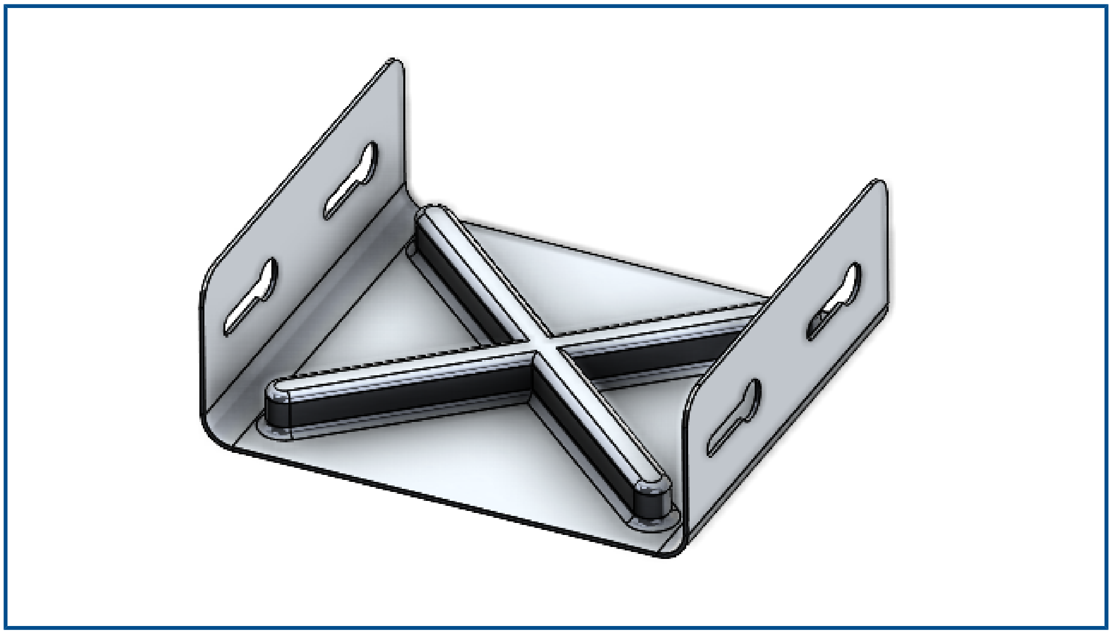
To open this PropertyManager:

1. In an assembly with a component that has a tab and slot, right-click the component and click **Propagate Slots**.

Selection

Propagate slots for these component(s)	Lists the components to apply the slots to.
Instances for slot propagation	Specifies which components to propagate slots to: • **Only selected**. Propagates slots to the selected components. With this option, you can delete specific components from the list. • **All instances in the same parent assembly**. Propagates slots to all instances of the selected components that are in the same parent assembly. • **All instances**. Propagates slots to all instances of the selected components. With this option, if some components already have a slot, they are ignored.

Stamp Tool



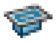
You can use the **Stamp** tool to create sketch-based parametric forming tools to apply to sheet metal parts. With sketch-based forming tools, you can create a sketch with a few parameters to stamp or form the sheet metal.

In earlier releases, you had to define all sketches and features, save the forming tool as a part (`.SLDFTP`), then apply it to sheet metal.

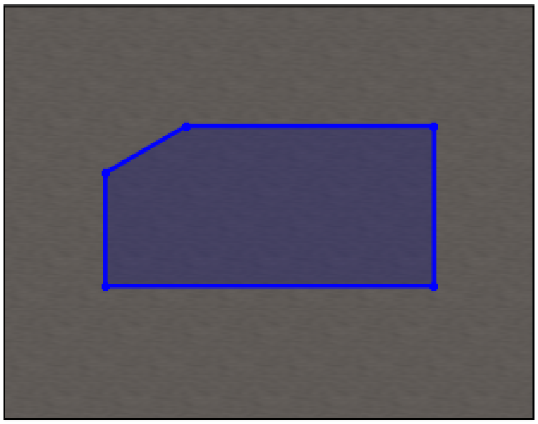
Using sketches to create forming tools is a faster way to apply forming tools to sheet metal parts. The **Stamp** tool allows more flexibility to experiment with different designs and parameters.

Using the Stamp Tool

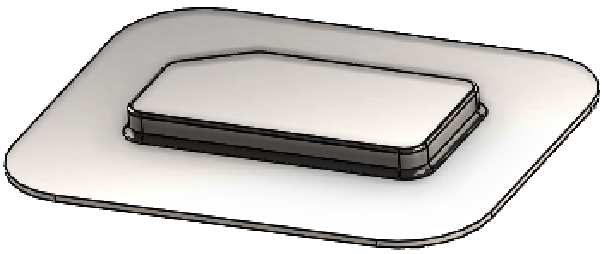
To use the stamp tool:

1. In a sheet metal part, click **Stamp**  (Sheet metal toolbar) or **Insert** > **Sheet Metal** > **Stamp**.

2. Sketch a closed profile sketch on the part for the stamp shape.



3. In the PropertyManager, specify options and click .



Stamp PropertyManager

To open this PropertyManager:

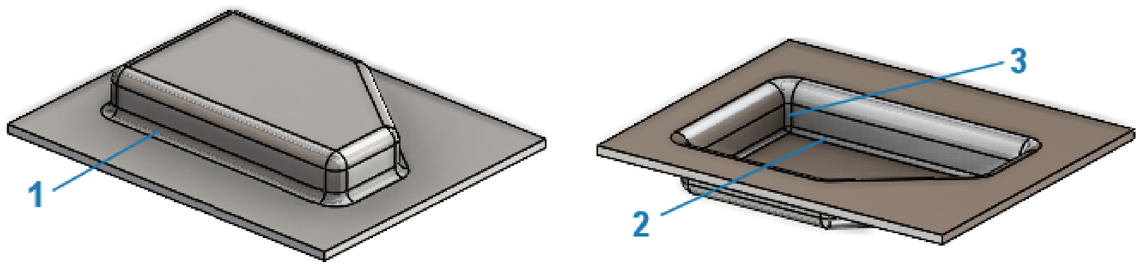
1. In a sheet metal part, click **Stamp** (Sheet metal toolbar) or **Insert** > **Sheet Metal** > **Stamp**.

Stamp Parameters

	Depth	Specifies the stamp depth from the top or bottom of the sheet metal face.
	Reverse Direction	Reverses the direction of the stamp.
	Draft Angle	Specifies the taper angle to apply to the stamp side faces.

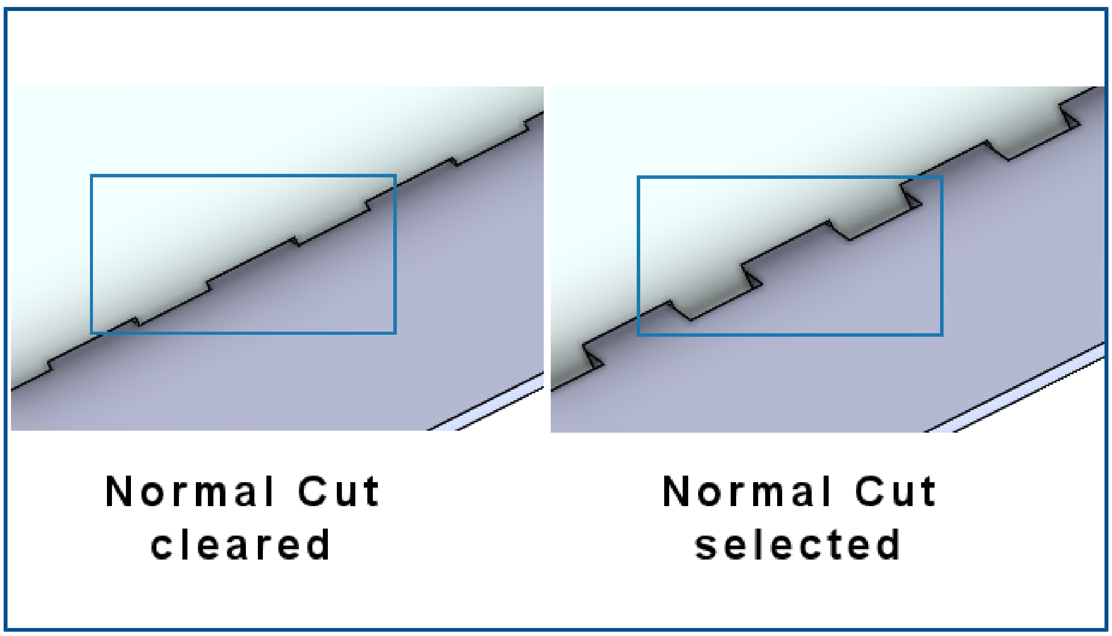
Fillet

If you specify a radius in the sketch before creating a stamp, the sketch radius is prioritized when creating the stamp.



1		**Die Radius (R1)**	Specifies the radius created by the die.
2		**Punch Radius (R2)**	Specifies the radius created by the punch.
3		**Punch Side Corner Radius**	Adds a corner punch radius. Specify the **Radius** created by the corner punch.

Normal Cut in Tab and Slot



When you use the **Tab and Slot** tool, you can specify that the slot is normal to the sheet even if the tab is at an angle to the slot. Slots that are normal to are essential in the manufacturing process.

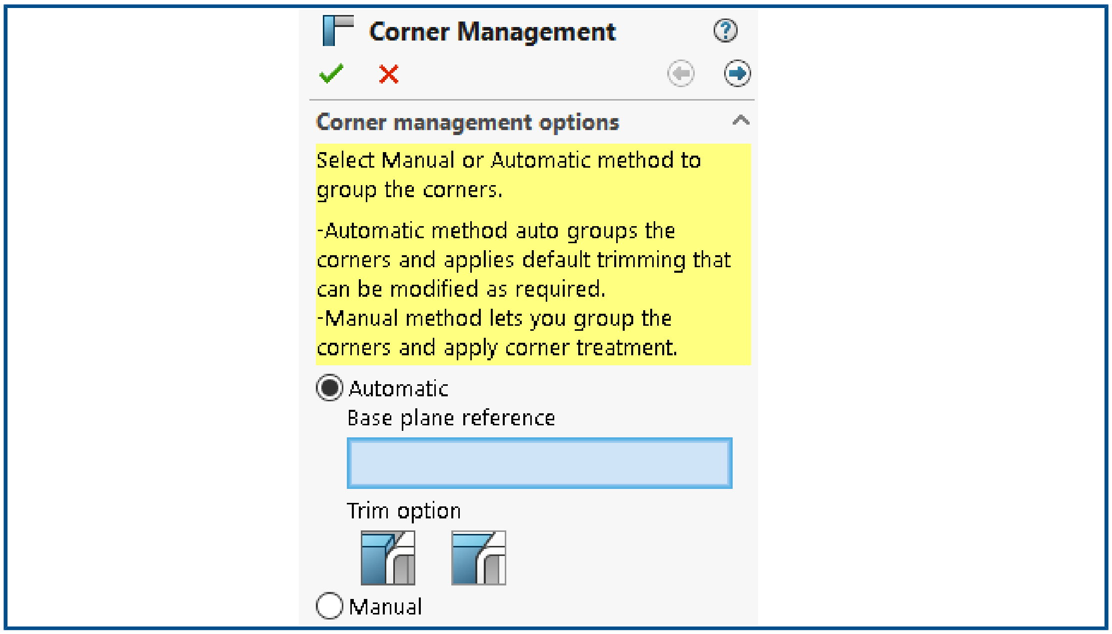
In the Tab and Slot PropertyManager, under **Slot**, select **Normal Cut**.

10

Structure System and Weldments

This chapter includes the following topics:

Corner Management



You can apply corner treatments manually or automatically.

To open the Corner Management PropertyManager:

1. Open a part and click **Structure System** > **Primary Member**.
2. Create primary members and exit the structure system mode.
3. In the PropertyManager, specify an option:
   - **Automatic**. Groups similar corners and applies the corner treatment.
   - **Manual**. Lets you group similar corners and apply the corner treatment.
4. Select **Automatic**.

> SOLIDWORKS selects a plane that determines the trim order of members. You can then modify the base plane reference, groups, and corner treatment, if required.

5. Specify a **Trim option**.
6. Click **Next** to continue with the corner treatment.

Two Member PropertyManager

The user interface of the Two Member PropertyManager is enhanced.

Enhancements include:

- Changes to trim types and trim options under **Corner Treatment**. You can select one of the following trim types:

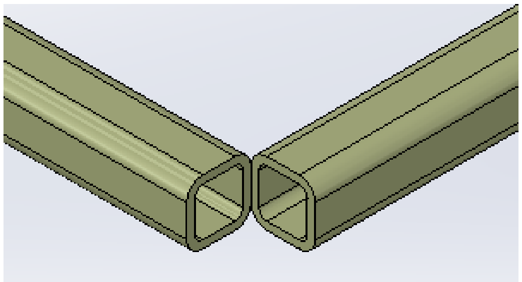
Icon	Trim type	Trim options
	End Butt1	**Planar Trim** or **Body Trim**
	End Butt2	**Planar Trim** or **Body Trim**
	Miter Trim	
	Open Corner	**First contact planar trim** or **Full contact planar trim**

- You can use the **End Butt1** and **End Butt2** trim options for swapping. Previously, you could swap the tool and body to trim using the arrows.

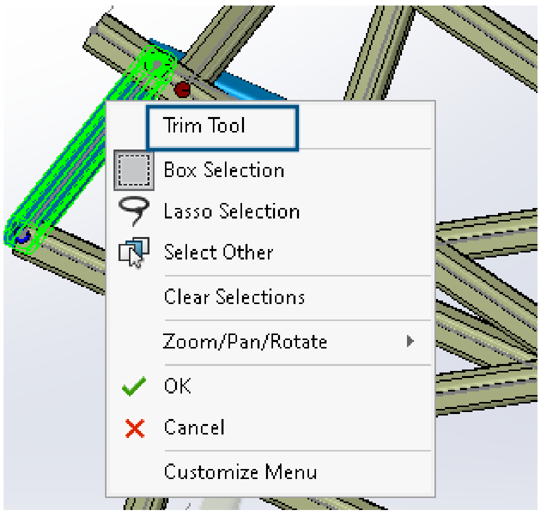
- Updated icons:

Icon	Trim Option
	Planar Trim
	Body Trim
	Miter Trim

- **Open corner** . Trims both members and creates an open corner.



- The **Trim Tool** shortcut menu is available in the graphics area. It lets you swap the member to trim.

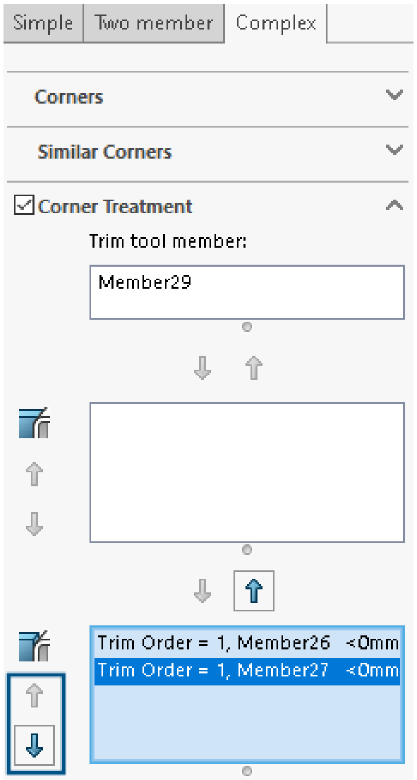


- In the PropertyManager, for **Trim Tool**, you can select **Automatic** or **User Defined**. The **User Defined** option lets you select a face or a plane to trim.

Complex Corner PropertyManager

The user interface of the Complex Corner PropertyManager is enhanced.

You can use **Trim order** for **Planar Trim**. Previously, you could use it only for **Body Trim**.



Editing the Corner Management Options

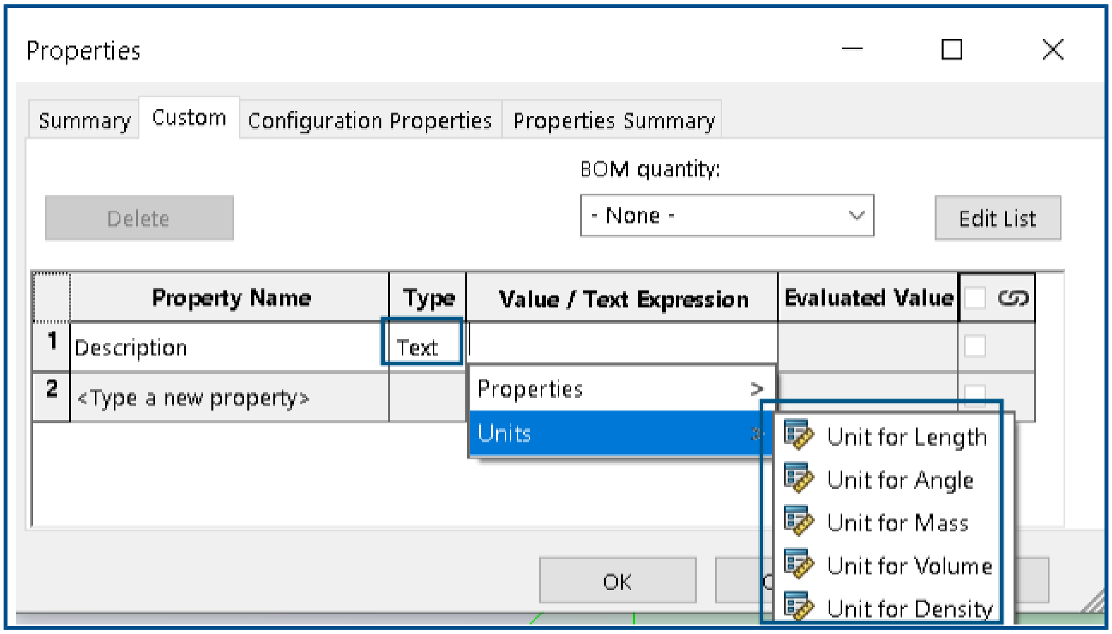
You can modify the corner treatment.

To edit the corner management options:

1. In the FeatureManager designTree, right-click **Corner Management** and click **Edit Feature**.
2. In the PropertyManager, click **Back** .
3. Click **Reset all corners** to clear all corner management settings.

> If you edit the structure system and add new corners, the corner management settings apply to the new corners.

Displaying Units in File Properties



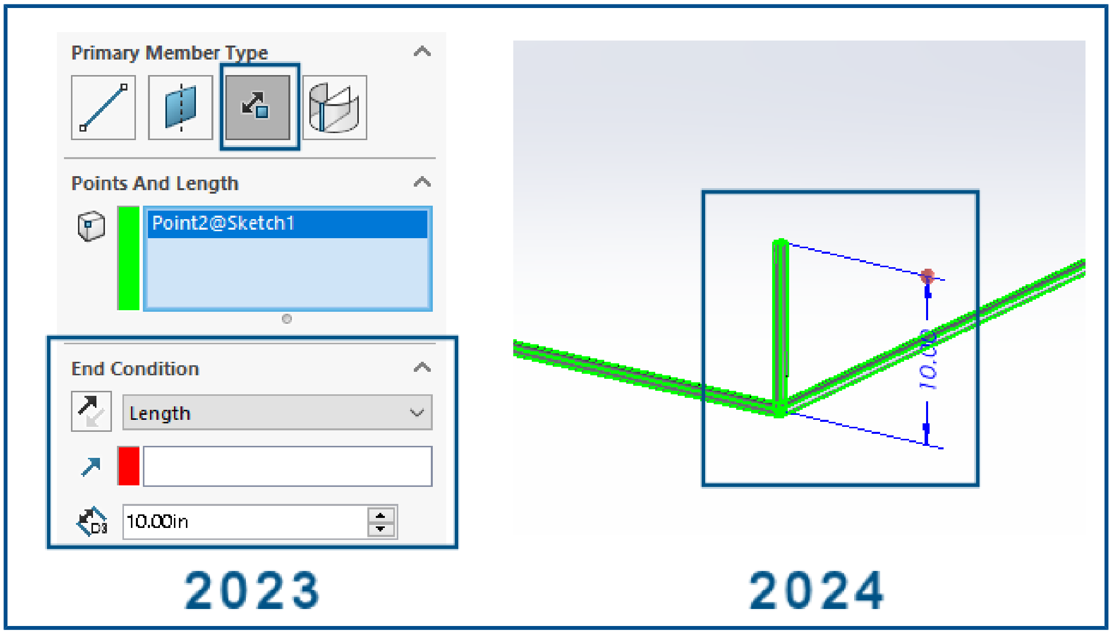
You can capture and display the units for the **Text** type of file properties.

To display units in file properties:

1. Click **Properties** (Standard toolbar).
2. In the Properties dialog box, on the Custom and Configuration Properties tabs, select a property name.
3. For **Type**, select **Text**.
4. Click in **Value/Text Expression**.
5. From the **Properties** flyout, select a property to display the evaluated value.
6. From the **Units** flyout, select a unit.

In previous versions, you could not capture the units for file properties.

Structure System



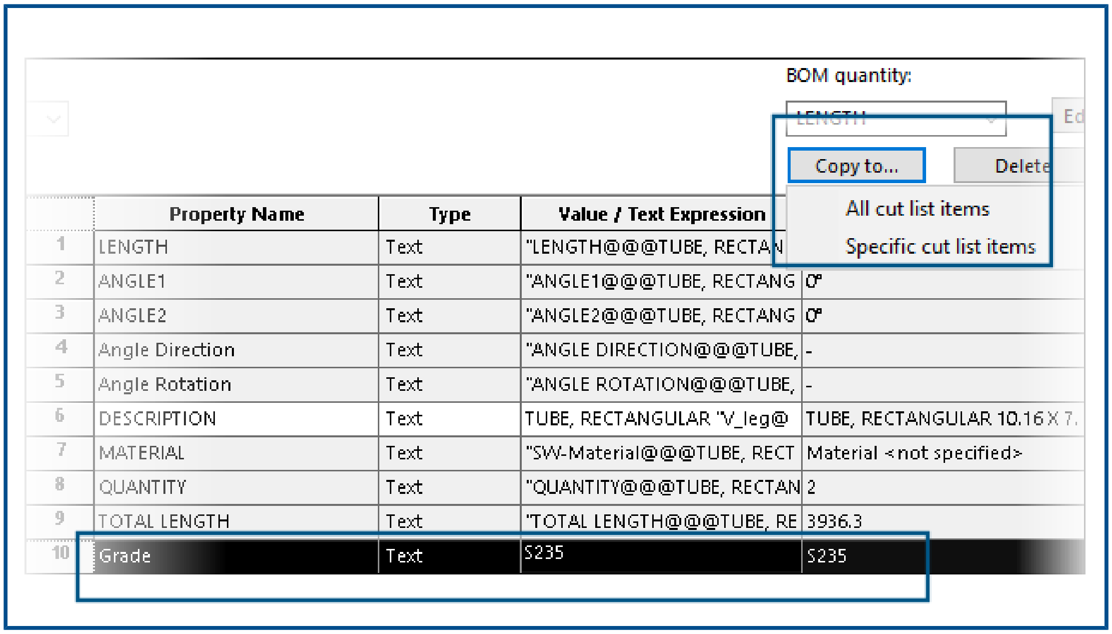
Structure system has improved usability in the graphics area and PropertyManager.

- When editing the structure system in the graphics area, you can change the length of the point length member.

  To change the length, double-click the member and click the dimensions. Previously, you had to edit the length of the point length member from the Primary Member PropertyManager.

- You can use corner management for profiles of less than 2mm.

Copying Cut List Properties to Cut List Items (2024 SP1)



You can create cut list properties and copy them to other cut list items.

To copy cut list properties to cut list items:

1. Open a part.
2. In the FeatureManager design tree, right-click a cut list item and select **Properties**.
3. In the Cut-List Properties dialog box, on the Cut List Summary tab, create a cut list property.
4. Select the property, click **Copy to**, and select one of the following:

All cut list items	Copies the selected property to all cut list items.
Specific cut list items	Copies the selected property to specific cut list items.

Copy to is available for user-defined properties only for files that use a new architecture.

Copy to copies the property of a cut list item to:

- All or specific cut list items that are available in the active configuration.
- The same cut list items that are available in the remaining configurations.

Copy Property to Cut List Items Dialog Box

You can use this dialog box to copy a cut list property to specific cut list items.

To access this dialog box, in the Cut-List Properties dialog box, on the Cut List Summary tab, click **Copy to** > **Specific cut list items**.

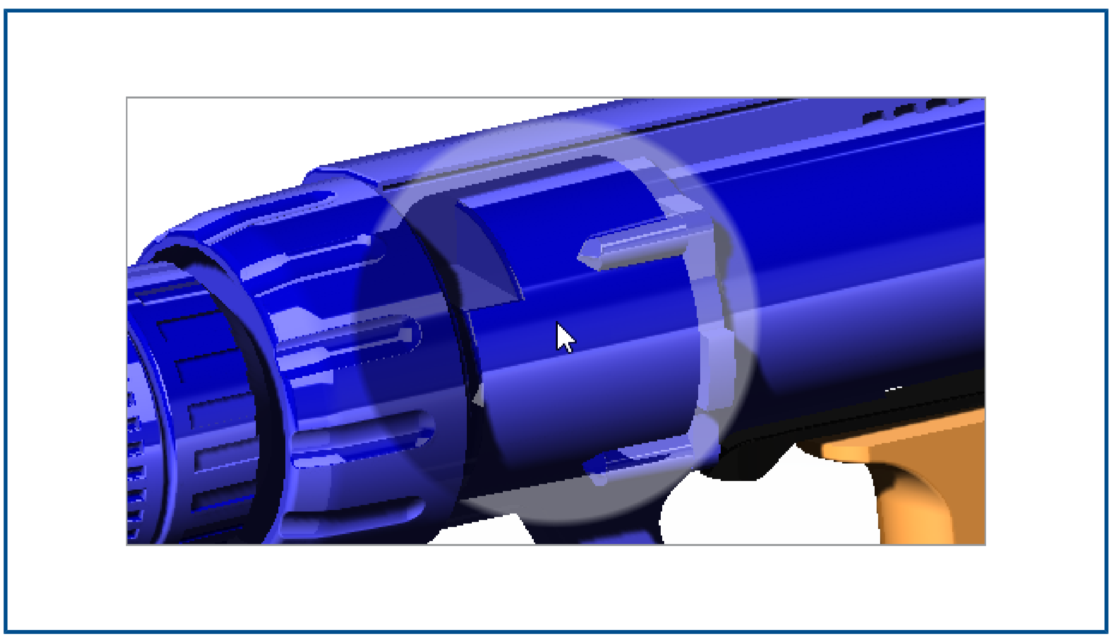
Option	Description
Select All	Selects all cut list items
Reset Selection	Resets the selection
OK	Copies the cut list property to the selected cut list items

11

Assemblies

This chapter includes the following topics:

Changing the Transparency of the SpeedPak Graphics Circle (2024 SP3)

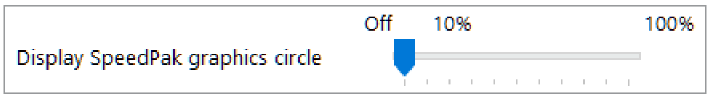


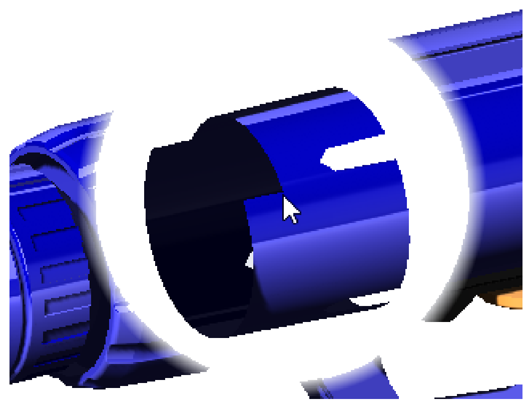
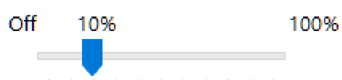
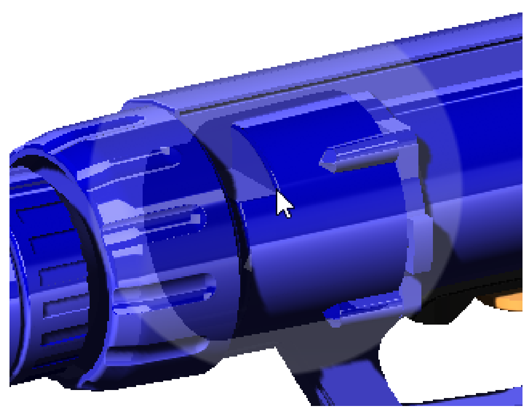
You can use the **Display SpeedPak graphics circle** slider to change the transparency of the SpeedPak circle.

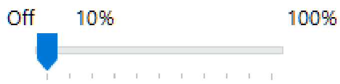
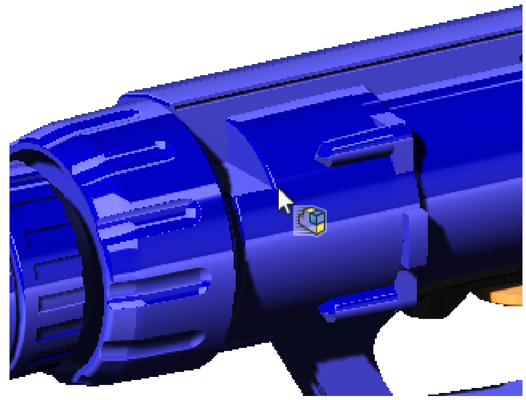
When the slider is at **100%**, the graphics are transparent. When the slider is **Off**, the SpeedPak graphics circle does not display and the pointer changes to an arrow with a SpeedPak image, .

To change the transparency of the SpeedPak graphics circle:

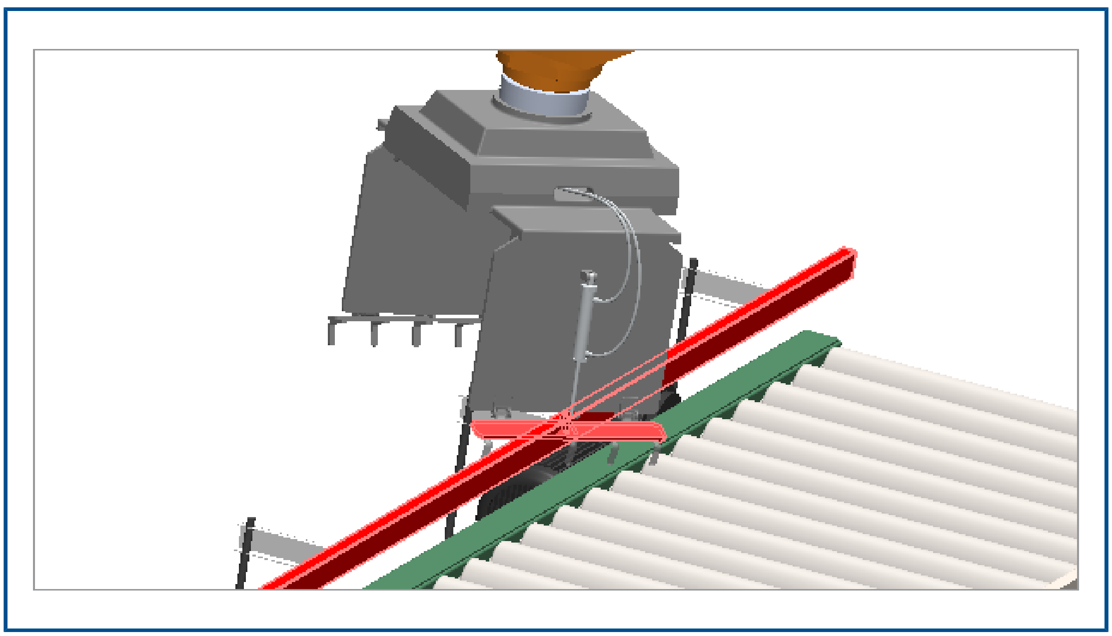
1. Click **Tools** > **Options** > **System Options** > **Display**.
2. For **Display SpeedPak graphics circle**, move the slider to change the transparency.



Slider	SpeedPak Graphics Circle
	
	

Slider	SpeedPak Graphics Circle
	

Detecting Interference between Surface Bodies (2024 SP3)



You can use interference detection between surface bodies for assemblies and multibody parts.

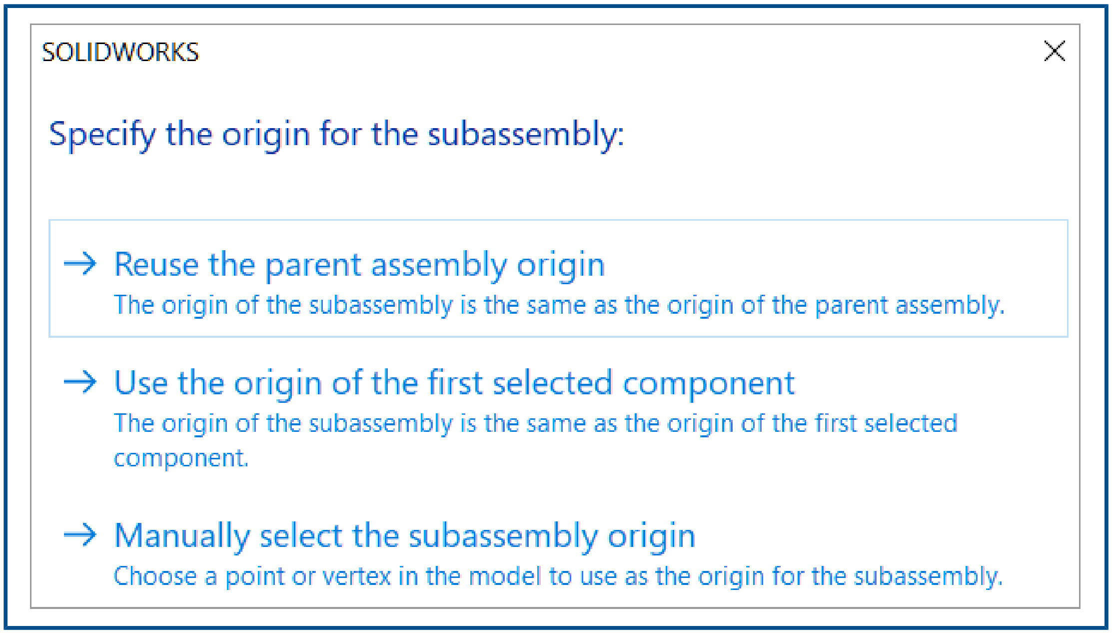
Benefits: You can find and fix interference issues for surface bodies.

To detect interference between surface bodies:

1. Open a model or multibody part that has an interference between surface bodies.
2. Click **Tools** > **Evaluate** > **Interference Detection**.
3. In the PropertyManager, under **Options**, click **Include surface bodies**.
4. Under **Selected Components**, click **Calculate**.
5. Under **Results**, scroll to the end for the surface body results.

When you select the surface interference, the intersecting faces appear in red in the graphics area.

Selecting an Origin for a New Subassembly (2024 SP2)



You can select an origin when creating a subassembly.

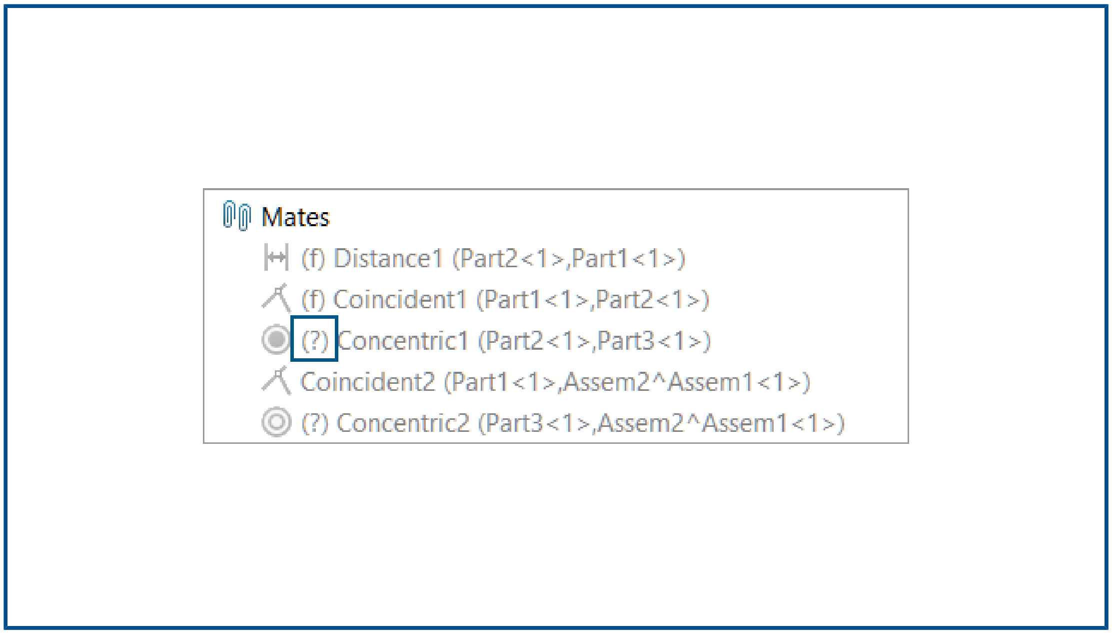
Origin options:

Origin of parent assembly	Uses the origin of the parent assembly as the origin of the subassembly.
Origin of the first selected component	Uses the origin of the first selected component as the origin of the subassembly.
Point or vertex	Uses a point or a vertex as the origin of the subassembly.

To select an origin for a new subassembly:

1. Open a model and select a component.
2. Right-click the selected component and click **Form New Subassembly**.
3. In the dialog box, select an option for the origin of the subassembly.

Unsolved Prefix Displays for Suppressed Mates (2024 SP2)



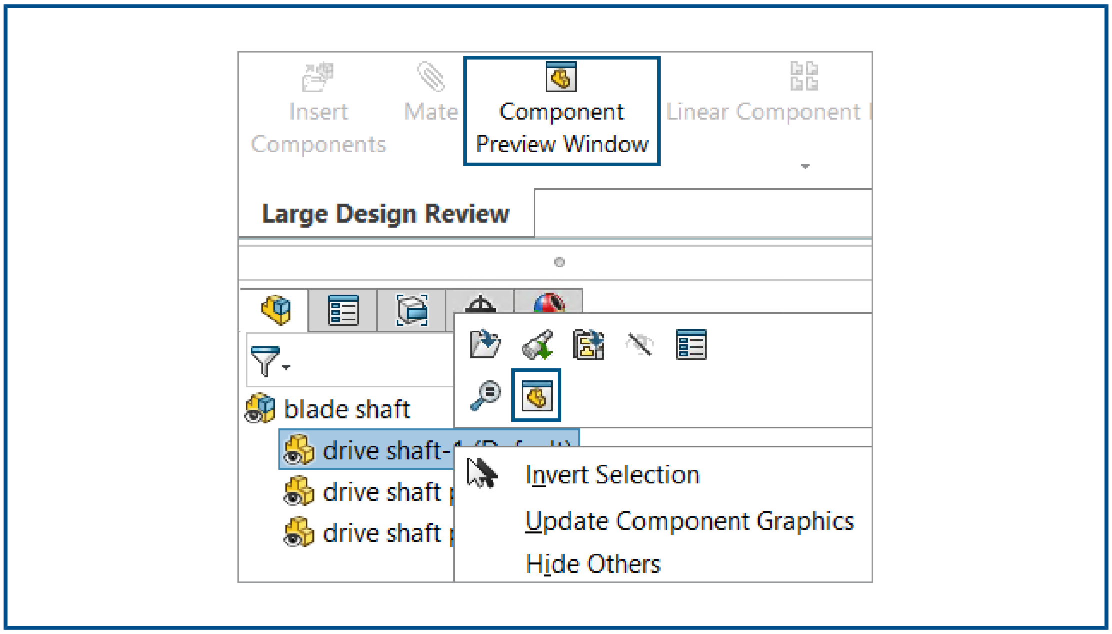
In a model, the unsolved prefix **(?)** displays in the mate name when a suppressed mate has a missing reference.

To view the unsolved prefix:

1. Open a model that has a suppressed mate with a missing reference.
2. In the FeatureManager design tree, expand the Mate folder.

   The unsolved prefix **(?)** shows in the mate name.

Component Preview Window Available in Large Design Review (2024 SP2)

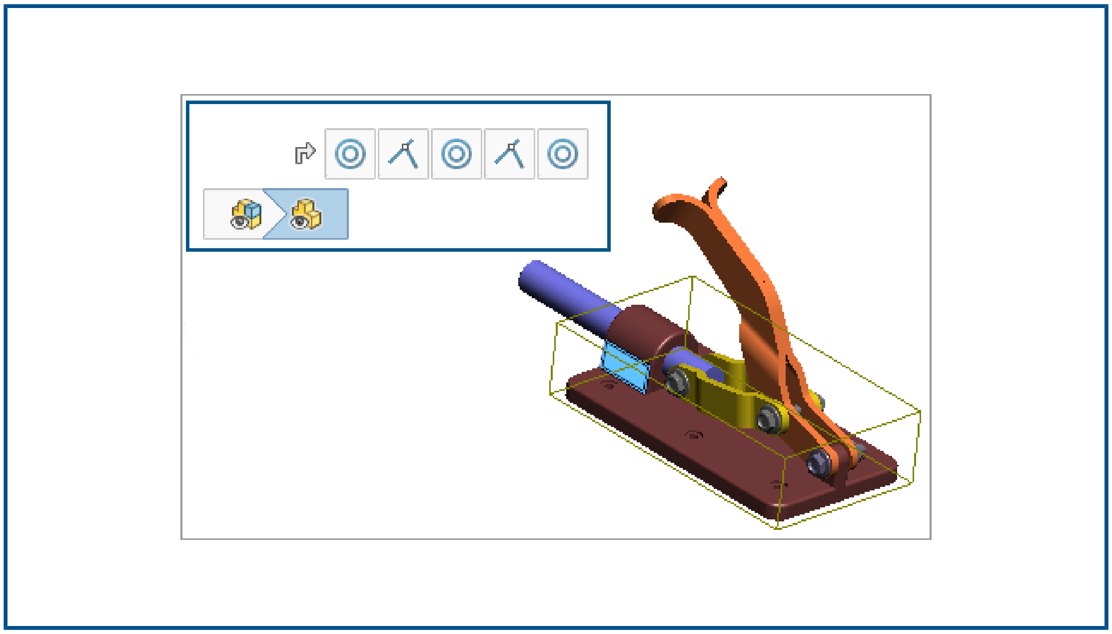


You can use the Component Preview window when you open an assembly in Large Design Review mode.

To open the Component Preview Window:

1. Open a model in Large Design Review mode.
2. Right-click a component and click **Component Preview Window** .

Selection Breadcrumbs Available in Large Design Review (2024 SP1)



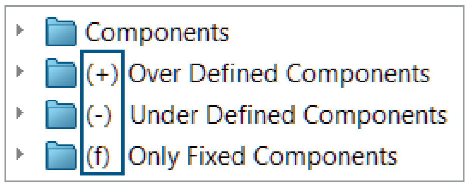
You can use breadcrumbs when you open a model in Large Design Review mode. With **Edit Assembly** selected, mates for the selected item show in the breadcrumbs.

To use selection breadcrumbs:

1. Enable breadcrumbs by clicking **Tools** > **Options** > **System Options** > **Display** and selecting **Show breadcrumbs on selection**.
2. Open a model in Large Design Review.
3. In the graphics area or in the FeatureManager design tree, select a component.

   The breadcrumbs display in the upper left corner.

Folder Prefixes (2024 SP1)



In a model, prefixes show in a folder name when the folder contains over defined components, under defined components, and only fixed components.

Folder prefixes:

(+)	Contains at least one over defined component.
(−)	Contains at least one under-defined component.
(f)	Contains only fixed components. If a folder contains a component that is not fixed, the fixed prefix does not show in the folder name.

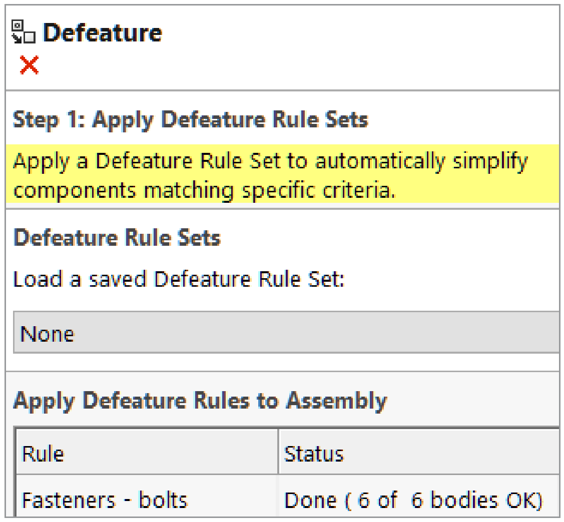
Prefixes do not show for folders that contain only well-defined components.

To view a folder prefix:

1. Open a model that has an under-defined component.
2. In the FeatureManager design tree, right-click an under-defined component and click **Add to New Folder**.
3. Enter a folder name and click **Enter**.

   The under-defined prefix shows in the folder name.

Defeature Rule Sets



Using the Defeature Silhouette method, you can create a set of rules to simplify the components in a model. You can specify criteria for component selection, defeature method, and a defeature orientation. You can enclose the components in one body and propagate visual properties.

For example, you can create a rule to simplify fasteners as cyclinders when the filename for a fastener contains bolt, nut, or washer.

You can save the rule set to use with other models. You can specify a file location for saved rule sets. You can use a rule set with a defeature group to defeature a model.

Specifying a File Location for Defeature Rule Sets

You can save defeature rule sets and log files to a designated folder.

You can use a saved defeature rule set with a different model. A log file shows the outcome of applying a defeature rule set to a model. The log file includes a list of components with a status of **OK** or **Failed**.

To specify a file location for defeature rule sets:

1. Click **Tools** > **Options** > **System Options** > **File Locations**.
2. Under **Show folders for**, select **Defeature Rule Sets**.
3. Click **Add** and select a location.

Creating Defeature Rule Sets

You can use a defeature rule set to simplify your model.

To create a defeature rule set:

1. Open a model, and click **Defeature** (Tools toolbar) or **Tools** > **Defeature**.

2. In the PropertyManager, select **Silhouette**.
3. Click **Next**.
4. Under **Apply Defeature Rules to Assembly**, click **Edit Rules**.
5. In the Defeature Rules Editor dialog box, under **Name**, enter a name.

Defeature Rules Editor New rule set

List of Rules:

	Name	Selection Criteria	Defeature Type	Enclose in one ...	Defeature Orie...	Visual prop...
1	*Click to a...*	Define selection cr...	*Choose type*	*Off*	*Choose orientat...*	*Propagate*

6. Under **Selection Criteria**, click **Define selection criteria**.
7. In the Advanced Component Selection dialog box, select search criteria.

   For example, search for fasteners where the filename contains bolt.

Advanced Component Selection

Define Search Criteria

Delete Clear All

	And/Or	Category1	Category2	Condition	Value
1		File Type		=	Fastener
2	And	Document name -- SW Sp		contains	bolt

8. In the Defeature Rules Editor dialog box, specify the **Defeature Type** and **Defeature Orientation**.

> For each rule, **Name**, **Selection Criteria**, **Defeature Type**, and **Defeature Orientation** must be populated.

Defeature Rules Editor New rule set

List of Rules:

	Name	Selection Criteria	Defeature Type	Enclose in one ...	Defeature Orie...	Visual prop...
1	Fastener	File Type = "Faste...	Cylinder	Off	Automatic	Propagate

9. Optional: Click **Save** to save the rules as a defeature rule set, `.slddrs`.
10. In the Defeature Rules Editor dialog box, click **OK** to return to the PropertyManager.

    Under **Apply Defeature Rules to Assembly**, the rule status is **Pending**.

Apply Defeature Rules to Assembly

Rule	Status
Fasteners - bolt	Pending

11. Click **Apply**.

    After SOLIDWORKS® applies the rule to the model, the status changes to **Done (x of y bodies OK)**.

Apply Defeature Rules to Assembly

Rule	Status
Fasteners - bolts	Done (6 of 6 bodies OK)

12. Optional: Click **Save log** to save the results to a log file.

    When you open the log file, you see a list of the defeatured components and the defeatured status.

```
Log for defeature silhouette rules applied to C:\Lifts\LIFT.SLDASM

### Rule: Fasteners - bolts ###
Hex@12mm NOM x 70mm LG, METRIC SHOULDER BOLT-3@4545: OK
Hex@12mm NOM x 70mm LG, METRIC SHOULDER BOLT-2@4545: OK
Hex@12mm NOM x 70mm LG, METRIC SHOULDER BOLT-1@4545: OK
Hex@12mm NOM x 70mm LG, METRIC SHOULDER BOLT-2@4568: OK
Hex@12mm NOM x 70mm LG, METRIC SHOULDER BOLT-3@4568: OK
Hex@12mm NOM x 70mm LG, METRIC SHOULDER BOLT-1@4568: OK
Rule complete: 6 OK, 0 Failed
```

Defeature - Apply Defeature Rule Sets PropertyManager

In assemblies, you can create a defeature rule set to simplify a model.

You can use a rule set with a defeature group to defeature a model.

To open the Defeature - Apply Defeature Rule Sets PropertyManager:

1. Open a model and click **Defeature** (Tools toolbar) or **Tools** > **Defeature**.
2. In the PropertyManager, select **Silhouette** .
3. Click **Next** until the **Apply Defeature Rule Sets** page appears.

Defeature Rule Sets

Load a saved Defeature Rule Set	Specifies the rule set to load. **None** displays when there are no loaded rule sets. Saved rule sets display in the list. To specify the file location for the saved rule set, click **Tools** > **Options** > **System Options** > **File Locations**. Under **Show folders for**, select **Defeature Rule Sets**. Click **Add** to specify a location.

Apply Defeature Rules to Assembly

Rule	Lists the rules.

Status	Displays the results of applying the rule: • **Pending**. Displays when the rule is not applied or when an existing rule is modified but not reapplied. • **Done (x of y bodies OK)**. After applying the rule, displays the number of components processed, **x**, and the number of components, **y** that meet the criteria.
Apply	Applies all rules to the model in the order that the rules are listed. The defeatured geometry generates and a preview displays in the graphics area. After a rule is applied to a component, no other rules are applied to that component. After saving the model as a part, the defeature components display in the FeatureManager design tree. The log file includes a list of components with a status of **OK** where components are defeatured or **Failed** where components are not defeatured. Rules apply to part level components. Rules do not apply to subassemblies.
Clear	Removes all the rules and deletes the simplified geometry applied to the model.
Edit Rules	Opens the Defeature Rules Editor dialog box.
Save log	Saves the log file.

Defeature Rules Editor Dialog Box

You can create a set of rules to automatically simplify the components in a model.

To open the Defeature Rules Editor dialog box:

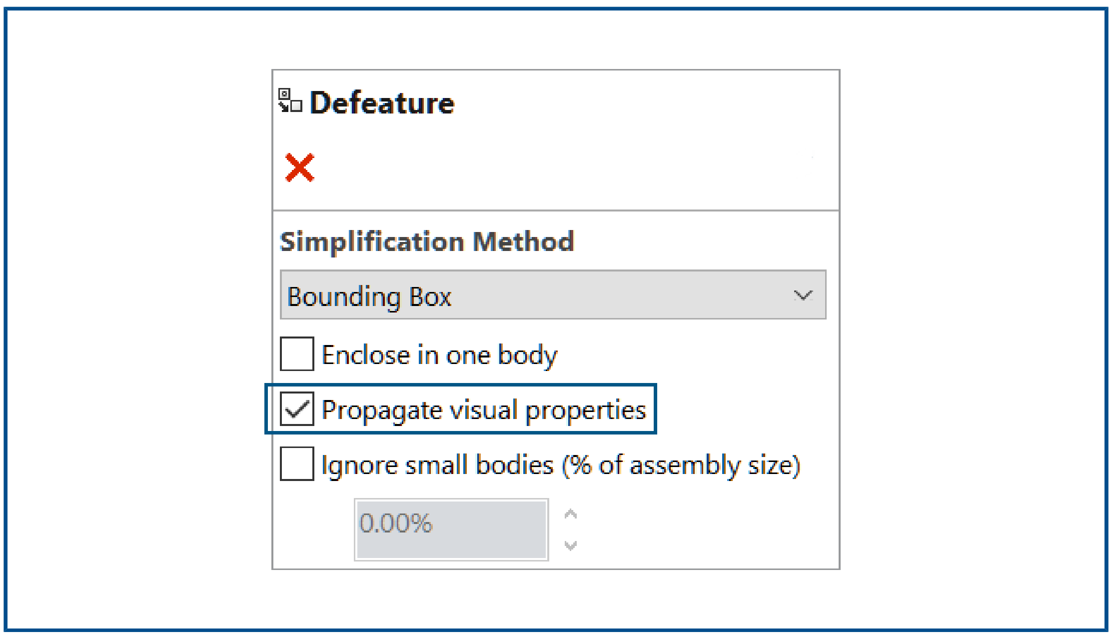
1. Open a model and click **Defeature** (Tools toolbar) or **Tools** > **Defeature**.
2. In the PropertyManager, select **Silhouette**.
3. Click **Next** until the Apply Defeature Rule Sets page appears.
4. Under **Apply Defeature Rules to Assembly**, click **Edit Rules**.

	New	Creates a new rule set.
	Open	Opens an existing rule set.
	Save	Saves the rule set in a Defeature Rule Set file, `.slddrs`.

Name	Specifies a name for the rule set.
Selection Criteria	Displays the selection criteria. For a new rule, click **Define selection criteria** to open the Advanced Component Selection dialog box where you define the selection rules. To modify a rule, click the selection criteria for the rule. Under **Rule Definition**, click **Selection Criteria**. In the Advanced Component Selection dialog box, the following functionality is not available when you open the dialog box from the Defeature PropertyManager: • Manage Searches tab • **Name of Search** • **Apply**
Defeature Type	Specifies a simplification method: **Bounding Box** – Creates a cuboid bounding box. **Cylinder** – Creates a cylinder derived from the dimensions of a cuboid bounding box. **Polygon Outline** – Creates an extruded polygon that fits around the outline of the selected bodies and components. **Tight Fit Outline** – Creates an extruded body by using the outlines of the selected bodies and components. **None (Copy Geometry)** – Creates an exact copy of the selected bodies and components.
Enclose in one body	Creates a single body that includes the specified components. • **Off** • **Per part** • **Entire group** When you select **Cylinder** or **None (Copy Geometry)** for **Defeature Type**, **Enclose in one body** is set to **Off**.

Defeature Orientation	Specifies a defeature orientation: • **Automatic** • **Component XY** • **Component YZ** • **Component XZ** • **Global XY** • **Global YZ** • **Global XZ**
Visual properties	**Propagate**: Includes appearances and textures in the defeatured model. **Don't propagate**: Omits appearances and textures from the defeatured model.
Rule Definition	Displays the selected rule. Click **Selection Criteria** to modify the rule.

Propagating Visual Properties in Defeature Groups



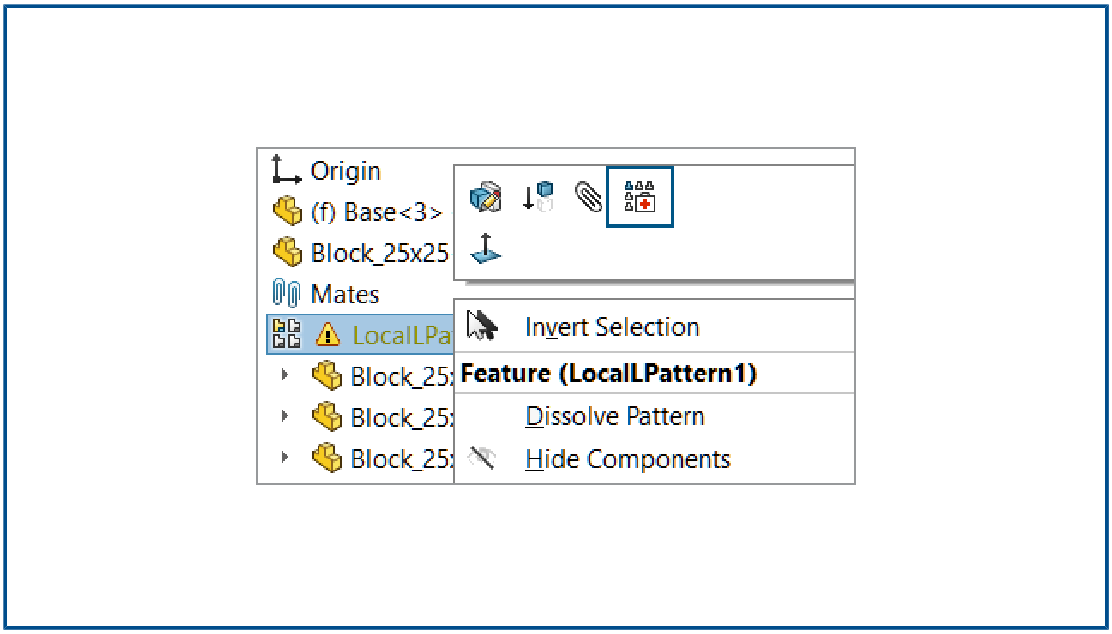
You can include appearances and textures in a defeatured group.

To propagate visual properties in defeature groups:

1. Open a model, and click **Defeature** (Tools toolbar) or **Tools** > **Defeature**.
2. In the PropertyManager, select **Silhouette**.
3. Click **Next** until the Defeature - Define Groups page appears.

4. Under **Simplification Method**, select **Propagate visual properties**.

Repairing Missing References in Linear or Circular Component Patterns



You can repair missing direction references in linear component patterns and circular component patterns.

For linear component patterns, SOLIDWORKS repairs the missing direction reference by selecting a reference on the component that is the same type and orientation, and is either the same location or the closest entity to the missing reference.

For circular component patterns, SOLIDWORKS repairs the missing direction reference by selecting a reference on the component that is the same entity and is coaxial with the missing axis. If there are multiple options for a replacement axis, SOLIDWORKS selects the closest to the missing axis.

