



# WHAT'S NEW

## SOLIDWORKS 2025



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# 1

## Welcome to SOLIDWORKS 2025

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This chapter includes the following topics:

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



SOLIDWORKS® 2025 contains user-driven enhancements that help streamline and accelerate your product development processes from concept to manufacturing:

- Accelerate time to market with enhanced collaboration and data management
- Streamline workflows for parts, assemblies, drawings, MBD, electrical and pipe routing, ECAD-MCAD collaboration, and rendering
- Work faster with import/export, user experience, and performance improvements
- Streamline drafting workflows with accuracy and clarity with DraftSight® updates
- Increase data efficiency with SOLIDWORKS PDM updates
- Ensure performance and accuracy with SOLIDWORKS Simulation updates
- Streamline electrical design with SOLIDWORKS Electric Schematic and Electrical Schematic Designer updates
- Continue to design anywhere with the latest in browser-based product development on the 3DEXPERIENCE® platform



This document covers all enhancements that affect how you interact with the **3DEXPERIENCE** 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® 2025 provide improvements to existing products and innovative new functionality.

- |                    |  |
|--------------------|--|
| Fundamentals       | • <a href="#">Specifying a Z-Up Template</a> on page 89  |
| Parts and Features | • <a href="#">Defeature Silhouette Method for Parts</a> on page 115<br>• <a href="#">Patterning Reference Geometry</a> on page 116<br>• <a href="#">Repairing Dangling Relations</a> on page 109 |
| Assemblies         | • <a href="#">Assembly Visualization</a> on page 151<br>• <a href="#">SpeedPak Instances</a> on page 154<br>• <a href="#">Interference Detection in Large Design Review Mode</a> on page 155     |
| SOLIDWORKS MBD     | • <a href="#">Creating DimXpert Dimensions from Sketch Dimensions</a> on page 267  |

## Performance

SOLIDWORKS® 2025 improves the performance of specific tools and workflows.

Some of the highlights for performance and workflow improvements are:

### Features

The quality and performance of pattern features is improved, especially for editing and rebuilding. Examples:

- If the seed feature of a pattern is another pattern, the seed feature is not highlighted.
- If the seed feature has more than 100 faces, the seed feature is not highlighted.
- For newly created patterns that use the **Instances to Vary** option, performance and accuracy are improved.
- The performance is improved when you edit or click **OK** to create patterns that have a large number of instances or faces.

### Assemblies

Performance is improved when calculating mass properties for an assembly.

## SOLIDWORKS PDM

SOLIDWORKS PDM performance is improved during the file check in to the SOLIDWORKS PDM database when the data transfer over the internet is slow. The file check in operation is two times faster than before.

## Sheet Metal

You can experience improved performance while working with multibody parts with a large number of cosmetic thread features when you enable the **Shaded cosmetic threads** option.

For sheet metal parts with multiple cosmetic thread features, performance is improved for these operations:

- Opening parts
- Creating new features
- Editing features
- Updating and rebuilding parts

Performance is improved while working with drawings that contain drawing views of sheet metal parts with many holes and forming tools. When working with such drawings, you can experience improved performance for:

- Opening drawing files
- Making new drawings from the sheet metal part
- Updating drawing views after making edits to the sheet metal part

## Sketching

Performance has been enhanced when zooming, panning, and rotating complex sketches, particularly when dealing with large sketches imported from DWG file conversions or those containing thousands of splines.

## Parts

Performance is improved for parts with highlighted edges when you select features or bodies.

## 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 3DEXPERIENCE® 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

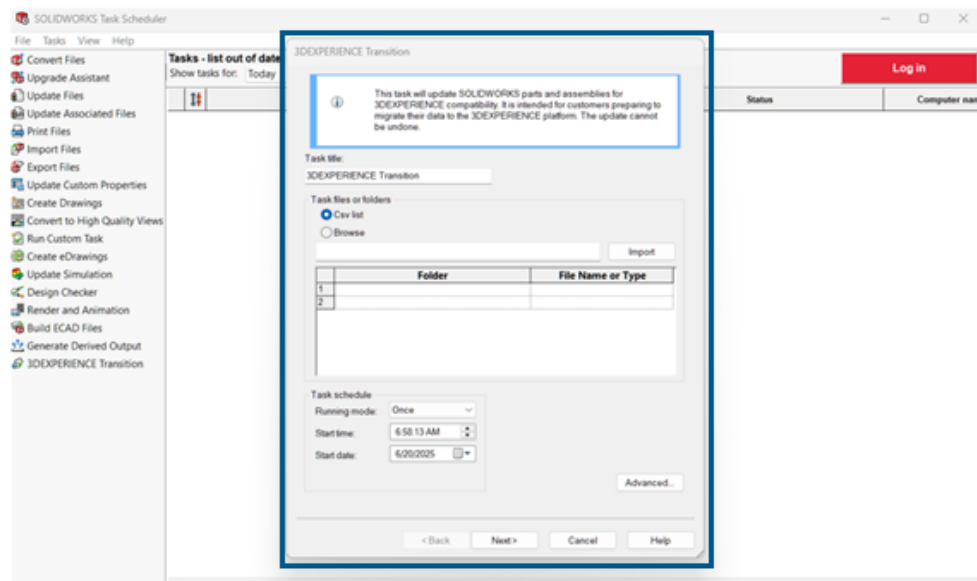
This chapter includes the following topics:

- **SP4 and FD04**
- **SP3 and FD03**
- **SP2 and FD02**
- **SP1 and FD01**
- **SP0 and 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).

### SP4 and FD04

#### 3DEXPERIENCE Transition Task in SOLIDWORKS Task Scheduler



The **3DEXPERIENCE** Transition task lets you update SOLIDWORKS files for compatibility with the **3DEXPERIENCE** platform. The **3DEXPERIENCE** Transition task works the same as the **3DEXPERIENCE** Compatibility task, but it can use a `.csv` file to select content from your computer and run macros.

**Benefits:** You can save time using `.csv` files to add content to the task.

With the **3DEXPERIENCE** Transition task, you can:

- Upgrade files without enabling **3DEXPERIENCE** compatibility by saving them in a current version.
- Upgrade custom properties.
- Add rebuild marks.
- Add display data marks.

Creating a 3DEXPERIENCE Transition Task

**To create a 3DEXPERIENCE Transition task:**

1. In SOLIDWORKS Task Scheduler, click **3DEXPERIENCE Transition**.
2. Under **Task title**, create a name for your task.
3. Under **Task files or folders**, select the content you want to update by doing one of the following:
  - Browse for a file or folder to add to **Task Files or Folders**.
  - Import a `.csv` file that specifies the content to add to **Task Files or Folders**.

The format of the `.csv` file is *path, filename*. For example to add `clamp.sldprt` and `bracket.sldprt`, write:

- "C:\Users\Public\Documents\SOLIDWORKS\SOLIDWORKS 2025\samples\tutorial\assemblymates","clamp.sldprt"
- "C:\Users\Public\Documents\SOLIDWORKS\SOLIDWORKS 2025\samples\tutorial\assemblymates","bracket.sldprt"

4. Run the task immediately or schedule the task (see [Scheduling the Task](#) on page 81).
5. Click **Next**.
6. In the Options dialog box, specify options:

Option	Description
<b>Configuration option</b>	<p>Saves only the active configuration or activates all configurations before saving.</p> <div> <p>Activating all configurations before saving can add significant time to the task.</p> </div>
<b>3DEXPERIENCE Compatibility</b>	<p>Updates SOLIDWORKS content for compatibility with the <b>3DEXPERIENCE</b> platform. See <a href="#">3DEXPERIENCE</a></p>

Option	Description
	<b>Compatibility and 3DEXPERIENCE Integration Options.</b>
<b>File Upgrade Settings</b>	<ul style="list-style-type: none"> <li>Upgrades custom properties.</li> <li>Adds rebuild mark to all configurations.</li> <li>Adds display data mark to all configurations.</li> </ul> <div> <b>Add display data mark to all configurations</b> is unavailable if you selected <b>3DEXPERIENCE Compatibility</b>. </div>
<b>Backup Files</b>	Specifies the location to back up the updated files.

- To run a macro, see [Running a Macro with the 3DEXPERIENCE Transition Task](#) on page 81.
- Click **Finish**.

Scheduling the Task

**To schedule the task:**

- Under **Task Schedule**, set:

Option	Description
<b>Running mode</b>	How often the task runs. Select <b>Once</b> , <b>Daily</b> , <b>Weekly</b> , or <b>Monthly</b> .
<b>Start time</b>	What time the task starts.
<b>Start date</b>	What date the task starts.

- Click **Options** to specify backup locations.
- Click **Advanced** to change the working folder, time-out values, and other options.
- Click **Finish**.

The task and its title, scheduled time, scheduled date, and status appear in the Tasks panel. The status of the task is **Scheduled**.

Running a Macro with the 3DEXPERIENCE Transition Task

**To run a macro with the 3DEXPERIENCE Transition task:**

- In the **3DEXPERIENCE Transition** task, select the files you want to run the macro on. See [Creating a 3DEXPERIENCE Transition Task](#) on page 80.

- a. Click **Next**.
2. In the Options dialog box, under **Custom Actions**, select **Run macro:**.
3. Browse for a SOLIDWORKS macro (.swp).
4. Click **Finish**.

The macro appears in the Task Scheduler with the title you set for the task.

## Sample SOLIDWORKS Macro

To test this functionality, you can paste the following text into a SOLIDWORKS macro (.swp).

This sample macro adds a property named "Hello" with a value of "Hello World" to any part, assembly, or drawing in the list of task files.

- For parts and assemblies, it adds a configuration-specific property to the active configurations.
- For drawings, it adds a custom property, because drawings do not contain configurations.

```
Dim swApp As SldWorks.SldWorks
Dim swModel As SldWorks.ModelDoc2
Dim config As SldWorks.Configuration
Dim cusPropMgr As SldWorks.CustomPropertyManager
Dim lRetVal As Long
Dim boolstatus As Boolean
Dim longstatus As Long, longwarnings As Long

Sub main()

    Set swApp = Application.SldWorks
    Set swModel = swApp.ActiveDoc

    If swModel Is Nothing Then
        ' If no model is currently loaded, then exit
        Exit Sub
    End If
    If (swModel.GetType <> swDocDRAWING) Then

        ' Add a Configuration Property named "Hello" to the active
        configuration for a Part or Assembly

        Set config = swModel.GetActiveConfiguration
        Set cusPropMgr = config.CustomPropertyManager

        lRetVal = cusPropMgr.Add3("Hello",
swCustomInfoType_e.swCustomInfoText, "Hello World",
swCustomPropertyAddOption_e.swCustomPropertyDeleteAndAdd)

    Else

        ' Add a Property named "Hello" for a Drawing

        Set cusPropMgr = swModel.Extension.CustomPropertyManager("")
        lRetVal = cusPropMgr.Add3("Hello",
swCustomInfoType_e.swCustomInfoText, "Hello World",
```

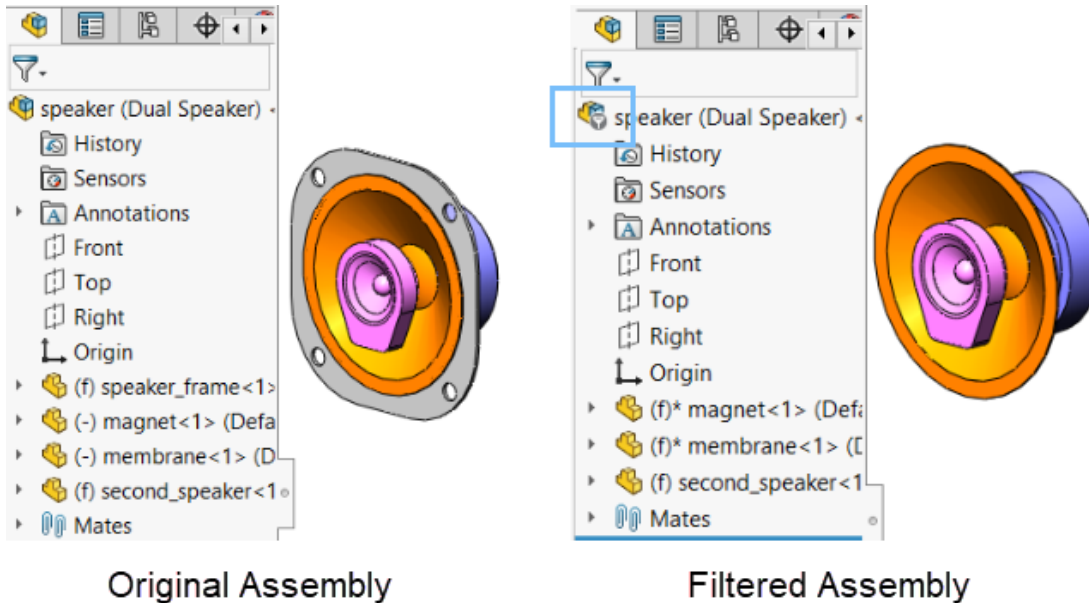


```
swCustomPropertyAddOption_e.swCustomPropertyDeleteAndAdd)
```

```
End If
```

```
End Sub
```

## Opening Filtered Assemblies (2025 FD04)



**3DEXPERIENCE** users can open filtered assemblies from the platform, edit them, and save them locally or back to the platform.

**Benefits:** You can open and work on a specific subset of components from a large assembly without opening the entire assembly.


Notes:


- Filtered assemblies do not support lightweight or Large Design Review mode. The filtered assemblies that you open must always be resolved.
- If you open the full assembly model, you cannot open the filtered assembly at the same time.

### To work with partial assemblies:

1. In your **3DEXPERIENCE** app, open the 3DEXPERIENCE tab in the task pane.
2. In the Compass, under **My Apps**, start **Product Structure Explorer** and do as follows:
  - a. Under **Access your work**, click **Open Content** and browse to an assembly that has filters applied to it.  
You or others can apply filters to an assembly.
  - b. To open the filtered assembly, drag it into the graphics area or right-click it and click **Open**.

The selected filtered assembly opens in SOLIDWORKS. In the FeatureManager design

tree, at the top node, this icon  indicates a filtered assembly. Filtered-out components do not appear in the FeatureManager design tree but the software retains these components in the product structure for subsequent filtering. A notification alerts you that this is a filtered assembly.

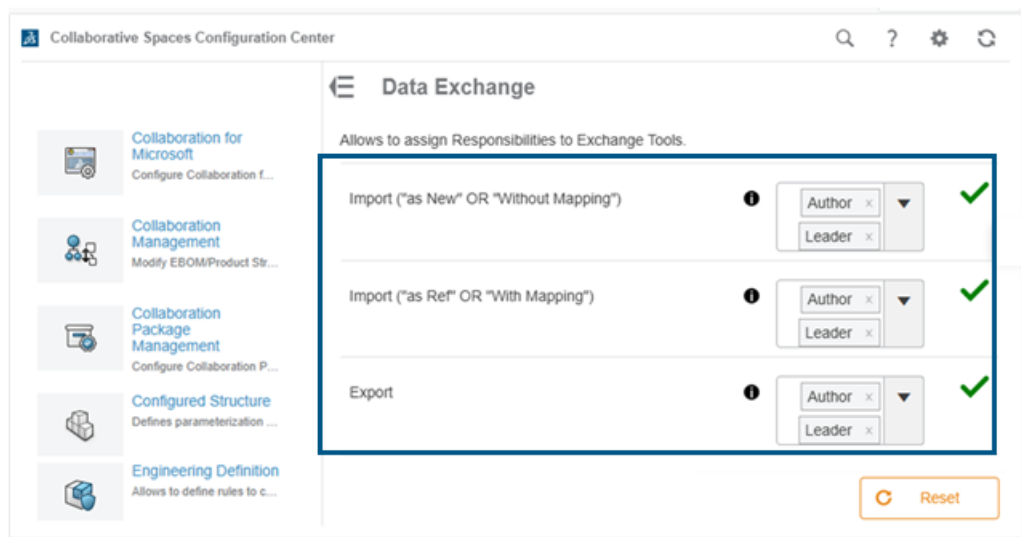
 **Filtered Assembly** Some components are not loaded because the assembly was opened from a 3DEXPERIENCE filter.

3. Modify the filtered assembly as required. Most commands are permitted except for the following blocked items:
  - New assembly features, such as cuts or fillets
  - Design tables
  - You cannot modify the top-level configuration of the assembly.
  - You cannot add a CAD Family to the ConfigurationManager.
4. Save the filtered assembly locally or back to the platform.

For more information about filtering, see [Filtering Data](#).

For more information about Product Structure Explorer, see [Product Structure Explorer](#).

## Specifying User Access Controls for Exporting and Importing CAD Packages (2025 FD04)



The 3DEXPERIENCE platform administrator can specify the user access controls for the **Export as Package** and **Import Package** commands in MySession. These commands are available when users enable the 3DEXPERIENCE Exchange add-in in SOLIDWORKS.

**Benefits:** Access controls define what users can and cannot do, and protect data integrity.

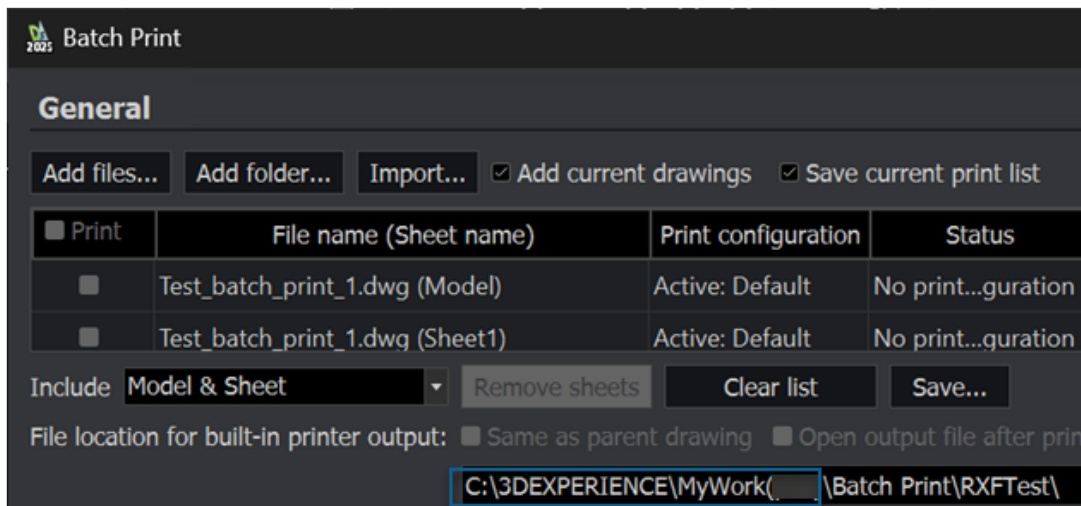
You must have administrator privileges to specify user access controls.

**To specify user access:**

1. In the **3DEXPERIENCE** platform, go to the **Collaborative Spaces Configuration Center > Data Exchange**.
2. For the Export and Import commands, select permissions for users, such as **Author**, **Leader**, **Reader**, or **Contributor**.

## SP3 and FD03

### Batch Print for 3DEXPERIENCE Drawings (DraftSight Connected Only) (2025 FD03)



You can add files from the **3DEXPERIENCE** platform and files from bookmarks to a batch print list. You can also save the batch print output of PDF files to the **3DEXPERIENCE** platform.

#### To add files from the 3DEXPERIENCE platform to a batch print list:

1. Type `BATCHPRINT` in the command window.
2. In the Batch Print dialog box, click **Add files**.
3. In the Specify File Names dialog box, click **Open from 3DEXPERIENCE**.
4. In the Open dialog box, select files and click **Open**.

#### To add files from bookmarks to a batch print list:

1. Type `BATCHPRINT` in the command window.
2. In the Batch Print dialog box, click **Add folder**.
3. In the Specify Folder dialog box, click **Select from 3DEXPERIENCE**.
4. In Select a Bookmark dialog box, select bookmarks and click **Select**.

#### To save the batch print output of PDF files to the 3DEXPERIENCE platform:

You can save the batch print output of PDF files only.

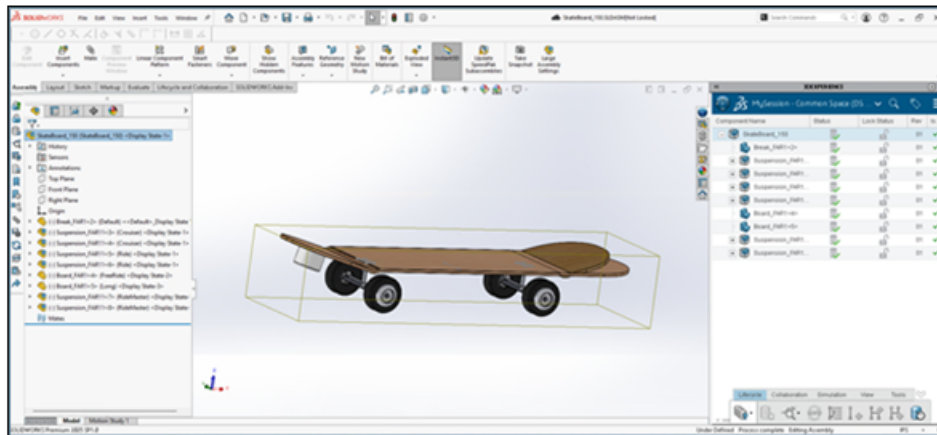
1. Type `BATCHPRINT` in the command window.

2. In the Batch Print dialog box, for **File location for built-in printer output**, click **Browse**.
3. In the Select a Bookmark dialog box, select a bookmark and click **Select**.

**Add current drawings** lets you add all the current drawings that you have opened from the **3DEXPERIENCE** platform to the batch print list.

For details, see [Processing Print Output in a Batch](#).

## Partial and Open Save for Filtered Assemblies (2025 FD03)



You can open and save only part of a SOLIDWORKS assembly using filters in the Design with SOLIDWORKS app. You do not need to load the entire model. You only need to load the parts you want to work with.

**Benefits:** Filters help reduce load time and memory usage by avoiding the need to open the entire assembly.

For example, if you are working on a skateboard model, you can create a filter that includes only the deck and wheels. Or you can create another filter that includes the trucks and bearings. You can choose what to load.

There are two types of filters:

- **Transient** filters are temporary. You define them, use them, and they disappear after the session. You do not save them to the platform.
- **Persisted** filters are saved on the platform. You can reuse them, search for them, and open them like any other item.

You can apply filters only to SOLIDWORKS model saved on the platform. Filters let you work faster by focusing only on the parts you need.

On the **3DEXPERIENCE** platform, you define filters directly on the product structure of the model. These filters control which components load in SOLIDWORKS. When you apply a filter, the software opens only the selected parts.

You can create filters in the Product Structure Editor or MySession apps, and then use them with the Design with SOLIDWORKS app. You can define a filter based on selection, attributes, geometry, or product configuration.

Filters support both one-time use (transient) or reusable workflows (persisted). Persisted filters behave like saved objects and are available across sessions.

When you open a filtered assembly:

- A filter icon appears in both the SOLIDWORKS FeatureManager design tree and the MySession pane.
- An informational banner appears in SOLIDWORKS to indicate the model is filtered.

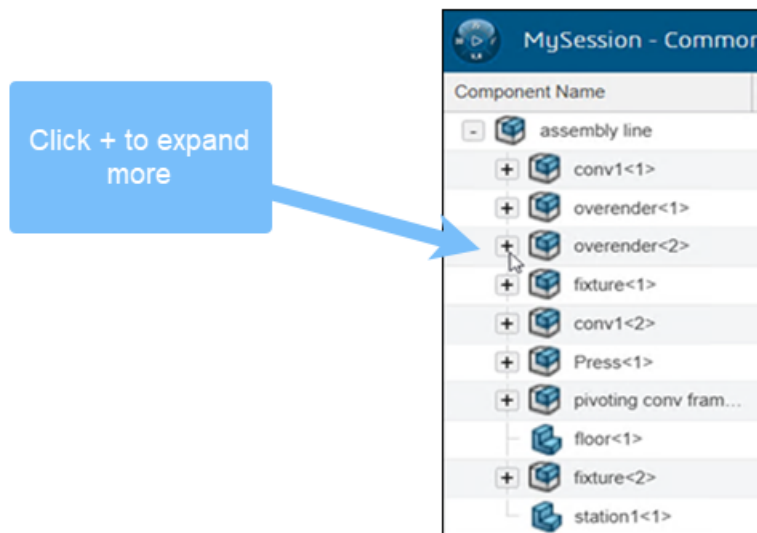
You can also filter assemblies based on PLM configurations, such as variants and options. This capability extends filtering beyond selection or geometry. It allows you to open specific configurations directly in SOLIDWORKS, based on how you define the product.

You can also open drawings that reference filtered assemblies, but their behavior depends on the filter type:

- Drawings based on persisted filters. You can open and save these drawings. They stay connected to the filtered definition of the assembly.
- Drawings based on transient filters. You can open them, but you cannot save them to the platform. This restriction prevents saving drawings based on temporary, unsaved filter states.

When you open a drawing referencing a filtered assembly, keep in mind that components excluded by the filter (such as hidden mates or parts) may be removed from the drawing view. If you save the drawing in that state, those changes will be saved to the platform and may affect how the drawing looks the next time it is opened.

## Dynamic Tree Expansion in MySession (2025 FD03)



MySession loads only the first level of children for assemblies by default, making it faster and easier to navigate, especially in large assemblies. The tree expands dynamically as you work, showing more structure only when you need it.

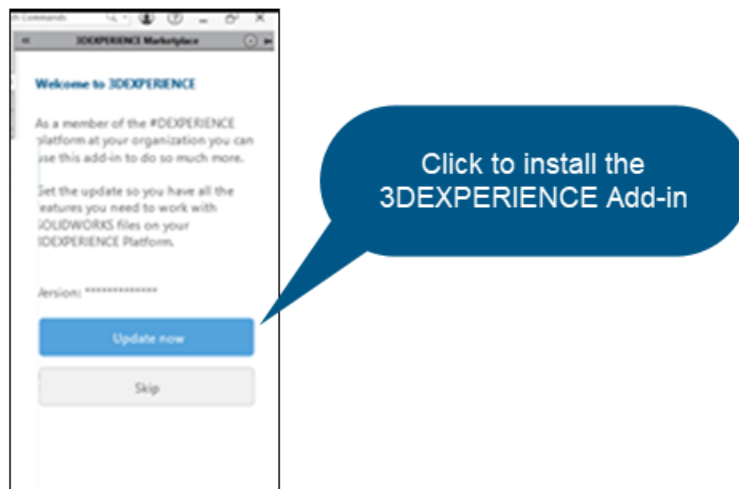
**Benefits:** Dynamic loading makes managing large, complex assemblies easier without overloading the system.

Prerequisite: To enable Dynamic Tree Expansion, go to the MySession action bar and navigate to **Tools > Options > Refresh MySession after opening files**, and make sure to clear the option.

To view more of the file structure, you can:

- Right-click a node and select **Expand All** or **Collapse All**.
- Click the **+** icon next to a node.
- Use the **View** options in the action bar:
  - **Expand All**
  - **Expand in Levels**
  - **Collapse All**

## Installing the 3DEXPERIENCE Add-In from 3DEXPERIENCE Marketplace (2025 FD03)



If you have the Collaborative Designer for SOLIDWORKS role, you can install the **3DEXPERIENCE** add-in directly from **3DEXPERIENCE** Marketplace in the SOLIDWORKS Task Pane. This method no longer requires a separate download.

**Benefits:** You can save time by installing the add-in directly within SOLIDWORKS.

To access this functionality, load the **3DEXPERIENCE** Marketplace from the SOLIDWORKS Add-In dialog box under the **Tools** menu.

To install the **3DEXPERIENCE** add-in from the **3DEXPERIENCE** Marketplace:

1. In the SOLIDWORKS Task Pane, click the SOLIDWORKS Resources tab and select the **3DEXPERIENCE Marketplace**.

The 3DEXPERIENCE Marketplace tab opens and displays the Welcome to 3DEXPERIENCE page.

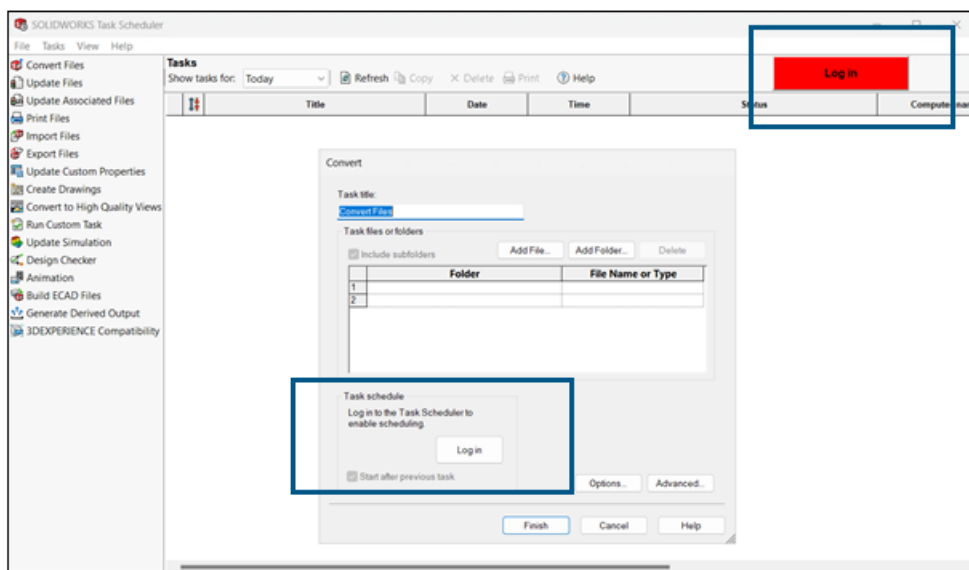
The **Update Now** option appears if you are assigned the Collaborative Designer role.

2. Click **Update Now** to begin installing, or **Skip** to continue using Marketplace without the update.

Before updating, make sure to save and close all open SOLIDWORKS files.

3. Follow these steps when the Installation Wizard opens:
  - a. Log in again, if prompted.
  - b. Install the **3DEXPERIENCE** Launcher if it is not already installed.
  - c. Proceed with installing the **3DEXPERIENCE** add-in.
4. After the installation is complete, restart SOLIDWORKS to apply the changes.

## SOLIDWORKS Task Scheduling Restored in SOLIDWORKS Connected (2025 FD03)



You can schedule tasks with local files in SOLIDWORKS Connected. Previously, tasks could only run immediately after creation. With this update, SOLIDWORKS can run scheduled tasks in the background, even when you are away from your machine.

**Benefits:** This update restores the ability to schedule tasks for a later time or on a recurring basis. In addition, the **username** and **password** fields have been removed from the **Generate Derived Output** task when running it with SOLIDWORKS Connected

To enable task scheduling:

1. Open the SOLIDWORKS Task Scheduler in SOLIDWORKS Connected by clicking **Tools > SOLIDWORKS Applications > SOLIDWORKS Task Scheduler**.
2. Click **Log in**, then enter your 3DEXPERIENCE platform **username** and **password**.  
Your credentials are saved and used to authorize scheduled tasks.
3. If your login information changes:
  - a. Click **Log Out**.  
Your initials will display if you have already signed in.
  - b. Click **Log in** again and enter your updated credentials.



The SOLIDWORKS Task Scheduler remembers your new credentials for future tasks.

## Stamping Drawings with User Names or Email Addresses (2025 FD03)

**3DEXPERIENCE** users can stamp drawings using extended attributes for user names or email addresses that display in **3DPlay**.

**Benefits:** This expands your possibilities for including useful information for drawings.

When you open a SOLIDWORKS drawing in **3DPlay** and it has PLM or extended attributes, you can preview those properties as annotations. In **3DPlay**, you can display the user names or email addresses of the users who participated in the drawing release process. The stamped information is also visible in the .pdf file-derived output.

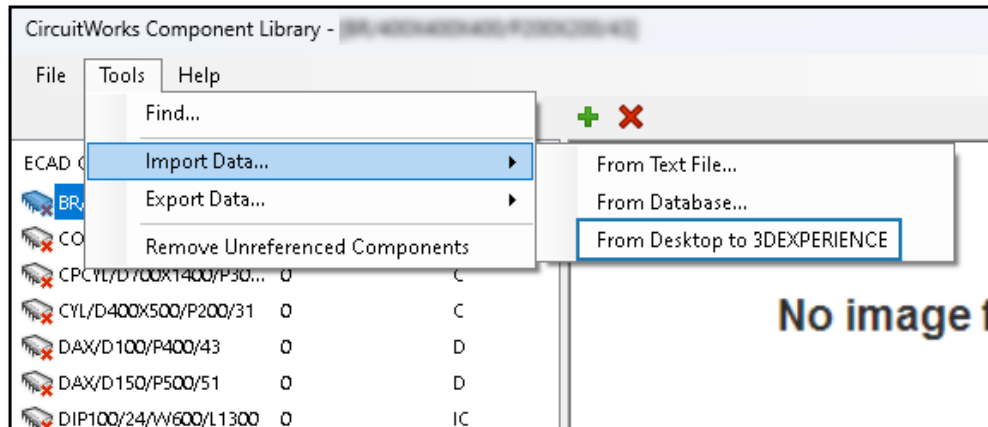
### To stamp drawings with user names or email addresses:

1. In a SOLIDWORKS drawing, add annotations that reference the following PLM properties:

ea_releasedby_name	User who performed the release process.
ea_createdby_name	User who created the drawing.
ea_changestatusby_name	User who performed the maturity change.
ea_releasedby_mail	Email address of the user who performed the release process.
ea_createdby_mail	Email address of the user who created the drawing.
ea_changestatusby_mail	Email address of the user who performed the maturity change.

2. Save the drawing to the **3DEXPERIENCE** platform to register the attributes.
3. In SOLIDWORKS, rebuild the drawing to ensure the annotations display correctly.
4. In MySession, use the **Change Maturity** command to update the state of the drawing, such as **In Work**, **Frozen**, **Released**, or **Obsolete**.
5. In **3DPlay** or any supported web viewer, open the drawing to see the updated annotations and verify that the information is accurate.

## Synchronizing CircuitWorks Component Libraries between SOLIDWORKS and SOLIDWORKS Connected (2025 SP03)



You can synchronize CircuitWorks component libraries between SOLIDWORKS Desktop and SOLIDWORKS Connected to keep your libraries up to date.

Use the following steps to copy components in either direction:

1. Go to **Tools > CircuitWorks > Component Library**. In **CircuitWorks Component Library**, select **Tools**.
2. Do one of the following:
  - Select **Import Data > From Desktop to 3DEXPERIENCE**.

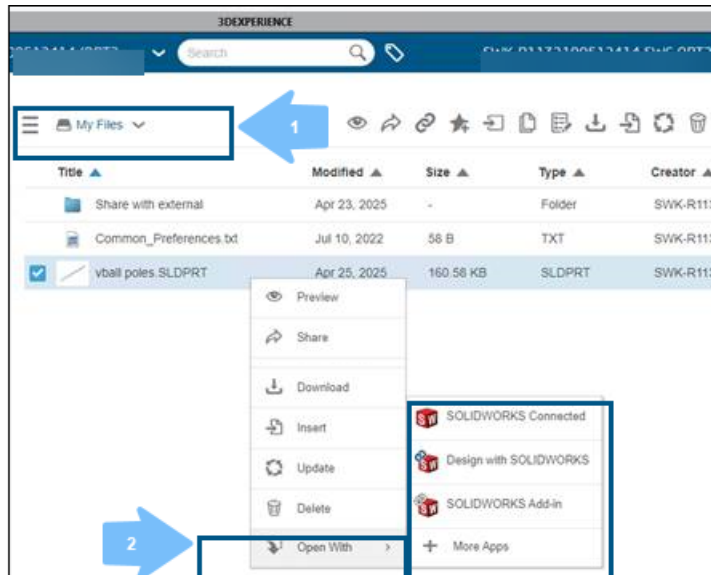
This synchronizes the entire desktop component library with the **3DEXPERIENCE** component library.

- Select **Export Data > From 3DEXPERIENCE to Desktop**.

This synchronizes the entire **3DEXPERIENCE** component library with the desktop component library.

3. Click **Yes**.

## Share Designs Using Open With in 3DDrive (2025 FD03)



You can use the **Open With** command in 3DDrive to open SOLIDWORKS files directly in SOLIDWORKS Connected, the Design with SOLIDWORKS app, or the SOLIDWORKS add-in.

**Benefits:** The **Open With** command in 3DDrive simplifies access to your files by reducing the steps to open and share models. You can collaborate directly from within SOLIDWORKS.

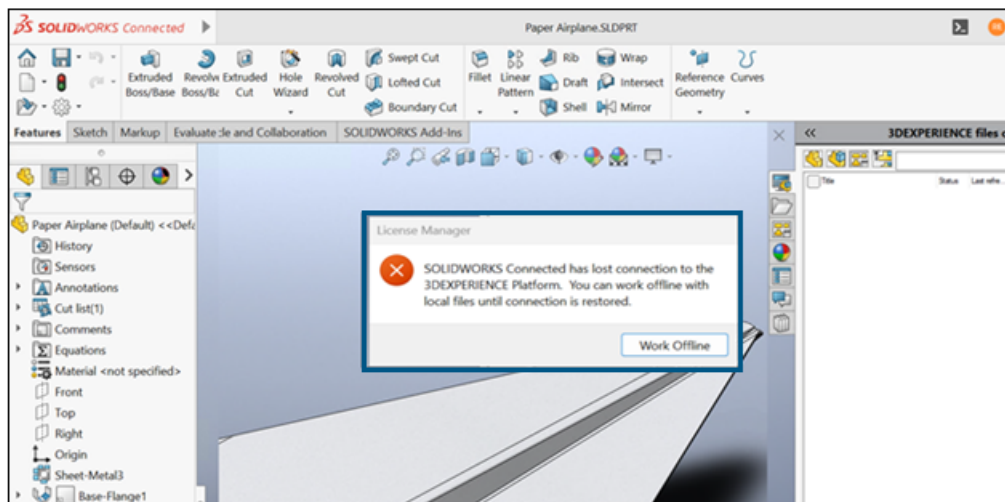
There is no need to generate a separate link. Recipients can preview the design, add markups, and send feedback to you, streamlining the review process. The existing method of uploading a model to 3DDrive, generating an external link, and sharing it with suppliers or external users is still available. Yet, this method only allows for one-way sharing, with limited feedback options.

If 3DDrive is not installed on your machine, a prompt lets you install it or continue without the installation.

### To open a SOLIDWORKS file using Open With in 3DDrive:

1. Navigate to 3DDrive in the 3DEXPERIENCE platform, and select a SOLIDWORKS file.
2. Right-click the file and select **Open With > SOLIDWORKS Connected**.
3. If 3DDrive is not installed, a prompt appears with the following options:
  - **Install (Recommended)**
  - **Continue without install**

## Work Offline When a Connection Is Unavailable (2025 FD03)



You can run SOLIDWORKS Connected, along with other **3DEXPERIENCE** Connected apps, even if the app cannot connect to the **3DEXPERIENCE** platform.

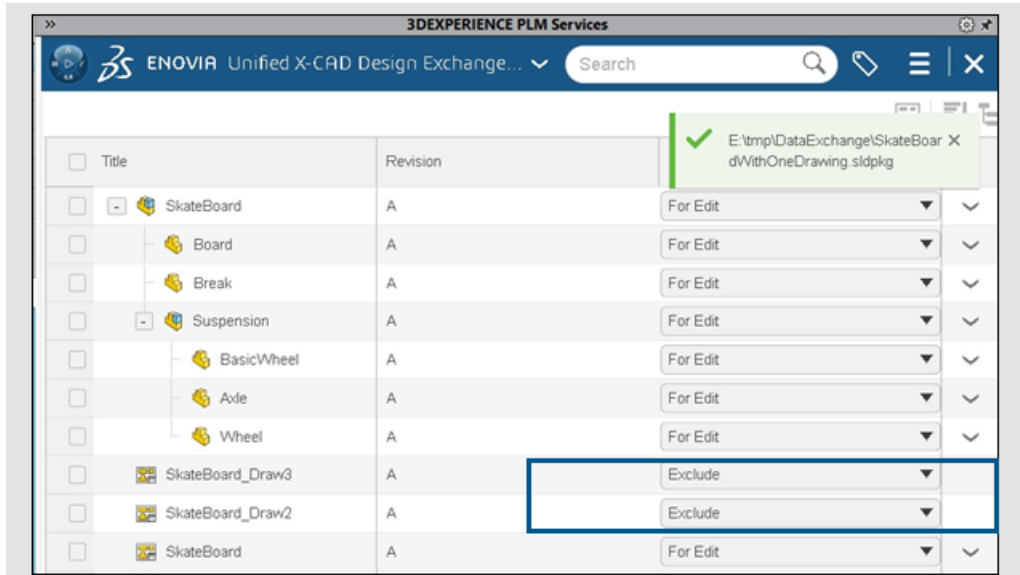
**Benefits:** This enhancement ensures uninterrupted access by allowing you to keep working in the app with your local files if you experience connectivity issues.

- **Introduced R2025x FD02:** If the app cannot connect to the **3DEXPERIENCE** platform and you have run SOLIDWORKS Connected in the last 30 days, the app prompts you to work offline.
- **Introduced R2025x FD03:** If you start your session as usual but lose connection during use, the app enters temporary offline mode and prompts you to work offline during disconnection.

While running in temporary offline mode, SOLIDWORKS Connected continues checking for a connection in the background. When the connection is restored, the app prompts you to restart to reconnect fully.

SOLIDWORKS Visualize Connected, DraftSight Connected, **3DEXPERIENCE** DraftSight Professional, and Electrical Schematic Designer also support temporary offline mode, allowing continued work with local files when the **3DEXPERIENCE** platform is unavailable.

## Excluding Drawings in an Export Package (2025 FD03)



You can control which drawings to include or exclude when using the **Export as Package** tool, located in the Collaboration tab of the MySession action bar. This tool lets **3DEXPERIENCE** users export data packages from SOLIDWORKS.

**Benefits:** The **Exclude** option helps you avoid exporting unnecessary or outdated drawings and reduces the size of the exported package.

When setting up your export package, you can manually select which drawings to include.

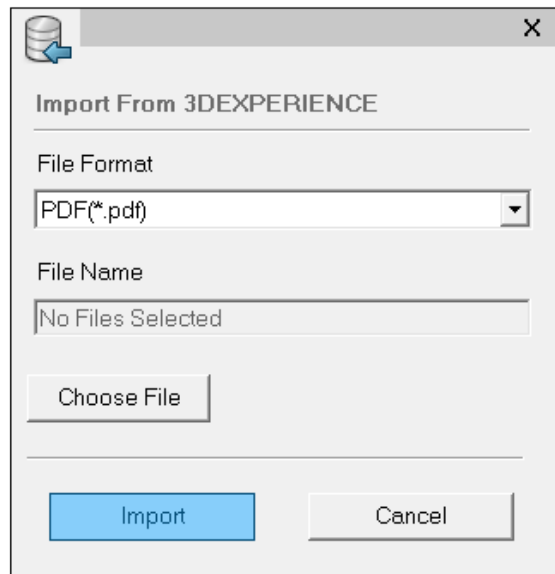
To include or exclude drawings in an export package:

1. With the parts or assemblies open in MySession, select a file and click the **Collaboration > Export as Package** in the action bar.
2. Select a node from the file and click **Add Drawings** to display any linked drawings in the view.
3. For each drawing, under the **Purpose** column:
  - Choose **For Edit** to include it in the package.
  - Choose **Read-Only** to include it as non-editable.
  - Choose **Exclude** to leave it out of the package.

You can only include one revision of a drawing in each export package. The tool does not support exporting multiple revisions of the same drawing.

## SP2 and FD02

### Importing a PDF File as a Block from the 3DEXPERIENCE Platform (DraftSight Connected Only) (2025 FD02)



You can use the **IMPORTPDFFROM3DEXPERIENCE** command to import a PDF file as a block from the 3DEXPERIENCE platform.

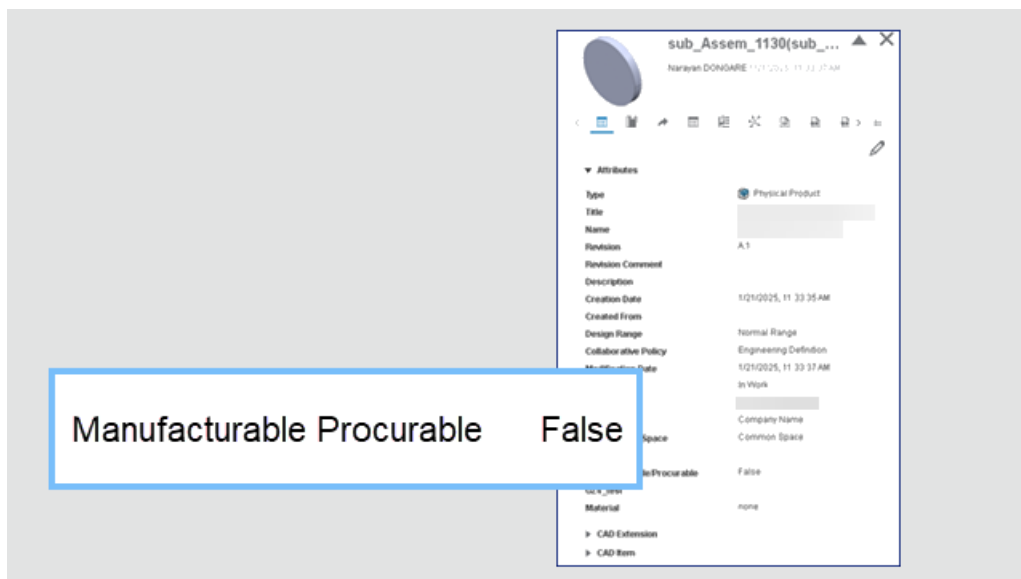
#### To import a PDF file as a block from the 3DEXPERIENCE platform:

1. Do one of the following:
  - Click **Import** > **Import from 3DEXPERIENCE**.
  - Click **File** > **Import** > **Import from 3DEXPERIENCE**.
  - Type `IMPORTPDFFROM3DEXPERIENCE` in the command window.
2. In the Import from 3DEXPERIENCE dialog box:
  - a. In **File Format**, select **PDF (\*.pdf)**.
  - b. Click **Choose File**.
3. In the Open dialog box:
  - a. Select a PDF file.
  - b. Click **Open**.

In the Import from 3DEXPERIENCE dialog box, **File Name** displays the selected file.

4. Click **Import**.
5. In the PDF Import dialog box, click **OK**.

## Mapping Promote to Manufacturing and Procurable Property (2025 FD02)



When you save an assembly to **3DEXPERIENCE** for the first time and select **Promote**, **3DEXPERIENCE** maps the **Manufacturable/Procurable** property to the assembly file.

**Benefits:** The **Manufacturable/Procurable** property appears in the **3DEXPERIENCE** file properties of the assembly. You can filter items in the **Engineering Release** app on the platform based on their manufacturability and procurability status.

To promote an assembly in SOLIDWORKS to **3DEXPERIENCE**:

1. Open the **FeatureManager** design tree for the assembly.
2. Select the **ConfigurationManager**.
3. Select the configuration and right-click **Physical Product** > **Edit Physical Product** > **Promote**.