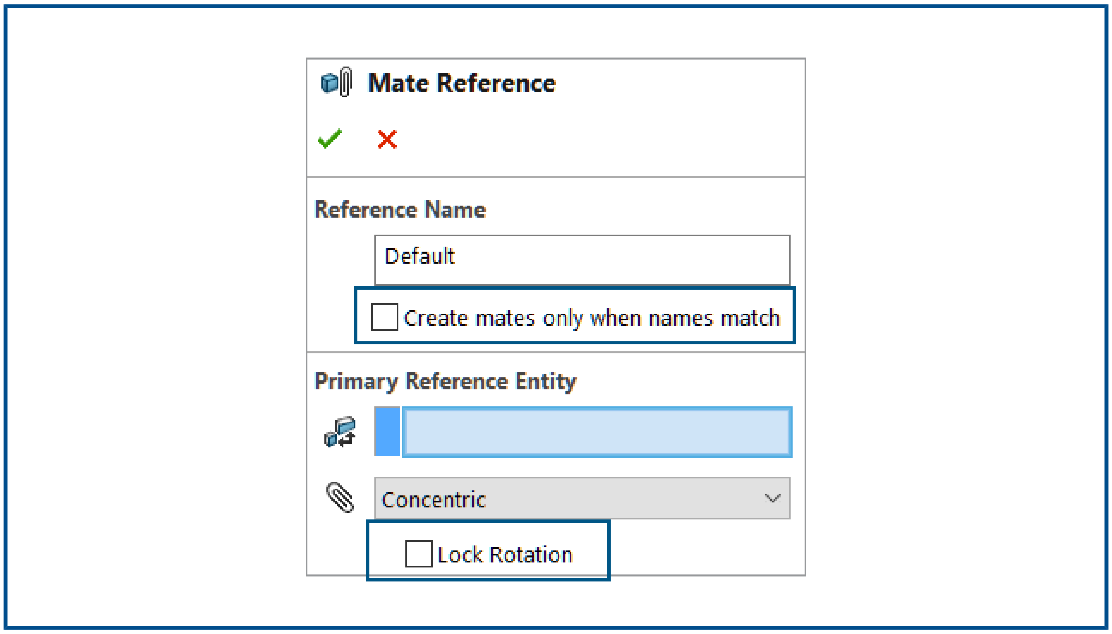
> You cannot use **Auto Repair** in Large Design Review mode.

To repair missing references in linear or circular component patterns:

1. Open a model that contains a linear component pattern or a circular component pattern with a missing direction reference.
2. Right-click the pattern and in the context toolbar, click **Auto Repair**.

   > If SOLIDWORKS cannot repair the error, you are prompted to repair the pattern manually.

Mate References



When creating mate references, you can select **Create mates only when names match** to create mate references only when the mate reference names are the same. The name match applies to primary, secondary, and tertiary reference entities.

To use **Create mates only when names match**, you must select this option on both components in the mate reference.

When more than one mate reference is available, the Select Mate Reference dialog box displays a list of mate references.

The dialog box can appear when using these workflows:

- Inserting a component.
- Dragging a component from the FeatureManager® design tree.
- Dragging a file from the File Explorer tab in the Task Pane.
- Dragging a file from the Design Library tab in the Task Pane.

In the Mate Reference PropertyManager, you can select **Lock Rotation** for **Concentric** mates.

To create mates only when the names match:

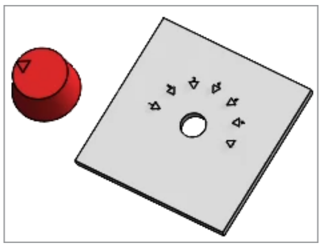
1. Open a model with a mate reference where the name of the mate reference is different for each component.
2. Open one of the components from the mate reference.
3. In the FeatureManager design tree for the component, under the **Mate References** folder, right-click a mate reference and click **Edit Definition**.
4. In the Mate Reference PropertyManager, under **Reference Name**, select **Create mates only when names match**.
5. Copy the **Reference Name** value to use later.

6. Open the other component in the mate reference and repeat the steps to enable **Create mates only when names match**.
7. For **Reference Name**, enter the name from the first component.
8. Close both components.
9. In a model, click **Insert** > **Reference Geometry** > **Mate Reference**.
10. Under References, select **Create mates only when names match**.
11. Select the two components to mate.

To select a mate reference in the Select Mate Reference dialog box:

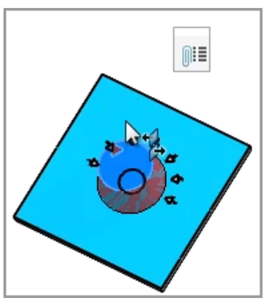
1. Open a model where multiple references are available between two components.

   In this example, you create a mate reference between a knob and a plate. The plate has several positions that you can select.

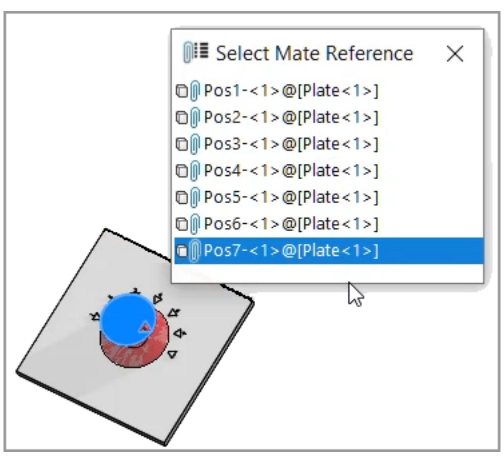
   

2. Drop the knob over the plate.

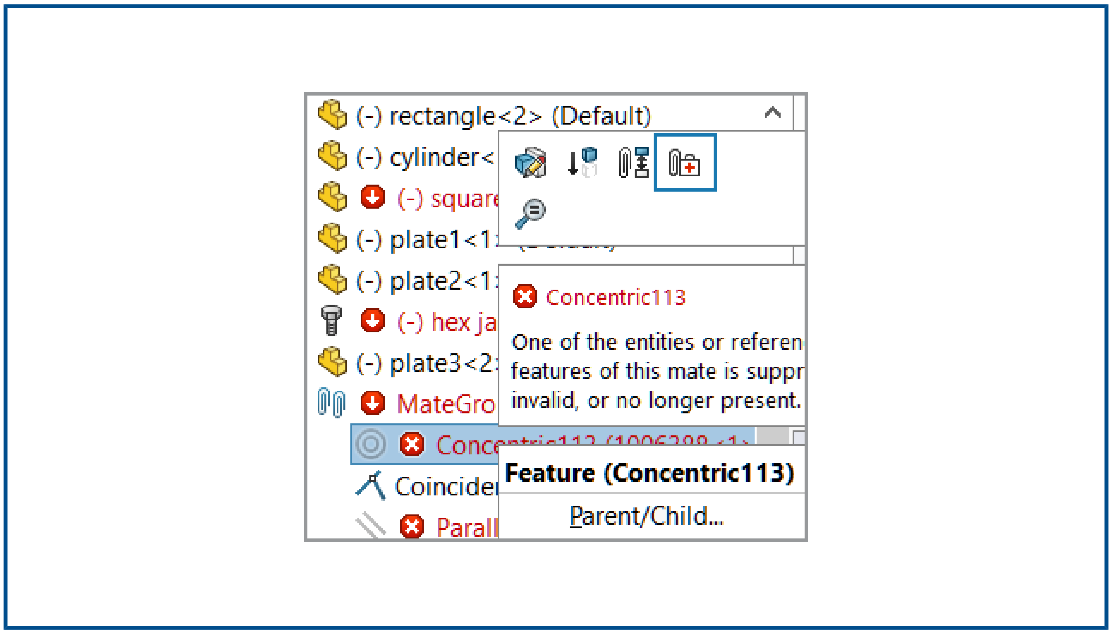
   Select Mate Reference appears when the knob is over the plate.

3. In the Select Mate Reference dialog box, select a reference.

Auto-Repair for Missing Mate References



Improvements to **Auto Repair** for concentric and parallel mates add more criteria for identifying replacement entities.

For concentric mates, SOLIDWORKS repairs the missing reference by selecting a face on the same component that has a different diameter and the same axis position.

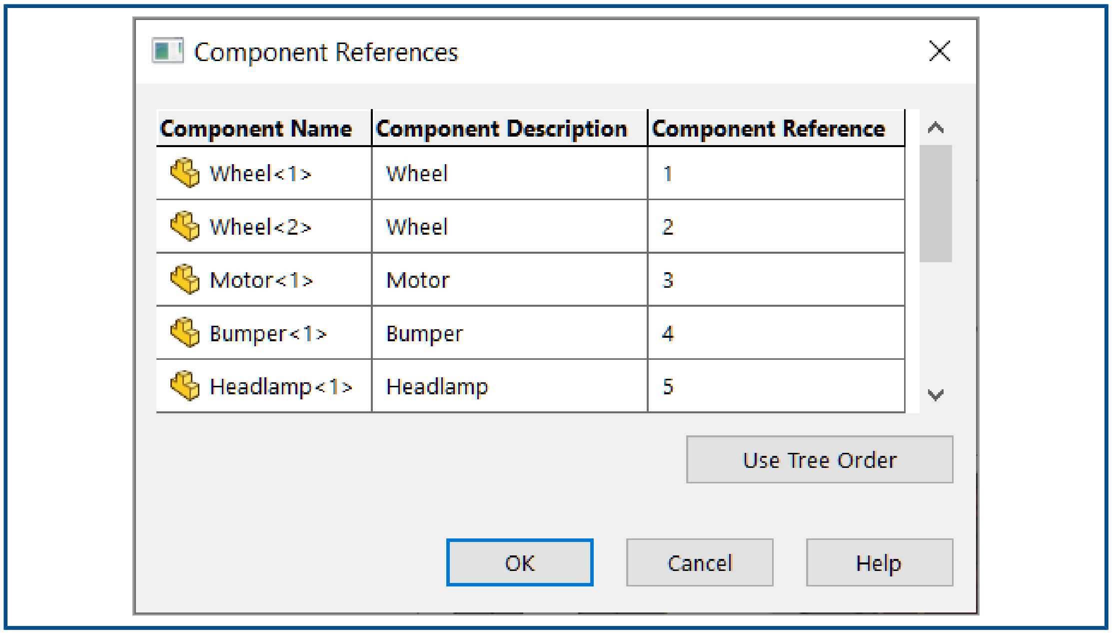
For parallel mates, SOLIDWORKS repairs the missing reference by selecting a reference on the same component that has a different position. For planar faces, the missing reference is repaired with a different planar face that has the same orientation. For plane references, the missing reference is repaired with a different plane that has the same orientation. If a matching plane is not available, SOLIDWORKS uses a planar face that has the same orientation to repair the missing plane reference.

To auto-repair missing mate references:

1. Open a model that contains a concentric mate error.
2. Right-click the mate and in the context toolbar for the mate, click **Auto Repair** .

> If SOLIDWORKS cannot repair the error, you are prompted to solve the mate manually.

Assigning Component References to Top-Level Components



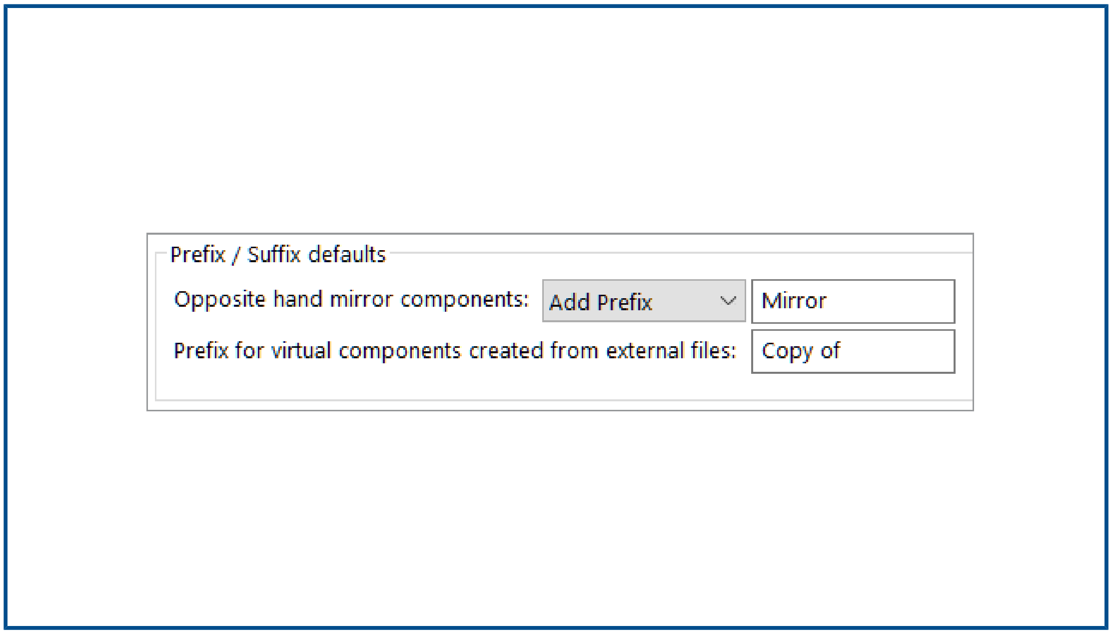
In the Component References dialog box, you can enter component references for all top-level components. You can use the tree order from the FeatureManager design tree as the component reference.

To assign component references to top-level components:

1. Open a model.
2. Right-click the assembly name in the FeatureManager design tree and click **Edit Component References**.
3. In the Component References dialog box, under **Component Reference**, enter a component reference for each component.

> To use the component order from the FeatureManager design tree, click **Use Tree Order**. Existing component references are overwritten.

Specifying a Prefix and Suffix for Components



You can use a system option to specify a default prefix and a default suffix for opposite-hand versions of mirrored components. You can also specify a default prefix for virtual components created from external files.

To specify a prefix andsuffix for components:

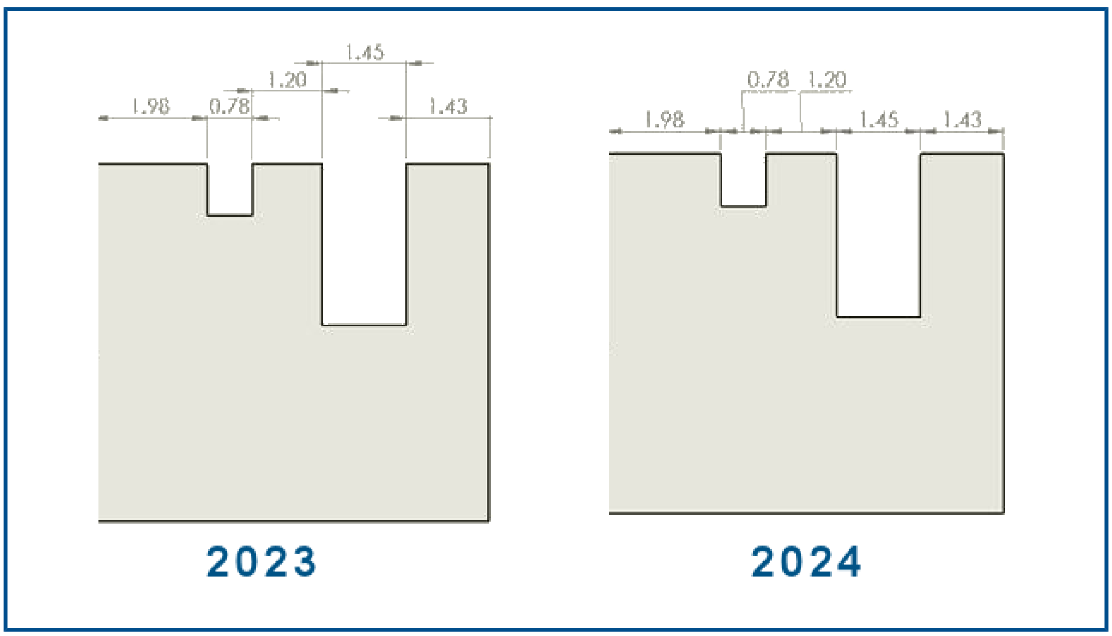
1. Click **Tools** > **Options** > **System Options** > **Assemblies**.
2. Under **Prefix / Suffix defaults**, specify options:
   a. For **Opposite hand mirror components**, select **Add Prefix** or **Add Suffix**, and enter text.
   b. For **Prefix for virtual components created from external files**, enter text.
3. Click **OK**.

12

Detailing and Drawings

This chapter includes the following topics:

Keeping Chain Dimensions Collinear



You can ensure that chain dimensions remain collinear even with limited space.

When dimension text and arrowheads overlap, you can select options for the best fit.

To keep chain dimensions collinear when dimension text overlaps:

1. Click **Tools** > **Options** > **Document Properties** > **Dimensions** > **Linear** > **Chain Dimension**.
2. Under **Collinearity Options**, select **Offset text automatically when space is limited**.

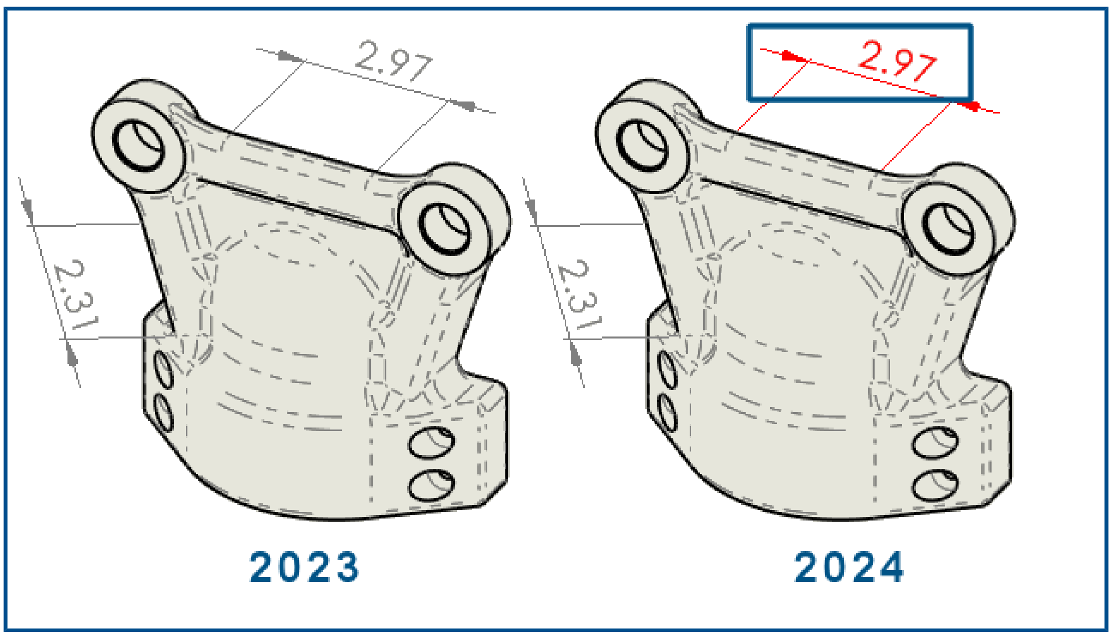
   For ISO and ANSI, this option is selected by default.

To keep chain dimensions collinear when arrowheads overlap:

1. Click **Tools** > **Options** > **Document Properties** > **Dimensions** > **Linear** > **Chain Dimension**.
2. Under **Collinearity Options**, select **When arrowhead overlaps substitute arrowhead termination automatically with:** and specify an option:
   - **Points**. Replaces arrowheads with points.
   - **Oblique Strokes**. Replaces arrowheads with oblique strokes.

   For ISO, this option is selected by default.

Overridden Dimensions



You can choose to automatically change the color of overridden dimensions.

Previously, you had to click every dimension and view its properties to see overrides.

You can:

- Change the color of overridden dimensions automatically.

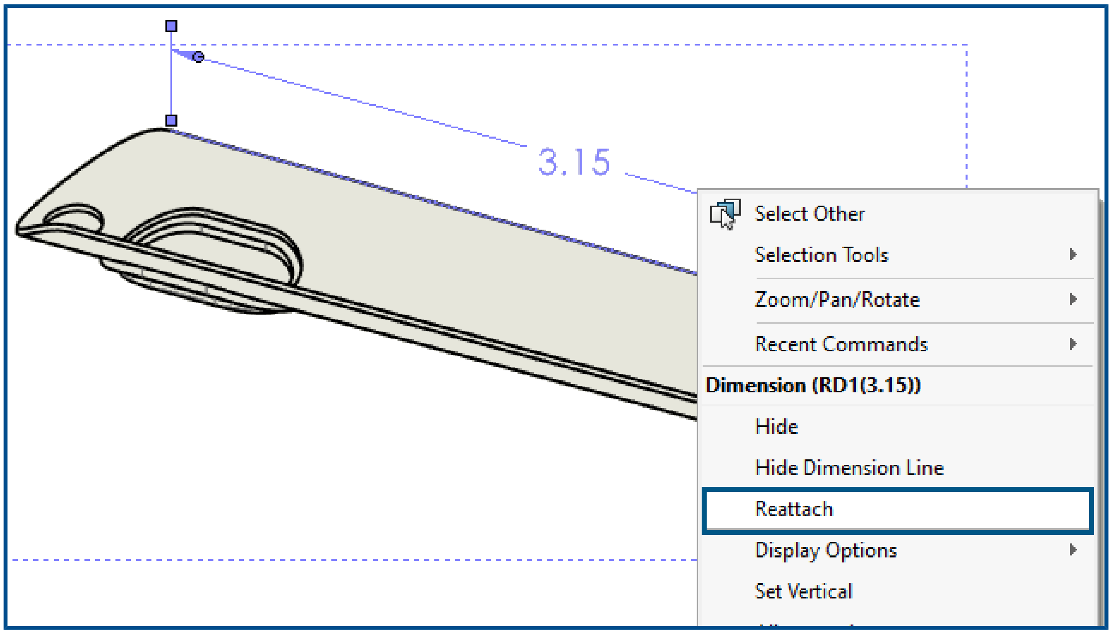
  To specify the color, click **Tools** > **Options** > **System Options** > **Colors**. Under **Color scheme settings**, edit the color for **Drawings, Overridden dimensions**.

  To display the color, click **Tools** > **Options** > **Document Properties** > **Dimensions** and select **Highlight overridden dimensions in a different color**.

- Restore the overridden dimension values to their original values.

  Right-click the overridden dimension and select **Restore Original Value**.

Reattaching Dangling Dimensions



You can reattach dangling dimensions in a way that makes the process more reliable. You can reattach dimensions that are not dangling the same way.

The feature does not support:

- Imported dimensions
- DimXpert dimensions
- Chain dimensions
- Symmetric linear diameter dimensions
- Path length dimensions

To reattach dangling dimensions:

1. Right-click the dangling dimension and click **Reattach**.

   SOLIDWORKS® highlights the dangling point with an X on the first extension line.

2. Select a point on the model to reattach the dangling point to.

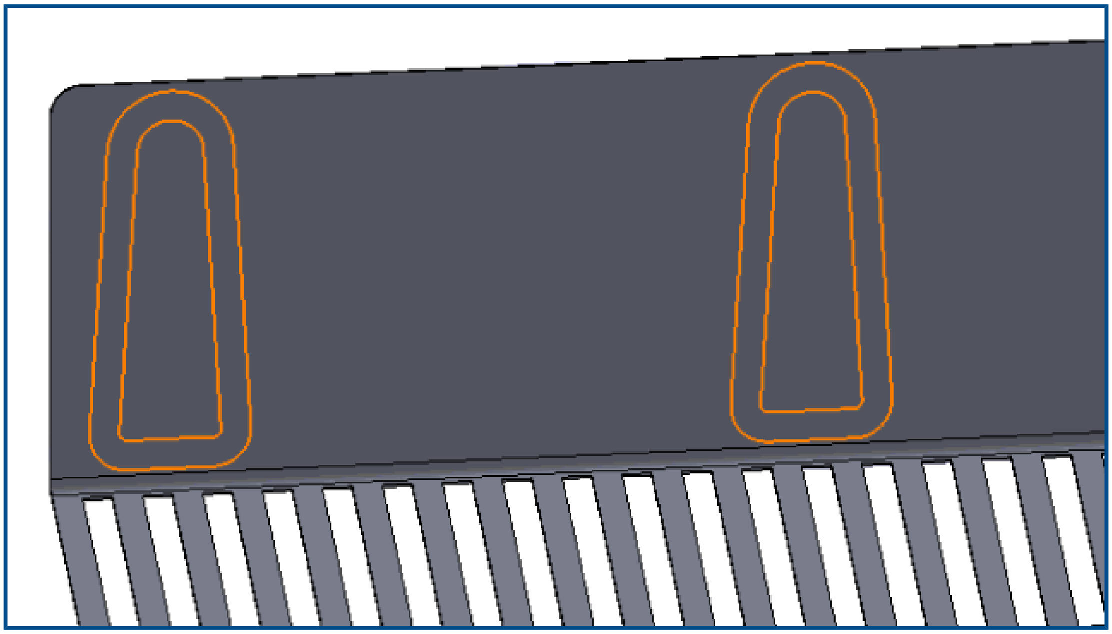
   The dangling point reattaches to the new selection.

   SOLIDWORKS highlights the dangling point with an X on the next extension line.

3. Select a point on the model to reattach the dangling point to.

   The dangling point reattaches to the new selection.

Excluding Hidden Sketches from Flat Pattern DXF Files

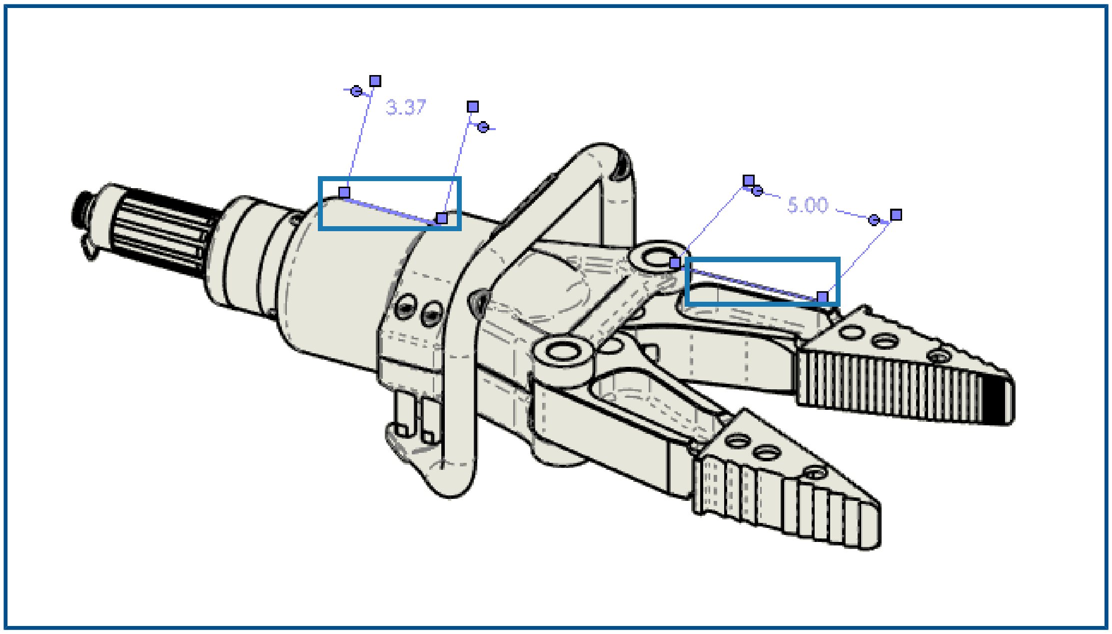


In the DXF / DWG Output PropertyManager, when you export a sheet metal flat pattern as a `.dxf` file, you can exclude hidden sketches.

To exclude hidden sketches from flat pattern DXF files:

1. In the PropertyManager:
   a. Under **Export**, select **Sheet metal**.
   b. Under **Entities to Export**, select **Sketches** and under **Sketches**, select **Exclude hidden sketches**.

Highlighting Referenced Elements



When you select a dimension, you can highlight the associated elements as well.

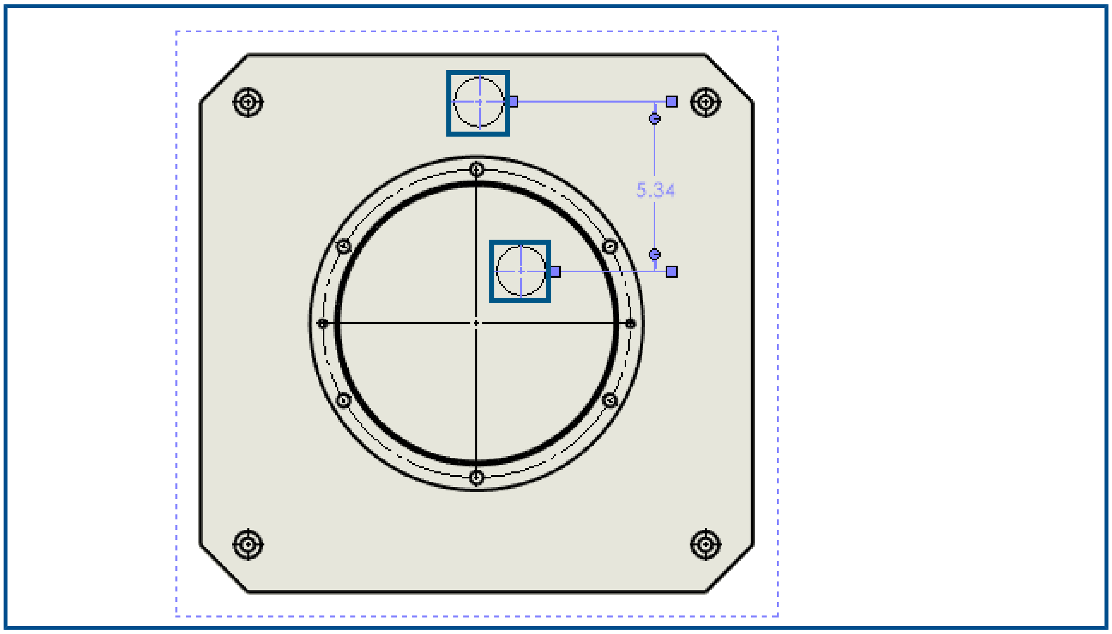
The feature does not support the following dimensions:

- DimXpert or sketch dimensions, such as angular running dimensions and ordinate dimensions
- Cosmetic threads
- Feature dimensions
- Blocked highlight for silhouette edge endpoints
- Referenced edges or points blocked for break view and Detailing mode legacy dimensions

To highlight referenced elements:

1. Click **Tools** > **Options** > **Document Properties** > **Detailing**.
2. Select **Highlight associated elements on reference dimension selection**.

Highlighting Associated Center Marks on Center Mark Dimensions

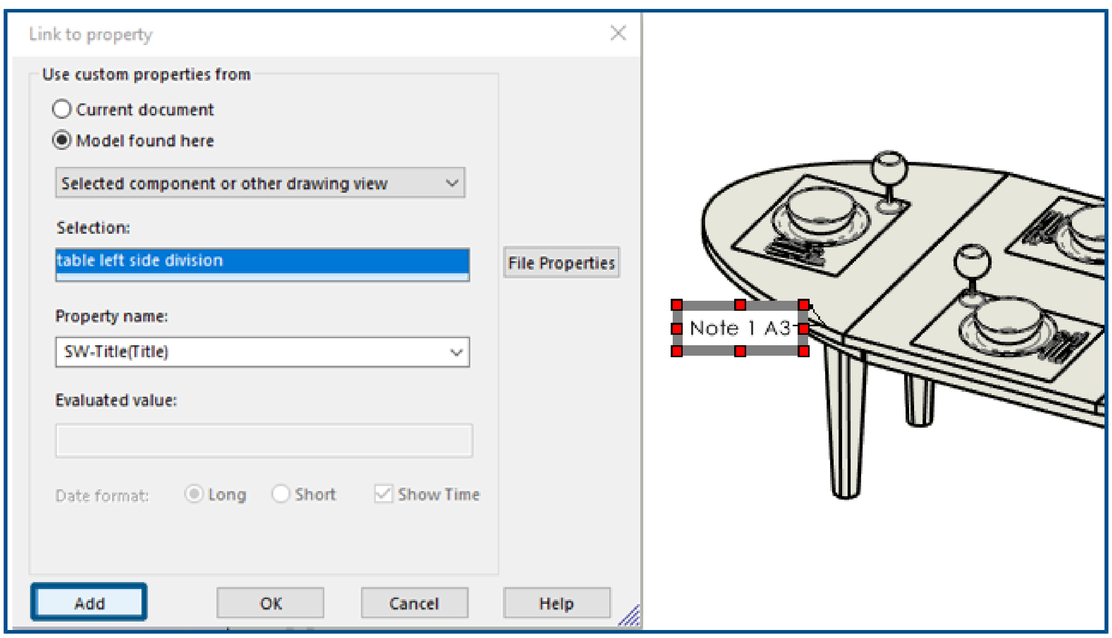


When you select a center mark dimension, the associated center marks highlight as well.

To highlight associated center marks on center mark dimensions:

1. Click **Tools** > **Options** > **Document Properties** > **Detailing**.
2. Select **Highlight associated elements on reference dimension selection**.

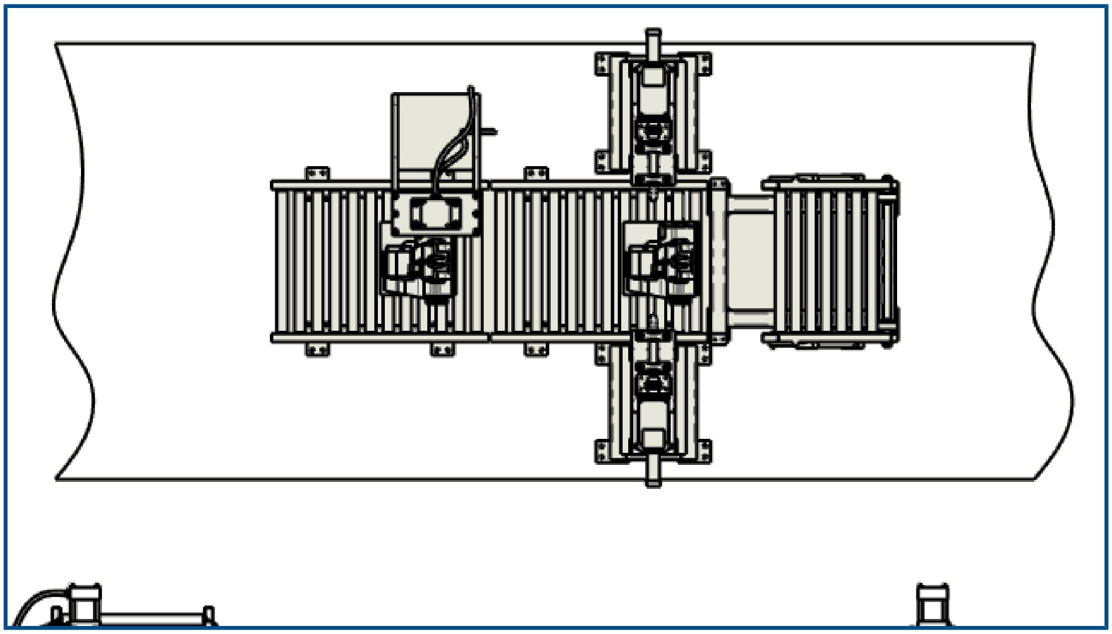
Keep Link to Property Dialog Box Open



When you create a note in a drawing, in the Link to Property dialog box, you can click **Add** to keep the Link to Property dialog box open. You can enter more text or select another property. The dialog box remains open until you click **OK** or exit the note.

Previously, you had to close the dialog box and reopen it. Now you can do everything at once.

Opening a Drawing in Detailing Mode by Default



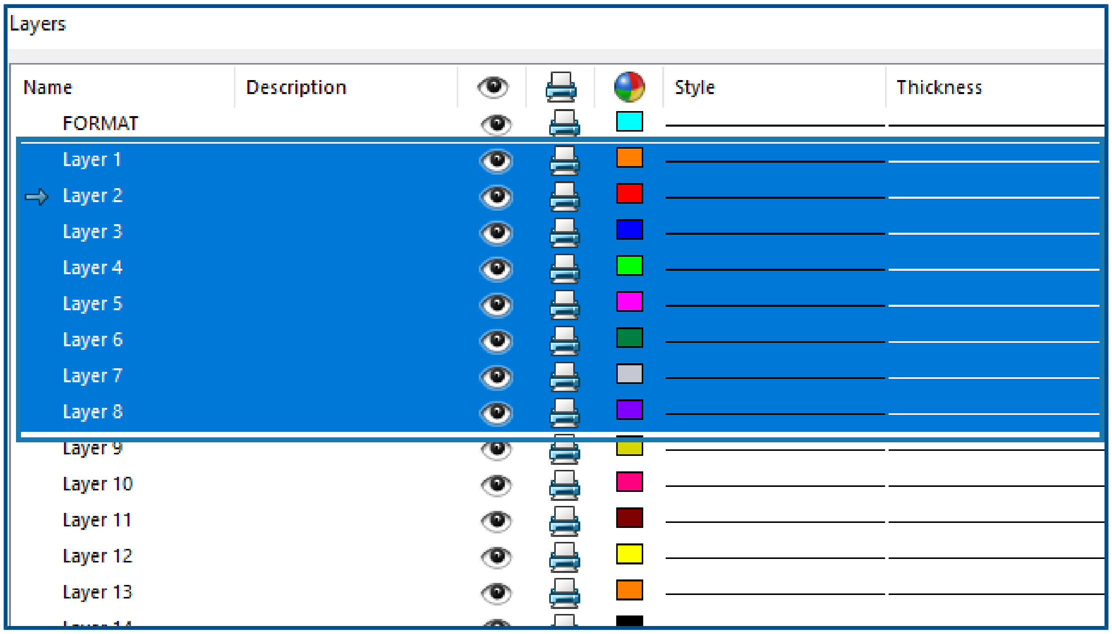
You can open a drawing in Detailing mode by default.

You can use this to automatically open large drawings quickly.

To open a drawing in Detailing mode by default:

1. Click **Tools** > **Options** > **System Options** > **Drawings** > **Performance**.
2. Select **Always open a drawing in detailing mode**.

Select Multiple Layers



You can select multiple layers at once to modify.

Previously, you had to select one layer at a time to modify.

You can:

- **Ctrl** + select each layer that you want.
- **Shift** + select a range of layers.

13

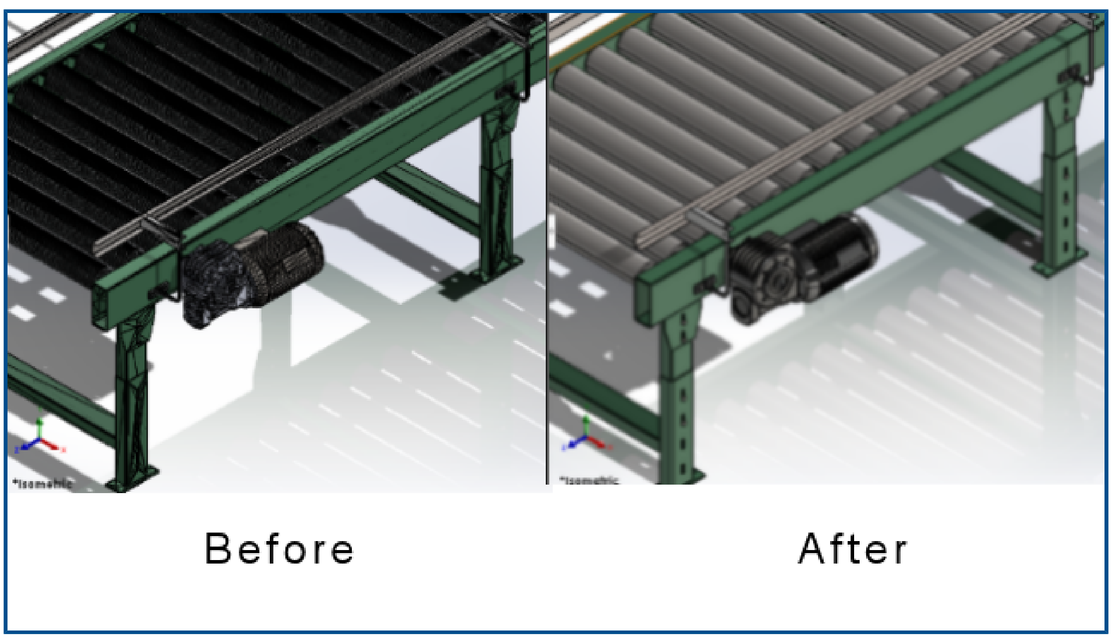
Import/Export

This chapter includes the following topics:

Performance Improvements When Opening 3MF Files (2024 SP3)

Improved performance when opening 3MF files.

Exporting IFC File - Support for Advanced Surface BREP (2024 SP2)



You can export BREP IFC files with cleaner faces.

For example, in the exported files, you can view:

- Planar faces instead of multiple coplanar facets
- Cylindrical faces instead of multiple facets that represent a cylinder

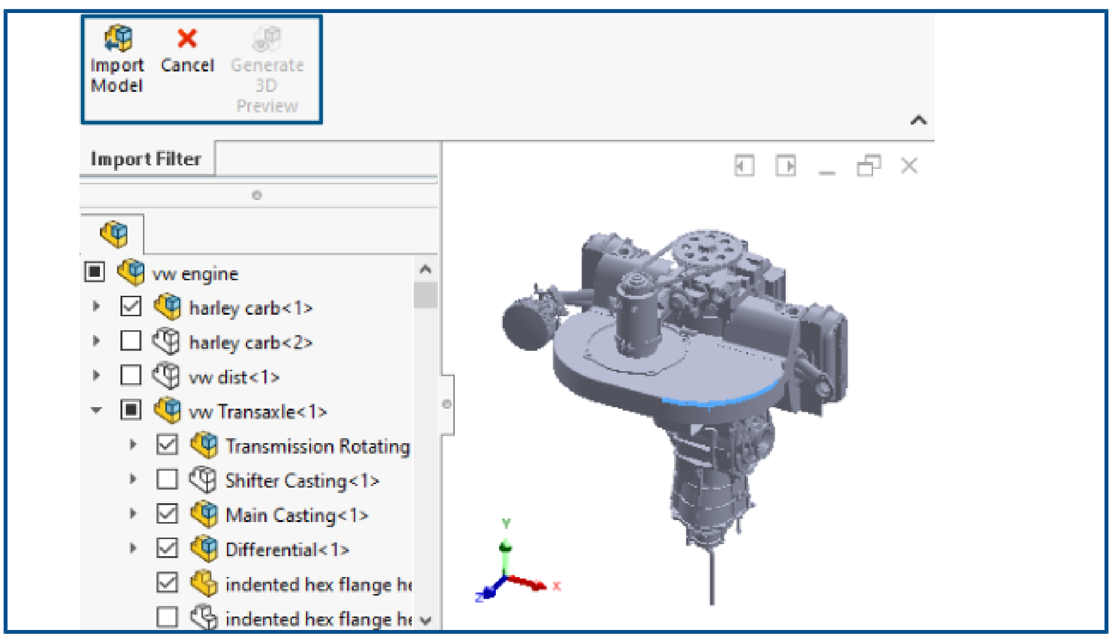
Opening Third-Party CAD Files (2024 SP2)

When importing file formats, SOLIDWORKS uses the latest conversion technology even if you clear **Enable 3D Interconnect** in **Tools** > **Options** > **System Options** > **Import**.

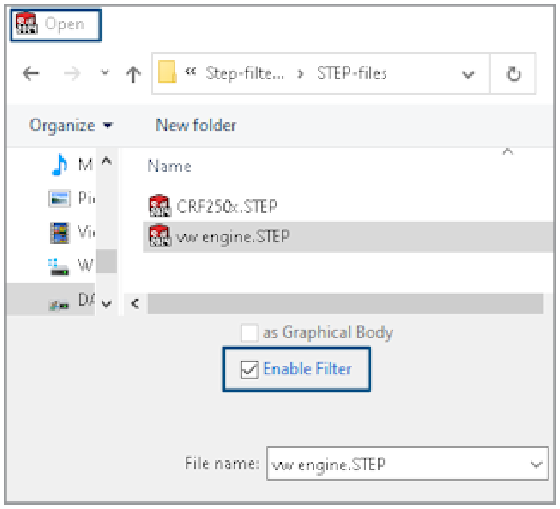
The conversion technology applies to these file formats:

- ACIS™
- Autodesk Inventor®
- CATIA® V5
- PTC Creo®
- IFC
- IGES
- Solid Edge®
- STEP
- NX™ software
- xDesign SLDXML

Using Filters to Import STEP Files (2024 SP1)



While importing a large STEP file using 3D Interconnect, you can apply filters before import. This lets you import selected components from the file using the Import Filter window.

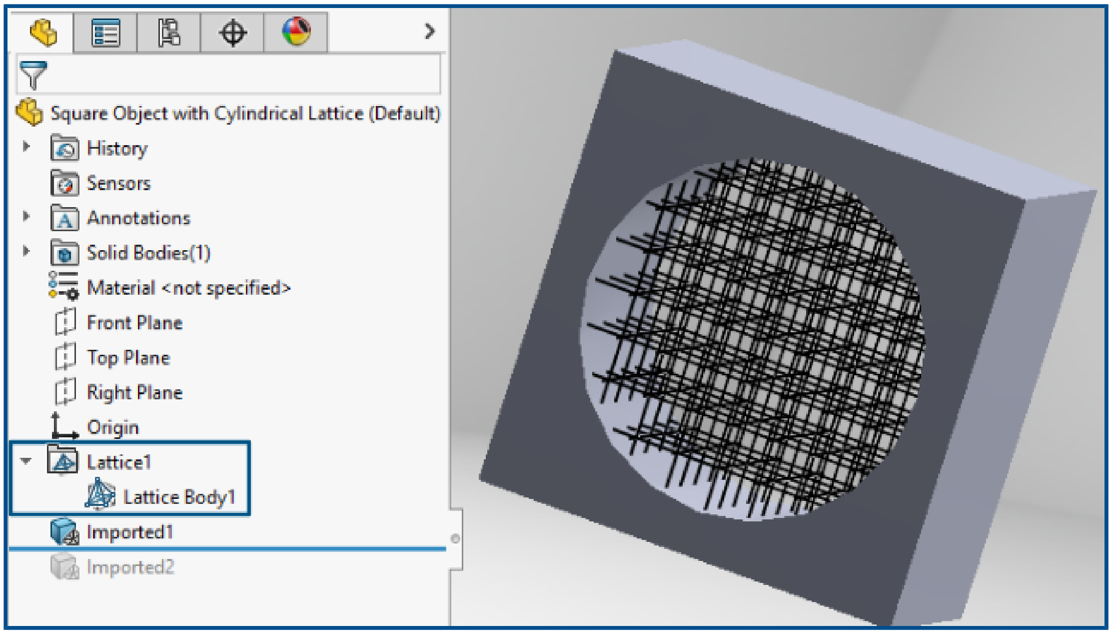


When you select **Enable Filter** while importing STEP file (**File** > **Open**), you can:

- View the STEP product structure similar to the FeatureManager design tree.
- Select and remove components from the STEP product structure.
- Right-click components and click **Keep Components** or **Exclude Components** to select or remove multiple components at once.
- Generate a minimalistic graphics preview (with fewer details such as excluding appearances) in the graphics area with **Generate 3D Preview** .
- Click **Import Model**  or **Cancel**. after previewing the filtered minimalistic model or directly without generating the graphics preview.

Importing a large STEP file is faster with improved performance depending on the number of objects that you select while applying filters. It also helps working with a simplified model.

Importing 3MF Files - Support for 3MF Beam Lattice Extension (2024 SP1)



When importing 3MF files containing beam lattices, you can import `.3mf` beam lattices.

In the FeatureManager design tree, each lattice in the imported file appears as an independent lattice feature containing one or more disjoint lattice bodies. Lattice bodies are lightweight bodies with thin lines representing the centerline of the beams.

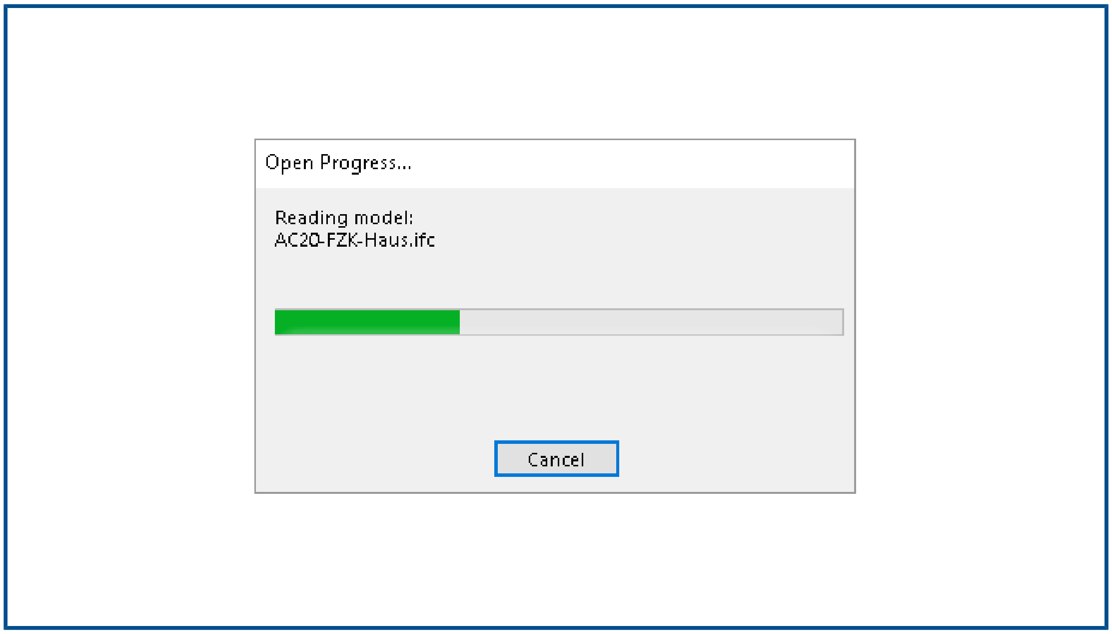
With the lattice bodies and features, you can:

- Convert them to mesh bodies

> This generates the full geometry of the lattice (including the beam diameter, variable beam diameter, and connecting spheres) as mesh BREP geometry. For more information, see *SOLIDWORKS Help*: *Graphics Mesh and Mesh BREP Bodies*.

- Hide or show them in the graphics area
- Create section views

Canceling the Import of Third-Party CAD Files



You can cancel the import of a third-party CAD file with 3D Interconnect if importing takes too long.

To cancel the import of third-party CAD files:

1. Click **File** > **Open**.
2. Optional: **3D**EXPERIENCE® Users: If the Open from 3DEXPERIENCE dialog box appears, click **This PC**.
3. In the Open dialog box, select a third-party CAD file and click **Open**.
4. In the Open Progress dialog box, while the import status is **Reading model**, click **Cancel** or press **Esc**.

   > You cannot cancel when the import status changes to **Loading model**.

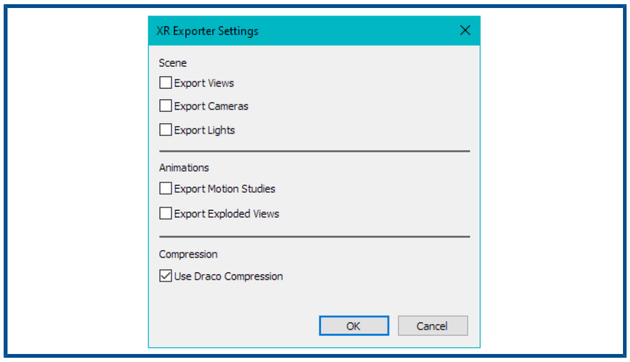
5. In the confirmation dialog box, click **Yes**.

Importing STEP Assemblies as Multibody Parts

Enhancements related to importing `STEP`, `IGES`, and `IFC` assemblies as multibody parts include:

- Import is available with a SOLDWORKS® parts-only OEM version.
- The performance of importing `STEP`, `IGES`, and `IFC` assemblies as multibody parts is improved up to 30%.

Exporting to Extended Reality



You can export SOLIDWORKS CAD files to `.glb` or `.gltf` file formats.

The files contain information such as geometry, appearances, textures, animations, motion studies, configurations, display states, exploded views, lights, and metadata. For large files, the export supports Draco, the standard file compression mechanism for `.glb` and `.gltf` files.

14

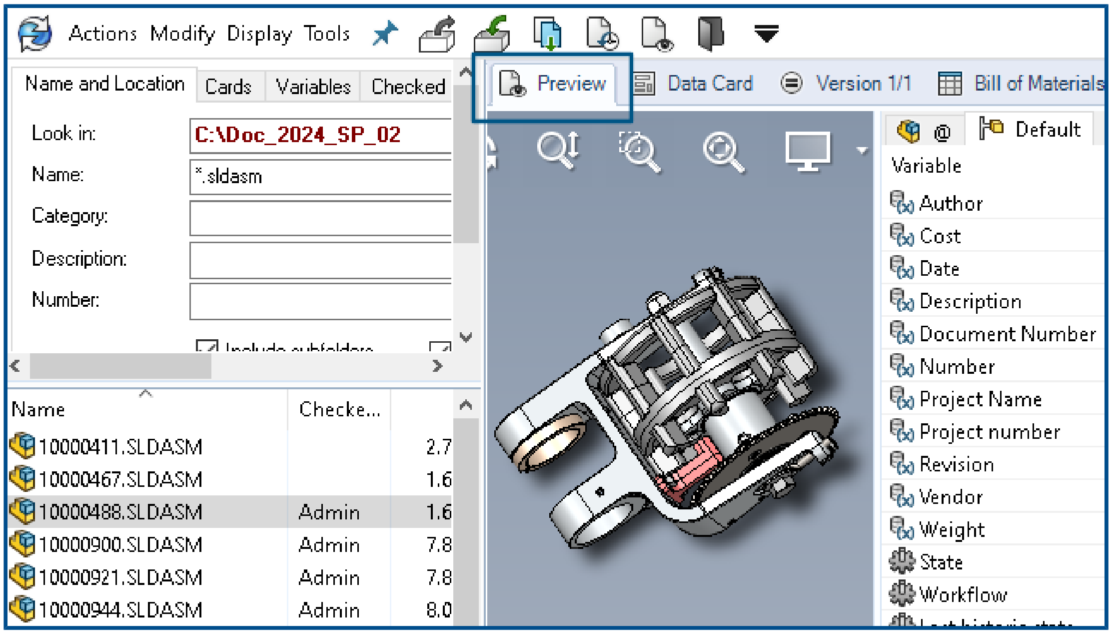
SOLIDWORKS PDM

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.

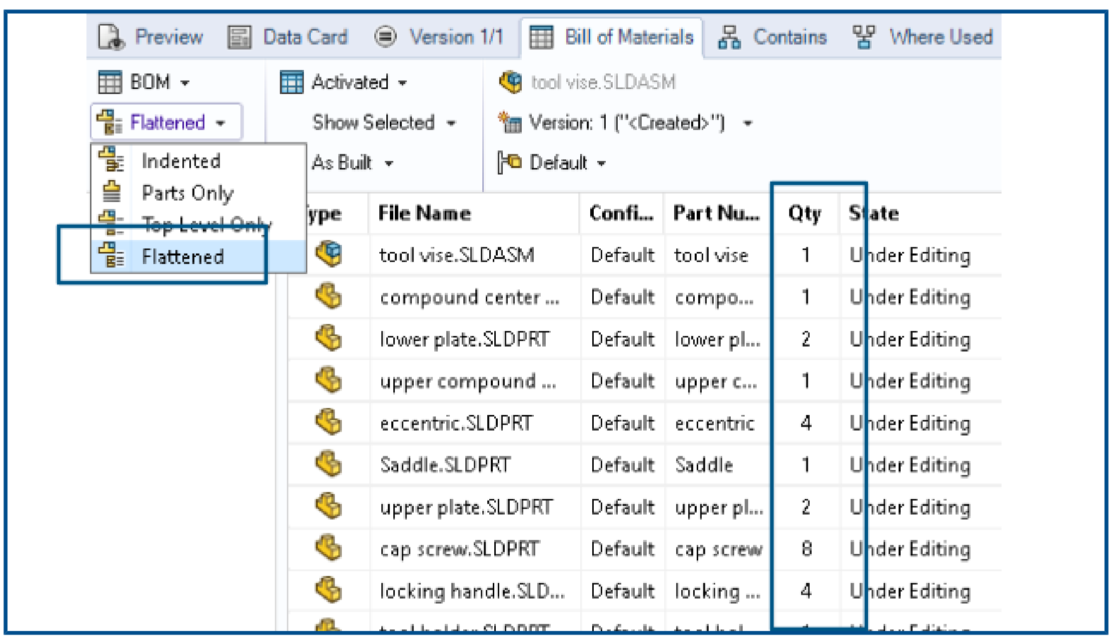
SOLIDWORKS PDM Professional is a full-featured data management solution for a small and large number of users, and is available as a separately purchased license.

Displaying the Preview Tab for Search Results (2024 SP2)



In the SOLIDWORKS PDM File Explorer, you can display the **Preview** tab for an item in the search result (Quick, Integrated, and Standalone search) at the bottom or to the right side of the window using the existing **Preview Placement** option.

Bill of Materials (BOM) View - Flattened Type (2024 SP2)



In the SOLIDWORKS PDM File Explorer, in the BOM view of the **Bill of Materials** tab, you can use the new type **Flattened** to view the total number of quantities required of a component present in the product structure.

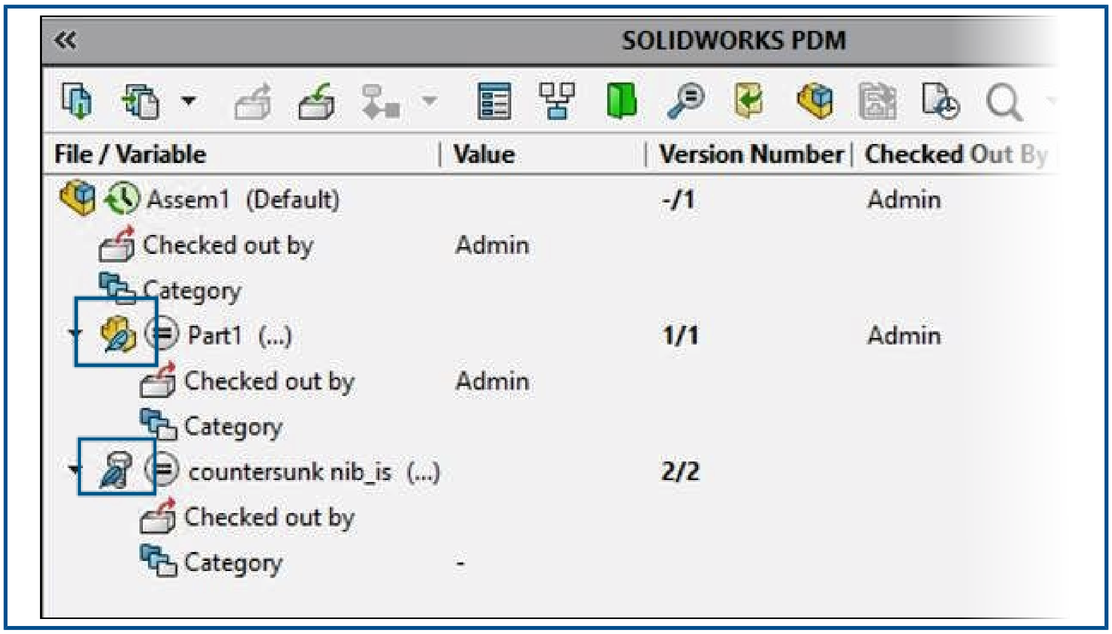
This option saves time and effort in calculating the total number of quantities of the components.

The **Flattened** BOM view displays:

- The product structure as a list of components without indentation.
- The component only once if it is present at multiple levels of the product structure.
- The quantity of the component by adding the quantities at each level.

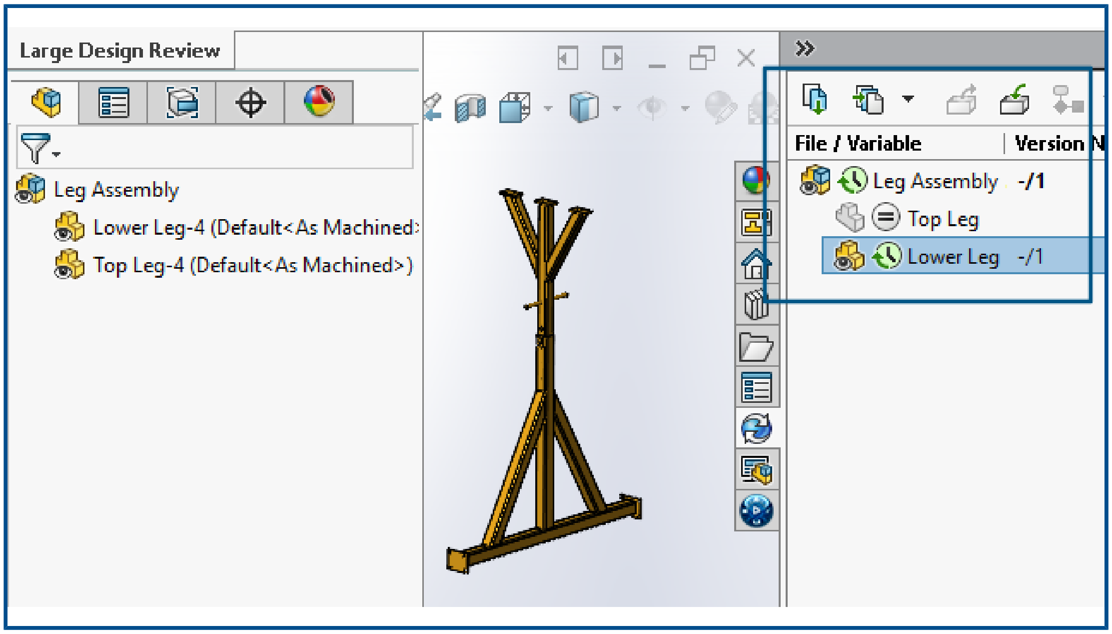
The **Flattened** type is available when viewing the computed BOMs in the desktop client and in Web2.

SOLIDWORKS PDM Add-in Enhancements (2024 SP1)



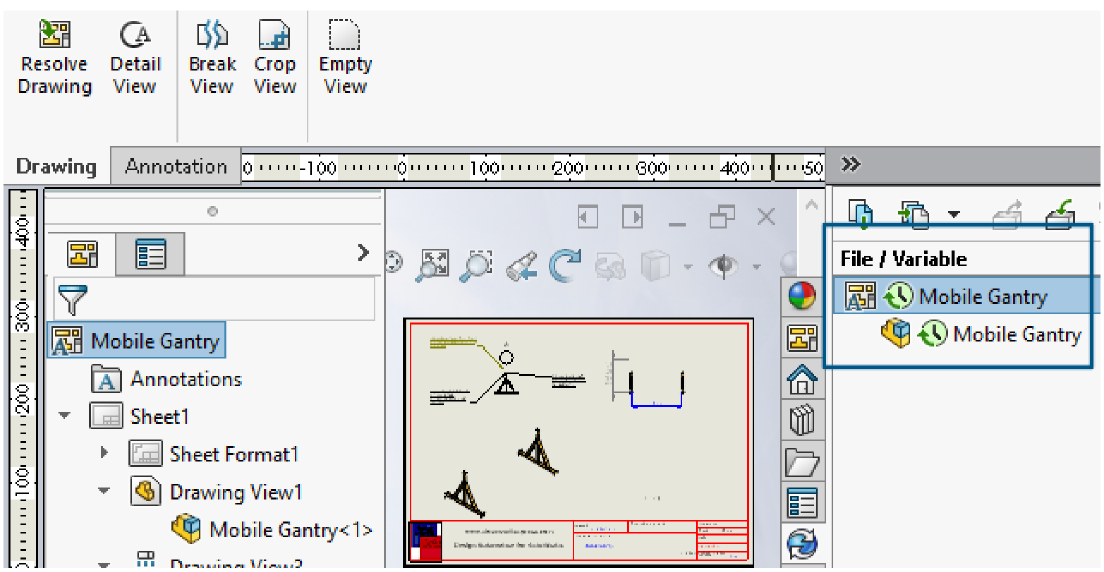
- When you save an assembly file as a part file, an internal component (saved as an external file in the vault), or a mirror component using the **Save as** command, a data card for the new file displays generating serial numbers and default values if set in the card.
- The SOLIDWORKS PDM add-in displays an icon overlay and supports all SOLIDWORKS PDM operations for components that are open in lightweight mode.
- You can enable the **Automatically optimize resolved mode, hide lightweight mode** option even when the SOLIDWORKS PDM add-in is active.

Handling Large Design Review (LDR) and Detailing Mode in the SOLIDWORKS PDM Add-in (2024 SP2)



For assemblies opened in **Large Design Review (LDR)** mode and for drawings opened in **Detailing** mode, you can view the SOLIDWORKS files structure in the SOLIDWORKS PDM Task Pane (along with icons) similar to the FeatureManager design tree.

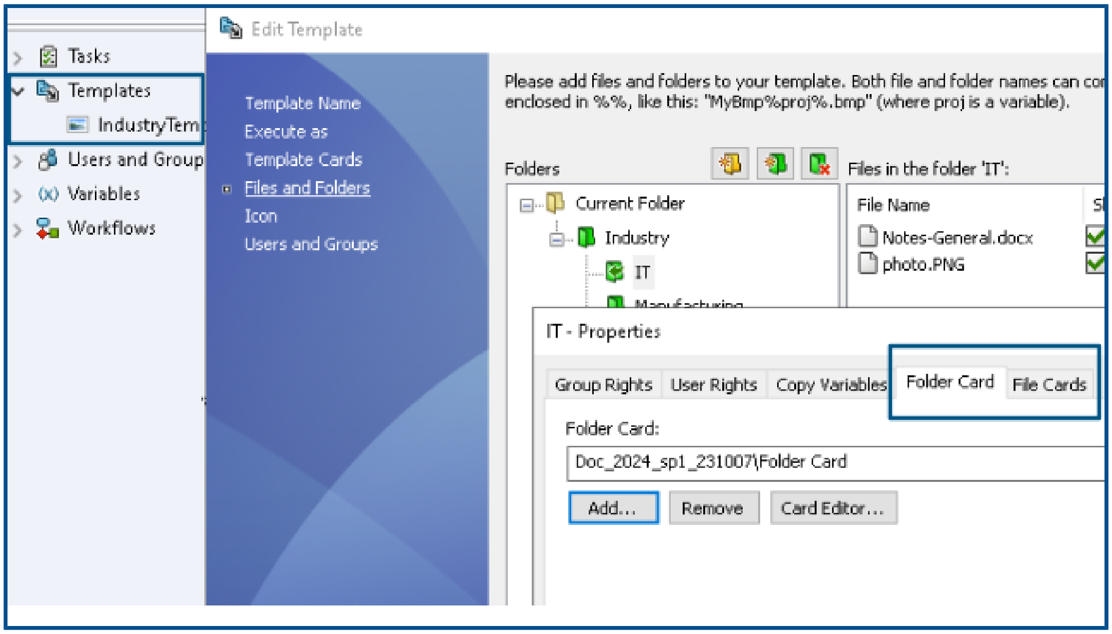
Because the display of both the FeatureManager design tree and the Task Pane tree are identical, you can work on the product structure with more clarity and ease.



For the **Detailing** mode, the PDM Task Pane tree displays child components only to the first level similar to the FeatureManager design tree.

For the **Large Design Review (LDR)** mode, you can perform SOLIDWORKS PDM operations such as **Check in** and **Check out** on the components from both the FeatureManager design tree and the Task Pane assembly tree.

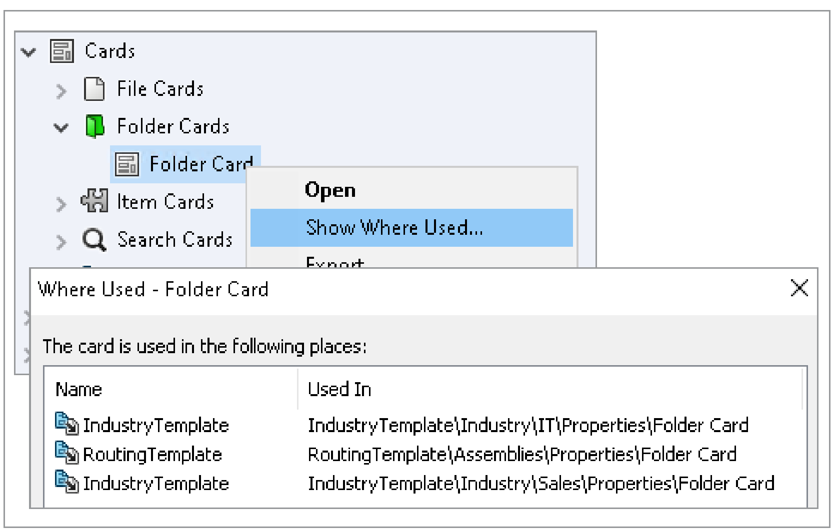
Assigning Data Cards to Files and Folders of a Template (2024 SP1)



In the SOLIDWORKS PDM Administration tool, while creating and editing a template, you can assign a folder card and multiple file cards to a folder.

In SOLIDWORKS PDM File Explorer, right-click and click **New** in the right pane. When the software creates the files and folders structure, the respective data cards are assigned automatically.

Changes to the file extensions for a card, assigned to a template, outside of the template configuration are not recognized.



In the SOLDWORKS Administration tool, under **Cards**, for each file, folder, and template card, you can right-click and see where the card is used. For example, click **Cards** > **Folder Cards** > ***Folder Card*** > **Show Where Used**. This option is useful when deleting a file or a folder data card.

Where Used Card Dialog Box

You can use this dialog box to display where a file, folder, or template card is used.

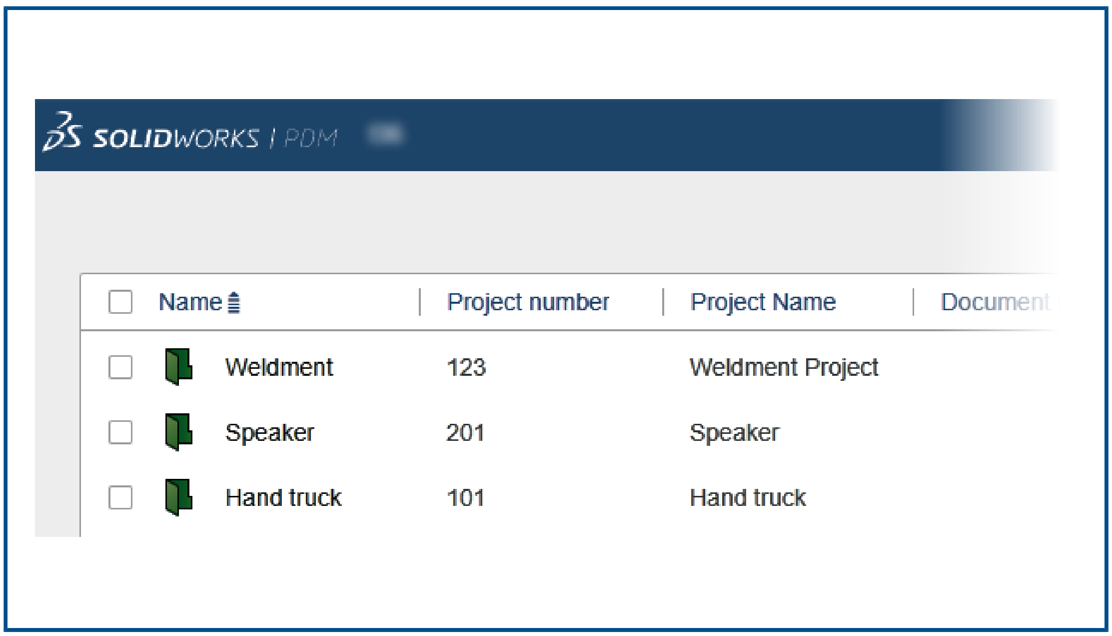
To open this dialog box:

1. In the Administration tool, expand **Cards**.
2. Expand a file, folder, or template card menu, for example **Folder Card**
3. Right-click the card.

You can see a list of all the places where the card is used:

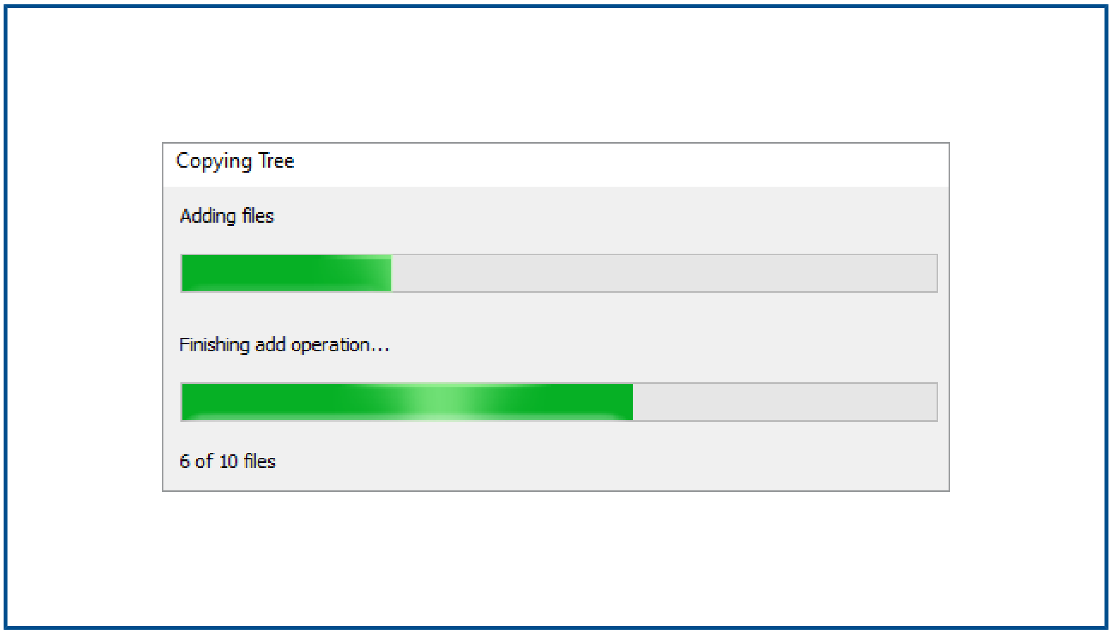
Name	Displays the template using the card.
Used In	Displays where the card is used.

Folder Card Variables in Web2 (2024 SP1)



In Web2, you can view data card variables for folders in a folder list. The values for custom columns for the folders are displayed in the list view of the large screen layout.

Progress Dialog Boxes (2024 SP1)



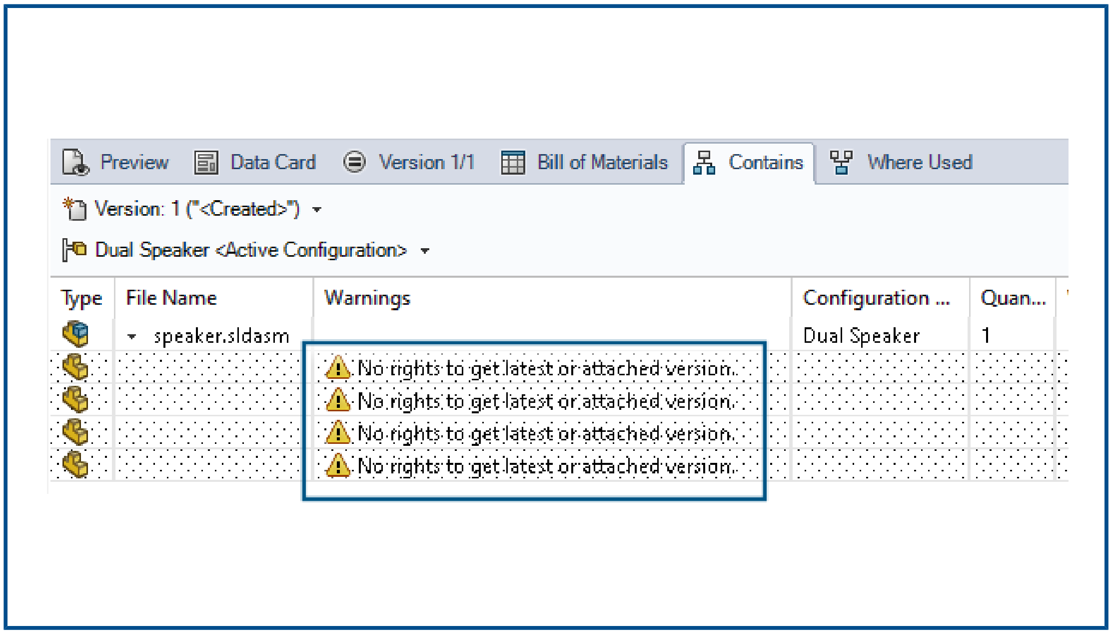
In the SOLIDWORKS PDM File Explorer, the progress dialog box of certain operations displays more information.

The Change State and Copy Tree progress dialog boxes have two progress bars:

- The first progress bar has the primary steps or actions of the overall operation, such as **Copying Files** and **Copying Variables**.
- The second progress bar has detailed information such as secondary steps, total number of files, etc.

The Check In and Reading File References progress dialog boxes have a single progress bar that displays the current action and file names.

Data Security Enhancements (2024 SP1)

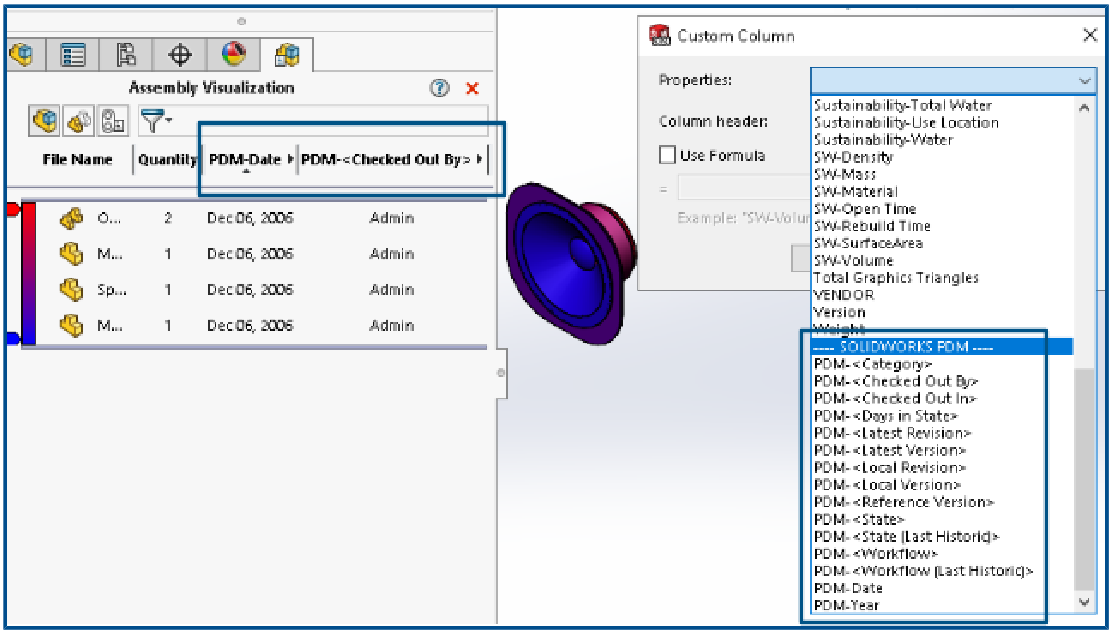


In SOLIDWORKS PDM File Explorer and Web2, unauthorized users cannot view file information in file view tabs or in file operations and file reference dialog boxes.

The warning message **No rights to get latest or attached version** displays for the following:

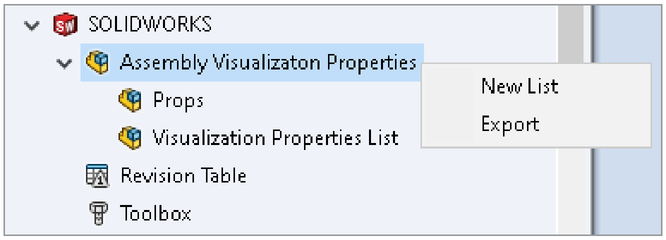
- File view tabs:
  - Contains
  - Where Used
  - Bill of Materials (Computed BOMs and Named BOMs)
- File operations dialog boxes
- File reference dialog boxes

Assembly Visualization



You can access SOLIDWORKS PDM variables in the SOLIDWORKS Assembly Visualization tool.

The SOLIDWORKS PDM variables are listed under **Properties** in the **Custom Column** dialog box of the Assembly Visualization tool. You can select variables, for example, **PDM-<Checked Out By>** or **PDM-Date**under the **SOLIDWORKS PDM** section in **Properties** and then view them in the Assembly Visualization panel.



To view SOLIDWORKS PDM custom variables in Assembly Visualization:

1. In the SOLIDWORKS PDM Administration tool, right-click **SOLIDWORKS** > **Assembly Visualization Properties**and click **New List**.
2. In the Customize Assembly Visualization Properties - Visualization Properties List dialog box, create a property list from the available variables.You can create multiple lists of properties and view them in Assembly Visualization depending on the permissions.

Customize Assembly Visualization Properties Dialog Box

You can use this dialog box to specify variables for specific users or groups that they can view in the SOLIDWORKS Assembly Visualization tool.

To open this dialog box:

1. In the Administration tool, expand **SOLIDWORKS**.
2. Right-click **Assembly Visualization Properties** and select **New List**.

Name

Specifies the name of the new properties list.

Variables

Variable	Displays the selected variable.
Name	Displays the name of the selected variable.

Add	Adds the selected variable.
Delete	Deletes the selected variable.
Up and down arrows	Moves the selected variables up or down.

Selected Variable

Variable	Displays the list of available variables and lets you select a variable from the list.
Name	Displays the name of the selected variable and lets you update the name.

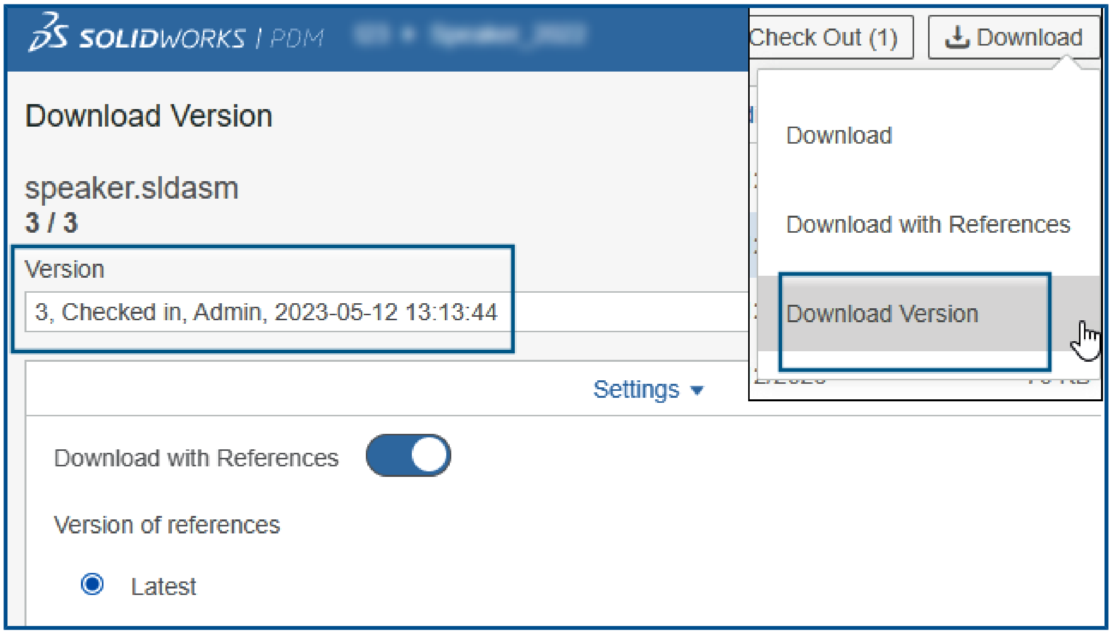
Users

Lists users and lets you specify users who can select the variables and view the list.

Groups

Lists groups and lets you specify groups whose members can select the variables and view the list.

Downloading Specific Versions of a File in Web2



SOLIDWORKS PDM Web2 lets you download a specific version of a file and its references.

You cannot select and download multiple files in a single operation.

The Download Version dialog box lets you select the version and settings for download.
To access this dialog box:

1. In the File list, select a file:
   - Large screen layout. Click **Download** > **Download Version**.
   - Small screen layout. Touch **Download** and then touch **Download Version**.

Download Version Dialog Box

You can use the Download Version dialog box to download a specific version of a file and its references.

To open this dialog box:

- Select a file and click **Download** > **Download Version**.

Version

Select the version of the file to download.

Settings

The collapsible option that displays the download settings options for files.

Download with references	Downloads the file with its references.
Version	**Latest** – Downloads the latest version. **Referenced** – Downloads the referenced versions.
Preserve relative paths	Preserves the paths of references relative to the parent file and creates a folder structure as required. When cleared, the folder hierarchy is flattened, and all referenced files are uploaded to the same destination folder as the parent file.
Include drawing	Downloads the drawing files associated with the file selected to download.
Include simulation	Downloads the SOLIDWORKS Simulation results associated with the selected files.

Files

Lists the file references to download. The file list includes customizable columns such as **State**, **Version**, **Size**, and **Path**. Click **Show More** and specify the columns to display.

Total Files to Download

Displays the total number of files and the count of individual files to download.

Download

Downloads the selected files. When the download is complete, a message appears with the number of downloaded files on the upper bar. If Web2 cannot download any references, a warning message appears.

Download Version Dialog Box - Small Screen Layout

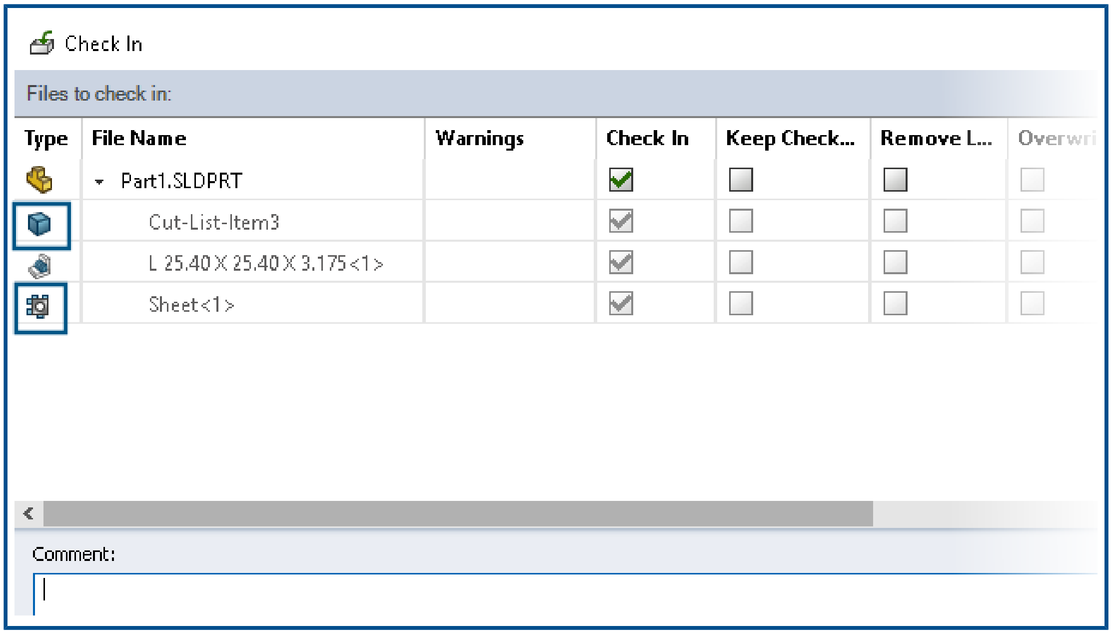
You can use the Download Version dialog box to download a specific version of a file and its references.

To open this dialog box:

1. Select a file and touch **Download**.
2. Touch **Download Version**.

Filename and latest version	Displays the version list and where you can select a version to download.
Settings	Lets you specify options.

File Type Icons



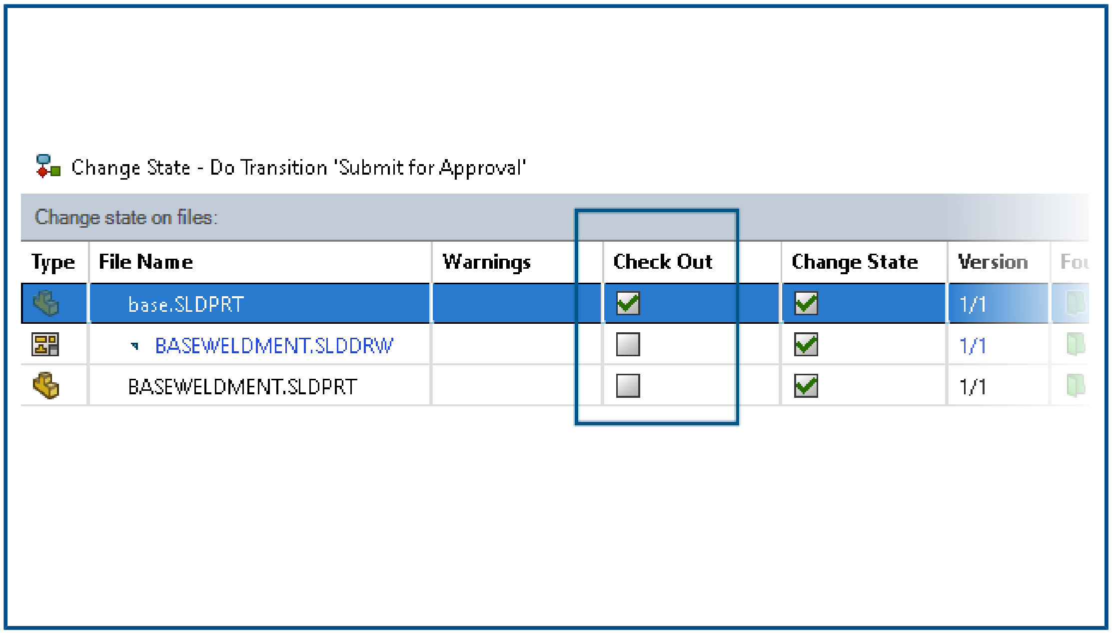
You can view the file type icons for weldment cut list items and the files that were shared using pasted shared overlays.

These icons are available in the dialog boxes for:

- File Details
- File Operations
- Web2

The type icons for cut list items are not available for SOLIDWORKS BOMs.

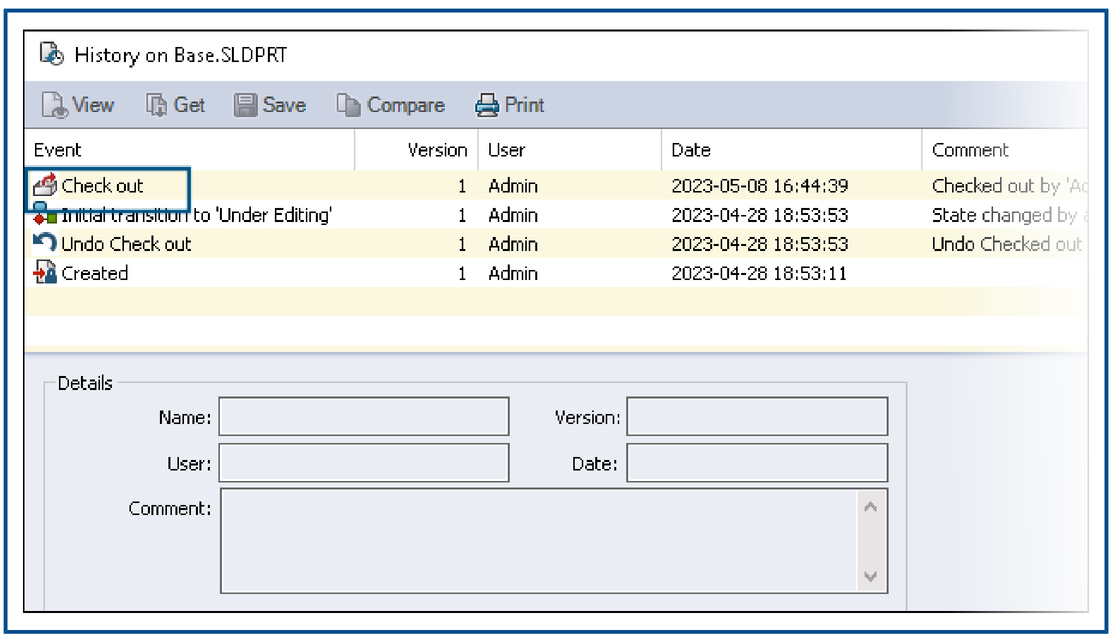
Check Out Option in Change State Command



You can check out a file after the change state operation completes.

You can customize the column set of the Do Transition dialog box to include the **Check Out** system variable. If you select **Change State** and **Check Out** for a file, the file is checked out after its state changes.

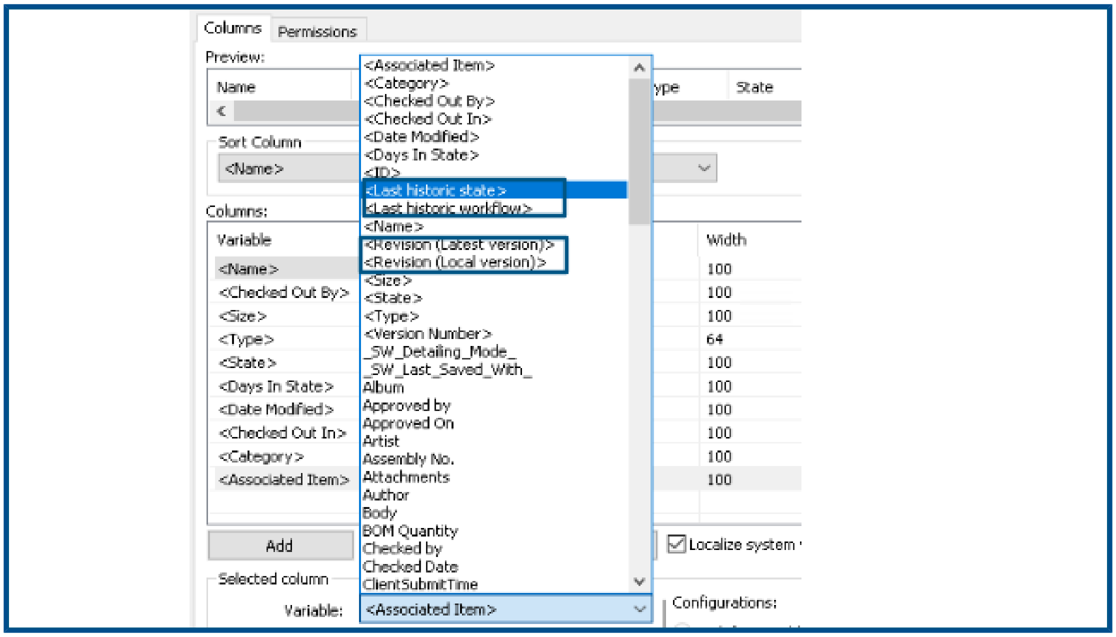
Viewing Check-Out Event Details



In SOLIDWORKS PDM File Explorer, you can view details of check-out and undo check-out events in the History dialog box of a file.

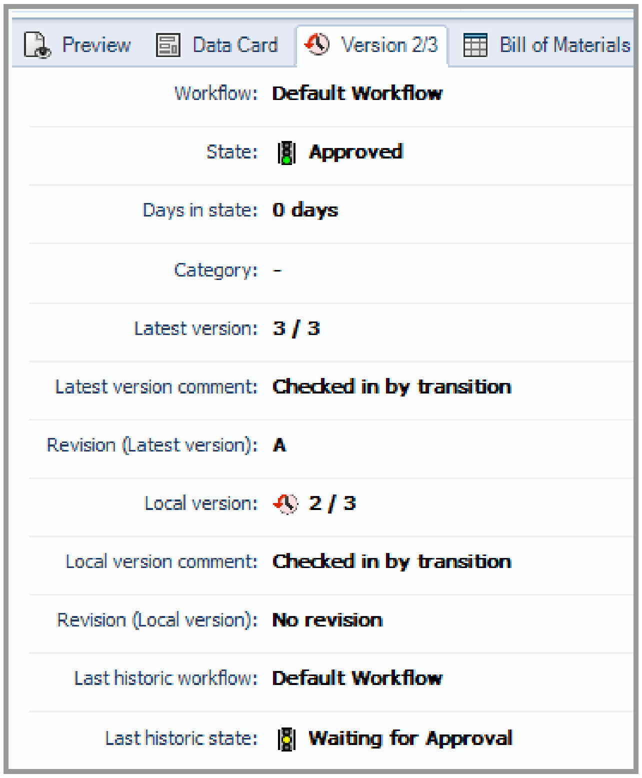
Along with the other details, you can see which user has performed the operation.

System Variables

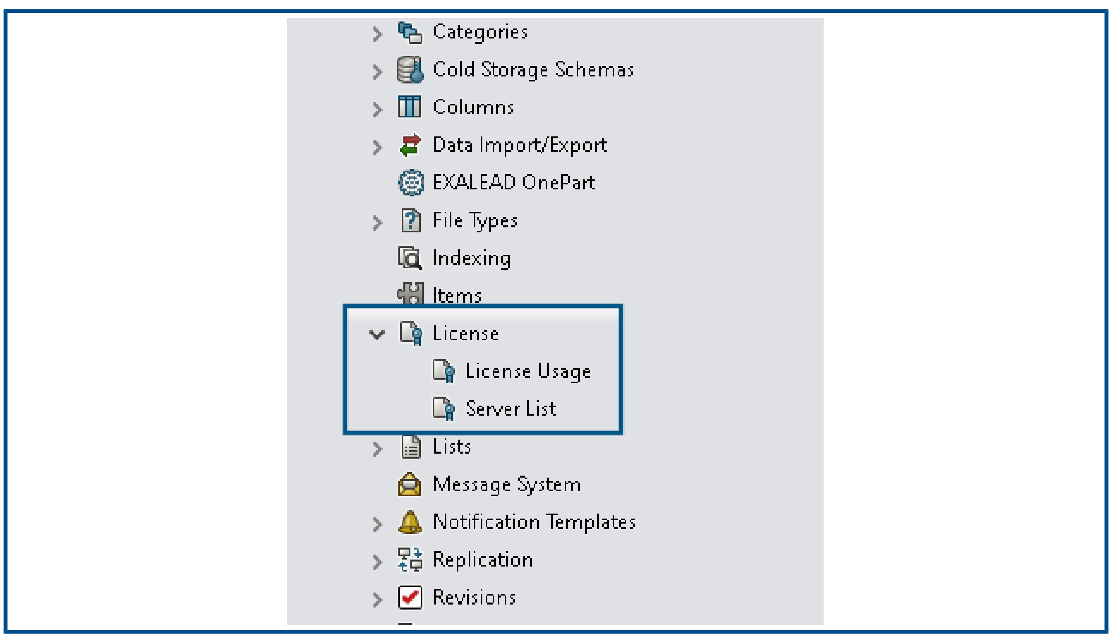


System variables are more available and easier to access.

- The following system variables are available in the **File List**, **Quick Search Result**, and **Search Result** column set types:
  - **<Last historic state>**
  - **<Last historic workflow>**
  - **<Revision (Latest version)>**
  - **<Revision (Local version)>**
- The **<Days in State>** system variable is available as a default column in **File list**.
- The SOLIDWORKS PDM task pane add-in has more system variables.
- In SOLIDWORKS PDM File Explorer, the addition of more system variables improves the user interface of the Version tab.



Viewing License Usage



You can view license details without any special administrative permissions.

In the Administration tool, the **License** node has the following subnodes:

- **Server List**. Lets you edit license servers.

> The administrative permission **Can update license keys** is renamed as **Can update license server**. You need this permission to edit license servers.

- **License Usage**. Lets you view license details. This helps you to ask users to log out if they are not using the tool, request more licenses from the administrator, or decide whether you need to switch to a different license type.

SOLIDWORKS PDM Performance Improvements

SOLIDWORKS PDM 2024 has improved the performance of file-based operations.

The following operations are approximately two times faster:

- Add files
- Change state
- Copy tree

  The copy tree to compressed archive operation is orders of magnitude faster.

15

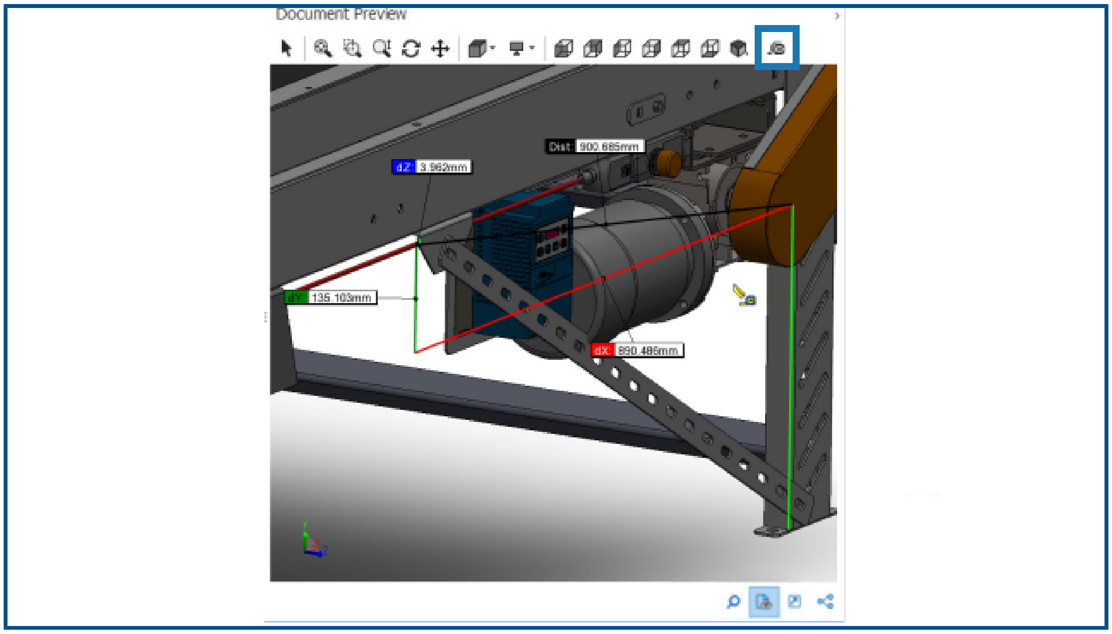
SOLIDWORKS Manage

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.

Measuring in a Document Preview



You can measure geometry in the **Document Preview** area.

You can use the measure tool when you preview a document supported by the eDrawings Viewer.

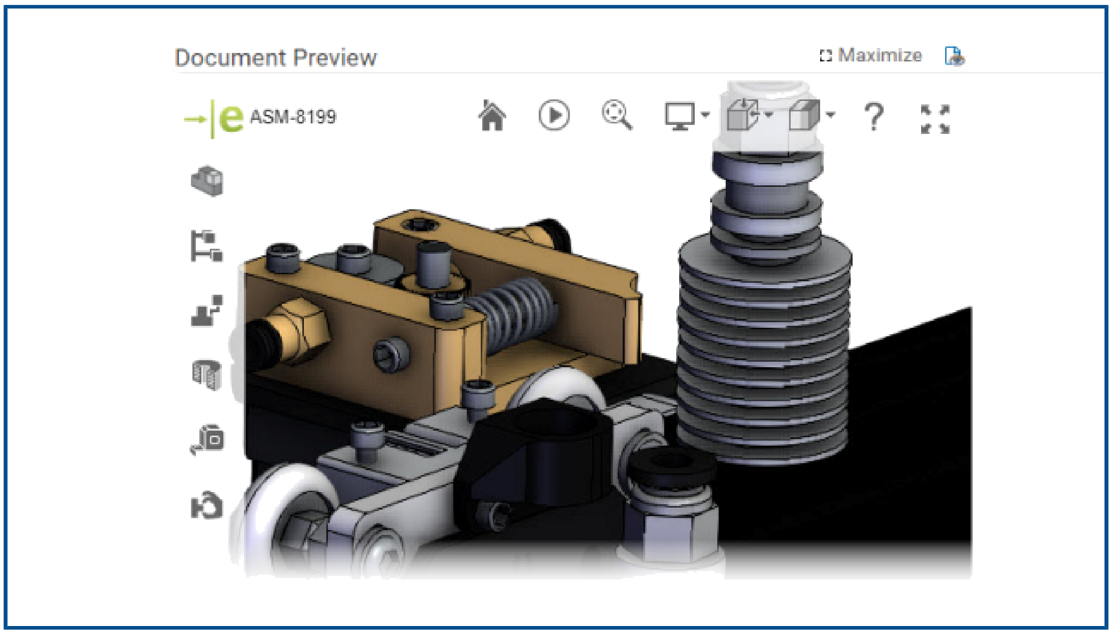
To measure in a Document Preview:

1. In the main grid, select a part, assembly, or drawing record.
2. Click **Document Preview** .

   The eDrawings® preview displays the selected SOLIDWORKS record.

3. Click **Measure** .
4. Select geometry to measure in the preview.

Plenary Web Client CAD File Preview

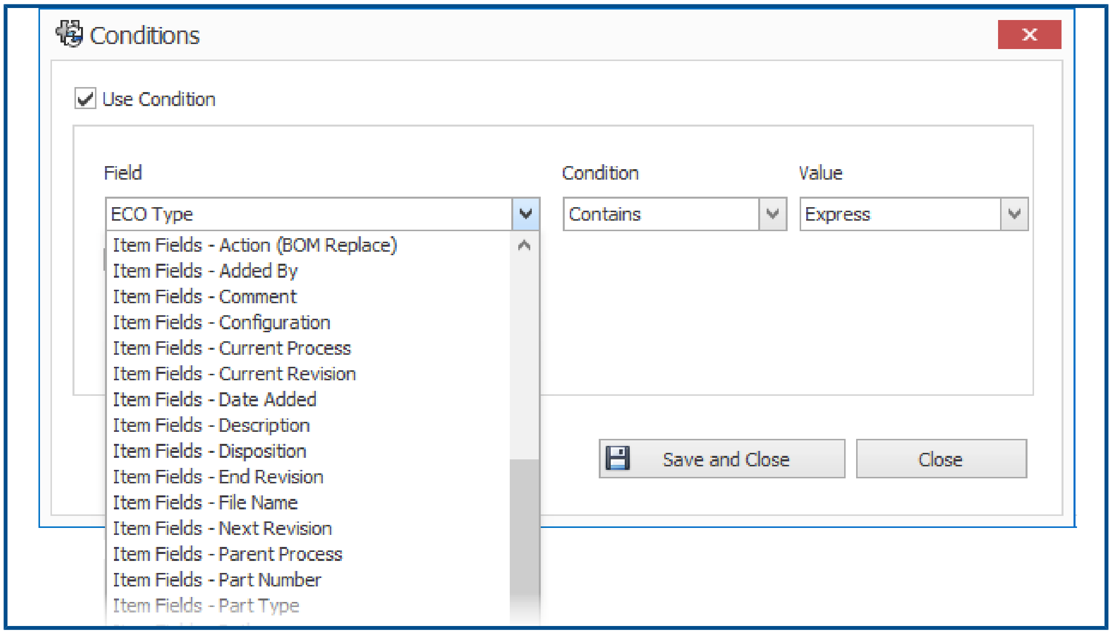


You can dynamically preview CAD files in the Plenary web client windows.

The preview is based on eDrawings and supports the same file type and functionality.

In previous releases, to get a dynamic preview you had to click a preview link to open the SOLIDWORKS PDM Web 2 client.

Field Conditions for Affected Items



You can add conditions for **Affected Items** mapped fields to control their existence and default values.

When a field has a condition for its existence, that is, if the condition is required or not, a blue asterisk appears in the column name. If you do not define a condition, the field is always available, and a red asterisk appears.

Adding Required Fields to an Affected Item Field

To add required fields to an affected item field:

1. In the System Administration tool, open the Process Wizard.
   To open the Process Wizard, right-click a process and click **Administration**.
2. If the process does not have at least one custom field, open the Item Fields wizard and add a custom field.

   > You cannot define mapped fields as required fields.

3. Open the Workflow Properties wizard and select a stage in the workflow diagram.
4. Click **Item Fields**.
5. Select **Required**.
   To add a condition, click ellipses in the first **Condition** column to open the Conditions dialog box.

   > You can also add **Item Fields** to define the condition.

6. Click **Save**.

Adding Default Values to an Affected Item Field

To add default values to an affected item field:

1. In the Administration Options tool, open the Process Wizard.
   To open the Process Wizard, right-click a process and click **Administration**.
2. If the process does not have at least one custom field, open the Item Fields wizard and add a custom field.

   > You cannot define mapped fields as required fields.

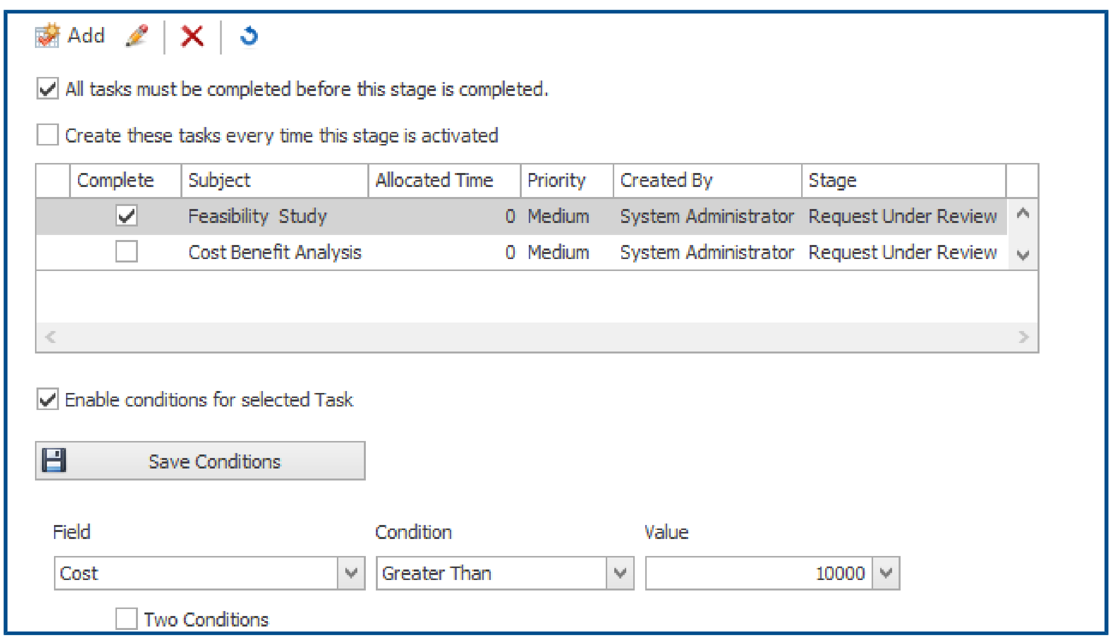
3. Open the Workflow Properties wizard and select a stage in the workflow diagram.
4. Click **Item Fields**.
5. Click the **Default** column and select a value from the list or enter a value.

   > Mapped fields cannot have a default value.

6. In the **When** column, select **Start** or **Finish** to specify when to enter the default value to the field.
   To add a condition, click ellipses in the second **Condition** column to open the Conditions dialog box.

   > You can also add **Item Fields** to define the condition.

Task Automation



Task automation streamlines the preconfiguration process of handling tasks.

You can add conditions to control the creation of individual tasks. This helps to create tasks that are based on process field values. For example, if multiple departments can participate in a process, each with their own task, you can add conditions to create the tasks for the required departments.

Adding Task Conditions

You can add conditions to control the creation of individual tasks.

To add task conditions:

1. Open the Process Wizard for an existing process and navigate to the Workflow Properties wizard.
2. Select a stage and click **Tasks**.
3. Click a task and select **Enable conditions for selected Task**.
4. Specify the task conditions.

Defining Task Completion Requirements

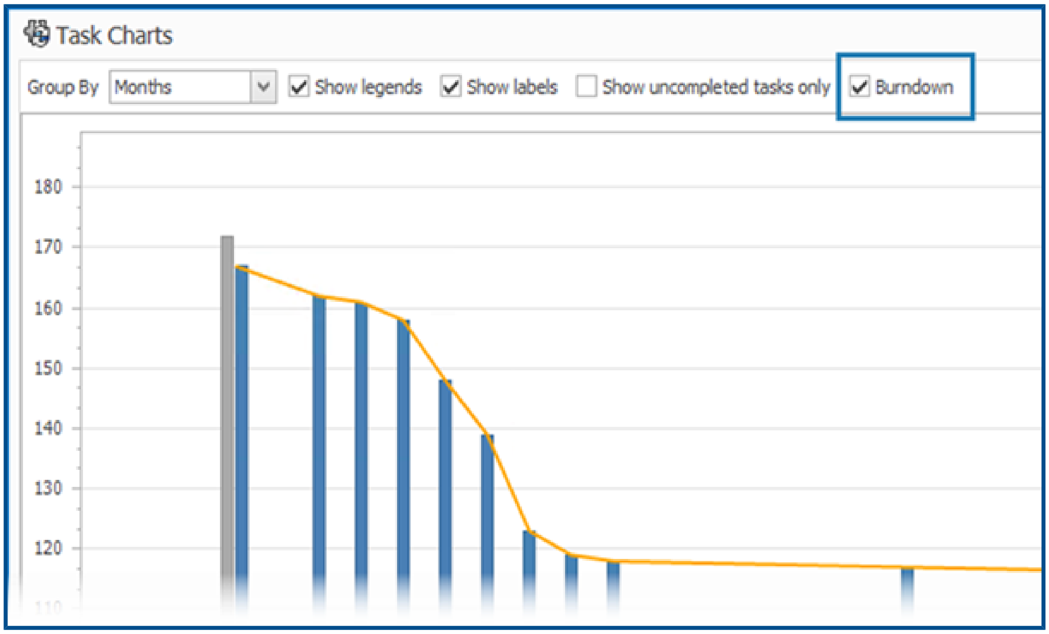
You can define individual tasks to complete before processes can move forward.

In previous releases, the only options to move a process forward was to complete all tasks.

To define task completion requirements:

1. Open the Process wizard for an existing process and navigate to the Workflow Properties wizard.
2. Select a stage and click **Tasks**.
3. Select a task.
4. Clear **All tasks must be completed before this stage is completed**.
5. In the task list, select the check box in the **Complete** column for each task to complete.

Task Burn Down Chart



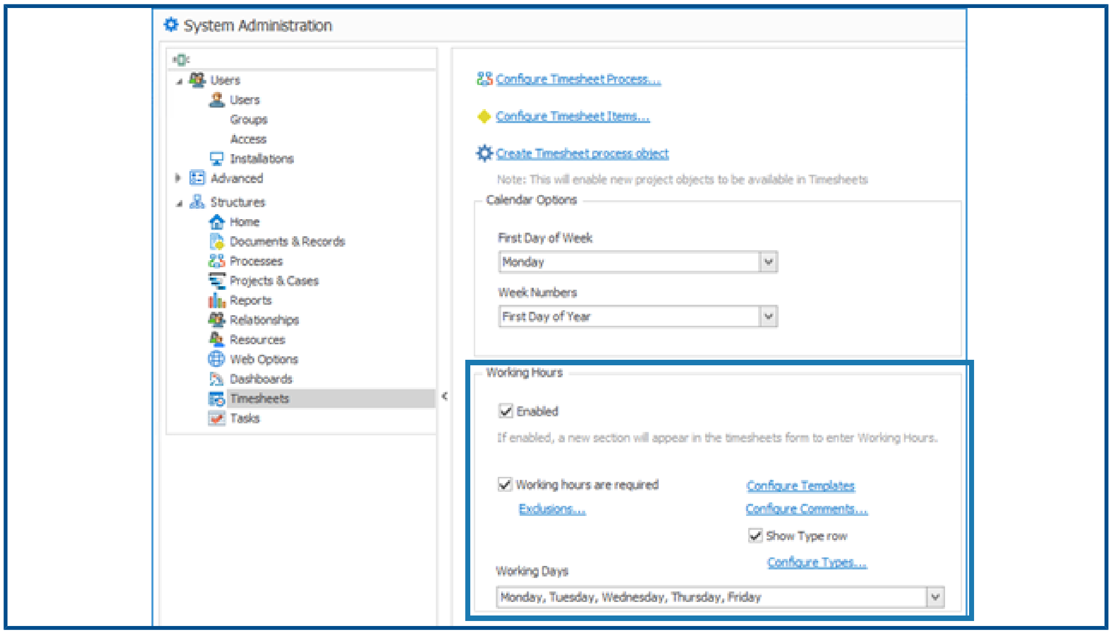
The task burn down chart shows the progression of all project tasks.

The chart shows the number of tasks at the start of the project and the number of remaining tasks at the end of the selected period. You can see only uncompleted tasks using the option, **Show uncompleted tasks only**.

> The burn down chart does not show canceled tasks.

To open the burn down chart, in the **Home** module, click **Tasks**.

Timesheet Working Hours



The **Working Hours** in a timesheet lets employees enter their daily working time for a week.

This helps employers track the working hours and breaks of employees.

Configuring Timesheet Working Hours

To configure timesheet working hours:

1. In the **System Administration** tool, click**Structures** > **Timesheets**.
2. Under **Working Hours**, select **Enabled**.

   Working Hours appears in all new and existing timesheets.

3. Specify **Working Hours** options:

Option	Description
Enabled	Lets you specify working hours options.
Working hours are required	Allows total hours for a day other than zero. If you select **Show Type row** and if the value for **Exclusions** matches the type you enter, you can enter total hours as 0.
Exclusions	Lets you enter values corresponding to the **Type**.
Configure Templates	Creates workweek templates to reduce the number of entries in a template.
Configure Comments	Lets you add comments for each day and time slot.
Show Type row	Displays a **Type** row for you to select a type from the list.
Configure Types	Specifies the required **Type** options.
Working Days	Specifies the days in the workweek.