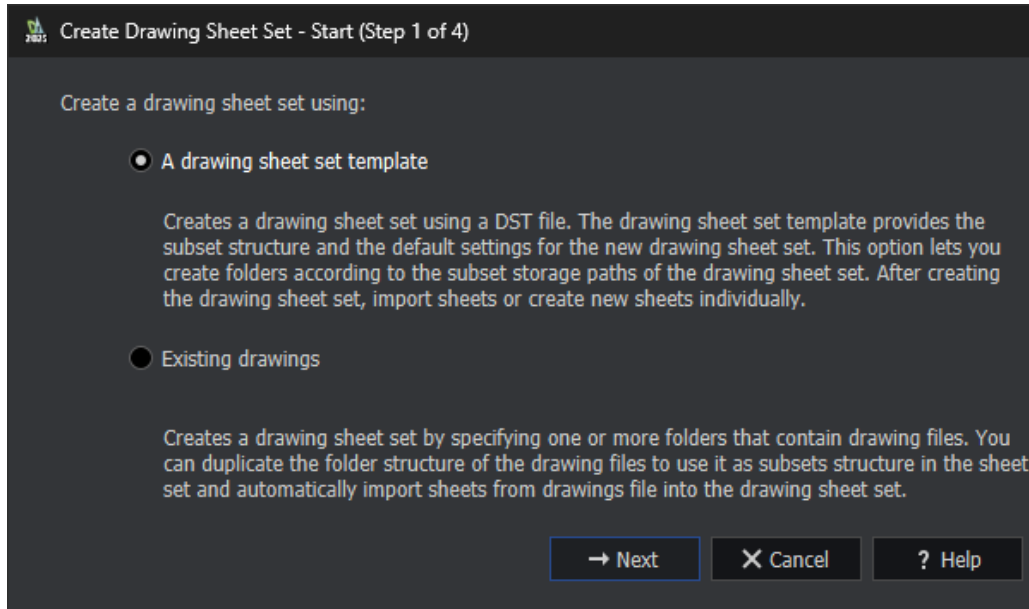
When you promote an assembly file to **3DEXPERIENCE**, the **Manufacturable/Procurable** property is set to **False**.

Similarly, if you set the **Show/Hide** property in SOLIDWORKS, **Manufacturable/Procurable** is set to **True** in **3DEXPERIENCE**.

This update also applies to the **Batch Save to 3DEXPERIENCE** tool.



## Sheet Set Manager on the 3DEXPERIENCE Platform (DraftSight Connected Only) (2025 FD02)




**3DEXPERIENCE** DraftSight lets you create the Sheet Set Data (DST) files and save them to bookmarks. You can open the saved DST files from bookmarks.

You can also define the properties of the Sheet Set Manager. See [Working with Drawing Sheet Sets](#). You can create DST files using an existing drawing or drawing sheet set template. **3DEXPERIENCE** DraftSight creates DST files as PLM objects.

### Creating Drawing Sheet Sets Using an Existing Drawing

You can use the Create Drawing Sheet Set wizard to create drawing sheet sets from an existing drawing.

#### To create drawing sheet sets using an existing drawing:

1. In the **Sheet Set Manager** palette, click **New Drawing Sheet Set** .
2. In the Create Drawing Sheet Set - Start wizard, select **Existing drawings** and click **Next**.
3. Click **Browse** for **Location for drawing sheet set data file (\*.dst)**.
4. In Browse for Drawing Sheet Set Folder dialog box, click **Select from 3DEXPERIENCE**.
5. In the Select a Bookmark dialog box:
  - a) Select an existing bookmark or create a bookmark to save the DST file to.
  - b) Click **Select**.


Alternatively, you can select a folder from **This PC**.
6. Click **Drawing Sheet Set Properties** to select a bookmark for the **Model view** from the **3DEXPERIENCE** platform.  
You can select a bookmark for **Label block for views** and **Callout blocks**.
7. In the Create Drawing Sheet Set - Drawing Sheet Set Details wizard, click **Next**.

8. In the Create Drawing Sheet Set - Choose Sheets wizard, click **Browse**.
  - a) In the Browse for Folders dialog box, select a folder from your computer or a bookmark that contains drawings.
  - b) Click **Specify Folder**.
9. In the Create Drawing Sheet Set - Choose Sheets wizard, click **Next**.
10. In the Create Drawing Sheet Set - Finalize wizard, click **Finish**.

#### Creating Drawing Sheet Sets Using a Drawing Sheet Set Template

You can use the Create Drawing Sheet Set wizard to create drawing sheet sets using a drawing sheet set template.


#### To create drawing sheet sets using a drawing sheet set template:

1. In the **Sheet Set Manager** palette, click **New Drawing Sheet Set** .
2. In the Create Drawing Sheet Set - Start wizard, select **A drawing sheet set template** and click **Next**.
3. In the Create Drawing Sheet Set- Drawing Sheet Set template wizard:
  - a) Select **Browse to another drawing sheet set to use as a template**.
  - b) Click **Browse**.
4. In the Browse for Drawing Sheet Set dialog box, click **Open from 3DEXPERIENCE**.
5. In the Open dialog box:
  - a) Select a drawing sheet set template (.DST) from **3DSearch** or **Bookmarks**.
  - b) Click **Open**.

In the Create Drawing Sheet Set - Drawing Sheet Set Template wizard, the name of the drawing sheet set template (DST) appears.
6. In the Create Drawing Sheet Set - Drawing Sheet Set Template wizard, click **Next**.
7. In the Create Drawing Sheet Set - Drawing Sheet Set Details wizard, click **Next**.
8. In the Create Drawing Sheet Set - Finalize wizard, click **Finish**.

#### Opening Drawing Sheet Sets

#### To open drawing sheet sets:

1. In the **Sheet Set Manager** palette, click **Open Drawing Sheet Set** .
2. In the dialog box, do one of the following:
  - Select the drawing sheet set (DST) and click **OK**.
  - From **Bookmarks** or **3DSearch**, click **Open from 3DEXPERIENCE**, select the sheet set manager file, and click **Open**.

The **Sheet Set Manager** palette displays the DST file's references.

## Global Rules in the 3DEXPERIENCE Integration Rules Editor (2025 FD02)

3DEXPERIENCE Integration Rules Editor

Parts Assemblies

**Sub-typing rules**

☒ Enable a global rule for all non-sub-typed parts

ID	Action	Sub-Type Name	Sub-Type Rule Description
1		Global rule (all non-sub-typed parts)	The global rule will apply to all parts which do not match the sub - typing rules below.

<

**Configuration mapping rules for sub-type ID 1 "Global rule (all non-sub-typed parts) "**

Single configuration part will always be single physical product. Some child configurations such as exploded views will always be mapped as representations.

Mapping for multi-configuration parts :

☒ Single physical product with representations

Use the  as the physical product

or use the  if that configuration name does not exist

☐ Multiple physical product

In the **3DEXPERIENCE** Integration Rules Editor, **3DEXPERIENCE** users can create global rules that apply to all non-sub-typed parts and assemblies.

**Benefits:** You can easily create a global rule instead of having to use workarounds.

**To create a global rule:**

1. To open the Editor, click **Tools > Options > System Options > 3DEXPERIENCE Integration > 3DEXPERIENCE Integration Rules Editor**.
2. In the dialog box, on the Parts or Assemblies tab, under **Sub-typing rules**:
  - a. Select **Enable a global rule for all non-sub-typed parts or assemblies**.

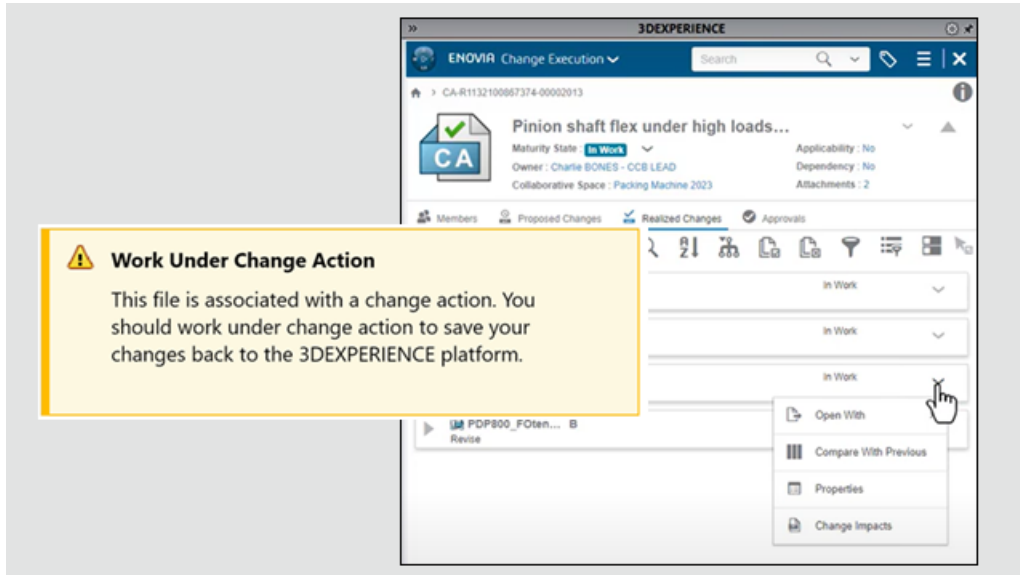
Row **0** (the default) is hidden and row **1** appears. The **Edit** and **Delete** tools are unavailable because you cannot change the sub-type rule definition of a global rule.

The global rule is always row **1**.

- b. In the table, under **ID**, click the **1**.  
The bottom section of the dialog box expands for you to specify configuration mapping rules for the global rule.
- c. Specify the configuration mapping rules for the global rule and click **OK**.  
The software stores the global rule in the **.XML** file at the location specified in the System Options dialog box for **3DEXPERIENCE Integration Rules Folder**.

If you select this option and invoke the **Update for 3DEXPERIENCE Compatibility** command, the system applies all user-defined sub-typing rules. Parts or assemblies that were not included in the defined sub-typing rules use the configuration-mapping logic defined in the global rule. If you clear the global rule, such parts or assemblies use the default logic for configuration mapping.

## Warning for Saving Files Associated with Change Action Restrictions (2025 FD02)



When you try to save a locked file in SOLIDWORKS, a warning appears if the file is blocked by an active **Change Action**. The warning means you cannot save the file to the platform until the Change Action is addressed.

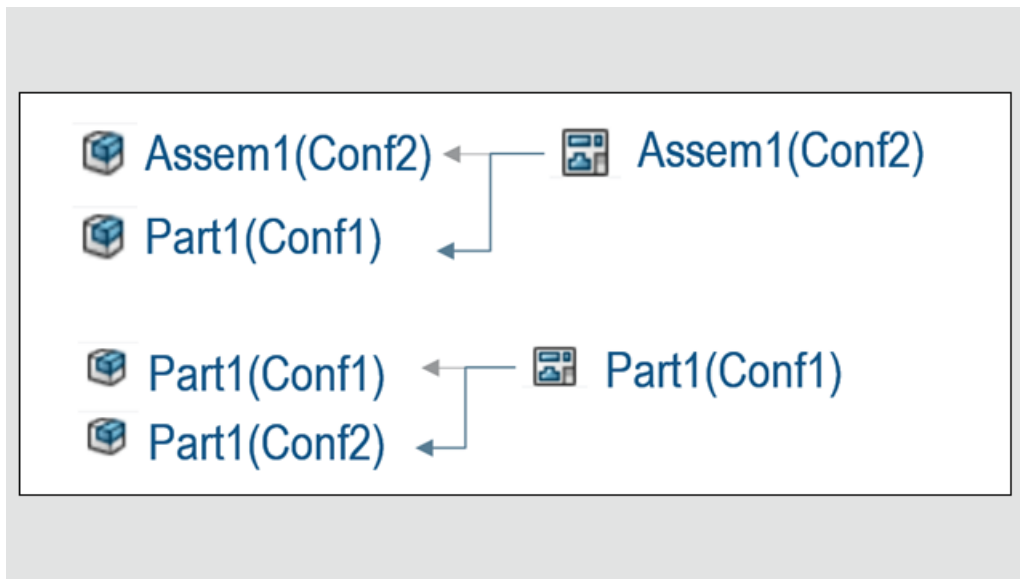
**Benefits:** This update ensures you resolve any active Change Actions associated with your files before proceeding.

To resolve a Change Action:

1. Activate **Work Under Change** through the relevant **Change Action**:
  - a. In MySession, click the highlighted **Change Action** and then click **Change Action** again.
  - b. From the list, select the relevant **Change Action** to activate **Work Under Change**.

If you do not have access to the **Change Action**, it may not appear in the list. Consult your platform administrator for assistance.
  - c. Click **OK** and save the file to the platform.
2. Alternatively, you can open the **Change Execution** app to review and resolve the **Change Action** status.
3. Request approval or complete any required steps before saving.

## Set Drawing Title from First Model View (2025 FD02)



When saving a drawing, SOLIDWORKS can automatically assign the drawing title based on the first referenced model view.

**Benefits:** This update helps maintain consistency by keeping the drawing title aligned with its parent model.

**You can enable this functionality by selecting the Use part or assembly title as new drawing title option, located under Tools > Options in the action bar.**

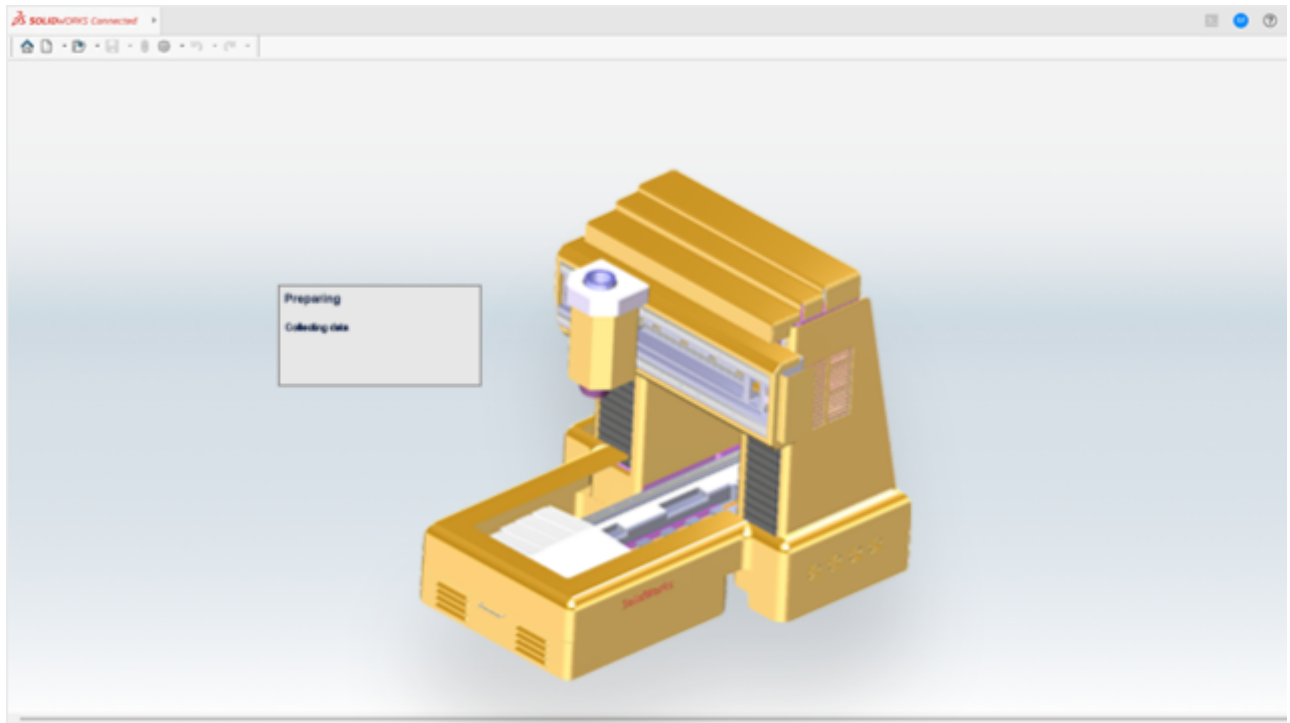
When you enable this option, the drawing title is determined by the first model view added to the drawing. This behavior can apply to when you first save the file, or on every save, depending on your configuration.

The rules for assigning a drawing title are:

- If the referenced model has one configuration, the drawing title matches the model name.
- If the referenced model has multiple configurations, the drawing title includes the model name and the configuration from the first view. For example:  
`Assembly1 (Config1).`

This option also applies to the **Batch Save to 3DEXPERIENCE** tool, helping to streamline filenaming during bulk operations.

## Improved Method for Opening 3DEXPERIENCE Files (2025 FD02)



SOLIDWORKS has improved how it opens **3DEXPERIENCE** files not stored in your local cache.

**Benefits:** This method improves performance by first retrieving metadata for the selected configuration on the client. It then caches the complete configuration metadata and downloads the required files, even if they are not yet loaded in the SOLIDWORKS session.

Some actions remain temporarily disabled until all the files are fully available. These actions include opening other files, locking and unlocking, replacing revisions, modifying properties, saving, and refreshing. You can continue using **3DSearch** while downloading the rest of your files.

**Note:**

- Switching configurations in an assembly is blocked until all components download.
- If the download process is interrupted, SOLIDWORKS prompts you to reload the files or restart to clear the cache.

## Displaying the First Revision in a 3DEXPERIENCE Revision Table (2025 FD02)

3DEXPERIENCE Revision Table				
ZONE	REV.	DESCRIPTION	CREATION DATE	REVISED BY
A	1	Created drawing	1/1/2025	Tom
A	17	Added fillets	1/14/2025	Tom
A	18	Added chamfers	1/14/2025	Tom
A	19	Added dimensions	1/14/2025	Tom
A	20	Changed 0.40 in. to 0.50 in.	1/14/2025	Tom

You can keep the first revision in the first row regardless of the amount of rows displayed.

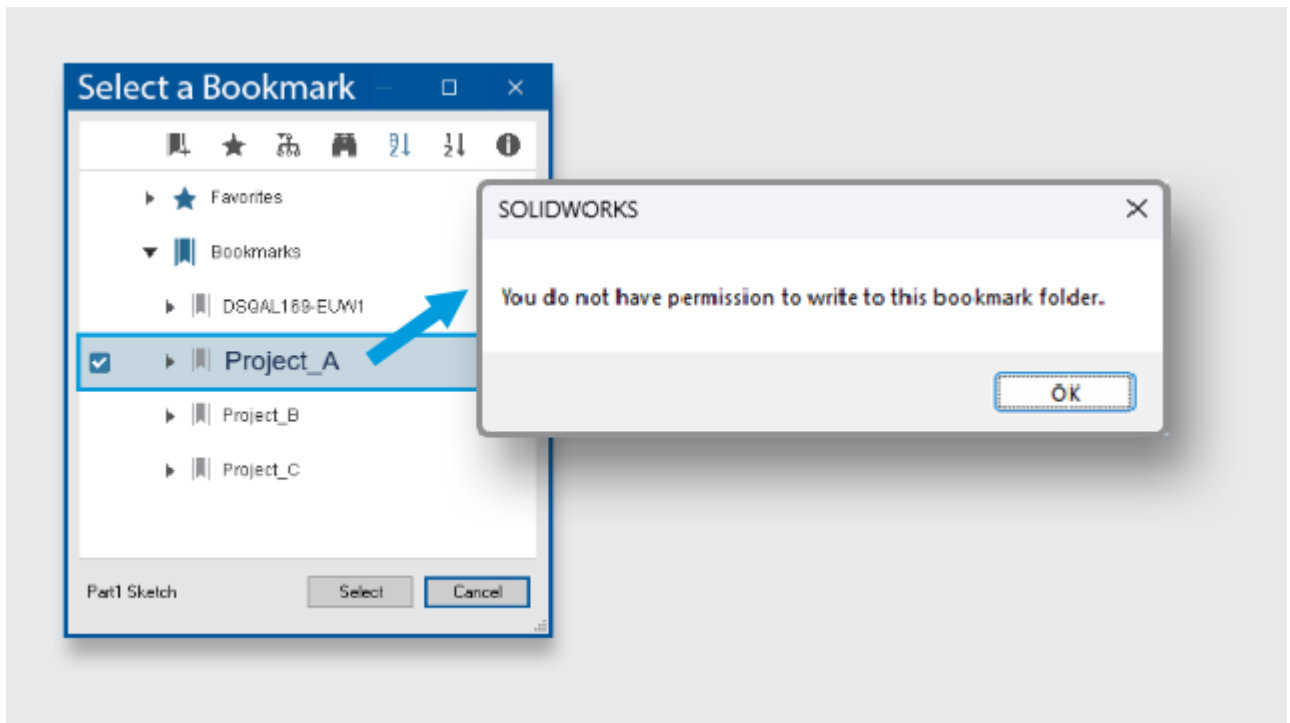
**Benefits:** You can always see the first revision, even if there are several revisions in the table.

**To display the first revision in a 3DEXPERIENCE revision table:**

1. Click **Tools > Options > Document Properties > Tables > Revision**.
2. Under **Types**, select **3DEXPERIENCE driven Revision Table**.
3. Select **Always show First Revision** and click **OK**.



## Notifications for Restricted Bookmarks (2025 FD02)



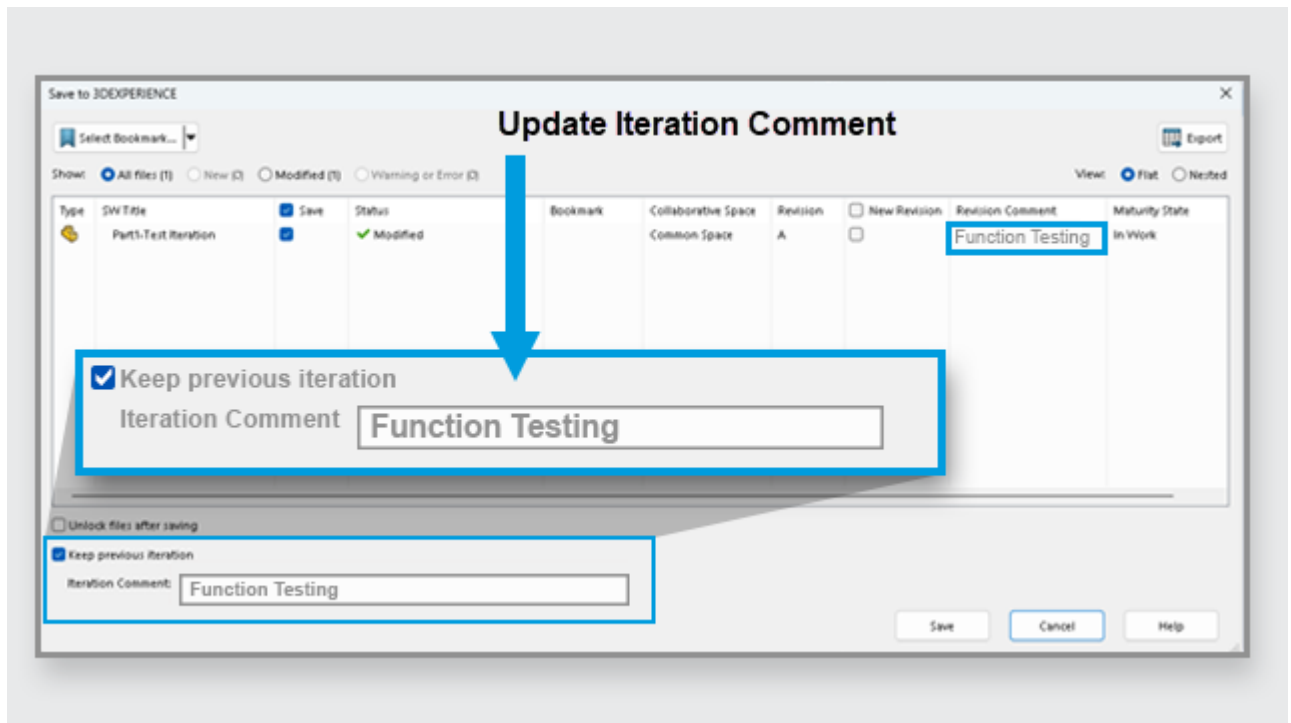
A notification appears when you attempt to use bookmarks that require write permissions.

**Benefits:** This update prevents you from accessing bookmarks that require write access.

When bookmarks are read-only, options such as **Select**, **Apply**, **Apply to All**, and **Apply to Selected** appear grayed out. This behavior applies to the Select a Bookmark dialog box, the Save to 3DEXPERIENCE dialog box, the Batch Save to 3DEXPERIENCE tool, and the default bookmark selection in **Tools > Options**.

While some actions need write access, others such as downloading from a bookmark, only need read access. Bookmarks in **Frozen**, **Completed**, or **Archived** states are usually read-only. Even when performing an action that only requires read access, the notification still appears to keep you informed.

## Adding Comments to File Iterations (2025 FD02)

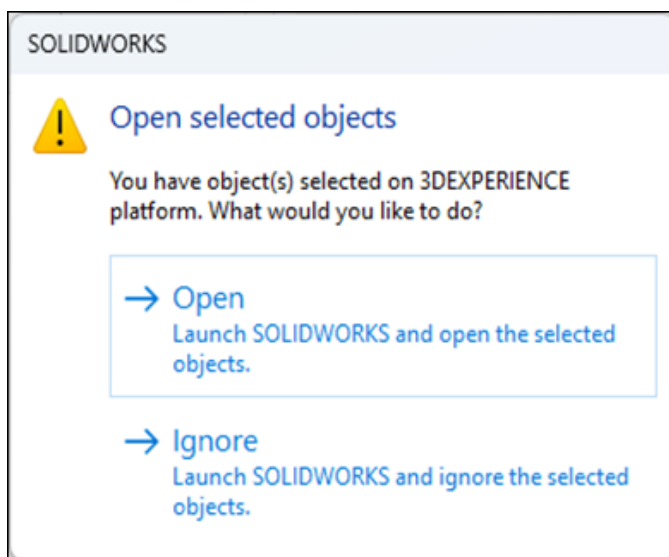


When saving a file to the **3DEXPERIENCE** platform, you can add comments to each iteration.

**Benefits:** This update makes it easier for you to find specific iterations.

If a **Revision Comment** is available for the parent file, the software automatically fills the **Iteration Comment** with the same text. If you did not add a **Revision Comment**, the **Iteration Comment** remains empty so you can add one if needed.

## Verifying Object Selection (2025 FD02)



When launching SOLIDWORKS from the **3DEXPERIENCE** platform, a user may unintentionally select an object, causing SOLIDWORKS to open it once the session is ready. After SOLIDWORKS starts, a dialog box appears, letting the user to either continue opening the selected object or interrupt the process.

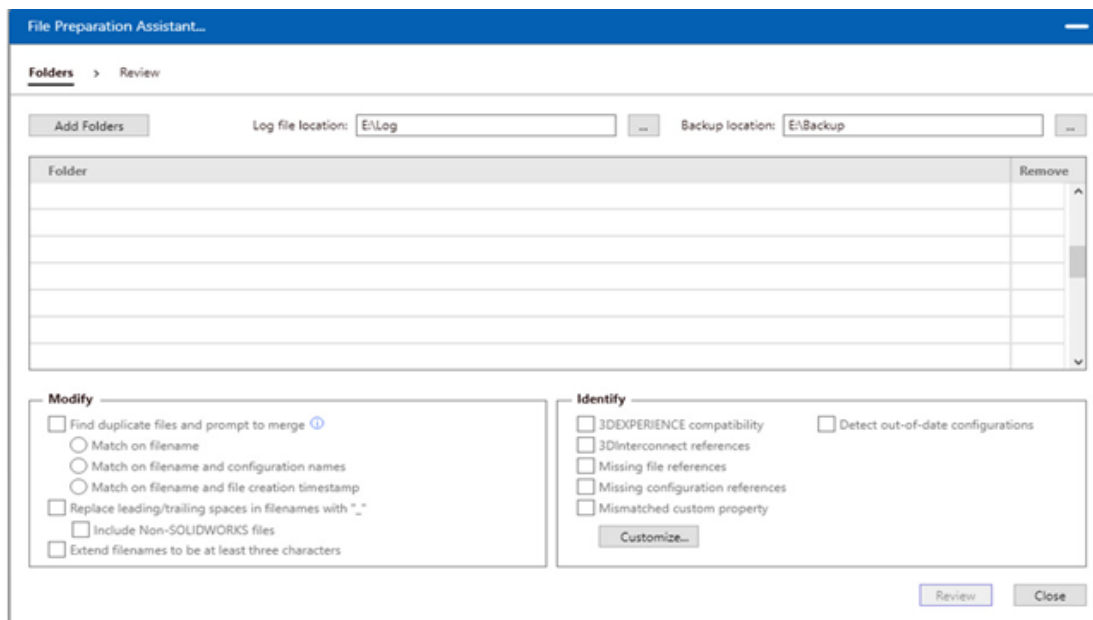
**Benefits:** This update helps prevent unintended openings when launching SOLIDWORKS.

You can choose:

- **Open:** Proceeds with loading the selected object in SOLIDWORKS.
- **Ignore:** Opens SOLIDWORKS without loading the object chosen.

The behavior does not apply when launching SOLIDWORKS from a desktop shortcut or script.

## File Preparation Assistant User Interface Changes (2025 FD02)





**3DEXPERIENCE** users can use the File Preparation Assistant with a simplified user interface.

**Benefits:** The simplified user interface helps you streamline the workflow.

Changes to the File Preparation Assistant user interface include:

- In the dialog box, **Add Folders** replaces **Add Folder**.
- **Log File:** and **Backup:** at the top of the screen replace **Backup and Logs** including:
  - **Select a folder in which to create a backup**
  - **Select a folder in which to create log files**
- There is no **Options**.
- There is no **Start** at the bottom of the screen.
- **Modify** includes:
  - **Find duplicate files and prompt to merge**
    - **Match on filename**
    - **Match on filename and configuration names**

- **Match on filename and time creation timestamp**
- **Replace leading/trailing spaces in filenames with "\_"**
  - **Include non-SOLIDWORKS files**
- **Identify** includes:
  - **3DEXPERIENCE compatibility**
  - **3DInterconnect references**
  - **Missing file references**
  - **Mismatched custom property**

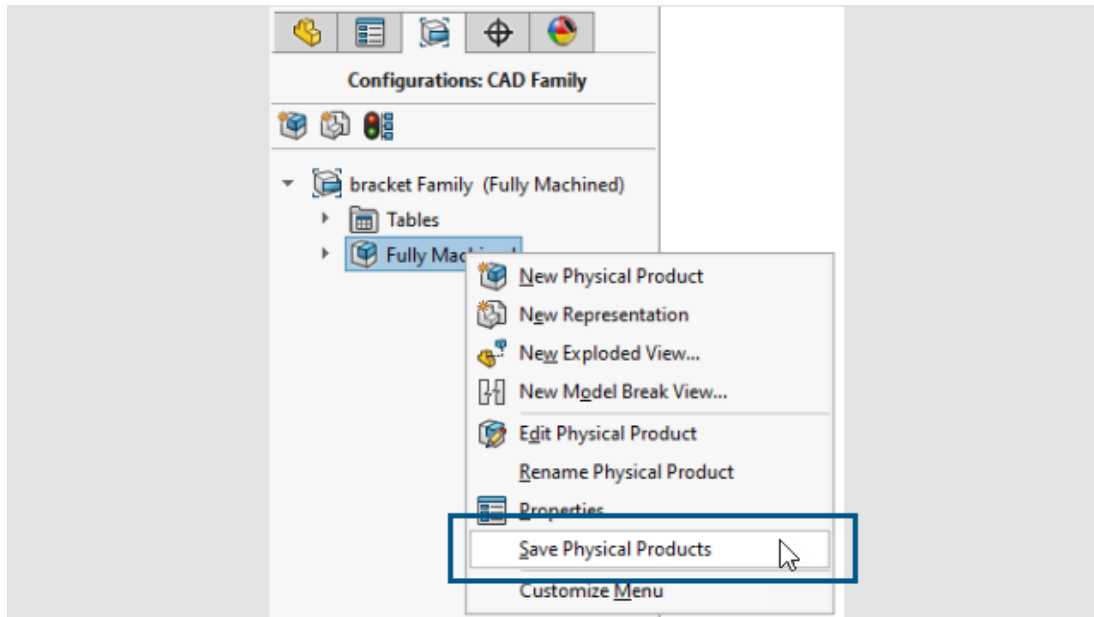
	<b>Delete Folder</b>	It displays a black X icon to indicate you can remove the folder.
	<b>Delete Folder Hover</b>	When you hover over the black X icon, it turns red.

File Preparation Assistant includes an option to include non-SOLIDWORKS files.

**To include non-SOLIDWORKS files:**

1. In SOLIDWORKS, click **Tools > File Preparation Assistant**.
2. In the dialog box, click **Add Folders**.
3. In the Browse for Folder dialog box, select a folder and click **OK**.
4. Click **Log File:** and choose a location where the software downloads the log file.
5. Click **Backup:** and choose a location where the software downloads the backup file.
6. Under **Modify**, select two options:
  - a. **Replace leading/trailing spaces in file names with "\_"**.
  - b. **Include Non-SOLIDWORKS Files**
7. The File Preparation Assistant automatically performs additional checks.

## Saving Physical Products and Configurations (2025 FD02)

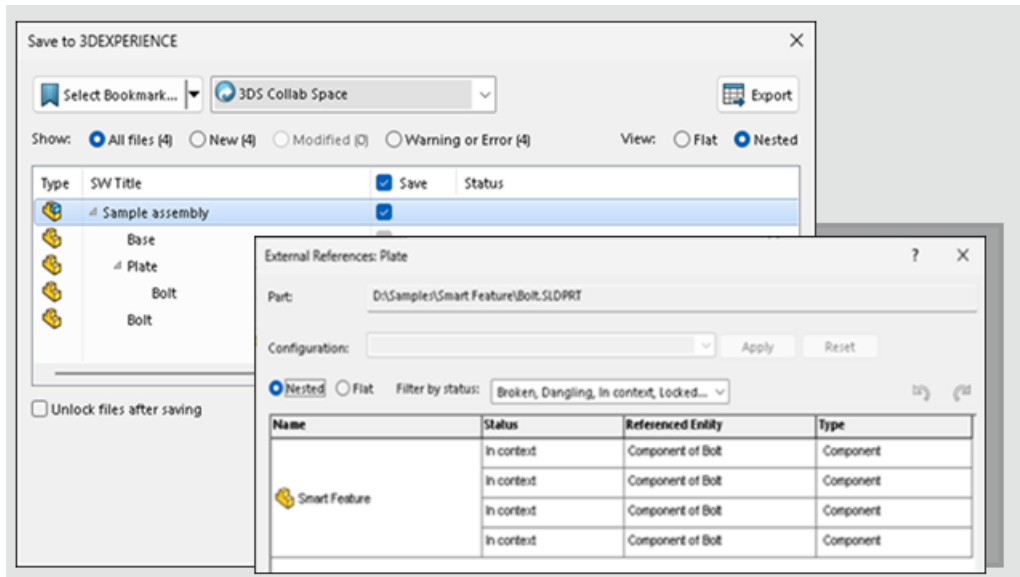


When you update a file for **3DEXPERIENCE** compatibility, the **Save Physical Products** command on the shortcut menu lets you directly save physical products. In the ConfigurationManager, when you right-click a configuration and click **Save Configurations**, in the Save As dialog box, you can click **Save to 3DEXPERIENCE** to save the file to the platform.

**Benefits:** This functionality improves performance by letting you save physical products from directly under the ConfigurationManager. Previously, this was not available.

The **Save Physical Products** command is available only for physical products. If a physical product contains representations, when you save the physical product, the software saves the representations under the physical product.

## Enhanced Support for Smart Components References (2025 FD02)

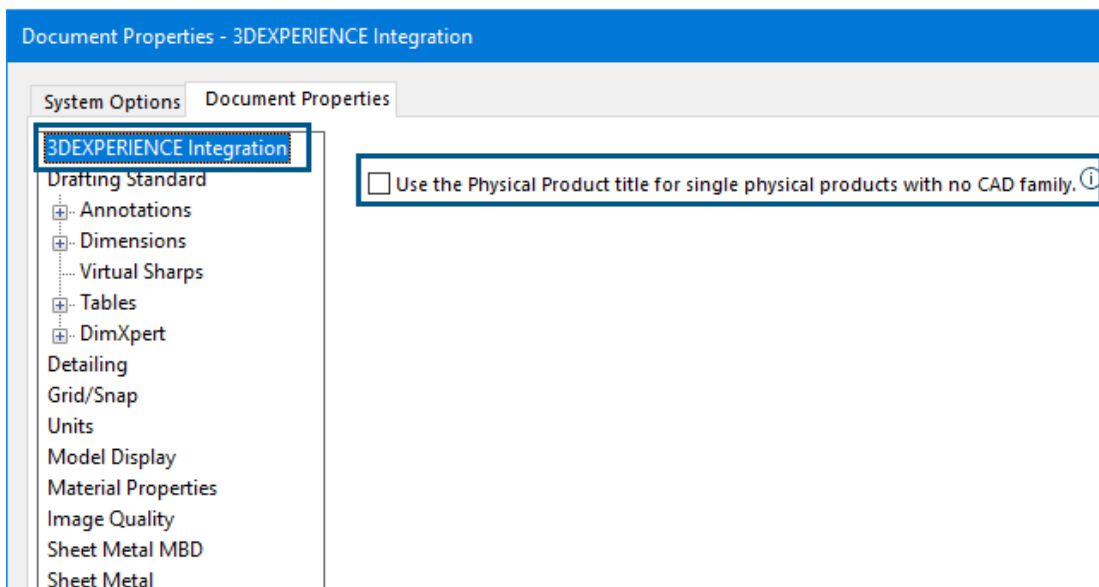


When you save a smart component to the **3DEXPERIENCE** platform, its references to additional components remain intact.

**Benefits:** This improvement helps preserve component relationships, making it easier to manage assemblies without losing connections.

For example, in the Save to 3DEXPERIENCE dialog box, right-click a subcomponent and select **External References** to view associated components.

## Synchronizing the Title of Single Physical Products (2025 FD02)



In a single physical product file with no CAD family, **3DEXPERIENCE** users can sync the **SOLIDWORKS** title to the physical product title.

**Benefits:** This approach avoids title sync issues for single physical products with no CAD family.

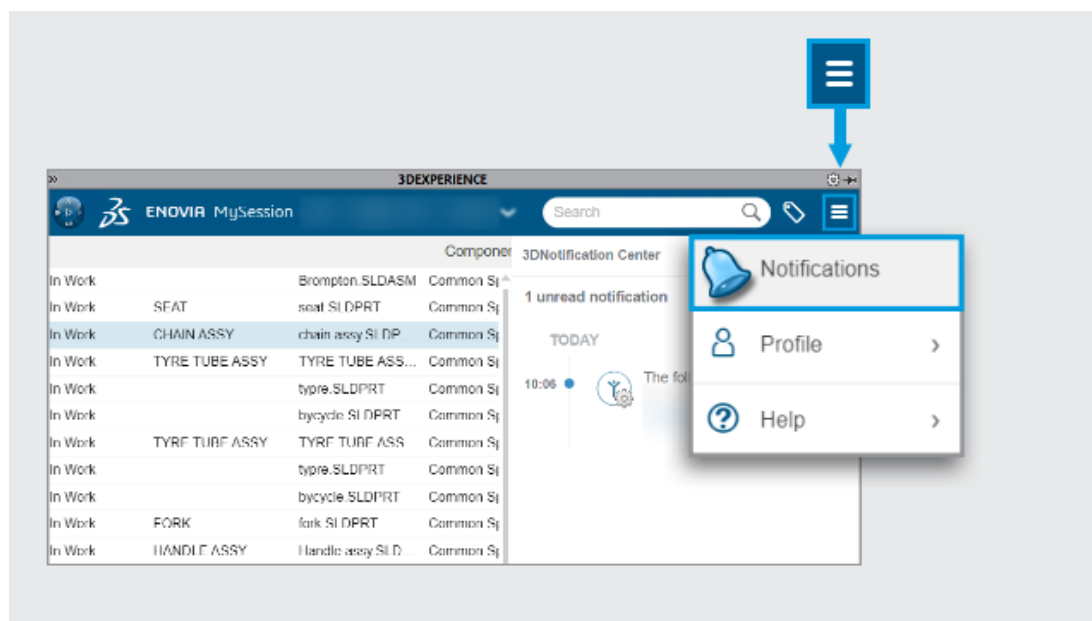
### To sync a single physical product file:

1. Open a part or assembly that is a single physical product file. It must not have a CAD family on the Configurations tab.
2. Click **Tools > Options > Document Properties > 3DEXPERIENCE Integration**.
3. In the dialog box, select **Use the Physical Product title for single physical products with no CAD family**, then click **OK**.

In offline mode, the software uses the last known values for the physical product title.

The Rename Title dialog box remains unchanged for models that are not single physical products. If you change a single physical product model to a multiple physical product model, for example by adding a CAD family, the title reverts to the name you define. Also, subsequent changes use the existing style of the Rename Title dialog box.

## Managing Platform Notifications in the SOLIDWORKS Task Pane (2025 SP2)




You can view and interact with notifications from platform apps directly in the 3DEXPERIENCE tab of the Task Pane.

**Benefits:** This feature lets you manage notifications without switching to the platform, keeping tasks and updates accessible within SOLIDWORKS.

You can open notifications for apps like 3DDrive, 3DSwym, and PartSupply. Clicking a notification shows its details within the same interface. Supported apps also include:

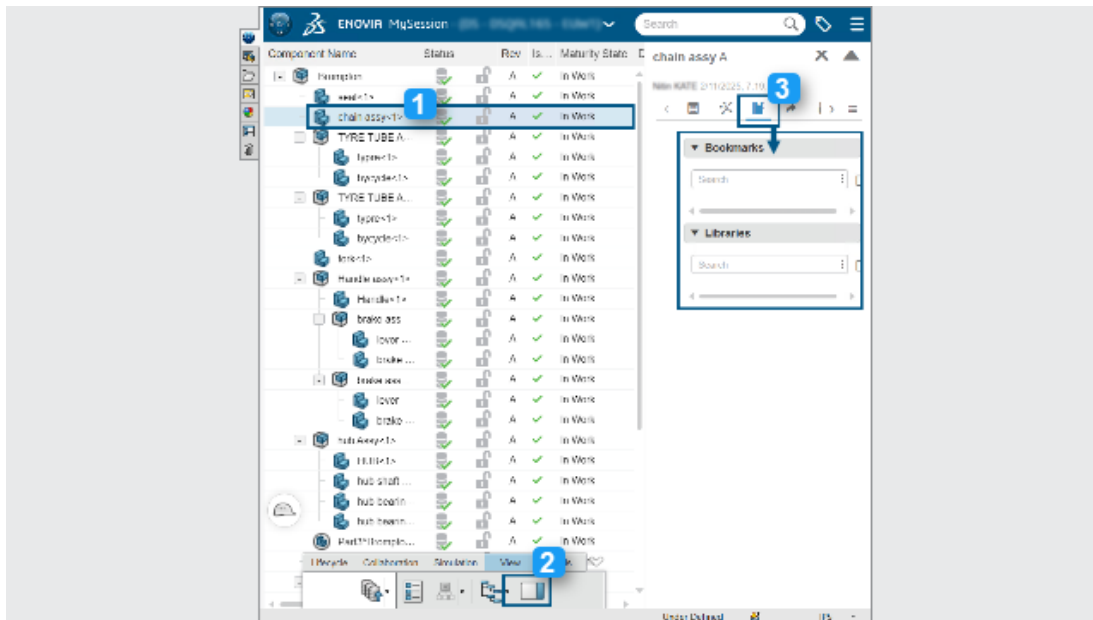
- 3DSearch
- Collaborative Tasks

- Collaborative Lifecycles
- Bookmark Editor

To view notifications, click the 3DEXPERIENCE tab in the Task Pane. Then in MySession, right-click the **Main Menu** in the top bar and click **Notifications** .

If you do not see notifications for an app, check the subscription settings in the **3DNotification Center**. Those settings are available in **Notifications Settings > Preferences**.

## Classifications Tab in MySession (2025 SP2)



You can use the Classifications tab in MySession to search and manage physical product classifications.

**Benefits:** This feature integrates data from 3DEXPERIENCE classification apps, such as IP Classify and Reuse.

To access the Classifications tab:

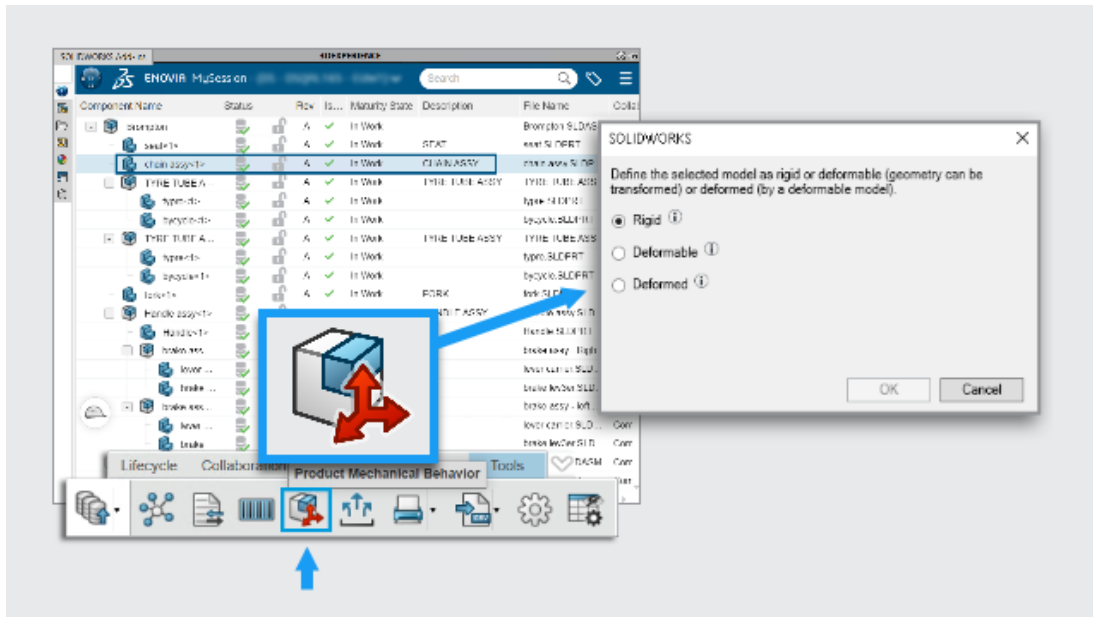
1. Select a component from the tree in MySession.
2. In the action bar, click **View > Display Panel**.
3. In the expanded tab, click **Classifications**.


You can search for bookmarks, and libraries of general classes and security classes in which the component is classified.

You can also access classifications from 3DSearch. When searching for a component, click **Classifications** to open the expanded tab.



## Managing Deformable Components (2025 SP2)




You can use the **Product Mechanical Behavior**  command in MySession to define how a component behaves in an assembly. You can classify it as rigid, deformable, or deformed while keeping a single part number.

**Benefits:** This approach gives SOLIDWORKS users a simple way to manage flexible components without leaving their workflow.

In real-world design, some components, such as hydraulic hoses or springs, start with a fixed shape but deform when placed in an assembly. Engineers need a way to track these changes without switching between applications.

To define a component as deformable:

1. Open an assembly saved in the **3DEXPERIENCE** platform.
2. In MySession, select a component in the tree.
3. On the action bar, click **Tools > Product Mechanical Behavior** .
4. In the dialog box, select an option:

<b>Rigid</b>	The component does not change shape.
<b>Deformable</b>	The component can take different shapes in an assembly.

## Deformed

A deformable product deforms a deformed product.

Click **Select Deformable** to choose a deformable product.

For a deformable 3DPart, you can only set 3DParts as deformed. For a deformable product, you can only set products as deformed.

When a component is marked as deformable or deformed, it remains linked to the original part number in the bill of materials.