Configuring Templates

You can create and configure workweek templates to reduce the number of entries in a template.

To configure templates:

1. Click **Configure Templates**.
2. In the Templates dialog box, click **New**.
3. In the Template Properties dialog box, enter a name for the template.
4. Optional: Select **Default** to specify this template as the default whenever you create a new timesheet.

5. Enter time values in each day or click arrows to select values for the following:

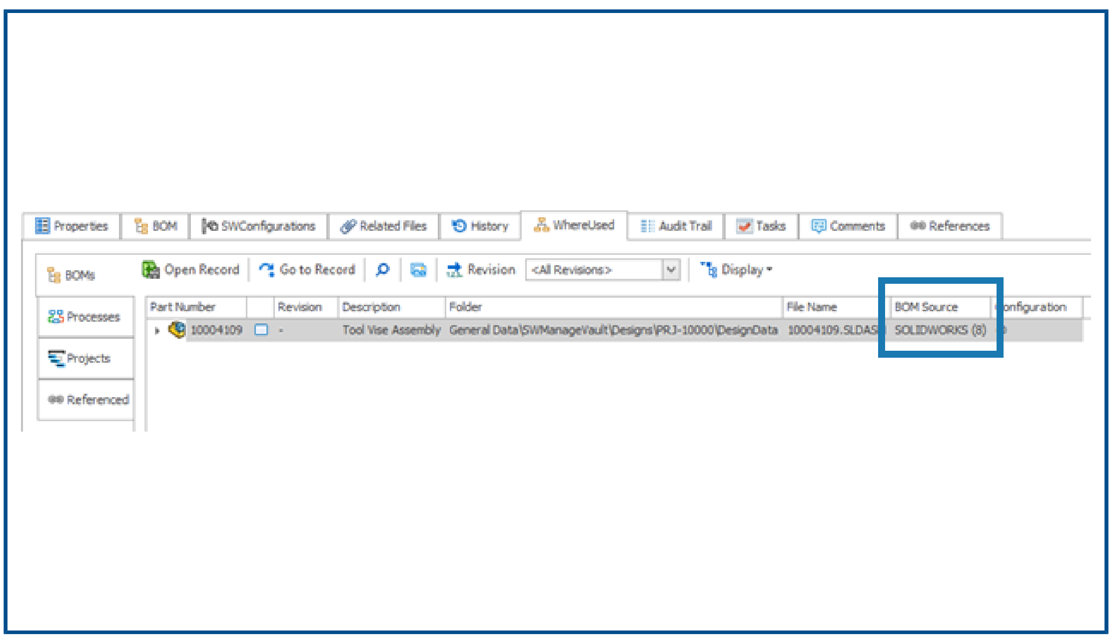
Option	Value	Format
Start	Work start time for a day	24-hour
Pause duration	Break time during the day	hh:mm
End	Work end time for a day	24-hour
Total Time	Calculated based on the other values you specify	

Configuring Comments

You can add comments for each day and time slot.

Administrators can add comments by clicking **Configure Comments** and entering values in a list format. You can modify a comment from the list or enter new text.

Bill of Materials Quantity



You can see the number of component BOMs on the Where Used tab.

On the Where Used tab, under **BOM Source**, you can see the number of BOMs displayed in parenthesis. In previous releases, you had to open the parent record to search for component BOMs.

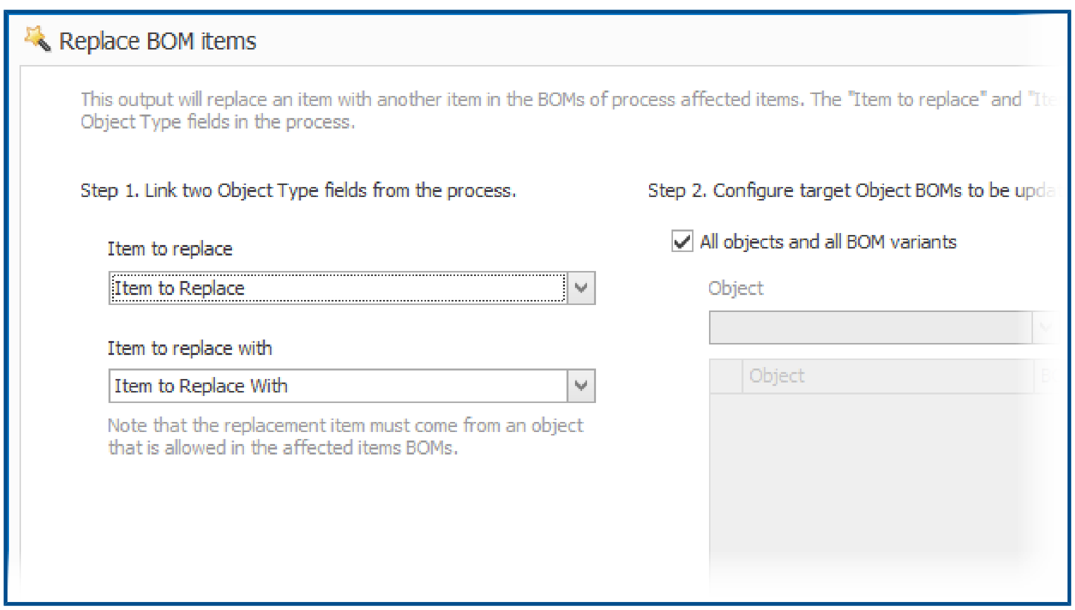
Adding Custom Columns to the Where Used Tab

You can define custom field columns on the Where Used tab. This displays the custom field information with the standard system fields.

To add custom columns to the Where Used tab:

1. Log in to the SOLIDWORKS Manage desktop client as an administrator.
2. Open the property card for a record in the object to which you want to add a custom column.
3. Select the Where Used tab.
4. Select the BOM tab.
5. Click (Where Used toolbar).
6. In the Custom Fields dialog box, click **New**.
7. In the Field Properties dialog box, enter a **Display Name**.
8. Click **Type** and select a data type.
9. Click a cell in the **Field** column of the required object and select a field to display.
10. Repeat the previous step for required objects to get field values from.
11. Click **Save and Close**.
12. Add additional custom fields as required.

Process Output for Replacing BOM Items



In BOMs, you can replace a record with another record.

You can replace a line item used in many assemblies without editing each assembly. The output is called **Replace BOM items**. To use **Replace BOM items**, you need two object type fields: one object type field holds a source item and other holds a target item.

Mass replace works only for record objects and not for SOLIDWORKS CAD references.

Enabling Mass Replace in a Process

To enable mass replace in a process:

1. In the System Administration tool, under **Structures** > **Processes**, edit an existing **Process** object.
2. In the Process Wizard, open the **Fields** page.
3. Click **New Field** to create a new object type field.
4. Enter a display name and select **Object Type** as the field type.
5. Click **Finish**.
6. In the Object Type Field Properties dialog box, click **Next**.

   Do not select **Allow Multiple Items**. You can replace a single record only.

7. Click **Next** again.
8. On the Select Object(s) page, select the objects where the items to replace come from.
9. Click **Next**.
10. On the Select Columns page, specify options.
11. Click **Next**.
12. On the Choose User Rights page, specify access permissions for the field.
13. Click **Finish**.
14. Repeat steps 3 to 13 to add an object type field to hold the target item.
15. In the Process Wizard, open the Workflow Properties wizard.
16. Select the stage where you want to replace the record.
17. Click **Outputs** and click **Add**.
18. In the Outputs dialog box, in **Select Type**, select **Replace BOM items** and click **Save**.
19. In the Replace BOM items dialog box, under **Step 1**, select the object type field for the source item in **Item to replace** and the target object type field in **Item to replace with**.
20. Under **Step 2**, specify the behavior for the target parent objects to update.

    Select the parent objects to add as affected items in the process.

21. Click **Save and Close**.

Replacing BOM Items

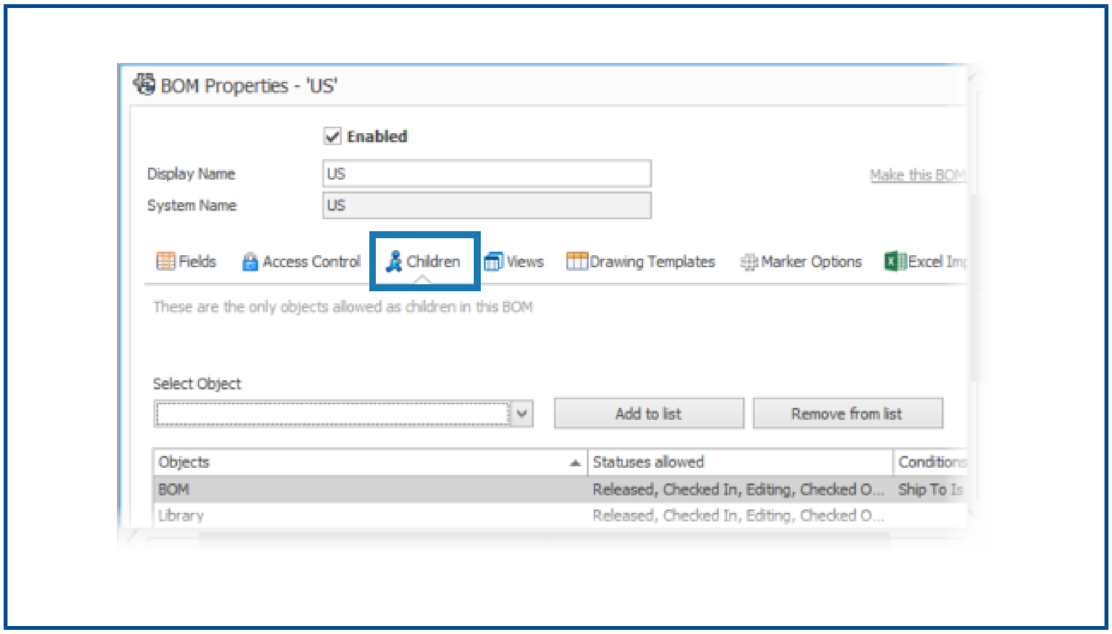
To replace BOM items:

1. In SOLIDWORKS Manage, navigate to the process object of the **Replace BOM items** output.
2. Click **New** (Main toolbar).
3. Select the item to replace and the item to replace with in the object type fields.
4. On the Affected Items tab, click **BOM replacements analysis**.
5. In the Replacement Analysis dialog box, select the required parent records to have the items replaced.

6. Click **Add to list** to close the dialog box and add the selected records to the affected item list.
7. Move the process through its workflow past the stage where you added the **Replace BOM items** output.

   To see the updated BOMs, open the record for an affected item.

Adding Child Conditions to BOMs



You can add conditions to restrict the addition of child item records based on the record's status and field values. This helps apply company policies for adding records to BOMs.

To add child conditions to BOMs:

1. In the System Administration tool, under **Structures**, select an object and click **Edit** .
2. Open the Bill of Materials wizard.

   If you edit a record or document object other than a SOLIDWORKS PDM object, click the BOM tab.

3. Select the **Bill of Material** object in the list and click **Edit** .
4. In the BOM Properties dialog box, click the Children tab.
5. Click the cell under **Statuses allowed** for the BOM variant and select the required status.
6. In the **Conditions** column for a BOM object, click ellipses in the cell to add conditions that restrict items to add to the BOM.
7. In the Do not allow adding items to BOM if these conditions are met dialog box, enter the required conditions and warning message.
8. Click **Save and Close**.

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SOLIDWORKS Simulation

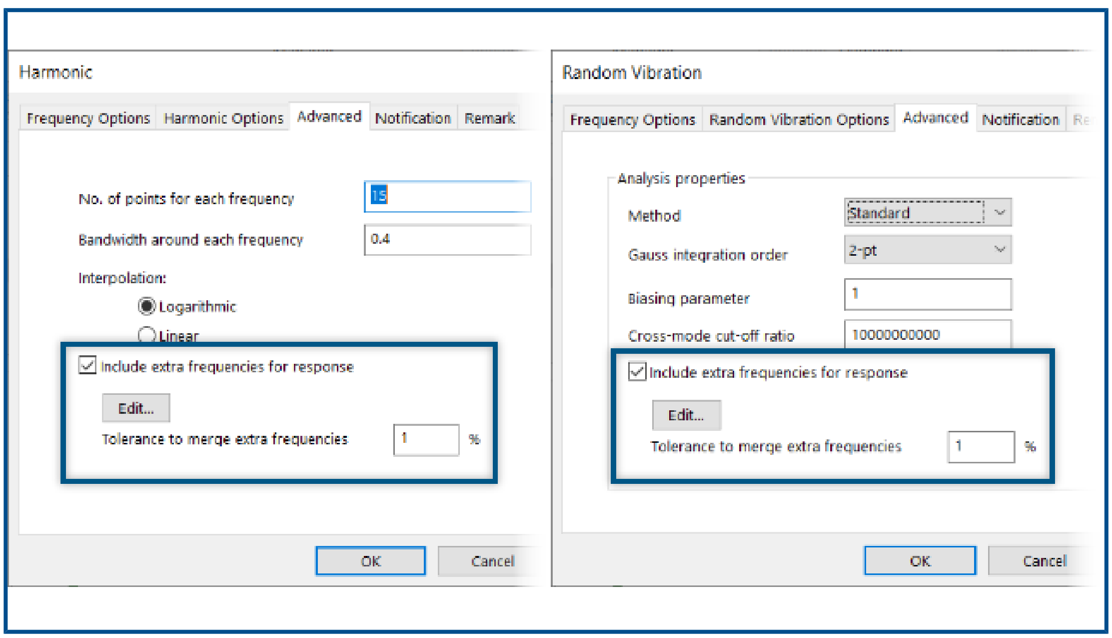
This chapter includes the following topics:

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.

3DEXPERIENCE SOLIDWORKS Simulation Designer Role (2024 SP1)

3DEXPERIENCE SOLIDWORKS roles, such as 3DEXPERIENCE SOLIDWORKS Standard, 3DEXPERIENCE SOLIDWORKS Professional, and 3DEXPERIENCE SOLIDWORKS Premium, now support SOLIDWORKS Simulation Standard, SOLIDWORKS Simulation Professional, SOLIDWORKS Simulation Premium, and SOLIDWORKS Motion licenses.

Extra Frequencies for Harmonic and Random Vibration Response (2024 SP1)

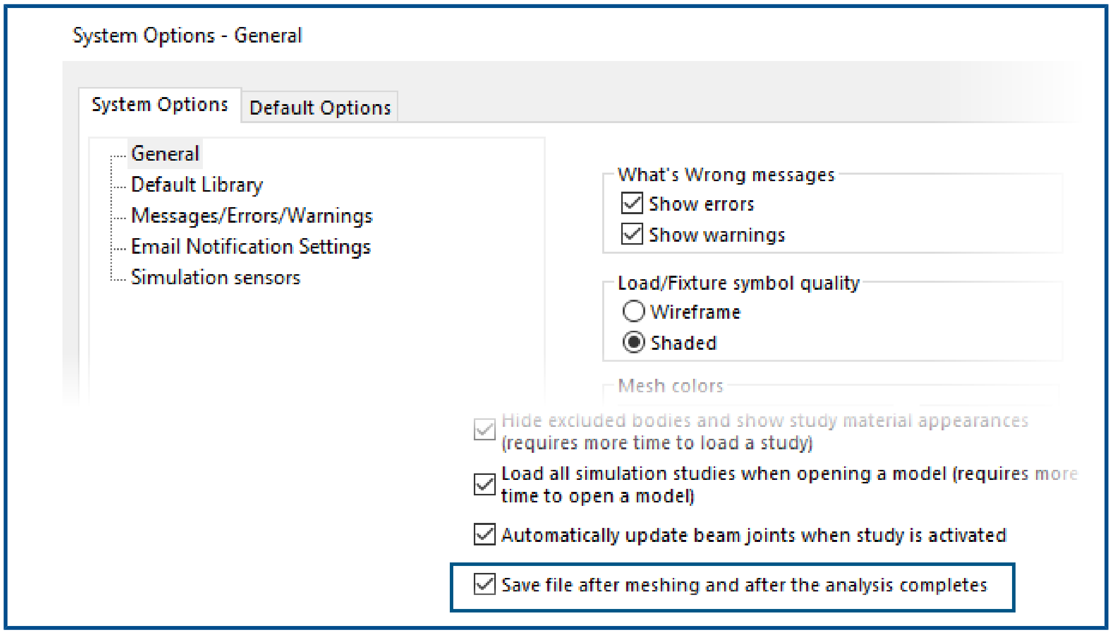


You can include up to 20 extra frequencies of interest when calculating the response parameters for harmonic and random vibration studies.

From the **Harmonic** > **Advanced Options** or **Random Vibration** > **Advanced** dialog boxes, select **Include extra frequencies for response**.

For more information, see *Harmonic - Advanced Options* or *Random Vibration - Advanced*.

Automatic Saving of a Model File



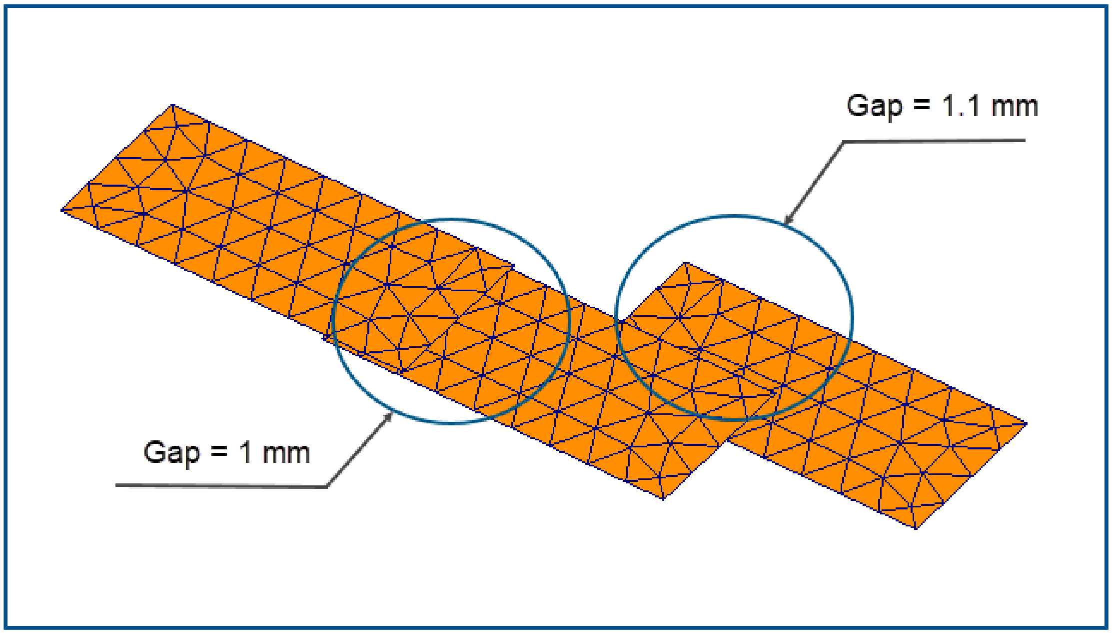
You can save a model file after meshing and after the analysis completes.

To turn on automatic saving of a model file:

From the **System Options** > **General** tab, select **Save file after meshing and after the analysis completes**.

Saving a model file automatically after meshing and after the completion of analysis prevents data loss in case of unexpected system crashes or power outages.

Bonding Interactions for Shells

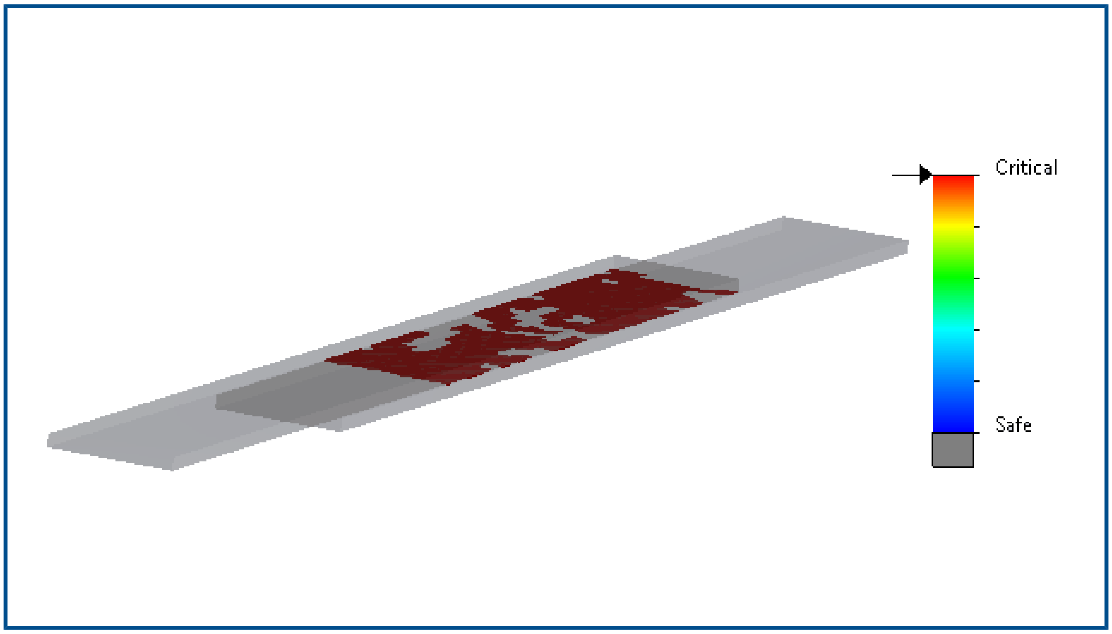


The enforcement of bonding interactions between sets of shell elements that have a physical gap is more robust.

The image above shows a model with three shell surfaces. One pair of shells has a physical gap of 1mm, while the second pair of shells has a gap of 1.1mm. By setting a user-defined **Maximum gap** for bonding to 1mm (the maximum gap between geometric entities to enforce local bonding interactions), only the pair of shells with a gap of 1mm should be bonded.

An improved algorithm enforces the proper bonding interactions irrespective of the mesh size. In previous releases, if you applied a coarse shell mesh to the three surfaces, the algorithm erroneously enforced a bonding interaction to the second shell pair with a 1.1mm gap.

Convergence Check Plot



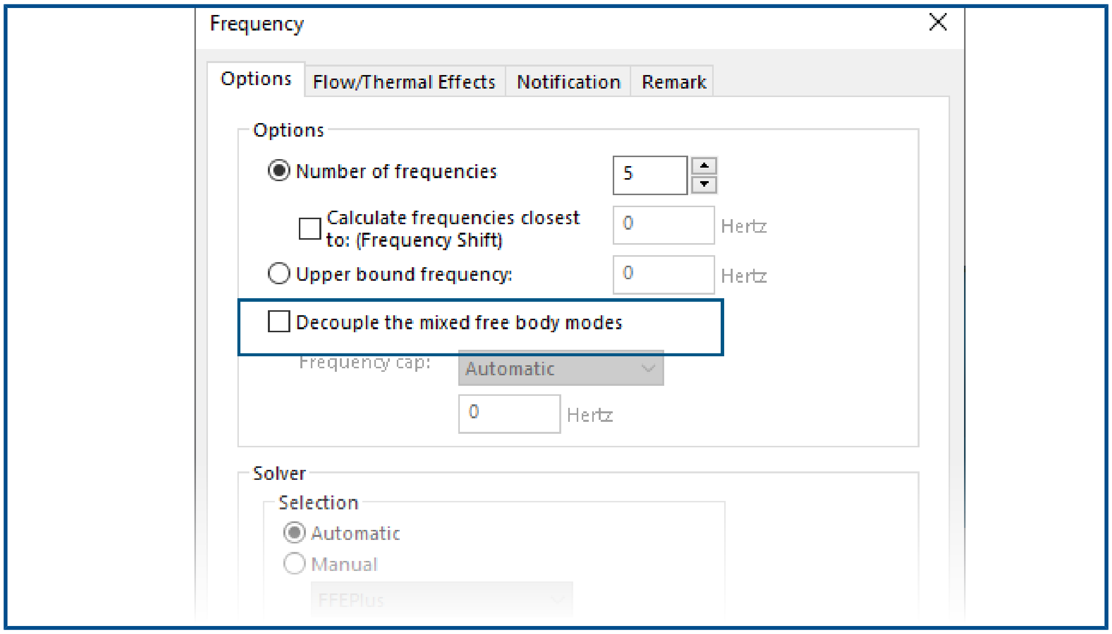
The **Convergence Check Plot** detects regions of the model where the solver has encountered contact convergence issues.

To access the Convergence Check Plot:

Do one of the following:

- Click **Diagnostic Tools** > **Convergence Check Plot** (Simulation CommandManager).
- In a simulation study tree, right-click **Results** and click **Convergence Check Plot**.

Decoupling Mixed Free Body Modes

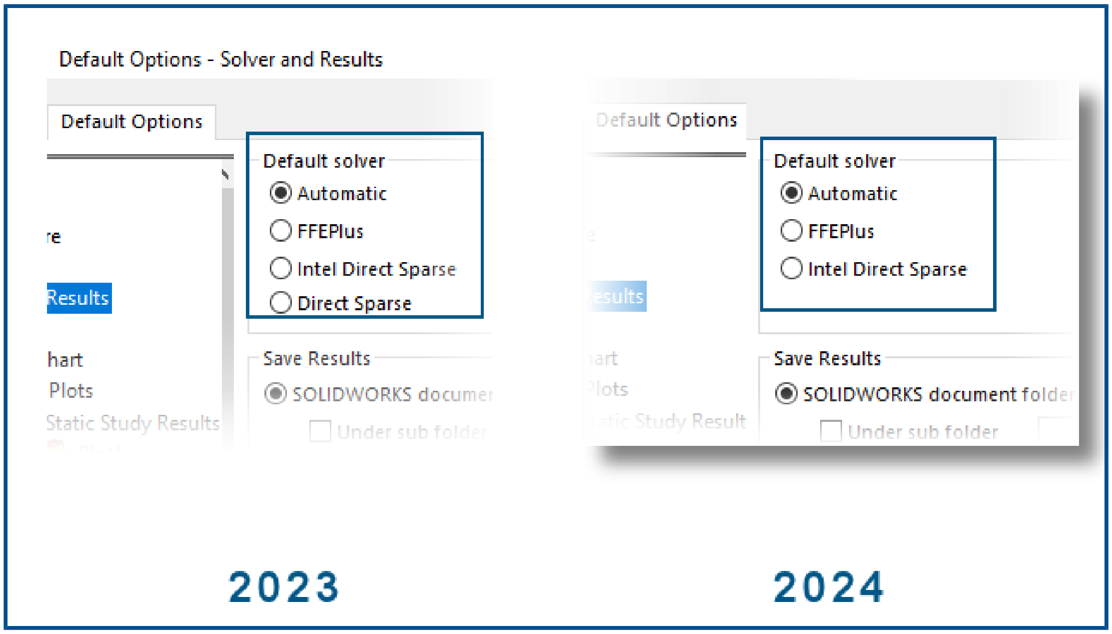


An algorithm can detect and decouple the mixed free body modes while calculating mode shapes.

From the Study Properties dialog box, select **Decouple the mixed free body modes**. In cases where mixed free body modes exist in a model, the algorithm resolves the mixed motion associated with a rigid body mode and provides the precise mode shape of a rigid body mode.

> The option to decouple the mixed free body modes is available in Frequency, Linear Dynamic, Harmonic, Random Vibration, and Response Spectrum Analysis studies.

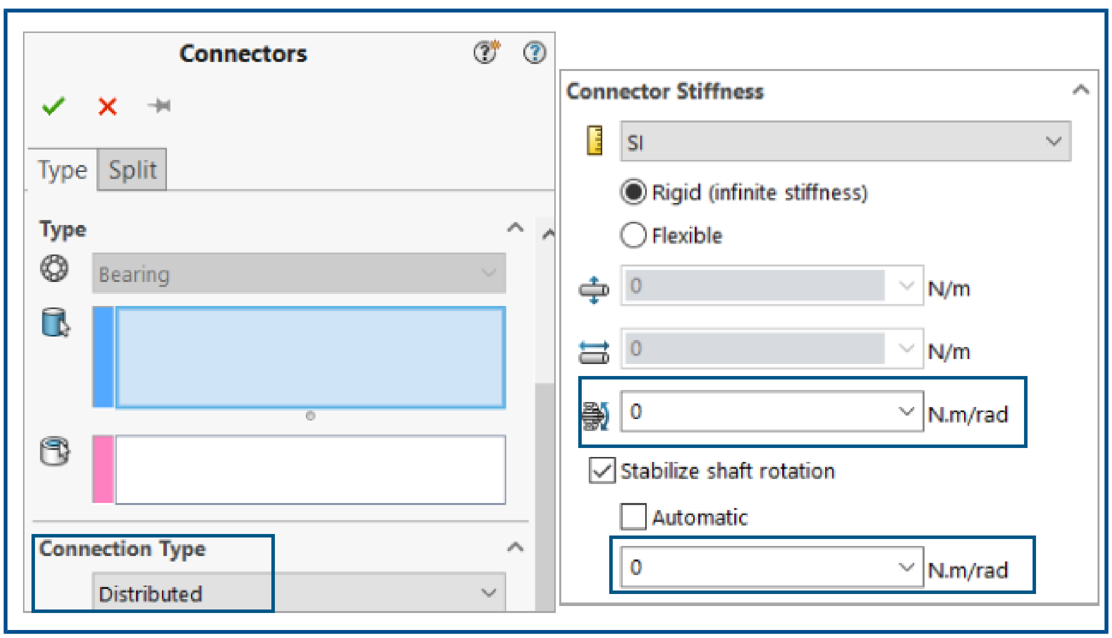
Direct Sparse Solver Retired



The Direct Sparse solver is removed from the list of solvers for simulation studies.

For legacy studies that use the Direct Sparse solver, SOLIDWORKS Simulation uses the Intel® Direct Sparse solver.

Enhanced Bearing Connectors



The introduction of **Distributed** coupling and **Tilt stiffness** enhances the formulation of bearing connectors.

The bearing connector is enhanced as follows:

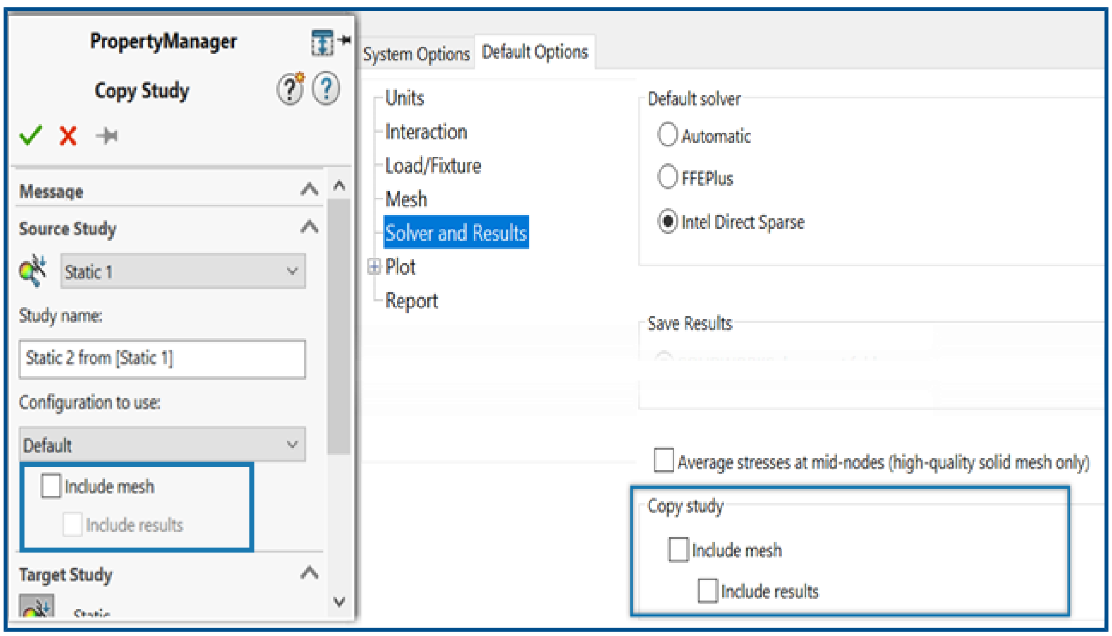
- A **Distributed** type is added to the connector's **Connection Type** options. For a new bearing connector definition, the default **Connection Type** is **Distributed**.
- The addition of **Tilt stiffness** accounts for the bending stiffness of the shaft.

> To simulate the **Allow Self-alignment** option, which was available in prior releases, set the **Tilt stiffness** to zero.

- You can apply a user-defined torsional stiffness to stabilize the shaft rotation.

The bearing connector enhancements are available for Linear static, Frequency, Buckling, and Linear dynamic studies.

Excluding Mesh and Results When Copying a Study

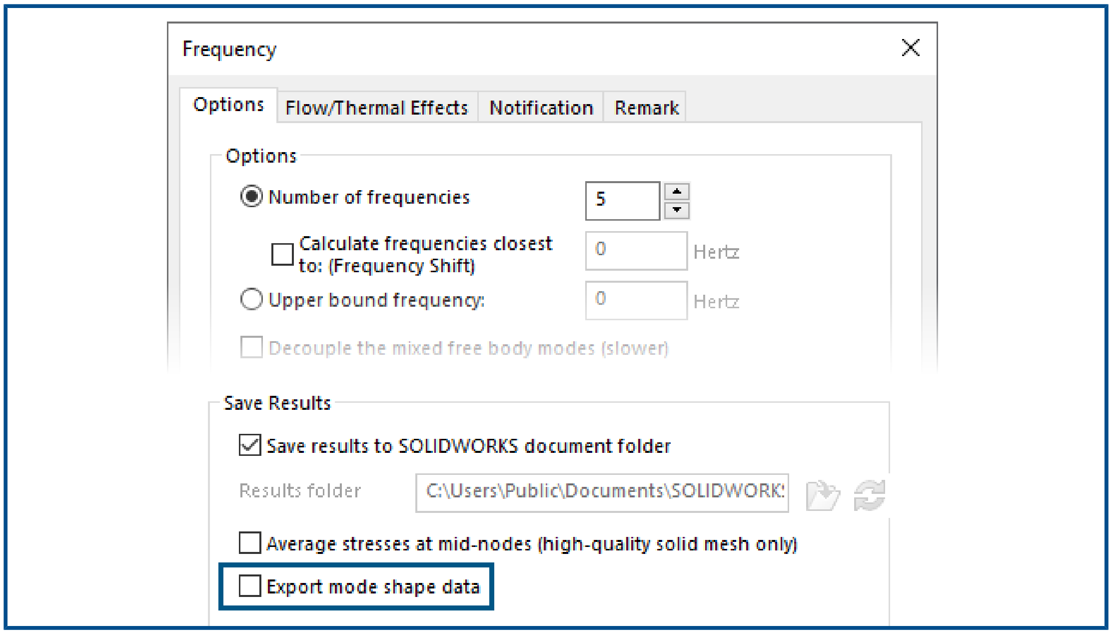


You can save time by excluding mesh and results data when copying a simulation study to a new study.

You can specify global default settings to include or exclude mesh and results when copying a study from the **Default Options** > **Solver and Results** > **Copy study** dialog box.

For individual studies, you can modify the default settings for **Include mesh** and **Include results** in the Copy Study PropertyManager.

Exporting Mode Shape Data

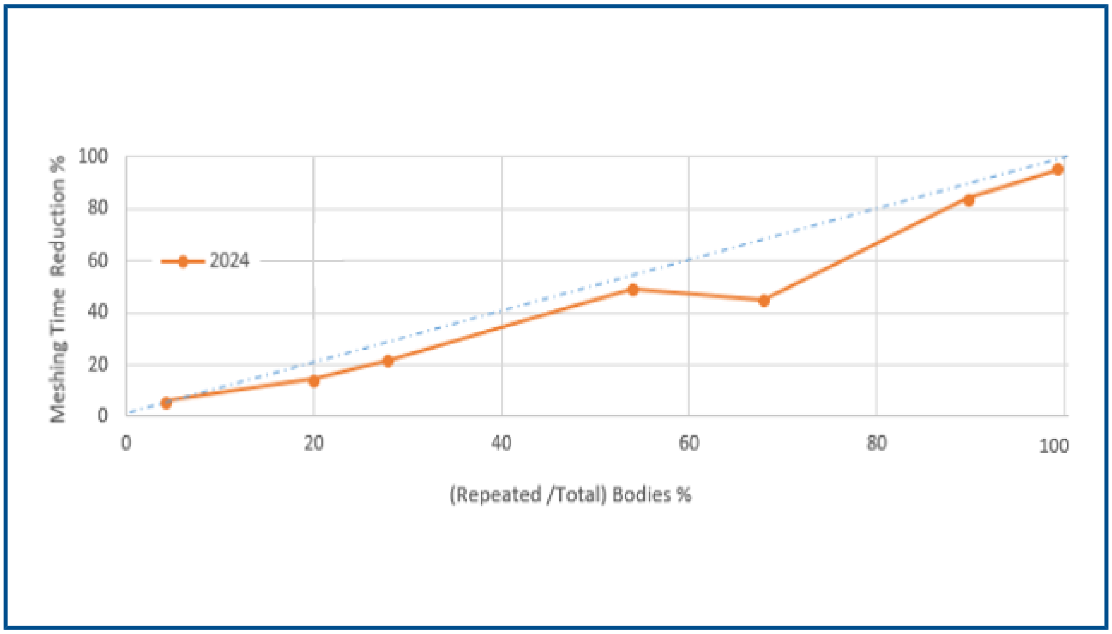


You can export mode shape data to the study's `study_name.out` file.

From the **Frequency** > **Options** dialog box, select **Export mode shape data**.

The mode shape data are saved to the study's `.out` file, located in the **Results** folder.

Mesh Performance



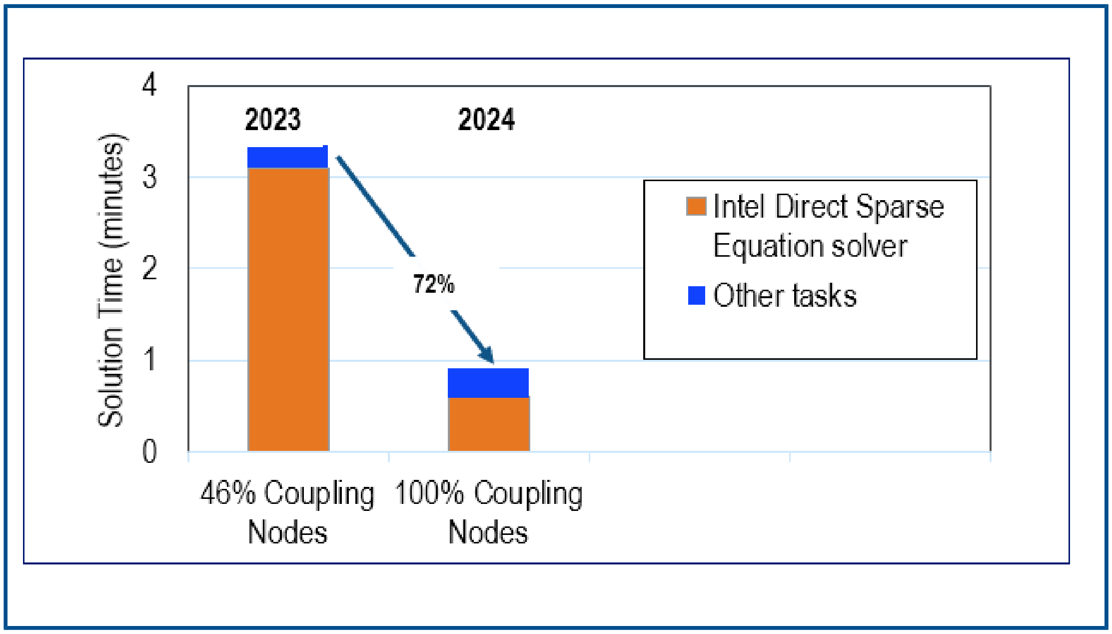
The meshing time with the Blended curvature-based mesher is reduced for assemblies that have multiple identical parts.

> This mesh enhancement is available with the SOLIDWORKS Simulation Premium and SOLIDWORKS Simulation Professional licenses.

An improved mesh algorithm based on the Blended curvature-based mesher identifies identical parts that are repeated in an assembly. The algorithm reuses the same mesh for the identical parts instead of meshing each of them independently, thus saving meshing time.

To use the improved mesh algorithm, from the **Default Options** > **Mesh** dialog box, select **Reuse mesh for identical parts in an assembly (Blended curvature-based mesher only)**.

Performance Enhancements



Several feature enhancements improve the performance and accuracy of simulation studies.

- Results from studies with remote displacements or remote rotations that are applied to large faces with the **Distributed** connection are more accurate.

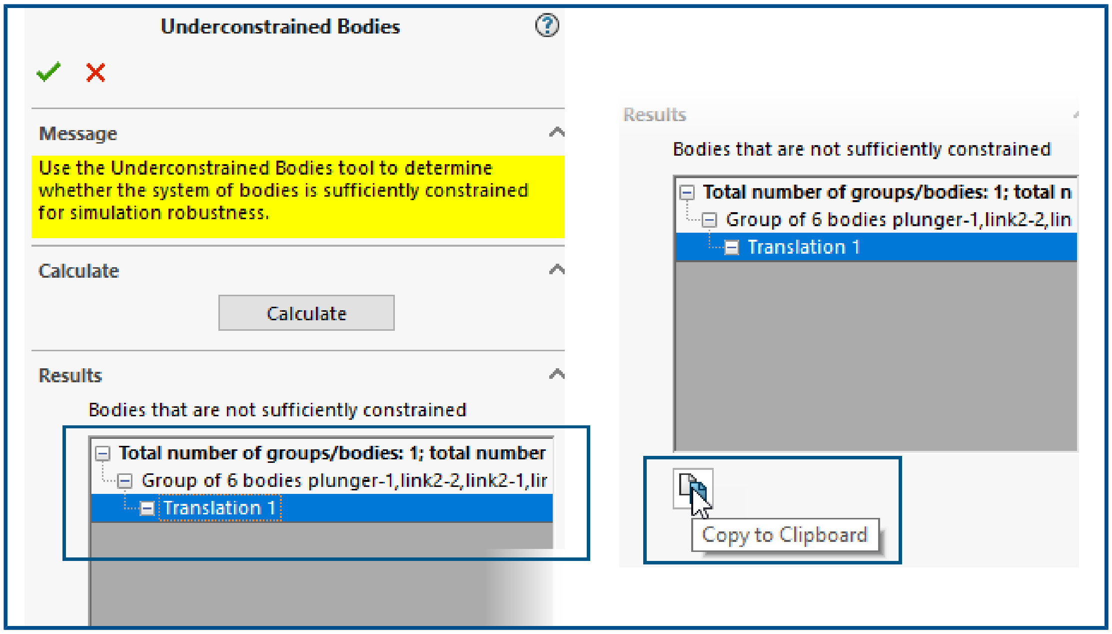
  The solution time for these studies is shorter with the Intel Direct Sparse solver. In previous releases, when the number of coupling nodes was very large, only a subset of the coupling nodes participated in the distributed coupling constraints. In SOLIDWORKS Simulation 2024, the distributed coupling constraints for remote displacements or remote rotations include all coupling nodes.

  The image illustrates the performance gain of the Intel Direct Sparse solver for a model that has a remote displacement applied with distributed coupling to approximately 29,600 coupling nodes.

  > The solution time with the FFEPlus iterative solver for similar studies is not faster in SOLIDWORKS Simulation 2024. However, the stress results are more accurate because all coupling nodes are considered in the distributed coupling formulation.

- Running larger linear dynamic studies is more efficient. The stress calculation of larger linear dynamic studies is optimized because of improved memory allocation by the solver.
- Improved memory estimate, allocation, and management by the solver allows the completion of large surface-to-surface bonded interaction sets that previously failed because of insufficient memory. This improvement applies to the SOLIDWORKS Simulation Professional and SOLIDWORKS Simulation Premium licenses.
- The total solution time for most static and thermal studies solved with the Intel Direct Sparse solver is reduced by more than 10%. Updating the Intel Direct Sparse solver with the new Intel MKL libraries and using parallel reordering with the variable block sparse row (VBSR) format improved the solver's performance.

Underconstrained Bodies Detection



There are several usability enhancements for the Underconstrained Bodies PropertyManager.

- You can copy the results of the underconstrained bodies detection tool to the clipboard.
- The list that shows the bodies that are not sufficiently constrained in the **Results** section is expandable for improved readability.
- It takes less time to show the animations of underconstrained bodies. The graphics quality of the animations that highlight underconstrained bodies is improved.

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SOLIDWORKS Visualize

This chapter includes the following topics:

- **Transformative Performance with Stellar Render Engine (2024 FD02)**
- **Turkish Language Support (2024 FD02)**
- **File Export Formats (2024 SP1)**
- **Enhanced Capabilities for Creating Compelling Appearances**

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.

Transformative Performance with Stellar Render Engine (2024 FD02)

Significant improvements to the Stellar render engine have measurably enhanced the render performance in SOLIDWORKS Visualize.

This functionality enhances the Viewport experience, particularly for larger resolutions and high-end GPUs.

Benefits: Interactions with the Viewport are smoother and more interactive. This improvement also results in a more responsive user interface.

Turkish Language Support (2024 FD02)

SOLIDWORKS Visualize Connected offers full support for the Turkish language in the user interface.

Benefits: If you install SOLIDWORKS Visualize Connected on a Turkish version of Windows, it automatically configures to Turkish.

You can also change the language in **Tools** > **Options** > **User Interface** > **Language**.

File Export Formats (2024 SP1)

The `.GLTF`, `.OBJ`, and `.FBX` file formats support the exporting of DSPBR appearance parameters.

The `.GLTF` and `.OBJ` file formats export the following DSPBR parameters and associated textures:

- Albedo

- Metallic
- Roughness
- Alpha
- Normal

The `.FBX` file format exports these DSPBR parameters:

- Diffuse color
- Diffuse texture

Enhanced Capabilities for Creating Compelling Appearances

SOLIDWORKS Visualize uses Dassault Systèmes' Enterprise PBR Shading Model (DSPBR) to closely replicate the realistic appearance of metal, glass, plastic, and other surfaces.

DSPBR is an appearance model for physically based rendering, supported by many renderers in the **3D**EXPERIENCE® platform. The shading model is easy to use and renderer independent. It combines parameters to describe metallic and nonmetallic appearances, including transparency for thin-walled and volumetric objects. It also provides effects, such as emission, clear coat, metallic flakes, and sheen, to cover a wide range of appearances.

SOLIDWORKS Visualize provides appearances for an expanded range of material types and subtypes. The full **Enterprise PBR Shading Model** consists of more than 30 parameters, which can be complex. The software organizes these parameters into categories that are relevant to specific **Appearance Types**. This simplifies the user interface and enhances usability while keeping unnecessary parameters hidden. The **Appearance Types** available are **Car Paint**, **Metal**, **Basic**, **Emissive**, **Textile**, **Leather**, **Wood**, **Glass**, and **Plastic**.

Enhancements include:

- A simplified interface for selecting appearance types and optimizing their parameters. You can select appearance types from a list or by clicking thumbnail images.

- The ability to adjust textures and texture maps for almost all parameters, with greater control and fidelity.
- The ability to combine normal and displacement maps and to apply vector displacement.
- Sample projects and other assets are updated and improved for showcasing DSPBR appearances. Additional appearances and assets are available in the Cloud content library.

You do not need to convert existing files to the DSPBR appearances. You can continue working with files created with legacy appearance types or convert them to the DSPBR types. New files must use the DSPBR appearance types.

Parameters for Basic Appearance Type

The **Basic Appearance Type** is made up of a few parameters that are sufficient to simulate the most commonly used real-world appearances.

If you are new to applying appearances, start with **Basic**. Descriptions for all the DSPBR appearances and how to apply textures are available in the SOLIDWORKS Visualize help.

Parameter	Description	Value
Albedo	Specifies the overall RGB color of a material. You can use it to apply color to thin walled transparent materials.	RGB color
Metallic	Determines the level of metallicness of a surface.	Decimal. [0..1]
Roughness	Controls the level of shininess or roughness of a surface.	Decimal. [0..1]
Normal	Adds the appearance of details such as bumps and dents to the surface of a model without changing the size of the geometry.	Texture
Displacement	Modifies the position of surface points using a texture that specifies the length and direction of displacement for each point.	Texture
Cut-Out Opacity	Adds a texture of holes to a surface without adding extra polygons to the geometry.	Decimal. [0..1]

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SOLIDWORKS CAM

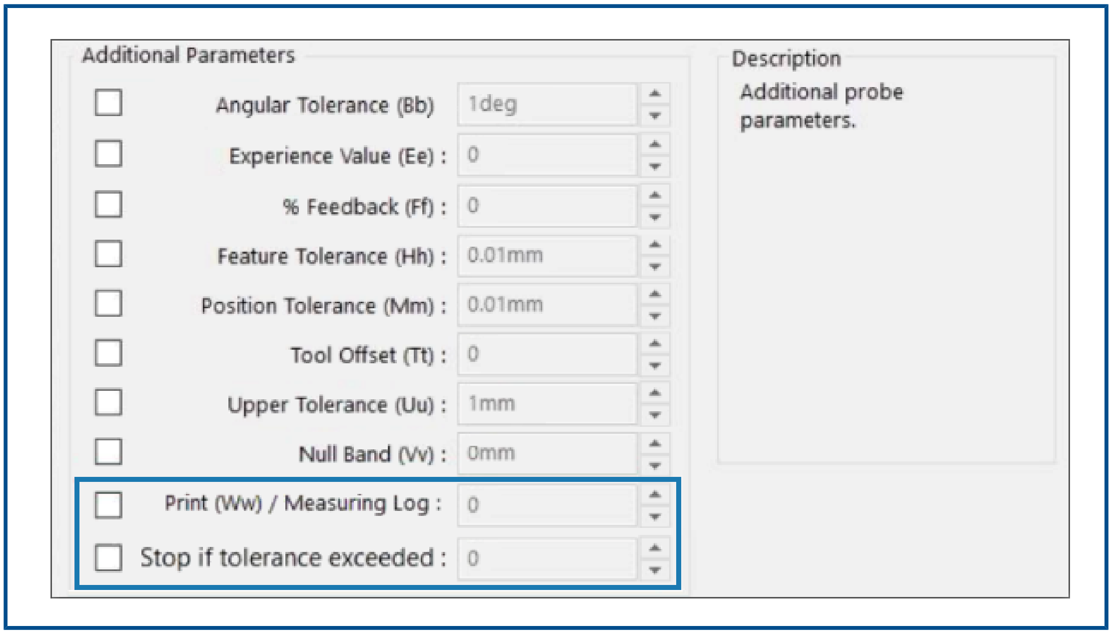
This chapter includes the following topics:

- **Additional Probe Cycle Parameters**
- **Canned Cycle Threading for Reverse Cuts**
- **Correct Feed/Speed Data for Parts Comprising Assemblies**
- **Heidenhain Probe Type**
- **End Conditions for Islands in the 2.5 Axis Feature Wizard**
- **Leadin and Leadout Parameters for Linked Contour Mill Operations**
- **Minimum Hole Diameter for Thread Mill Operations**
- **Post Processor Path**
- **Probe Cycles**
- **Probe Tool Output Options**
- **Probing Cycles in Assembly Mode**
- **Setup Sheets**
- **Shank Types for Mill Tools**
- **Tool Select Filter Dialog Box**
- **Tool Selection - Flute Length**
- **Tool Selection - Tool Crib Priority**

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.

Additional Probe Cycle Parameters



The Additional Probe Cycle Parameters dialog box contains options for **Stop if tolerance exceeded** and **Print (Ww) / Measuring Log**.

Stop If Tolerance Exceeded

If a probe cycle goes beyond tolerance limits, the **Stop if tolerance exceeded** parameter specifies whether to interrupt the program and display the details of the violation.

Values you can specify for this parameter:

- `0`. Does not interrupt the machining program or display the violation details if tolerance limits are violated.
- `1`. Interrupts the machining program and displays the violation details on the controller.

The command associated with this parameter in the posted code is

```
Q309=1 ;PGM STOP TOLERANCE
```

Print (Ww) / Measuring Log

The **Print (Ww)** parameter is renamed to **Print (Ww) / Measuring Log**.

The functionality for **Print (Ww) / Measuring Log** depends on the **Probe Type** selected.

Probe Type	Print (Ww) / Measuring Log Functionality
Renishaw	Indicates whether the data is output in the post-processed code.
Heidenhain	Indicates whether to create, save, or display the measuring log.

Values you can specify for this parameter:

- 0. Does not create the measuring log.
- 1. Creates the measuring log and saves it to the controller.
- 2. Interrupts the NC program and displays the measuring log.

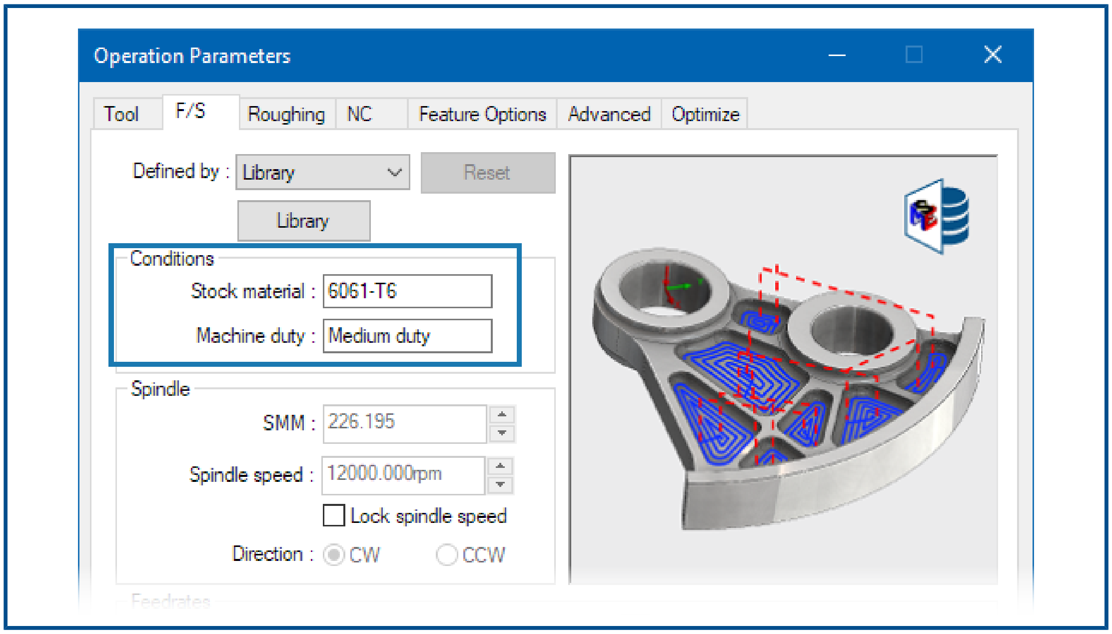
Canned Cycle Threading for Reverse Cuts

For threading operations, SOLIDWORKS CAM supports the **Canned cycle output** option for reverse cut types.

In the Operation Parameters dialog box, on the Thread tab, under:

- **Cut type**, select **Reverse**.
- **Program point**, select **Canned cycle output**.

Correct Feed/Speed Data for Parts Comprising Assemblies

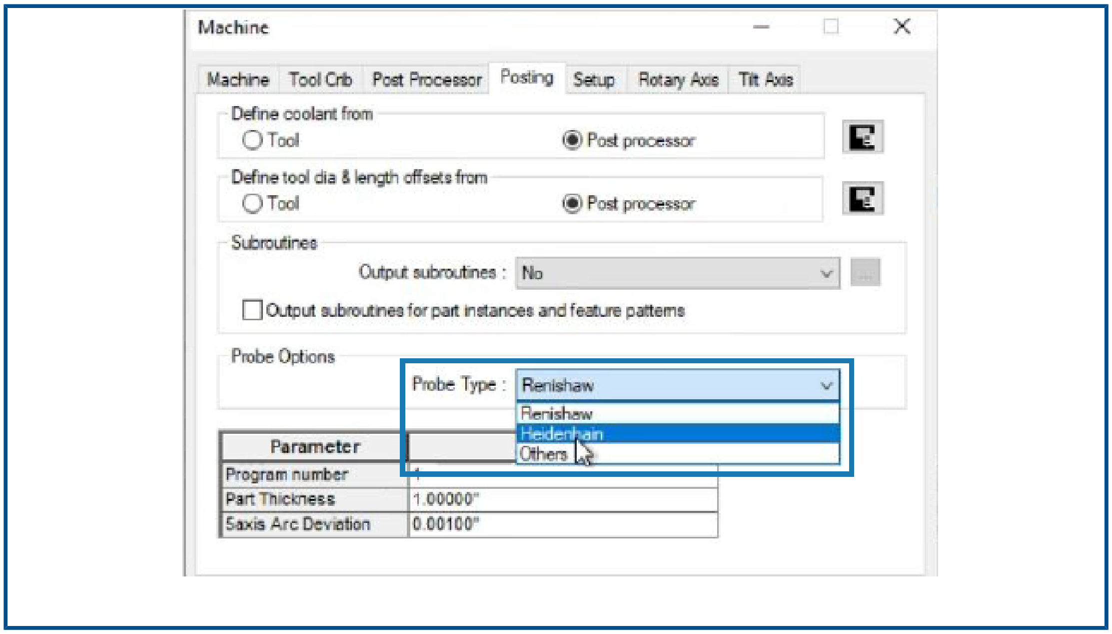


In Assembly mode, if the different parts or the multiple instances of a part comprising an assembly have different stock materials, then for each part or instance, the correct stock material appears.

The associated stock material appears in the Operation Parameters dialog box on the F/S tab for **Stock material**. The Feed/Speed Editor uses the **Stock material** for feed/speed calculation.

In previous releases, in Mill Assembly mode, when an assembly contained parts that had different stock materials or split part instances had different stock materials, the feed/speed computations were often inaccurate. This occurred because SOLIDWORKS CAM only considered the stock material assigned to the first part listed in the Part Manager for feed/speed computation. SOLIDWORKS CAM assigned the calculated the feed/speed values to the other parts that constituted the assembly though they had different stock materials. This resulted in erroneous feed/speed values.

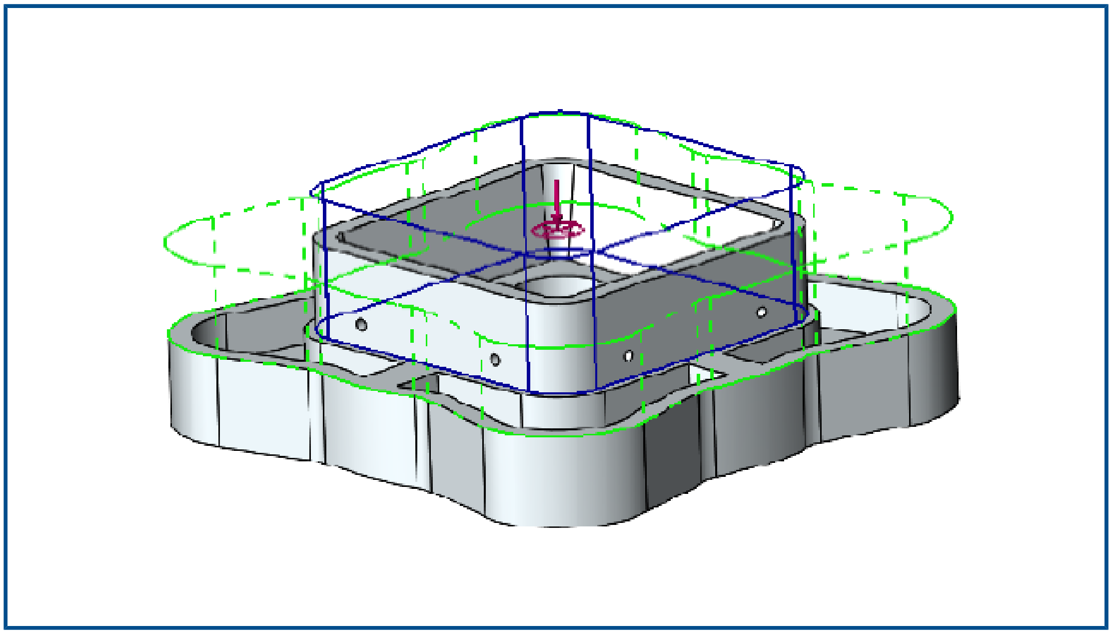
Heidenhain Probe Type



SOLIDWORKS CAM supports probing operations on machine tools that use Heidenhain controllers.

In the Machine dialog box, on the Posting tab, under **Probe Options**, in **Probe Type**, select **Heidenhain**.

End Conditions for Islands in the 2.5 Axis Feature Wizard

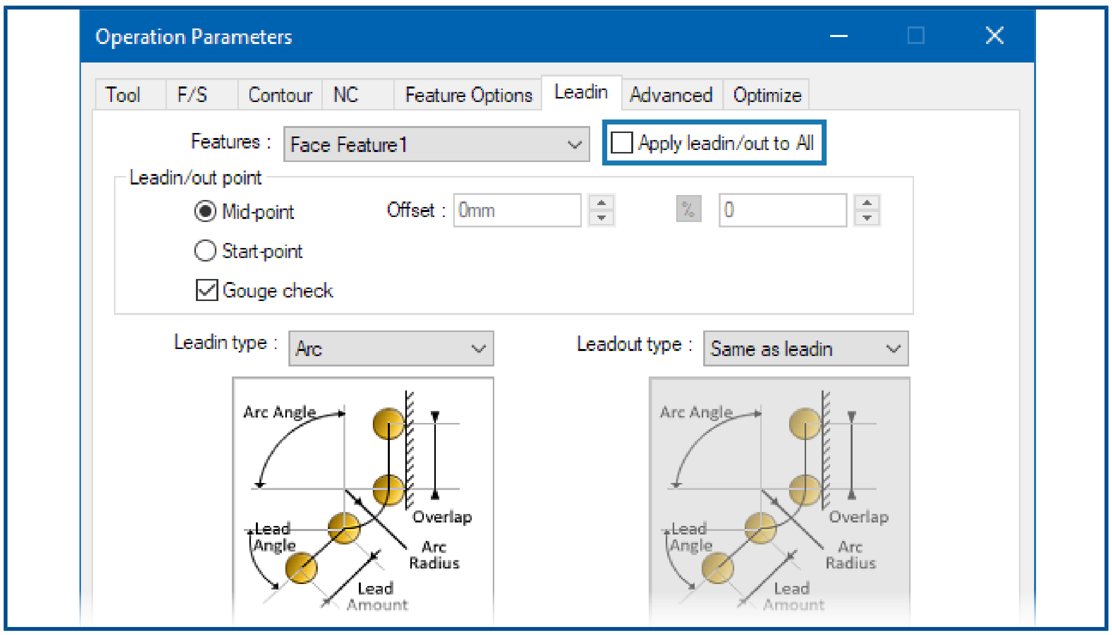


You can define the height of islands for 2.5 Axis features in two directions.

In previous releases, SOLIDWORKS CAM automatically specified island height from the topmost point of the island face to the bottom of the feature. If the island face was a different height than the top face of the feature, the resultant island was shorter compared to the feature height. You could not increase the island height in the other direction to match the feature height.

In the 2.5 Axis Feature: Island Entities PropertyManager, you can specify island height under **End condition - Direction 2**. You can define the height in Z+ and Z- directions. The direction associated with **End Condition - Direction 2** is opposite to the bottom profile of the island feature.

Leadin and Leadout Parameters for Linked Contour Mill Operations

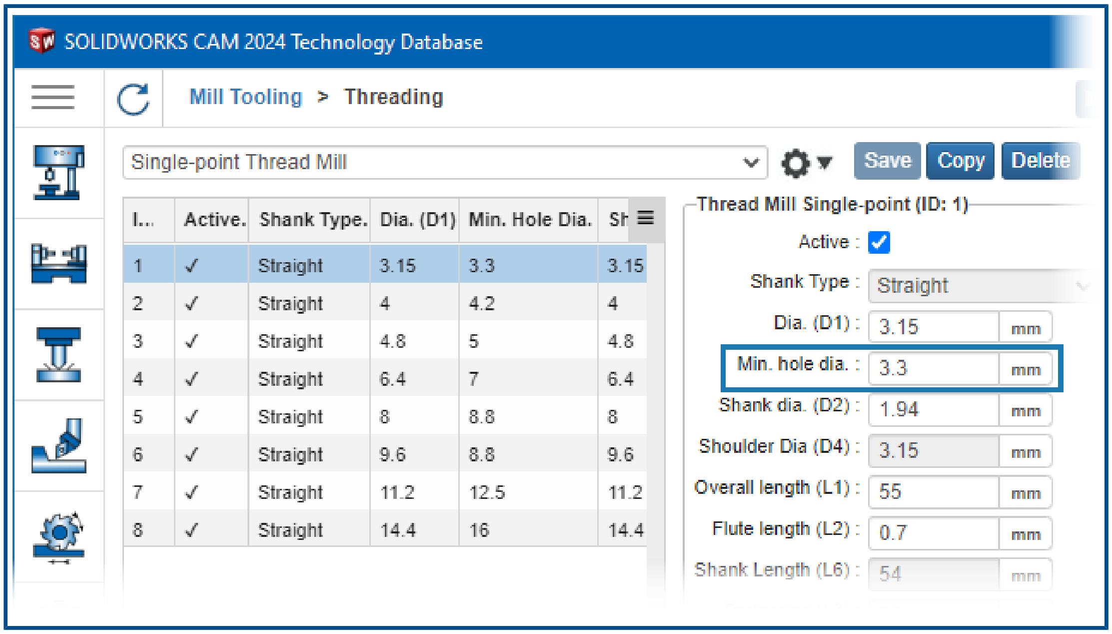


For linked Contour Mill operations, you can specify an option to copy the **Leadin** and **Leadout** parameters of the first Contour Mill operation to the other linked operations.

In the Operation Parameters dialog box, on the Leadin tab, select **Apply leadin/out to All**. SOLIDWORKS CAM does not link these operation parameters because they are feature-specific:

- **Leadin/out point**
- All parameters under **Links between**

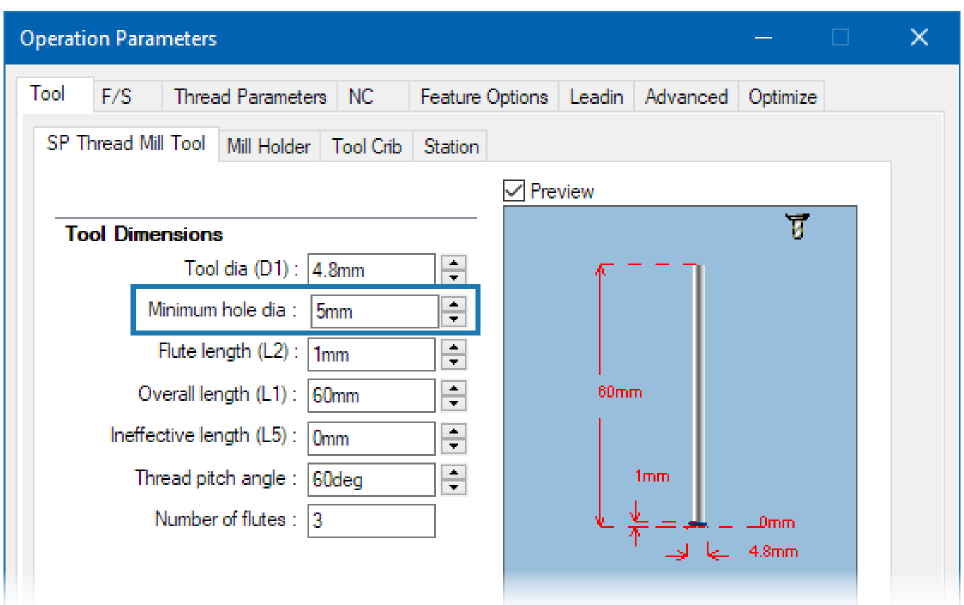
Minimum Hole Diameter for Thread Mill Operations



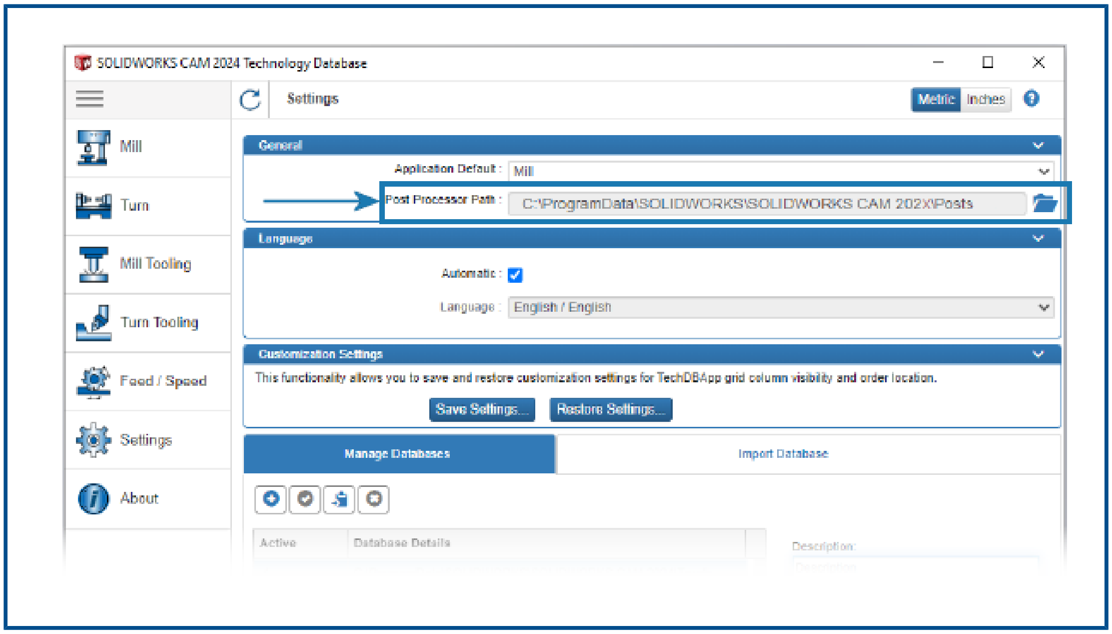
You can specify the minimum hole diameter for thread mill operations. In previous releases, this parameter was read-only.

In the Technology Database (TechDB), on the Mill Tooling tab, select a **Threading Tool** and specify **Min. hole dia.**

You can also specify **Minimum hole dia** in the Operation Parameters dialog box, on the Tool tab, on the Thread Mill Tool secondary tab, under **Tool Dimensions**. Changes in the Operation Parameters dialog box are not saved to the TechDB.



Post Processor Path



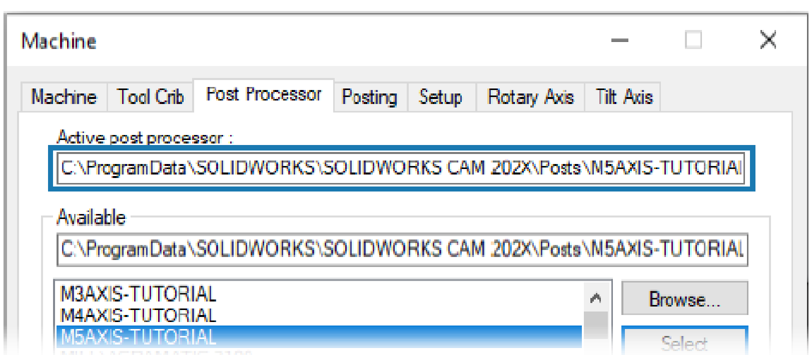
You can specify the default location of the folder containing post processors on the Settings tab of the Technology Database (TechDB). Under **General**, specify **Post Processor Path**. You do not need to reselect the post processor for every part or assembly.

When you change the location of the folder containing post processors and you open a previously programmed part or assembly in SOLIDWORKS CAM, the following occurs:

1. SOLIDWORKS CAM determines whether the post processor file is available in the folder for **Active post processor**.

   If the folder is unavailable, the software loads the **Post Processor Path**.

2. SOLIDWORKS CAM searches for the post processor file in **Post Processor Path**.
3. When SOLIDWORKS CAM finds the post processor file, it displays the file path of the post processor file in the Machine dialog box on the Post Processor tab for **Active post processor**.



Probe Cycles

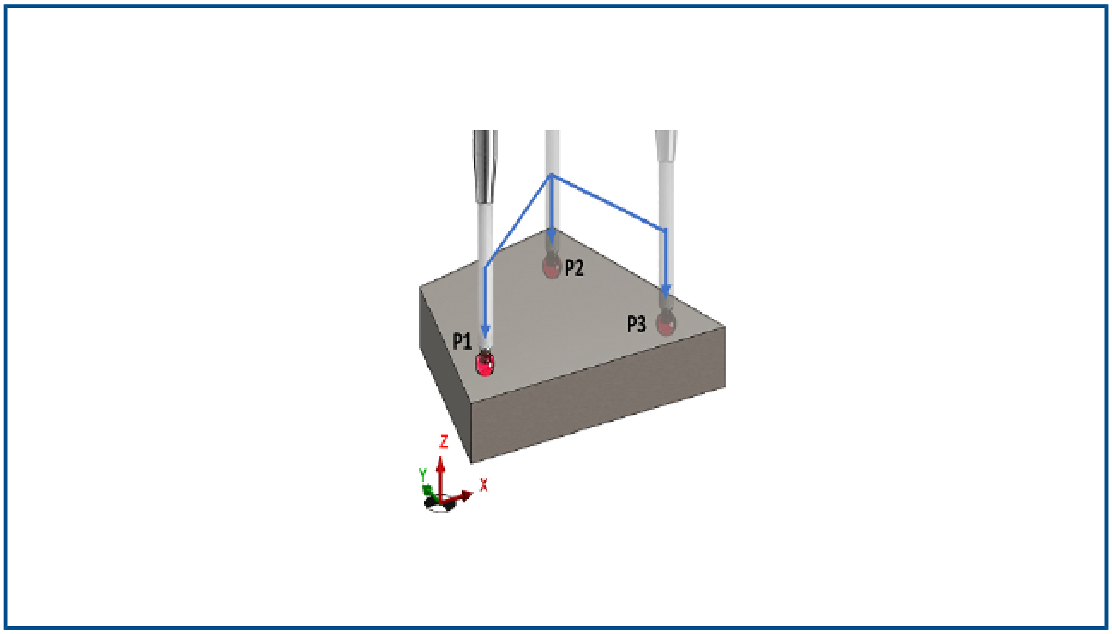
SOLIDWORKS CAM includes additional probing cycles to calibrate and measure planes and axes.

Probing cycles include:

- **3 Point Plane**
- **Angle Measurement (X Axis)**
- **Angle Measurement (Y Axis)**
- **4th Axis Measurement (X Axis)**
- **4th Axis Measurement (Y Axis)**

You can access probe cycles in the Operation Parameters dialog box on the Probe tab, under **Probe Cycle**.

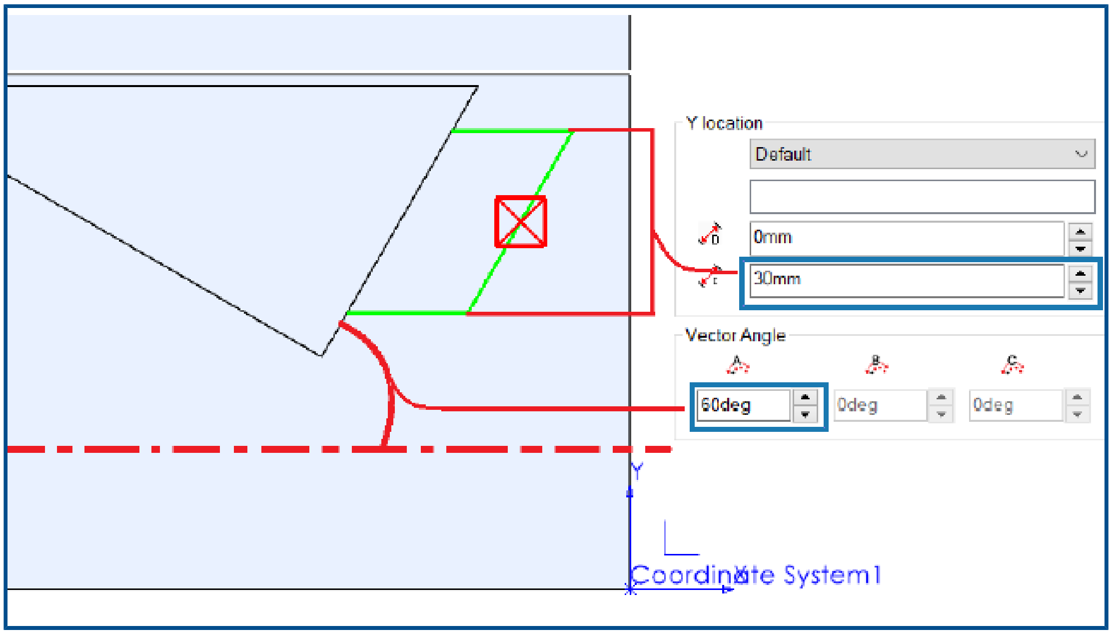
Three Point Plane



With the **3 Point Plane** probe cycle, SOLIDWORKS CAM measures the selected surface using three points on that surface. The probed points establish a plane.

When you select **3 Point Plane**, SOLIDWORKS CAM positions the three points at default offset values. You can modify the offset values and probe the points at the required locations.

Angle Measurement (X/Y Axis)

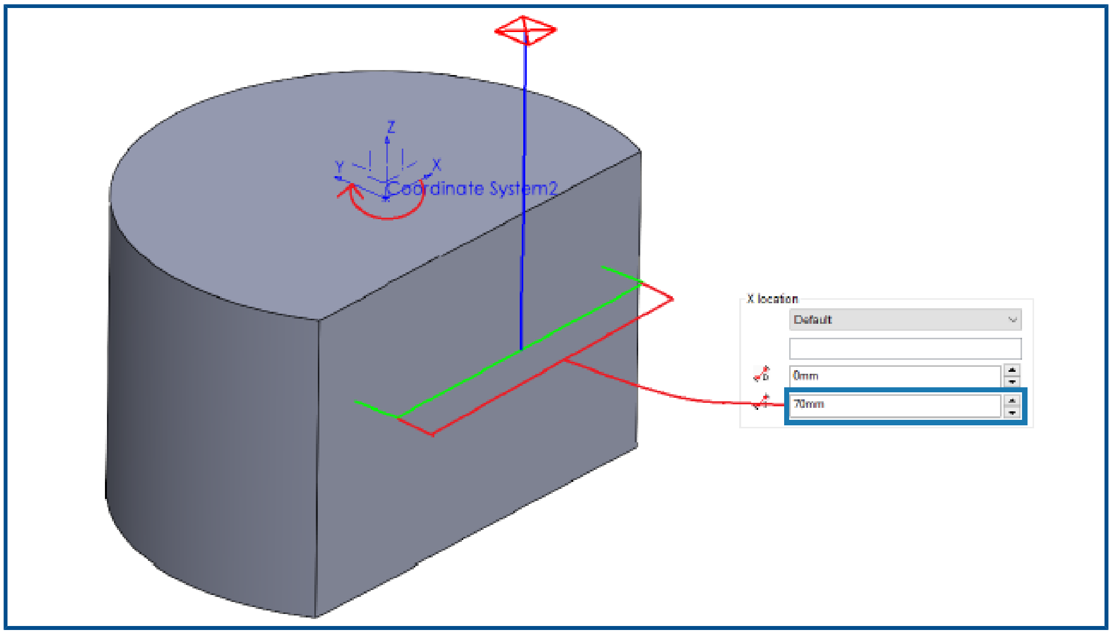


The **Angle Measurement (X Axis)** and **Angle Measurement (Y Axis)** probe cycles probe two points on a selected surface and calculate the angle of the face with respect to the X or Y axis, respectively.

SOLIDWORKS CAM positions the two points symmetrically around the centroid of the selected face. In the Operation Parameters dialog box, on the Probe tab, under **Probe Cycle**, you can specify the distance between the points in **Incremental Distance** for **X location** and **Y location**.

The normal of the selected planar face must be perpendicular to the Z axis of the setup where you insert the probe.

4th Axis Measurement (X/Y Axis)



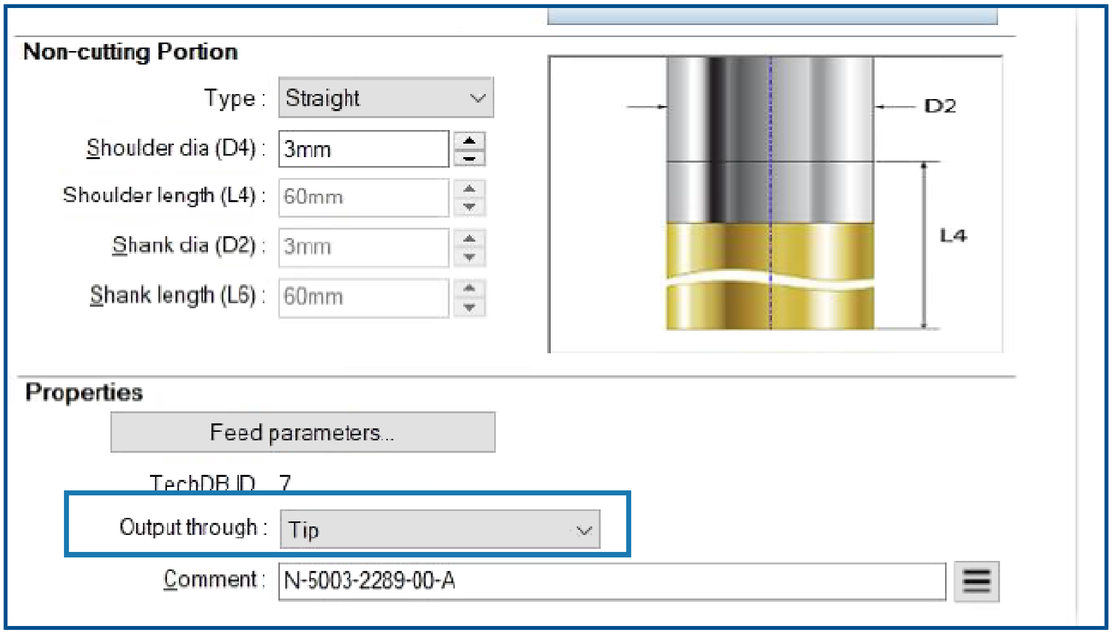
This probe cycle measures the slope of a selected surface between two points with respect to the fourth axis.

The selected surface must be such that the slope between the probed points is measured in the X or Y axis. You can use the resultant value to compensate the rotary axis.

The X and Y coordinates of the centroid of the surface are the start point of the toolpath. SOLIDWORKS CAM positions the probing points symmetrically about this start point based on the assigned distance between the two probing points.

The probe movements are parallel to the axis. SOLIDWORKS CAM measures the clearance distance from the reference point on the surface. For the probing moves, the clearance distance can be more or less than the defined.

Probe Tool Output Options

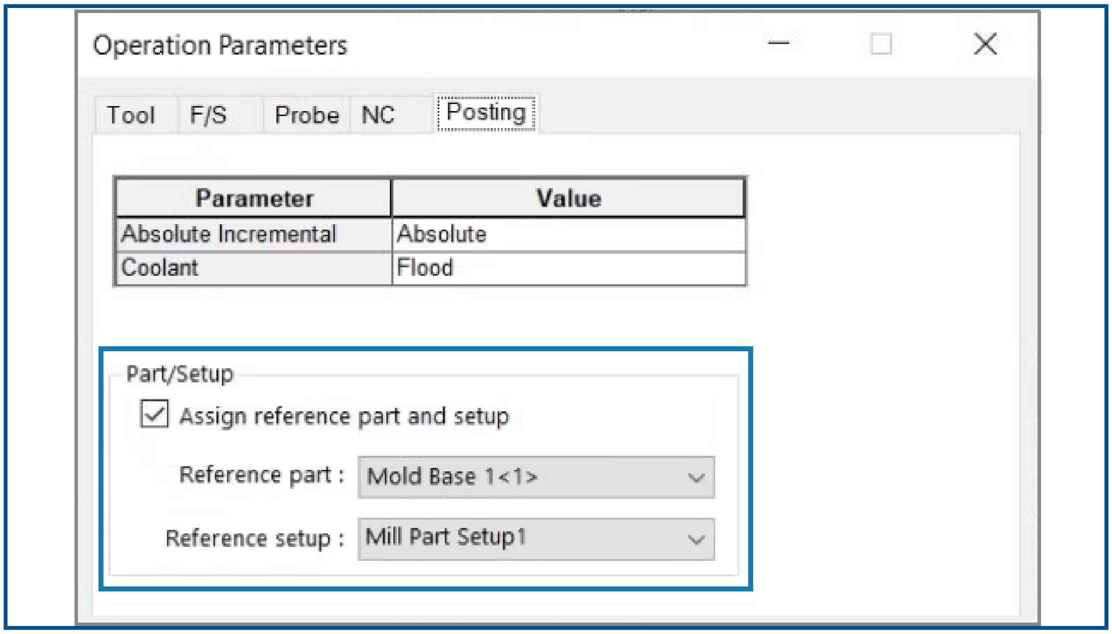


You can specify the **Output through** parameter for probe tools. This parameter generates the toolpath and G-code with the set tool reference point.

In the Operation Parameters dialog box, on the Tool tab, on the Probe Tool tab, under **Properties**, you can specify options for **Output through**:

- **Tip**. Generates the toolpath with reference to the tip of the probe tool.
- **Center**. Generates the toolpath with reference to the center of the probe tool.

Probing Cycles in Assembly Mode



You can assign appropriate part instance and mill part setups for each probe operation generated in Assembly mode. This ensures an accurate **Part Setup Origin** while posting the toolpath of the probe operation.

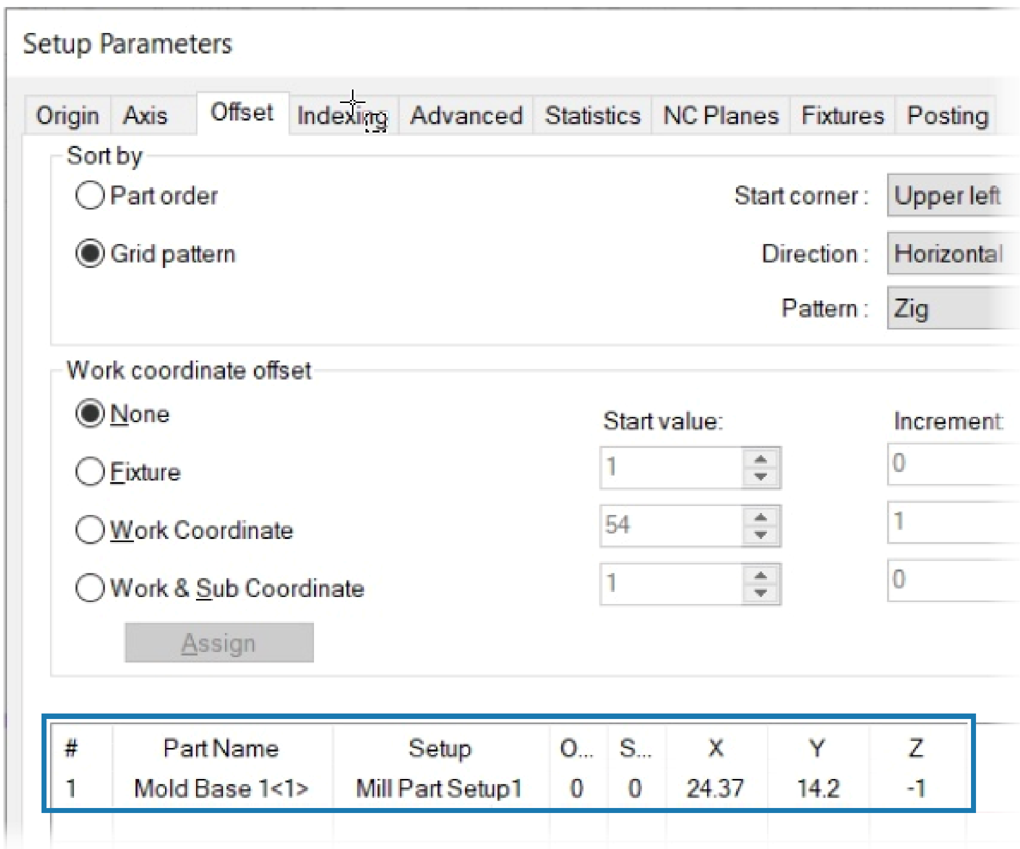
In previous releases, if only probe operations existed under an operation setup of an assembly, SOLIDWORKS CAM measured their coordinates from the fixture coordinate system (FCS). SOLIDWORKS CAM did not list the instance and relevant feature setup on the Offset tab in the Setup Parameters dialog box. Even if you specified the output origin as **Part Setup Origin**, the toolpath coordinates referred to the FCS, leading to inaccurate posted code.

In the Operation Parameters dialog box, on the Posting tab, under **Part/Setup**, you can specify parameters in Assembly mode.

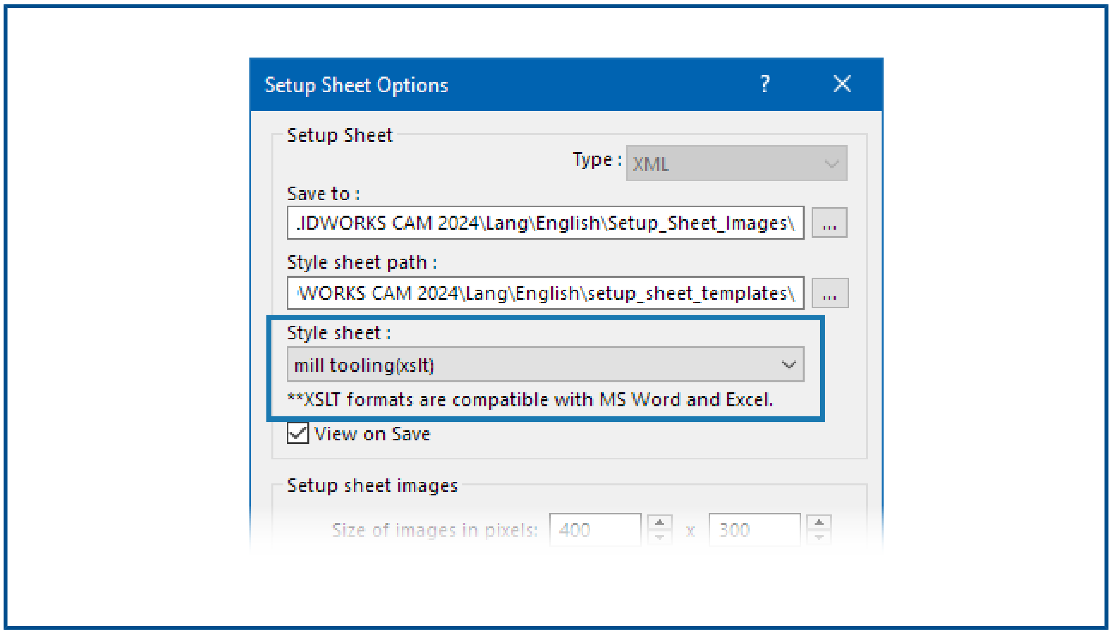
Parameter	Description
Assign reference part and setup	Enables the **Reference part** and **Reference setup** parameters.
Reference part	Lists all parts in the Part Manager. The default selection is the part (with the part instance as a suffix if there are multiple part instances) whose face you selected in the Probe tab for the **Probe** operation. If you did not select a face, SOLIDWORKS CAM uses the first part listed in the Part Manager. If the post processing requires you to specify the **Part Setup Origin**, SOLIDWORKS CAM uses the values of the origin of the selected part as reference. SOLIDWORKS CAM also uses the **Part Setup Origin** to calculate the coordinates when executing **Step Through Toolpath** and simulation commands.

Parameter	Description
Reference setup	Lists all the part setups associated with the part or part instance selected in **Reference Part**. The default selection is the valid feature setup for the part or part instance selected for **Reference Part** whose features can be machined from the selected operation setup. SOLIDWORKS CAM uses the origin of the part setup that you select to compute the coordinates of the toolpath while posting.

For **Probe** operations, the selections you make for **Reference part** and **Reference setup** are displayed in the part instances and work coordinates on the Offset tab of the Setup Parameters dialog box.

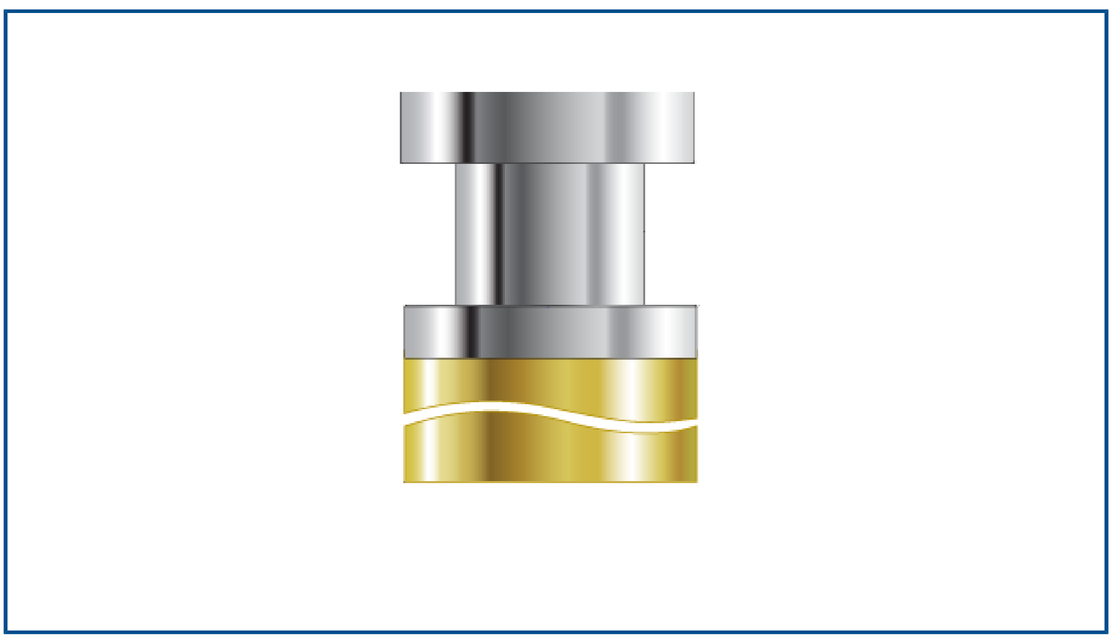


Setup Sheets



The default format for Setup Sheets is `.xslt` for compatibility with the latest browsers.

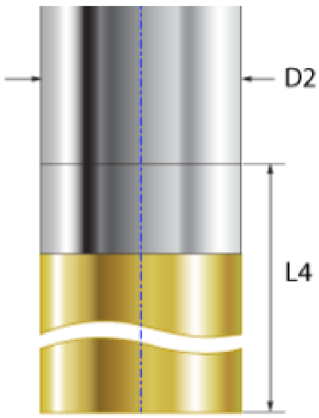
Shank Types for Mill Tools



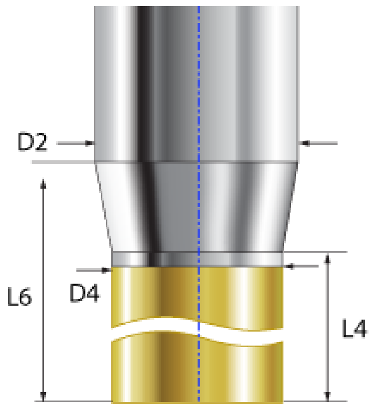
You can define shank types (**Straight**, **Tapered**, or **Neck**) for any Mill Tool.

In previous releases, only certain Mill Tools could have shank types. You can specify shank types for the noncutting portion of these additional tools:

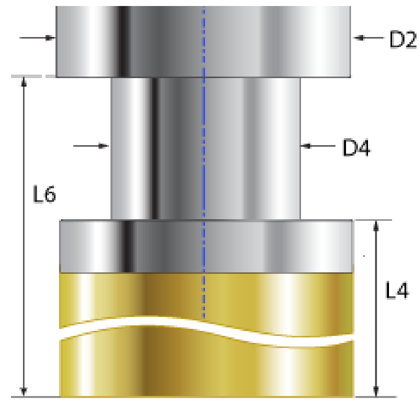
- Bore Tool
- Center Drill
- Countersink Tool
- Dovetail Tool
- Keyway Tool
- Lollipop Tool



Straight. You can define the shoulder length and shank diameter.



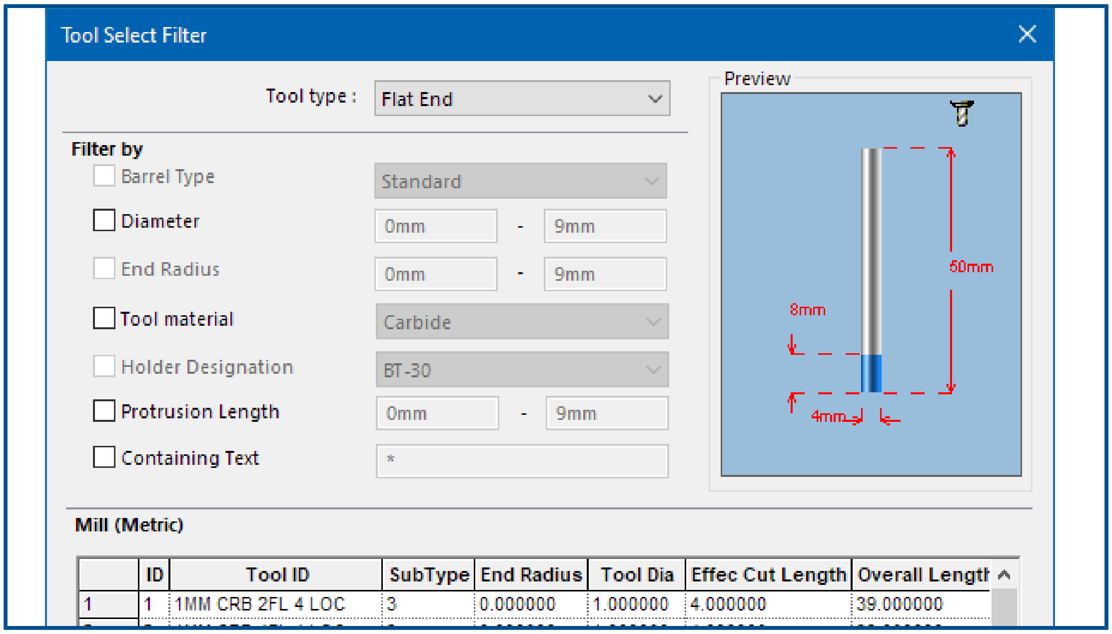
Tapered. You can define the shoulder diameter, shoulder length, shank diameter, and shank length. The tapered portion of the tool is the noncutting portion of the cutting tool.



Neck. You can define the shoulder diameter, shoulder length, shank diameter, and shank length. The neck portion of the tool is the noncutting portion of the cutting tool.

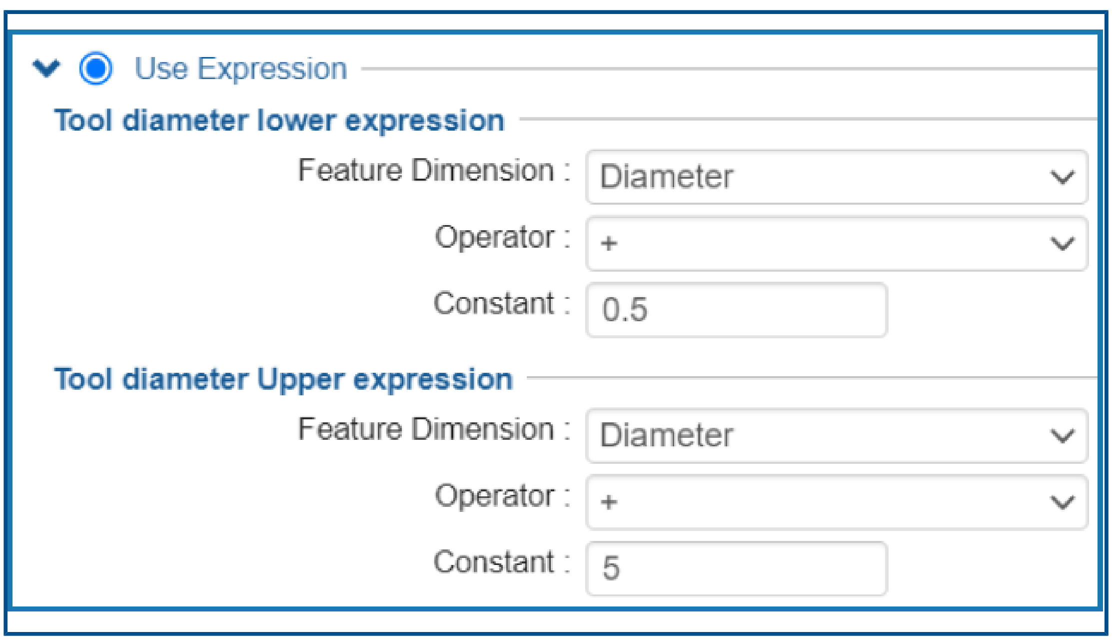
- D2 = Shank diameter
- D4 = Shoulder diameter
- L4 = Shoulder length
- L6 = Shank length

Tool Select Filter Dialog Box



You can resize the Tool Select Filter dialog box to see additional table columns.

Tool Selection - Flute Length



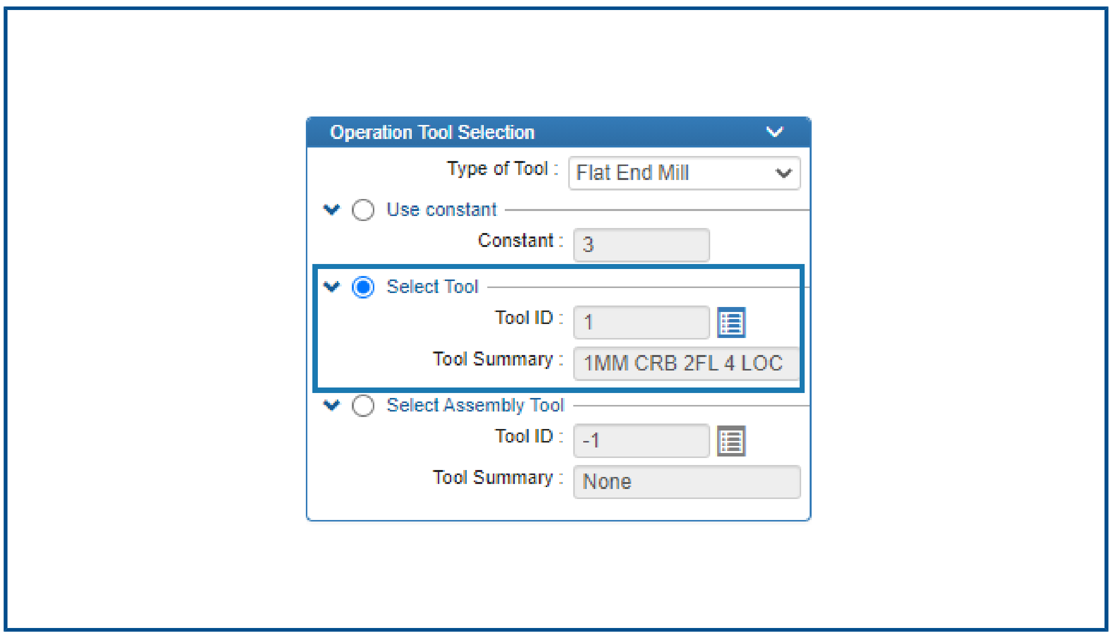
When you specify tool selection criteria based on **Use Expression** and not on a specific tool, SOLIDWORKS CAM accounts for the tool's flute length.

When you run **Generate Operation Plan**, for each operation that you define the tool selection criteria with a Tool diameter lower/upper expression, the following rules apply:

- If the tool crib has two or more tools with identical diameter values matching the expression criteria, SOLIDWORKS CAM accounts for the flute length to assign the tool. It selects the tool with a flute length more than the feature depth. If all tools have a flute length more than the feature depth, SOLIDWORKS CAM selects the tool with a flute length closest to the feature depth.
- If SOLIDWORKS CAM still finds two or more tools, it uses the rules of Stock/Tool Material Mapping to select a tool.

For example, consider a rectangular pocket with a feature depth of 75mm. Based on the feature strategy assigned to this feature, the tool selection criteria selects a 25mm Flat End Mill. The tool crib has two Flat End Mill tools with identical diameters of 25mm. However, one tool has a 50mm flute length and the other has an 80mm flute length. SOLIDWORKS CAM selects the tool with the 80mm flute length because it is closer in value to the feature depth.

Tool Selection - Tool Crib Priority



SOLIDWORKS CAM has better tool selection logic when you select **Tool crib priority** in the Technology Database (TechDB).

SOLIDWORKS CAM has optimized tool selection logic so appropriate tools are available in the active tool crib:

- If the tool assigned in the TechDB for a specific operation is not in the active tool crib, SOLIDWORKS CAM adds it to the tool crib even though smaller tools might be in the active tool crib. (If you selected a tool by referencing it to a specific **Machine ID** in the TechDB.) If another tool with similar parameters is in the active tool crib, SOLIDWORKS CAM uses that tool.
- If you specify that the resultant tool derived from the expressions defined in the TechDB as inactive, SOLIDWORKS CAM does not add it to the active tool crib. It uses the subsequent tool selection rules to add an active tool to the tool crib.

19

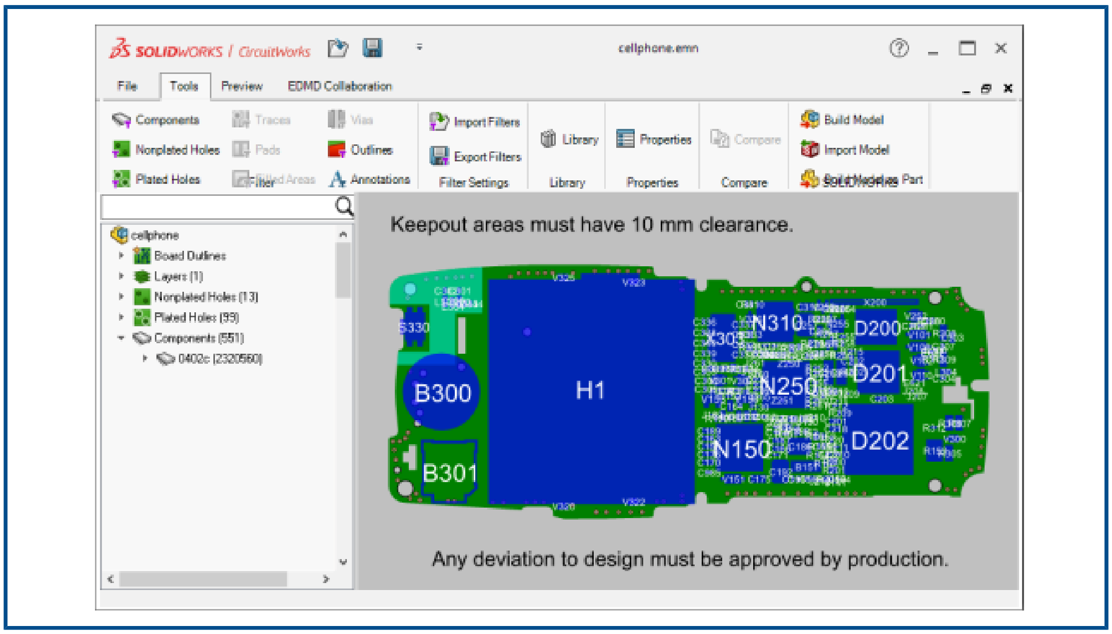
CircuitWorks

This chapter includes the following topics:

- **User Interface Redesign (2024 SP4)**
- **CircuitWorks in SOLIDWORKS Standard (2024 FD02)**
- **SOLIDWORKS Connected Support for CircuitWorks (2024 FD01)**

CircuitWorks™ is available in SOLIDWORKS® Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium.

User Interface Redesign (2024 SP4)



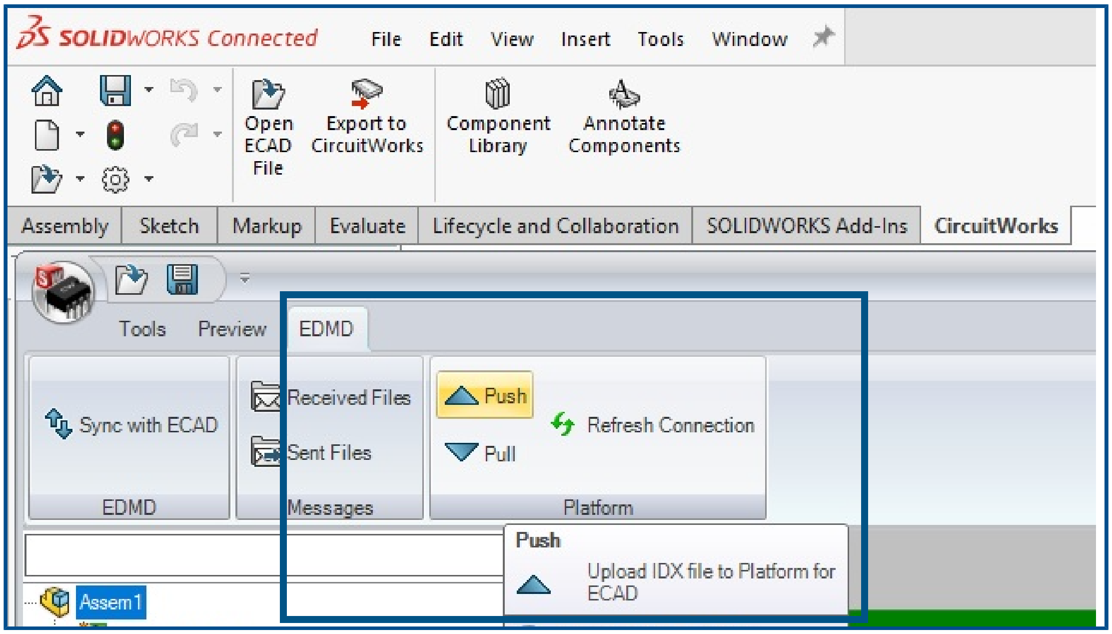
The user interface for CircuitWorks is redesigned to be more consistent with SOLIDWORKS.

The Quick Access toolbar, CommandManager, and the CircuitWorks tree look and work similarly to those in SOLIDWORKS.

CircuitWorks in SOLIDWORKS Standard (2024 FD02)

CircuitWorks is available in all versions of SOLIDWORKS, including SOLIDWORKS Standard.

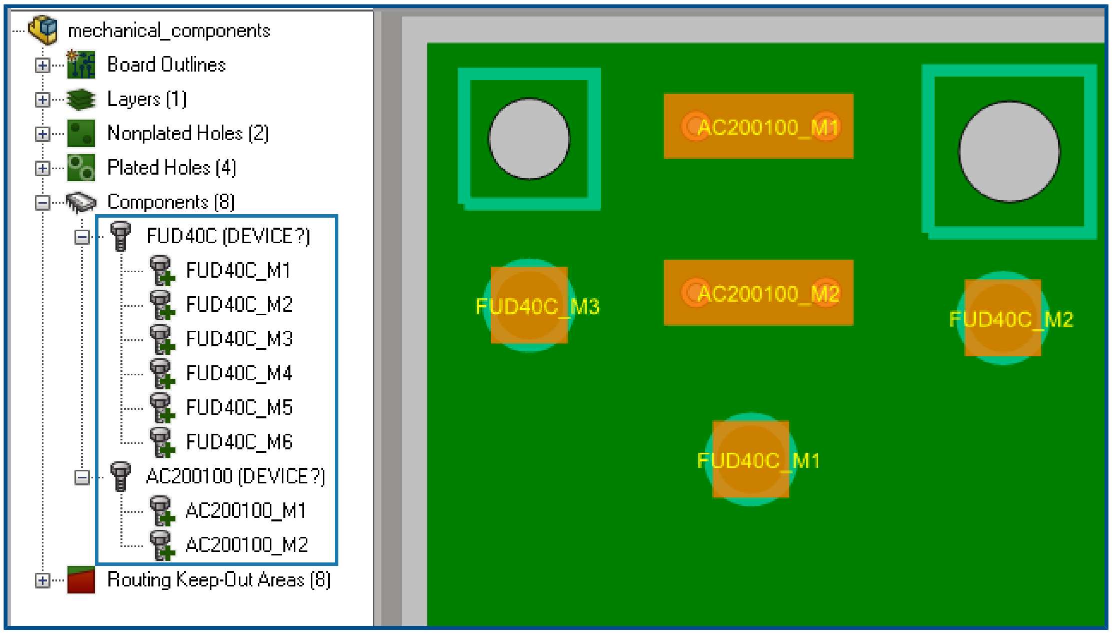
SOLIDWORKS Connected Support for CircuitWorks (2024 FD01)



SOLIDWORKS Connected supports additional CircuitWorks functionality.

- The **Push** and **Pull** tools (EDMD toolbar) let you send and receive IDX 3 files from ECAD.
- **Associate Model** lists electronic component data models from the **3D**EXPERIENCE platform. You can associate each CircuitWorks tree component with SOLIDWORKS part or assembly files. After you associate a model from the **3D**EXPERIENCE platform, the asterisk in the CircuitWorks tree disappears.
- In the Component Properties panel and the CircuitWorks Component Library, for **SOLIDWORKS component**, click **Browse for component** to list electronic component data models from the **3D**EXPERIENCE platform.
- When you create an assembly in SOLIDWORKS Connected, the Open dialog box lists electronic component data models from the **3D**EXPERIENCE platform that you can use in the assembly.

Reference Designators for Comparing Mechanical Component Modifications (2024 SP3)



CircuitWorks assigns a temporary reference designator (Ref. Des.) to each instance of a mechanical component if the component does not have a Ref. Des. already associated with it.

When you open an `IDX 3` file in CircuitWorks, the software assigns the Ref. Des. that is also available in SOLIDWORKS when you build the model. The Ref. Des. appears in the CircuitWorks tree with the instance name. The same Ref. Des. appears in the SOLIDWORKS FeatureManager design tree after you model the mechanical components in SOLIDWORKS.

By having Ref. Des. indicators on each component, you get:

- More accuracy when viewing modification results when you export the board assembly from SOLIDWORKS to CircuitWorks using the **Export to CircuitWorks** tool. Any modifications to the mechanical components in SOLIDWORKS appear in the Sync with ECAD dialog box and in the Changes tree in the CircuitWorks window.
- More accurate results when viewing modification results when you import or export the board assembly from CircuitWorks to an ECAD designer using the **Sync with ECAD** tool. Any modifications to the mechanical components appear in the Sync with ECAD dialog box.

Pushing Tasks to the 3DEXPERIENCE Platform

To push tasks to the 3DEXPERIENCE platform:

1. From CircuitWorks, click **File** > **Options**.

2. On the Prostep EDMD tab:
   - Select **Use Prostep EDMD**.
   - In **Read and write Prostep version**, select **v 3.0**.
   - In **Shared folder**, specify where to share Prostep EDMD files between CircuitWorks and the ECAD application. Ensure that you have write permission for this folder.
   - Select **Use GMT style date in IDX communication**.
   - (Optional) Select **Reverse rotation direction of components on the underside of the board**. When cleared, the component does not rotate - it goes on the underside of the board instead of on top, as a mirror image of the component.
3. On the SOLIDWORKS Import tab, under **Conductive layer modeling**, select **Complete (slower)**.

   SOLIDWORKS creates all the layers so you can see each layer of the board.
4. Click **OK**, then restart SOLIDWORKS.
5. From CircuitWorks, click **Push** (EDMD toolbar).
6. In the EDMDPushPull dialog box, under **Ready to push change**:
   a) For **Collaborator**, enter a name.

      You can enter the first, last, or both names.

   b) Click **Check Name** and search for a name to add.
   c) (Optional) Enter **Comments**.
   d) Click **OK**.

   The baseline data is pushed to the **3D**EXPERIENCE platform in Prostep EDMD IDX 3 format through the **3D**EXPERIENCE Collaborative Tasks. The task is assigned to the ECAD engineer. If you push a change or response file, the software prepopulates the **Collaborator** or you can change the name.

Building Models (2024 FD01)

In CircuitWorks Connected, you can use the **Build Model** tool to build and save board models and components to the **3D**EXPERIENCE platform. In earlier releases, you had to save the board model and each component separately.

CircuitWorks Connected builds the board model and corresponding components regardless of whether you already built the board model and components.

Scenario	After CircuitWorks builds the model
First time building the model	CircuitWorks saves the board and its components to the local cache. Choose options: • **Save to 3DEXPERIENCE**. Saves all models to the **3D**EXPERIENCE platform. • **Don't Save**. Closes the dialog box. You can save the models to the **3D**EXPERIENCE platform later on in the SOLIDWORKS software.