**For more information on Product Mechanical Behavior, see [Action Bar of MySession](#).**

## Recent File List (2025 SP2)

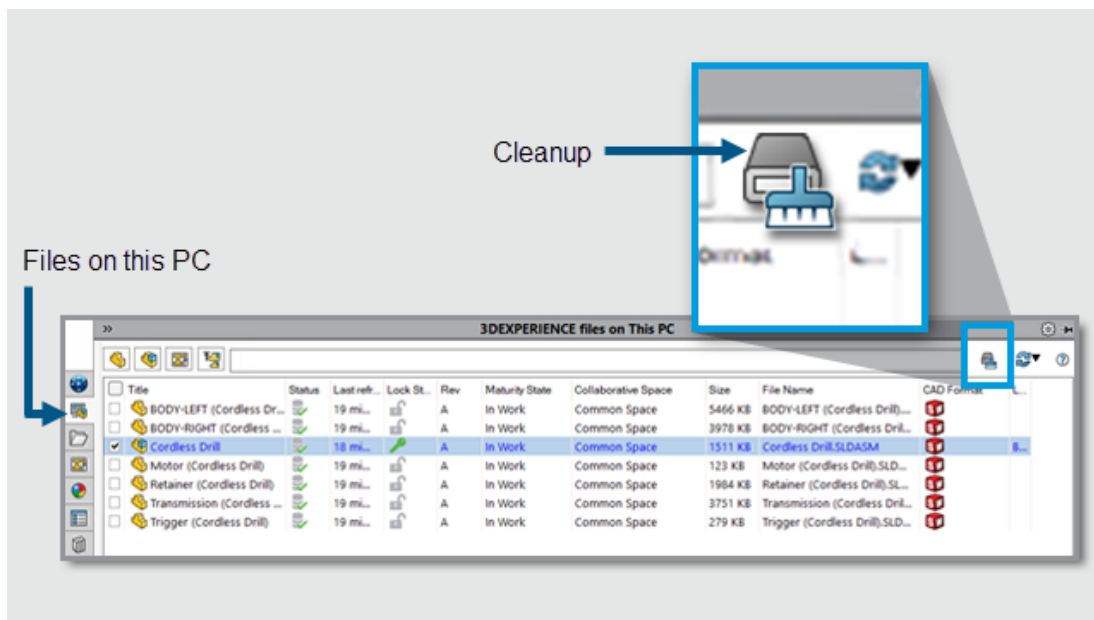
In the Welcome and Open dialog boxes, the Recent tab displays files from the current tenant only.


**Benefits:** This provides a clearer view and avoids the possibility of saving a file to another tenant.

You can also open a recent file even after it is cleared from the cache.

This feature is not available in offline mode.


## Cleanup Local Cache in the 3DEXPERIENCE Files on This PC Tab (2025 SP2)



You can remove unused files from your local cache in the 3DEXPERIENCE Files on This PC tab using the **Cleanup**  tool.

**Benefits:** This tool helps you free up disk space on your local machine and simplifies file organization without affecting files stored on the **3DEXPERIENCE** platform.

To use the Cleanup tool:

1. In the Task Pane, click the 3DEXPERIENCE Files on this PC tab. Then click **Cleanup**  in the toolbar.
2. In the dialog box, select a timeframe to delete files based on their **Last Refreshed** date on the platform.

You can also remove files manually using the **Delete from this PC** command:

1. Right-click the files and select **Delete from this PC** in the 3DEXPERIENCE Files on This PC tab.
2. If the files contain an assembly or multibody parts, select one of these options:
  - **Delete only the selected files.** Removes the chosen files but keeps the referenced files intact.
  - **Delete the selected files and their references.** Removes the chosen files along with their references.

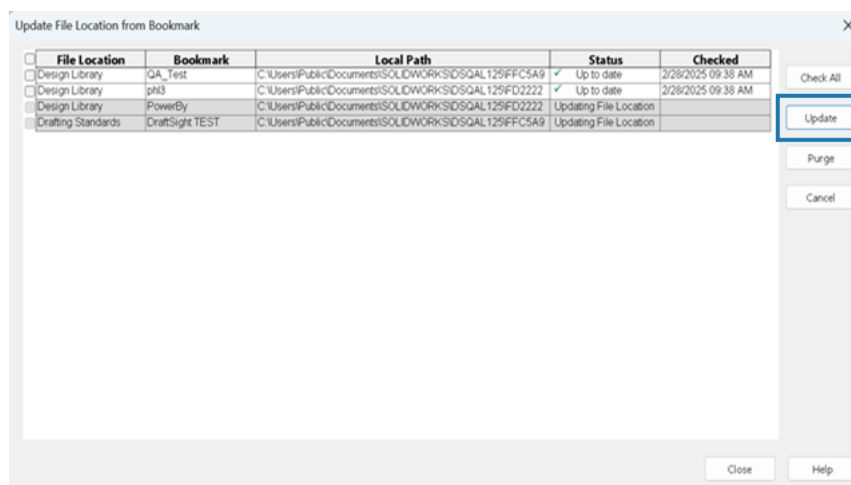
Deleting files removes them from your local cache but remain available on the **3DEXPERIENCE** platform. The 3DEXPERIENCE Files on This PC tab refreshes automatically.

Files cannot be deleted if:

- An assembly contains references that do not meet deletion conditions.
- Files have local modifications.
- Files are open in the current session.
- Files are locked by you.

A warning message will appear, if no files meet the criteria for deletion.

## Automatic Update of Bookmarked File Locations (2025 FD02)



SOLIDWORKS automatically updates bookmarked file locations to ensure that local content matches the latest information in the bookmarks on the **3DEXPERIENCE** platform.

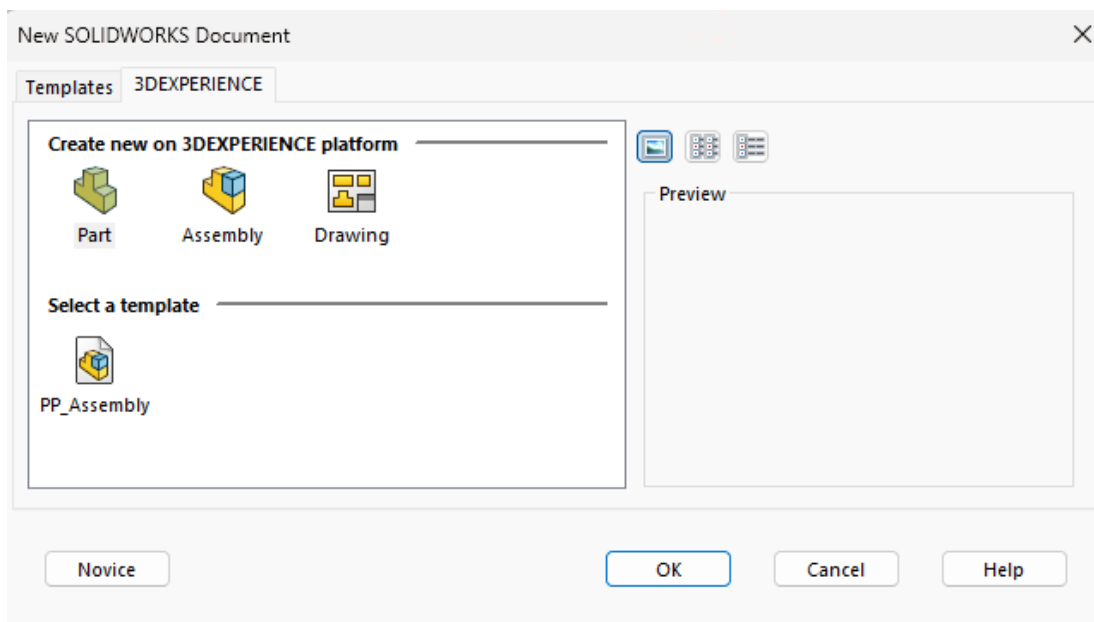
When you start SOLIDWORKS or switch from offline to online mode, SOLIDWORKS checks the mapped bookmarks for changes once per session. If the check finds modified bookmarks, SOLIDWORKS updates the local copy of the content. The update runs in the background, allowing you to continue working in SOLIDWORKS.

The app displays the updated status and time stamp of the selected files.

**Benefits:** Your workflow becomes simpler because the bookmarked file locations update automatically in the background, allowing you to continue working in SOLIDWORKS.

## SP1 and FD01


### Filling in Custom Property Values on File Creation (2025 FD01)



When you create a part, assembly, or drawing, the interface provides an easier way for you to fill in custom properties for files.



**Benefits:** This streamlines the workflow by filling in custom properties during file creation.

#### To fill in custom property values on file creation:

1. Click **New**  (Standard toolbar) or **File > New**.
2. In the New SOLIDWORKS Document dialog box, click **Advanced**.

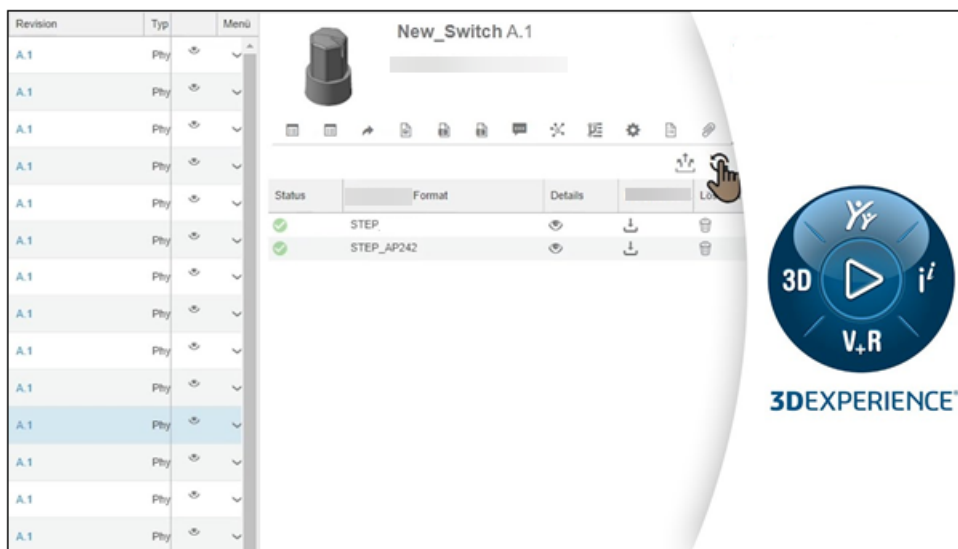
In **Tools > Options > System Options > Default Templates**, you can change the **Default Templates** to the **3DEXPERIENCE** template so that this workflow also applies to the **Novice** version of the dialog box.

3. On the 3DEXPERIENCE tab, under **Create new on 3DEXPERIENCE platform**, select **Part**, **Assembly**, or **Drawing**.

4. Click **OK**.
5. In the New Part/New Assembly/New Drawing dialog box, on the Properties  and Classifications  tabs, specify custom properties for the file.
6. Click **Create**.

An empty physical product is created on the **3DEXPERIENCE** platform.

## Saving Transient Components to the Platform (2025 FD01)



When opening non-SOLIDWORKS files, you can upload transient SLDPRT files to the platform as Derived Outputs (DOs).

**Benefits:** This update simplifies how you work with non-SOLIDWORKS data and improves performance during file opening workflows. This enhancement applies to various MCAD formats, including CATIA V5, NX, CREO, INVENTOR, and SOLIDEDGE.

A transient SLDPRT file is a temporary file that SOLIDWORKS generates when importing non-native CAD data. Instead of reimporting the original file each time, SOLIDWORKS saves the transient SLDPRT file to the platform. For future opens, SOLIDWORKS downloads and uses the saved SLDPRT DO directly, eliminating the need to reimport the file.

When you open a non-SOLIDWORKS file for the first time, SOLIDWORKS imports the data and uploads the transient SLDPRT file as a DO in the background. This upload allows other users or sessions to reuse the file without reimporting. For subsequent opens, SOLIDWORKS retrieves the SLDPRT DO from the platform, simplifying the process and saving time.

If you are using different versions of SOLIDWORKS on the same platform, older versions cannot reuse DOs created by newer versions. An error message will appear in these cases.

While you can edit the transient SLDPRT file, these edits will not update the original non-SOLIDWORKS file. The process for accessing current files from the local cache remains unchanged.

## Tracking Maturity Changes with Evaluated Attributes in SOLIDWORKS Drawings (2025 FD01)

	Property Name	Type	Value / Text Expression
1	Approval task [1]	Text	\$PLMPPR:"ea_releasedtask.1"
2	Approval task [2]	Text	\$PLMPPR:"ea_releasedtask.2"
3	Approved on [1]	Text	\$PLMPPR:"ea_releaseddate.1"
4	Approved on [2]	Text	\$PLMPPR:"ea_releaseddate.2"
5	Approver [1]	Text	\$PLMPPR:"ea_releasedby.1"
6	Approver [2]	Text	\$PLMPPR:"ea_releasedby.2"
7	Change Status Action Name	Text	\$PLMPPR:"ea_changestatusaction"
8	Created By	Text	\$PLMPPR:"ea_createdby"
9	Creation Date	Text	\$PLMPPR:"created" ...
10	Latest Maturity Change Actor [1]	Text	\$PLMPPR:"ea_changestatusby.1"
11	Latest Maturity Change Actor [2]	Text	\$PLMPPR:"ea_changestatusby.2"
12	Latest Maturity Change Date	Text	\$PLMPPR:"ea_changestatusdate"
13	Latest Maturity Change Date [1]	Text	\$PLMPPR:"ea_changestatusdate.1"
14	Latest Maturity Change Date [2]	Text	\$PLMPPR:"ea_changestatusdate.2"
15	Latest Maturity Change Task [1]	Text	\$PLMPPR:"ea_changestatustask.1"
16	Latest Maturity Change Task [2]	Text	\$PLMPPR:"ea_changestatustask.2"
17	Maturity State	Text	\$PLMPPR:"status"
18	Released on	Text	\$PLMPPR:"ea_releaseddate"

Evaluated attributes automatically track and display maturity changes for SOLIDWORKS drawings saved to the **3DEXPERIENCE** platform.

**Benefits:** Evaluated attributes make it easy to track maturity changes and see the history of a drawing without manual updates.

A typical use case starts with creating a route template with approval tasks for a designer and manufacturer. Each task is approved using the **Change Action** command in MySession, updating the maturity of the drawing. After all tasks are approved, the state of the drawing changes to **Released**.

The workflow for this feature is as follows:

1. Open a drawing file in SOLIDWORKS.
2. Add annotations that reference the following PLM properties:
  - `ea_changestatusaction`: The Change Action used to promote the drawing.
  - `ea_changestatusdate`: The date of the maturity changes.
  - `ea_changestatusby[i]`: The user who performed the maturity change.
  - `ea_changestatustask[i]`: The task used to execute the maturity change.
3. Save the drawing to the **3DEXPERIENCE** platform so that the attributes are registered.
4. Rebuild the drawing in SOLIDWORKS to ensure the annotations display correctly.
5. In MySession, use the **Change Action** or **Change Status** command to update the state of the drawing such as **In Work**, **Frozen**, **Released**, and **Obsolete**.
6. Open the drawing in 3DPlay or any supported web viewer to see the updated annotations and verify that the information is accurate.

Using evaluated attributes has these limitations:

- It only works for stand-alone annotations and does not support properties in tables or combined with others.

- Due to indexing, regular attributes may show delays, but `ea_` attributes update instantly.
- Empty attributes, like task or date fields, appear as a "-" in SOLIDWORKS.
- It supports only UDL and PDF formats, not DXF/DWG.
- Tasks for maturity transitions are evaluated only after the drawing is released or marked obsolete.


## Opening Drawings in Detailing Mode (2025 FD01)

You can open and save drawings from the **3DEXPERIENCE** platform in Detailing mode without loading the references.


**Benefits:** Detailing mode improves performance for opening and editing large assembly drawings.

There are two ways to open drawings in Detailing mode.

### To open drawings in Detailing mode using Open from 3DEXPERIENCE:

1. Click **File > Open > Open from 3DEXPERIENCE > 3DSearch**.
2. In the dialog box, select a drawing.
3. Under **Mode**, select **Detailing** .
4. Click **Open**.

### To open drawings in Detailing mode using MySession:

1. In MySession, click **Tools > Options > Open**.
2. In the dialog box, select **Choose the mode before opening files** and click **OK**.
3. In 3DSearch, search for a drawing.
4. Right-click the drawing and click **Open**.
5. Under **Mode**, select **Detailing** .
6. Click **Open**.

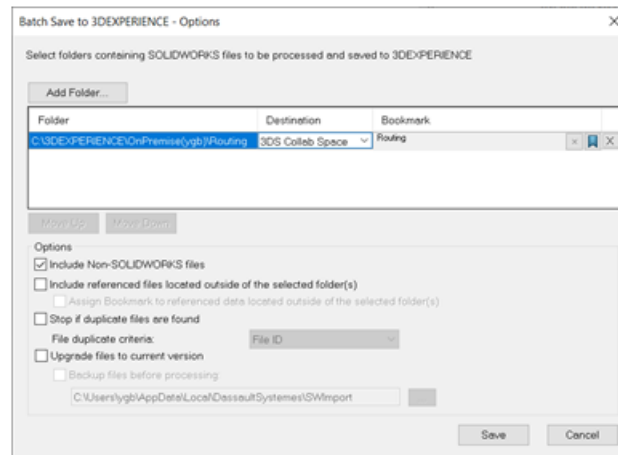
## Saving Drawings in Detailing Mode (2025 FD01)

You can save drawings to the **3DEXPERIENCE** platform in Detailing mode.

### To save drawings in Detailing mode:

1. Click **File > Save to 3DEXPERIENCE**.

## Batch Upload Non-SOLIDWORKS Files to the 3DEXPERIENCE Platform (2025 FD01)



You can use the Batch Save to 3DEXPERIENCE option to upload non-SOLIDWORKS files, such as .xml, .xls, .db, and more, directly to a selected bookmark on the **3DEXPERIENCE** platform.

This option organizes different file types into a folder structure, making the upload process simpler especially for extensive routing libraries. The Batch Save add-in keeps non-SOLIDWORKS files like RoutingLib.db and Components.xml are current.

#### To upload files using the Batch Save to 3DEXPERIENCE option in the Routing Library Manager:

1. In SOLIDWORKS, click **Tools > Add-ins** to enable the Routing add-in.
2. Open the **Routing Library Manager** from the Windows Start menu by clicking **SOLIDWORKS Tools > SOLIDWORKS Routing Library Manager**.
3. Navigate to the Routing File Locations and Settings tab and click **Batch Save to 3DEXPERIENCE**.

The Batch Saves to 3DEXPERIENCE Options dialog box opens. The **Include Non-SOLIDWORKS Files** option is selected by default.

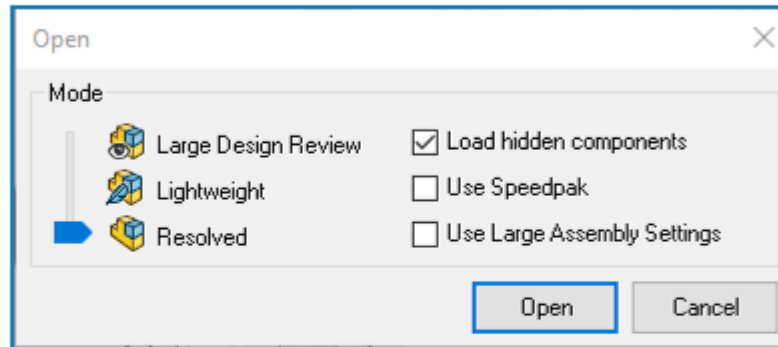
4. Select additional options in the dialog box.
5. Click **Add Folder** to select the folder containing the files. All files, including non-SOLIDWORKS files like .xml, .xls, and .db, are included for upload.
6. Choose the **Bookmark** for the upload.
7. Click **OK**.

#### Limitations:

- The Batch Save option uploads non-SOLIDWORKS files as separate documents that are not linked to SOLIDWORKS files..
- It does not detect file modifications and only works for first-time uploads.



## Improved Open Mode for Files Saved on the 3DEXPERIENCE Platform (2025 FD01)



Updates to file open modes when working with files saved on the **3DEXPERIENCE** platform offer more control and consistency.

**Benefits:** These updates give you more control over how files open with **3DEXPERIENCE** files in SOLIDWORKS.

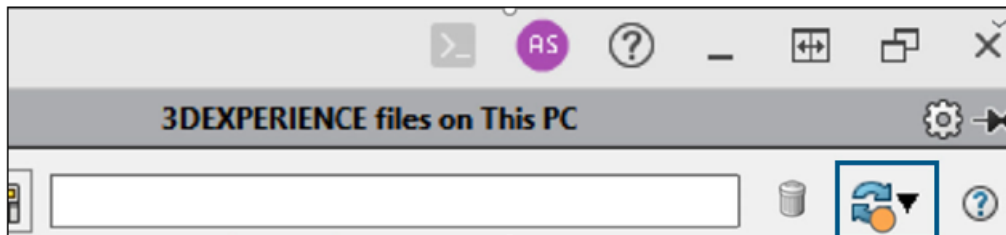
Updates to the 3DEXPERIENCE Files on This PC tab include:

- **Alt + Drag-and-Drop** Shortcut: Pressing **Alt** while dragging a file from the tab displays the Open Mode dialog box.
- **Locked Large Design Review Mode:** Files downloaded in Large Design Review mode always open in that mode.
- **Consistent Right-Click and Drag Behavior:** Right-clicking or dragging files follow your Open Mode dialog box settings.
- **Multi-File Selection:** Open Mode prioritizes options based on file types, such as assemblies over parts or drawings, when selecting multiple files.
- **Tooltip for Open Mode:** When dragging files from the tab, a tooltip with **Hold Alt for Open Dialog** appears.

Another improvement for 3DEXPERIENCE Search results includes:

- **Alt + Drag-and-Drop** Shortcut: Clicking **Alt** while dragging and dropping files displays the Open Mode dialog box before opening the file.

## Status and Refresh Enhancements for the 3DEXPERIENCE Files on this PC Tab (2025 FD01)



The 3DEXPERIENCE Files on This PC tab updates automatically to show the most current data.

**Benefits:** You no longer need to manually refresh the tab, making it easier to stay current.

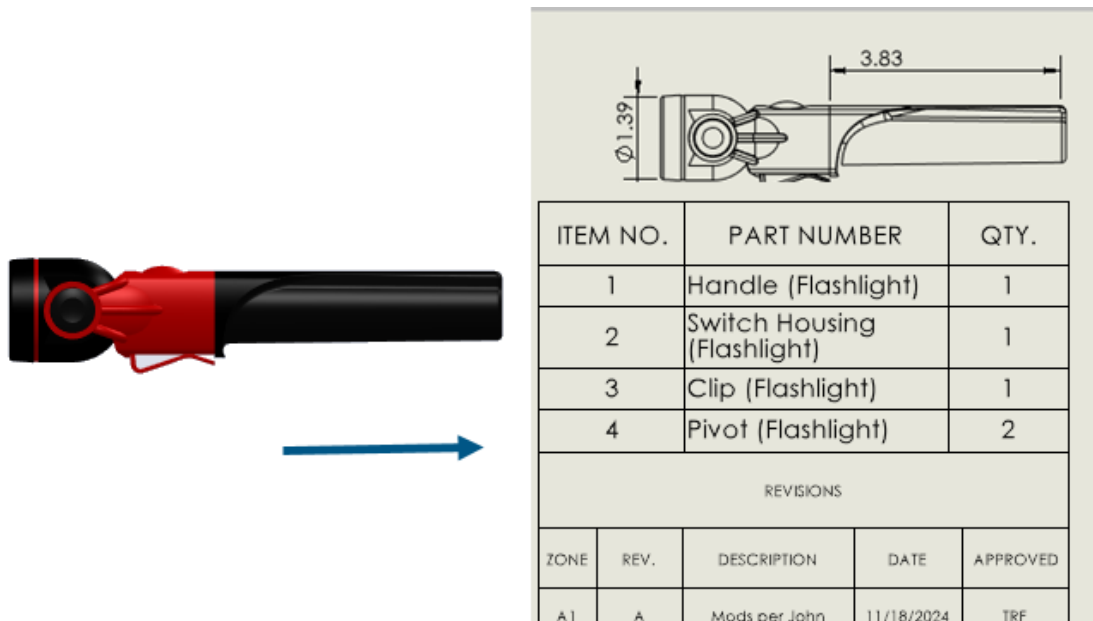
An orange status indicator on **Refresh** helps you monitor your files. It stays clear when your view is current but turns orange when you save new files, download files, or have missing files locally. This visual clue highlights when the tab needs your attention..

You can also filter top-level assemblies to show only the main nodes of assemblies in the cache. This option simplifies navigation and makes tracking changes easier, when combined with the status indicator.

Updated tooltips provide clear descriptions of the refresh options:

- **Refresh View:** Refreshes the entire view.
- **Refresh All from Server:** Updates lifecycle information for all files from the server.
- **Refresh Selected from Server:** Updates lifecycle information for selected files only.

## Auto-Generate Drawings (BETA) (2025 SP1)



**3DEXPERIENCE** users can auto generate drawings of parts and assemblies.

**Benefits:** Automatically generating drawings reduces errors and time spent on repetitive tasks.

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## Auto Generating Drawings (BETA)

**3DEXPERIENCE** users can automatically generate drawings of parts and assemblies.

**To auto generate drawings:**

- Do one of the following:
  - Click **File** > **Auto-Generate Drawing** (BETA).
  - In the FeatureManager design tree or graphics area, right-click a part, subassembly, or assembly, and click **Auto-Generate Drawing**(BETA).

- In the PropertyManager, specify options and click .

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## Tasks (Auto-Generate Drawings) (BETA) Tab



The Tasks (Auto-Generate Drawings) (BETA) tab lists the generated drawings and their progress.

The **Auto-Generate Drawings** (BETA) tool is available to **3DEXPERIENCE** users only.

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### To open this tab:

In a part or assembly, click **Tasks (Auto-generate drawings)** (BETA) from the Task Pane.

<b>Title</b>	Displays the name of the generated drawing.
<b>Status</b>	<p>Displays the status of the drawing generation.</p> <ul style="list-style-type: none"> <li> <b>Completed</b></li> <li> <b>Failed</b></li> </ul>
<b>Actions</b>	<ul style="list-style-type: none"> <li><b>Cancel.</b> (Available during drawing creation.) Cancels the auto-drawing generation for the selected item.</li> <li><b>Open.</b> (Available after completion of drawing.) Opens the selected drawing in Detailing mode.</li> <li><b>View Details.</b> (Available if drawing creation fails.) Opens a report to show why the auto-generated drawing failed.</li> <li><b>Queue Next.</b> Moves the task to the next in the queue. SOLIDWORKS does not cancel the current task in progress. If you click <b>Queue Next</b> on another task, SOLIDWORKS moves the task to the next in the queue.</li> <li>Right-click any row in the Tasks tab to:             <ul style="list-style-type: none"> <li><b>Clear.</b> Clears the selected row from the list.</li> <li><b>Clear All.</b> Clears all the rows from the Tasks tab except for the rows in progress. This includes rows where the status is <b>Completed</b> or <b>Failed</b>.</li> </ul> </li> </ul>

## MySession Behavior in Large Design Mode (2025 FD01)

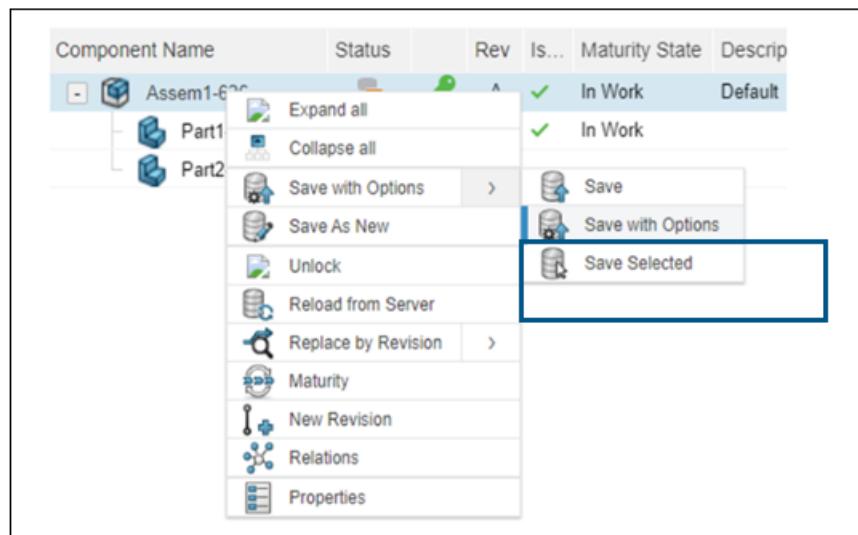
When you open data in **Large Design Review** (LDR) mode, MySession shows only one node for the opened assembly file. This node includes the same information as when the file is loaded in **Resolved** mode.

If the file has not yet been saved to the platform, it only shows SOLIDWORKS information. If the file is already saved to the platform, it displays both SOLIDWORKS and PLM information. In LDR mode, child nodes do not appear for the root assembly.

The following MySession commands are unavailable for this node. Trying to use them shows an error message:

- **Save as New**
- **Save Active Window As New**
- **Reload from Server**
- **Replace by Revision**
- **Replace by Latest Revision**
- **Update Revisions**

## Save Selected Files in MySession (2025 FD01)



In MySession, you can save individual parts, assemblies, or drawings to the **3DEXPERIENCE** platform without saving the entire model.

**Benefits:** This command lets you save only the necessary components while controlling what gets uploaded to the platform.

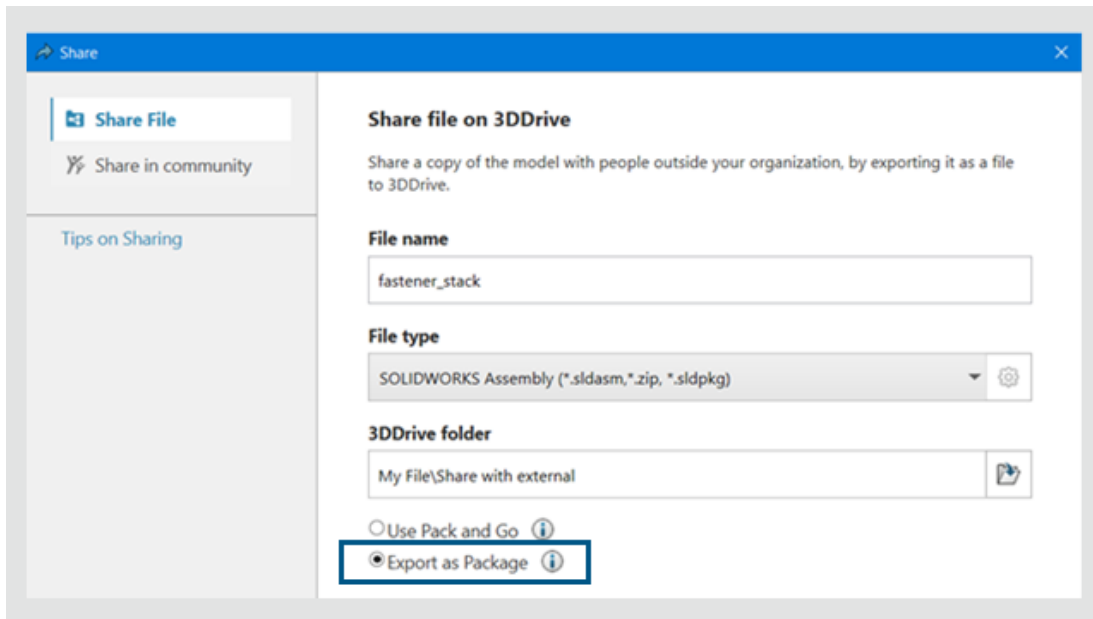
### To use Save Selected:

1. Open an assembly in SOLIDWORKS.
2. In MySession, right-click the component and select **Save Selected**.
3. In the Save dialog box, ensure that the component is selected.
4. Click **Save**.

Limitations include:

- **Saving Changes in Parts:** Changes to individual parts made at the assembly level are not saved unless you explicitly include those parts.
- **Top-Level Assemblies:** Use **Save with Options** when saving a new top-level assembly to handle graphical properties and flexible assemblies correctly.

## Sharing Files Using Export as Package (2025 FD01)



**3DEXPERIENCE** users can use the **Export as Package** option in the Share dialog box to share assemblies saved on the **3DEXPERIENCE** platform. You can share the package with external teams who can modify the files in SOLIDWORKS. You can then merge the returned files back to the platform.

**Benefits:** The **Export as Package** option gathers all referenced files that the **Pack and Go** option may not include, such as drawings that are not in the cache.

### To use the Export as Package option:

1. In SOLIDWORKS, open an assembly that is saved on the **3DEXPERIENCE** platform.
2. Click **File > Share**.
3. In the dialog box:
  - a. Click **Share File**.
  - b. Enter a **File name** and in **File type**, select **SOLIDWORKS Assembly**.
  - c. Click **Export as Package**.
  - d. Click **Continue**.

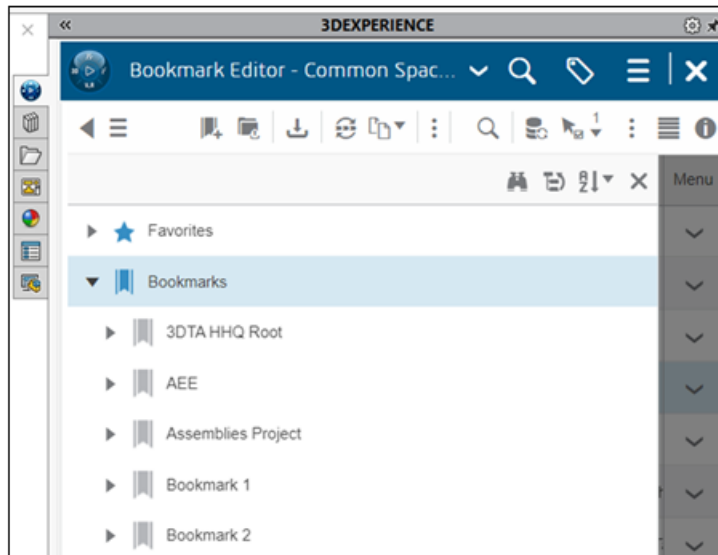
The **Export as Package** command opens in the 3DEXPERIENCE tab of the Task Pane.

For more information about this command, see [Exporting and Importing SOLIDWORKS Data](#).

4. Specify the options and click **Export**.

The software exports the package with the `.sldpkg` extension.

## Managing Bookmark Issues When Saving Data (2025 FD01)

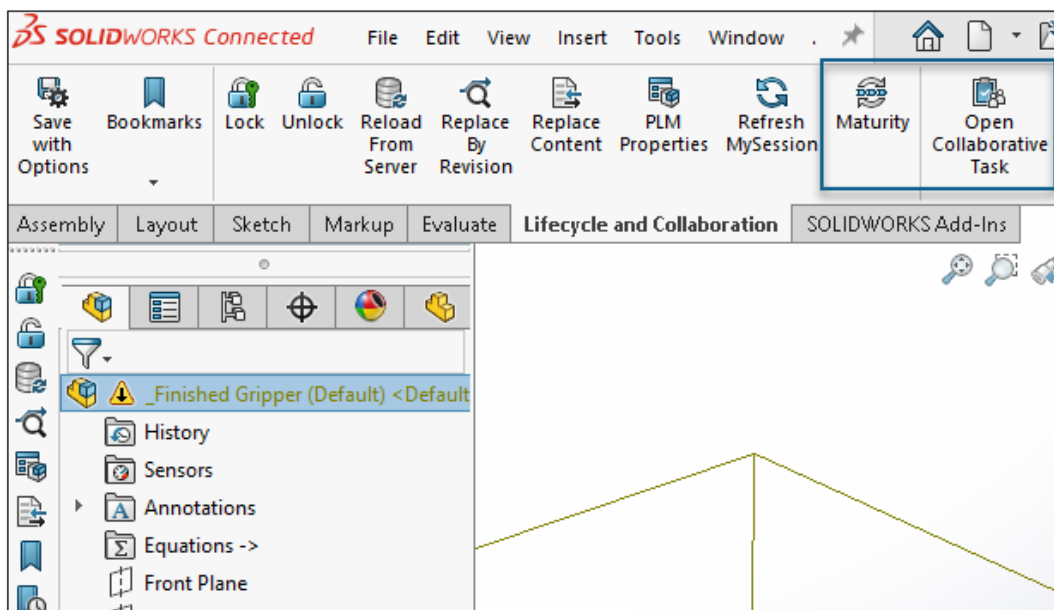


You can save data to the **3DEXPERIENCE** platform even if bookmarks are frozen, completed, archived, or deleted. If an assigned bookmark fails, a message indicates that the data is not bookmarked.

**Benefits:** Save operations run even when bookmarks are frozen, completed, archived, or deleted.

After saving, you can use the Bookmark Editor to manually resolve issues and assign bookmarks.



## Lifecycle and Collaboration Tab (2025 FD01)





You can use the **Open Collaborative Task** and **Maturity** tools on the Lifecycle and Collaboration tab.

The **Open Collaborative Task** tool opens collaborative tasks in the SOLIDWORKS Task Pane. The **Maturity** tool changes the maturity state of the selected file.

**To access the Open Collaborative Task tool:**

1. Do one of the following:
  - In the CommandManager, click **Open Collaborative Task** .
  - On the Lifecycle and Collaboration toolbar, click **Open Collaborative Task** .

**To access the Maturity tool:**



1. Do one of the following:
  - In the CommandManager, click **Maturity** .
  - On the Lifecycle and Collaboration toolbar, click **Maturity** .
  - Click **Tools** > **Lifecycle and Collaboration** > **Maturity**.

**Changing the Maturity State**

You can use the **Maturity** tool to change the maturity state of a selected file.

**To change the maturity state:**

In the FeatureManager design tree, select a file and do one of the following:

- In the CommandManager, click **Maturity** .
- On the Lifecycle and Collaboration toolbar, click **Maturity**.
- Click **Tools** > **Lifecycle and Collaboration** > **Maturity** .



The maturity states of selected files change.

**Opening Collaborative Tasks**

You can use the **Open Collaborative Task** tool to open collaborative tasks in the SOLIDWORKS Task Pane.

**To open collaborative tasks:**

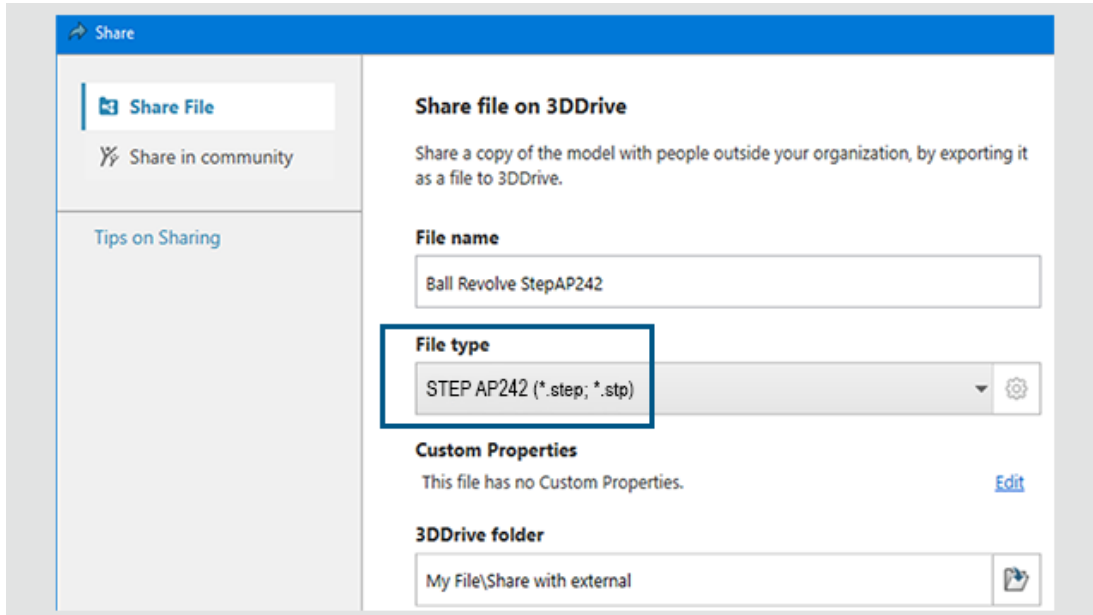
Do one of the following:

- In the CommandManager, click **Open Collaborative Task** .
- On the Lifecycle and Collaboration toolbar, click **Open Collaborative Task** .

Collaborative tasks appear in the SOLIDWORKS Task Pane.



## Sharing Models as the STEP242 File Type (2025 FD01)




When you enable the SOLIDWORKS MBD add-in, **3DEXPERIENCE** users can share parts or assemblies as the STEP242 file type to 3DDrive. All custom properties associated with the files appear in the Share dialog box under **Custom Properties**.

**Benefits:** The STEP242 file type is an update to the CAD-neutral STEP file standard and includes 3D Product and Manufacturing Information (PMI) in addition to the CAD data.

The SOLIDWORKS MBD add-in is not part of any role. You need a stand-alone license that you add during installation in the Install SOLIDWORKS Add-ins dialog box.

### To share models as the STEP242 file type to 3DDrive:

1. Click **Tools > Add-Ins**, select **SOLIDWORKS MBD**, and click **OK**.
2. Open a part or assembly and click **File > Share**.
3. On the Share File tab, under **Share file on 3DDrive**:
  - a. Specify the **File name**.
  - b. Under **File type**, select **STEP242 AP242 (\*.step;\*.stp)**.
  - c. **Optional:** To select from the available custom properties in the model, in the **Custom Properties** section, click **Edit**.

In the Publish to STEP242 on 3DDrive PropertyManager, select the custom properties to share in the model and click .

The Share with external dialog box opens. Go to step four.
  - d. Click **Upload**.

The Share with external dialog box opens.
4. Specify the sharing options and click **Share**.

The system notifies you that the file is uploaded to 3DDrive.

For more information, see [Sharing Files on 3DDrive](#).

## Working with Iterations (2025 FD01)

You can create iterations of SOLIDWORKS drawings, parts, or assemblies.

**Benefits:** You can access prior iterations of SOLIDWORKS files for recovery. This is convenient if you made a mistake and want to restore a file.

### Creating Iterations

You can create iterations of SOLIDWORKS parts, assemblies, or drawings.

#### To create iterations:

1. In a part, assembly, or drawing, click **File > Save to 3DEXPERIENCE**.
2. In the dialog box, select **Keep previous iteration**.
3. Click **Save**.

### Recovering Iterations

You can recover iterations of SOLIDWORKS parts, assemblies, or drawings.

#### To recover iterations:

1. On the MySession Lifecycle tab, click **Iteration**.
2. Select any iteration and click **Replace Content**.
3. Click **File > Save to 3DEXPERIENCE**.

## Linking 3DEXPERIENCE Revision Table Columns to Custom Attributes (2025 FD01)

The diagram illustrates the process of linking 3DEXPERIENCE revision table columns to custom attributes. It shows two tables. The top table, titled 'Revisions', has columns: Area, REV., DESCRIPTION, DATE, and APPROVED. The bottom table, titled 'Revision Table', has columns: ZONE, REV., DESCRIPTION, DATE, and APPROVED. A blue arrow points from the 'Area' column in the 'Revisions' table to the 'ZONE' column in the 'Revision Table', indicating the mapping.

Revisions				
Area	REV.	DESCRIPTION	DATE	APPROVED
B1	B	Mods per Joe	1/2/25	TRF

Revision Table				
ZONE	REV.	DESCRIPTION	DATE	APPROVED
B1	B	Mods per Joe	1/2/25	TRF

You can link **3DEXPERIENCE** revision table columns to custom attributes created on the platform.

**Benefits:** Linking attributes means you only need to input information in one place.

You can do the following in **3DEXPERIENCE** revision tables:

Functionality	Access
Edit titles	Double-click the title text.
Edit column names	Double-click the column text.

#### Creating Custom Attributes in 3DEXPERIENCE Revision Table Columns (2025 FD01)

Creating attributes lets you input information in one place.

##### To create custom attributes in 3DEXPERIENCE revision table columns:

1. Only users with administrator privileges can access and manage the Collaborative Spaces Control Center. SOLIDWORKS supports custom attributes in the 3DEXPERIENCE revision table that you create using Attribute Management under the Collaborative Spaces Configuration Center.


Click **Collaborative Spaces Control Center > Attributes Management > Drawing**.

2. (Optional) To add a new attribute, click +.
3. Enter a name for the attribute and click **OK**.
4. Click **Configuration Deployment**.
5. Under **Configuration and Server Utilities**, click **Upload Index Model with added or Removed Attributes** and **Reload Server Cache**.
6. (Optional) Click **CAD Collaboration > SOLIDWORKS**.
7. (Optional) Under **Drawing**, click + and select the attribute you created.

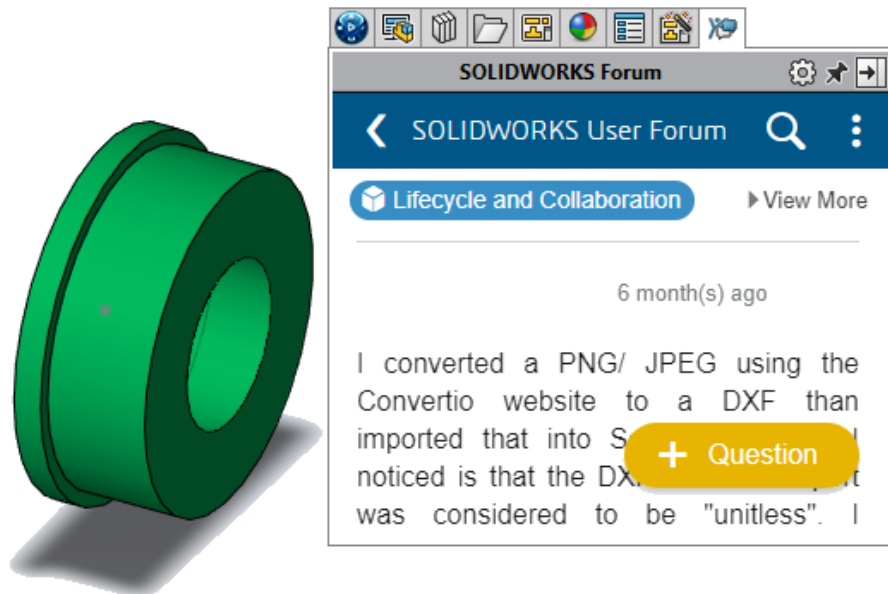
#### Linking Custom Attributes in 3DEXPERIENCE Revision Table Columns (2025 FD01)

Linking attributes lets you input information in one place.

##### To link custom attributes in 3DEXPERIENCE revision table columns:

1. In a **3DEXPERIENCE** revision table, click a column.
2. Under **Column Properties**, click **Custom Attributes**.
3. Click  and select the attribute.

## Accessing the SOLIDWORKS User Forum (2025 FD01)




You can access the SOLIDWORKS User Forum from the Task Pane.

**Benefits:** You can connect with the community of worldwide SOLIDWORKS experts without leaving SOLIDWORKS.

**To access the SOLIDWORKS User Forum:**

1. **Do one of the following:**

- From the title bar, click **Help** (?) > **User Forum**.
- Click the User Forum tab .