Scenario	After CircuitWorks builds the model
Board model may or may not be in the local cache but exists in the local CircuitWorks database	Choose options: • **Overwrite**. Creates a new board model and saves it to the **3D**EXPERIENCE platform. • **Use Existing**. Downloads the board model from the **3D**EXPERIENCE platform and uses it in the SOLIDWORKS assembly. • **Cancel**. Cancels the build model operation.
Board model's components exist in the local CircuitWorks database	Choose options for the components: • **Yes**. Uses the existing model. • **Yes to All**. Uses the existing models for all components in the board model. • **No**. Builds a new model. • **No to All**. Builds new models for all components in the board model.
Board model is in the local CircuitWorks database and already on the **3D**EXPERIENCE platform but not in the local cache	Choose options: • **Overwrite**. Creates a new board model and saves it to the **3D**EXPERIENCE platform. • **Use Existing**. Downloads the board model from the **3D**EXPERIENCE platform and uses it in the SOLIDWORKS assembly. • **Cancel**. Cancels the build model operation.

After the build model process finishes, you can specify an option to save the board model and its components to the **3D**EXPERIENCE platform automatically. In CircuitWorks, click **Options** > **SOLIDWORKS Import** and select **Automatically Save to 3DEXPERIENCE after Build Model is complete**.

If you decide not to save the board model right after building the board in CircuitWorks, you can save it later when in the SOLIDWORKS software. In SOLIDWORKS, click **Save to 3DEXPERIENCE** (CircuitWorks toolbar) or **Tools** > **CircuitWorks** > **Save to 3DEXPERIENCE**.

Board Outline and Cutout Changes from CircuitWorks (2024 SP2)

CircuitWorks can generate MCAD change files based on board outline and cutout changes. You can then send these changes as `IDX 3` files to Cadence® Allegro®.

ECAD either accepts or rejects each of these changes. Based on the ECAD `IDX 3` response file, rejected changes reappear in CircuitWorks. Click **Build Model** to apply those changes to the SOLIDWORKS assembly.

When you make board outline or cutout changes, any other changes are omitted from the same change file (such as components, holes, or keep in/keep out areas). You need to send those as additional changes later.

Board Outline and Cutout Changes from ECAD (2024 SP3)

ECAD designers can generate `IDX 3` change files based on board outline and cutout changes. You can then open these changes in CircuitWorks.

In CircuitWorks, you can accept or reject each of these changes. Click **Build Model** to apply those changes to the SOLIDWORKS assembly. Based on the CircuitWorks response file, rejected changes reappear in the ECAD system.

20

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.

Offline Help for SOLIDWORKS Composer Products

Offline Help for all SOLIDWORKS Composer products is available as a `PDF` instead of in `HTML` format.

In earlier releases, offline Help worked only in Microsoft Internet Explorer. Now it is browser independent.

Support for SpeedPak Configurations in SOLIDWORKS Composer

You can translate SOLIDWORKS assembly files containing components in SpeedPak configurations to SOLIDWORKS Composer.

The SpeedPak components are switched to their parent configurations to enable translation of these components to SOLIDWORKS Composer.

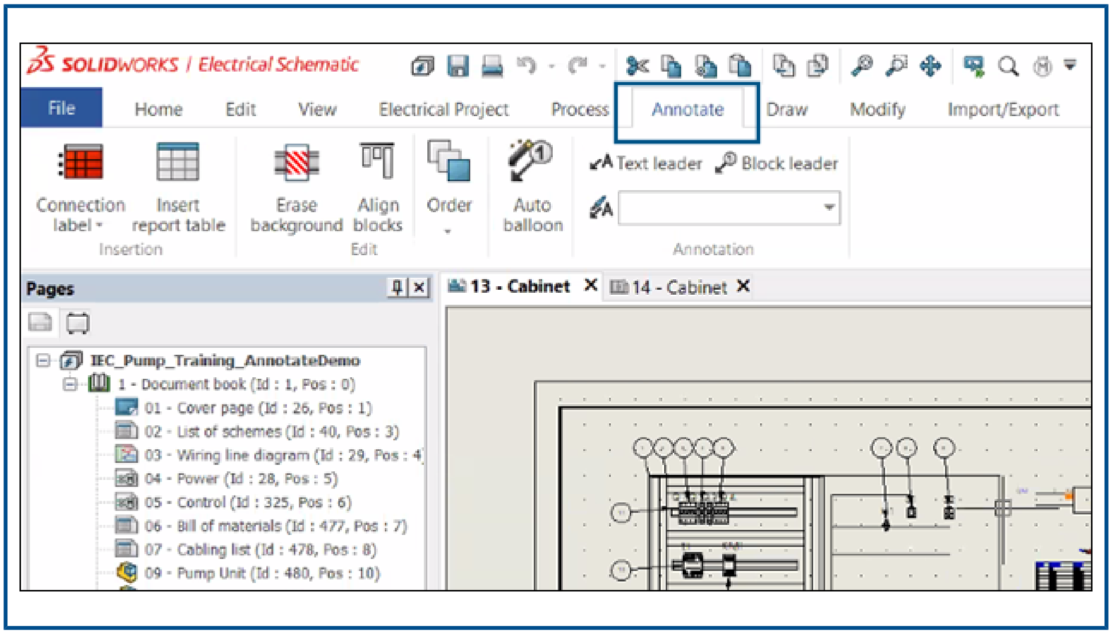
21

SOLIDWORKS Electrical

This chapter includes the following topics:

SOLIDWORKS® Electrical is a separately purchased product.

Annotate Tab (2024 SP3)

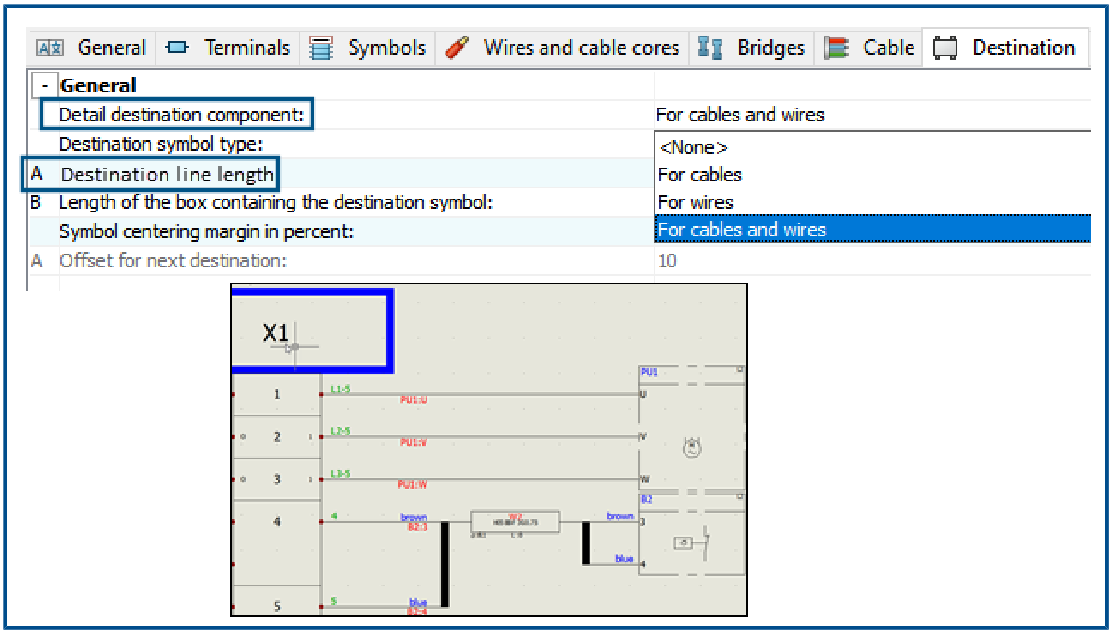


In SOLIDWORKS Electrical Schematic, the **Annotate** tab is added to the ribbon. From this tab, you can make changes to 2D drawings from 3D and flattened routing documents. It saves time and makes customization tasks simpler.

Several existing commands from the **Cabinet layout** tab are also available under the **Annotate** tab:

- **Connection label**
- **Insert report table**
- **Erase background**
- **Align blocks**
- **Order**
- **Auto balloon**
- **Text leader**
- **Block leader**
- **Leader style**

Terminal Strip Drawings (2024 SP3)



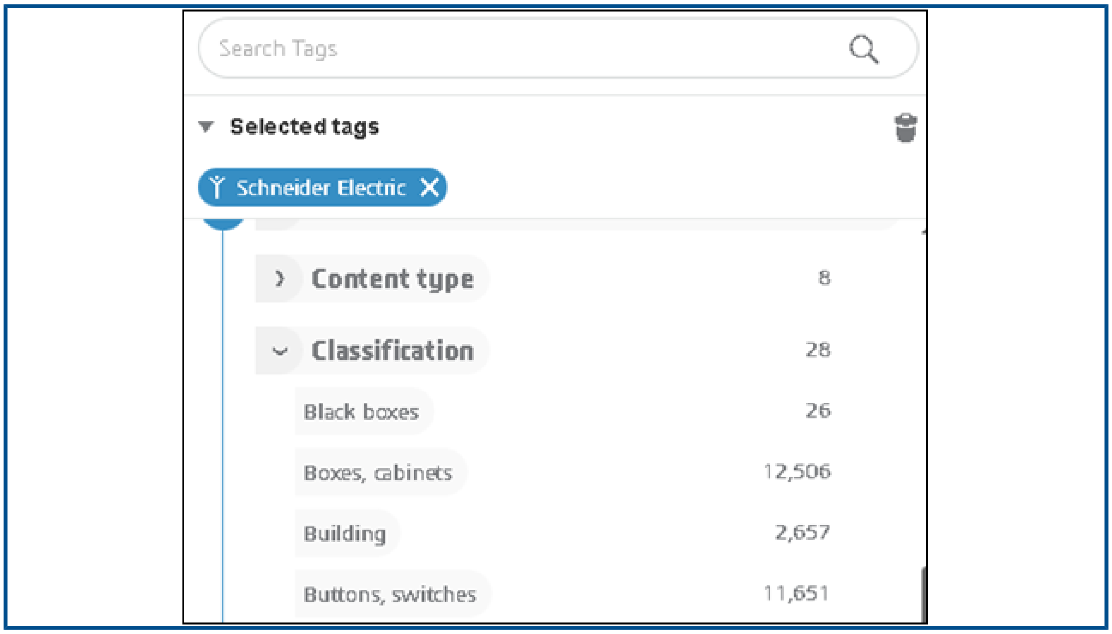
You can organize wires and cables by destination part. This makes terminal strip layouts tidier and more organized.

Enhancements:

- The option **Detail cable destination** is renamed to **Detail destination component**. It has the following options:
    - **None**
    - **For cables**
    - **For wires**
    - **For cables and wires**

    This option displays a box containing the destination symbol for cables and wires. For successive wires associated with the same component, the software draws only one component.
- **A Destination cable core length** is renamed to **A Destination line length**. This option applies to wire components too.
- In the Terminal strip editor dialog box, a new column  appears between **Destination** and **Cable**. It contains the mark of the component terminal where the wire is connected.

6W Tags Enhancements in ECP(2024 FD03)

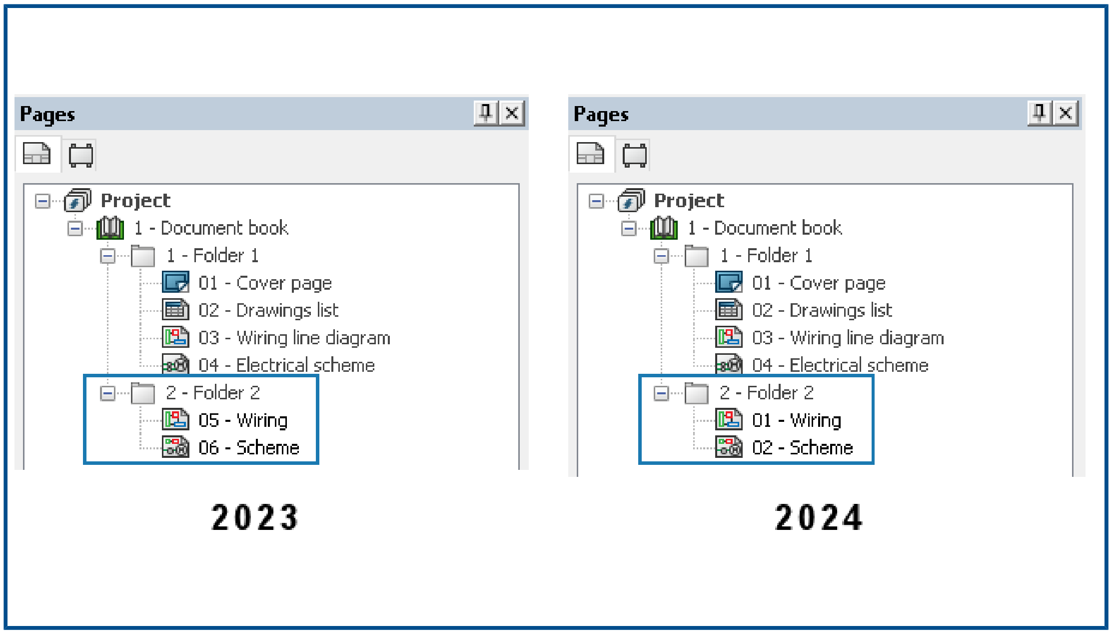


The 6W Tags feature in the **Electrical Content Portal** is enhanced to quickly find specific information in the 6WTags. This helps you organize data and track tasks more effectively.

Enhancements in the **Catalog Content** page:

- The **Classification** is available under the **What** node. When you select a classification, the associated sub-classes are displayed. When you select a sub-class, the next level is displayed. This helps you to filter and navigate through the structure systematically.
- The **Creation date** node in the **When** hierarchy is modified to display the year only. Once you select a year, the corresponding months and dates are displayed under it.
- The **Search Tags** field is added at the top of the 6W Tags area to search for specific values in 6WTags.

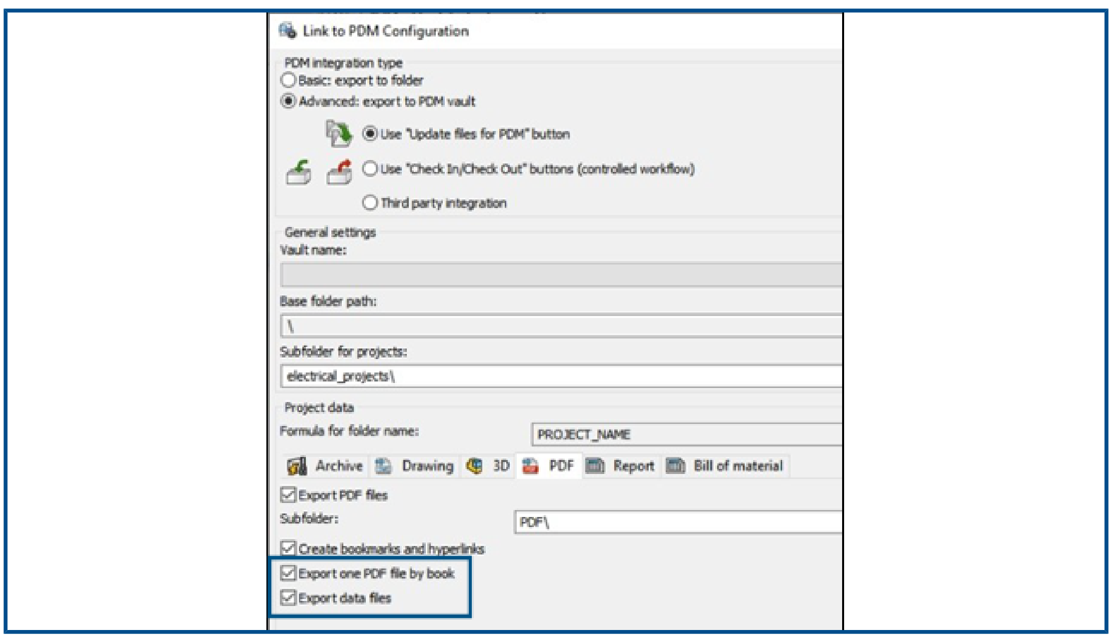
Drawing Mark Numbers (2024 SP2)



You can number drawings by folder.This lets you assign the same drawing number across multiple folders. Previously drawing marks were unique per book.

In the Electrical Project Configuration dialog box, under **Marks unique by**, for **Drawing**, specify **Electrical Project**, **Folder**, or **Book**.

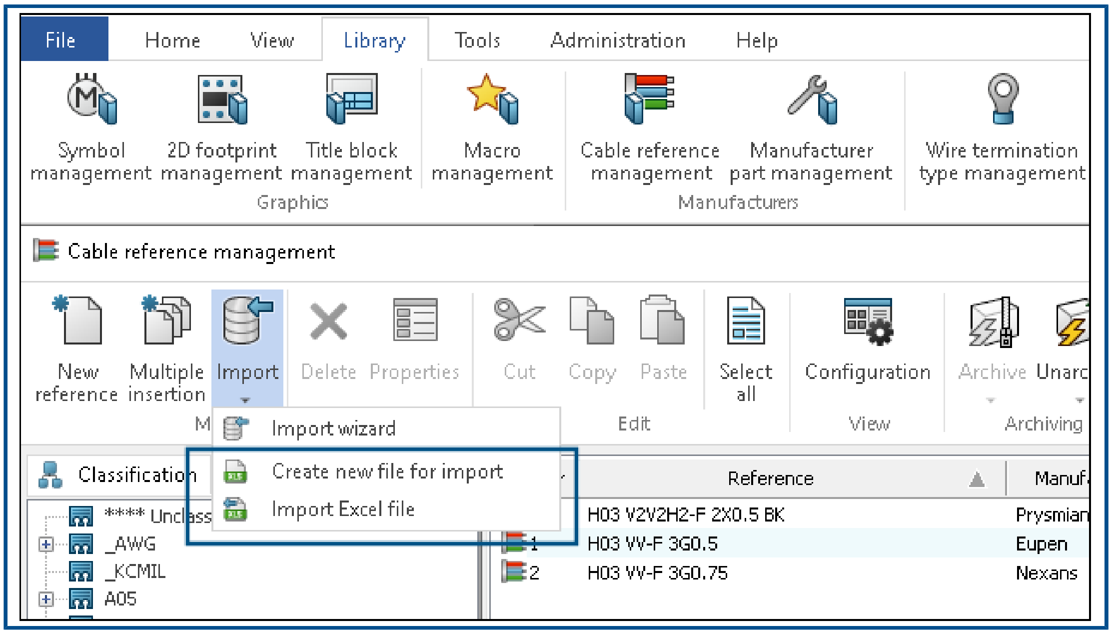
Exporting Data Files (2024 SP2)



In Link to PDM Configuration dialog box, you can include the data files in the exported PDF file.

To export data files, click **Link to PDM Configuration** > **PDF** and select **Export data files**. The option **One file per book** is renamed as **Export one PDF file by book**.

Import Options to Manage Cable References and Manufacturer Parts (2024 SP2)



Two new commands are available in **Cable reference management** and **Manufacturer part management**:

- **Create new file for import**
- **Import Excel file**

In **Cable reference management**, you can access the commands from:

- **Library** > **Cable reference management**. In Cable reference management, click **Import** > **Create new file for import** .
- **Library** > **Cable reference management**. In Cable reference management, click **Import** > **Import Excel file** .

In Manufacturer part management, you can access the commands from:

- **Library** > **Manufacturer part management**. In Manufacturer part management, click **Import** > **Create new file for import** .
- **Library** > **Manufacturer part management**. In Manufacturer part management, click **Import** > **Import Excel file** .

Creating a New Excel File from Template

You can create a new Excel file for import and adapt it to the input language and class of manufacturer parts or cable references.

You can import all the data from the cable references and manufacturer parts, which were previously missing in the file, like cable core details, complex cable core properties, circuits, and connections points in manufacturer parts.

To create a new Excel file from the template for cable references:

1. Click **Library** > **Cable reference management** .
2. In the Cable reference management dialog box, click **Import** > **Create new file for import** .
3. In the Create new Excel file for cable reference import dialog box, select the following:
   - For **Language**, select the language from the list. The default language is set to match the interface language. The list contains the 14 languages that correspond to the interface languages.
   - For **Class**, click to open the **Class selector** and select the base class for cable reference. If you do not select any class then all the classes and subclasses are available in the Excel file.
   - For **Template available**, select the Excel file found in the template folder.
   - Select **Open created template** to open the created template.
4. Click **OK**.
5. In the Save As dialog box, save the new Excel file in the required location. The file opens automatically.
6. Edit the data in the Excel file to import the new data to the cable references.
   - **Reference** is the mandatory field for the successful import of the data.
   - **Manufacturer**, **Class**, **Library**, **Family**, **Cable type**, etc. are **required** fields. If you leave these fields empty, the software warns you and imports the data with errors.
   - **Article Number**, **External ID**, **Translatable data**, etc. are **optional** fields. If you leave these fields empty, no errors occur.
   - **Column A** (can be hidden) contains key code, for example, to identify the language of the header.
   - The last row of header (can be hidden) contains the name of fields associated with columns like **#car_reference**. Do not remove this information.
   - You can add more columns for translated data to enter more languages at the same time. Modify the language code in the field name, like **.en** in **#car.ctr_0.en** for cable description.
   - The hidden page **_ValidationList_** contains the named range used to show drop-down items in some columns, based on the Excel feature **Data Validation**.

> You can also create a new Excel file for import of the Manufacturer part using the same steps as above. Access the command from **Library** > **Manufacturer part management**. In **Manufacturer part management**, click **Import** > **Create new file for import** .

Importing the Template

You can reimport the filled Excel file that you created earlier using the **Create new file for import** command. You can only import new data.

To import the Excel file:

1. Click **Library** > **Cable reference management** .
2. In **Cable reference management**, click **Import** > **Import Excel file** .
3. In the Open dialog box, select the Excel file to import and click **Open**.
4. In the Cable references import dialog box, do the following:

- Click **Select file** to open the Open dialog box and select the Excel file to import. **Excel import file** displays the path of the imported Excel file.
- Under **Format selection and separator**, for **Row format**, choose from:
  - **One line per cable core**
  - **One line per reference**

  For **Cable core separator**, choose from:

  - **Colon ':'**
  - **Line break**
  - **Pipe '|'**
  - **Semi colon ';'**

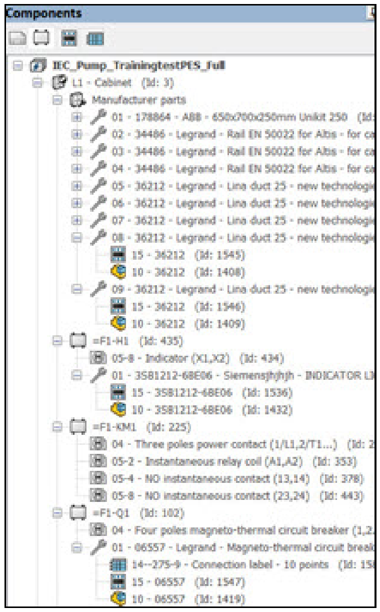
  This option appears only if you select **One line per reference** for **Row format**.

- Under **File preview**, the preview of the imported file appears.
- Click **Compare** to simulate cable reference import. A log file is created with the same name as the Excel file. If there are errors, you can open the Excel sheet and rectify the errors.
- Click **Open** to open the selected Excel file for editing.
- Click **Import** to import the manufacturer cable reference to the library.

You can also import the template for the Manufacturer part using the same steps as above. Access the command from **Library** > **Manufacturer part management**.

In Manufacturer part management, click **Import** > **Import Excel file** .

Restructuring the Electrical Component Tree



The electrical component tree is restructured and simplified to display the 2D footprints, 3D parts, and the connection labels associated with a manufacturer part. You can quickly identify these items for a particular manufacturer part in the electrical component tree.

In earlier releases, all the 2D footprints, 3D parts, and the connection labels inserted appeared as subitems in the electrical component tree. You could not distinguish between the 2D footprint and connection labels applicable to a particular manufacturer part.

Components

Under each component, there is a node for each manufacturer part associated with the component and an intermediate node for each symbol (2D footprint or connection label) representing that manufacturer part. The node for each manufacturer part contains all the corresponding 2D footprints, connection labels, and the 3D part or assembly items.

You can control the visibility of the tree items for the manufacturer parts. In the component tree, right-click the top item of the project, select **View** > **Manufacturer part**, and choose from the following three options:

- **Hide**. Hides the node for manufacturer parts. The tree items relative to the manufacturer parts appear directly under the component.
- **With graphics**. Creates intermediate tree items only for the manufacturer parts that have graphics (2D footprints, connection labels, etc.) associated with it. This is the default option.
- **All**. Creates items for all the manufacturer parts whether they have graphics associated with them or not.

Locations

An item in the component tree groups all manufacturer parts of the location. The node contains the 2D footprints and the connection labels associated with each manufacturer part associated with the location.

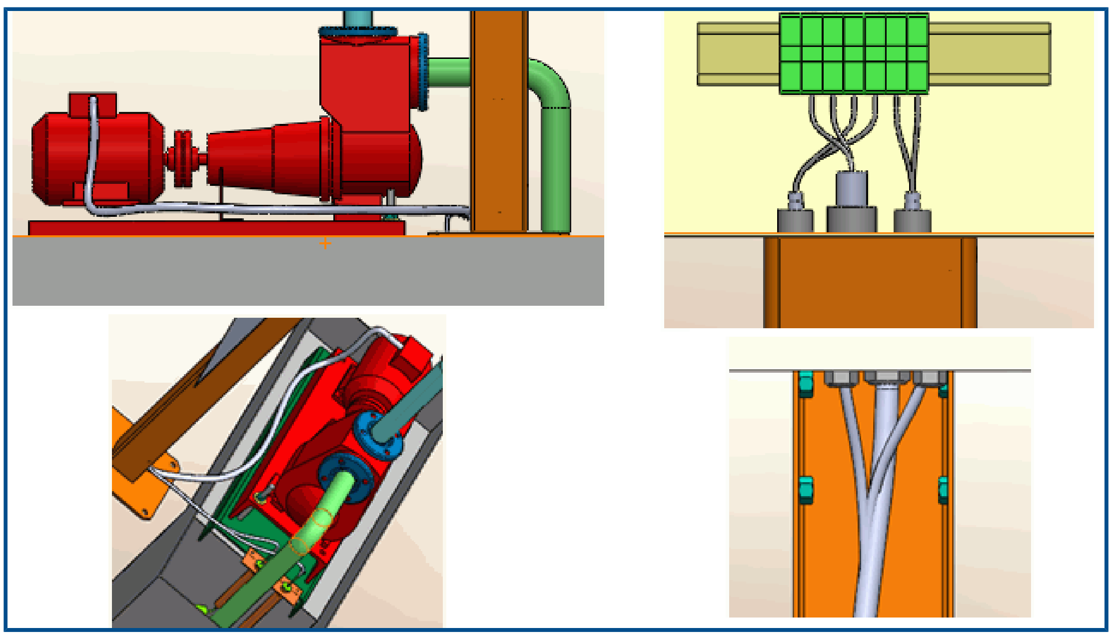
You can right-click the node and select the following:

- **Properties**. Opens the Properties dialog box of the selected manufacturer part. If you select several manufacturer parts, the Properties dialog box displays only the common properties.
- **Delete manufacturer parts**. Deletes the selected manufacturer parts.

Cabinet Layout

The Intermediate node for location parts is also applicable for the 2D or 3D cabinet layout tree. All the manufacturer parts appear even if they do not have any graphics associated with them.

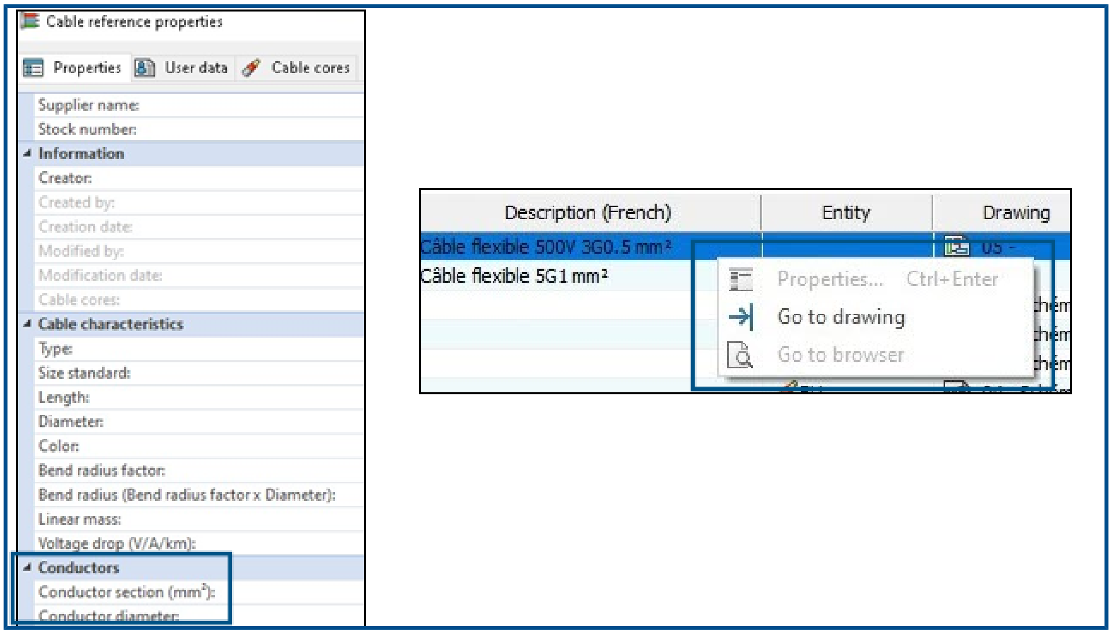
SOLIDWORKS Electrical Tutorials (2024 FD01)



SOLIDWORKS Electrical tutorials are integrated into the SOLIDWORKS Electrical help. The tutorials are more complete and consistent with existing SOLIDWORKS documentation.

At **http://help.solidworks.com**, click **SOLIDWORKS Electrical** > **SOLIDWORKS Electrical Tutorials**.

Cable Management (2024 SP1)



Cable Management has a streamlined workflow that saves you time.

The enhancements include:

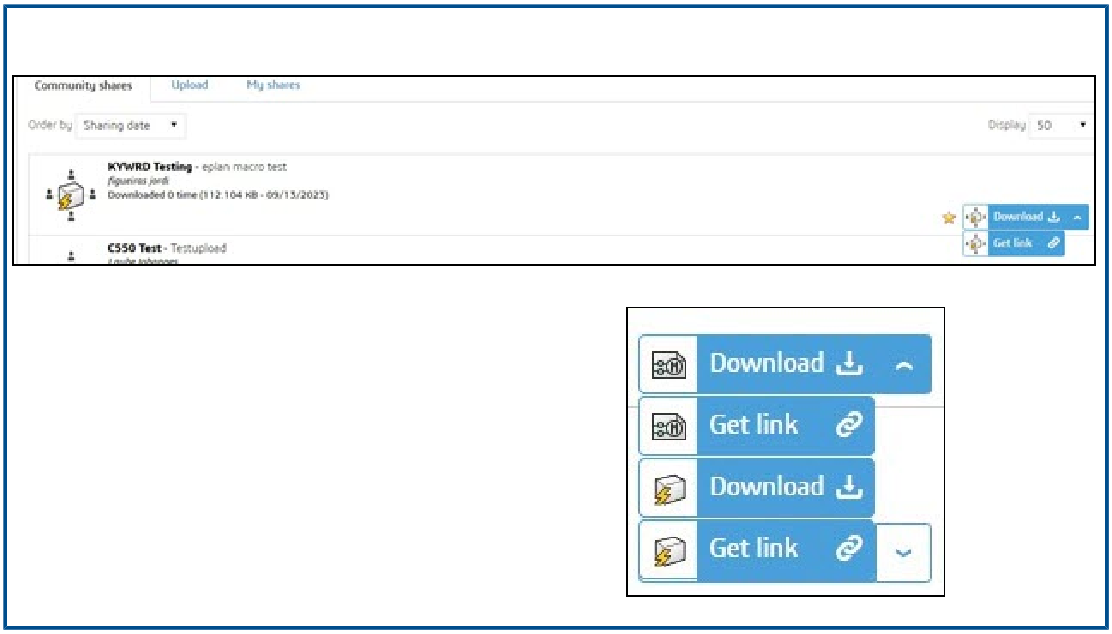
- **Replace** cable is more flexible. You can replace a miscellaneous cable core type with a neutral cable core type with no system warnings.
- New commands are available in the shortcut menu. You can use:
  - **Properties** to view the properties of the selected cable.
  - **Go to drawing** to go to the location of the drawing, generally a line diagram from cable core item.
  - **Go to browser** to show the origin component of the cable core.
- When you delete cables used in the scheme or line diagram, the wires associated with their cable cores are dissociated automatically.
- The Cable Reference properties dialog box includes a new **Conductors** section with **Conductor section** and **Conductor diameter** listed under it. The section **Characteristics** is renamed as **Cable characteristics**.

Dynamic Link Between Drawings (2024 SP1)

When you modify a `.SLDDRW` drawing file inside SOLIDWORKS® and save it, the software updates the corresponding drawing file (`.EWG`) inside the **SOLIDWORKS Electrical project** folder automatically.

In earlier releases, when you modified a drawing file inside SOLIDWORKS® and saved it, the corresponding drawing file inside **SOLIDWORKS Electrical project** folder was not updated automatically. You had to click **Create Project Drawing** command again to update the drawing file.

Sharing Links in the Electrical Content Portal (2024 SP1)



You can share links to an item (the manufacturer part, symbol, etc.) or the electrical package containing the item in the Electrical Content Portal.

You can select the list next to an item to:

- Download the item
- Link to the item
- Download the electrical package
- Link to the electrical package

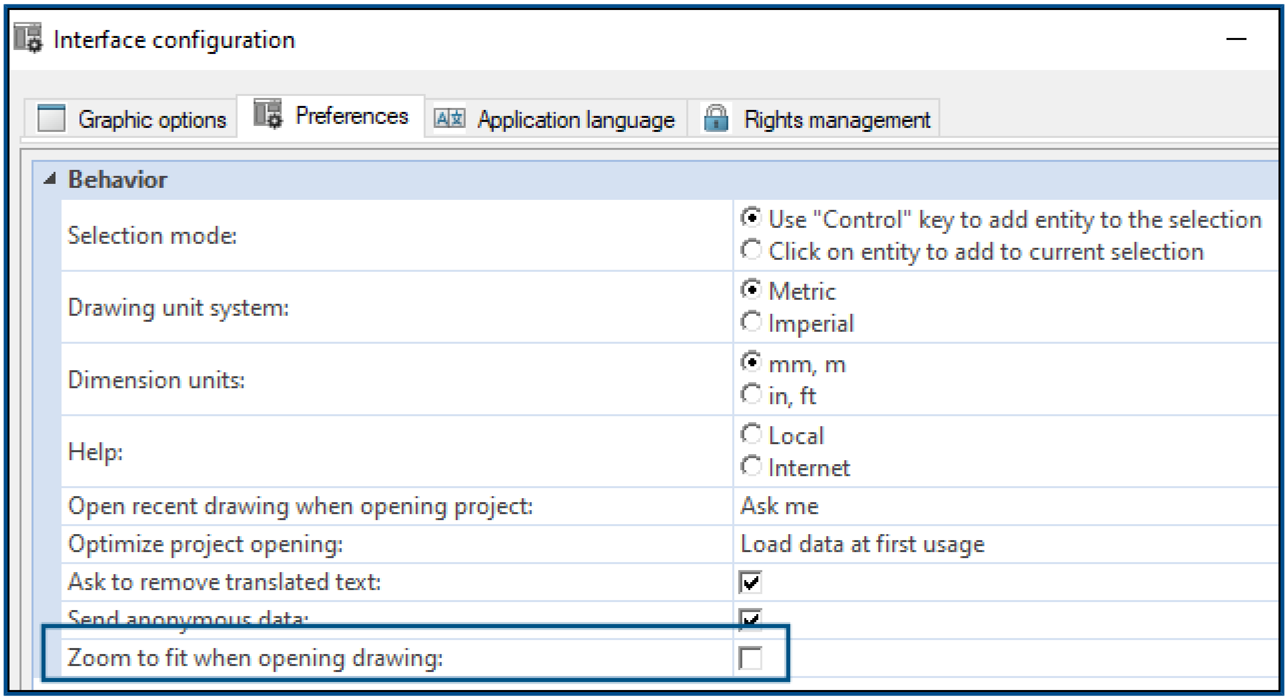
In previous releases, you could only download the content and automatically unarchive it into the respective libraries.

Single Entry for Cables or Wires in BOM Tables (2024 SP1)

The BOM table created for cables and wires after routing contains only one entry for each wire style or cable reference.

This single entry displays the sum of the length of each wire style or cable reference. You can have a cable or wire BOM table in PDM with the required length.

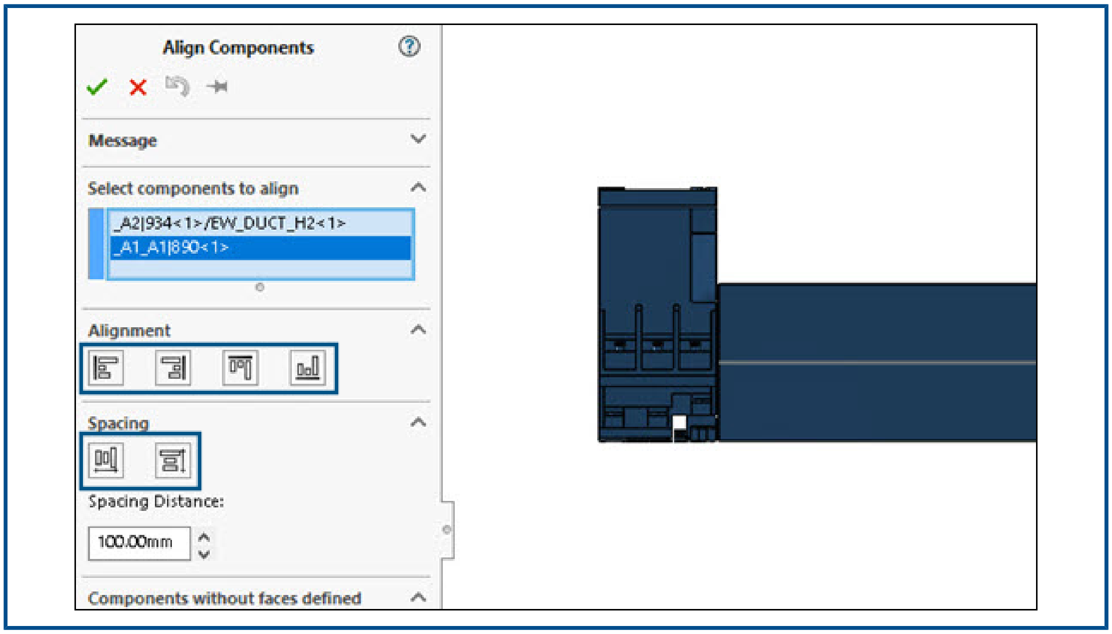
Zoom to Fit When Opening Drawings (2024 SP1)



When you open a drawing, you have the option to have it automatically zoom to fit your graphics area. The drawing can be a project drawing, a title block, a symbol, or a `dwg` file.

To activate this option, click **Interface configuration** > **Preferences**. Under **Behavior**, select **Zoom to fit when opening drawing**. This option helps you automatically view the entire extents of the drawing without additional **Zoom** commands.

Aligning Components

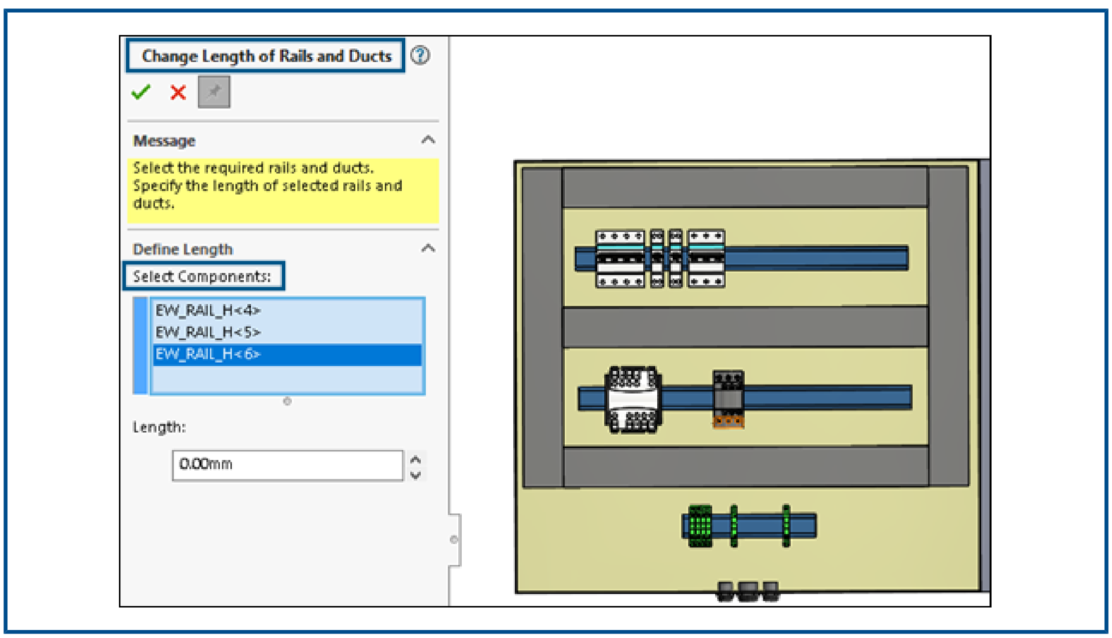


When you use **Align Components** while designing 3D cabinet layouts, you can preview changes in the graphics area.

This significantly reduces the effort required to align SOLIDWORKS components in 3D cabinet layouts.

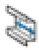
The Align Component PropertyManager has a simplified and improved workflow.

Changing the Length of Multiple Rails and Ducts

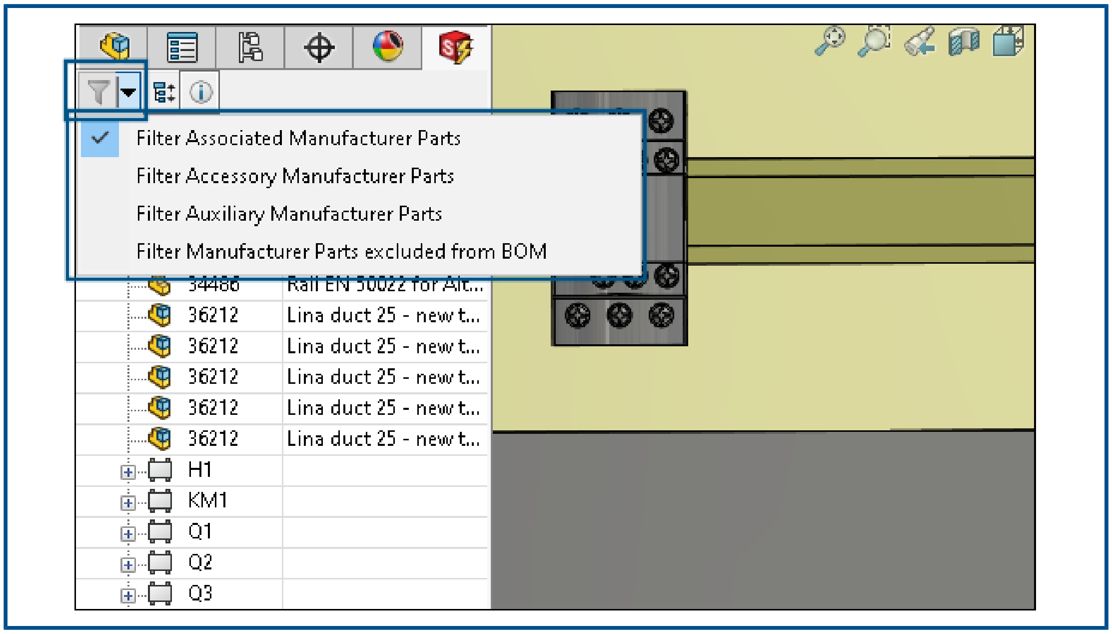


You can change the length of multiple rails and ducts simultaneously. In earlier releases, you could only change the length of a single rail or duct. The multiselection of rails and ducts makes the process of 3D cabinet creation faster.

To change the length of multiple rails and ducts:

1. In the SOLIDWORKS Electrical 3D menu, click **Change Length of Rails and Ducts** .
2. In the PropertyManager, under **Define Length** > **Select Components**, select multiple rails and ducts in the graphics area.

Filtering Auxiliary and Accessory Parts



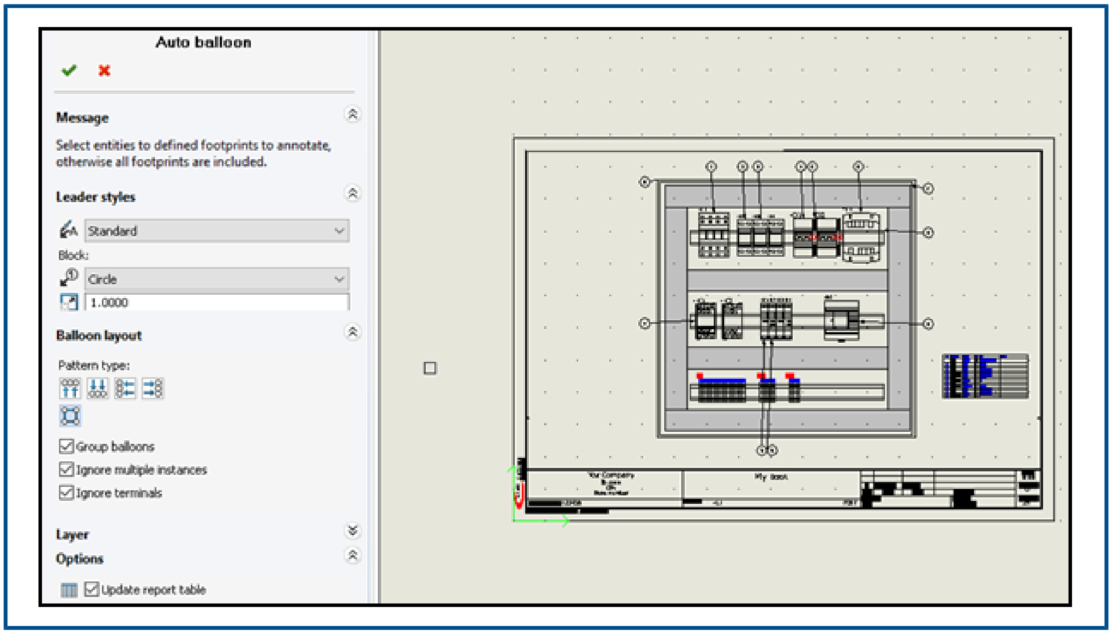
In SOLIDWORKS Electrical, you can filter manufacturer parts based on your selection. You can filter:

- Associated manufacturer parts
- Accessory manufacturer parts
- Auxiliary manufacturer parts
- Manufacturer parts excluded from the BOM

You can use the list in **Filter Manufacturer Parts** in the **Electrical Manager** tree to filter various types of manufacturer parts. **Show/Hide Associated Components** is replaced by this filter option.

This feature is also available in the 2D cabinet layout of SOLIDWORKS Electrical Schematic.

Auto Balloons in 2D Cabinets



You can insert auto balloons in SOLIDWORKS Electrical 2D cabinet layout drawings.

Inserting Auto Balloons in 2D Cabinets

To insert auto balloons in 2D cabinets:

1. Click **Cabinet Layout** > **Auto Balloon** .
2. Select a drawing view in which to insert the balloons.
3. In the PropertyManager, specify options and click .

Auto Balloon PropertyManager

To open this PropertyManager:

1. Click **Cabinet Layout** > **Auto Balloon** .

Leader styles

	Leader style	Specifies the predefined style to apply to leaders.
	Block	Specifies the block to use for the balloons.
	Scale	Specifies a number for the scale to apply to the block used for balloons.

Balloon layout

Specifies the **Pattern type**.

> For balloon marks, you can specify only the numeric values. Specifying formulas is not supported.

	Top	Displays balloons on the top of the cabinet drawing.
	Bottom	Displays balloons on the bottom of the cabinet drawing.
	Left	Displays balloons on the left of the cabinet drawing.
	Right	Displays balloons on the right of the cabinet drawing.
	Square	Displays balloons in a square surrounding the cabinet drawing.
	Group balloons	Displays the arrows of the grouped balloons with less tilt.
	Ignore multiple instances	Inserts balloons only for the first instance of the same manufacturer part.
	Ignore terminals	Does not insert balloons for the terminal strip.

Layer

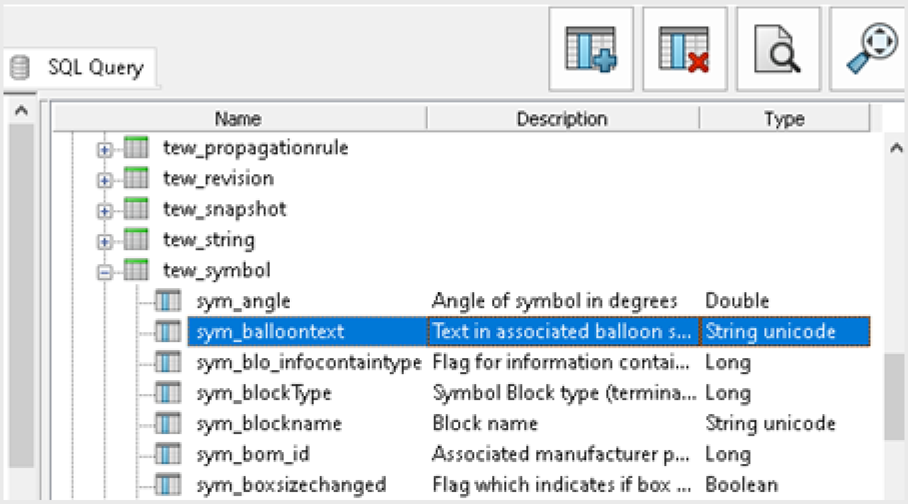
Specifies the layer on which to insert the balloons.

Options

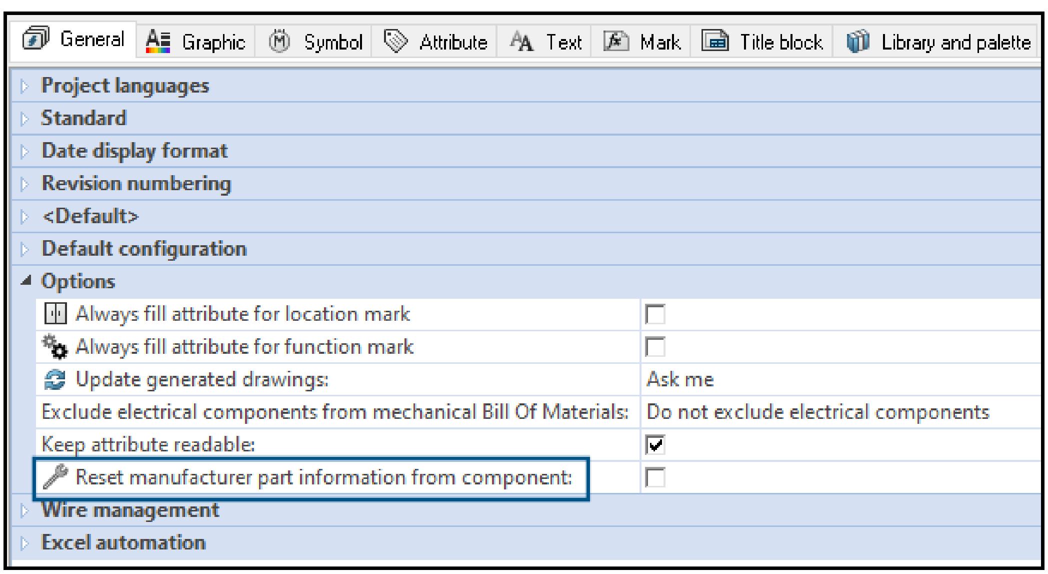
Insert report table. Inserts a report table filtered from the content of the current document.

To insert a report table, select **Insert report table** in the Auto Balloon PropertyManager. Click ✔, to open the panel to automatically insert the auto-ballooning report.

- If one or more report tables are already inserted, select **Update report table** to update the report tables.
- The Auto-balloon mark is data stored in the database, retrievable through a query, while the Report_Row is computed during report generation. There is no direct relationship between them.



Removing Manufacturer Part Data

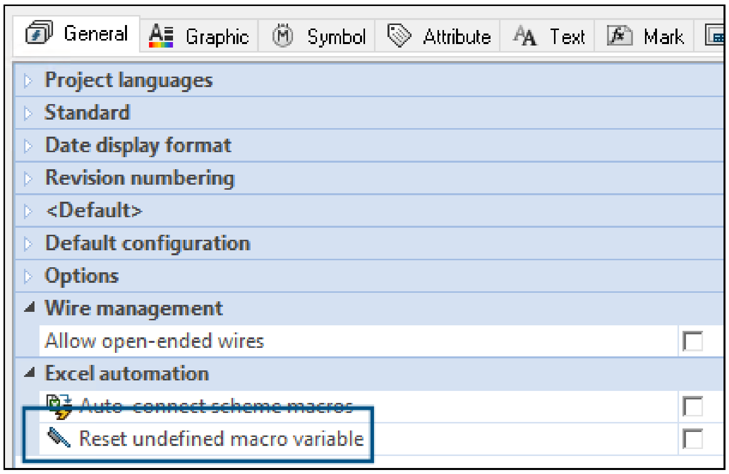


You can clear manufacturer part information when deleting or replacing a part from a component.

To remove manufacturer part data, click **Electrical Project** > **Configurations** > **Project.** In the Electrical Project Configuration dialog box, in the **General** tab, under **Options**, select **Reset manufacturer part information from component.** This resets the related information such as manufacturer data, terminal mark when you delete or replace it with a different part.

The option is cleared by default. If you clear this option, the part retains the terminal numbers even after you delete or replace it.

Resetting an Undefined Macro Variable

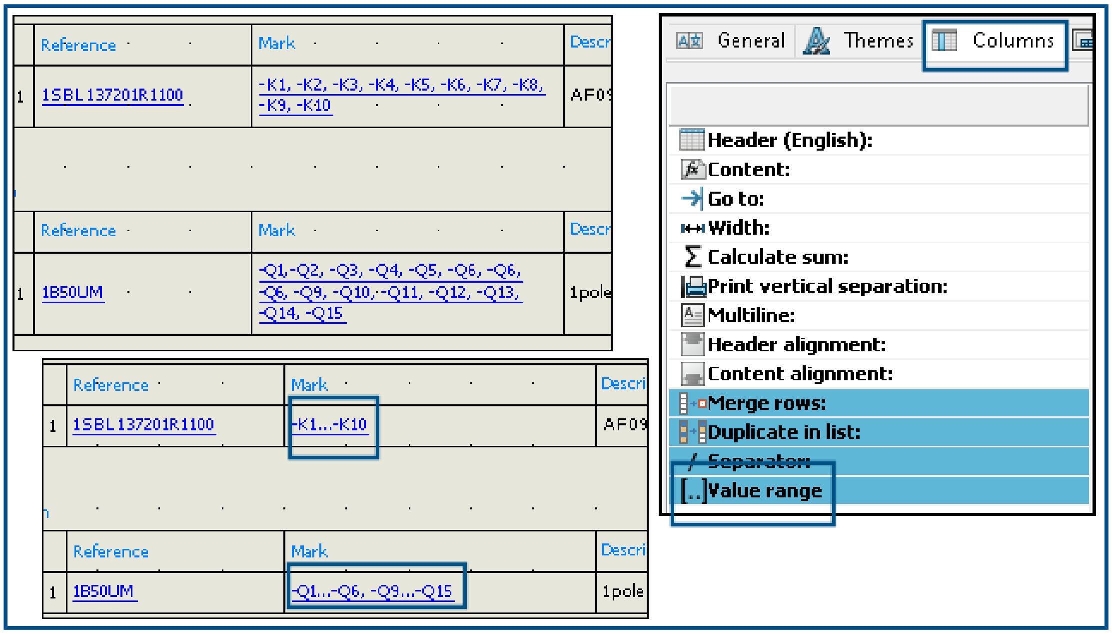


Excel automation lets you automatically reset undefined macro variables.

To reset undefined macro variables, click **SOLIDWORKS Electrical** > **Configurations** > **Project**. In the Electrical Project Configuration dialog box, on the **General** tab, under **Excel automation**, select **Reset undefined macro variable**. When you select this option, the `%xxx%` variable does not remain in the inserted macro. It is replaced by:

- An empty string
- A removed object
- Associated default object (like function or location)

Shortening Lists Using Ranges



In report configuration, when you merge rows, the software lists consecutive values as a range for merged rows instead of listing each individual value in the range.

In the Report configuration edition dialog box, under **Columns**, select **Value range**. To activate this option, select **Merge rows**. You can activate this option for multiple columns at once.

SOLIDWORKS Electrical Schematic Enhancements

SOLIDWORKS Electrical Schematic offers an improved user experience.

- In drawings, you can move entities using the arrow keys.
- The grid point size for the project sheets adapts automatically to the screen resolution.
- In a schematic project, when you set the side panels to **Auto Hide**, the panels retain the auto hide setting. This behavior increases the app's usability.

SOLIDWORKS Electrical Performance Improvement

Performance improvements include:

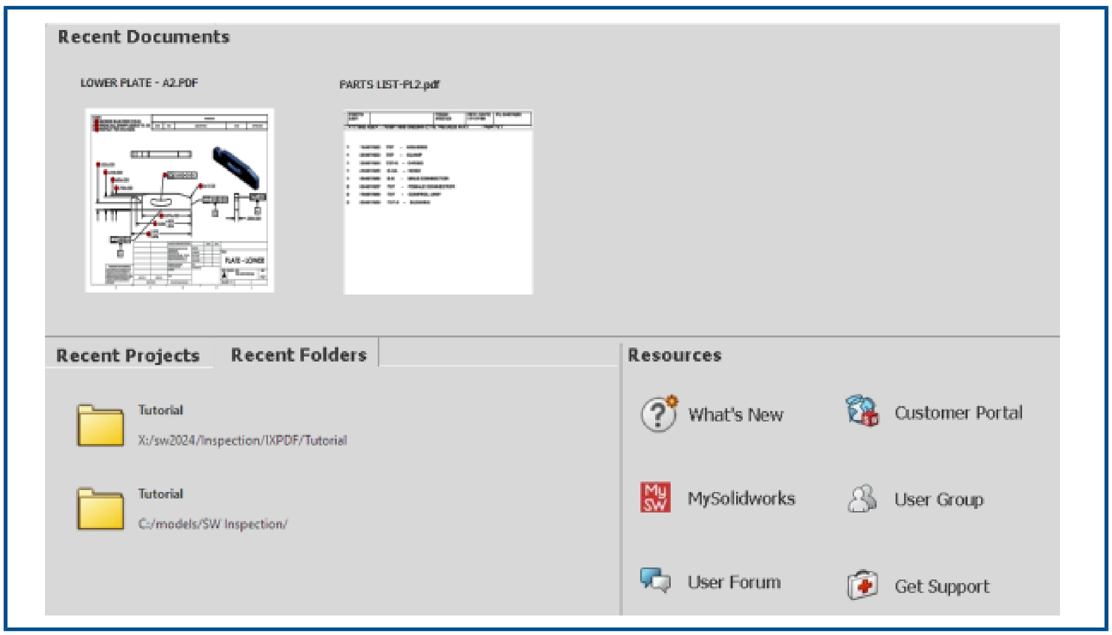
- Archiving a project for remote users (VPN connection) is improved and is much faster now.
- The automatic routing issue that caused the creation of loops while routing wires through splices is fixed. This allows cleaner and faster flattening of harnesses.

22

SOLIDWORKS Inspection

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*).

Welcome Page



The redesigned Welcome to SOLIDWORKS Inspection page in SOLIDWORKS Inspection Standalone improves usability.

The welcome page includes:

- **Recent Documents**
- **Recent Folders**
- **Recent Projects**
- **Resources**

23

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.

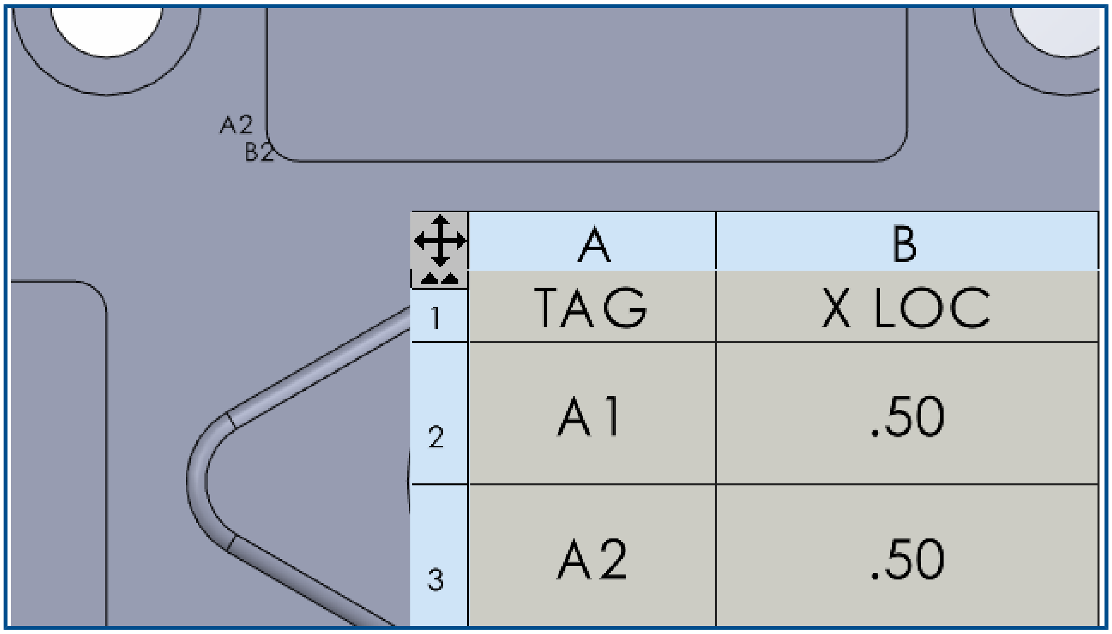
Specifying STEP Export Controls to STEP 242 (2024 SP3)

In the Publish to STEP242 PropertyManager, you can specify STEP export controls to add data to or remove data from a STEP 242 file.

To specify STEP export controls to STEP 242:

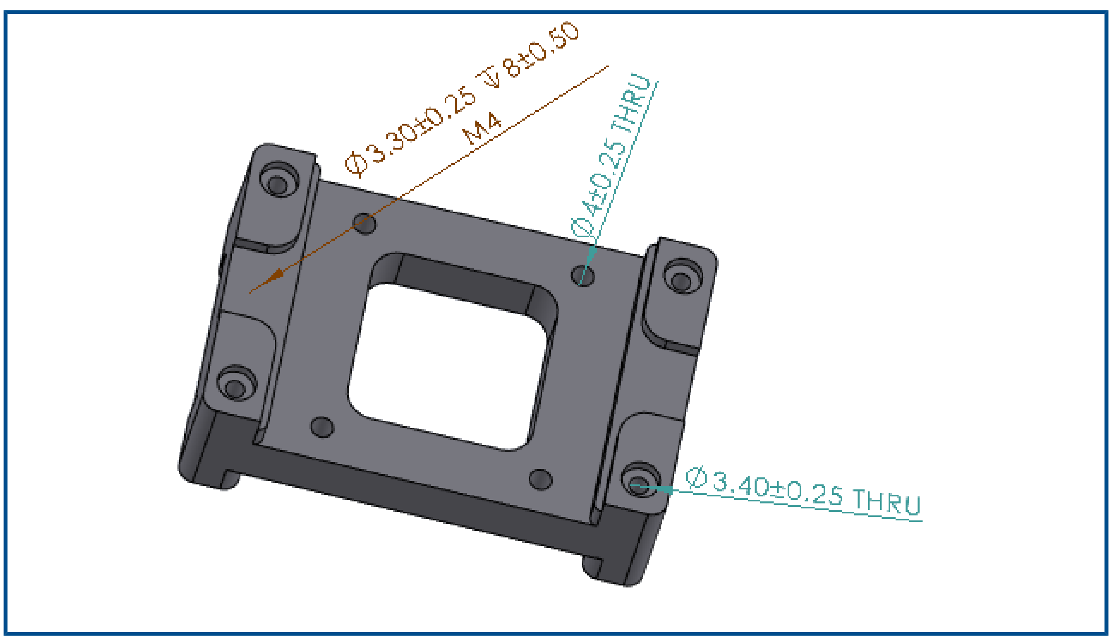
1. Click **Publish STEP 242 File** (MBD toolbar).
2. In the Publish to STEP242 PropertyManager, under **Step Export Settings**, specify an option:
   - **Split periodic faces**. Splits periodic faces, such as cylindrical faces, into two.
   - **Export face/edge properties**. Exports face and edge properties.
3. Click .
4. In the Save As dialog box, enter a file name.
5. Click **Save**.

Hole Tables



You can include a hole table when you publish a part to 3D PDF.

Repairing Dangling Dimensions



You can repair dangling DimXpert dimensions.

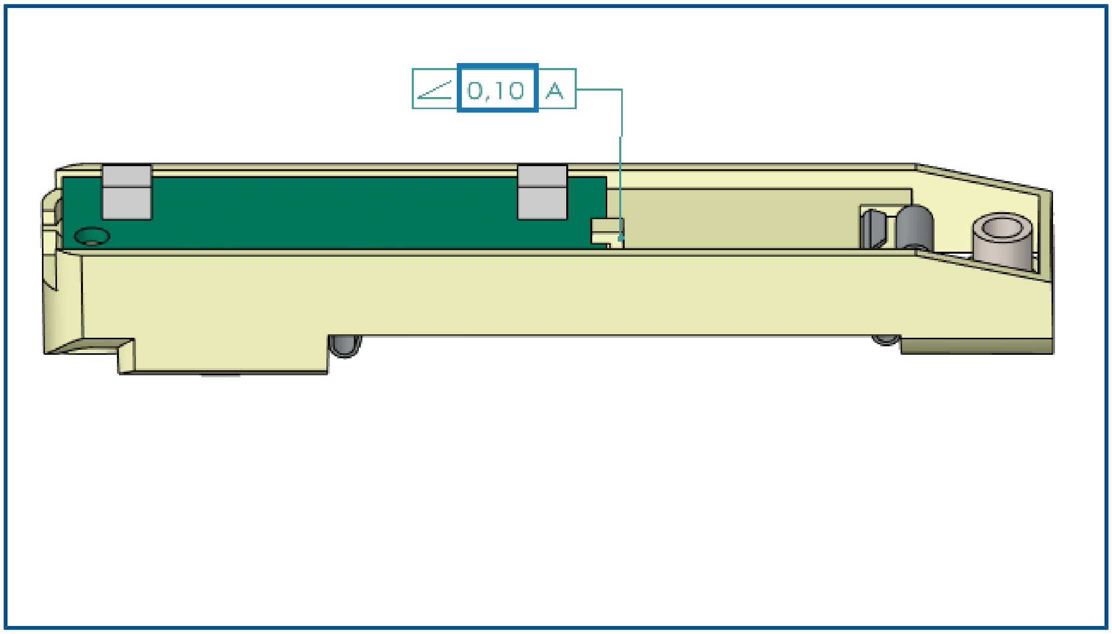
You can edit the dangling dimensions to reattach them to a feature in the model. This applies to dimensions created using the DimXpert tools, such as **Size Dimension** ,

Location Dimension, and the **Angle Dimension** tool. This tool is available for DimXpert dimensions only.

To repair dangling dimensions:

1. Open a part or assembly that contains dangling dimensions created with DimXpert tools.
2. In the DimXpertManager, right-click a feature and select **Edit Feature**.
3. In the PropertyManager, select the missing reference with the dangling dimension and click .

Adding a Decimal Separator in Geometric Tolerance Symbols

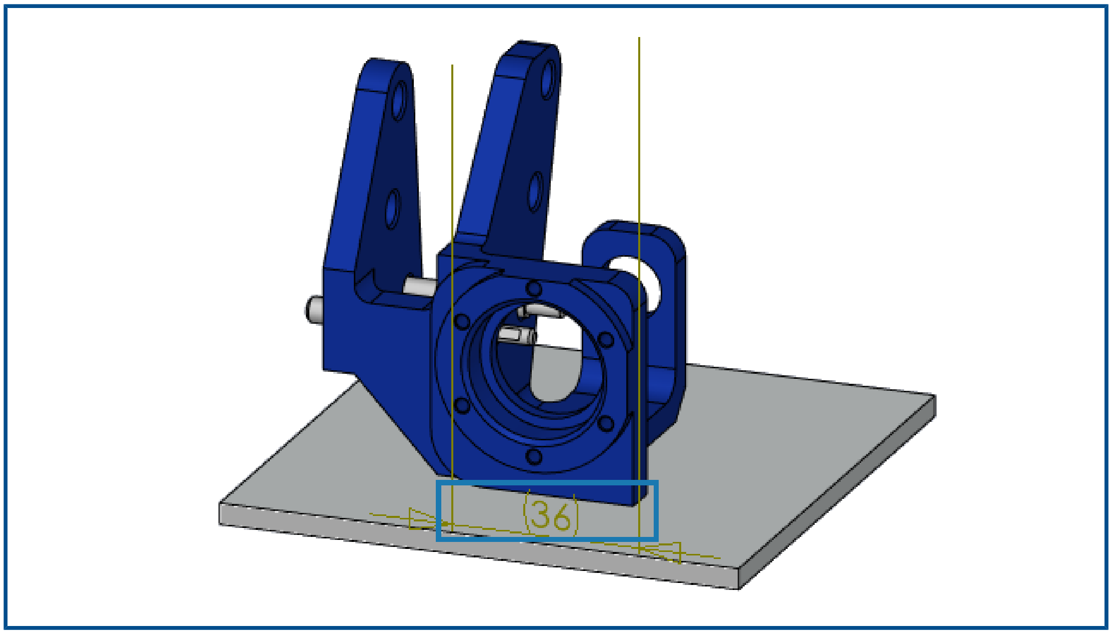


You can add a decimal separator in geometric tolerance symbols.

To add a decimal separator in geometric tolerance symbols:

1. Click **Tools** > **Options** > **Document Properties** > **Annotations** > **Geometric Tolerances**.
2. Under **Decimal Separator**, specify an option:
   - **Comma**. Inserts a comma.
   - **Period**. Inserts a period.

Controlling Visibility of Annotations through Solid Geometry

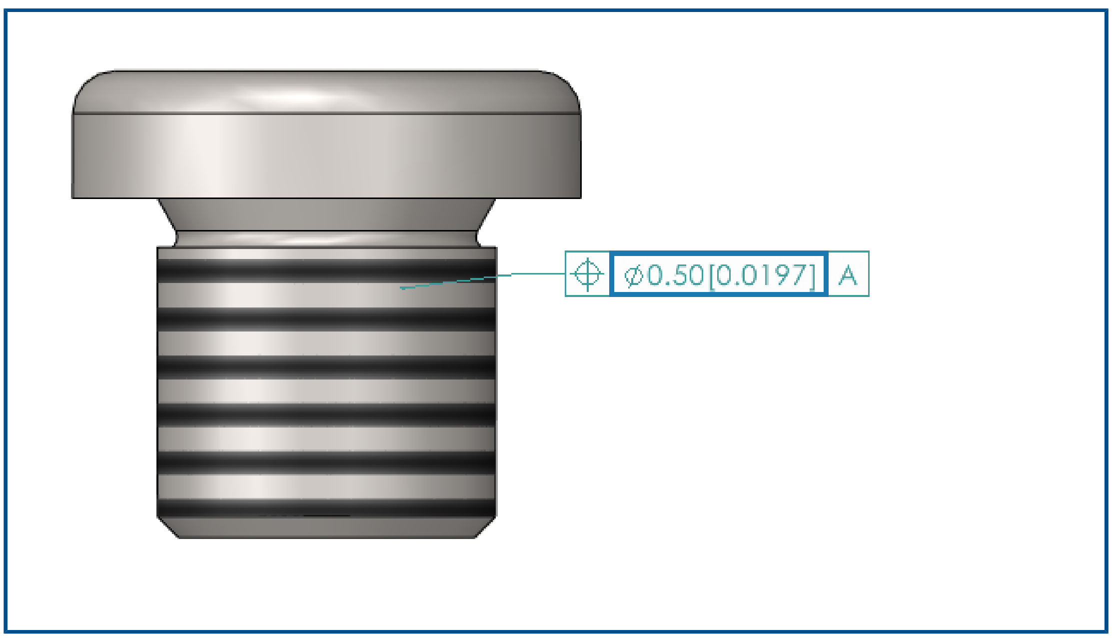


You can make annotations, such as dimensions, stay on top of the model. This lets you see dimensions and extension lines if you rotate the model.

To control visibility of annotations through solid geometry:

1. Click **Tools** > **Options** > **System Options** > **Display**.
2. Select **Display DimXpert dimensions on top of model**.

Displaying Dual Dimensions in Geometric Tolerance Symbols

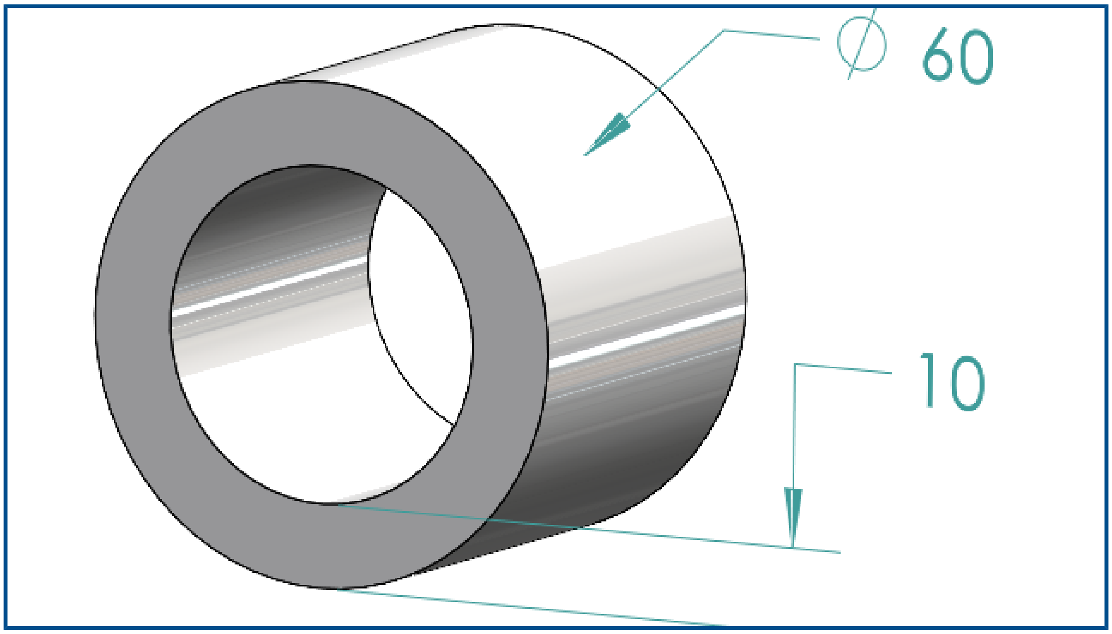


When you create geometric tolerance symbols, you can display dual dimensions, which show two sets of values, such as inches and millimeters, within a single dimension.

To display dual dimensions in geometric tolerance symbols:

1. In a part or drawing, click **Geometric Tolerance**  (MBD Dimension toolbar).
2. In the graphics area, click to place the symbol.
3. Select **Range** in the **Tolerance** dialog box and the **Geometric Tolerance** PropertyManager and select **Display Dual Dimensions**.

Creating Thickness Dimensions for Curved Surfaces



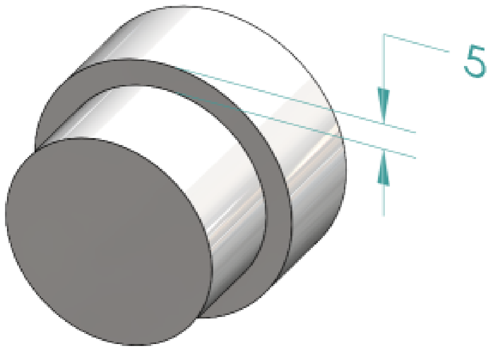
You can create thickness dimensions for curved surfaces.

This helps show the relationships between surfaces. You can apply thickness dimensions to:

- Cylinders
- Bosses
- Simple holes

You can create thickness dimensions between two concentric DimXpert features for:

- An inside and outside diameter, where the inside diameter is a cylinder or a simple hole, and the outside diameter is a cylinder or a boss.
- Two inside diameters of a cylinder or simple hole.
- Two outside diameters of a cylinder or boss. For example:



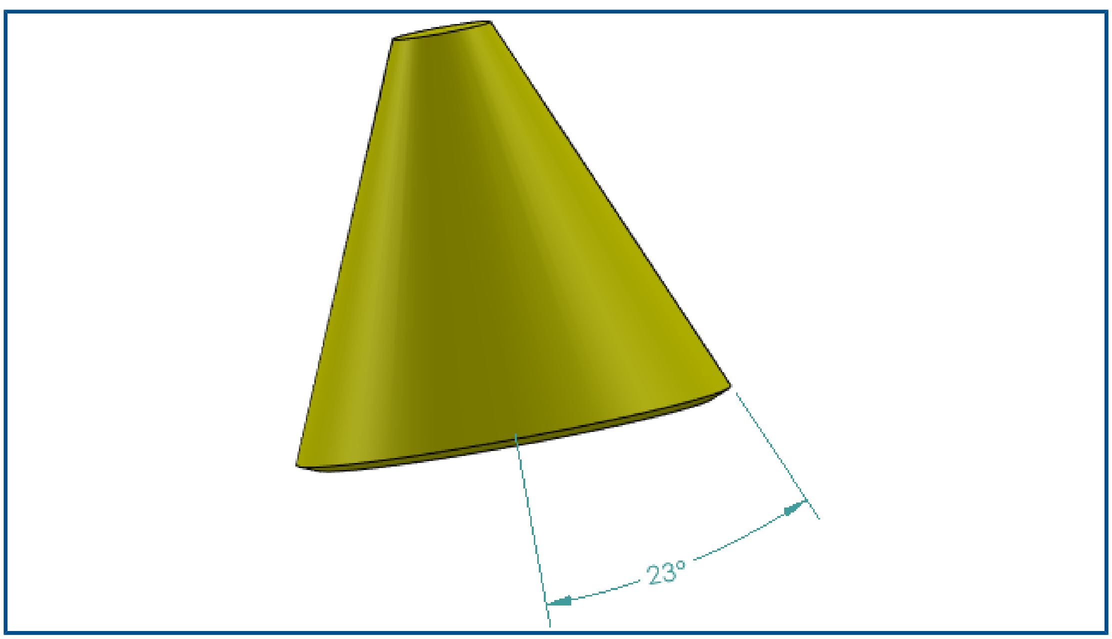
To create thickness dimensions for curved surfaces:

1. Click **Location Dimension** (MBD Dimension toolbar).

   Steps 2 and 3 require that you select two features. For thickness dimensions, the two features must be cylindrical, concentric, and have different diameters.

2. Select the face of the origin feature.
3. Select the face of the tolerance feature.
4. Click to place the dimension.
5. Specify options in the PropertyManager and click .

Displaying Half Angles of Conical Dimensions

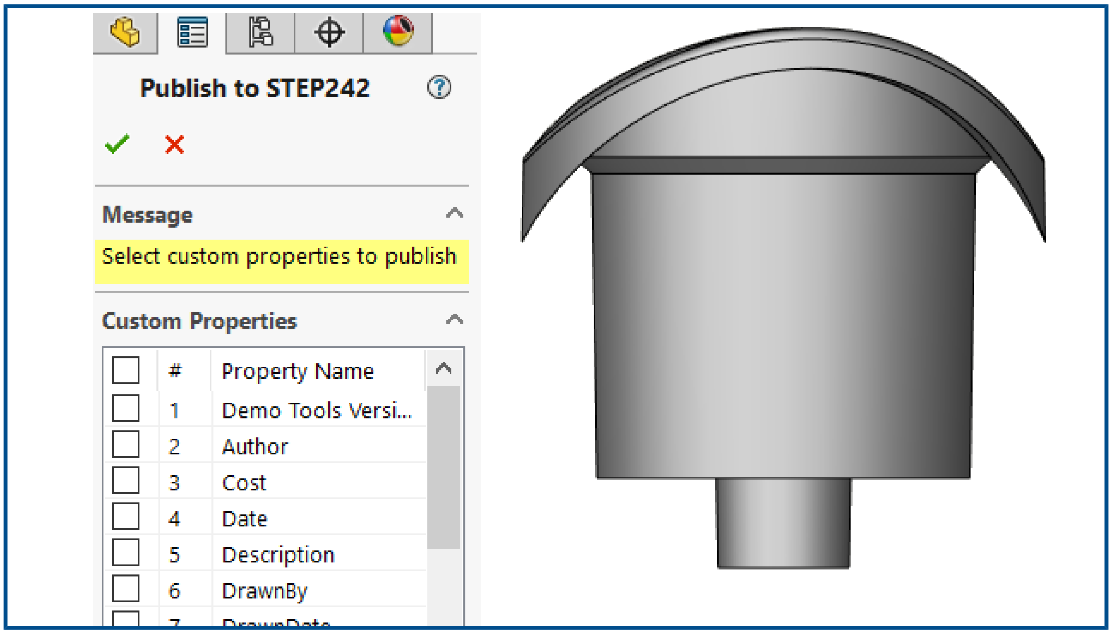


You can display a conical angle dimension as a half angle. This lets you convert a full angle of a cone to a half angle.

To display half angles of conical dimensions:

1. In the DimXpert Value PropertyManager, under **Primary Value**, select **Show as half angle**.

Exporting Custom Properties to STEP 242



You can export custom properties from a part or assembly to the STEP 242 format.

To export custom properties to STEP 242:

1. Click **Publish STEP 242 File** (MBD toolbar).
2. In the Publish to STEP242 PropertyManager, specify custom properties to export and click ✓.
3. In the Save As dialog box, enter a file name.
4. Click **Save**.

Viewing Annotations and Dimensions

You can view annotations and dimensions in a more organized way.

As of SOLIDWORKS 2024 and later, you do not need a SOLIDWORKS MBD license for this functionality.

You can use the following features:

- List annotations in a tree view. When you select an annotation in the FeatureManager design tree, it highlights the annotation in the graphics area, and you can hide or show annotations.

- Sort by annotation type. You can sort annotations by type, such as smart dimensions, weld symbols, and balloons for better organization.

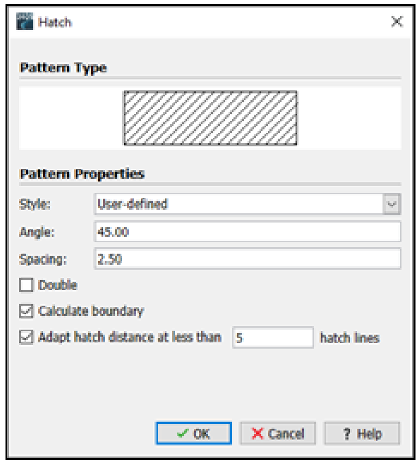
24

DraftSight

This chapter includes the following topics:

DraftSight® is a separately purchased product that you can use to create professional CAD drawings. It is available as DraftSight Professional, DraftSight Premium, and DraftSight Mechanical. In addition, DraftSight Enterprise and Enterprise Plus are available on network license. **3D**EXPERIENCE® DraftSight is a combined solution of DraftSight with the power of the **3D**EXPERIENCE platform.

Hatch Commands (DraftSight Mechanical Only) (2024 SP3)



You can run the **AM_UserHatch** command to apply user-defined or predefined hatches on enclosed geometry.

You can run the **AM_UserHatchEdit** command to edit the hatches.

When you run these commands, the Hatch dialog box opens where you can:

- Specify the angle of hatch lines.
- Specify spacing between hatch lines.
- Specify the number of hatch lines if the area to hatch is small enough to match the specified pattern.
- Calculate the new boundaries of an area when editing a hatch.

Applying User-Defined or Predefined Hatches

You can apply user-defined or predefined hatches on geometry in the graphics area.

To apply user-defined or predefined hatches:

1. Type `AM_UserHatch` in the command window.
2. In the dialog box, from **Style**, select **User-defined**.
   a) In **Angle**, enter the angle of the hatch lines.
   b) In **Spacing**, enter the spacing between hatch lines.

3. Optional: Select one of the following predefined hatches.

   The software creates hatch patterns with specific angle and spacing between hatch lines.

   You can override the **Angle** and **Spacing** values of predefined hatches.

Hatch	Angle	Spacing
	45°	2.5 mm or 0.1 in
	45°	5 mm or 0.22 in
	45°	13 mm or 0.5 in
	135°	2.7 mm or 0.12 in
	135°	4.7 mm or 0.19 in
	135°	11 mm or 0.4 in
	45°/135°	2.3 mm or 0.09 in

4. Optional: Select **Double** to create the cross pattern with hatch lines perpendicular to the primary lines.
5. Optional: In **Adapt hatch distance at less than**, enter the number of hatch lines if the area to hatch is small enough to match the specified pattern.

   The default number of lines is 5.
6. Click **OK**.
7. In the graphics area, specify an internal point in an enclosed area of the geometry.

Editing User-Defined Hatches

You can quickly edit user-defined hatches in the graphics area.

To edit user-defined hatches:

1. Type `AM_UserHatchEdit` in the command window.
2. In the graphics area, select a user-defined hatch.
3. In the dialog box, from **Style**, select a new predefined hatch pattern.
4. In **Angle**, edit the hatch angle value.
5. In **Spacing**, edit the distance between the hatch lines.
6. Select **Double** to create a cross pattern with hatch lines perpendicular to the primary lines.
7. Select **Calculate boundary** to create new boundaries of the hatch area.
   a) In the graphics area, specify a point in an area to hatch.

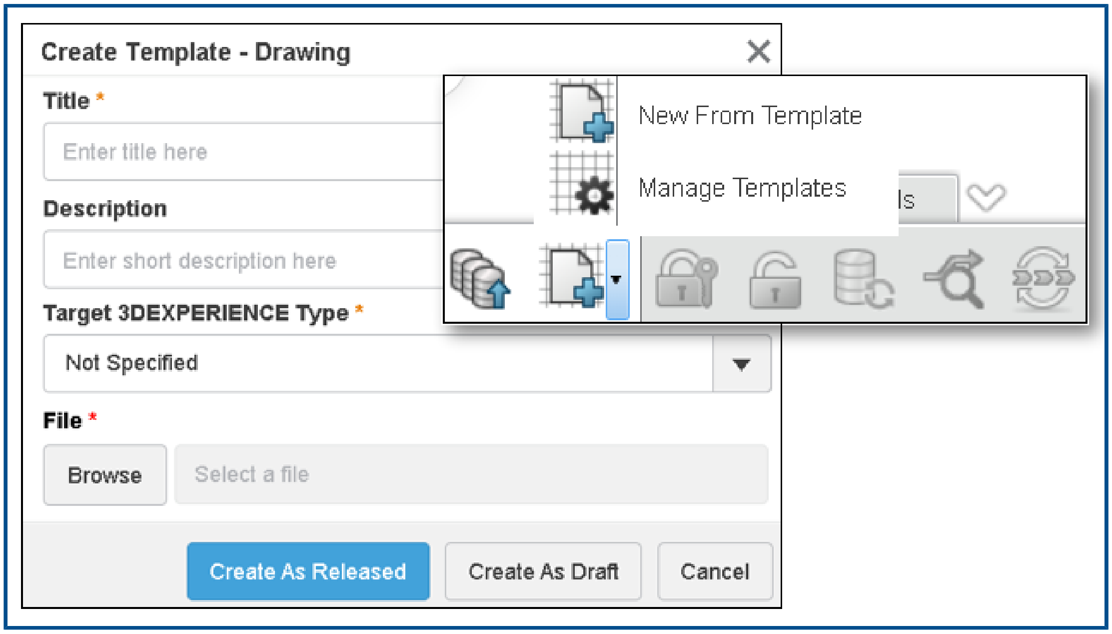
      Alternatively, you can select **Specify entities** and specify the entities to hatch.

   DraftSight deletes the hatch that you selected in step 2.

8. Optional: In **Adapt hatch distance at less than**, enter the number of hatch lines if the area to hatch is small enough to match the specified pattern.

   The default number of lines is 5.
9. Click **OK**.

Templates on the 3DEXPERIENCE Platform (DraftSight Connected Only) (2024 FD01)



You can create, save, and manage templates on the **3D**EXPERIENCE platform. You can access these templates to create new drawings.

Previously, you could save and access your templates locally only.

Creating a Template from a Drawing

You can create a new template from the locally saved drawing file.

To create a template from a drawing:

1. In the **My Session** widget, from the action bar, click **Manage Templates**.

   The Manage Templates dialog box displays the templates created on the platform.
2. Click **Add template**.
   a) In the Create Template - Drawing dialog box, enter the **Title** and **Description**.
      You can have multiple templates with the same name.
   b) For **Target 3DEXPERIENCE Type**, select **Drawing**.
   c) Click **Browse** and select a locally saved drawing file.
      You cannot attach one drawing file to multiple templates.
   d) Click **Create As Released** or **Create As Draft**.
3. Optional: Click **Edit Template** to edit the templates that are not in the released state.

4. Optional: Click **Download Template** to download the drawing file associated with the template.

   The software downloads the file to `C://3DEXPERIENCE/MyWork`.

5. Optional: Click **Maturity** to change the maturity state.
6. Optional: Click **Delete Template** to delete the template.
7. Optional: Click **Reload Template** to reload the list of templates.

   > If you create a template as released, you cannot edit or delete it, or change its maturity state.

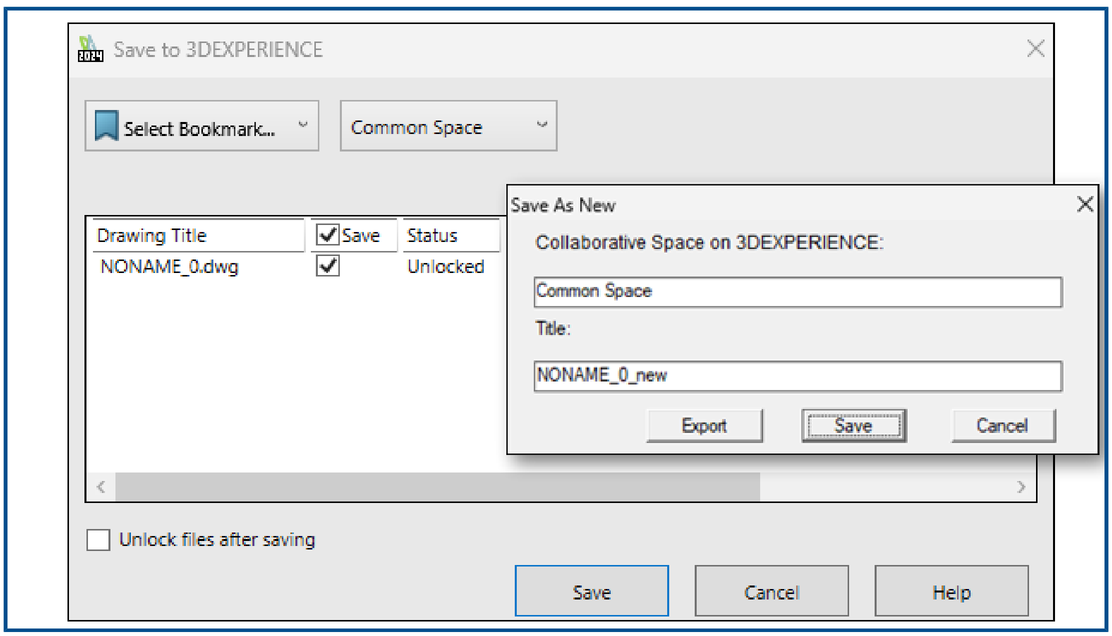
Creating a Drawing from a Template

You can create a drawing from the template saved on the **3D**EXPERIENCE platform.

To create a drawing from the template:

1. In the My Session widget, from the action bar, click **New From Template**.
2. In the dialog box, select the template saved on the platform.
3. Enter the file name and click **OK**.
4. Optional: Save the drawing file on the platform.

Saving a File to the 3DEXPERIENCE Platform (DraftSight Connected Only) (2024 FD01)



You can select a bookmark, change the collaborative space, and update the title of new files from the Save to 3DEXPERIENCE dialog box.

The Save as New dialog box lets you save a file that is saved on the **3D**EXPERIENCE platform with a new name.

When you save a file to the **3D**EXPERIENCE platform, the progress bar displays a message that includes the file name and name of the collaborative space.

Save as New Dialog Box

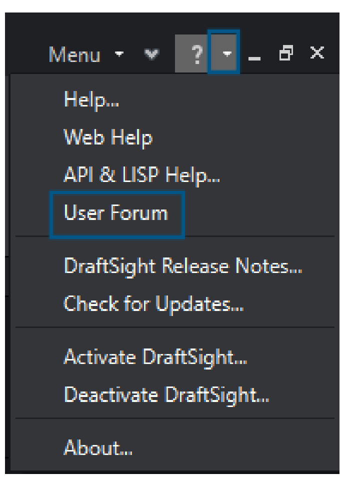
You can use this dialog box to save a file that is saved on the **3D**EXPERIENCE platform with a new name.

To access the dialog box, do one of the following:

- Right-click the drawing tab and click **Save as New**.
- Enter the `SAVEASNEW` command in the command window.

Option	Description
Collaborative Space on 3DEXPERIENCE	Displays the collaborative space on which you have saved the file.
Title	Displays the title with **new** as suffix. You can edit the title.
Include References	Available only when the file has references.
Export	Exports DraftSight files locally.
Save	Saves the file on the **3D**EXPERIENCE platform.

Accessing the DraftSight User Forum (2024 SP1)



You can access the DraftSight user forum that contains posts from the DraftSight user community.

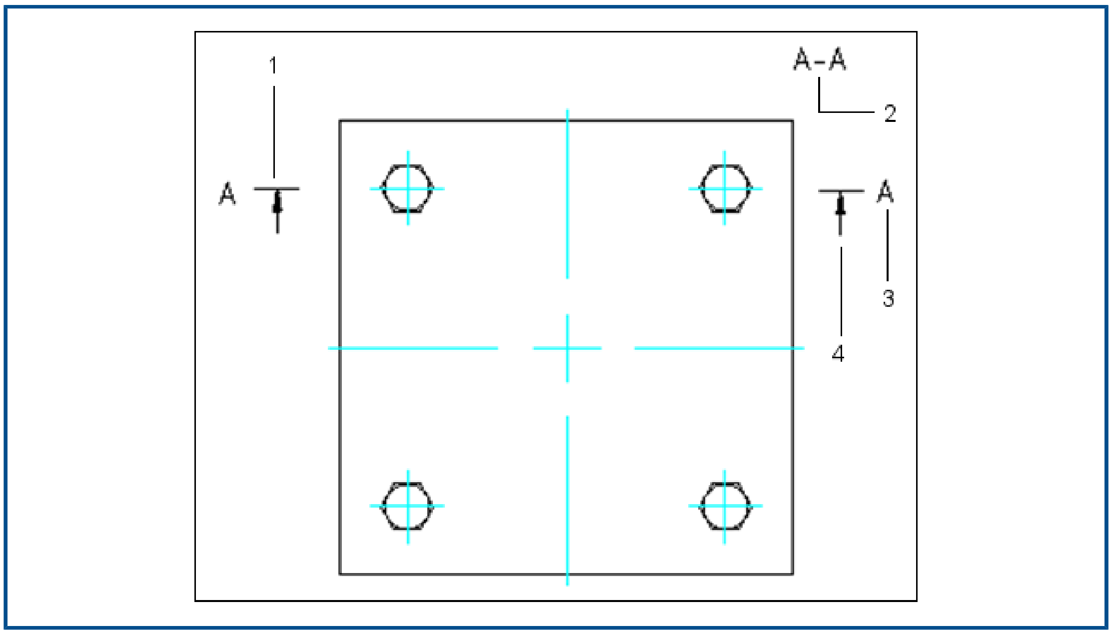
To access the user forum:

Do one of the following:

- Click ▾ and select **User Forum**.
- Type `UserForum` in the command window.

When you click **User Forum**, DraftSight redirects you to the **3D**EXPERIENCE platform. Access to the **3D**EXPERIENCE platform requires **3D**EXPERIENCE credentials.

Section Line Command (DraftSight Mechanical Only) (2024 SP1)



You can create a section line at the cutting plane of the section and insert the corresponding section view label in the drawing area.

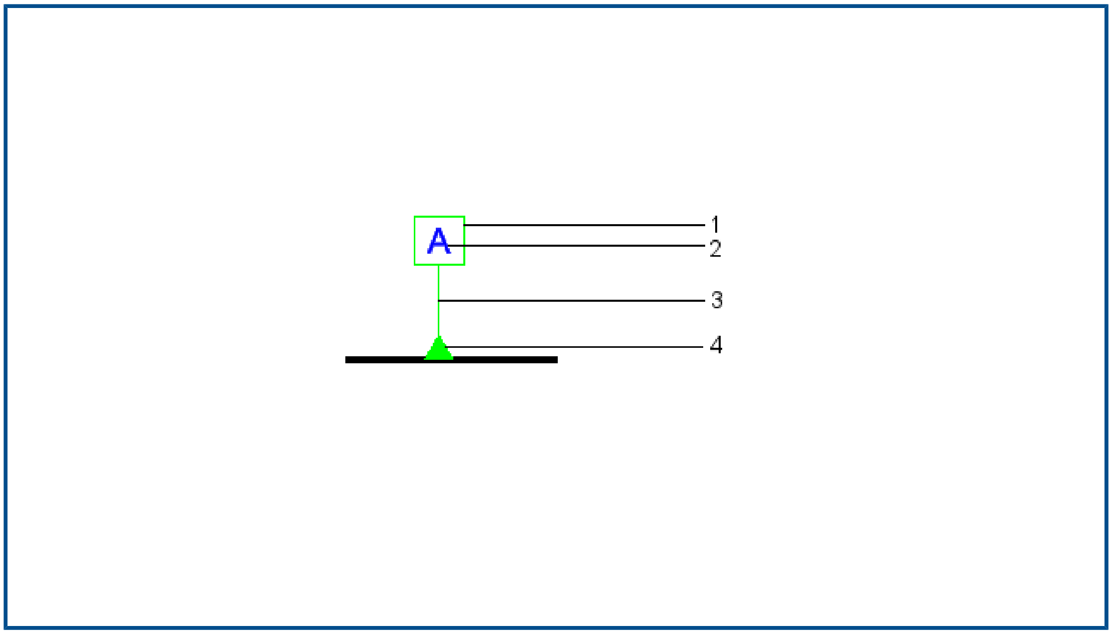
Enter the `AM_SectionLine` command to draw section lines. The command creates the following entities:

Entity	Description
1	Section line
2	Section view label
3	Section view identifier
4	Direction arrow

The command lets you control the appearance of different entities of the section line, such as arrows, lines, and name. You can create multiple sections on an entity for the following types of section views:

Type of section view	Description
Full section	The cutting plane passes through the entire length of the entity.
Aligned section	Two nonparallel cutting planes pass through the entity. Use these sections on cylindrical entities.
Half section	The cutting plane passes through a portion of the entity to section.
Offset section	The cutting plane bends to pass through the features of the entity. Use these sections on entities that are not in a straight line.

Datum Identifier Commands (DraftSight Mechanical Only) (2024 SP1)



You can use datum identifier commands to add a datum identifier and attach it to areas in a drawing.

A datum is a plane, a straight line, or a point used as a reference to measure and locate geometric entities and geometric tolerances. You can use the following commands:

- `AM_DatumIdentifier` to create datum identifier symbols.
- `AM_DatumIdentifierEdit` to edit datum identifier symbols.

Datum identifier symbols identify datum features for feature control frame symbols. For example, you can use a datum identifier symbol to mark the center of a hole.

Elements of datum identifier symbols include:

1	Square frame
2	Datum identifier of two capital letters maximum
3	Leader arrow
4	Triangle symbol

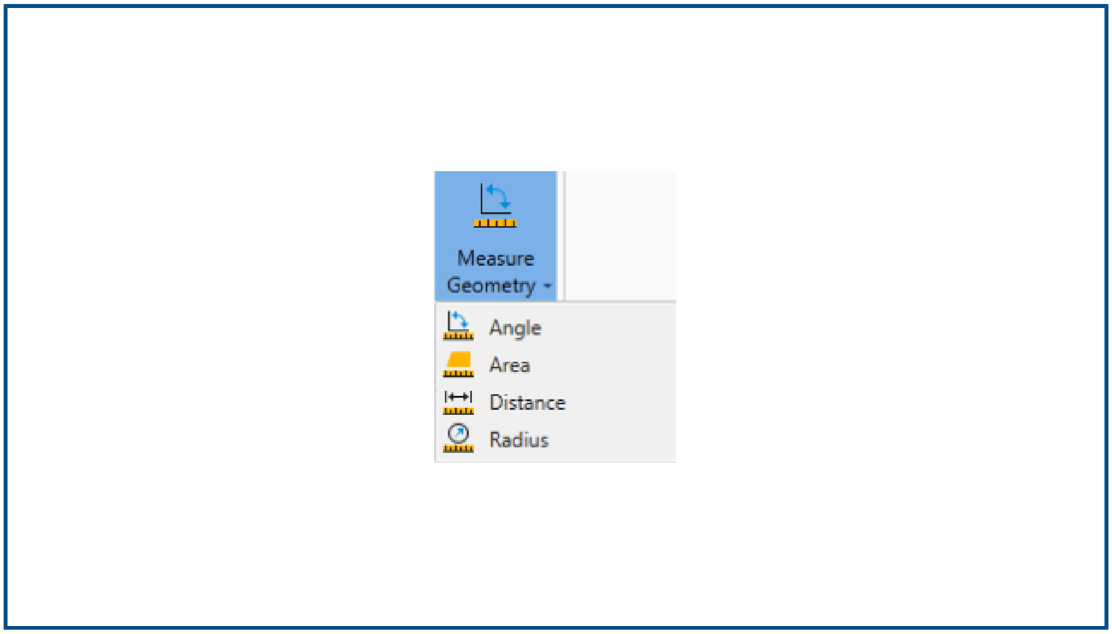
When you create a datum identifier symbol in a drawing, the software generates a label that contains the datum identifier enclosed in a rectangle. The datum identifier appears in all feature control frames that use the datum as a reference. A leader line connects the label to the datum on the drawing. The leader line may include a filled or empty triangle. The position of the triangle indicates the corresponding datum.

You can attach datum identifier symbols on:

- A surface or on one extension line of a surface

- Visible lines such as extension lines, dimensions, or axes
- A hole, leader pointing to a hole, or feature control frame

Measure Geometry Command



You can use the MEASUREGEOM command to measure an area, angle, distance, and radius.

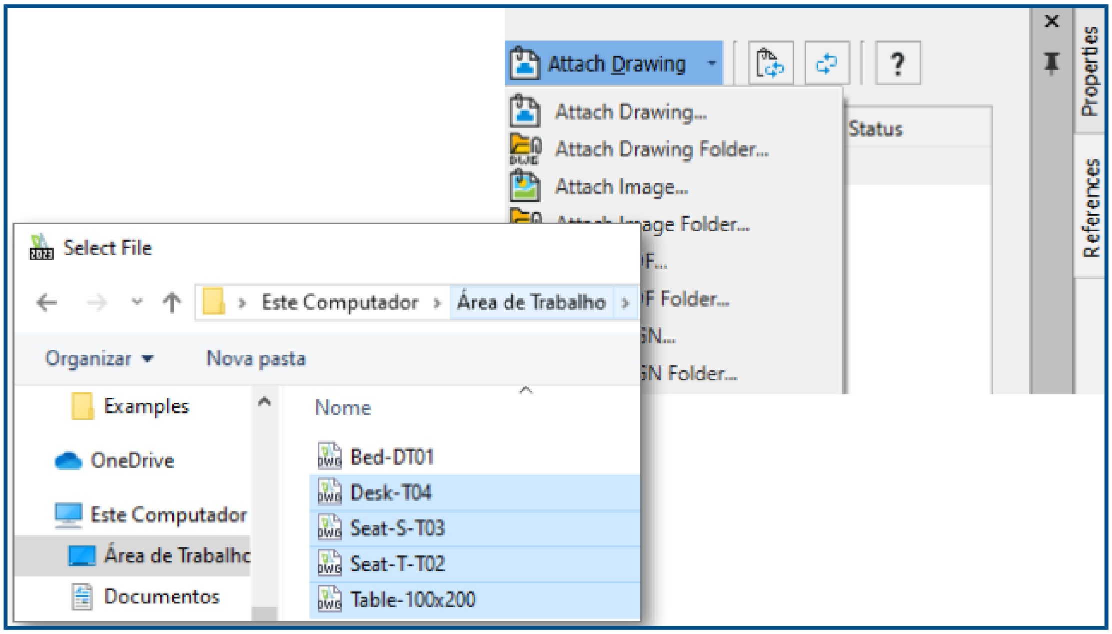
In previous releases, you had to run commands like AREA, DIST, and GETANGLE.

To access the Measure Geometry command:,

Do one of the following:

- On the ribbon, click **Home** > **Tools** > **Measure Geometry**.
- Enter MEASUREGEOM in the command window.

Selecting Multiple Files and Inserting as Reference



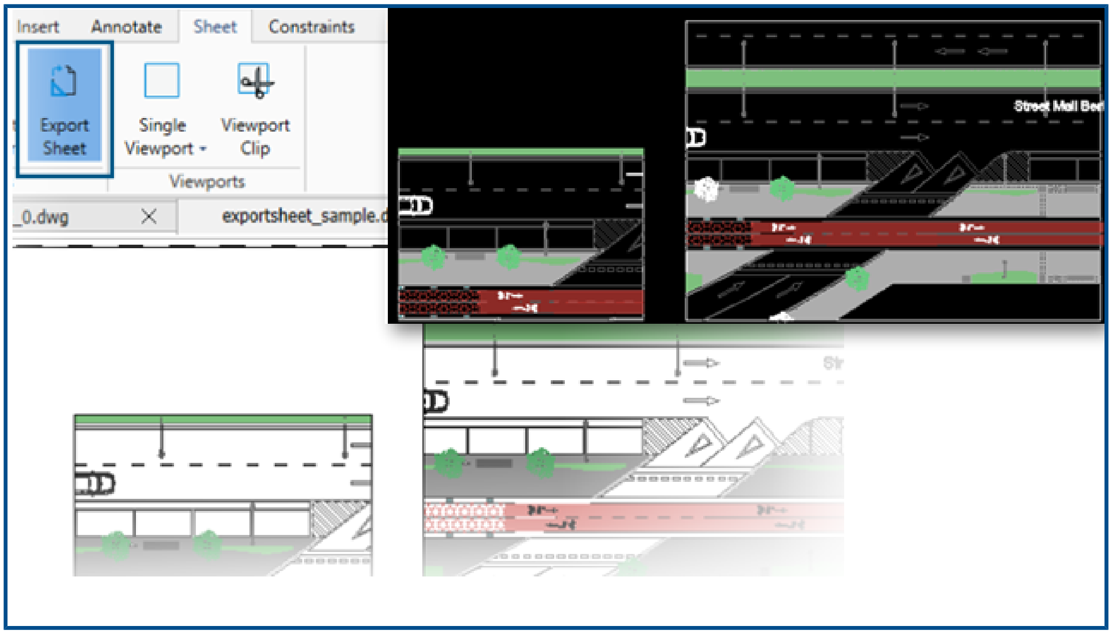
You can select multiple files and folders and insert them as external references to the DWG™ file. This reduces the number of clicks required to insert multiple files and the possibility of failing to insert a file.

To select multiple files and insert them as references:

Do one of the following:

- On the ribbon, click **Insert** > **Block** > **References Manager**.
- On the ribbon, click **Attach**.
- On the menu, click **Tools** > **References Manager**.
- Enter `REFERENCES` in the command window.

Export Sheet Command



You can export all visible entities from an active sheet viewport and entities from the sheets to the new drawing.

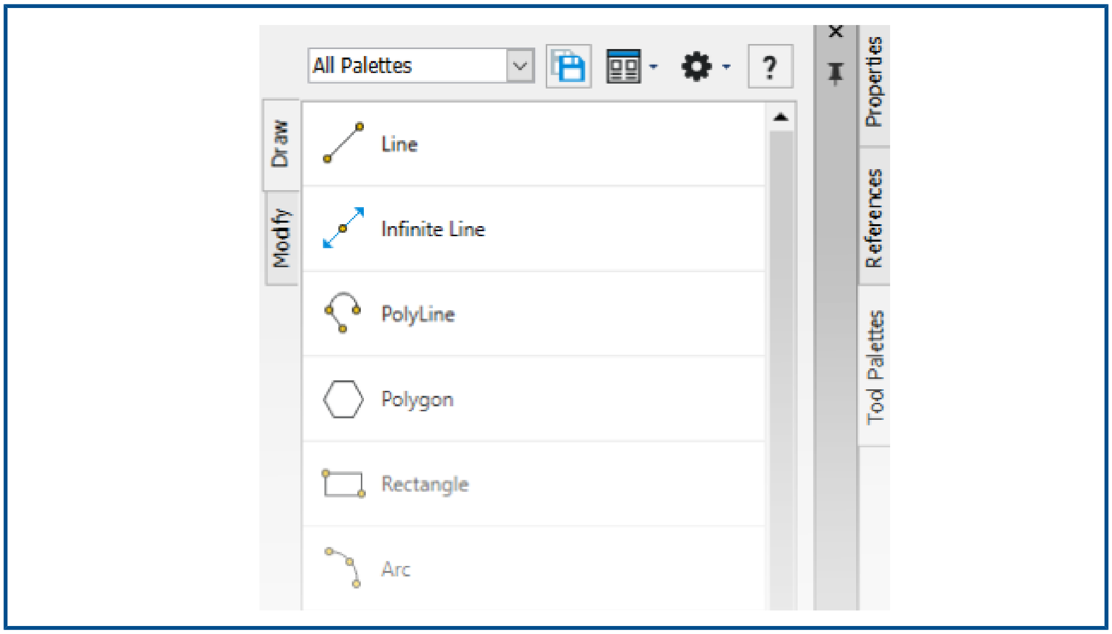
This lets you edit the representation created in the new drawing using commands like `TRIM`, `COPY/PASTE`, `EXPLODE`, `STRETCH`.

To access the Export Sheet command:

Do one of the following:

- On the ribbon, click **Sheet** > **Sheets** > **Export Sheet**.
- On the menu, click **File** > **Export** > **Export Sheet**.
- Enter `EXPORTSHEET` in the command window.

Tool Palettes



You can find frequently used tools and data in the Tool Palettes.

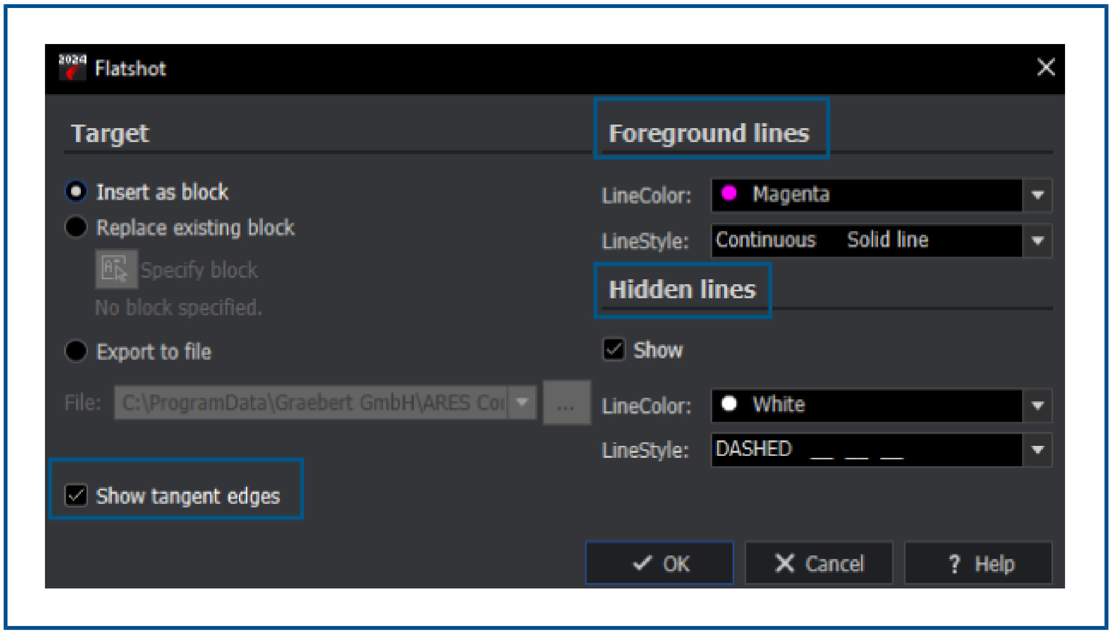
The palettes include all generic properties like docking and auto-hide. You can also create your own palette to store tools and data.

To access the Tool Palettes:

Do one of the following:

- On the ribbon, click **Insert** > **Palettes** > **Tool Palettes**.
- On the menu, click **Tools** > **Tool Palettes**.
- Enter `TOOLPALETTES` in the command window.

Make Flat Snapshot Command



You can use the enhanced features of the `MAKEFLATSNAPSHOT` command for formatting the foreground and hidden lines, and displaying tangent edges.

To access the Make Flat Snapshot command:

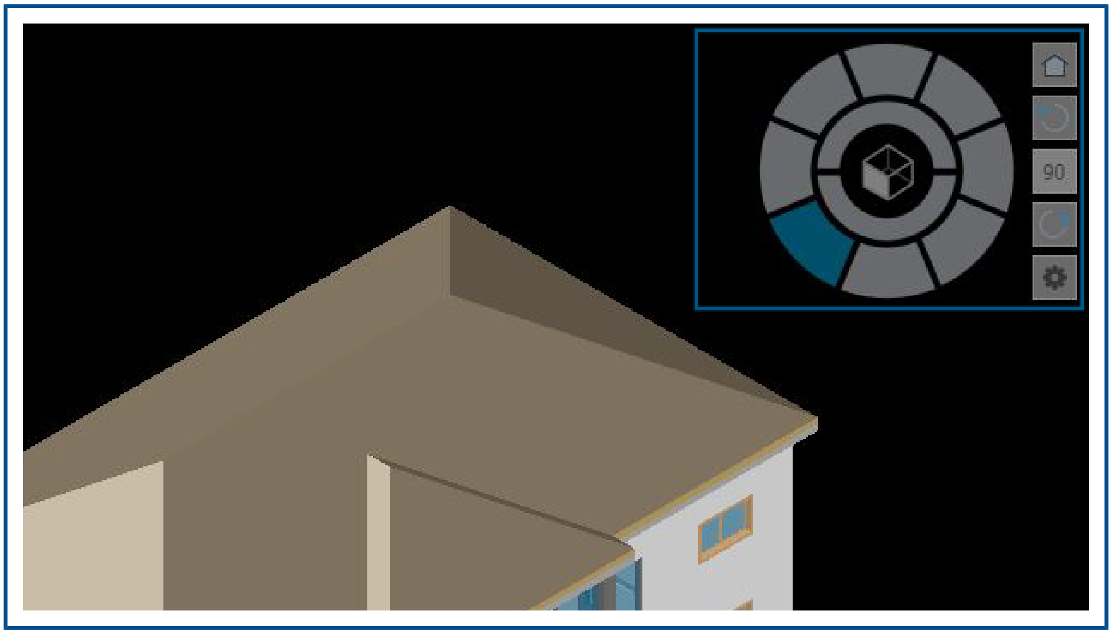
Do one of the following:

- On the ribbon, click **Home** > **Snapshot** > **Make Flat Snapshot**.
- On the menu, click **Solids** > **Solid Editing** > **Make Flat Snapshot**.
- Enter `MAKEFLATSNAPSHOT` in the command window.

The enhanced features include:

- **Foreground lines**. **LineColor** and **LineStyle** specify the line color and style of foreground lines.
- **Hidden lines**. **Show** displays the hidden lines. **LineColor** and **LineStyle** specify the line color and style of hidden lines.
- **Show tangent edges**. Displays tangent edges in the flat representation.