## Using Reload (2025 FD01)

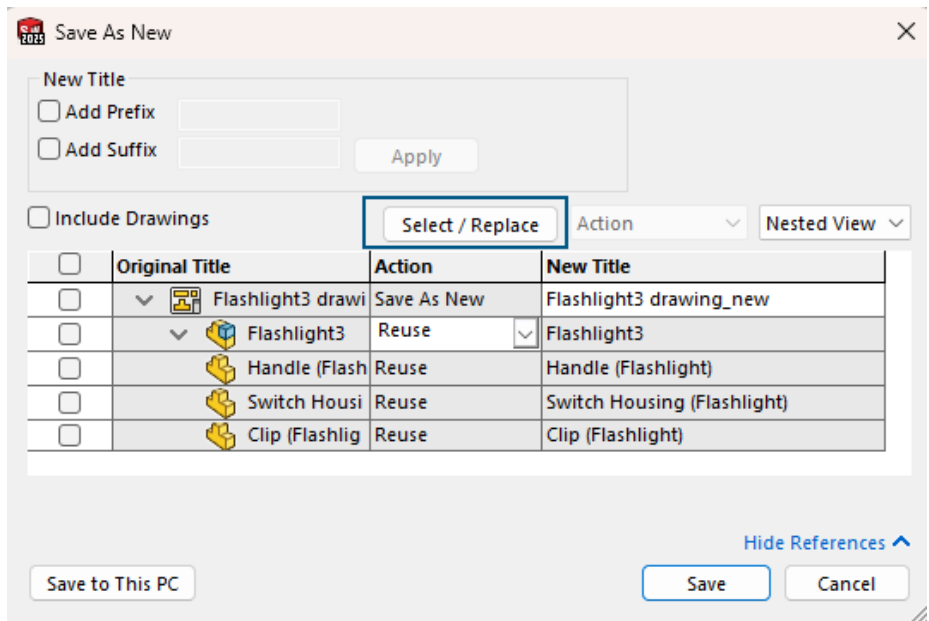
You can reload parts, assemblies, and drawings files in SOLIDWORKS Connected.

**Benefits:** You can undo the changes since the last **Save** operation.

**To reload a part, assembly, or drawing:**

1. In a part, assembly, or drawing, click **File** > **Reload**.

## Save As New Dialog Box (2025 FD01)



In the Save As New dialog box, you can rename file titles and multiselect in one step.

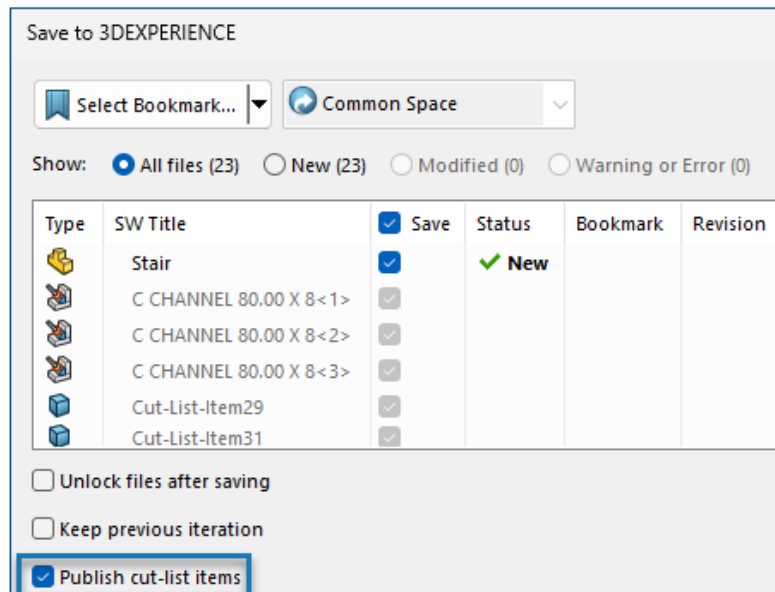
**Benefits:** The dialog box provides a simplified way to rename multiple files at once.

The Save As New dialog box supports a part within a part. It contains Show References to expand the interface. Previously, **Save As New** did not support the integrated part.

You can change the **Action** from **Reuse** to **Save As New** for referenced parts.

Select/Replace Functionality	Description
<b>Search original title for</b>	Searches for file titles that match the text you enter in the original title for the selected rows.
<b>Replace text with</b>	Replaces the file titles with the text that you enter for the selected rows.
<b>Select</b>	Selects the rows with matching values in the new title.
<b>Replace</b>	Replaces the value in the new title for the selected rows.

## Publishing Cut List Items on the 3DEXPERIENCE Platform (2025 SP1)



You can publish cut list items of a weldment part on the **3DEXPERIENCE** platform.

To publish the cut list items, save the SOLIDWORKS part as a weldment part to the **3DEXPERIENCE** platform. The side panel displays the extension of the weldment part as SW Weldment Part.

Prerequisites to save the SOLIDWORKS part as a weldment part:

- You must not have already saved the part on the **3DEXPERIENCE** platform.
- The part must contain a weldment feature.
- The part must be flagged as a Single Physical Product.

Prerequisites to publish cut list items on the **3DEXPERIENCE** platform:

- The part must be a weldment part.
- The cut list must be up to date.
- The cut list item property must have the CutlistID.

#### To publish cut list items on the 3DEXPERIENCE platform:

1. With a weldment part open, click **Options** (Standard toolbar), select the Document Properties tab, and then select **Weldments**.
2. In the Document Properties - Weldments dialog box, under **Cut list IDs**, select **Generate Cut list IDs** and click **OK**.
3. In the **3DEXPERIENCE Task Pane**, right-click the part and click **Save**.
4. In the Save to 3DEXPERIENCE dialog box, select **Publish cut-list items** and click **Save**.

MySession displays the cut list items of the weldment part. The side panel displays the properties of the cut list items.

Administrators can define custom PLM attributes and mapping between CAD items and PLM items for saving attributes on the **3DEXPERIENCE** platform.

## Accepting or Rejecting Parent-Child Relationships in IDX Files (2025 SP1)

☒ Open all ProStep files in folder automatically

☒ Sync with ECAD automatically on build

☐ Use email-based communication:

Default recipient email addresses:

☒ Animate change in preview image on tree selection

☐ Reverse rotation direction of components on the underside of the board

☒ Check for changes made in SOLIDWORKS before applying changes from ECAD

☒ Use GMT style date in IDX communication

☐ Use parent-child association in IDX communication

You can manage and accept or reject changes in parent-child associations, whether updates come from ECAD or MCAD.

CircuitWorks now supports parent-child associations between components and other board items, such as keep-in, keep-out, plated holes, and non-plated holes, when interacting with IDX3.0 files. You can accept or reject changes to these items from either ECAD or MCAD.

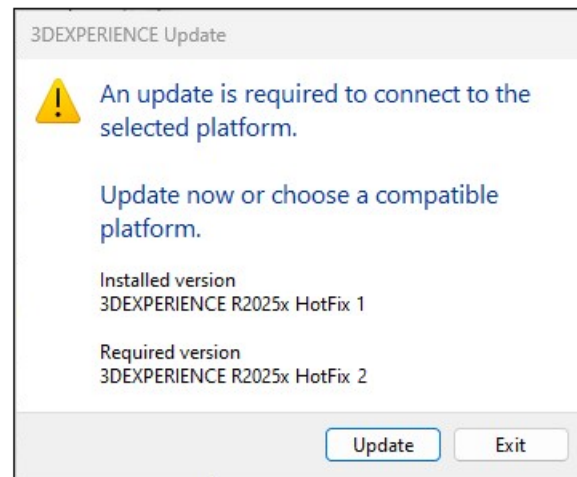
### Benefits:

- Accept or reject all associated changes in one action, regardless of whether the updates originate from ECAD or MCAD.
- When modifying parent components in MCAD, all related child items are updated automatically during the export to CircuitWorks.

To use this feature, do the following:

1. Click **Tools > CircuitWorks > CircuitWorks Options**.
2. Select **ProStep EDMD** and select **Use parent-child association in IDX communication**.

## Improved Update Notifications for Connected Apps (2025 SP1)



When you launch SOLIDWORKS Connected, Visualize Connected, or DraftSight Connected from a desktop shortcut, you can update the app directly from the message if an update is available or required.

The platform applies the new behavior to any major or minor updates after installing 3DEXPERIENCE 2025x FD01.

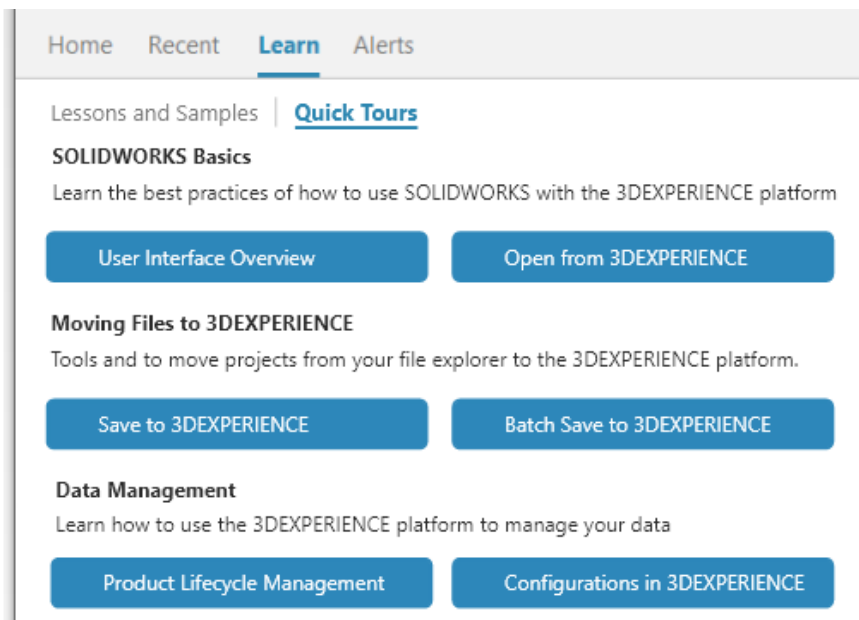
Previously, you had to navigate to the app in the Compass using a browser as a separate step.

**Benefits:** This improvement simplifies the update process and removes the need to switch between tools, making it faster to keep your apps up to date.



## SP0 and GA

### Quick Tours



**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 **3DEXPERIENCE** apps to help you quickly understand basic functionality and concepts. For information on best practices, see [SolidPractices](#).

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

To start a Quick Tour, click a tour, 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.

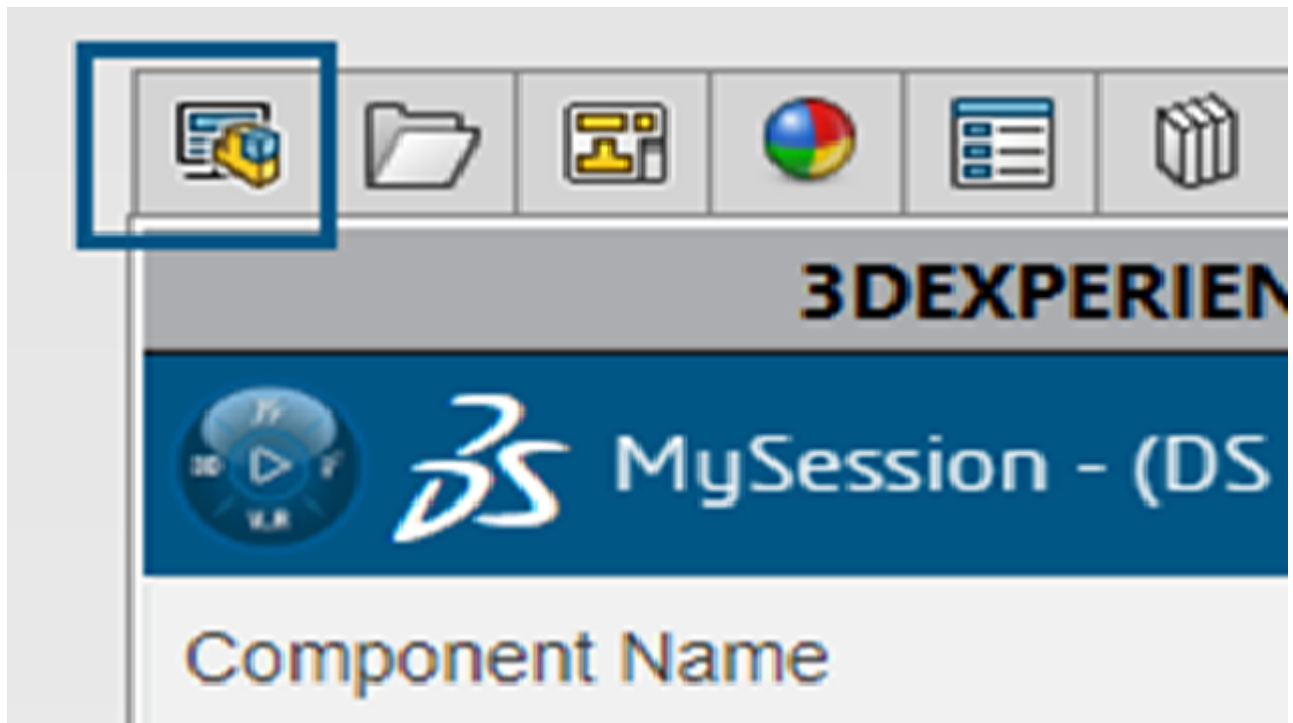
### Removal of Option to Generate 3D Format

The **Compute 3D format for all configurations** option is removed.


**Benefits:** You can continue working in SOLIDWORKS while the output is getting generated.

The option was added on the **Settings Page** of **Collaborative Spaces Configuration Center** > **CAD Collaboration** > **SOLIDWORKS**. The CGRs are now generated using the Conversion Service for cloud environment and Derived Format Converter for on-premises environment.

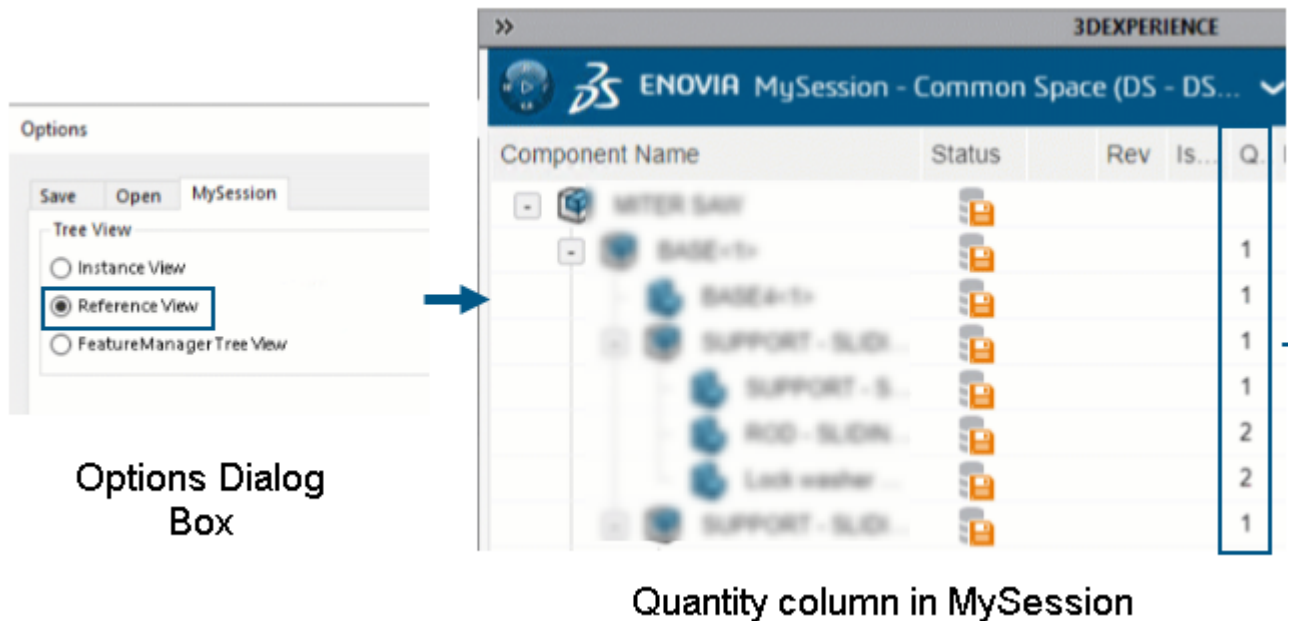
## Task Pane



Enhancements to the user interface help improve productivity.

In Design with SOLIDWORKS® and SOLIDWORKS Connected, the Task Pane shows **3DEXPERIENCE** Files on this PC as the second tab. When you turn off the **3DEXPERIENCE**  tab, **3DEXPERIENCE** Files on this PC is the first tab. In earlier releases, **3DEXPERIENCE** Files on this PC was the last tab.

## Visibility of Quantity Column



The **Quantity** column in MySession is visible or hidden based on the tree view option selected in the **Options** dialog box.

**Benefits:** You get the flexibility to show or hide the **Quantity** column.

The **Quantity** column displays the number of instances associated with an object. The values displayed are based on the selected **Tree View** type in the **Options** dialog box. The column is visible when you select **Reference View** or **FeatureManager Tree View** option.

## Licensing Support for SOLIDWORKS CAM, SOLIDWORKS Inspection, and SOLIDWORKS MBD Add-Ins

If you own licenses for SOLIDWORKS CAM, SOLIDWORKS Inspection, and SOLIDWORKS MBD, 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 an add-in and enter your serial number. If you are using a network license, you must specify the address (port@server) of your SolidNetWork (SNL) License server.

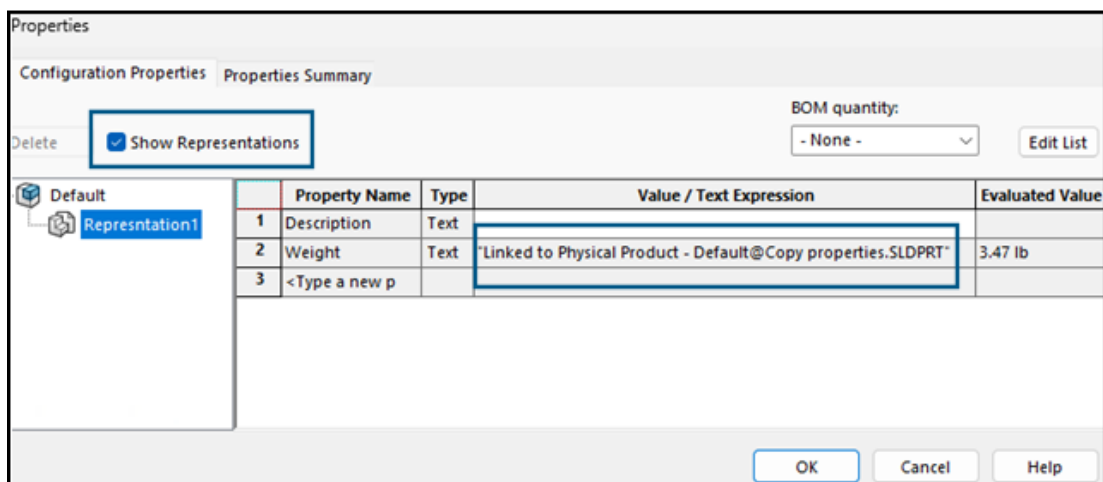
Once you install the add-in:

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

For SOLIDWORKS Inspection specifically, when you install SOLIDWORKS Inspection with SOLIDWORKS Connected, the add-in and stand-alone app are installed and updated. The

stand-alone app offers the same features as the SOLIDWORKS Installation Manager version. You can start the stand-alone app from the desktop shortcut or the Windows **Start** menu, not from the **3DEXPERIENCE** Compass. The stand-alone app also supports the same activation methods and SolidNetWork (SNL) licensing.

## Linking Configuration Properties of Representations to Physical Products



SOLIDWORKS links the configuration properties of representations to their physical products.

You can overwrite the values of representations that are linked from the physical products. **Show Representations** lets you display the representations of physical products in the left panel.

For linking between physical products and representations of legacy files that are compatible with the **3DEXPERIENCE** platform and saved:

1. In the FeatureManager® design tree, right-click the file.
2. Select **Link properties in representations**.

# 3

## Installation

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This chapter includes the following topics:

- **Convert SolidNetWork License Server to 64-Bit**
- **Installing the SOLIDWORKS Manage Web API**

### Convert SolidNetWork License Server to 64-Bit

The SOLIDWORKS® SolidNetWork License Manager 2025 installs as a 64-bit application. This change does not affect functionality or user experience.

### Installing the SOLIDWORKS Manage Web API

You can install the Manage Web API in the SOLIDWORKS PDM InstallShield Wizard. During the installation, you can either use the default port or specify another value for the Http port.

In addition, in the SOLIDWORKS Installation Manager, you can install the Manage Web API on the SOLIDWORKS Manage Server page and specify the Http port there as well.

# 4

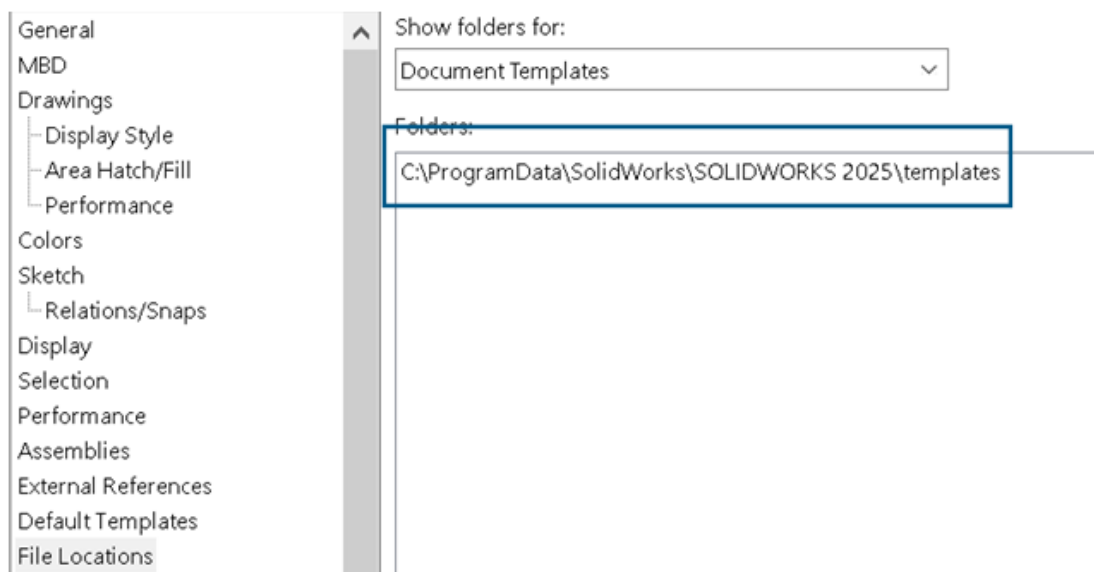
## Administration

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This chapter includes the following topics:

- **Inheriting Default File Locations When Upgrading to SOLIDWORKS 2025**
- **SOLIDWORKS Login Manager**

### Inheriting Default File Locations When Upgrading to SOLIDWORKS 2025



The logic for inheriting file locations from previous installations has improved. Previously, you had to modify or reset file locations when upgrading because of default file locations from earlier installations.

Default file locations now follow this logic:

- If you kept the default file location in an earlier installation, SOLIDWORKS® 2025 creates and uses a new default file location when you first run the software.
- Any new sheet formats and document templates added in the previous default location are integrated into the 2025 default file locations. The integration includes any new files referenced in `ProgramData\SOLIDWORKS\SOLIDWORKS version`.

There is no change if you customized file locations to custom paths. SOLIDWORKS 2025 continues to inherit custom paths from earlier installations. Custom paths exist outside of `ProgramData\SOLIDWORKS` or the SOLIDWORKS installation folders.

The data in the SOLIDWORKS installation directory is updated only if SOLIDWORKS is installed in the Windows Program Files folder. If SOLIDWORKS is installed outside this folder, file locations inherit custom paths.

## SOLIDWORKS Login Manager

The SOLIDWORKS Login Manager, installed by the SOLIDWORKS Installation Manager, allows login to the **3DEXPERIENCE** Marketplace and **3DEXPERIENCE** apps.

When installing an administrative image using the command line or through Microsoft Active Directory, you must include the SOLIDWORKS Login Manager file in the image. For example: `administrative_image_directory\swloginmgr\SOLIDWORKS Login Manager.msi`.

# 5

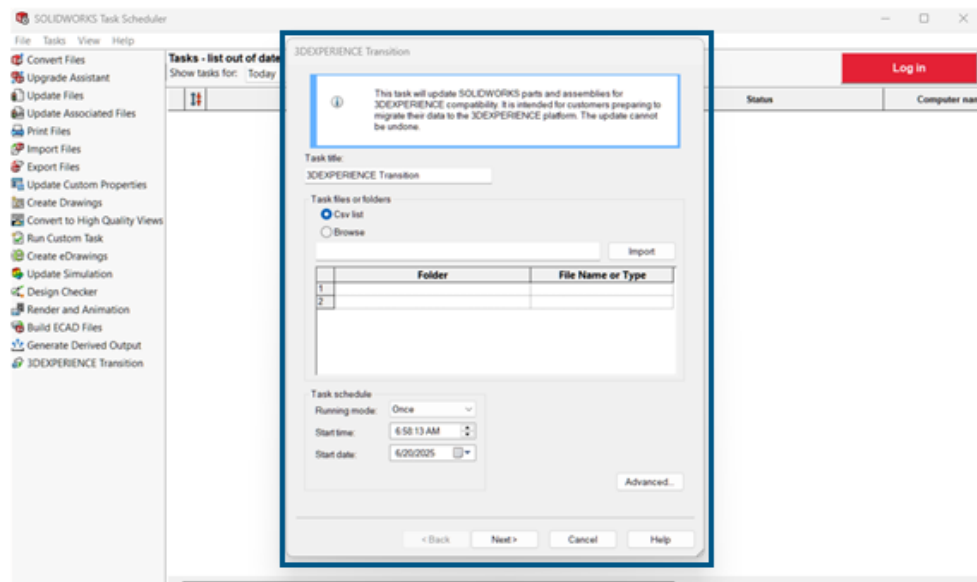
## SOLIDWORKS Fundamentals

---

This chapter includes the following topics:

- **3DEXPERIENCE Transition Task in SOLIDWORKS Task Scheduler**
- **Generating STEP Derived Objects for SOLIDWORKS Assemblies Using SOLIDWORKS Task Scheduler (2025 FD02)**
- **Performance in Multibody Parts (2025 SP2)**
- **Renaming Notes, DimXpert and Feature Dimensions in Annotations View under FeatureManager Design Tree (2025 SP2)**
- **Sharing Files on 3DDrive and 3DSwym (2025 SP1)**
- **Changes to System Options and Document Properties**
- **Application Programming Interface**
- **Specifying a Z-Up Template**
- **Saving SOLIDWORKS Inspection Files Using Bookmarks**

### 3DEXPERIENCE Transition Task in SOLIDWORKS Task Scheduler



The **3DEXPERIENCE** Transition task lets you update SOLIDWORKS files for compatibility with the **3DEXPERIENCE** platform. The **3DEXPERIENCE** Transition task works the same



as the **3DEXPERIENCE** Compatibility task, but it can use a `.csv` file to select content from your computer and run macros.

**Benefits:** You can save time using `.csv` files to add content to the task.

With the **3DEXPERIENCE** Transition task, you can:

- Upgrade files without enabling **3DEXPERIENCE** compatibility by saving them in a current version.
- Upgrade custom properties.
- Add rebuild marks.
- Add display data marks.

## Creating a 3DEXPERIENCE Transition Task

**To create a 3DEXPERIENCE Transition task:**

1. In SOLIDWORKS Task Scheduler, click **3DEXPERIENCE Transition**.
2. Under **Task title**, create a name for your task.
3. Under **Task files or folders**, select the content you want to update by doing one of the following:
  - Browse for a file or folder to add to **Task Files or Folders**.
  - Import a `.csv` file that specifies the content to add to **Task Files or Folders**.

The format of the `.csv` file is *path, filename*. For example to add `clamp.sldprt` and `bracket.sldprt`, write:

- "C:\Users\Public\Documents\SOLIDWORKS\SOLIDWORKS 2025\samples\tutorial\assemblymates","clamp.sldprt"
- "C:\Users\Public\Documents\SOLIDWORKS\SOLIDWORKS 2025\samples\tutorial\assemblymates","bracket.sldprt"

4. Run the task immediately or schedule the task (see [Scheduling the Task](#) on page 81).
5. Click **Next**.
6. In the Options dialog box, specify options:

Option	Description
<b>Configuration option</b>	<p>Saves only the active configuration or activates all configurations before saving.</p> <div> <p>Activating all configurations before saving can add significant time to the task.</p> </div>
<b>3DEXPERIENCE Compatibility</b>	<p>Updates SOLIDWORKS content for compatibility with the <b>3DEXPERIENCE</b> platform. See <a href="#">3DEXPERIENCE</a></p>

Option	Description
	<b>Compatibility and 3DEXPERIENCE Integration Options.</b>
<b>File Upgrade Settings</b>	<ul style="list-style-type: none"> <li>Upgrades custom properties.</li> <li>Adds rebuild mark to all configurations.</li> <li>Adds display data mark to all configurations.</li> </ul> <div> <b>Add display data mark to all configurations</b> is unavailable if you selected <b>3DEXPERIENCE Compatibility</b>. </div>
<b>Backup Files</b>	Specifies the location to back up the updated files.

- To run a macro, see [Running a Macro with the 3DEXPERIENCE Transition Task](#) on page 81.
- Click **Finish**.

### Scheduling the Task

#### To schedule the task:

- Under **Task Schedule**, set:

Option	Description
<b>Running mode</b>	How often the task runs. Select <b>Once</b> , <b>Daily</b> , <b>Weekly</b> , or <b>Monthly</b> .
<b>Start time</b>	What time the task starts.
<b>Start date</b>	What date the task starts.

- Click **Options** to specify backup locations.
- Click **Advanced** to change the working folder, time-out values, and other options.
- Click **Finish**.

The task and its title, scheduled time, scheduled date, and status appear in the Tasks panel. The status of the task is **Scheduled**.

### Running a Macro with the 3DEXPERIENCE Transition Task

#### To run a macro with the 3DEXPERIENCE Transition task:

1. In the **3DEXPERIENCE** Transition task, select the files you want to run the macro on. See [Creating a 3DEXPERIENCE Transition Task](#) on page 80.
  - a. Click **Next**.
2. In the Options dialog box, under **Custom Actions**, select **Run macro:**.
3. Browse for a SOLIDWORKS macro (.swp).
4. Click **Finish**.

The macro appears in the Task Scheduler with the title you set for the task.

## Sample SOLIDWORKS Macro

To test this functionality, you can paste the following text into a SOLIDWORKS macro (.swp).

This sample macro adds a property named "Hello" with a value of "Hello World" to any part, assembly, or drawing in the list of task files.

- For parts and assemblies, it adds a configuration-specific property to the active configurations.
- For drawings, it adds a custom property, because drawings do not contain configurations.

```
Dim swApp As SldWorks.SldWorks
Dim swModel As SldWorks.ModelDoc2
Dim config As SldWorks.Configuration
Dim cusPropMgr As SldWorks.CustomPropertyManager
Dim lRetVal As Long
Dim boolstatus As Boolean
Dim longstatus As Long, longwarnings As Long

Sub main()

  Set swApp = Application.SldWorks
  Set swModel = swApp.ActiveDoc

  If swModel Is Nothing Then
    ' If no model is currently loaded, then exit
    Exit Sub
  End If
  If (swModel.GetType <> swDocDRAWING) Then

    ' Add a Configuration Property named "Hello" to the active
    configuration for a Part or Assembly

    Set config = swModel.GetActiveConfiguration
    Set cusPropMgr = config.CustomPropertyManager

    lRetVal = cusPropMgr.Add3("Hello",
    swCustomInfoType_e.swCustomInfoText, "Hello World",
    swCustomPropertyAddOption_e.swCustomPropertyDeleteAndAdd)

  Else

    ' Add a Property named "Hello" for a Drawing
```

```
Set cusPropMgr = swModel.Extension.CustomPropertyManager("")
lRetVal = cusPropMgr.Add3("Hello",
swCustomInfoType_e.swCustomInfoText, "Hello World",
swCustomPropertyAddOption_e.swCustomPropertyDeleteAndAdd)

End If

End Sub
```

## Generating STEP Derived Objects for SOLIDWORKS Assemblies Using SOLIDWORKS Task Scheduler (2025 FD02)

**3DEXPERIENCE** users can use the STEP format for assemblies in a Generate Derived Output task.

**Benefits:** You can share the derived output from an assembly without having to assign a CAD license to users of other departments.


You can use the Generate Derived Output task to include STEP AP203 or AP214 Derived Objects attached to SOLIDWORKS assemblies.

The STEP format is not available for the Design with SOLIDWORKS app in on-premise installations.

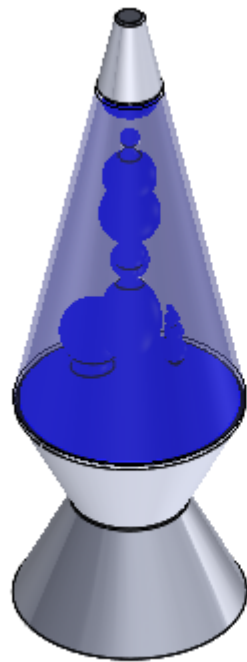
The Generate Derived Output task requires that you enter your **3DEXPERIENCE** platform password when creating the task. This allows the task to run SOLIDWORKS on your behalf at a future date and time. For example, you can set up a task to run nightly and automatically generate Derived Outputs for any assemblies or drawings added each day that match your search.

Previously, you could run the task only once and within the same day.

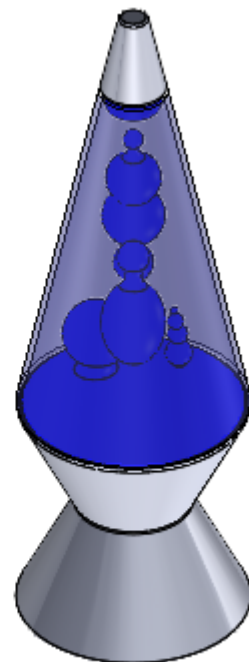
### To generate STEP derived objects for SOLIDWORKS assemblies:

1. In SOLIDWORKS, click **Tools > SOLIDWORKS Applications > SOLIDWORKS Task Scheduler**.
2. Click **Generate Derived Output**  on the sidebar, or click **Tasks > Generate Derived Output**.
3. In **Task title**, enter a new title for the task or leave the default.
4. Select one of the following STEP formats for **Derived Output Format**.
  - **STEP AP203**
  - **STEP AP214**
5. Select a **Collaborative Space**.
6. For **Maturity**, select **Released only** or **Frozen and Released**.
7. For **Owner**, select **All Content** or **My Content** from the collaborative space.
8. Enter your **3DEXPERIENCE** platform password in the **Password** field.

## Performance in Multibody Parts (2025 SP2)



Option On



Option Off

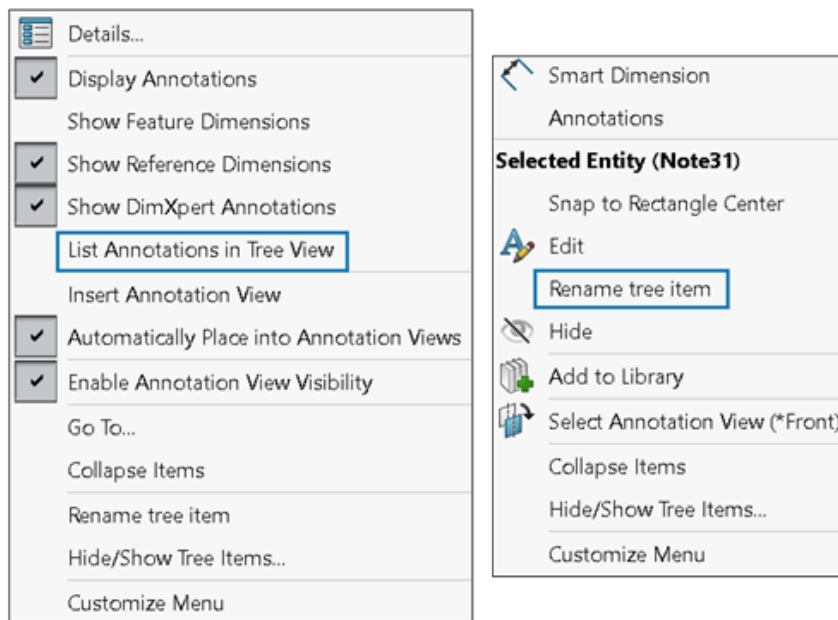
You can turn the display of silhouette edges off to improve performance in large multibody parts.

You can specify a threshold for the number of bodies for the part to be considered large. When you open a part whose number of bodies exceeds this threshold, SOLIDWORKS automatically turns the display of silhouette edges off.

### To turn the display of silhouette edges off:

1. Click **Tools > Options > System Options > Performance**.
2. Select **Do not display silhouette edges in parts when the number of bodies exceeds**.
3. Specify a value for the minimum number of bodies.
4. Click **OK**.
5. Save the model, close the model, and then reopen it for the option to take effect.

## Renaming Notes, DimXpert and Feature Dimensions in Annotations View under FeatureManager Design Tree (2025 SP2)



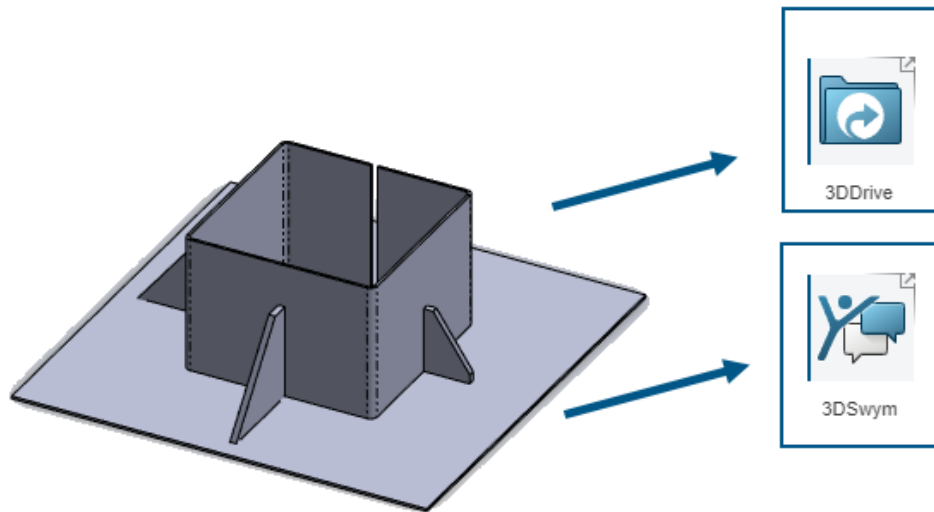
You can rename notes and dimensions in annotations with specific titles instead of generic names like *note1* and *note2*.

To rename the notes and dimensions in annotations:

1. In the FeatureManager design tree, right-click **Annotations** and select **List Annotations in Tree View**.
2. Do one of the following:
  - Right-click a note or dimension and select **Rename tree item**
  - Select a note or dimension and press F2.
3. Type the name and then click the graphics area.

The name can include letters, numbers, and special characters.

## Sharing Files on 3DDrive and 3DSwym (2025 SP1)



You can use the **Share** tool to share SOLIDWORKS files on 3DDrive and 3DSwym without installing the Design with SOLIDWORKS app.

3DDrive and 3DSwym let you securely share data with your team directly from SOLIDWORKS.

### To share files on 3DDrive and 3DSwym:

1. In a SOLIDWORKS document, click **File > Share**.
2. Specify the app.
  - To share on 3DDrive, select **Share File**.
  - To share on 3DSwym, select **Share in Community**.
3. If you did not sign in, click **Log In** and enter your **3DEXPERIENCE** credentials to access the app.

If you experience issues with accessing 3DSwym or 3DDrive, see [Steps to activate your 3DEXPERIENCE platform](#).

4. In the app, complete the required fields and click **Upload** for 3DDrive or **Publish** for 3DSwym.

## Changes to System Options and Document Properties

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

### System Options

Option	Description	Access
<b>Use AI fastener recognition to create SmartMates when inserting components</b>	(2025 FD03) Turns on automatic fastener recognition when you insert components into assemblies. SOLIDWORKS recognizes components that appear as nuts, bolts, or washers to automatically add mates to components.	<b>Assemblies</b>
<b>Override document level tree display</b>	(2025 SP3) Specifies options in the Component Name and Description dialog box at the system level.  When selected, you can click <b>Component Name and Description</b> to open the Component name and Description dialog box.	<b>FeatureManager</b>
<b>Scene, Animations, and Compression</b>	(2025 SP2) The export options for GLTF and GLB extended reality files are moved from the XR Exporter Settings dialog box to System Options. In <b>File Format</b> , select <b>GLTF/GLB</b> and specify the options.	<b>Export</b>
<b>Auto-resolve lightweight components upon expansion in FeatureManager tree</b>	(2025 SP2) Resolves lightweight components when you expand the components in the FeatureManager design tree.	<b>FeatureManager</b>
<b>Recognized mesh face Unrecognized mesh face</b>	Specifies the colors shown for the mesh faces when you use the <b>Insert &gt; Mesh &gt; Segment Imported Mesh Body</b> or <b>Convert Mesh to Standard</b> tool. See <b>Colors &gt; Color scheme settings</b> .	<b>Colors</b>
<b>Use Property Set mapping file</b>	Maps custom properties to IFC™ property sets. See <b>Export &gt; File Format: IFC &gt; Output as</b> .	<b>Export</b>



Option	Description	Access
<b>File Locations</b>	The logic for inheriting file locations from previous installations has improved. See <a href="#">Inheriting Default File Locations When Upgrading to SOLIDWORKS 2025</a> on page 77	<b>Installation</b>
<b>Zoom to fit on open</b>	When you open a drawing, you have the option to have it automatically zoom to fit your graphics area.	<b>Drawings</b>

## Document Properties

Option	Description	Access
<b>Automatically add Flange Length dimension to flange profiles</b>	SOLIDWORKS® automatically adds length dimensions to all edge flange profiles, where the sketch dimension (not the feature dimension) controls the flange length.	<b>Sheet Metal</b>
<b>Surface symbol standard</b>	Select a standard: <ul style="list-style-type: none"> <li>• 21920-1</li> <li>• 1302 (1992)</li> <li>• 1302 (2002)</li> </ul>	<b>Surface Finishes</b>
<b>Tolerance type</b>	Select a tolerance: <ul style="list-style-type: none"> <li>• None</li> <li>• Bilateral</li> <li>• Limit</li> <li>• Symmetric</li> <li>• MIN</li> <li>• MAX</li> <li>• Fit</li> <li>• Fit with tolerance</li> <li>• Fit (tolerance only)</li> </ul>	<b>Chamfer Dimension Tolerance</b>

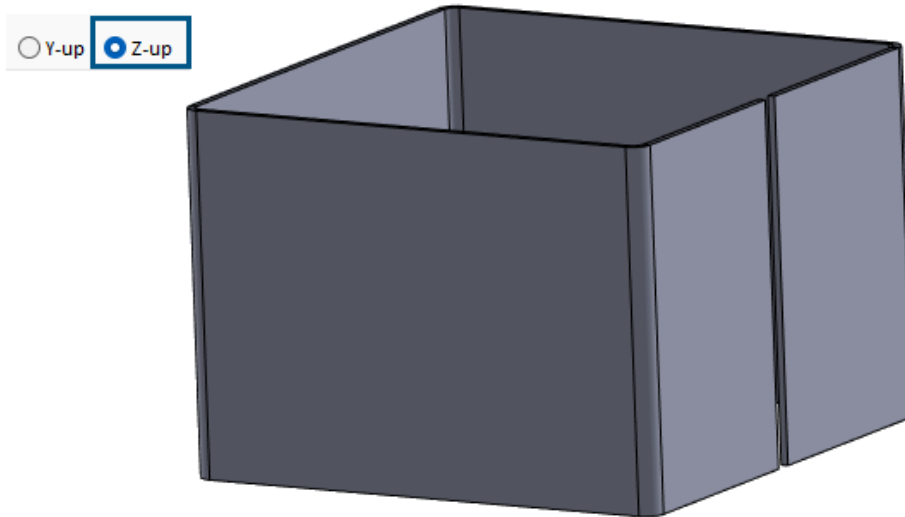
## Application Programming Interface

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

- Ability to import annotations into drawings
- Photorealistic rendering with SOLIDWORKS Visualize through the SOLIDWORKS API. Appearance support for the SOLIDWORKS Visualize API add-in includes:

- Access to new IRenderMaterial properties
- Ability to add or edit floor appearances of model scenes
- Texture mapping of nonlinear surfaces, including surface projections
- Improved performance:
  - When reloading a SOLIDWORKS model from disk
  - With component objects


## Specifying a Z-Up Template



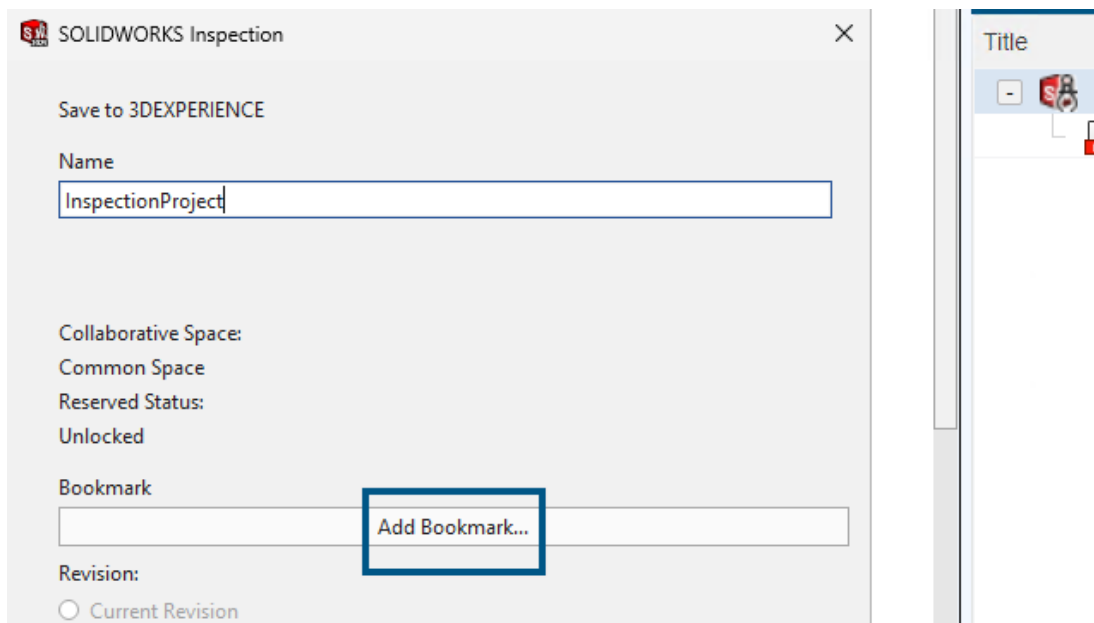
When you create a part or assembly, you can choose a template for a Z-up orientation. In earlier releases, SOLIDWORKS had a default Y-up orientation only.

The Y-up and Z-up orientation settings are available only for the default templates that SOLIDWORKS creates.

### To specify a Z-up template:

1. Click **New**  (Standard toolbar) or **File > New**.
2. In the dialog box:
  - a. Select a document type.
  - b. Specify an option:
    - **Y-up**. The Y-axis points upward.
    - **Z-up**. The Z-axis points upward.
  - c. Click **OK**.

## Saving SOLIDWORKS Inspection Files Using Bookmarks



You can save SOLIDWORKS Inspection files to the **3DEXPERIENCE**® platform using bookmarks.