View Navigator



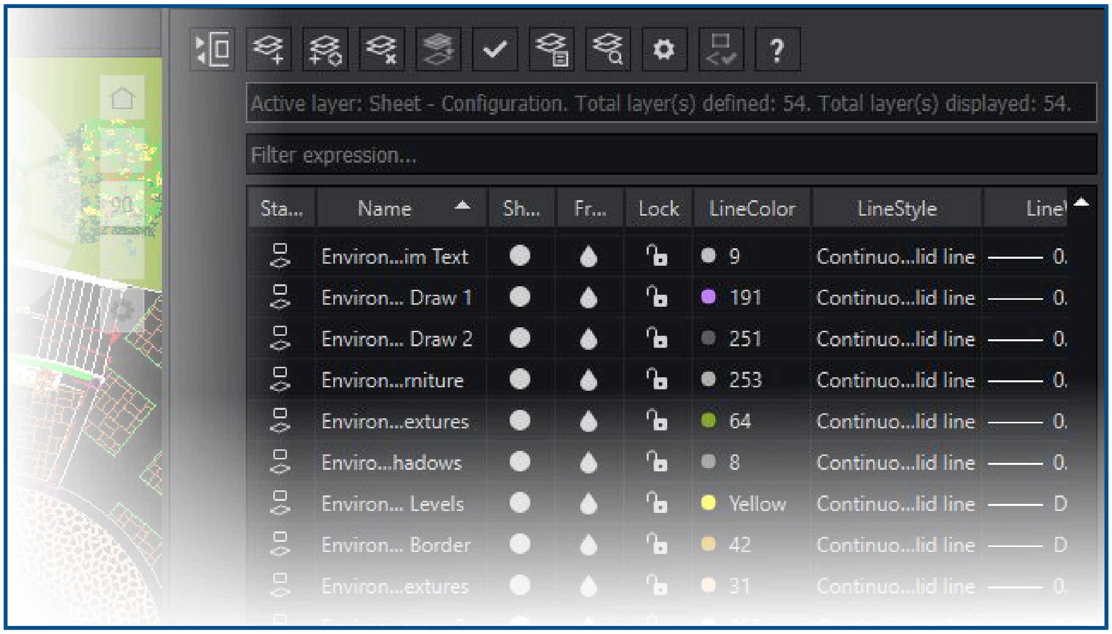
View Navigator lets you switch between standard and isometric views or parallel and perspective views of a model.

Its interface acts as a 3D orientation indicator that lets you see the current view direction.

To access the View Navigator command:

- On the ribbon, click **View** > **Views** > **View Navigator**.
- On the menu, click **View** > **View Navigator**.
- Enter `VIEWNAVIGATOR` in the command window.

Layer Manager Palette



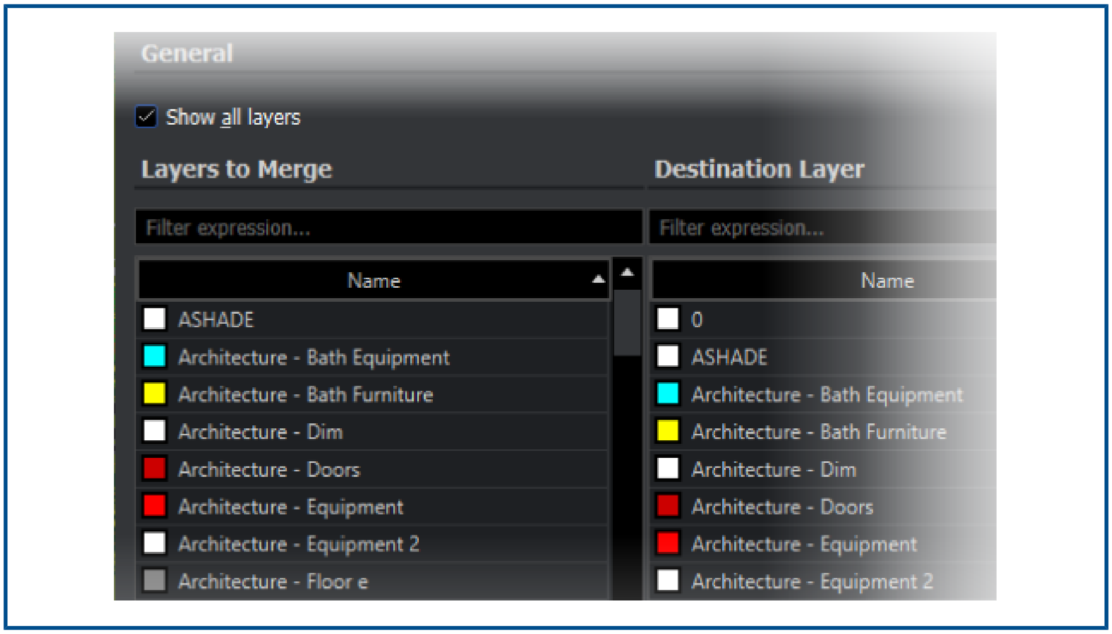
You can use the Layer Manager dialog box as a palette that you can float or dock on the side.

In the Layer Manager palette, you have quick access to layers, layer states, layer previews, or isolating layers.

To open the Layer Manager palette:

- On the ribbon, click **Home** > **Layer** > **Layers Manager**.
- On the menu, click **Format** > **Layer**.
- Enter `LAYER` in the command window.

Merge Layer Command



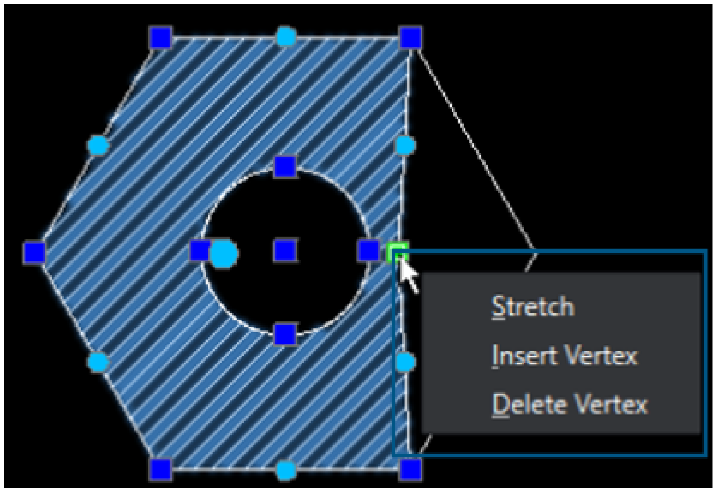
You can use the `MERGELAYER` command to reorganize layers.

This command is available from the Layer Manager palette that helps you merge the content of selected layers into other layers.

To access the Merge Layer command:

- On the ribbon, click **Home** > **Layers** > **Merge Layers**.
- On the menu, click **Format** > **Layer Tools** > **Merge Layers**.
- Enter `MERGELAYER` in the command window.

Reshaping Hatches



You can adjust the contour of hatches or gradient hatches.

When you select a hatch entity, the grips appear that help you adjust the shape. When you hover over a grip, the shortcut menu appears with editing options.

25

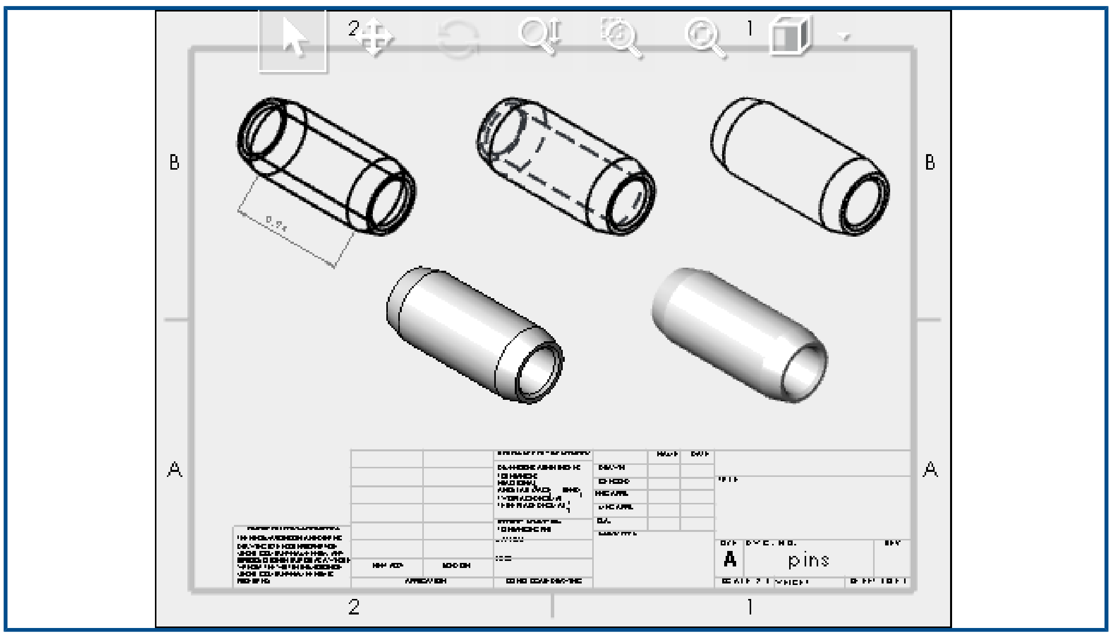
eDrawings

This chapter includes the following topics:

- **Display Styles in Drawings**
- **Supported File Types**
- **eDrawings Performance Improvements**

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

Display Styles in Drawings



If you saved a SOLIDWORKS drawing with specific display styles in drawing views, eDrawings supports each display style for any `.EDRW` file that you save in eDrawings 2024 and later.

In the Heads-up View toolbar, eDrawings shows all display states if the drawing views have shaded data: **Shaded with Edges**, **Shaded**, **Hidden Lines Removed**, **Hidden Lines Visible**, and **Wireframe**. The **Display Style** tool is only available for drawings with shaded data.

If you change the display style of a drawing view in eDrawings, only the selected view updates with the new display style. All other views remain the same. However, if you change the display style when you have not selected a drawing view, all views change to the selected display style.

If you rotate a drawing view, the display style is unaffected.

Supported File Types

eDrawings has updated the supported versions for several file types.

Format	Version
ACIS (`.sat`, `.sab`)	Up to 2021
Autodesk® Inventor® (`.ipt`, `.iam`)	Up to 2023
CATIA® V5 (`.CATPart`, `.CATProduct`)	Up to V5_V62023
Creo® - Pro/Engineer® (`.ASM`, `.NEU`, `.PRT`, `.XAS`, `.XPR`)	Pro/Engineer 19.0 to Creo 9.0
JT (`.jt`)	Up to v10.6
NX™ (Unigraphics®) (`.prt`)	NX1847 Series to NX2212
Parasolid™ (`.x_b`, `.x_t`, `.xmt`, `.xmt_txt`)	Up to 35.1
Solid Edge® (`.asm`, `.par`, `.pwd`, `.psm`)	V19 - 20, ST - ST10, 2023

eDrawings Performance Improvements

eDrawings performance is improved with various tools, rendering, printing, and file closure times.

Performance improvements include:

- **Measure** tool. Up to 20 times faster when opening the Measure pane, entity selection, and changing units.
- **Markup** tool. Up to 10 times faster when creating markups.
- **Reset** tool. Up to 1.5 times faster when resetting a model.
- Faster rendering and printing with software OpenGL.
- Faster times for closing files.

26

SOLIDWORKS Flow Simulation

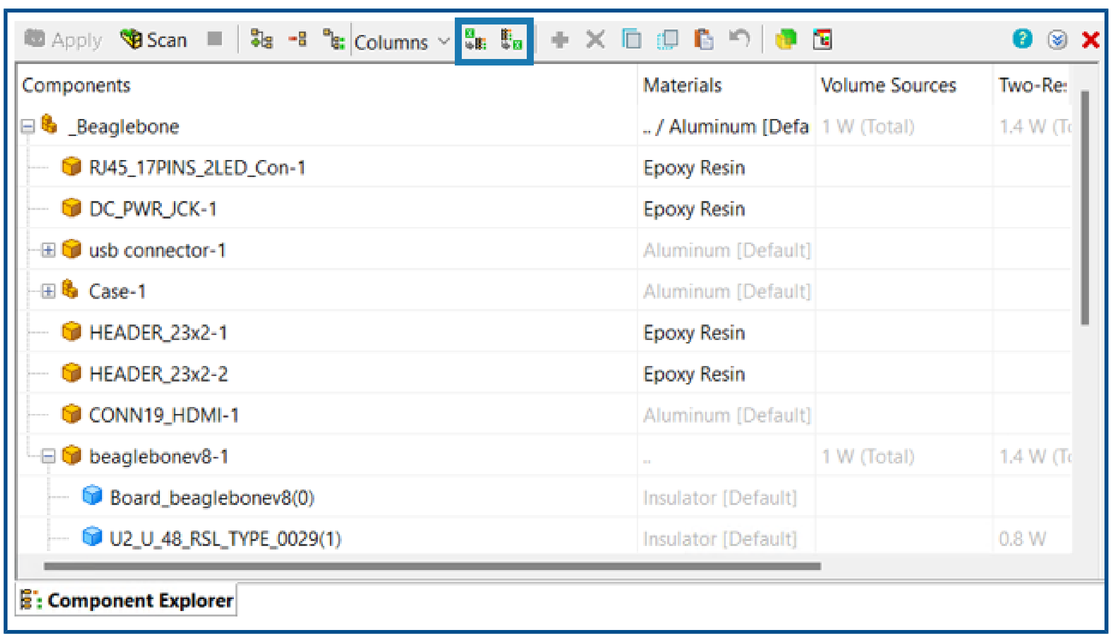
This chapter includes the following topics:

- **Importing and Exporting Component Lists**
- **Mesh Generation**
- **Mesh Boolean Operations**

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

For installation of SOLIDWORKS Flow Simulation, see **Load SOLIDWORKS Flow Simulation Modules**.

Importing and Exporting Component Lists



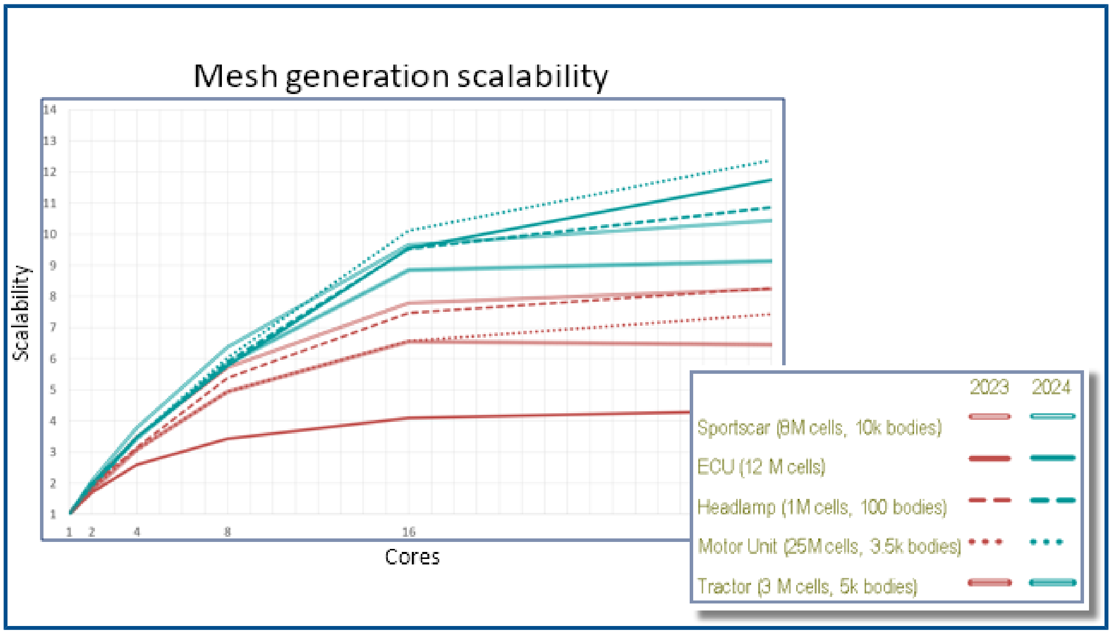
In the Component Explorer dialog box, you can export component lists to a Microsoft® Excel® spreadsheet, edit the properties, and import the component lists back.

By using a spreadsheet, you can manage component properties. You can edit:

- **Materials**
- **Volume Sources**
- **Two-Resistor Components** (library and power)

- **LEDs** (library and current)

Mesh Generation



With the Smart Cell Cartesian mesh generator, you can generate meshes faster and with smaller file sizes.

Speeds are 9-12 times faster on 32 cores for 10-20M cell models in Flow Simulation 2024 compared to 3-7 times faster in 2023. Meshing speed is about 2-3 times faster on 32 cores in 2024 because of scalability.

Mesh Boolean Operations

The Mesh Boolean Operation (MBO) handles complex and extremely bad geometry faster and easier. When SOLIDWORKS cannot conduct Boolean operations successfully because of bad geometry (such as bad topology with missing entities or self-intersecting faces), you can use MBO.

MBO meshes bodies separately and then conducts Boolean operations on the meshed bodies without using CAD Boolean operations.

This technology prepares and meshes even very bad models 5-15 times faster without prior user adjustments or automatic model healing. You can use MBO with the CAD Boolean diagnostic, combining the power of Mesh Boolean and the convenience of getting additional information, such as a diagnostic of the fluid domain.

If the CAD Boolean diagnostic fails to detect the fluid domain, you can still mesh the model with Mesh Boolean. In these cases, the Solver Monitor dialog box shows additional subdomain diagnostics. You can specify how to handle the geometry (CAD Boolean, Preprocessor Boolean (formerly called Improved Geometry Handling), or Mesh Boolean), and you can turn off the CAD Boolean diagnostics.

27

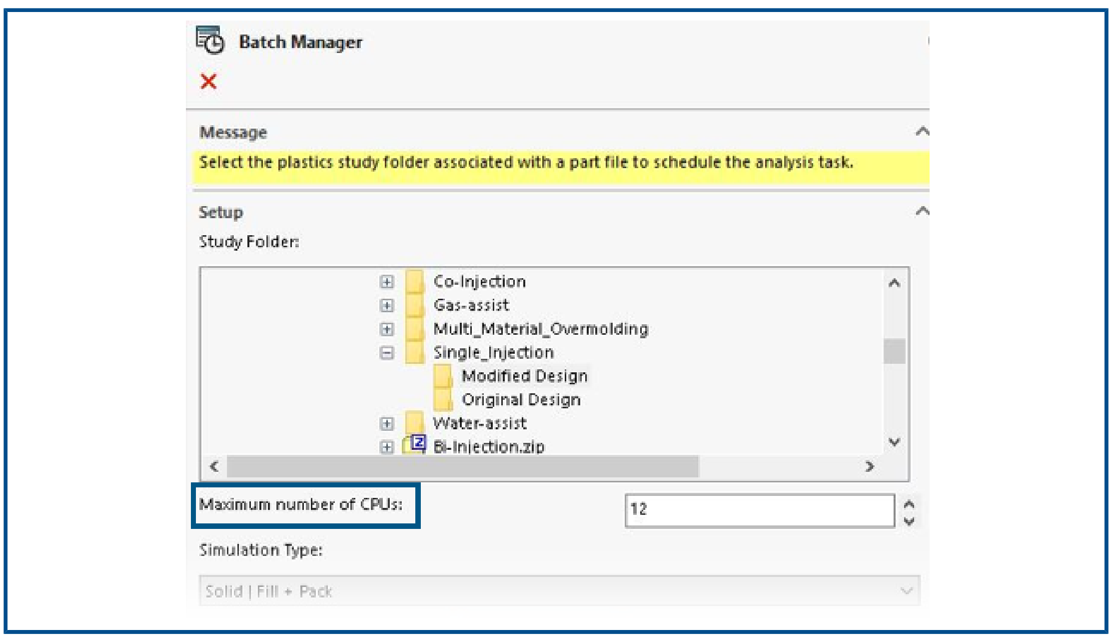
SOLIDWORKS Plastics

This chapter includes the following topics:

- **Batch Manager**
- **Compare Results**
- **Cool Solver**
- **Hot and Cold Runners**
- **Injection Location Advisor**
- **Materials with Pressure-Dependent Viscosity**
- **Material Database**
- **Mesh Enhancements**

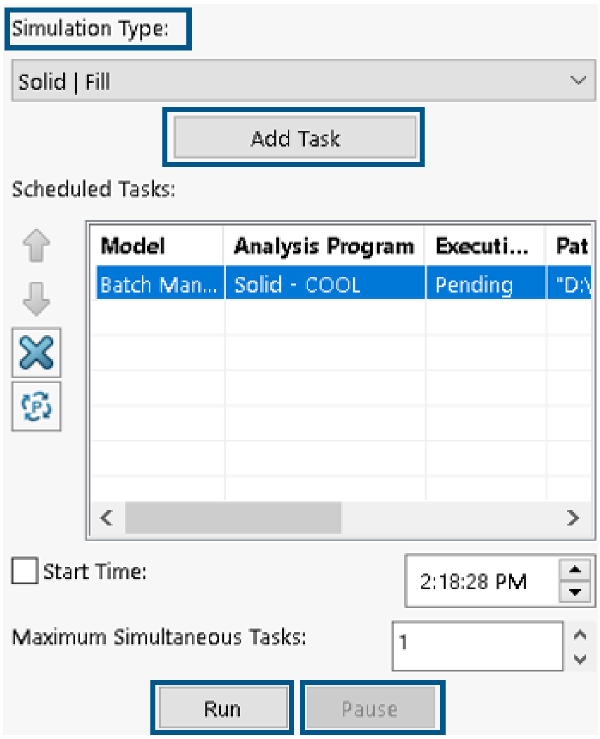
SOLIDWORKS® Plastics Standard, SOLIDWORKS Plastics Professional, and SOLIDWORKS Plastics Premium are separately purchased products that you can use with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium.

Batch Manager

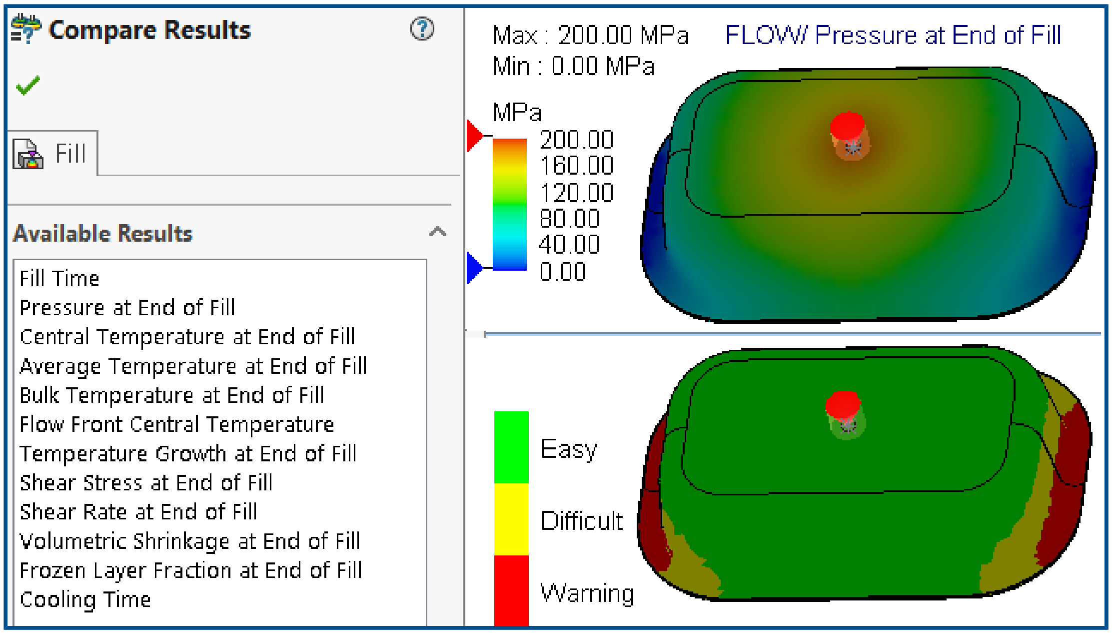


The Batch Manager PropertyManager is redesigned to improve usability.

- Rearrangement of the user interface elements in sections provides a streamlined workflow for the Batch Manager.
- Ability to specity the maximum number of CPUs for an analysis task.
- Improved visibility for the simulation type assigned to an analysis task and for controls to add, run, and pause an analysis task.



Compare Results



You can display four different results plots from one study using split view panes.

To display multiple result plots after running a study:

Do one of the following:

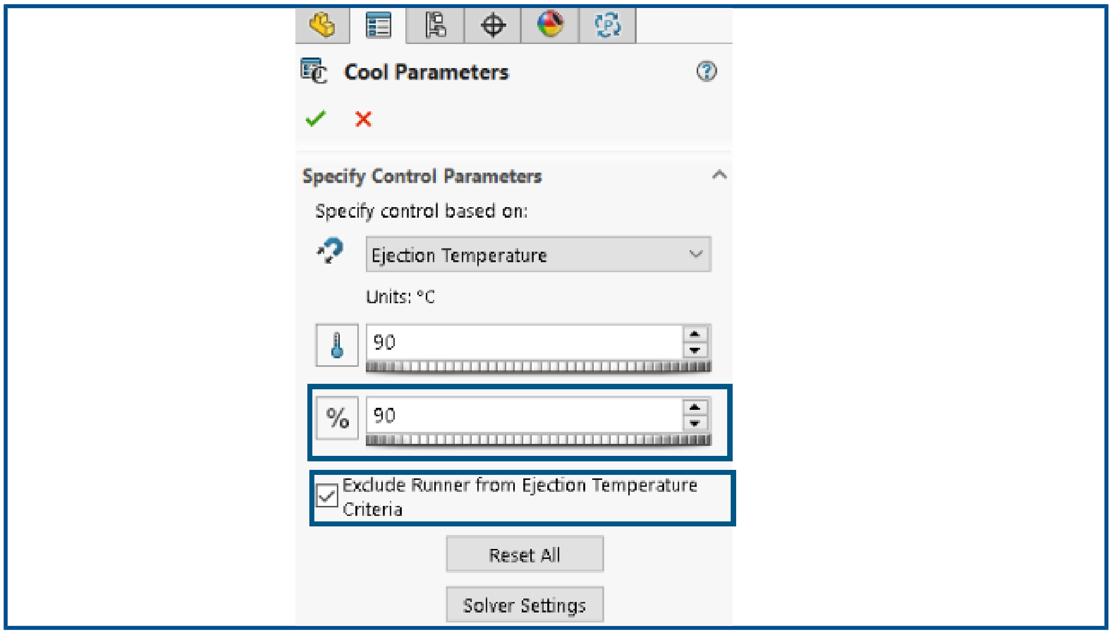
- Click **Compare Results** (Plastics CommandManager).
- In a study's PlasticsManager tree, right-click **Results** and click **Compare Results**.

In the Compare Results PropertyManager, you have these options:

Option	Description
Synchronize Views	Applies the same view orientation to all view panes.
Save Image	Saves the split view of the multiple result plots to a `.png` image format.

You can also specify the maximum and minimum values of the results shown on the view panes, view an isosurface mode, and use available tools to display animations.

Cool Solver



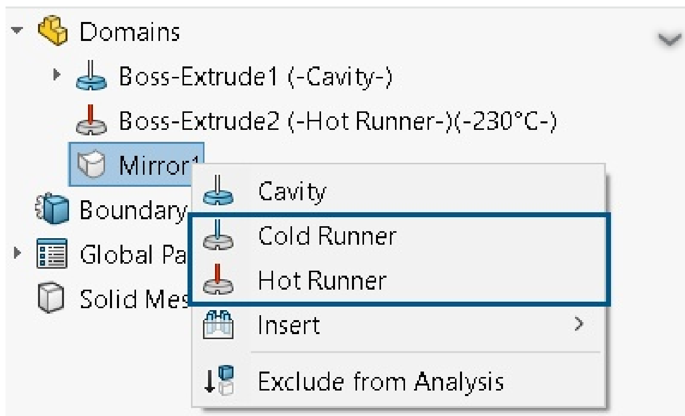
Solver options for ejection criteria enhance the performance of plastic injection simulations for thermoplastic materials.

You can either specify the cooling time or let the cool solver estimate a cooling time based on the following temperature ejection criteria for thermoplastic materials.

Option	Description
Volume % frozen at ejection	Specifies the percentage of the mold's volume that needs to cool down below the ejection temperature. The default is 90%.
Exclude Runner from Ejection Temperature Criteria	Excludes the cooling state of the sprue and runner segments from the ejection criteria. It is common to reduce the overall manufacturing time by ejecting the part

Option	Description
	before the sprue and runner segments have cooled completely.

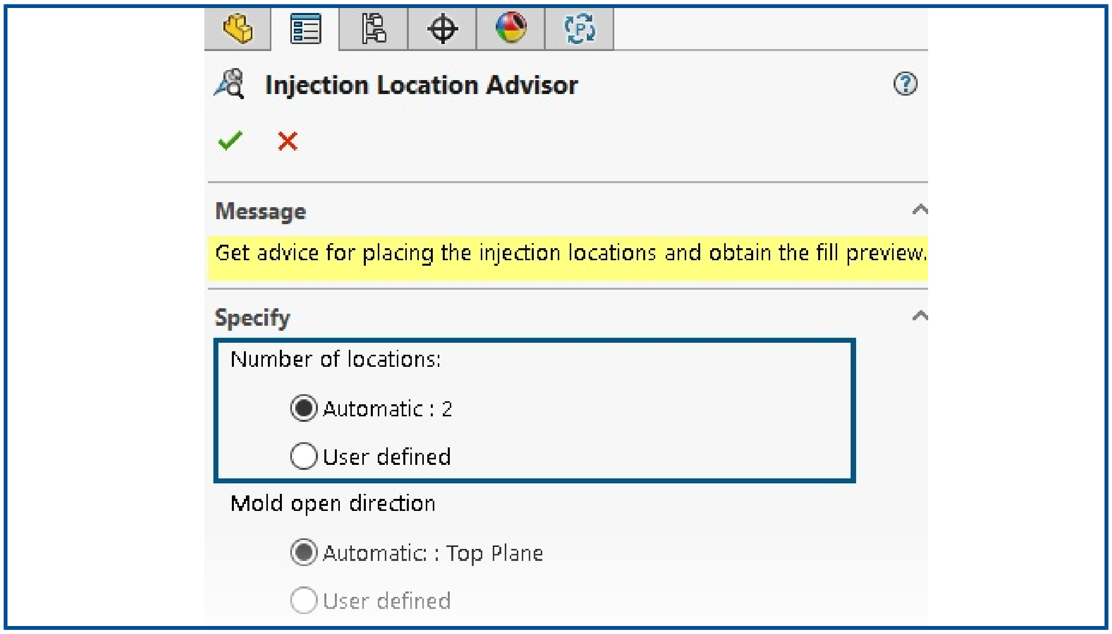
Hot and Cold Runners



You can more easily assign hot or cold runner domains to components of a plastic injection simulation.

To assign a runner domain type to a body listed under the **Domains** node, right-click the body and click **Hot Runner** or **Cold Runner**.

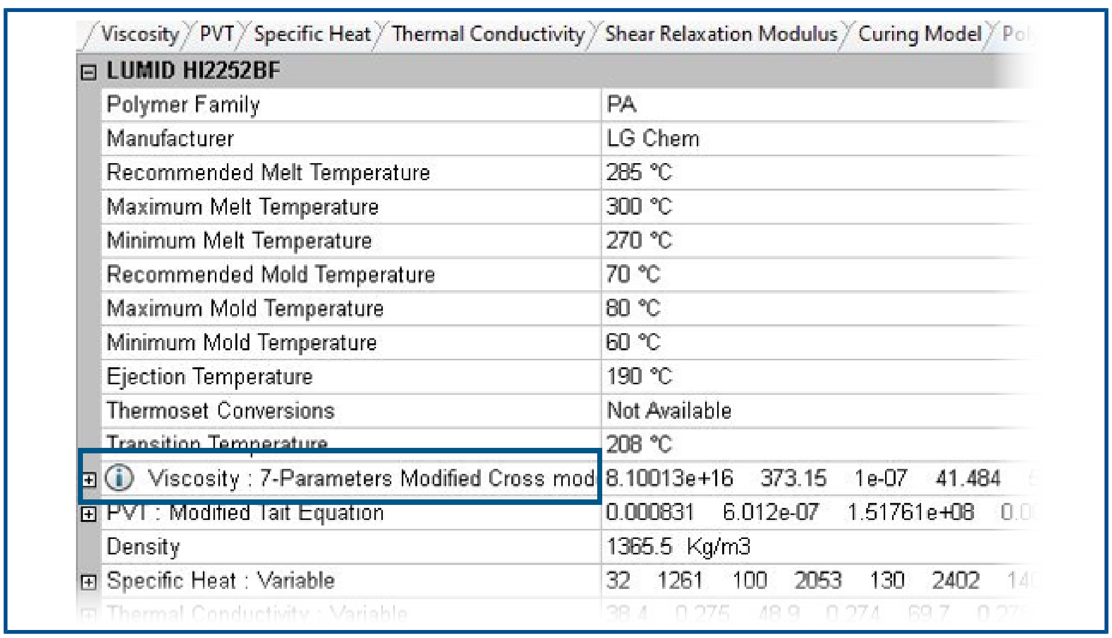
Injection Location Advisor



The Injection Location Advisor can iteratively determine an optimal number of injection locations (maximum of 10) to fill a cavity.

The default for **Number of locations** is **Automatic**, which activates the iterative approach for finding an optimal number of injection locations. To specify a custom number of injection locations, select **User defined**.

Materials with Pressure-Dependent Viscosity



Fill and Pack simulations support materials with pressure-dependent viscosity.

Materials that have pressure-dependent viscosity are listed in the Plastics Materials Database with an information icon ⓘ.

Accounting for pressure-dependent viscosity is important for parts that contain long flow lengths or very thin walls, or for cases where you need high injection pressures.

For more information, see *Material Properties (Polymer, Mold, and Coolant Domains)*.

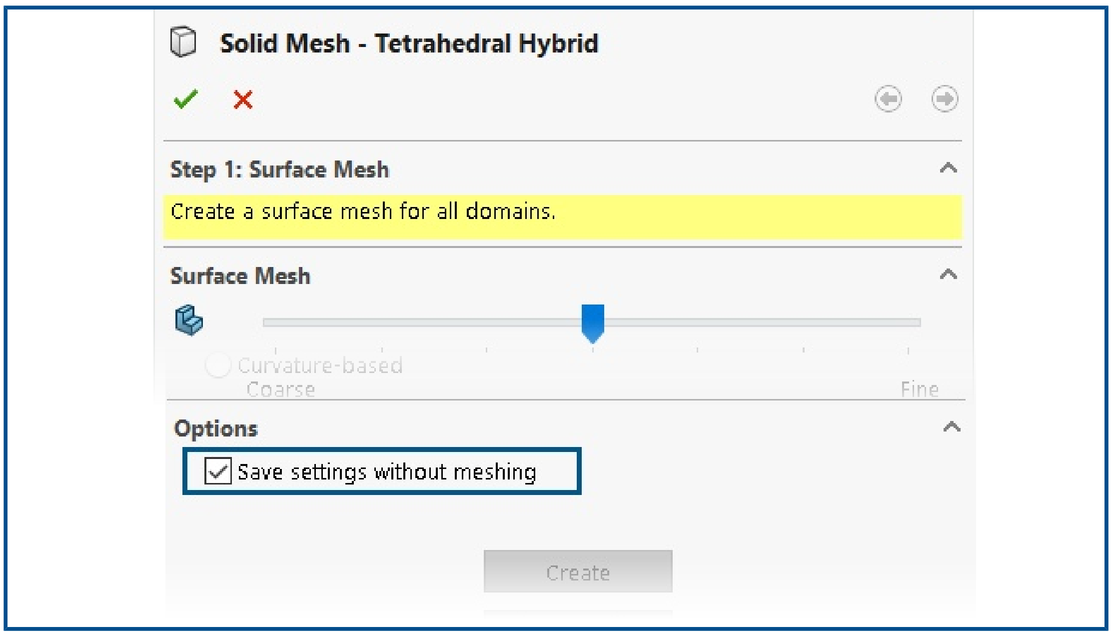
Material Database

The Plastics material database includes the latest data from the material manufacturers.

Materials	Description
New Materials	Added 417 new material grades from the following material suppliers: • CHIMEI: 42 • DuPont: 2 • EMS-GRIVORY: 4 • KRAIBURG TPE: 4 • LG Chem: 85 • MOCOM: 128 • ORLEN Unipetrol RPA: 20 • RadiciGroup High Performance Polymer: 2 • SABIC Specialties: 126 • Solvay Specialty Polymers: 1 • Trinseo: 3
Modified Materials	Updated 40 material grades with the latest material property values provided by the following material suppliers: • Borealis: 1 • CHIMEI: 2 • EMS-GRIVORY: 10 • ORLEN Unipetrol RPA: 20 • SABIC Specialties: 7

Materials	Description
Removed Materials	Removed 292 obsolete material grades from the following material suppliers: • 3M: 1 • ALBIS: 4 • Borealis: 1 • DuPont: 2 • DuPont Engineering Polymers: 2 • KRAIBURG TPE: 1 • LANXESS GmbH: 3 • LG Chemical: 56 • SABIC Specialties: 211 • Solvay Specialty Polymers: 11

Mesh Enhancements



You can save the mesh settings of a study without creating a mesh. You can also preview a surface mesh before creating a solid mesh.

The meshing options are available from the Solid Mesh - Tetrahedral, Solid Mesh - Hexahedral, and Shell Mesh PropertyManagers.

Option	Description
Save settings without meshing	You can save the mesh settings of a model (mesh size, refinement method, and advanced mesh control) without creating the mesh. When you run a study, the mesh settings

Option	Description
	are applied automatically to generate the mesh. In a study's PlasticsManager tree, the icon next to **Solid Mesh** or **Shell Mesh** indicates that you saved the mesh settings for the model.
Show preview	You can preview a surface mesh before creating a solid mesh to check the mesh validity for a model.

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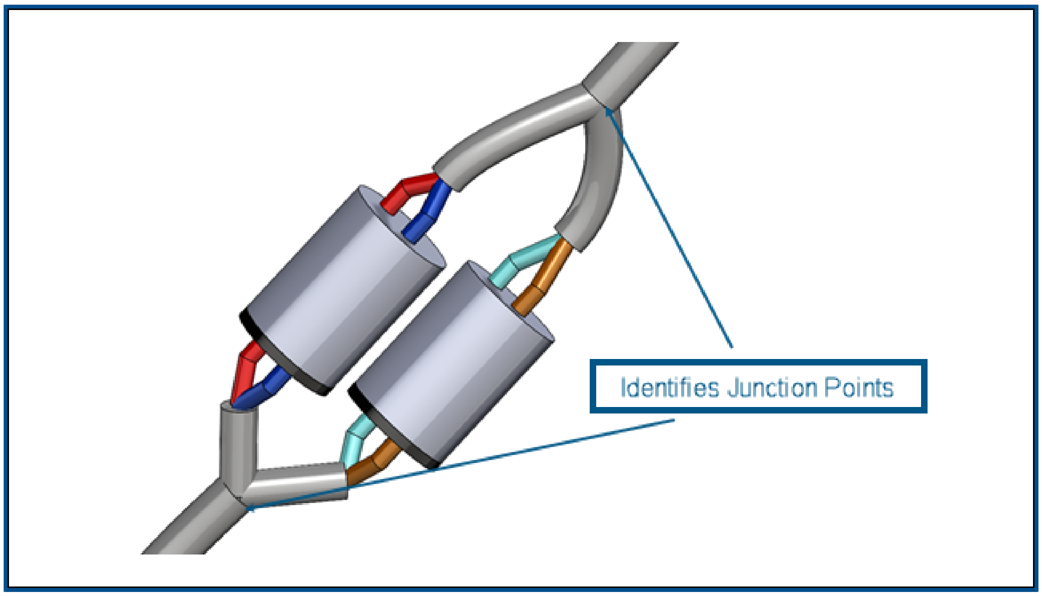
Routing

This chapter includes the following topics:

Routing is available in SOLIDWORKS® Premium.

Better Positioning of Complex Splices and Loop Segments in Flattened Routes (2024 SP3)

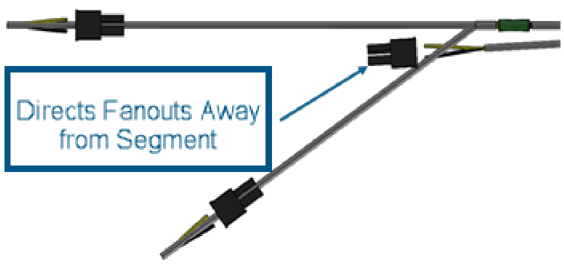


The **Flatten Route** tool offers improved support for complex and multi-circuit splices.

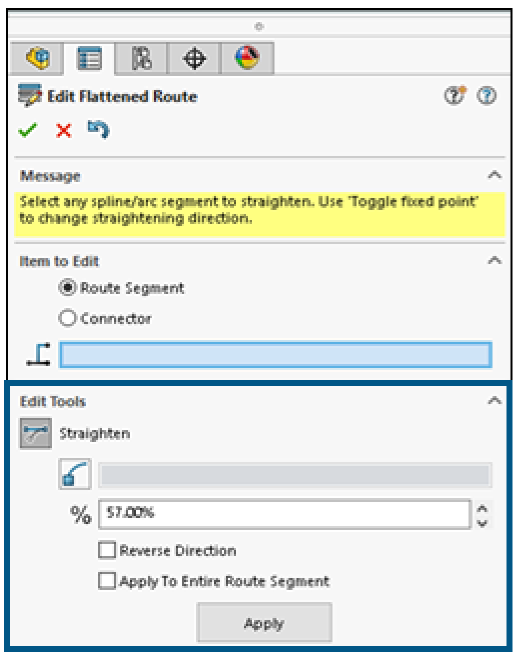
The **Flatten Route** tool automatically performs the following functions:

- Identifies the junction points in loop segments and moves them to the flattened plane.

- Directs fanouts away from the route segment rather than integrating them into the route segment.



Reverse Direction and Specify Percentage Options for Discrete Wires (2024 SP3)

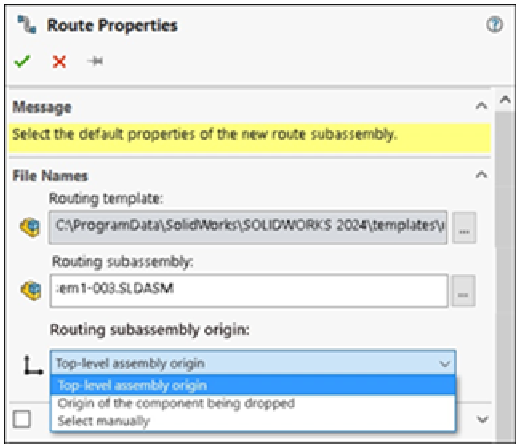


The Edit Flattened Route PropertyManager lets you reverse the direction of route segments when you straighten flattened discrete wires.

You can also specify a percentage to straighten segments instead of straightening an entire discrete wire segment.

To access these options, open a manufactured route assembly of discrete wires and click **Edit Flattened Route**. In the PropertyManager, click **Route Segment** and select a spline from the subassembly or flyout tree. Then click **Straigthen** , enter a value for **%**, and select **Reverse Direction**.

Aligning a Route Subassembly to the Origin (2024 SP3)



When creating a new route subassembly, you can align and position it according to your design requirements using the Route Properties PropertyManager.

Choices for defining the origin include:

- **Top-Level Assembly Origin**

  The origin of the routing subassembly aligns coincidentally with the origin of the top-level assembly.

- **Origin of the component being dropped**

  The origin of the routing subassembly aligns coincidentally with the origin of the fitting being added.

- **Select manually**

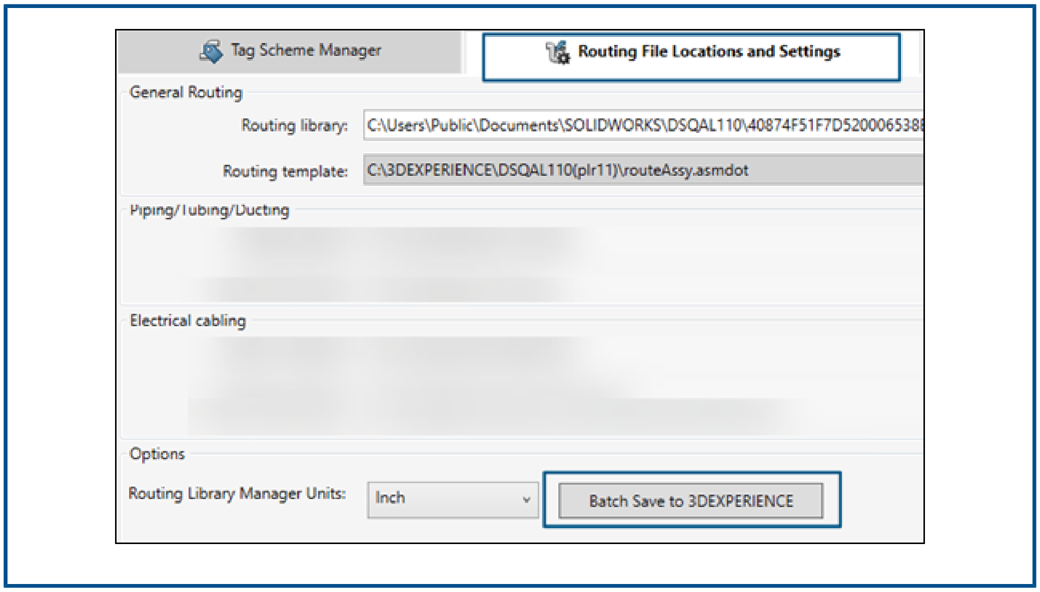
  The origin of the routing subassembly aligns coincidentally with a sketch point or vertex that you specify. You can also select the C-point or R-point of the fitting.

Quality Improvements to Flattened Route Updates (2024 SP3)

Continuing efforts to enhance quality and consistency while working with flattened routes in 3D, the Routing add-in has implemented the following updates:

- Changes made in the 3D route instantly reflect in the flattened route, reducing differences between them.
- The software accurately mirrors re-imported changes in the flattened route.
- Enhanced flexibility for edited and non-open end route segments allows them to adapt to changes in length without affecting the entire segment.
- Implemented the Split Route segment functionality for managing edits in a flattened configuration.

Using the 3DEXPERIENCE Add-In with Routing (2024 SP1)



The **3D**EXPERIENCE add-in enables you to store and manage your routing components and assemblies from a collaborative space on the **3D**EXPERIENCE platform. Additionally, you can access services, including free 3D routing components, through the **3D**EXPERIENCE Marketplace | PartSupply app.

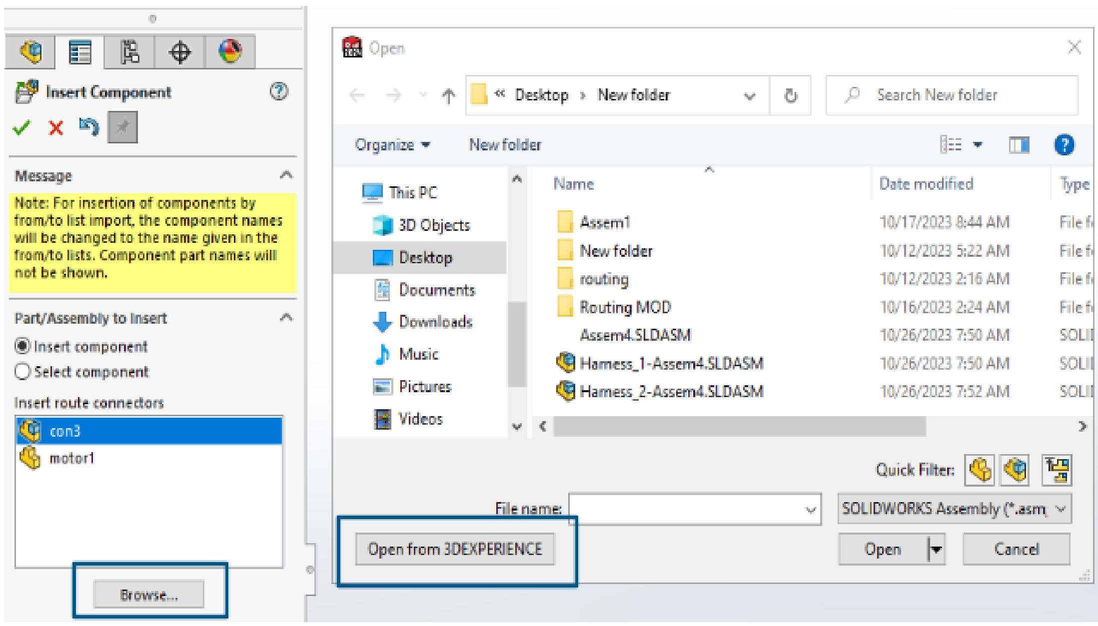
Within the Routing Library Manager, using the 3DEXPERIENCE add-in, you can perform the following tasks:

Tab	Task
Routing File Locations and Settings	• Batch upload the routing component library from a local computer to the **3D**EXPERIENCE platform. Click **Batch Save to 3DEXPERIENCE**. You can save only SOLIDWORKS files to the **3D**EXPERIENCE platform with batch uploading. • Batch download the routing component library from the **3D**EXPERIENCE platform. For **Routing template**, click **Browse** to locate a folder. In the dialog box, click **Select from 3DEXPERIENCE**.
Component Library Wizard	Create new or modify existing components in the library on the local computer or **3D**EXPERIENCE platform.

Tab	Task
Routing Component Wizard	Save the defined component on the local computer or the **3D**EXPERIENCE platform.
Piping and Tubing Database	Access all configurations of the components, **Uploaded** or **Not uploaded** to the **3D**EXPERIENCE platform, using **Component status**.

You can also open a routing assembly or component from the **3D**EXPERIENCE platform from the:

- Route Properties PropertyManager for pipes and elbows. For example, click **Browse** for **Custom elbow** in the Bend - Elbows dialog box.

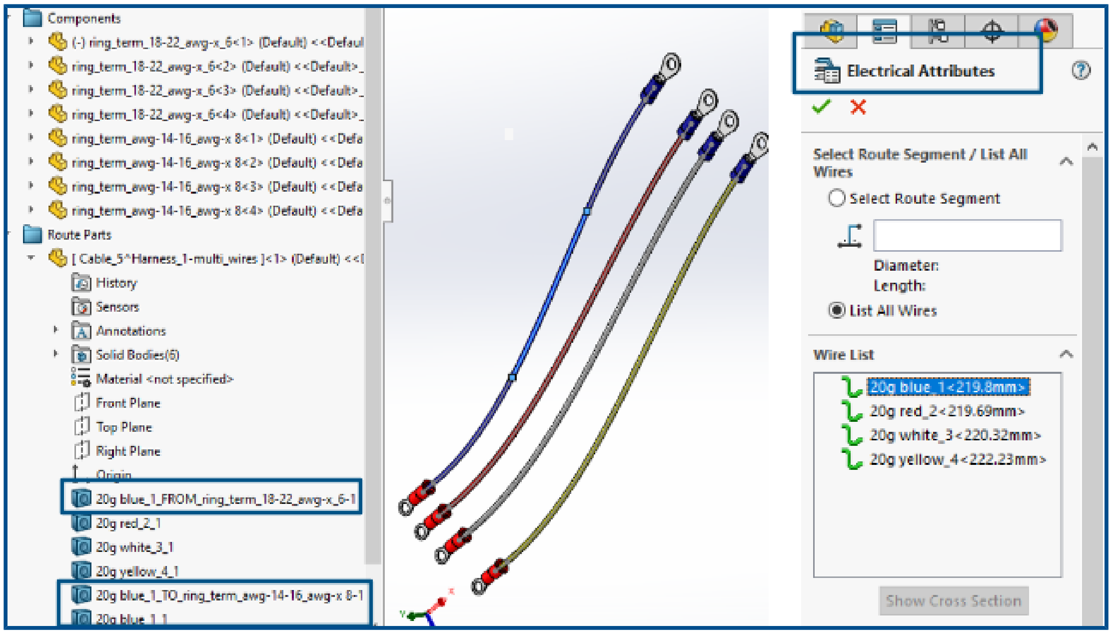


- **Start by From/To**, for example after clicking **Browse** for **Insert Component**.
- **Reuse Route** tools.
- **Add Splice** and **Edit Splice** options.

To learn more about the platform, see **Working with the 3DEXPERIENCE Platform and 3DEXPERIENCE Apps**.

To access free 3D components from the platform, see **Using 3DMarketplace | Part Supply**.

Naming Wires and Cables in the FeatureManager Design Tree



You can view the marks or names of 3D wires, cables, and their cores under **Route Parts** in the FeatureManager® design tree for a routing assembly. The Electrical Attributes PropertyManager automatically preassigns the marks or names.

This helps you correlate the 3D routes in the FeatureManager design tree with the marks or names of the wires, cables, and their cores displayed on the schematic drawing.

The naming convention uses the following to uniquely identify different routes:

- Wire, cable, and cable core marks from the Electrical Attributes PropertyManager.
- Sequential numbers as suffixes (*n*). Where *n* is proportional to the number of splits (with split route) and 1 (without split route).
- The directions (FROM/TO) that they connect to the components.

For example, the above image shows the naming for a routing assembly with four wires as follows:

- The three wires red, white, and yellow do not have **Split Route** applied and the naming convention is:

  Wire mark_1

  For example, 20g_red_2_1

- The blue wire has a **Split Route** applied at two points with three split bodies created and the naming convention is:

  - For the two extreme ends connected to the components:

    Wire Mark_FROM/TO_Component Mark

    For example,

    20g blue_1_FROM_Component1

    20g blue_1_TO_Component2

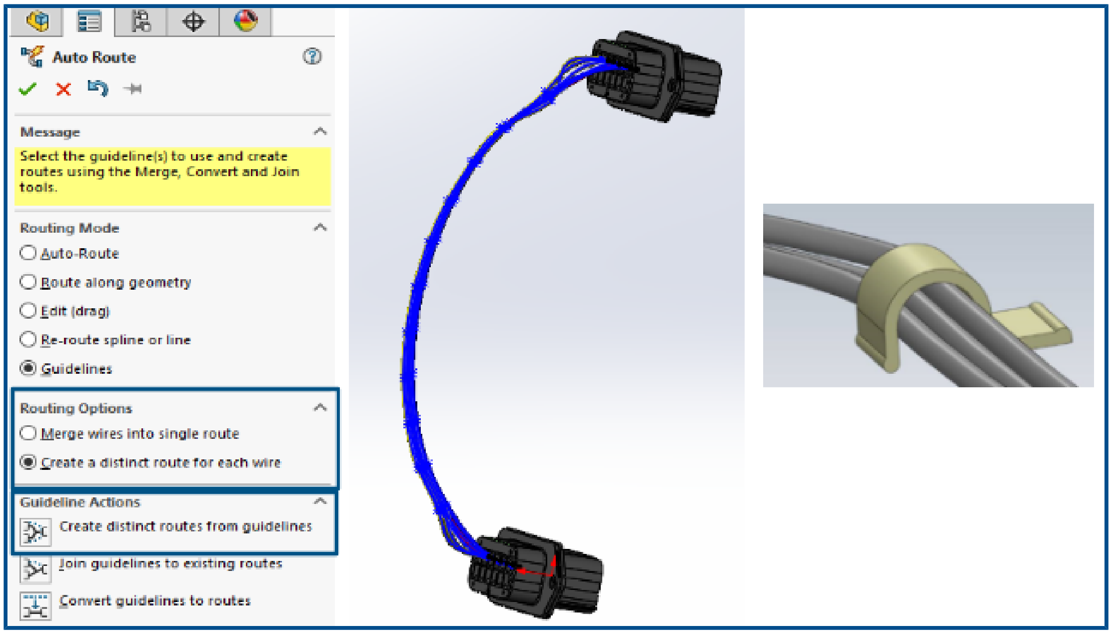
- In-between cable bodies not connected to the components:

  Wire Mark_*n*

  For example,

  20g blue_1_1

Discrete Wires with Auto Route



You can visualize each wire in a bundle distinctly in 3D and flatten them.

The Auto Route PropertyManager, **Routing Options** include:

- **Merge wires into single route**. Routes the selected wires along a single route.
- **Create a distinct route for each wire**. Routes the selected wires as distinct routes.

You can edit discrete wires by:

- Adding a route to the bundle with **Add Route to Discrete Bundle**.
- Removing a route from the bundle with **Remove Route from Discrete Bundle**.
- Moving the bundle by dragging a spline point on the discrete wire.
- Merging two bundles with **Merge Discrete Bundle**.
- Splitting a single route segment from the bundle.
- Creating a single junction point for multiple discrete bundles coming out from the connector or separate junction point for each discrete bundle.
- Routing the bundle through a clip by selecting one of its splines.

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SOLIDWORKS Toolbox

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

Additional Toolbox Hardware

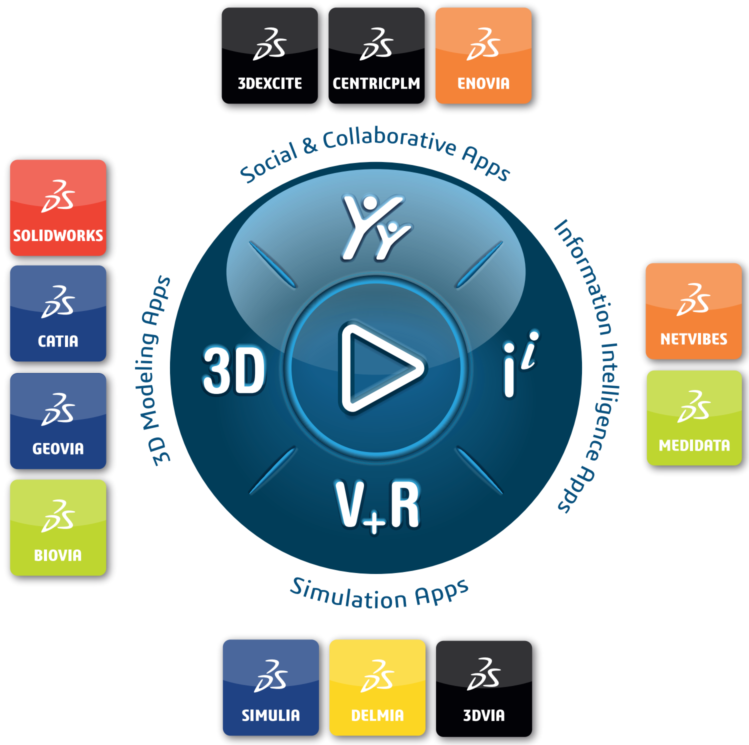
More hardware is available in the ANSI Inch and Metric Toolbox libraries.

Standard	Additional Folders	Additional Hardware

ANSI Inch	• The **Washers** folder includes:
	  • **Circular Washers**
	  • **Square Beveled Washers**
	• The **Nuts** folder includes subfolders for:
	  • **Hex Nuts - Prevailing Torque Nuts**
	  • **Wing Nuts**
	• The **Pins** folder includes subfolders for:
	  • **Clevis Pins**
	  • **Cotter Pins**
	  • **Grooved Pins**
	  • **Spring Pins**
	  • **Straight Pins**
	  • **Tapered Pins**
		• The **Bolts and Screws** > **Self Tapping Screws** folder includes a large hex head tapping screw.
		• The **Bolts and Screws** > **Machine Screws** folder includes a large hex screw.

ANSI Metric	**Pins**. Includes coiled spring pins.	

In the ANSI Inch standard, the `hex head tapping screw_ai.SLDPRT` in **Bolts and Screws** > **Self Tapping Screws** > **Hex Head Tapping Screw** has been updated. If you copy the updated file, you will lose any customization to the existing file.



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