### To save SOLIDWORKS Inspection files using bookmarks:

1. Open a project or create a new project, and from **MySession**, right-click the file and click **Save**.
2. In the Save to 3DEXPERIENCE dialog box, click **Add Bookmark**.
3. In the Bookmark Editor, right-click **Bookmarks** and select **New Bookmark**.
4. In the New Bookmark window, for **Title** enter a name for the bookmark and click **Create**.
5. Select the new bookmark.
6. Specify an option and click **Apply**.
  - a. **Add Existing**. Adds the newly created bookmark to existing bookmarks.
  - b. **Upload file**. Uploads an existing file.
7. To save the bookmark to the **3DEXPERIENCE** platform, click **Save**.

# 6

## User Interface

---

This chapter includes the following topics:

- **Specifying Component Name and Description Options at the System Level (2025 SP3)**
- **Search Commands (2025 SP2)**
- **Simplified Interface (2025 SP1)**
- **Command Predictor**
- **Reorganize Components**
- **Usability**
- **Hole Wizard**
- **Save and Auto Save Progress**
- **Create Document Group**

### Specifying Component Name and Description Options at the System Level (2025 SP3)

☒ Override document level tree display Component Name and Description ...

Component Name and Description

Select primary, secondary and tertiary name and description elements to show in the FeatureManager Tree. Certain elements appear inside ( ) or < > as shown.

Primary	( Secondary )	< Tertiary >
<input type="radio"/> Component Name	<input type="checkbox"/> Component Description	<input type="checkbox"/> Display State Name
<input checked="" type="radio"/> Component Description	<input type="checkbox"/> Configuration Name	
	<input checked="" type="checkbox"/> Configuration Description	

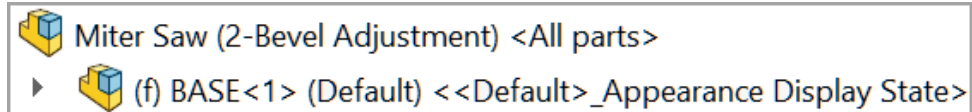
You can use **Override document level tree display** to specify options in the Component Name and Description dialog box at the system level.

When enabled, the system level options override the document level options for the component names in the FeatureManager design tree. The system level options do not overwrite the document level options in the document.

#### To specify component name and description options at the system level:

1. Open a model.

For example, in the FeatureManager design tree, the component name shows the component name, the configuration name, and the display state name.

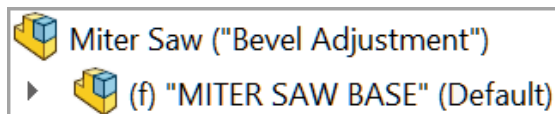


2. Click **Tools > Options > System Options > FeatureManager**.
3. Select **Override document level tree display**.

Selecting **Override document level tree display** disables the options in the Component Name and Description dialog box at the document level.

4. Click **Component Name and Description**.
5. In the Component Name and Description dialog box, select different options.  
For example, under **Primary**, select **Component Description**. Under **Secondary**, clear **Configuration Name** and select **Configuration Description**. Under **Tertiary**, clear **Display State Name**.
6. Click **Apply** and then click **OK** to close the dialog box.
7. Close the System Options dialog box.

In the FeatureManager design tree, the component name shows the component description and configuration description.



8. In the FeatureManager design tree, right-click the assembly and click **Tree Display > Component Name and Description**.

The options are disabled at the document level when **Override document level tree display** is selected.

Component Name and Description

Select primary, secondary and tertiary name and description elements to show in the FeatureManager Tree. Certain elements appear inside ( ) or < > as shown.

Primary

☒ Component Name
 ☐ Component Description

( Secondary )


☐ Component Description
 ☒ Configuration Name
 ☐ Configuration Description

< Tertiary >

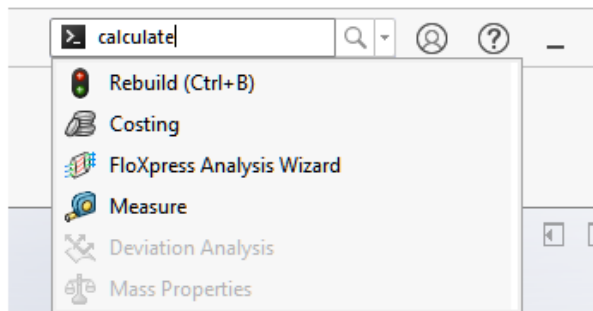
☒ Display State Name

☐ Do not show Configuration or Display State name if only one exists

**Name Preview :** Miter Saw (2-Bevel Adjustment) <All parts>

 Options for component names and descriptions are set in system options. To enable the options at the document level, click Options > System Options > FeatureManager and clear the Override document level tree display option.

## Search Commands (2025 SP2)



The **Search Commands** functionality provides better results because of enhanced terminology mapping. Terminology from other CAD packages is mapped to SOLIDWORKS tools to help you find the tools you need. The search results also include keyboard shortcuts for quicker access to the tools.

You can map multiple keywords to SOLIDWORKS tools. Previously only one keyword per tool was supported.

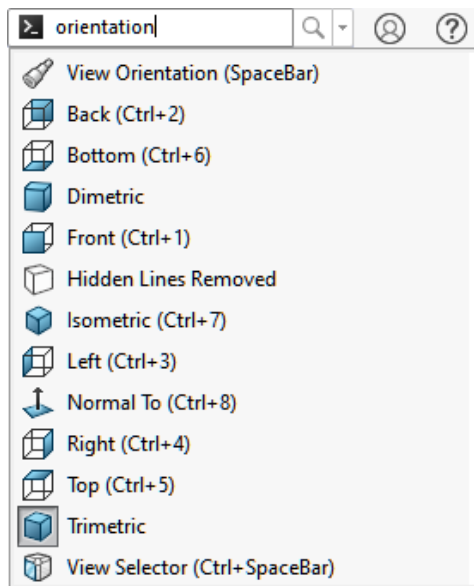
93

## Keyword Mapping

The software includes more keywords that map to SOLIDWORKS tools. This is helpful when you search for a tool that does not necessarily use the SOLIDWORKS names. For example, if you search for a term used in a different CAD product, the corresponding SOLIDWORKS tool may appear in the search results.

## Keyboard Shortcuts

When you use **Search Commands**, the results include the tools' keyboard shortcuts in parenthesis if they exist. If you use the **S** key to search for tools, the results also list the keyboard shortcut.

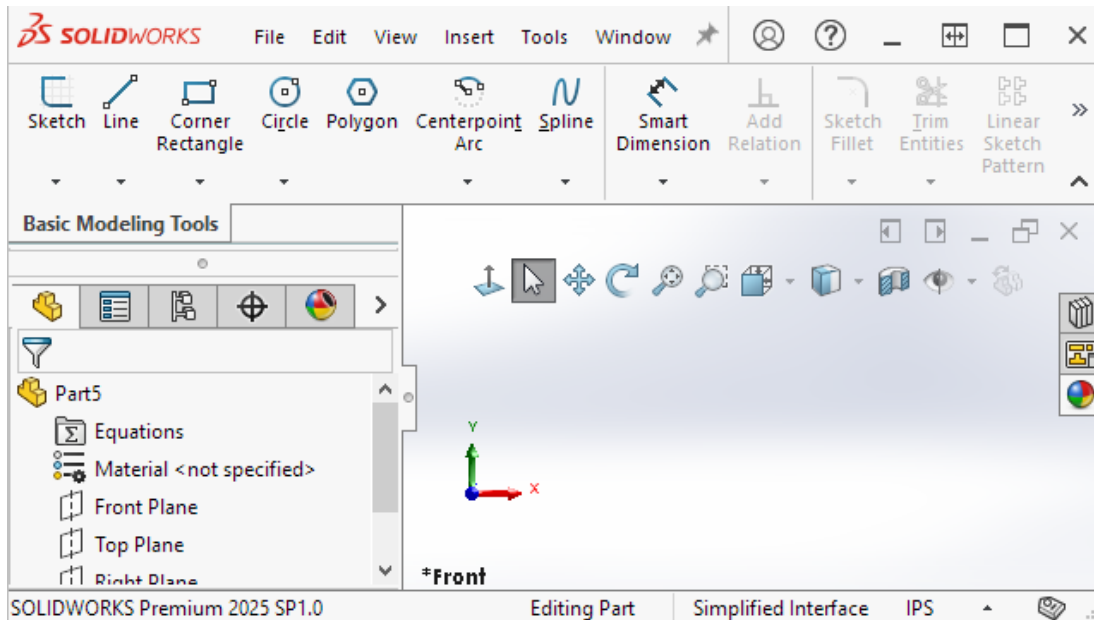


## Multiple Keyword Support

You can specify multiple keywords to use when searching for tools in the **Search Commands** functionality.

Click **Tools > Customize**. In the dialog box, on the Keyboard tab, in the **Search terms** column, specify keywords for tools separated by commas.

## Simplified Interface (2025 SP1)



**Simplified Interface** is a workspace that presents the SOLIDWORKS window with an abridged user interface. The window includes basic user interface elements tailored to the type of document that you open.

With a document open, click **View** > **Workspace** > **Simplified Interface**.

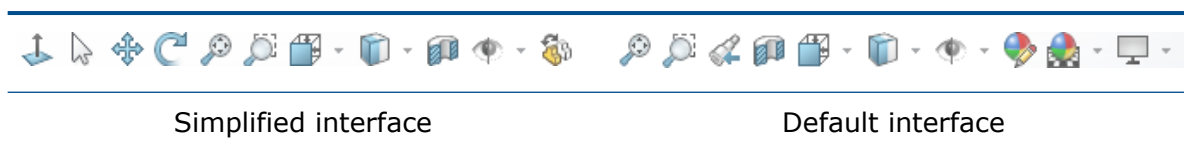
Without a document open, click **View** > **Simplified Interface**.

When selected, the status bar indicates the **Simplified Interface** workspace.

If you use the **Simplified Interface** workspace, customize the interface for your needs, then turn off **Simplified Interface**, SOLIDWORKS saves any customizations that you made if you turn on **Simplified Interface** again.

## Heads-Up View Toolbar

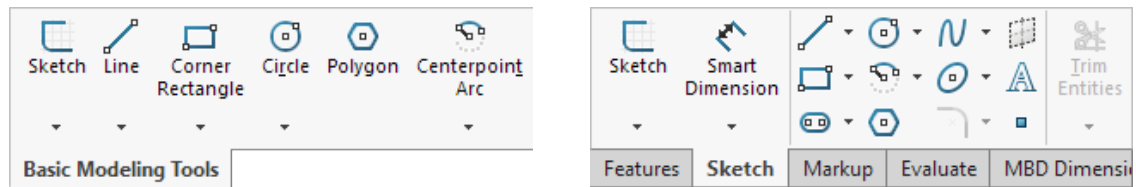
For parts and assemblies, the Heads-up View toolbar contains tools to manipulate views. It does not include appearances, scenes, or view settings.



## CommandManager

The CommandManager displays one tab per document type. The tabs are Basic Modeling Tools, Basic Assembly Tools, and Basic Drawing Tools which contain commonly used tools for those document types.





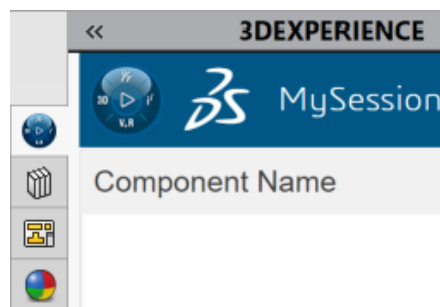
Simplified interface

Default interface

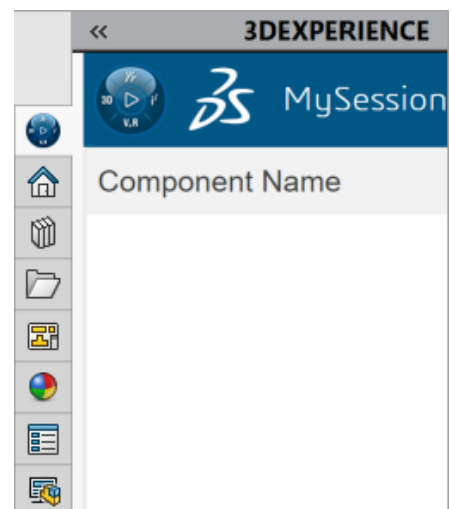
## Task Pane

The Task Pane contains the following tabs:

- 3DEXPERIENCE
- Design Library
- View Palette
- Appearances, Scenes, and Decals



Simplified interface

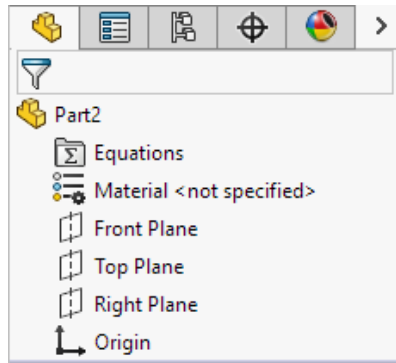


Default interface

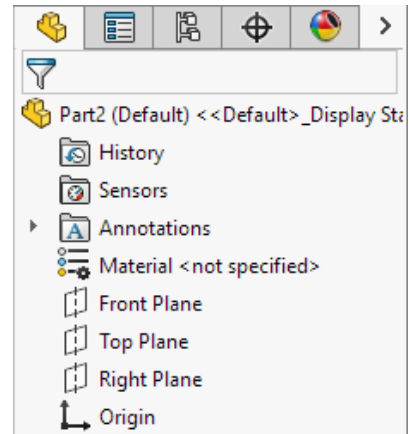
## FeatureManager Design Tree

The FeatureManager design tree contains the following items:

- Equations
- Material
- Planes
- Origin



Simplified interface

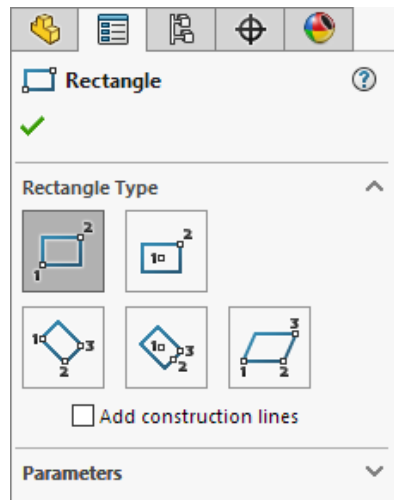


Default interface

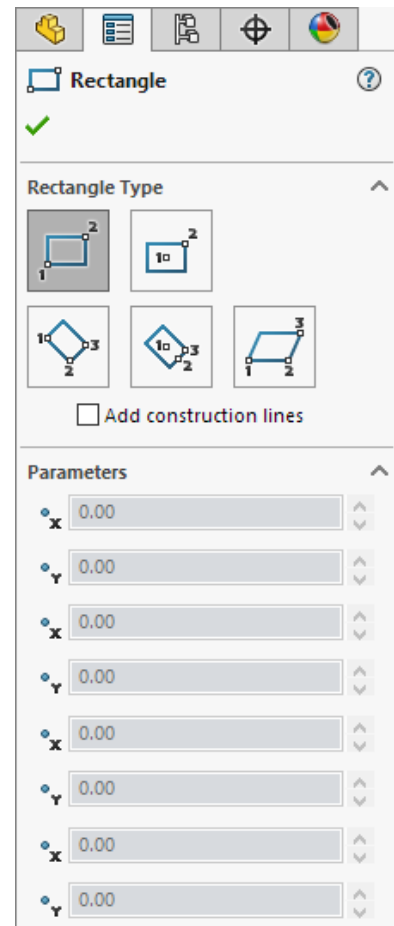
Items in the FeatureManager design tree do not include configuration or display state names if only one exists.

## PropertyManagers

Several PropertyManagers contain collapsed sections:



Simplified interface



Default interface

## Sketch Mode

For parts, the **Simplified Interface** opens a new part with an active sketch on the Front plane.

## MotionManager Design Tree

The MotionManager design tree is hidden.

## Command Predictor



The Command Predictor predicts the tools that are most relevant based on the tools you used in the current SOLIDWORKS session. It reduces the time that you spend searching for tools that you are likely to use next.

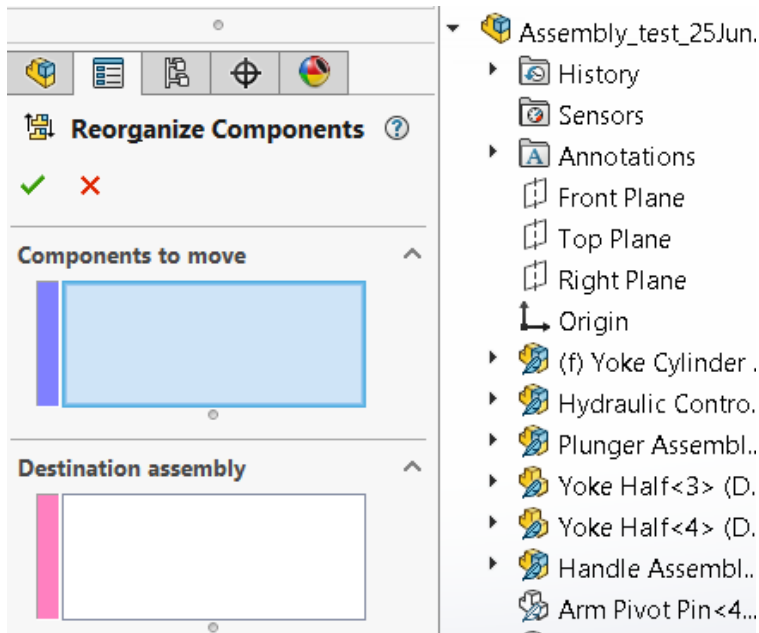
### To use the Command Predictor:

1. In a SOLIDWORKS document, click the Command Predictor (beta) tab (CommandManager).
2. Click a tool on the tab.

The Command Predictor is beta functionality and tool suggestions are based on a machine learning model.

## Reorganize Components

Enhancements to the user interface help improve productivity.

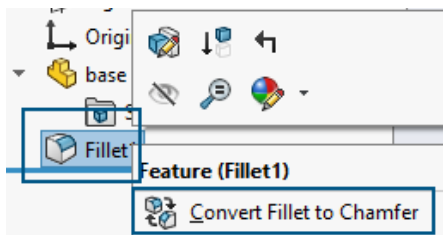


The Reorganize Components dialog box has moved to a PropertyManager. The dialog box no longer obscures the graphics area.

## Usability

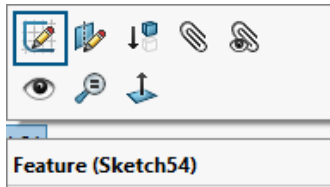
The user interface is enhanced to improve productivity.

### Fillet to Chamfer Naming

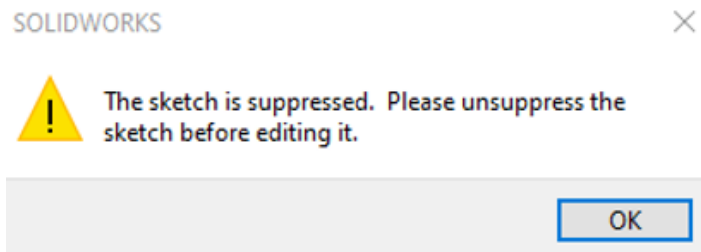


In the FeatureManager® design tree, when you right-click a fillet and select **Convert Fillet to Chamfer**, the FeatureManager design tree changes the fillet name to chamfer. You can use **Convert Chamfer to Fillet** as well and the software updates the name accordingly. In earlier releases, the fillet name remained in the FeatureManager design tree.

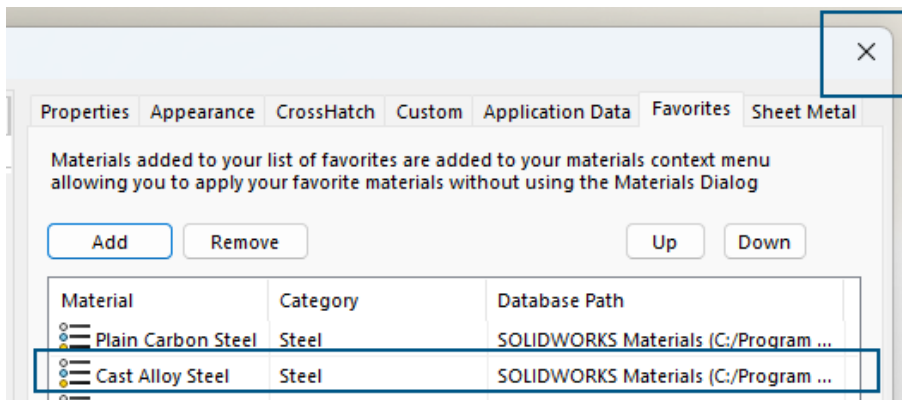
## Unsuppress the Sketch Automatically



In the FeatureManager design tree, you can right-click a suppressed sketch you want to edit, select **Edit Sketch**, and the software unsuppresses the sketch automatically. In earlier releases, you received this notification:

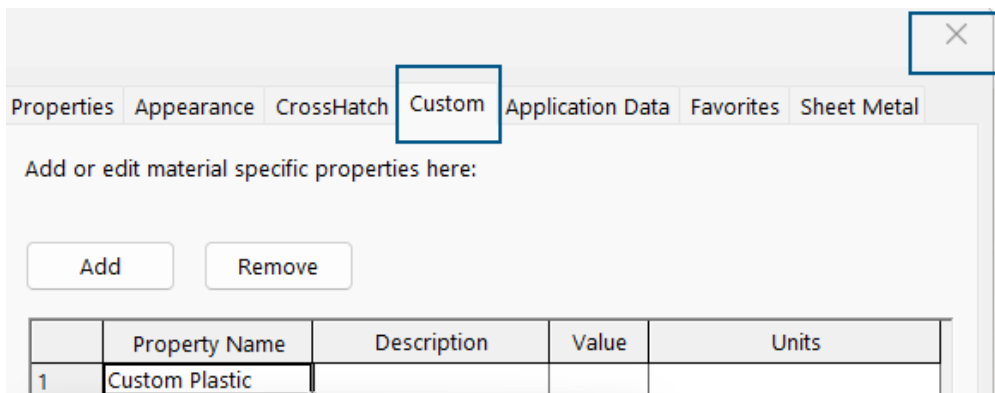


## Material Dialog Box - Favorites Tab

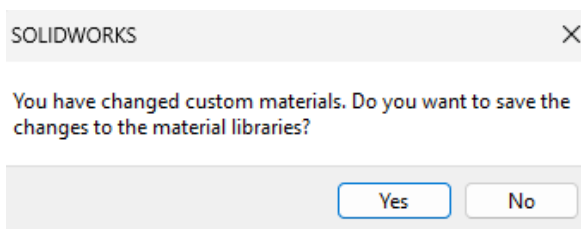


In the Material dialog box, after you add a new material to **Favorites**, you can click **Close** or **x** in the upper-right corner to save the changes and close the dialog box. In earlier releases, when you clicked **x**, the software did not save the changes.

## Material Dialog Box - Custom Tab

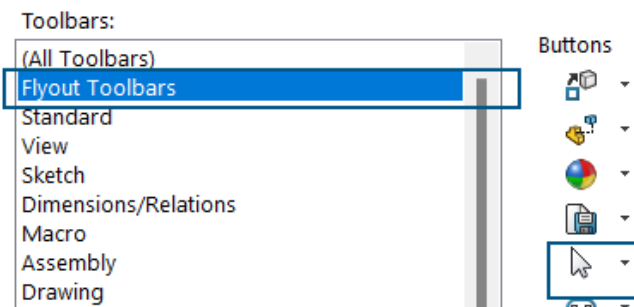


In the Material dialog box, after you add a new material from **Custom Materials** to **Custom**, and click **X**, you receive the following notification:



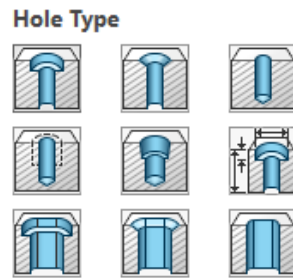
In earlier releases, you only received the notification when you clicked **Close**.

## Flyout Menu in Customize Dialog Box




In the Customize dialog box, the **Select** tool is available under the flyout toolbars.

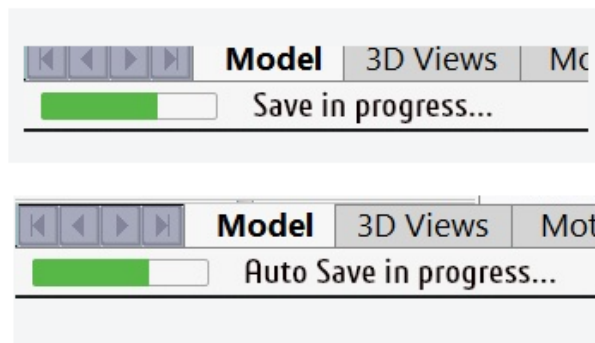
## Hole Wizard



Enhancements to the user interface help improve productivity.

When you click **Hole Wizard**  (Features toolbar), the **Hole Type** icons are clearer to distinguish.

## Save and Auto Save Progress

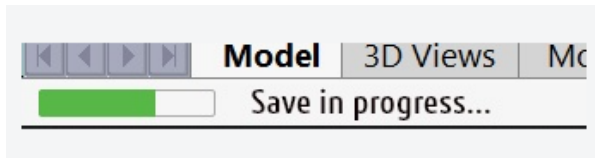




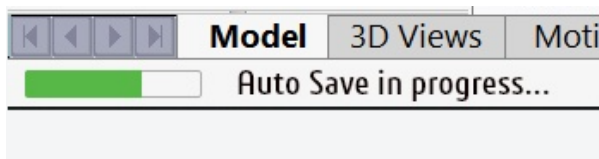
Enhancements to the user interface help improve productivity.

When you save files to the **3DEXPERIENCE** platform, the software shows messages to indicate the software is saving your files.

While you save a file on the **3DEXPERIENCE** platform, the software shows a progress bar and displays "Save in progress..." in the status bar.



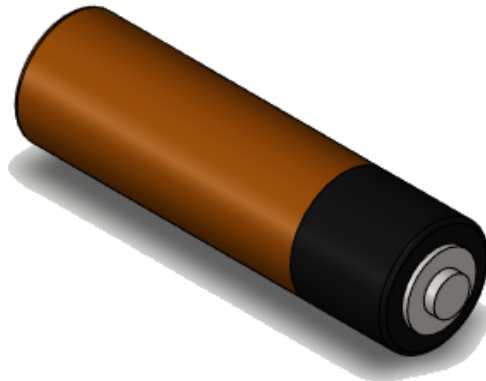
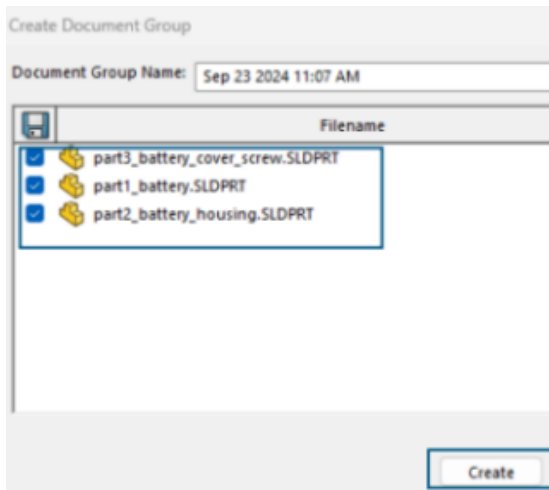
While a file autosaves on the **3DEXPERIENCE** platform, the software shows a progress bar and displays "Auto Save in progress..." in the status bar.



## Create Document Group

You can save all open files in SOLIDWORKS as a single document group. This lets you open all the files saved in that group at once. In earlier releases, you had to open every file individually.

### Creating Multiple Files as a Document Group



### To create a document group:

1. In a SOLIDWORKS document, click **Window > Create Document Group**.

The Create Document Group dialog box contains a list of open files in SOLIDWORKS.

2. In the dialog box:
  - a. Select the required files.
  - b. Click **Create**.

The software shows a success notification. A message notifies you that SOLIDWORKS created Document Group and you can access it from the Recent tab in the Welcome dialog box.

## Updating a Document Group

When you create new parts, you can save the parts as part of a previously created document group.

### To update a document group:

1. Open the parts to include in a document group.
2. Click **Window > Create Document Group**.
3. In the dialog box:
  - a) In **Document Group Name**, select a document group.

The software populates the list with the open files and the files saved under the selected document group.
  - b) Click **Create**.

# 7

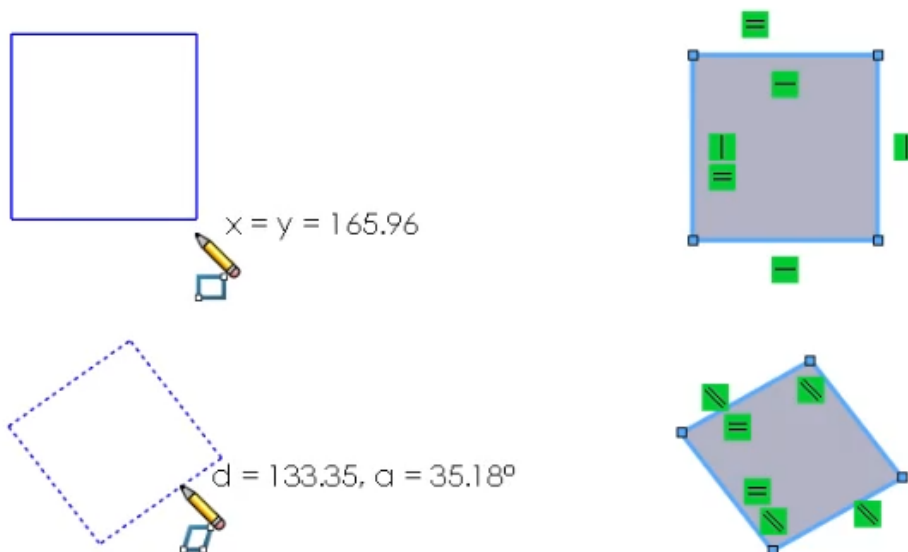
## Sketching

---

This chapter includes the following topics:



- **Creating Squares Using Rectangle Tools (2025 SP2)**
- **Flip Endpoint Tangent (2025 SP1)**
- **Repairing Dangling Relations**
- **Linear and Circular Sketch Patterns**



### Creating Squares Using Rectangle Tools (2025 SP2)



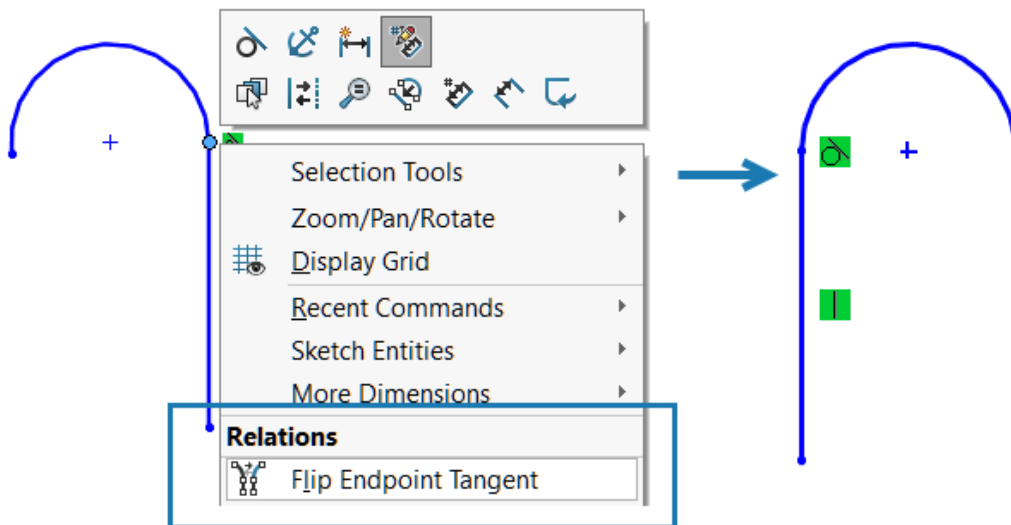
When using the rectangle tools, you can create squares by pressing **Shift** while sketching.

#### To create squares using the rectangle tools:

1. Open a part or an assembly.
2. In the FeatureManager design tree, select a plane.
3. Click **Sketch**  (Sketch toolbar).
4. Create a corner rectangle:
  - a. On the Sketch tab of the CommandManager, click **Corner Rectangle** .
  - b. In the sketch, click to place the first corner.

- c. Press **Shift**, and then drag and click to create a square.  
The adjacent sides of the square have equal relations.
5. Create a parallelogram:
  - a. Click **Parallelogram** .
  - b. Click to place the first corner.
  - c. Press **Shift** and then drag, rotate, and click to specify the length and angle of the first edge.
  - d. Continue pressing **Shift**, and drag and click to specify the angle of the other three edges.
6. Create a 3 point corner rectangle and enter a value:
  - a. Click **Options > System Options > Sketch** and select **Enable on screen numeric input on entity creation**.
  - b. In the sketch, click **3 Point Corner Rectangle** .
  - c. Click to place the first corner and drag the cursor.
  - d. Enter a value for the size of the square and press **Enter**.
  - e. Press **Shift**, drag the cursor and click to create the square.


## Flip Endpoint Tangent (2025 SP1)

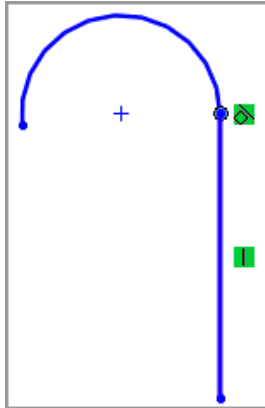



You can flip the endpoint of a tangent arc that is connected to a line. The radius of the arc does not change.

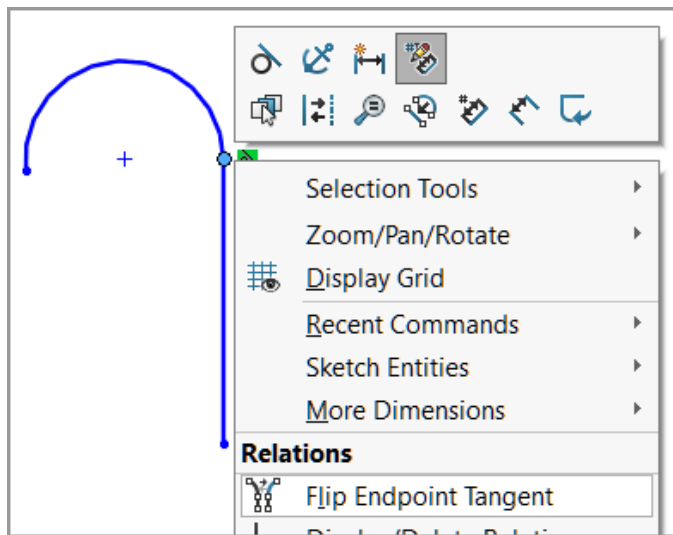
The functionality is not available for 3D sketches.

### To flip the endpoint of a tangent arc:

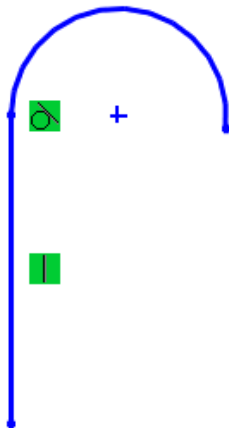
1. Open a new sketch and sketch a line.
2. Click **Tangent Arc**  and create an arc from the end point of the line.



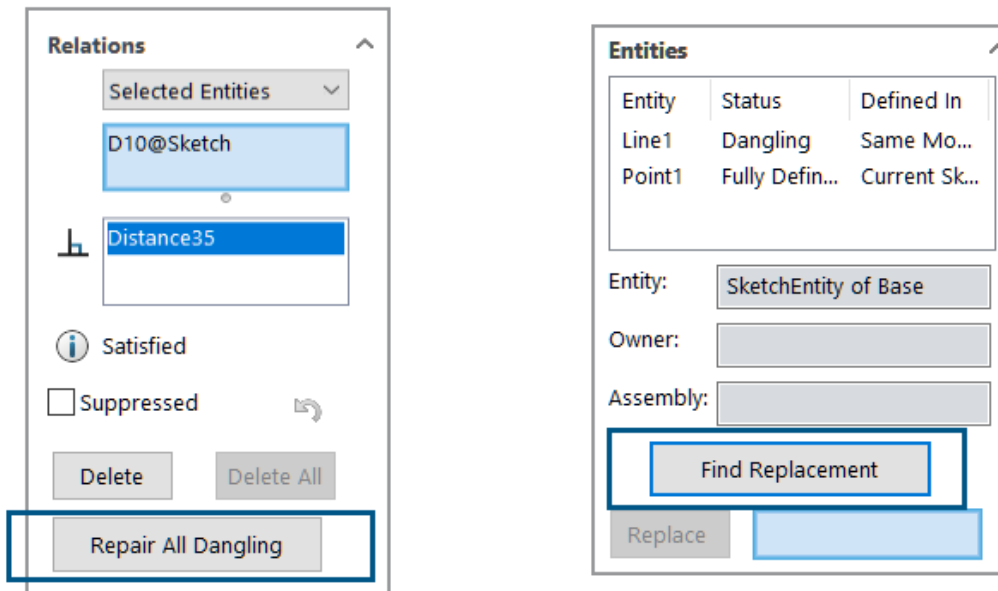
3. Right-click the point where the arc and the line meet, and click **Flip Endpoint Tangent** .



The tangent arc flips position:



## Repairing Dangling Relations




In the Display/Delete Relations PropertyManager, you can use **Find Replacement** to fix dangling relations in a sketch. Use **Repair All Dangling** to automatically fix all dangling relations.

You can use **Auto Repair Sketch Relation or Dimension**  to repair the selected dangling relation from the context toolbar.



These options are available only for 2D sketches. Dangling relations that have external references cannot be repaired using **Repair All Dangling** and **Find Replacement**. You must manually repair these dangling relations.

### To repair dangling relations:

1. Open a model that has a dangling relation.
2. Click **Display/Delete Relations**  (Dimensions/Relations toolbar) or **Tools > Relations > Display/Delete**.
3. In the PropertyManager, under **Relations**, select a dangling relation.
4. Under **Entities**, click **Find Replacement**.

SOLIDWORKS® searches for a replacement. A message appears if a replacement is not found.

**Repair All Dangling** and **Find Replacement** are available when a sketch has dangling relations.

5. When a replacement is found, review the replacement listed in **Entity to replace the one selected above** and then click **Replace**.

## Linear and Circular Sketch Patterns

For linear and circular patterns, you can generate a fully defined sketch pattern.

For a linear sketch pattern of a fully defined entity, select these options in the Linear Pattern PropertyManager to generate a fully defined pattern:

- **Dimension X spacing**
- **Fix X-axis**
- **Dimension Y spacing**
- **Dimension angle between axes**

For a circular sketch pattern, a coincident relation is applied automatically between a selected point and the center of the pattern when the origin point is not the selected point.

# 8

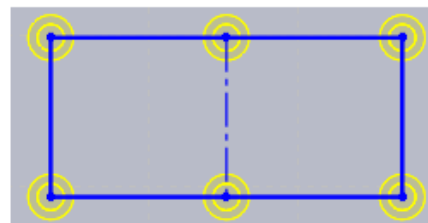
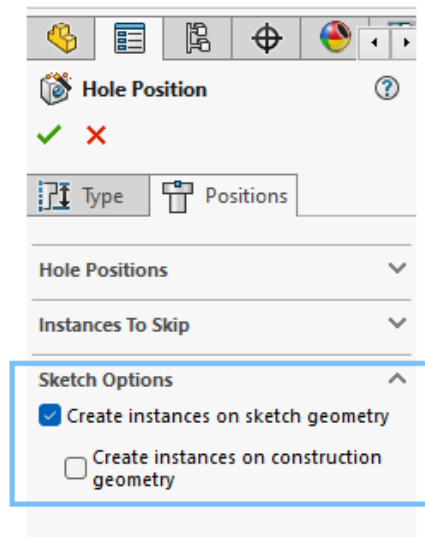
## Parts and Features

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This chapter includes the following topics:

- **Retaining Hole Wizard Sketch Options (2025 SP3)**
- **Pinning the Fillet or Chamfer PropertyManager (2025 SP2)**
- **Exiting Part Processes with the Escape Key (2025 SP2)**
- **Defeature Silhouette Method for Parts**
- **Patterning Reference Geometry**
- **Converting Mesh BREP to Standard BREP**
- **Segment Mesh Enhancements**
- **Move/Copy Body Features**
- **Variable Size Fillets**
- **Curve Through XYZ Points Enhancements**

### Retaining Hole Wizard Sketch Options (2025 SP3)



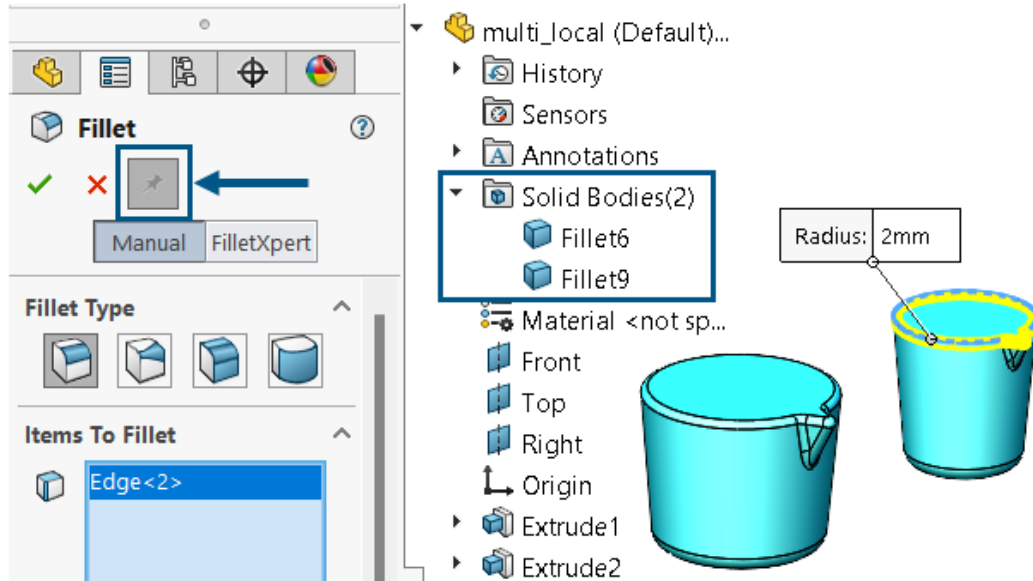
For Hole Wizard features, the software retains the **Sketch Options** settings for new holes, new parts, or new sessions of SOLIDWORKS.

On the Hole Wizard Positions tab, this functionality applies to the options **Create instances on sketch geometry** and **Create instances on construction geometry**.



By default, these **Sketch Options** settings are cleared.




## Pinning the Fillet or Chamfer PropertyManager (2025 SP2)



You can pin the Fillet or Chamfer PropertyManager.

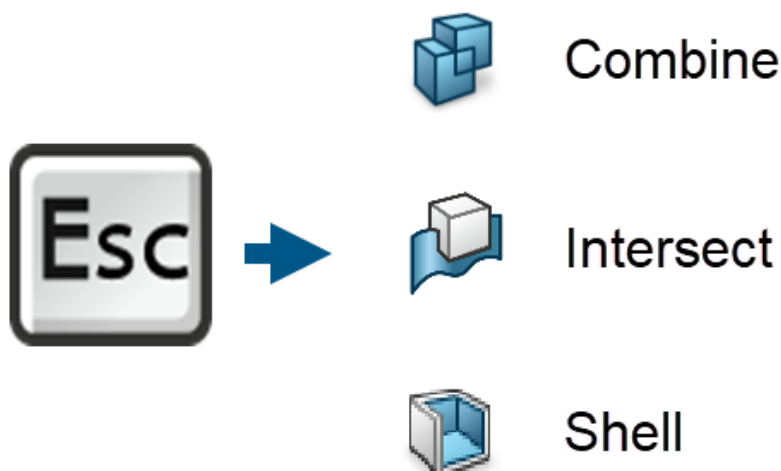
**Benefits:** You can apply multiple fillets or chamfers with the same or different parameters to different bodies in sequence without having to reopen the PropertyManager each time. The fillet or chamfer features can also be different types.

Pushpin availability

Feature	Information
Fillet pushpin	<ul style="list-style-type: none"> <li>Available only for <b>Manual</b> mode.</li> <li>Available for these fillets: <ul style="list-style-type: none"> <li> <b>Constant Size</b></li> <li> <b>Face</b></li> <li> <b>Full Round</b></li> </ul> </li> </ul> <p>During a session, the software retains the settings under <b>Items to Fillet</b>, <b>Fillet Parameters</b>, and <b>Fillet Options</b>.</p> <div style="border: 1px solid gray; padding: 5px; margin-top: 10px;"> <p>The pushpin is unavailable when editing an existing fillet.</p> </div>

Feature	Information
Chamfer pushpin	<ul style="list-style-type: none"><li>• Available for all five types of chamfers.</li><li>• During a session, the software retains the settings under <b>Items to Chamfer</b>, <b>Chamfer Parameters</b>, and <b>Chamfer Options</b>.</li></ul>

## Exiting Part Processes with the Escape Key (2025 SP2)








To immediately exit out of lengthy part processes, press the **Esc** key to cancel the ongoing command and revert the model to its previous state. This applies to the part commands **Combine**, **Intersect**, and **Shell**.

**Benefits:** You can exit processes that may take a long time to complete or that you started by mistake.

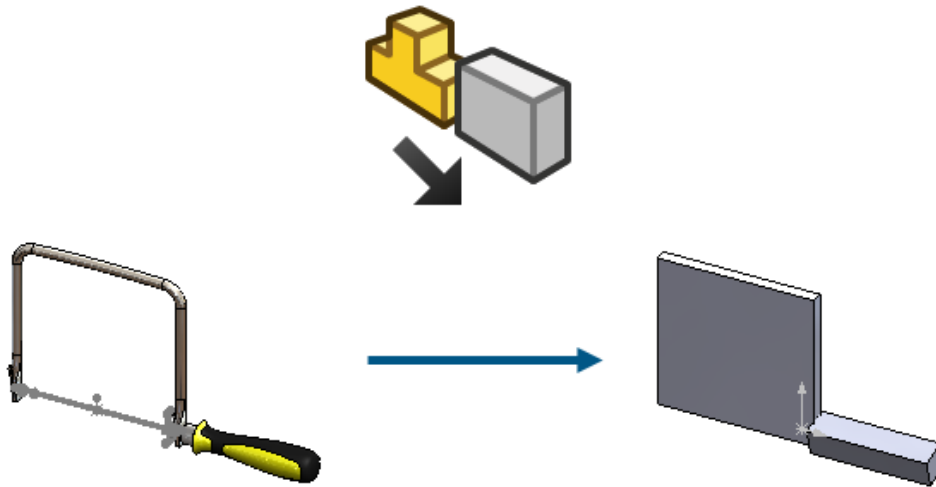
Status bar messages during a preview or main operation alert you that this functionality is available: Press <ESC> to cancel Preview **OR** Press <ESC> to cancel <Combine/Intersect/Shell> command.

Press the **Esc** key during these commands to exit the described processes.

Command	PropertyManager Actions That You Can Exit
<b>Combine</b>	<ul style="list-style-type: none"> <li>Click <b>Show Preview</b> with the <b>Add</b>, <b>Subtract</b>, or <b>Combine</b> operation.</li> <li>Click  to start running the command.</li> </ul>
<b>Intersect</b>	<ul style="list-style-type: none"> <li>Click <b>Intersect</b>.</li> <li>Click  to start running the command.</li> </ul>
<b>Shell</b>	<ul style="list-style-type: none"> <li>Click <b>Show Preview</b> when you select a face or solid body.</li> <li>Click  to start running the command.</li> <li>First click <b>Show Preview</b> then do one of the following: <ul style="list-style-type: none"> <li>Under <b>Parameters</b>, do one of the following: <ul style="list-style-type: none"> <li>Change the shell <b>Thickness</b> .</li> <li>Select a face.</li> <li>Select a solid body.</li> <li>Select <b>Shell outward</b>.</li> </ul> </li> <li>Under <b>Multi-thickness Settings</b>, change the <b>Multi-thickness(es)</b>  value or select a face.</li> </ul> </li> </ul>

The software returns you to the PropertyManager state before clicking **OK** and remembers all settings.

## Defeature Silhouette Method for Parts




For single body and multibody parts, you can use the Silhouette defeature method to create a highly simplified part and make it associative to the parent part.

In previous releases, the Silhouette defeature method was available only for assemblies. You define groups of bodies and then define a simplification method for these groups.

Simplification methods include:

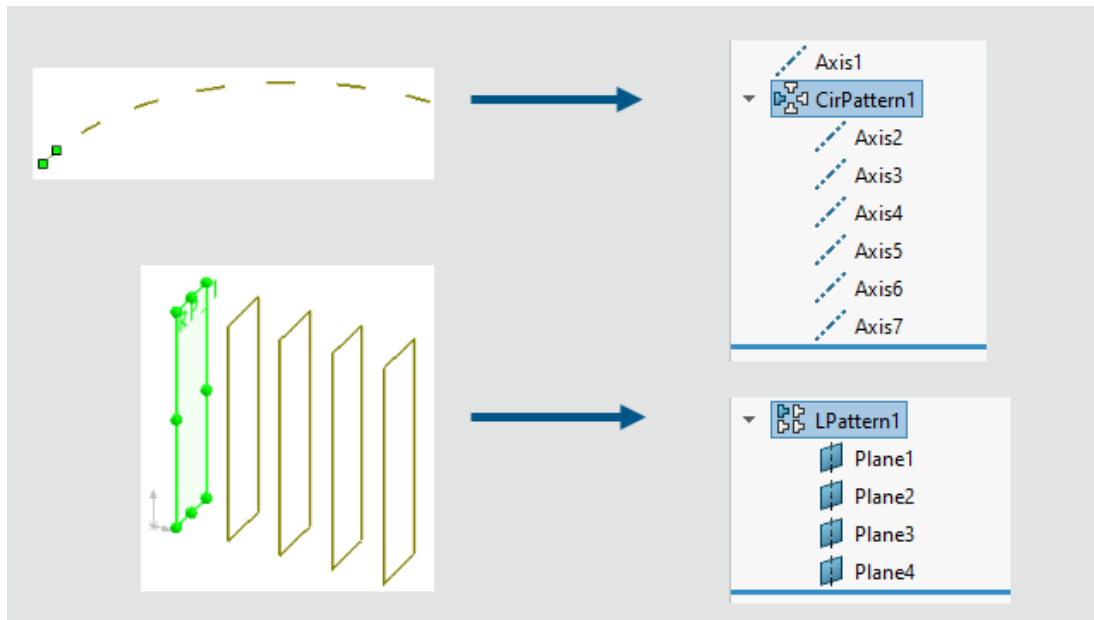
- **Bounding Box**
- **Cylinder**
- **Polygon Outline**
- **Tight Fit Outline**
- **None (Copy Geometry)**

You can retain a link to the original model so if you update the original, the defeatured model is updated. In the Results PropertyManager tab, when you select **Create a new configuration**, in the ConfigurationManager, you can right-click the defeature configuration and select **Edit Defeature** or **Update Defeature**.

To access the Silhouette defeature method, in a part, click **Tools > Defeature** and under **Defeature Method**, click **Silhouette** .



Click  or  to navigate the modes and finalize the defeature process.

## Patterning Reference Geometry



You can create linear or circular patterns of planes and axes.

### To pattern reference geometry:

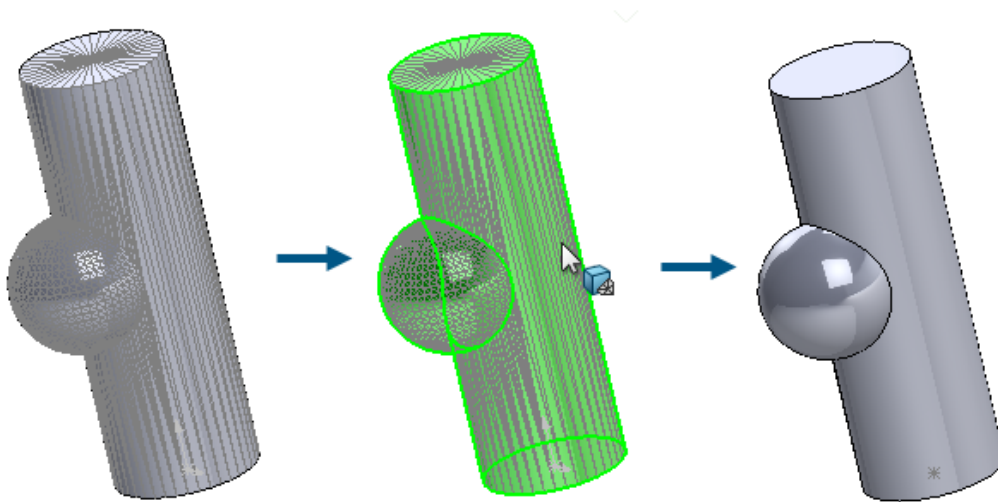
1. Open a part, click **Insert > Pattern/Mirror**, and select **Linear Pattern** or **Circular Pattern**.
2. In the PropertyManager, select **Reference Geometry**.
3. For **Reference Plane or Reference Axis to Pattern** , select the plane or axis to pattern.
4. Specify the parameters, then click .

You can modify the spacing and instance parameters for **Direction 1** and **Direction 2**. You can skip, vary, and delete instances.

#### Limitations:

- If a plane contains a sketch, the plane pattern does not pattern the sketch.
- A pattern can contain one reference geometry entity only, either one plane or one axis.

## Converting Mesh BREP to Standard BREP



You can use the **Convert Mesh to Standard** tool to convert mesh BREP faces with recognized geometry to standard BREP faces.

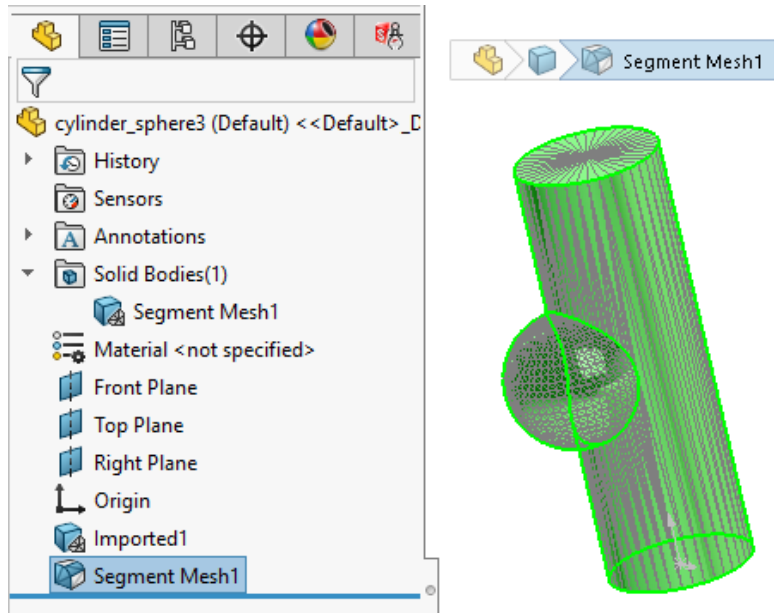
This functionality works for mesh BREP or hybrid mesh bodies that have recognized geometry. The functionality works best for meshes with well-defined planar, cylindrical, conical, and spherical geometry that do not have significant noise.




**Benefits:** Standard BREP geometry is more functionally complete than mesh or hybrid geometry.

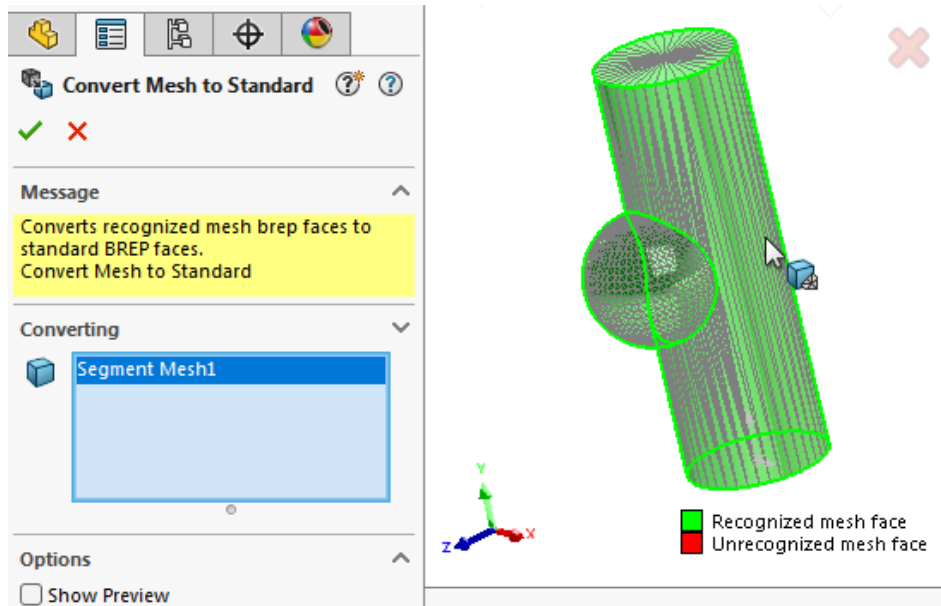
### To convert mesh BREP with recognized faces to standard BREP:


1. Open a model that has mesh BREP or hybrid mesh bodies with segmented and recognized faces.

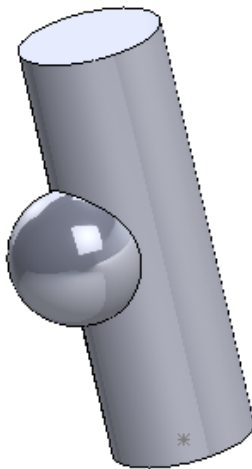
This meshed model has been segmented into cylindrical, spherical, and planar faces.




2. Do one of the following:
    - Right-click a body and select **Convert Mesh to Standard** .
    - Click **Insert > Mesh > Convert Mesh to Standard** .
    - Click **Convert Mesh to Standard**  (Mesh Modeling CommandManager).
  3. In the PropertyManager, under **Select Body**, select bodies to convert segmented, recognized mesh BREP faces to standard BREP faces.
- Colors indicate faces that are recognized or unrecognized. You can specify these **Recognized mesh face** and **Unrecognized mesh face** colors in **Tools > Options > System Options > Colors > Color scheme settings**.
- This entire model is recognized as one **Convert Mesh to Standard** feature, shown as a green **Recognized mesh face**, as indicated in the legend in the lower-right corner of the graphics area.



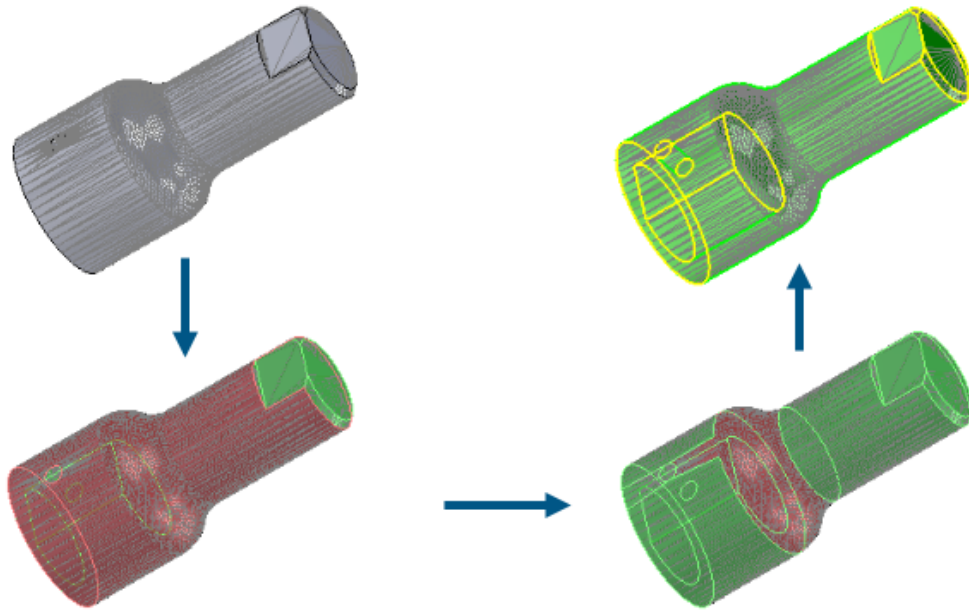
4. Click  to convert the recognized segmented mesh BREP faces to standard BREP faces.



The converted standard BREP faces appear in the FeatureManager® design tree with the **Convert to Standard BREP** name and icon .



## Segment Mesh Enhancements




The **Segment Mesh** tool recognizes additional face types and has an improved user interface.

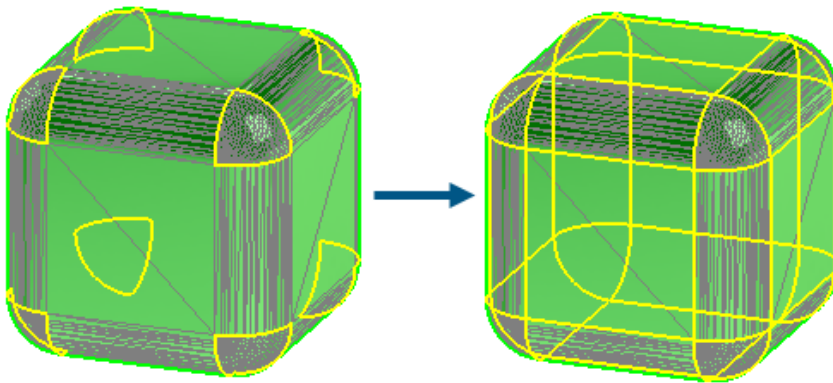
### Additional Face Types Recognized

When you segment meshes, the software can recognize faces that are conical or spherical, in addition to planes and cylinders. You can convert these recognized faces to standard BREP faces with the same geometric condition.

### Improved User Interface

In the Segment Mesh PropertyManager, under **Segmenting**, the **Facet Shape**  tool is available. This tool creates segments by grouping adjacent facets based on the shape difference, which typically indicates a boundary between two regions in the model used to create the mesh file.

Under **Options**, select **Show Preview** to preview the edges for segmented faces, shown as yellow. Under **Perimeter**, drag to adjust the value to refine the segmentation of faces.



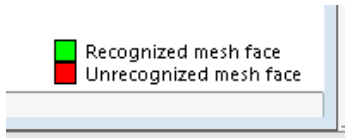
When you run the **Segment Imported Mesh Body** tool, improved graphical information helps you understand which faces have been recognized.

If you have not previously segmented the model, the display of mesh BREP bodies and hybrid mesh bodies does not change.

- Standard BREP and graphics bodies are optionally hidden.
- Selected faces are highlighted using the **Selected Item 1** color specified in **Tools > Options > System Options > Colors > Color scheme settings**.

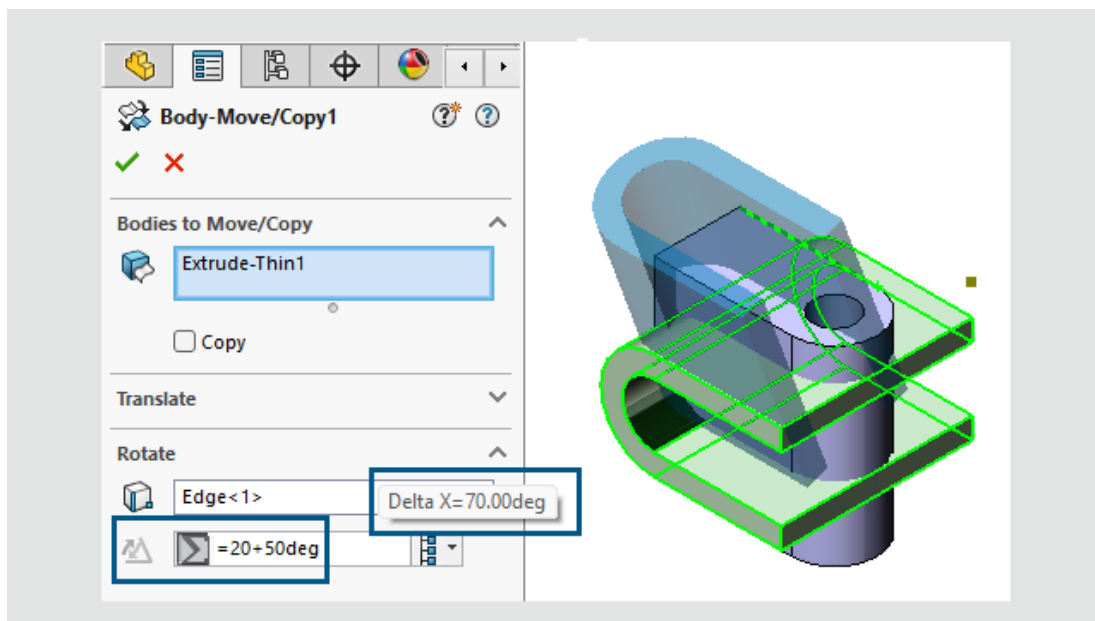
After the first round of segmenting the model, the following display changes apply:

- A legend appears to explain the colors used for recognized and unrecognized faces.











- The software uses the **Recognized mesh face** and **Unrecognized mesh face** colors specified in **Tools > Options > System Options > Colors > Color scheme settings**.




## Move/Copy Body Features

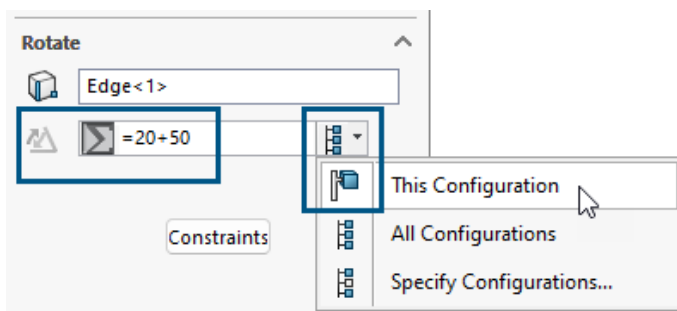


The **Move/Copy Body** feature offers enhanced support for equations and configurations.

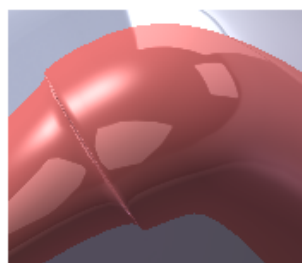
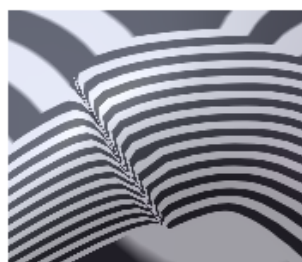
In the Move/Copy Body PropertyManager, you can use equations to specify values for the **Distance**  dimension under **Translate** and for the **Angle**  dimension under **Rotate**. In the PropertyManager, enter = and the equation. For example, enter =20+50. To access this equation in the Equations, Global Variables, and Dimensions dialog box, in the FeatureManager design tree, right-click **Equations** and select **Manage Equations**.

The **Distance**  and **Angle**  dimension icons are replaced with the icons  and . To flip the dimensions along the entity you selected, under **Translate**, click **Distance**  or under **Rotate**, click **Angle** .

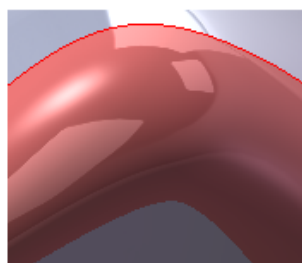
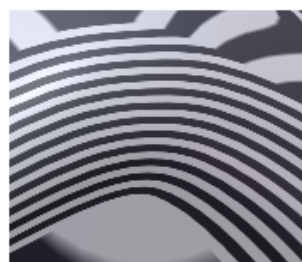
You can use configurations to specify the values for all dimensions, including these equation-driven values. **This Configuration** , **All Configurations** , and **Specify Configurations** .



## Variable Size Fillets




2024



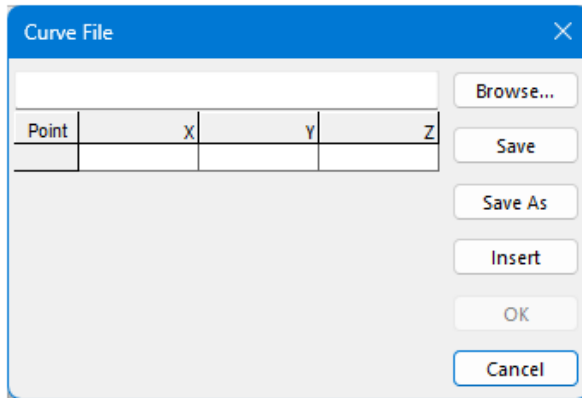
2025 Option

For variable size fillets, you can create continuously blended fillets with the **Continuous edge blend** option.

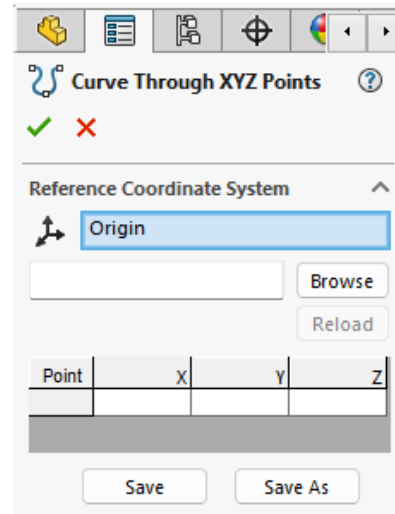
To access this option, in the Fillet PropertyManager, for **Fillet Type**, select **Variable Size Fillet**  and under **Fillet Options**, select **Continuous edge blend**.

This option uses an improved algorithm to create continuously blended edges that are extremely smooth.

## Curve Through XYZ Points Enhancements



2024




2025

The **Insert > Curve > Curve Through XYZ Points** functionality uses a PropertyManager in which you can select a different coordinate system. The points of the curve transform into the space of the coordinate system.

In earlier releases, this functionality used a dialog box and could only use the origin of the part for the curve.

In the PropertyManager, you can:

- Manually enter the XYZ coordinate data.
- Click **Browse** to select a `.sldcrv` or `.txt` file.
- Click **Reload** to update the curve based on any modifications made to the `.sldcrv` or `.txt` file used to create it.

When you open files created before SOLIDWORKS 2025 and edit curves created by XYZ points, in the PropertyManager, under **Reference Coordinate System**, the software uses the origin for **Coordinate System (Origin)** .

# 9

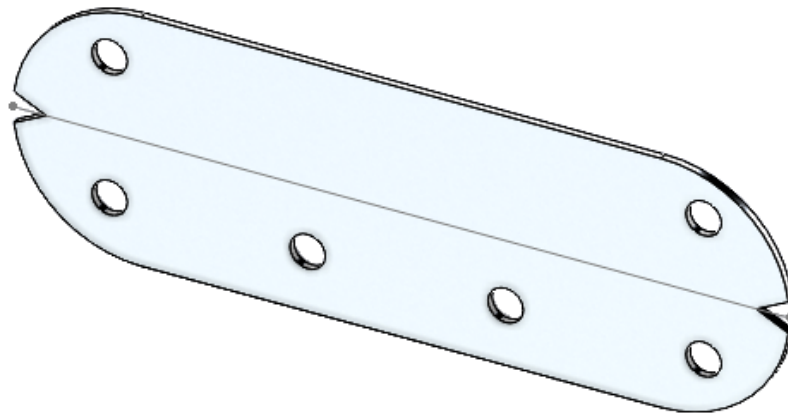
## Sheet Metal

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This chapter includes the following topics:

- **Bend Notches**
- **Tab and Slot**
- **Multi Length Edge Flanges and Automatic Flange Length Dimensions**
- **Performance Improvements in Cosmetic Thread Features**
- **Performance Improvements in Rebuilding Drawings**

### Bend Notches



You can create notches across bends in flattened sheet metal parts. In manufacturing, bend notches help manufacturers determine where to put the press brake. You can use

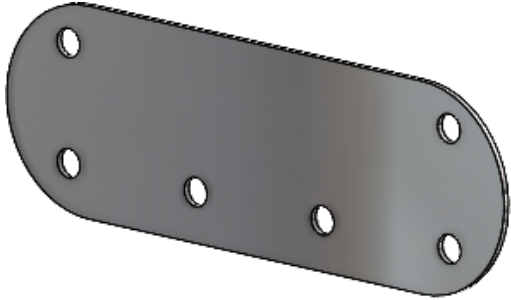
notch features on all bends so the bending operator can then use them to line up the bend with the tooling.

## Creating Bend Notches

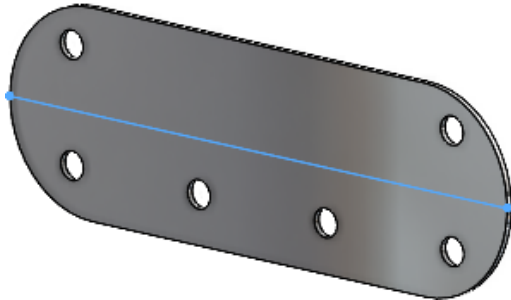
You can create bend notches on sheet metal parts in the flattened state.


### To create bend notches:

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

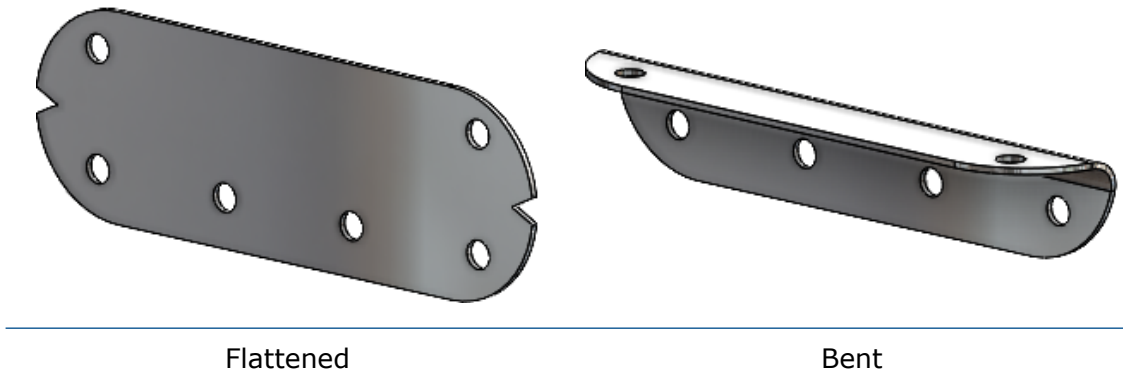


2. In the graphics area, select the bends where you want to add notches.



3. In the PropertyManager, specify options and click .

The notches appear in the flattened sheet metal part. You can edit the notches only when the part is flattened.



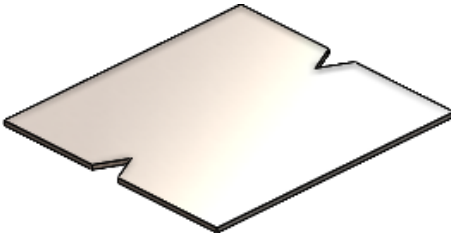

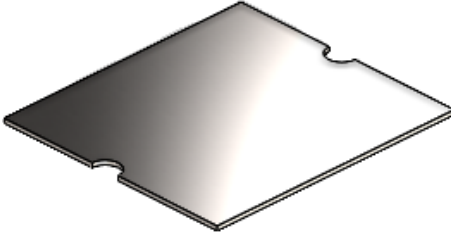


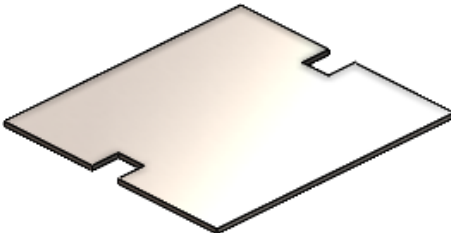


# Bend Notch PropertyManager

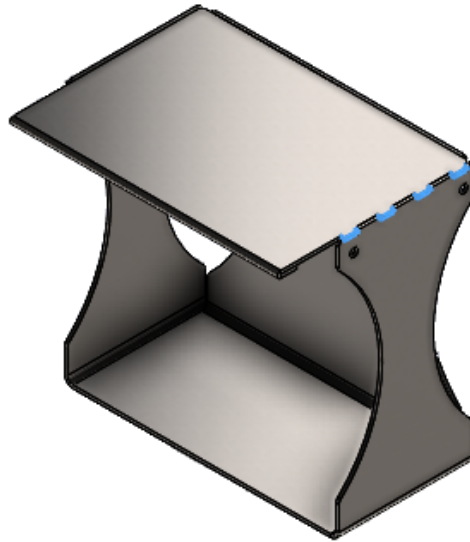
**To open this PropertyManager:**

- 1. In a flattened sheet metal part, click **Bend Notch**  (Sheet Metal toolbar) or **Insert > Sheet Metal > Bend Notch**.

## Bend Notch

<b>Bends</b>	Lists the bends to apply notches to.
<b>Collect all bends</b>	Selects all bends in the part to apply notches to.
<b>Notch type</b>	<p>Specifies the notch shape:</p> <ul style="list-style-type: none"><li>• <b>Triangular</b>. Specify the <b>Width</b>  and <b>Depth</b> .</li></ul>  <ul style="list-style-type: none"><li>• <b>Circular</b>. Specify the <b>Radius</b> .</li></ul>  <ul style="list-style-type: none"><li>• <b>Rectangular</b>. Specify the <b>Width</b>  and <b>Depth</b> .</li></ul> 

## Tab and Slot



The workflow for creating tab and slot features is simplified and provides more flexibility. Additional options let you create center-aligned tabs, offset tabs in equal increments, tab directions, and instances of tab and slot features to skip.

After you select the tab edge in a sheet metal part, SOLIDWORKS® automatically selects a slot face that is normal to the edge to streamline the process. For non-sheet metal parts, you need to select the slot face.



If you have nonintersecting regions of two bodies, the tab and slot feature applies only to the intersecting regions.

### Tab and Slot PropertyManager

#### Spacing

---

**Center Align**

Places the tabs from the center of the intersecting edge. Specify the **Number of Instances**  and **Spacing**  to define the number of instances based on the distance.

---

#### Offset

**Tab Start Reference**

Specifies the point, vertex, or edge where the offset begins.

**Tab End Reference**

Specifies the point, vertex, or edge where the offset ends.

---



**Equal Offset**

Creates an offset where the start and end distance is the same from the reference points.

## Tabs

**Tab Direction**


(Non-sheet metal parts only.) Creates the tab in a direction other than normal to the tab face based on your selection in the graphics area.

You can select points, planes, edges, axes, vertices, linear sketch entities, or planar faces.

## Instances to Skip

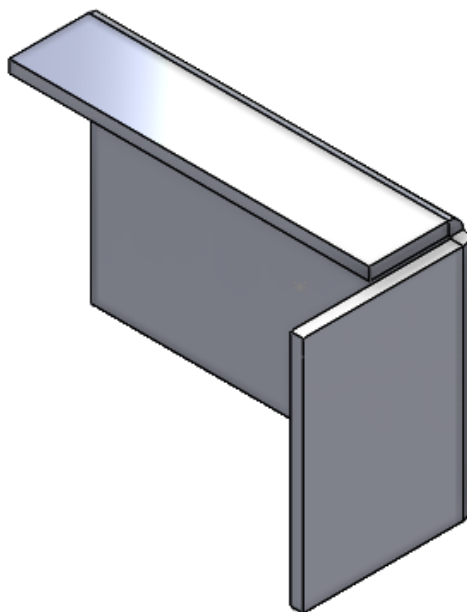
**Instances to Skip**

Skips the tab and slot instances that you select in the graphics area.


In the graphics area, pink selection orbs display on the tab and slot instances. The pointer changes to  when you hover over each instance and the coordinates of the instance appear. Click a selection orb.

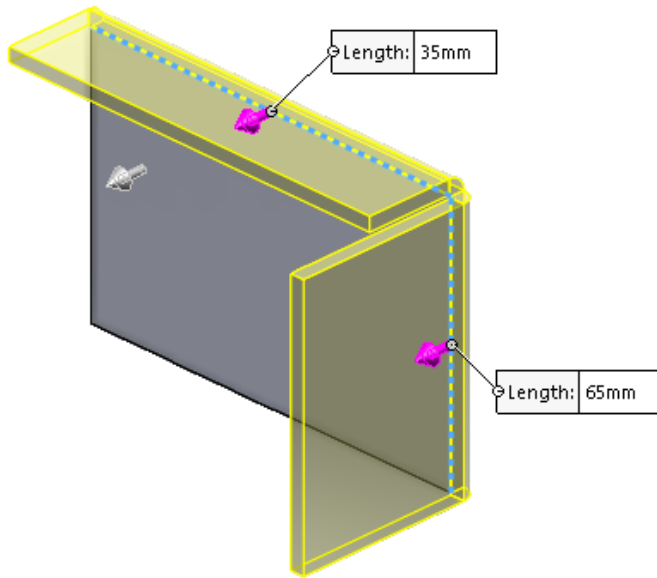
To restore a skipped instance, click the selection orb again.

## Multi Length Edge Flanges and Automatic Flange Length Dimensions



When you create edge flanges in sheet metal parts, you can create flanges with different lengths.

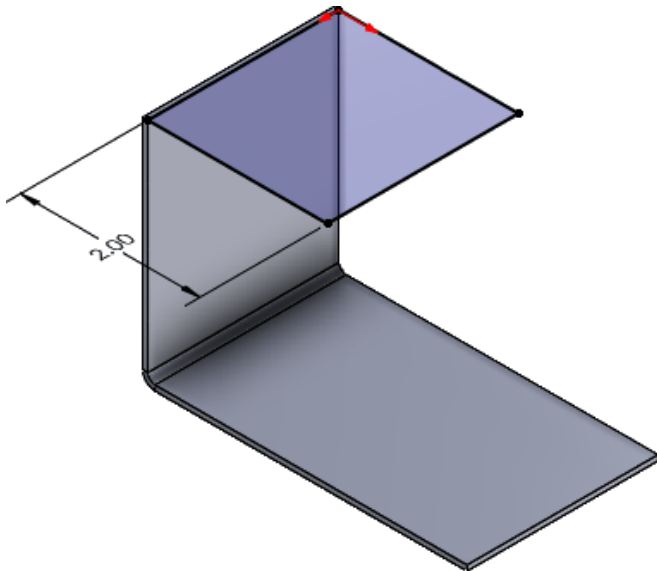
In the PropertyManager, you can select **Multi Length Flange** and specify the length of each flange in the feature. You can specify the **Length**  in the PropertyManager or in the graphics area.



In **Tools > Options > Document Properties > Sheet Metal**, under **Edge Flange Options**, you can select **Automatically add Flange Length dimension to flange profiles**.

When selected:

- SOLIDWORKS automatically adds length dimensions to all edge flange profiles
- The sketch dimension (not the feature dimension) controls the flange length



## Performance Improvements in Cosmetic Thread Features

You can experience improved performance while working with multibody parts with a large number of cosmetic thread features when you enable the **Shaded cosmetic threads** option.

For sheet metal parts with multiple cosmetic thread features, performance is improved for these operations:

- Opening parts
- Creating new features
- Editing features
- Updating and rebuilding parts

## Performance Improvements in Rebuilding Drawings

Performance is improved while working with drawings that contain drawing views of sheet metal parts with many holes and forming tools.

When working with such drawings, you can experience improved performance for:

- Opening drawing files
- Making new drawings from the sheet metal part
- Updating drawing views after making edits to the sheet metal part

# 10

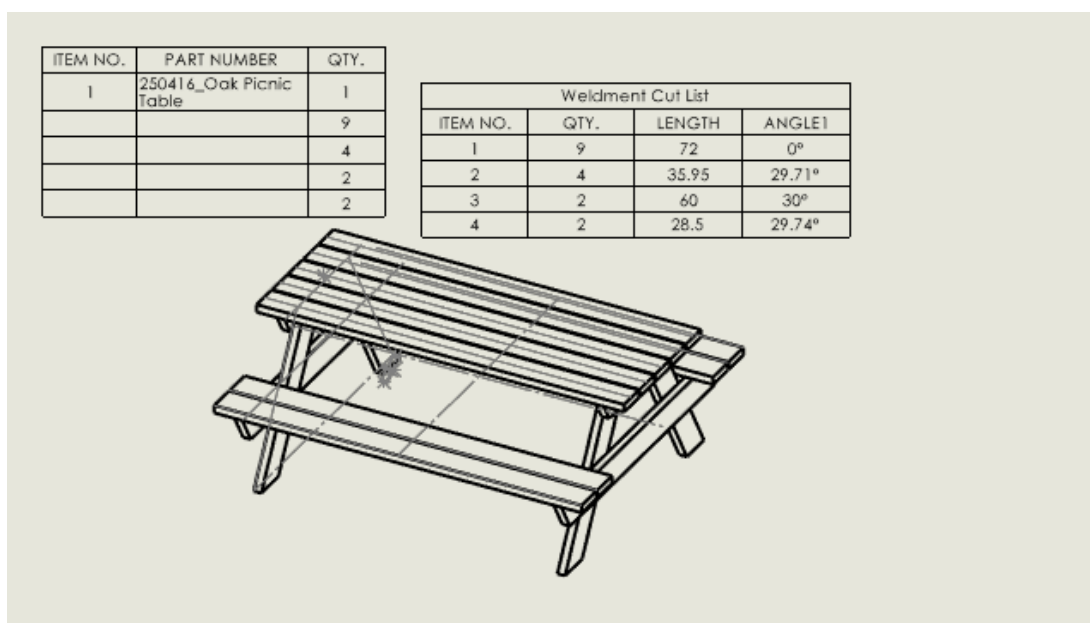
## Structure System and Weldments

---

This chapter includes the following topics:

- **Grouping Weldment Profiles and Quantities (2025 SP3)**
- **Applying Document Units to Cut List IDs (2025 SP2)**
- **Selecting a Profile Size from Design Tables and Configuration Tables (2025 SP2)**
- **Publishing Cut List Items on the 3DEXPERIENCE Platform (2025 SP1)**
- **Accessing and Working with Favorite Profiles**
- **Complex Corner PropertyManager and Structure System**
- **Trimming Attached Members**
- **Groove Beads**


### Grouping Weldment Profiles and Quantities (2025 SP3)



In the Bill of Materials (BOM) PropertyManager, you can group weldment profiles and quantities in a detailed cut list more efficiently.

**To group weldment profiles and quantities:**

1. In a drawing of a weldment or structure system, click **Bill of Materials**  (Table toolbar) or **Insert > Tables > Bill of Materials**.

2. In the Bill of Materials PropertyManager, under **BOM Type**:
  - a. Select **Indented**.
  - b. Select **Detailed cut list**.
  - c. Click **Weldment Grouping**.
3. In the Weldment Grouping dialog box, select items to group:
  - **PROFILE (Standard, Type, Size)**. Defines the structural members in a weldment model. This is a cross-sectional shape of a structural member, such as a beam or tube.
  - **UNIT OF MEASURE**. Specifies the units of measure for models in the BOM.
  - **MATERIAL**. Recognizes materials when determining if models are identical, grouping geometrically identical models with different materials into separate folders within the cut list.
  - **LENGTH**. Specifies the individual lengths of each model within a weldment, and the total length of identical models grouped together.
  - **ANGLE1**. Specifies the end face closest to the sketch profile.
  - **ANGLE2**. Specifies the end face opposite of **ANGLE1**.
  - **DESCRIPTION**. Provides details about each cut list item, such as material type and finish.
  - **PART NUMBER (SW-Part Number)**. Specifies a custom property in the cut list named SW-Part Number.
4. Click **OK**.
5. Click .

## Applying Document Units to Cut List IDs (2025 SP2)

Cut list IDs

☒ Generate Cut list IDs

Structure Cut list ID:

%Description%, %MATERIAL%, %LENGTH%, %ANGLE1%, %ANGLE2%, %Angle C

Sheet Metal Cut list ID:

%Description%, %MATERIAL%, %Bounding Box Length%, %Bounding Box Width

Generic Cut list ID:

%Description%, %MATERIAL%

☒ Apply Document Unit Settings to Cut list IDs

You can select **Apply Document Unit Settings to Cut list IDs** to apply document units to cut list IDs.

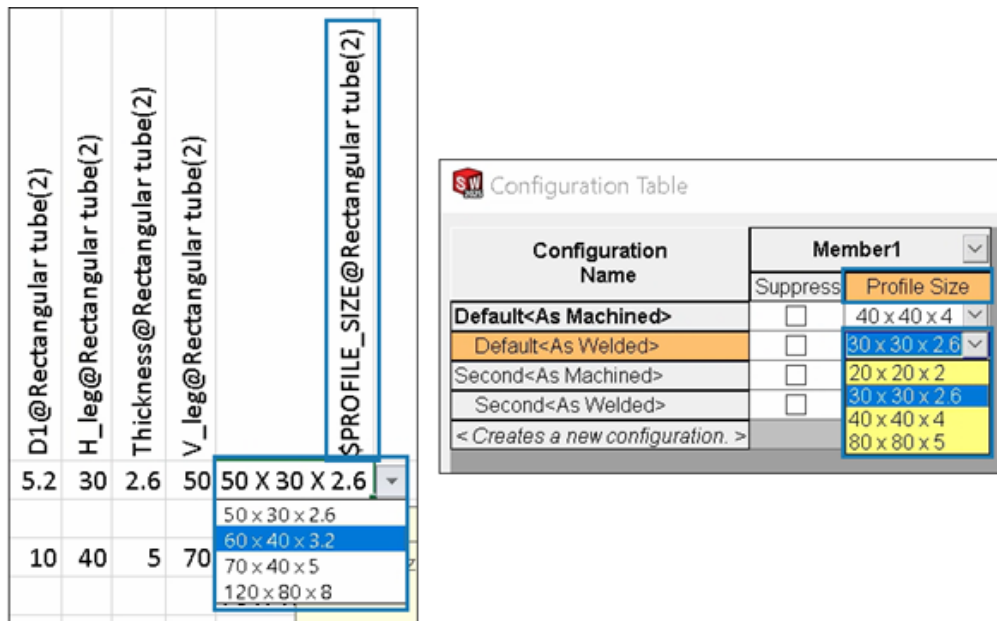
When you select this option, the units of cut list IDs are the same as document units. Earlier, the cut list IDs were in MKS units irrespective of the document units.

Click **Tools > Options > Document Properties > Weldments** and select **Apply Document Unit Settings to Cut list IDs**.

This option is available only if you select **Generate Cut list IDs**.

You can select this option for legacy files too. Units of cut list IDs change with the changes in document units.

## Selecting a Profile Size from Design Tables and Configuration Tables (2025 SP2)




For weldments and structure systems, you can select the size of the profile from design tables and configuration tables.

For the configured profiles, the design table and configuration table display the **Profile Size** column where you can select a size.

To control the profile size in design tables, the column header uses the following syntax:




- Weldments: \$PROFILE\_SIZE@feature\_name
- Structure systems: \$PROFILE\_SIZE@member\_name

### To insert a design table:

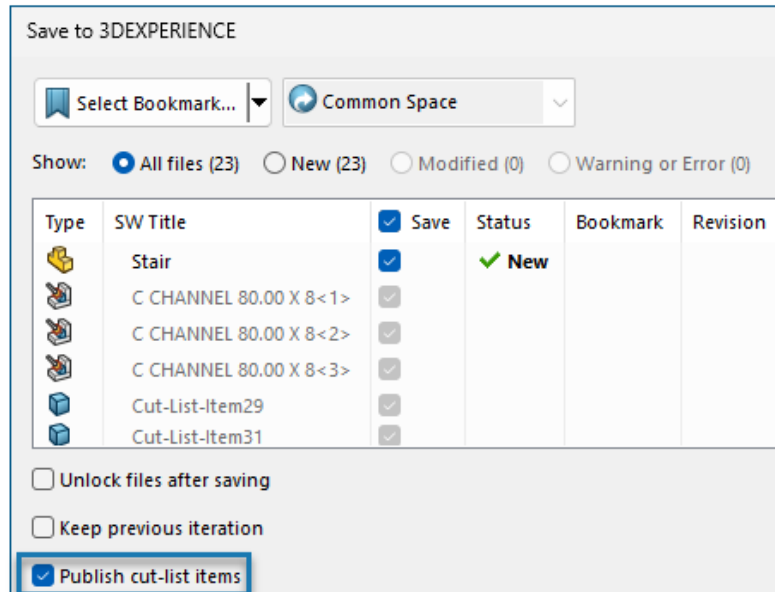
1. Open a part that has multiple configurations.
2. Click **Excel Design Table**  (Tools toolbar) or **Insert > Tables > Excel Design Table**.

You can also select the profile size by editing a design table.

**To access a configuration table:**

1. In a part that has multiple configurations, click the ConfigurationManager  tab.
2. Expand **Tables** .
3. Right-click **Configuration Table**  and click **Show Table**.

## Publishing Cut List Items on the 3DEXPERIENCE Platform (2025 SP1)



You can publish cut list items of a weldment part on the **3DEXPERIENCE** platform.

To publish the cut list items, save the SOLIDWORKS part as a weldment part to the **3DEXPERIENCE** platform. The side panel displays the extension of the weldment part as SW Weldment Part.


Prerequisites to save the SOLIDWORKS part as a weldment part:

- You must not have already saved the part on the **3DEXPERIENCE** platform.
- The part must contain a weldment feature.
- The part must be flagged as a Single Physical Product.

Prerequisites to publish cut list items on the **3DEXPERIENCE** platform:

- The part must be a weldment part.
- The cut list must be up to date.
- The cut list item property must have the CutlistID.

**To publish cut list items on the 3DEXPERIENCE platform:**

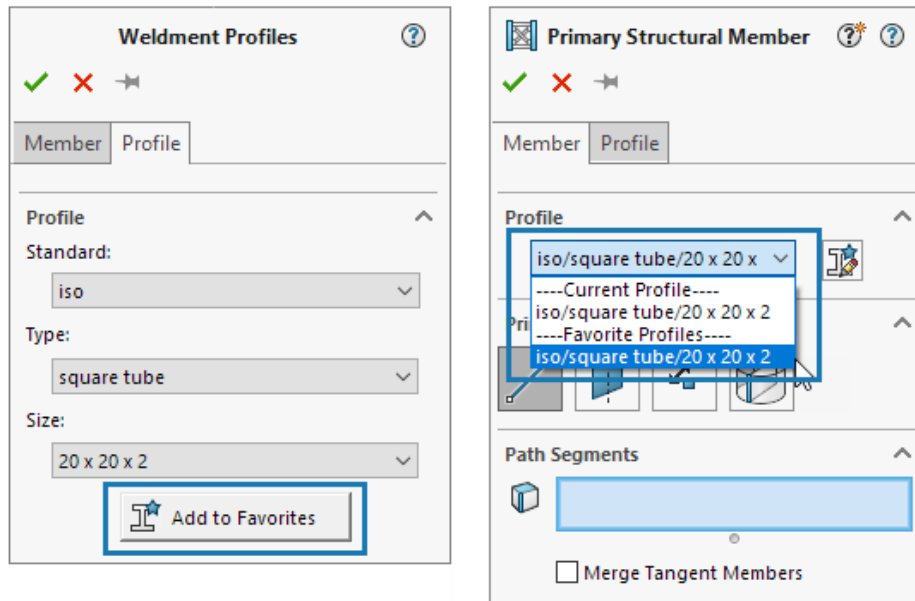
1. With a weldment part open, click **Options**  (Standard toolbar), select the Document Properties tab, and then select **Weldments**.
2. In the Document Properties - Weldments dialog box, under **Cut list IDs**, select **Generate Cut list IDs** and click **OK**.
3. In the **3DEXPERIENCE Task Pane**, right-click the part and click **Save**.

4. In the Save to 3DEXPERIENCE dialog box, select **Publish cut-list items** and click **Save**.

MySession displays the cut list items of the weldment part. The side panel displays the properties of the cut list items.



Administrators can define custom PLM attributes and mapping between CAD items and PLM items for saving attributes on the **3DEXPERIENCE** platform.

## Accessing and Working with Favorite Profiles



You can add favorite profiles in the Primary Structural Member and Secondary Structural Member PropertyManagers for quick access.

### To access and work with favorite profiles:

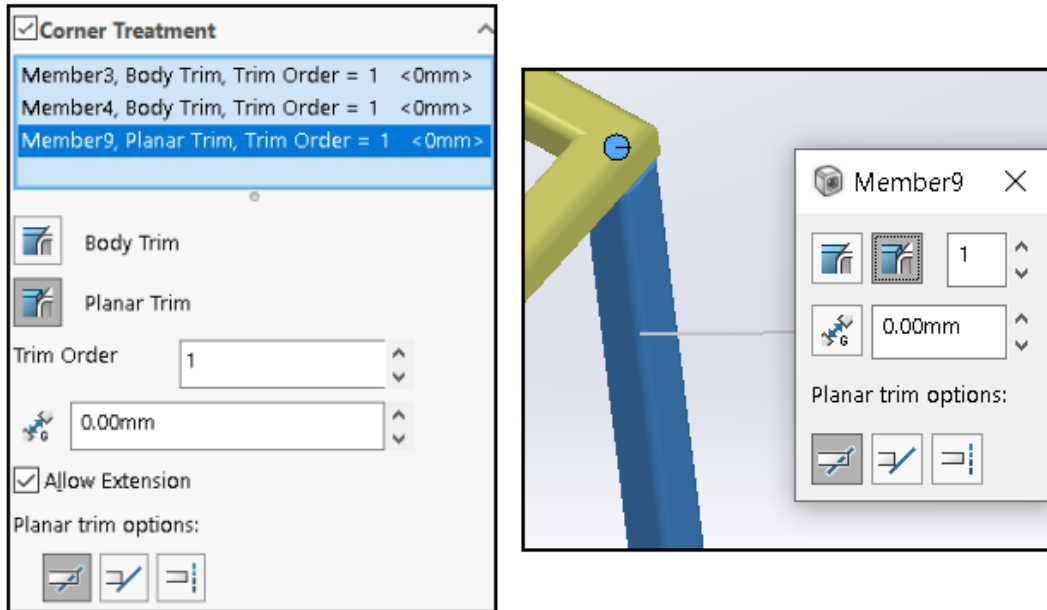
1. Open a structure system part and click the Structure System tab.
2. In the CommandManager, click **Create Structure System**.
3. In the Primary Structural Member PropertyManager, on the Profile tab, select the **Standard**, **Type**, and **Size** of the profile.
4. Click **Add to Favorites**  to add the profile as a favorite profile.  
★ as a suffix to the size indicates a favorite profile.
5. On the Member tab, under **Profile**, select the profile in **Favorite Profiles**.
6. Click  to modify the list of favorite profiles.
7. In the Favorite Profile List dialog box, select a profile and click the following:
  - **OK**. Accepts the changes, if any.
  - **Delete**. Deletes the selected profile.
  - **Move Up** or **Move Down**. Changes the sequence of profiles in the list.



## Complex Corner PropertyManager and Structure System

The Complex Corner PropertyManager provides enhanced **Corner Treatment** options. Also, you can create and edit the structure system more easily.

### Complex Corner PropertyManager



#### To open this PropertyManager:

1. Open a model that includes three or more intersecting members.
2. In the FeatureManager® design tree, expand **Corner Management**.
3. Right-click **Complex Corner Group** and select **Edit Feature**.

Enhancements include:

- Under **Corner Treatment**, the members box displays body trim members and planar trim members. You can select a member and click **Body Trim** or **Planar Trim** to change its trim type.
- Details of the selected member, such as body trim, planar trim, and trim order appear as callouts in the graphics area.
- Icons represent planar trim options.

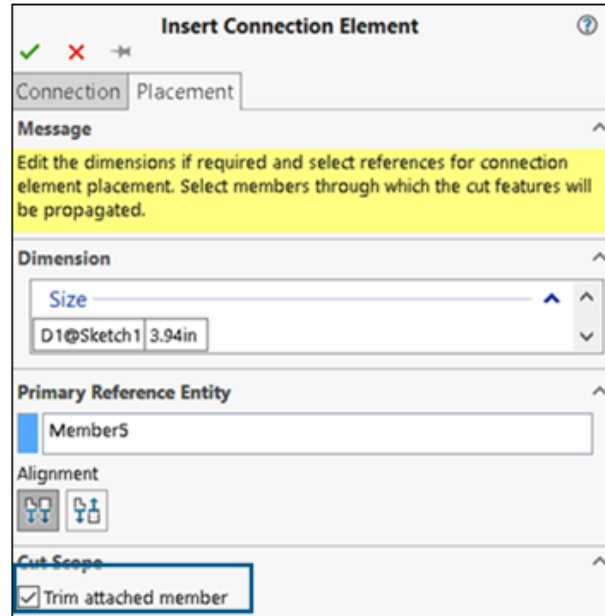
### Access to Structure System

Enhancements include:

- When you open a structure system model, SOLIDWORKS displays a message to activate the Structure System tab.
- For new files, the CommandManager displays **Create Structure System**. When you click **Create Structure System**, SOLIDWORKS displays Primary Member PropertyManager.

- For files that include a structure system, the CommandManager displays **Edit Structure System**.
- For files that include multiple structure systems, you must select the structure system to edit from the FeatureManager design tree.

## Trimming Attached Members

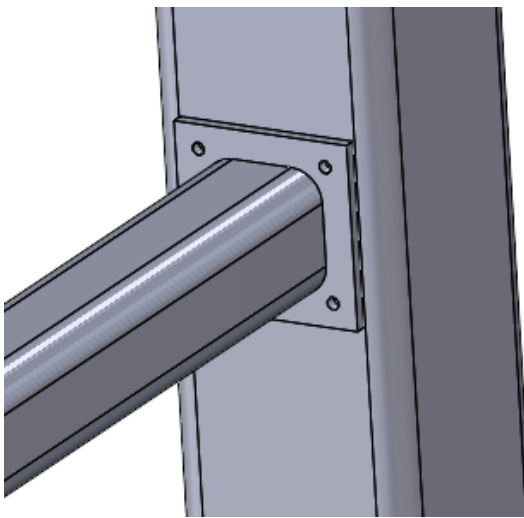


You can trim an attached member when you insert a connection element.

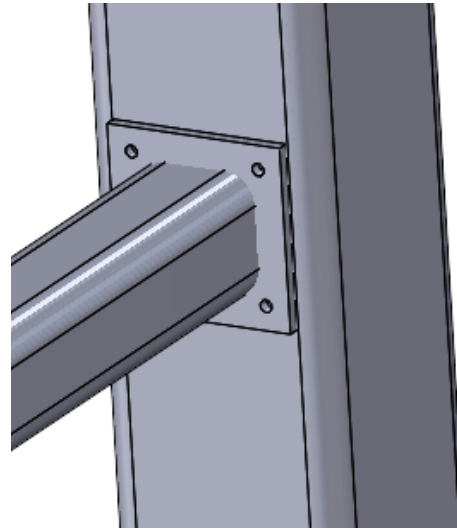
In the Insert Connection Element PropertyManager, **Trim attached member** trims the body member from its intersection point with the connection element.

### To trim attached members:

1. Open a structure system model and click the Structure System tab.
2. Click the **Insert Connection Element** tab on the CommandManager or **Insert > Structure System > Insert Connection Element**.
3. Select the connection element to insert.
4. In the PropertyManager, click the Placement tab.
5. In the graphics area, select the reference entities.
6. Select the alignment.
7. Under **Cut Scope**, select **Trim attached member**.
8. Click **✓**.

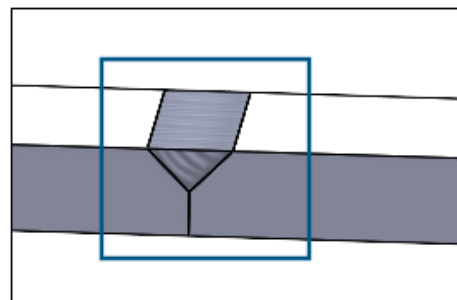
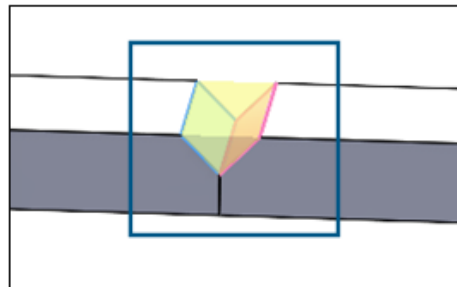
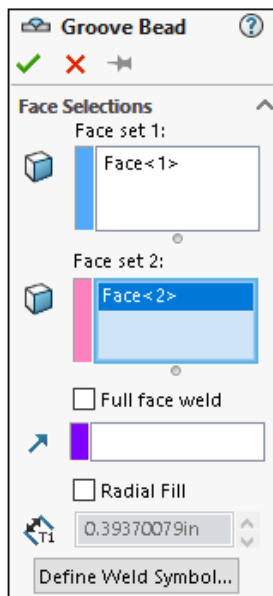


**Trim attached member** selected



**Trim attached member** cleared

## Groove Beads




You can create a groove bead to join two selected surfaces with a solid weld. SOLIDWORKS® creates a solid body in the gap based on the surfaces.

## Creating Groove Beads

You can create groove beads between the two surfaces.

### To create groove beads:

1. Open a part that has solid bodies to join.

2. Click **Insert > Weldments > Groove Bead**.
3. In the graphics area, select the faces to join.
4. Specify options in the PropertyManager and click .

## Groove Bead PropertyManager

The Groove Bead PropertyManager lets you create a solid weld between two solid bodies.

### To open this PropertyManager:

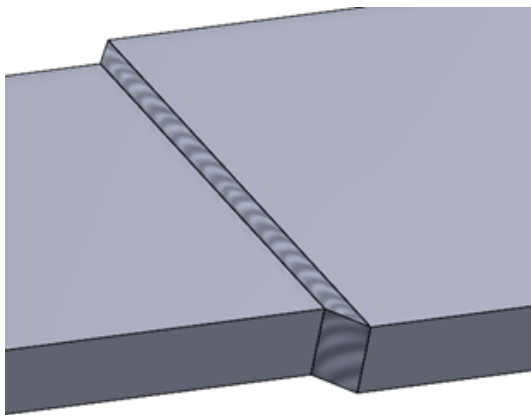
1. Open a multibody part and click **Insert > Weldments > Groove Bead**.

## Face Selections

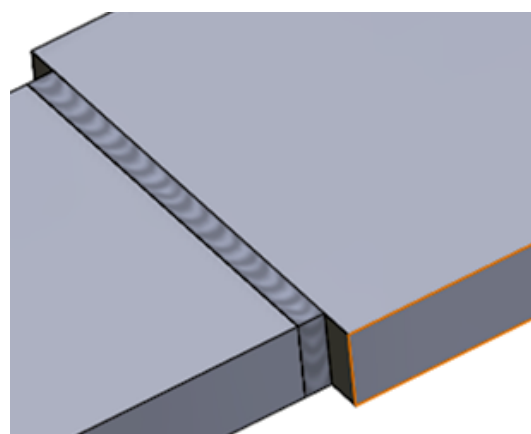
**Face set 1** and **Face set 2**. Specifies the faces of solid bodies to connect from the graphics area.

## Full face weld

Creates a weld on the entire surface. Otherwise, creates a weld on the surface where one surface projects on another.



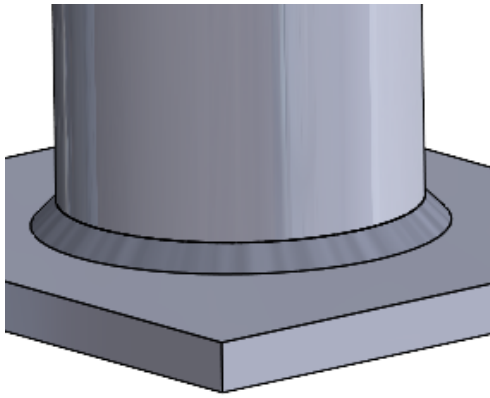
**Full face weld** selected



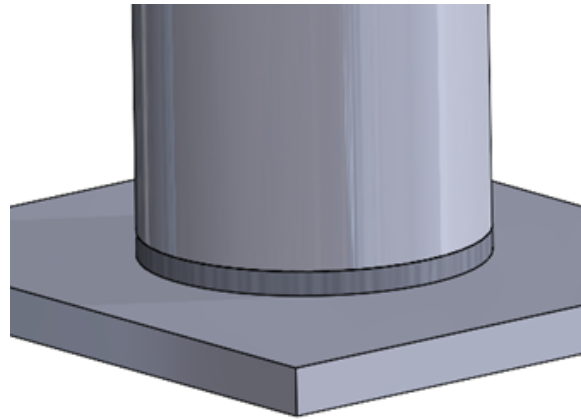
**Full face weld** cleared

## Radial Fill

Creates a weld on the surface including the radial fill distance.



**Radial Fill** selected



**Radial Fill** cleared

## Define Weld Symbol

Opens the Weld Symbol dialog box to define the weld symbol settings. The weld symbol attaches to the active weld bead.

See **Weld Symbol Properties**.

# 11

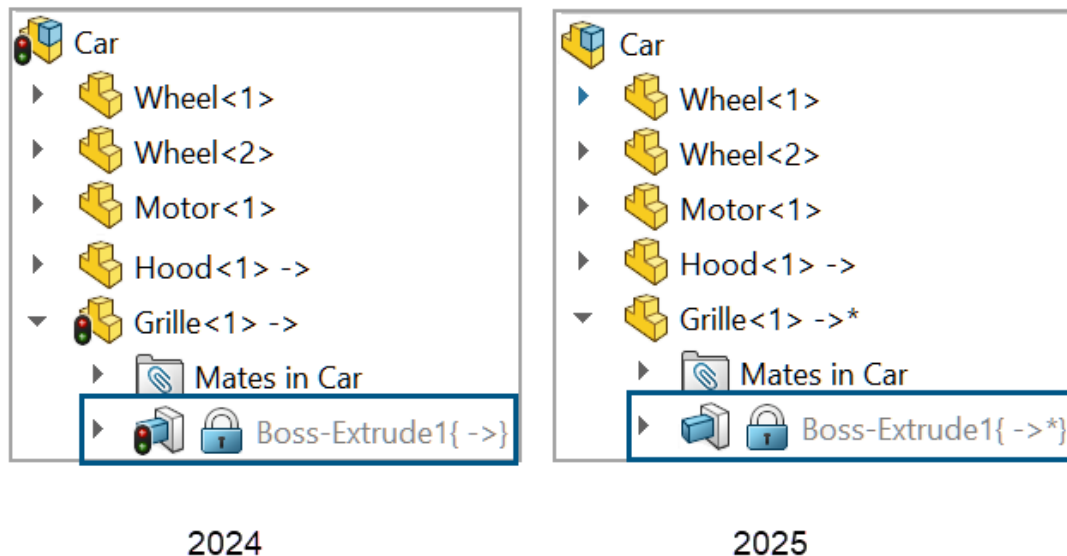
## Assemblies

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This chapter includes the following topics:

- **Locking External References in Frozen Features during a Rebuild (2025 SP3)**
- **SmartMates with AI Fastener Recognition (2025 SP3)**
- **Connected Design Libraries Included in Search for Referenced Documents (2025 SP3)**
- **Option for Automatically Resolving Lightweight Components (2025 SP2)**
- **Maintaining External References to Derived Sketches (2025 SP1)**
- **Warning When Moving Components (2025 SP1)**
- **Canceling Interference Detection Calculations (2025 SP1)**
- **Assembly Visualization**
- **SpeedPak Instances**
- **Interference Detection in Large Design Review Mode**
- **Performance Evaluation**
- **Linking Display State to the Patterned Seed Component**
- **Inserting Assemblies with Rolled-Back Features**
- **Copy with Mates**
- **Performance When Calculating Mass Properties**
- **Controlling the Visibility of Part Sketches in Assemblies**

## Locking External References in Frozen Features during a Rebuild (2025 SP3)



When you rebuild a model that has frozen features with out-of-date external references, SOLIDWORKS prompts you to lock the external references.

You can lock the external references or manage the external references as locked for the current session. SOLIDWORKS does not update locked external references during a rebuild. As a result, the model is not marked out-of-date.

Previously, the model was marked out of date because the external references were not updated.

### To lock external references in frozen features during a rebuild:

1. Open a model that has external references in a frozen feature.
2. Edit the feature referenced in the external reference.
3. Rebuild the model.

A message opens prompting you to lock the references.

4. Choose an option:

- **Lock the external reference**

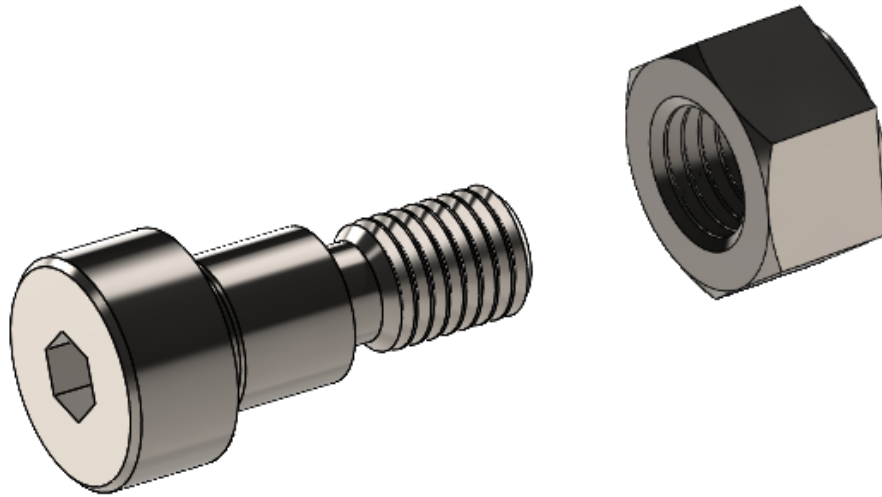
The external references of the frozen features are locked.

- **Manage the external references as locked for this session**

The external references of the frozen features are managed as locked only for this SOLIDWORKS session.

After the external references are locked, the rebuild finishes and the model is not marked out-of-date.

## SmartMates with AI Fastener Recognition (2025 SP3)






When inserting components into assemblies, SOLIDWORKS recognizes components that appear as nuts, bolts, or washers to automatically add mates to components.


SOLIDWORKS uses AI to recognize the fasteners. Automatic recognition is limited to:

- Fasteners with preview images
- Single body fasteners that are SOLIDWORKS parts
- Components to mate that are 20% or less than the geometry diameter
- Fasteners that do not have mate references and are not Toolbox parts

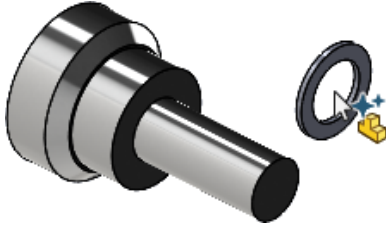
To turn on automatic recognition, in **Tools > Options > System Options > Assemblies**, select **Use AI fastener recognition to create SmartMates when inserting components**. (The option is on by default.) To turn off automatic recognition temporarily, press **ALT** while dragging a component into the assembly.


### To use SmartMates with AI fastener recognition:

1. From one of the following sources, drag a fastener (nut, bolt, or washer) onto an assembly component to mate:
  - FeatureManager® design tree (**CTRL+drag** if the component is in the same assembly)
  - **Insert Components**  (Assembly toolbar)
  - Task Pane (such as the Design Library  or File Explorer )








The pointer changes to .





2. Drop the fastener on the component to mate when the pointer changes to .

## Connected Design Libraries Included in Search for Referenced Documents (2025 SP3)

Find References		
Title	In Folder	Collaborative Space
▼  Car 2	C:\Users\User\AppData\Local\DassaultSyste	Common Space
▼  Fender	C:\Users\Public\Documents\SOLIDWORKS\	Common Space
 Car	C:\Users\Public\Documents\SOLIDWORKS\	Common Space
▼  Hood	C:\Users\Public\Documents\SOLIDWORKS\	Common Space
 Car	C:\Users\Public\Documents\SOLIDWORKS\	Common Space
▼  Grille	C:\Users\Public\Documents\SOLIDWORKS\	Common Space
 Car	C:\Users\Public\Documents\SOLIDWORKS\	Common Space

When you open a referenced document, SOLIDWORKS includes Connected Design Libraries



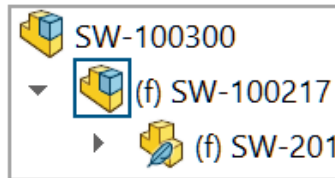
in the search for referenced documents.

In a model, you can click **File > Find References** to view the list of referenced documents that includes files from the Connected Design Library.

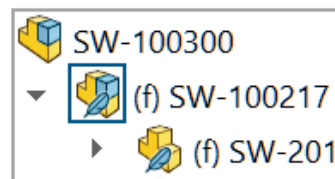
## Option for Automatically Resolving Lightweight Components (2025 SP2)



Auto-resolve lightweight components upon expansion in FeatureManager tree



Auto-resolve lightweight components upon expansion in FeatureManager tree



You can select **Auto-resolve lightweight components upon expansion in FeatureManager tree** to resolve expanded lightweight components in the FeatureManager design tree.

When you clear this option, expanded components remain in lightweight mode.

This option is applicable when **Manually manage resolved and lightweight modes** is selected.

### To automatically resolve lightweight components:

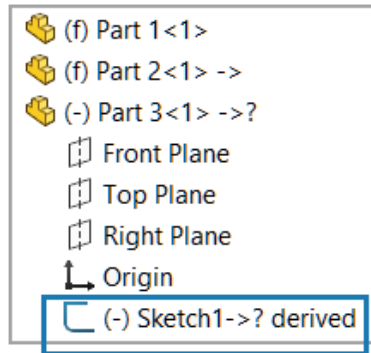
1. Click **Options > System Options > FeatureManager**.
2. Select **Auto-resolve lightweight components upon expansion in FeatureManager tree**.
3. Open a model in lightweight mode.

If lightweight mode is not available, click **Options > System Options > Performance** and select **Manually manage resolved and lightweight modes**.

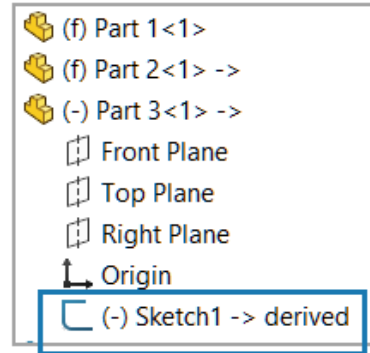
4. Expand a component.

The expanded component resolves in the FeatureManager design tree.

## Maintaining External References to Derived Sketches (2025 SP1)



2024



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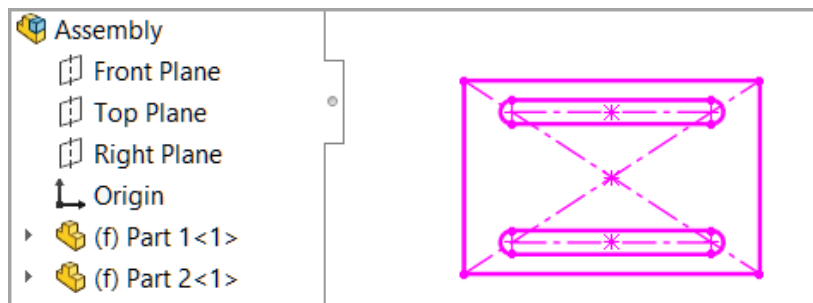
When you use **Save As Copy and Continue** to copy a part, external references to a derived sketch in the copied part are maintained.

The external references are maintained when you use Microsoft® File Explorer to copy a part with a derived sketch.

### To maintain external references to derived sketches:

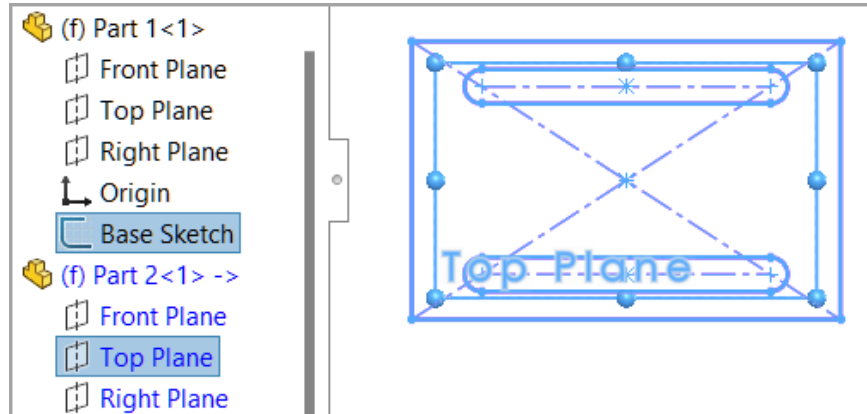
1. Open a model that contains two parts.

In this example, Part 1 shows in the graphics area.



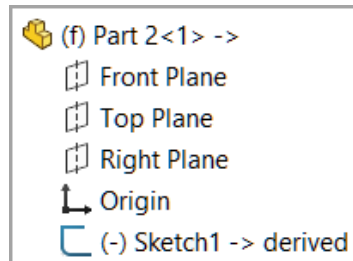
2. Create a derived sketch.


- a. Right-click Part 2 and click **Edit Part** .
- b. Press **Ctrl** and select a sketch from Part 1 and a plane from Part 2.



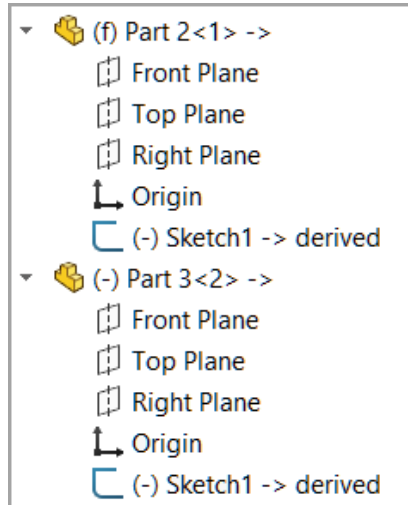
- c. Click **Insert > Derived Sketch**.
- d. Exit the edit-in-context mode by clicking in the confirmation corner.
- e. Click **File > Save All**.

Part 2 has a derived sketch from Part 1.




3. Create a copy of the part that has the derived sketch.
  - a. Right-click Part 2 and click **Edit Part** .
  - b. Click **File > Save As > Save as Copy and Continue**.
  - c. Save the new part as Part 3.
  - d. Exit the edit-in-context mode.
4. Insert the new part into the model.
  - a. Click **Insert > Component > Existing Part/Assembly**.
  - b. In the dialog box, select Part 3 and add the part.

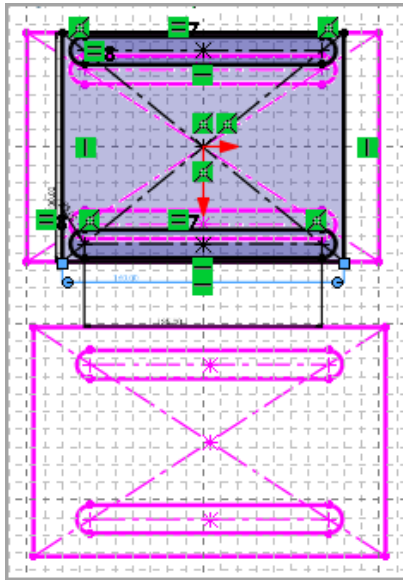
Part 2 and Part 3 have the derived sketch.



5. Update the first part.

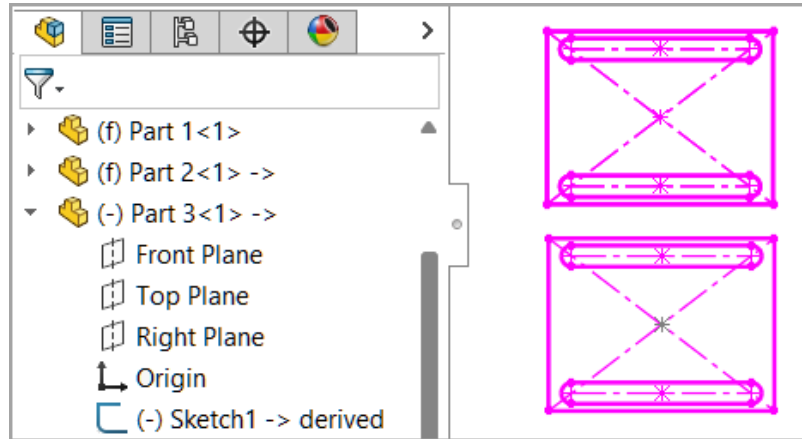
- For Part 1, right-click the sketch and click **Edit Sketch** .
- Modify a dimension.

A dimension in Part 1 changed from 200mm to 170mm.

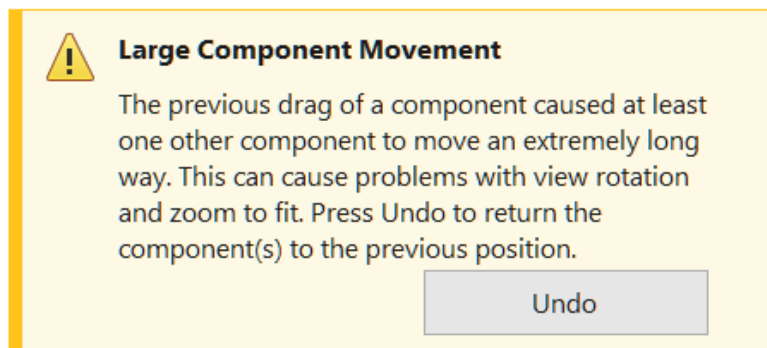


- Exit the edit-in-context mode.

Part 3 uses the updated dimension and the derived sketch remains defined.




## Warning When Moving Components (2025 SP1)



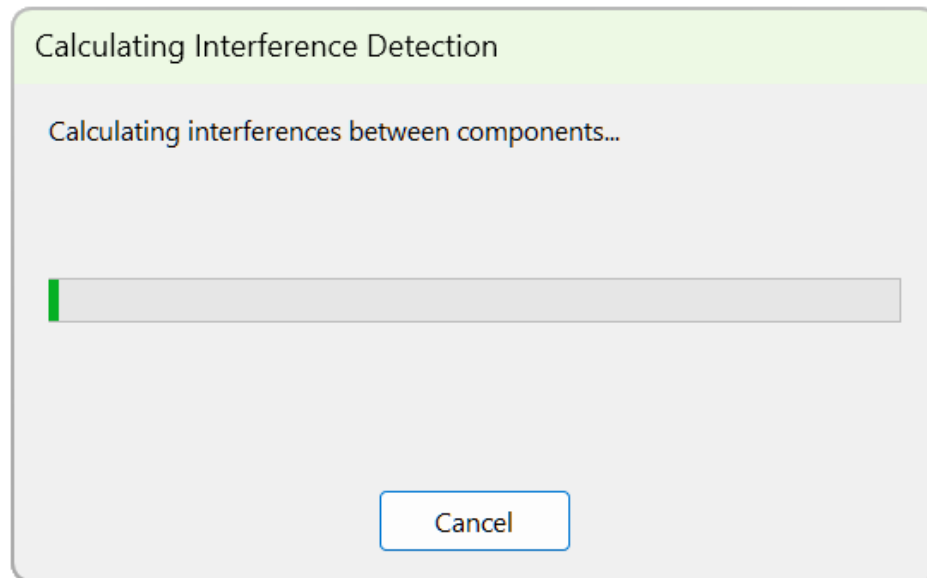
When a component moves a long distance from an assembly, SOLIDWORKS shows a warning message.

In some cases, a small drag of a component or changing mate settings can cause a component to move far away from the assembly.

The long distance between the component and the assembly can cause issues with view rotation and **Zoom to Fit** .


To return the component to the previous position, click **Undo** in the notification dialog box or click **Edit > Undo Move Component** .

## Canceling Interference Detection Calculations (2025 SP1)

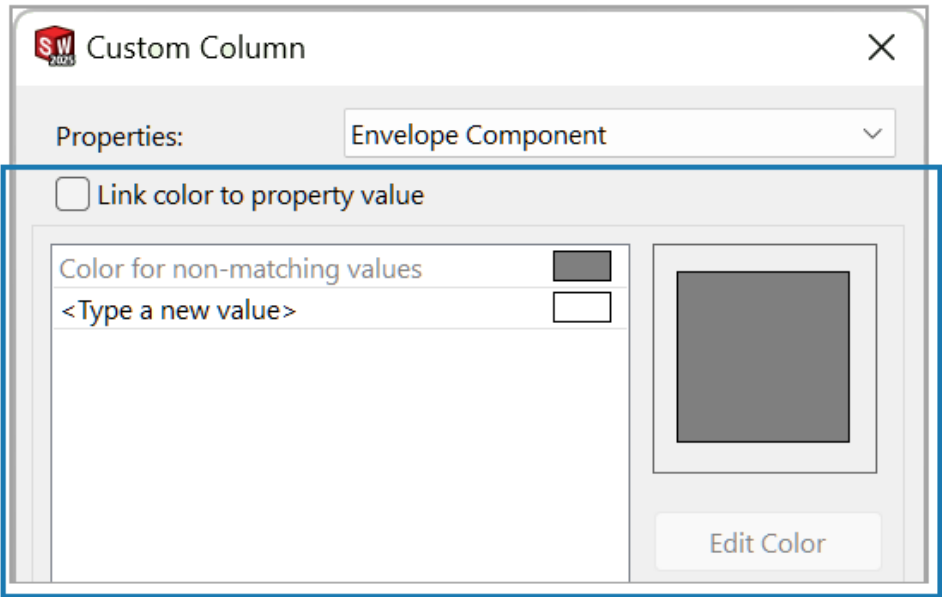


You can cancel calculations for interference detection.

**To cancel interference detection calculations:**

1. Open a large model.
2. Click **Interference Detection**  (Assembly toolbar) or **Tools > Evaluate > Interference Detection**.
3. In the PropertyManager, click **Calculate**.
4. Click **Cancel** in the dialog box or press **Esc**.

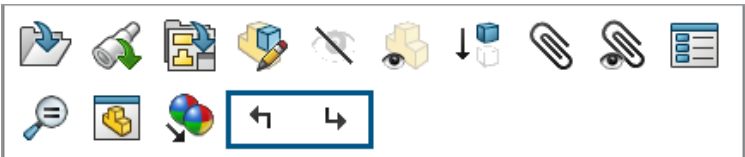
## Assembly Visualization



You can assign a color to a property value, select new properties, and roll up or roll down components.

In the Custom Column dialog box, you can select **Link color to property value** to specify a color for a component property. When you select this option, you cannot change the colors using the color slider.

In the context toolbar for a component, you can use **Roll up component** and **Roll down component** to hide components.



In the Custom Column dialog box, these properties are available:


The **3DEXPERIENCE** properties are available on the **3DEXPERIENCE** platform. For these properties, **Link color to property value** is always selected.

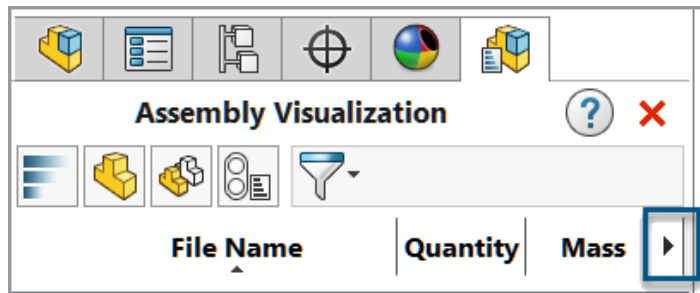
Property	Description
<b>Envelope Component</b>	Reports whether the component has an envelope component.
<b>Overridden Mass Properties</b>	Reports whether the component has overridden mass properties.



Property	Description
<b>3DEXPERIENCE - CAD Format</b>	Reports the CAD format of the component. Examples of CAD formats: <ul style="list-style-type: none"> <li>• 3DEXPERIENCE®</li> <li>• CATIAV5</li> <li>• X-CAD</li> <li>• SOLIDWORKS®</li> </ul>
<b>3DEXPERIENCE - Collaborative Space</b>	Reports the collaborative spaces where the component is saved.
<b>3DEXPERIENCE - Latest Revision</b>	Reports whether this is the latest revision of the component.
<b>3DEXPERIENCE - Lock status</b>	Reports the lock status of the component: <ul style="list-style-type: none"> <li>• Locked by me</li> <li>• Locked by other user</li> <li>• Not locked</li> </ul>
<b>3DEXPERIENCE - Maturity</b>	Reports the maturity level of the component: <ul style="list-style-type: none"> <li>• Frozen</li> <li>• In Work</li> <li>• Obsolete</li> <li>• Private</li> <li>• Released</li> </ul>
<b>3DEXPERIENCE - Updated for compatibility</b>	Reports whether the component is updated for compatibility with the 3DEXPERIENCE platform.

#### To link a color to a property value:

1. Open a model that contains components with overridden mass properties.
2. Click **Assembly Visualization**  (Tools toolbar or Evaluate tab on the CommandManager) or **Tools > Evaluate > Assembly Visualization**.
3. On the Assembly Visualization tab, click the arrow ▶ to the right of the column headers.



4. Click **More**.
5. In the Custom Column dialog box, under **Properties**, select a property like **Overridden Mass Properties**.
6. Select **Link color to property value**.
7. Double-click **Type a new value** and enter a value.
8. Click **Edit Color** and select a color for the value.

☒ Link color to property value

Color for non-matching values

Yes

<Type a new value>

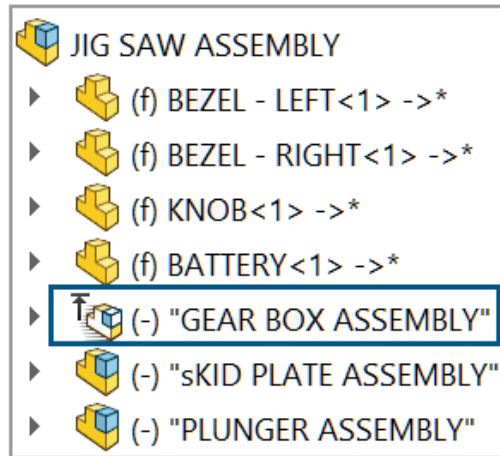
Edit Color

Delete

9. After closing the dialog boxes, on the Assembly Visualization tab, click the **Overridden Mass Properties** column header to sort the column by values.

File Name	Quantity	Overridden Mass Properties
DoorFrame	2	Yes
Column	3	No
Door	2	No

## SpeedPak Instances



You can create a SpeedPak instance from a subassembly without modifying the referenced subassembly. The SpeedPak instance is saved in the top-level assembly.

You can edit a SpeedPak instance by right-clicking the instance and clicking **SpeedPak Options** > **Edit SpeedPak**.



### Inserting a SpeedPak Instance

You can create a SpeedPak instance by adding an assembly to the model.

#### To insert a SpeedPak instance:

1. Open a model and click **Insert** > **Component** > **Insert SpeedPak Instance** .

**Insert SpeedPak Instance** is not available in Large Design Review mode.


2. In the PropertyManager, select an assembly to insert and specify options.
  3. Click **Next**  to open the SpeedPak PropertyManager and specify SpeedPak options.
- The SpeedPak instance  shows in the FeatureManager® design tree.

### Creating a SpeedPak Instance

You can create a SpeedPak instance from a subassembly that is in the model.


#### To create a SpeedPak instance:

1. Open a model that has subassemblies.
2. Right-click a subassembly and click **SpeedPak Options**.
3. Select an option: **Create Mated SpeedPak** or **Create Graphics SpeedPak**.

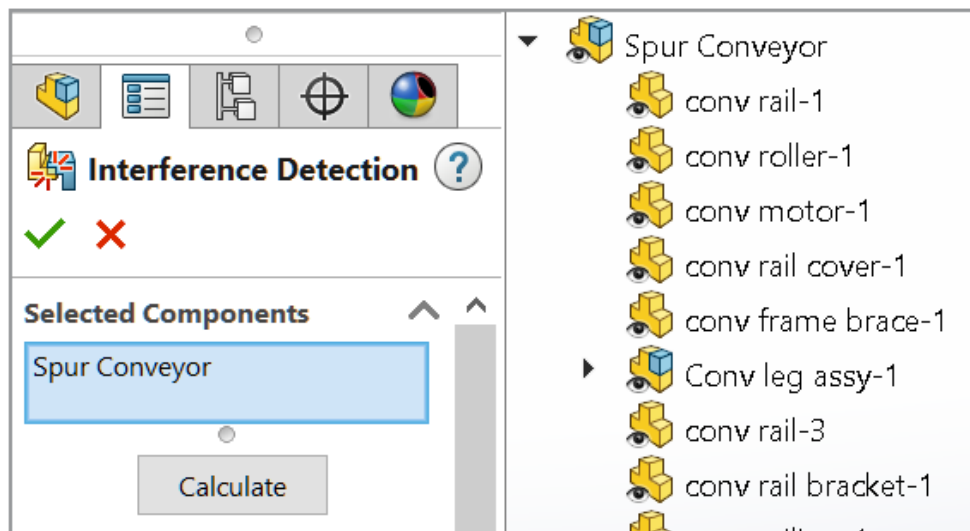
- When prompted, select **Create a SpeedPak Instance in the top level assembly**.  
The SpeedPak instance  appears in the FeatureManager design tree.

### Switching Between a SpeedPak Instance and a Parent Subassembly

To switch between a SpeedPak instance and a parent subassembly:

- In the FeatureManager design tree, right-click the SpeedPak instance  and click **SpeedPak Options > Set SpeedPak to Parent**.
- Optional: To switch back to the SpeedPak instance, right-click the subassembly and click **SpeedPak Options > Use SpeedPak**.


### Interference Detection in Large Design Review Mode



You can use interference detection on assemblies opened in Large Design Review mode.

In Large Design Review mode, the volume of interference is not available and calculations for interference detection are approximate. For accurate results, resolve the components and recalculate the interferences.

#### To use interference detection in Large Design Review mode:

- Open an assembly in Large Design Review mode.
- Click **Interference Detection**  (Large Design Review tab) or **Tools > Evaluate > Interference Detection**.
- Select options in the PropertyManager and click **Calculate**.

The following options are unavailable in the PropertyManager:

- **Create fasteners folder**
- **Create matching cosmetic threads folder**
- **Excluded Components**
- **Hide excluded components from view**
- **Ignore all smaller than**
- **Ignore hidden bodies/components**
- **Include surface bodies**
- **Remember excluded components**
- **Sort Largest to Smallest**
- **Sort Smallest to Largest**
- **Treat coincidence as interference**

## Performance Evaluation

### **Open Summary**

This assembly was last opened in 1 minutes and 7 seconds.

### **Graphics Triangles Details**

Total triangles in the assembly: 4,378,272



### **Previous Version References**

346 of 403 documents in this assembly have not been updated to the latest version of SOLIDWORKS


In the Performance Evaluation dialog box, you can see the number of outdated documents, the time required to open the assembly, and the total number of graphics triangles.

New options and information:

Options and Information	Description	Section
Time to open	Under <b>Open Summary</b> , displays the time taken to open the assembly.	Open Performance

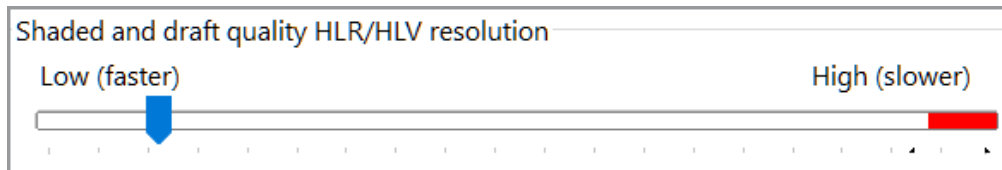
Options and Information	Description	Section
Searching for Referenced Documents	Lists the number of documents found in the <b>Referenced Documents</b> folders and the time taken to perform the search.	Open Performance
Total number of triangles in the assembly	Under <b>Graphics Triangles Details</b> , displays the total number of graphic triangles in the top-level assembly.  In the number, SOLIDWORKS uses the separator specified by the operating system to separate groups of thousands.	Display Performance
<b>Reduce Image Quality</b>	Under <b>Shaded Image Quality</b> , reduces the shaded image quality to 50% for the parts with higher image quality.  This option does not apply to subassemblies.  <div> <p>Not available for assemblies opened in lightweight mode except when the assembly has a flexible subassembly.</p> </div> <p>Clicking <b>Reduce Image Quality</b> moves the <b>Low (faster) - High (slower)</b> slider closer to the <b>Low (faster)</b> side.</p> <p>To view the slider, click <b>Tools &gt; Options &gt; Document Properties &gt; Image Quality</b>. The slider is under <b>Shaded and draft quality HLR/HLV resolution</b>.</p>	Display Performance
Time to solve mates	Under <b>Mate</b> , displays the time required to solve the mates when you rebuild the assembly.	Rebuild Performance
<b>Open and Isolate Components</b>	You can use <b>Open</b> and <b>Isolate Components</b> in the Mates dialog box.  Under <b>Mate</b> , click <b>Show These Files</b>  to open the dialog box.	Rebuild Performance
Flexible subassemblies	Lists the number of mates in the flexible subassemblies.	Rebuild Performance
Configurations Rebuilt on Save	Lists parts with more than 20 configurations that have the <b>Rebuild on Save</b> mark  .	Rebuild Performance
Statistics	Under <b>Assemblies</b> , the statistics do not include suppressed mates.	Statistics


**To use performance evaluation:**

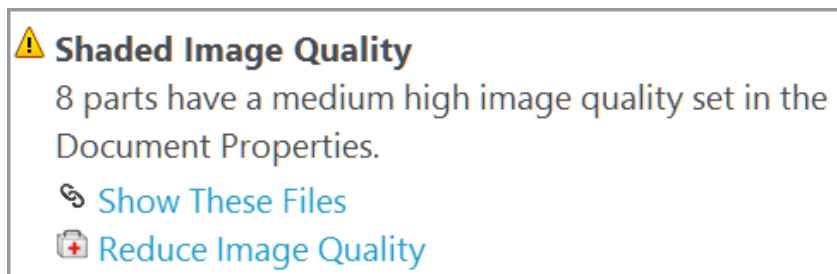
1. Open an assembly.
2. Click **Performance Evaluation**  (Evaluate toolbar) or **Tools > Evaluate > Performance Evaluation**.

**To reduce the image quality:**

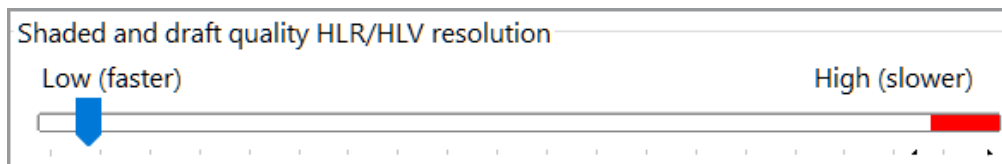
1. Open a model and click **Tools > Options > Document Properties > Image Quality**.
2. Review the slider position under **Shaded and draft quality HLR/HLV resolution**.



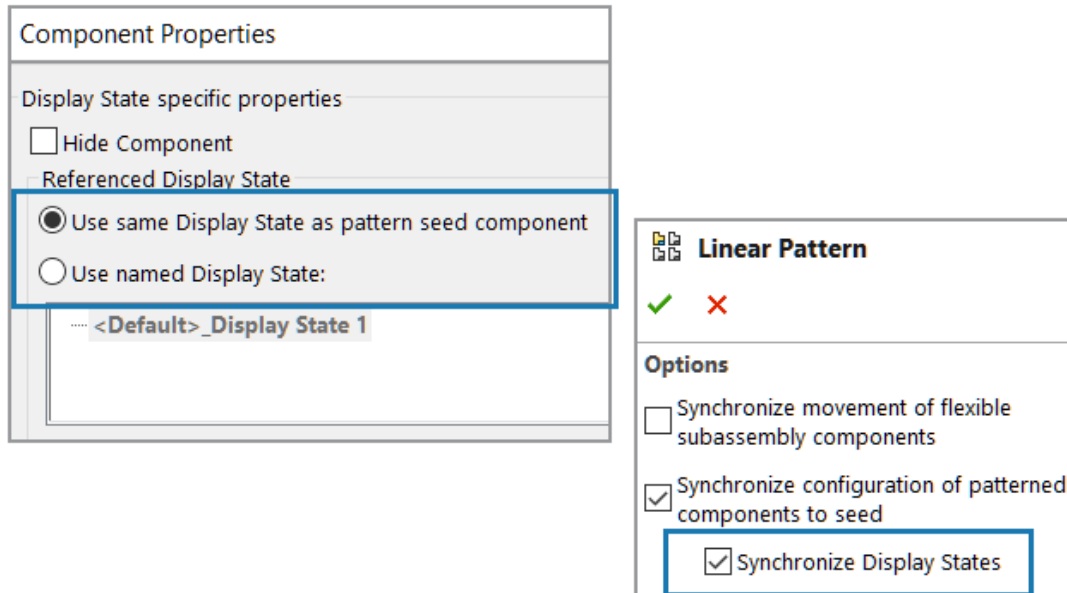
3. Click **Tools > Evaluate > Performance Evaluation**.
4. Under **Shaded Image Quality** in the **Display Performance** section, click **Reduce Image Quality** .



5. After the Performance Evaluation results update, check the slider position under **Shaded and draft quality HLR/HLV resolution**.



## Linking Display State to the Patterned Seed Component




You can link the display state of the patterned components to the patterned seed component.

Use the following options in the Component Properties dialog box to select the display state:

<b>Use same Display State as pattern seed component</b>	Links the display state of the patterned components to the patterned seed component. Disables the list of display states.
<b>Use named Display State</b>	Shows the list of display states. This option is available when the patterned component references a different configuration for the patterned seed component and the display type is a linked display state.

You can link the display state in any Component Pattern PropertyManager. In the PropertyManager, under **Synchronize configuration of patterned components to seed**, select **Synchronize display states**.

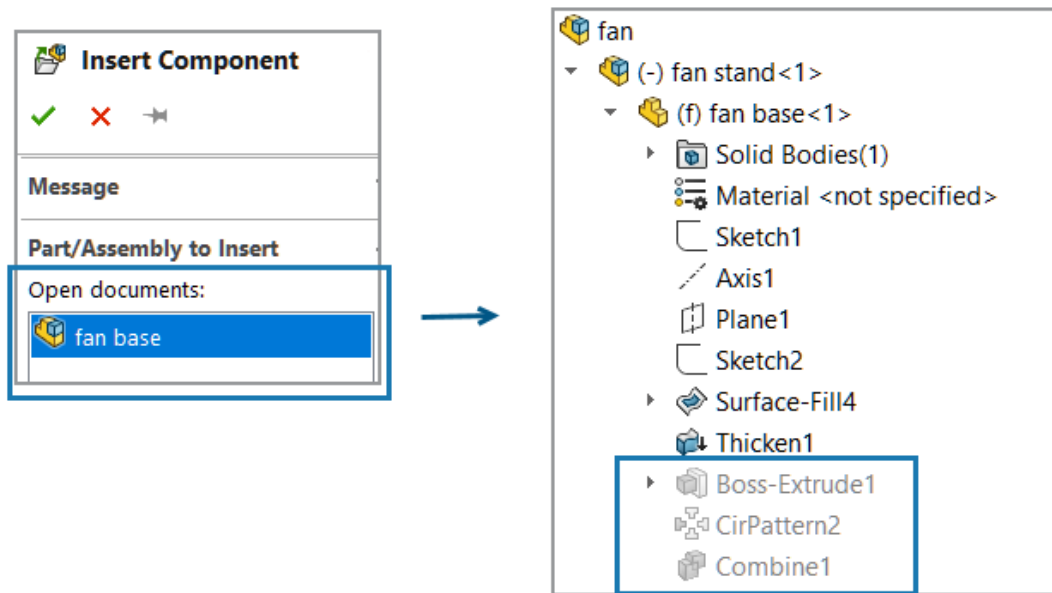
### To link the display state to the patterned seed component:

1. Open a model that contains patterned components.
2. In the FeatureManager design tree, expand a patterned component.
3. Under the expanded patterned component, right-click a component and click **Component Properties** .
4. In the dialog box, select **Use same Display State as pattern seed component**.




When **Synchronize display states** is selected in a Component Pattern PropertyManager, **Use same Display State as pattern seed component** is selected and cannot be cleared.

## Inserting Assemblies with Rolled-Back Features



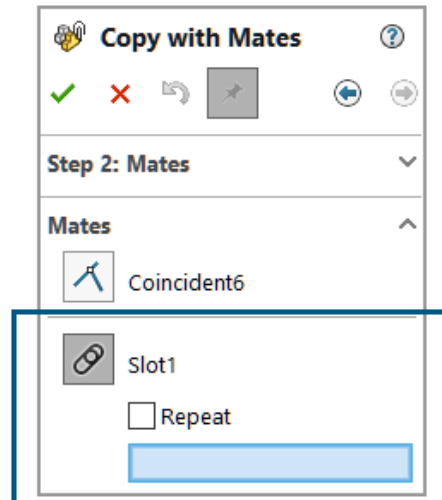
In a model, you can insert an assembly with a part reference that has rolled-back features.

### To insert an assembly with rolled-back features:

1. Open a model and click **Insert Components**  (Assembly toolbar) or **Insert > Component > Existing Part / Assembly**.
2. Select an assembly that contains a part with rolled-back features.

The assembly is added to the model.



## Copy with Mates



You can use **Copy with Mates** to copy components that have a lock mate, a path mate, a linear coupler mate, or a mechanical mate.

For hinge mates, you can copy a maximum of six hinge mates at the same time.

### To copy with mates:

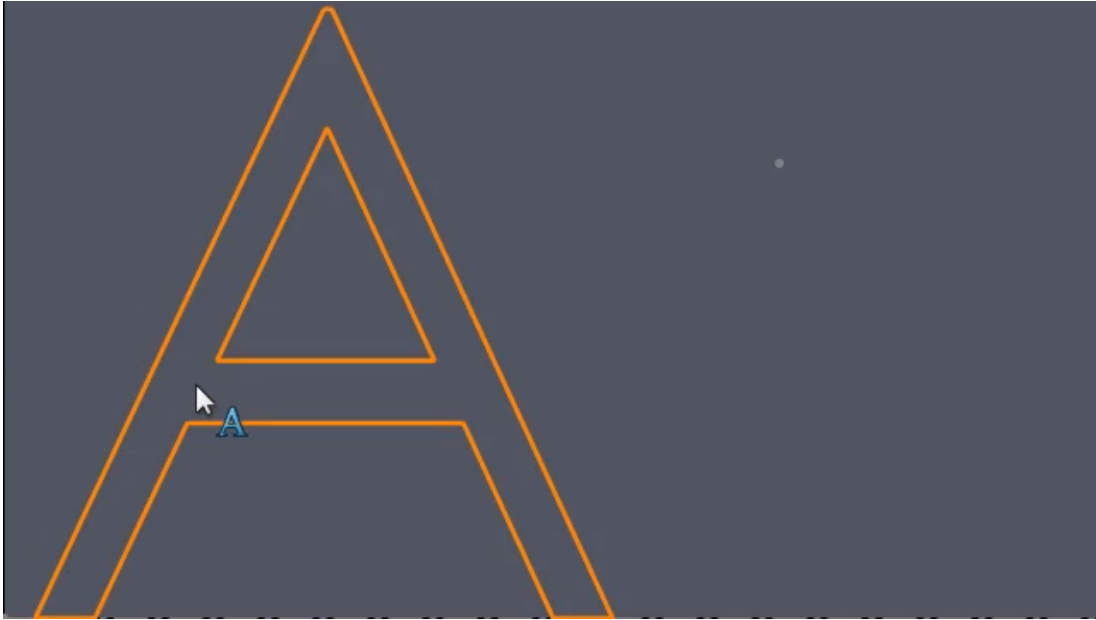
1. Open a model, and click **Copy with Mates**  (Assembly toolbar) or **Insert > Component > Copy with Mates**.
2. In the PropertyManager, select a component that has mechanical mates.
3. Click **Next** .

Under **Mates**, the mechanical mates are listed.

## Performance When Calculating Mass Properties

Performance is improved when calculating mass properties for an assembly.

## Controlling the Visibility of Part Sketches in Assemblies



You can control the visibility of part sketches in assemblies.

SOLIDWORKS maintains the visibility of sketch display states when you insert a part into an assembly. In earlier releases, the part took precedence over the sketch.

### **To control the visibility of part sketches in assemblies:**

1. Create a part with two sketches.
2. Create two display states in the part.
3. Make the sketch visibility such that one sketch is visible in one display state and the other sketch is visible in the other display state.
4. Insert two instances of the part in an assembly.
5. Make the sketch visibility such that each part display state is visible.

Each component shows the visibility of the sketch per its referred display state.

# 12

## Detailing and Drawings

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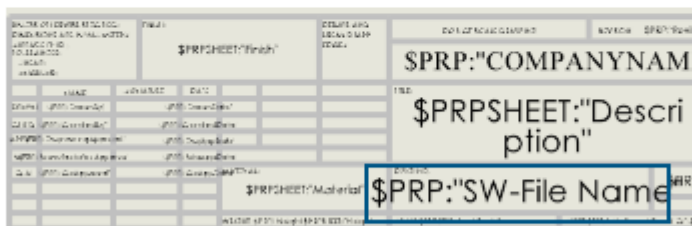
This chapter includes the following topics:

- **Hiding or Showing Annotation Text Expressions (2025 SP2)**
- **Inserting Family Tables in Drawings (2025 SP1)**
- **Creating Surface Finish Symbols in Conformance with ISO 21920 (2025 SP1)**
- **Linking Bills of Materials to Display States (2025 SP1)**
- **Creating Flattened BOMs (2025 SP1)**
- **Auto-Generate Drawings (BETA) (2025 SP1)**
- **Additional Tolerance Types for Chamfer Dimensions**
- **BOM Quantity Override for Detailed Cut Lists**
- **Reloading Drawings**
- **Exporting Drawing Views as Blocks to DXF/DWG Files**
- **Inserting and Viewing Cosmetic Threads in Assembly Drawings**

### Hiding or Showing Annotation Text Expressions (2025 SP2)



Hide



Show

You can hide or show annotation text expressions on drawing sheets.

Annotation text expressions are placeholder text linked to a custom property. This option is a quick way of seeing the properties that link to a note.

**To hide or show annotation text expressions:**

1. Click **View > Hide/Show > Annotation Text Expression**.

## Inserting Family Tables in Drawings (2025 SP1)

Family Table															
ITEM NO.	PART NUMBER	DESCRIPTION	A	B	D	D1	D8	D9	D10	C	D14	D16	D5	E	
1	Default		Ø 40	Ø 20	70	80	80	118.79	30	3	80	15	22	36	
2	B01001		Ø 40	Ø 20	70	80	80	118.79	30	3	80	15	22	36	
3	B02001		Ø 41	Ø 20.5	71	80	80	120.59	30	3	80	15	22	37	
4	B03001		Ø 42	Ø 21	72	80	80	122.39	30	3	80	15	22	38	
5	B04001		Ø 43	Ø 21.5	73	80	80	124.19	30	4	80	15	22	39	
6	B05001		Ø 44	Ø 22	74	80	80	126	30	4	80	15	22	40	
7	B06001		Ø 45	Ø 22.5	75	80	80	127.81	30	4	80	15	22	41	
8	B07001		Ø 46	Ø 23	76	80	80	129.62	30	5	80	15	22	42	
9	B08001		Ø 47	Ø 23.5	77	80	80	131.44	30	5	80	15	22	43	
10	B09001		Ø 48	Ø 24	78	80	80	133.25	30	5	80	15	22	44	
11	B10001		Ø 49	Ø 24.5	79	80	80	135.07	30	5	80	15	22	45	




You can use the **Family Table** command to insert configuration data in drawings.

You can specify the table parameters in **Tools > Options > Document Properties > Tables > Family**. You can specify the family table template location in **Tools > Options > System Options > File Locations > Show folders for > Family Table Templates**.

If you double-click a family table cell to edit it, the software prompts you to keep the link and have the external model inherit the changes, or break the link to override the value. You can restore the broken link by clearing the cell.

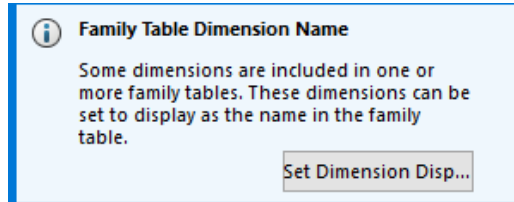
**Benefits:** You can quickly insert configuration data in drawings without needing any workarounds. Family tables display variations in part and assembly configurations or custom properties in a table in SOLIDWORKS drawings.

**To insert family tables in drawings:**

1. In a SOLIDWORKS drawing, click **Insert > Tables > Family Table** .
2. In the PropertyManager:
  - a. Select the file from which to create the family table and click **Next** .
  - b. Specify options to define the family table.
  - c. Click .
3. Click in the drawing file to place the table.

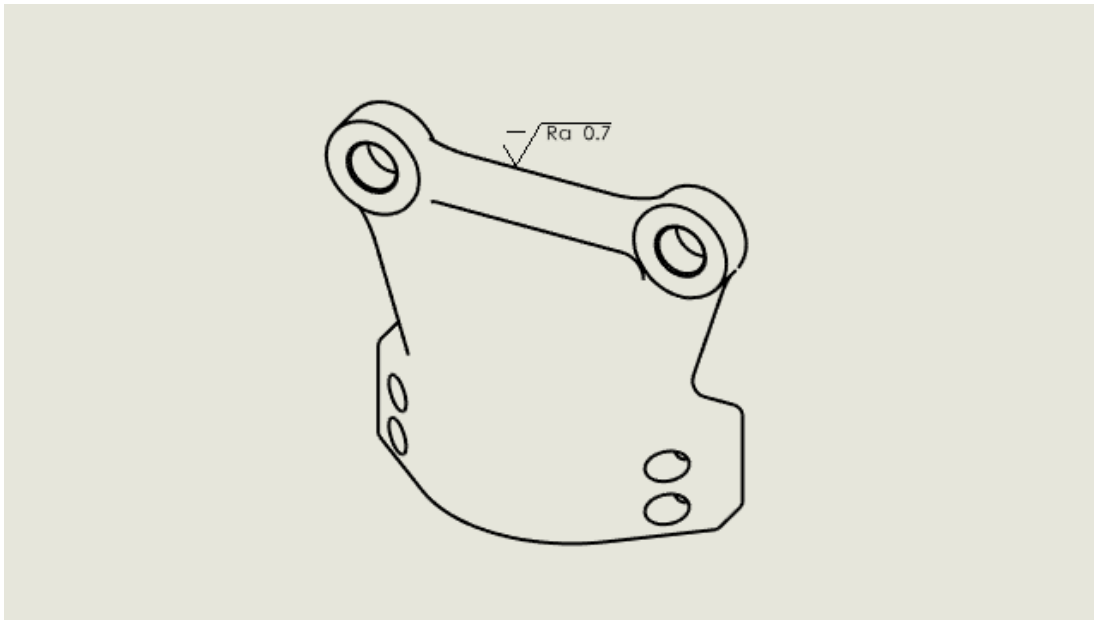
In the dimension column menu or dialog box for family tables, specify **Dimension Name** for the column name. The name is specific to the drawing. Changes to the column name apply to all family tables in the drawing referencing the same driving dimension.

4. Optional: If you click **Insert > Model Items** and insert items into the drawing, the **Family Table Dimension Name** notification appears. Click **Set Dimension Display** to display the inserted items using the dimension name from the family table.





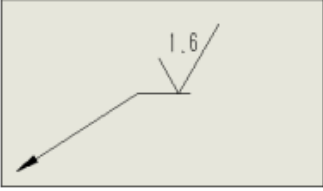
To switch the display of inserted items in the drawing, you can also select the items to open the Dimension PropertyManager. On the Value tab, under **Family Table Dimension Name**, select **Display as name in Family Table** to display the items using the names from the table. Clear the option to display the items using their values.

## Creating Surface Finish Symbols in Conformance with ISO 21920 (2025 SP1)



You can insert surface finish symbols that comply with the latest ISO standards including ISO 21920-1, ISO 1302:202, and ISO 1302:1992.

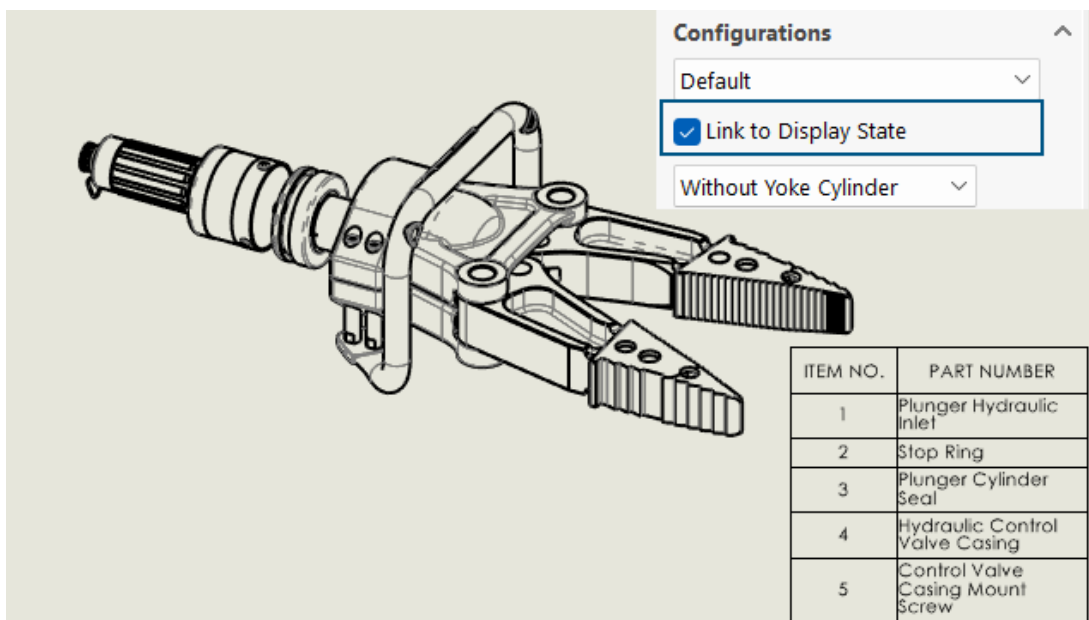
You can use the following symbols.

	<b>21920-1</b>
	<b>1302 (2002)</b>
	<b>1302 (1992)</b>

**To create surface finish symbols in conformance with ISO 21920:**

1. In a drawing, click **Tools > Options > Document Properties > Annotations > Surface Finishes**.
2. In the dialog box in Surface symbol standard, select a standard and click **OK**.

## Linking Bills of Materials to Display States (2025 SP1)



In the Bill of Materials (BOM) PropertyManager, you can link a BOM to display states.

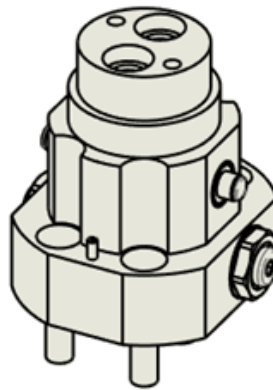
**Benefits:** You can see items in the BOM only for components that are visible in the view.

**To link BOMs to display states:**

1. In the Bill of Materials PropertyManager, under **Configurations**, select **Link to Display State**.
2. Click ▼ and select a display state.
3. Click ✓.

## Creating Flattened BOMs (2025 SP1)

ITEM NO.	PART NUMBER	QTY.
1	Valve Block	1
2	Check Valve	1
3	Check Valve Center Shaft	1
4	Piston Inlet Valve Washer	4
5	Piston Inlet Valve O-Ring	3
6	Check Valve Body	2
7	Piston Inlet Valve Ball	4
8	Check Valve Outer Washer	2




In the Bill of Materials PropertyManager, you can flatten a BOM to display the total quantities for all components.

**Benefits:** Flattened BOMs save time by calculating the total number of quantities of the components.

The flattened BOM displays the:

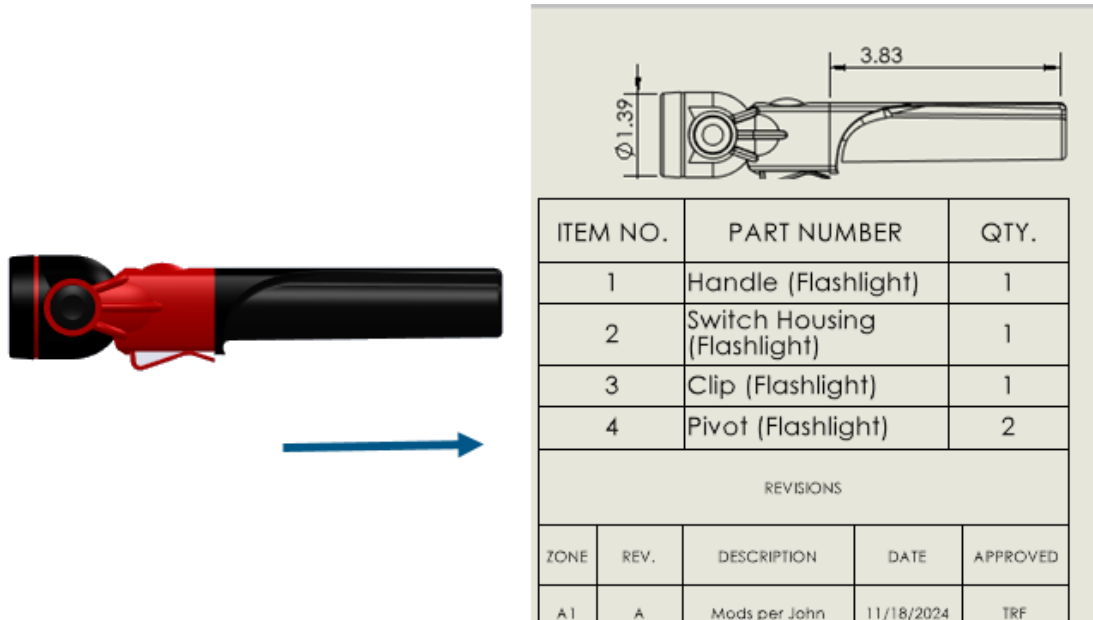
- Model as a list of components without indentation.
- Component only once if it exists at multiple levels of the model.
- Total quantity of the component by adding the quantities of every component.

**To create flattened BOMs:**

1. In a drawing, click **Bill of Materials**  (Table toolbar) or **Insert > Tables > Bill of Materials**.
2. In the PropertyManager, for **BOM Type** select **Flattened**.
3. Click ✓.



## Auto-Generate Drawings (BETA) (2025 SP1)



**3DEXPERIENCE** users can auto generate drawings of parts and assemblies.

**Benefits:** Automatically generating drawings reduces errors and time spent on repetitive tasks.

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### Auto Generating Drawings (BETA)

**3DEXPERIENCE** users can automatically generate drawings of parts and assemblies.

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#### To auto generate drawings:

1. Do one of the following:
  - Click **File** > **Auto-Generate Drawing** (BETA).
  - In the FeatureManager design tree or graphics area, right-click a part, subassembly, or assembly, and click **Auto-Generate Drawing** (BETA).
2. Optional: To select multiple part or assembly components, do one of the following:
  - In the FeatureManager design tree or graphics area, **Ctrl** + select components and click **File** > **Auto-Generate Drawing** (BETA).
  - In the Auto-Generate Drawings (BETA) task pane, click **Edit**.

3. In the PropertyManager, specify options and click .


### Auto-Generate Drawing (BETA) PropertyManager

In the Auto-Generate Drawing (BETA) PropertyManager, you can select parts or assemblies to automatically generate a drawing.

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#### To open this PropertyManager:

In a part or assembly, click **File > Auto-Generate Drawing (BETA)**.

<b>Selected components</b>	Specifies the components to include in the auto-generated drawing.
<b>Title</b>	Specifies a title for the auto-generated drawing.
 <b>Reset to filename</b>	Resets the title of the drawing to the part or assembly file name.
<b>Save location</b>	Specifies a folder to save the auto-generated drawing.
<b>Same as parent part/assembly</b>	Saves the auto-generated drawing in the same folder as the component selected for the drawing generation.

### Tasks (Auto-Generate Drawings) (BETA) Tab

The Tasks (Auto-Generate Drawings) (BETA) tab lists the generated drawings and their progress.

The **Auto-Generate Drawings** (BETA) tool is available to **3DEXPERIENCE** users only.

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#### To open this tab:

In a part or assembly, click **Tasks (Auto-generate drawings)** (BETA) from the Task Pane.

<b>Title</b>	Displays the name of the generated drawing.
--------------	---

## Status

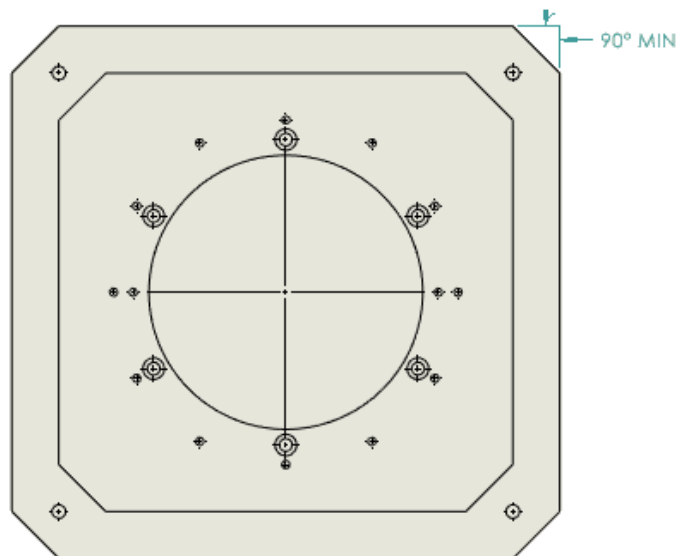
Displays the status of the drawing generation.

-  **Completed**
-  **Failed**

## Actions

- **Cancel.** (Available during drawing creation.) Cancels the auto-drawing generation for the selected item.
- **Open.** (Available after completion of drawing.) Opens the selected drawing in Detailing mode.
- **View Details.** (Available if drawing creation fails.) Opens a report to show why the auto-generated drawing failed.
- **Queue Next.** Moves the task to the next in the queue. SOLIDWORKS does not cancel the current task in progress. If you click **Queue Next** on another task, SOLIDWORKS moves the task to the next in the queue.
- Right-click any row in the Tasks tab to:
  - **Clear.** Clears the selected row from the list.
  - **Clear All.** Clears all the rows from the Tasks tab except for the rows in progress. This includes rows where the status is **Completed** or **Failed**.

## Additional Tolerance Types for Chamfer Dimensions

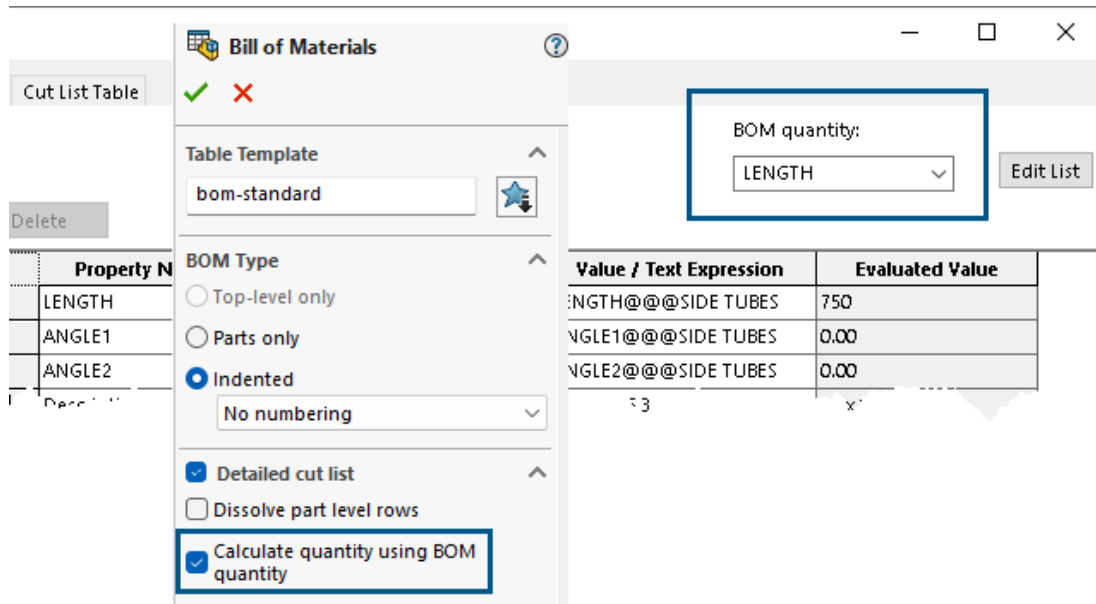


You can specify **MIN**, **MAX**, **Limit**, **Fit**, and **Fit with Tolerance** tolerance types for chamfer dimensions in drawings.

To access additional tolerance types for chamfer dimensions:

1. Click **Tools > Options > Document Properties > Dimensions > Chamfer**.
2. In the Document Properties - Chamfer dialog box, click **Tolerance**.
3. In the Chamfer Dimension Tolerance dialog box, in **Tolerance type**, select a tolerance and click **OK**.

## BOM Quantity Override for Detailed Cut Lists



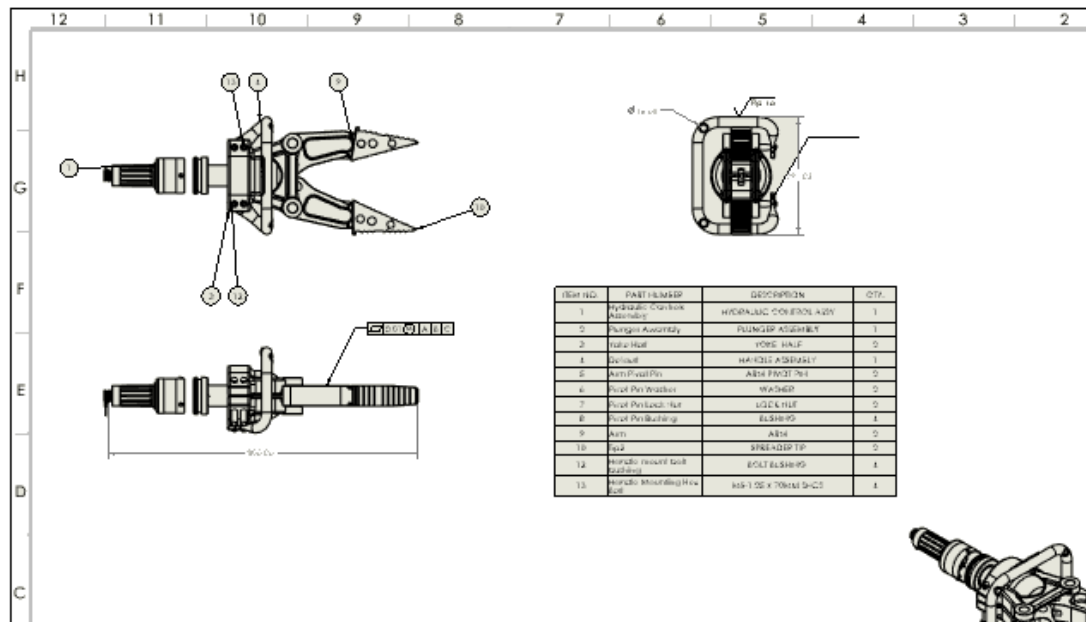
In the Bill of Materials PropertyManager, you can choose an option for **Detailed Cut List** to use the BOM quantity in weldments.

When you select the option, **Calculate quantity using BOM quantity**, the software takes the property that you select in the **BOM quantity** list and uses the value as the multiplier. If you clear the option, the BOM table displays the quantity as the number of instances.

**To use the BOM quantity override for detailed cut lists:**

1. Click **Bill of Materials** (Table toolbar) or **Insert > Tables > Bill of Materials**.
2. In the PropertyManager, select **Detailed Cut List** and **Calculate quantity using BOM quantity**.
3. Click **✓**.

## Reloading Drawings



You can reload SOLIDWORKS drawings. This is useful in multi-user environments if you have read-only access and want the latest version with changes made by another user.

This enhancement was first included in SOLIDWORKS 2024 SP2 but not fully documented at that time. We include it here to promote full customer awareness of the enhancement.

**Benefits:** You can use reload to undo changes since the last save operation. You can reload the latest version of a document, especially if you have read-only access and another user has made changes.

### To reload drawings:

1. In a drawing, click **File > Reload**.

## Exporting Drawing Views as Blocks to DXF/DWG Files

You can export drawing views as blocks to .dxf or .dwg files.

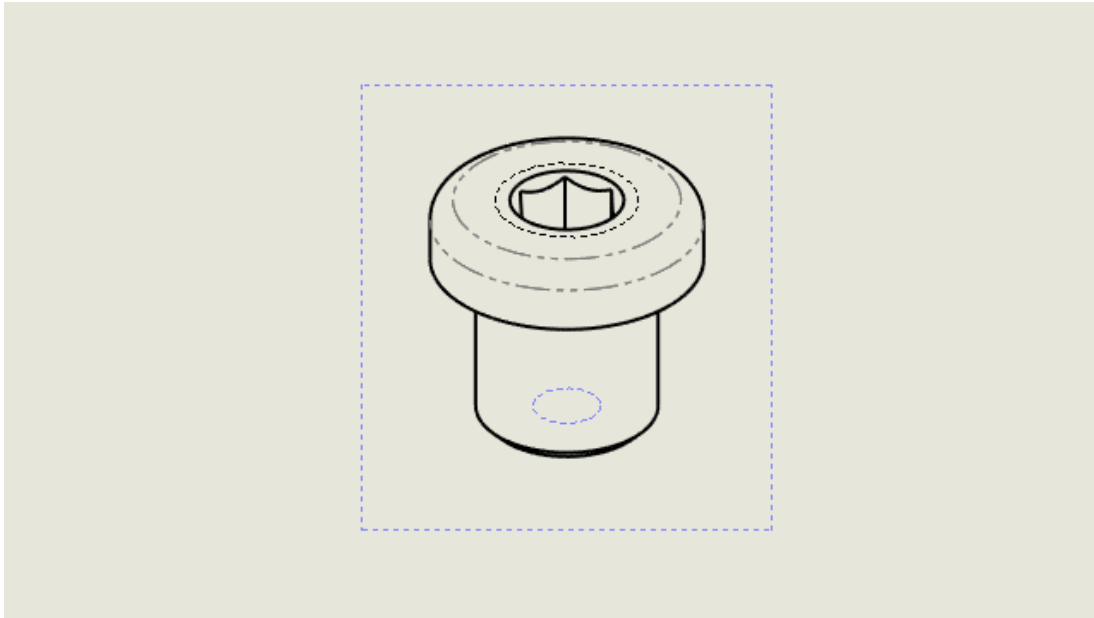
This enhancement was included in SOLIDWORKS 2024 SP2 but not fully documented at that time. We include it here to promote full awareness of the enhancement.

**Benefits:** Grouping related geometry into blocks helps organize drawings and makes it easier to navigate and manage complex designs.

### To export drawing views as blocks to DXF/DWG files:

1. In a drawing, click **Tools > Options > System Options > Export**.
2. Under **File Format**, select **DXF/DWG**.
3. Click **OK**.

## Inserting and Viewing Cosmetic Threads in Assembly Drawings



You can insert and view cosmetic threads in assembly drawings.

This enhancement was first included in SOLIDWORKS 2024 SP2 but not fully documented at that time. We include it here to promote full customer awareness of the enhancement.


**Benefits:** You have more control over whether or not you want to insert and view cosmetic threads in assembly drawings.

Previously, when you inserted cosmetic threads into an assembly, you did not see the cosmetic threads in drawings automatically. You had to click **Insert** > **Model Items** > **Cosmetic Thread** to see the cosmetic threads.

### To insert cosmetic threads into assembly drawings:

1. Click **Tools** > **Options** > **Document Properties** > **Detailing**.
2. Under **Auto insert on view creation**, select **Cosmetic Threads - assembly (may affect performance)**, and click **OK**.

### To import cosmetic threads in assembly drawings:

1. In the Drawing View PropertyManager, under **Import options**, select **Import annotations** and **Cosmetic threads**.
2. Click .

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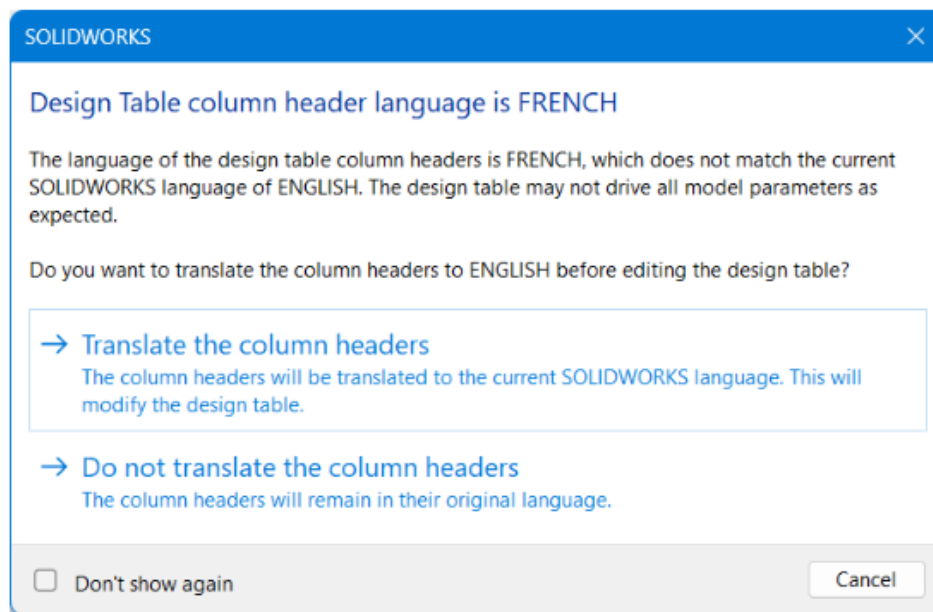
## Configurations

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This chapter includes the following topics:

- **Translating Design Table Column Headers (2025 SP2)**
- **Display State Tables**

### Translating Design Table Column Headers (2025 SP2)




You can automatically translate a design table's column headers into the current SOLIDWORKS language. This functionality is supported by all SOLIDWORKS languages.

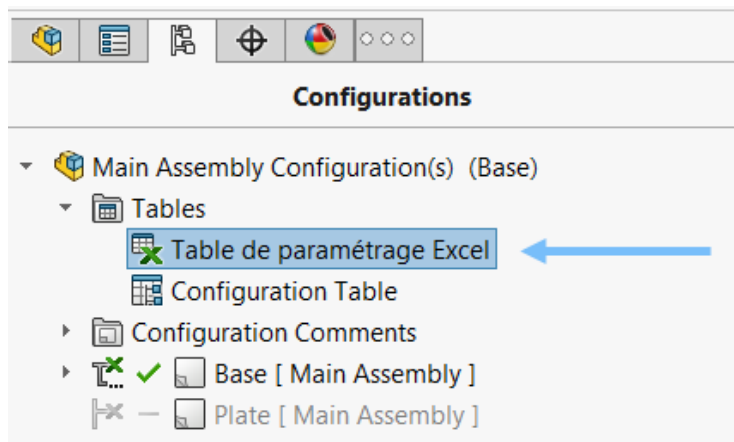
**Benefits:** You can view design table column headers in the local SOLIDWORKS language with no required workarounds.

For example, you create a design table in the German language with a column header **\$BESCHREIBUNG**. If you open the design table in an English version of SOLIDWORKS, you can automatically translate the column header as **\$DESCRIPTION**. If you open the same design table in an Italian version of SOLIDWORKS, you can automatically translate the column header as **\$DESCRIZIONE**.

The translation process is only temporary during the table editing process. The design table in the model remains in the original language.

**To translate design table column headers:**

1. Open a model with a design table that was created in another language. In this example, the original design table is in French.
2. In the ConfigurationManager , under **Tables**, right-click the foreign language Excel design table and click **Edit table**.



The Design Table column header language is *<foreign language>* dialog box alerts you that the design table language is different from your current language.

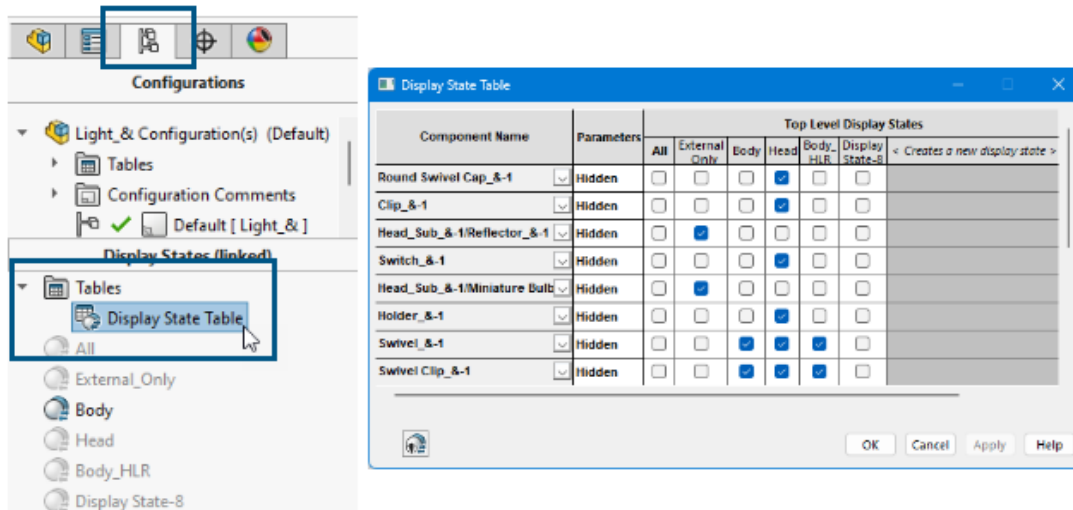


3. Click **Translate the column headers**.

The design table opens with the French column headers translated into English.



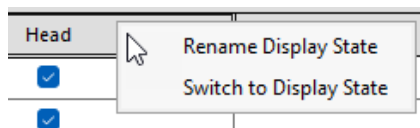
## Display State Tables



In assemblies with multiple display states, you can use the **Display State Table** to control the display states.

The **Display State Table** lets you:

- Control the Hide/Show state of a component
- Add a new display state by clicking in the **Creates a new display state** column
- Add a new component to the table by double-clicking the component in the PropertyManager or graphics area
- Double-click a display state's name cell to switch to that display state
- Right-click a display state's name cell to rename the display state or switch to it



To access the display state table, in the ConfigurationManager , under **Display States** > **Tables** , right-click **Display State Table**  and click **Show Table**.

Display State Table

Component Name	Parameters	Top Level Display States						
		All	External Only	Body	Head	Body_HLR	Display State-8	< Creates a new display state >
Round Swivel Cap_&-1	<div></div> Hidden	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
Clip_&-1	<div></div> Hidden	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
Head_Sub_&-1/Reflector_&-1	<div></div> Hidden	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
Switch_&-1	<div></div> Hidden	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
Head_Sub_&-1/Miniature Bulb	<div></div> Hidden	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
Holder_&-1	<div></div> Hidden	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
Swivel_&-1	<div></div> Hidden	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Swivel Clip_&-1	<div></div> Hidden	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	

OK

Cancel

Apply

Help

### General Information

- The table appears if the top-level assembly contains more than one display state.
- The table is available for unlinked and linked display states. For linked display states, the table shows the display states that are available for the active configuration.
- In the table, you can click **Hide/Show Referenced Display State** to hide or show the **Referenced Display State** row for each component in all top-level display states.

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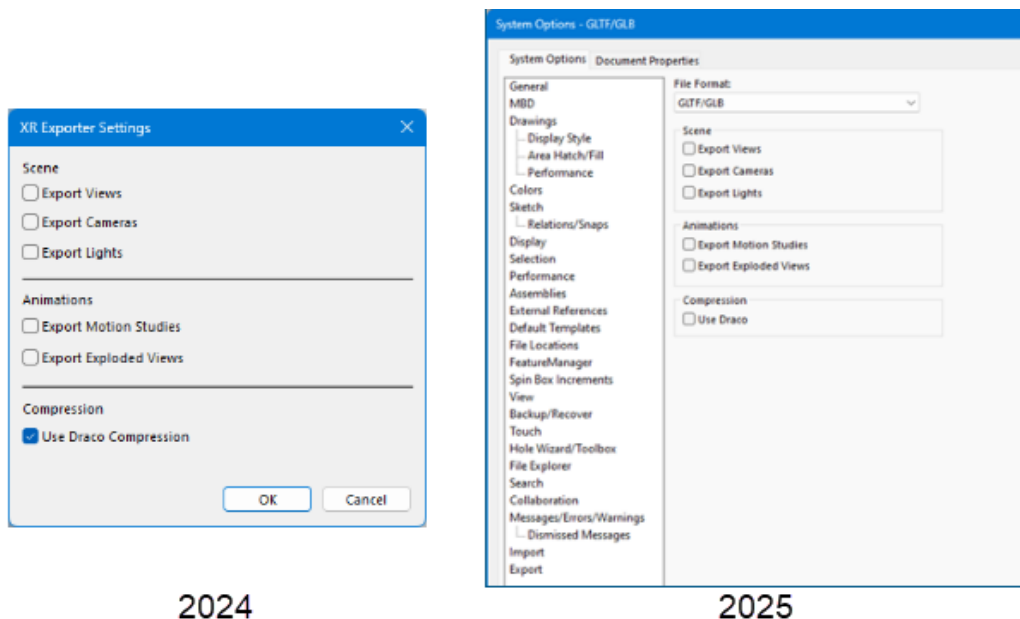
## Import/Export

---

This chapter includes the following topics:

- **Extended Reality Export Options (2025 SP2)**
- **Importing IFC and STEP Files (2025 SP2)**
- **Filtering Components When Importing IFC Files (2025 SP1)**
- **Exporting Custom Properties to IFC Files**
- **Importing Extended Reality Files**

### Extended Reality Export Options (2025 SP2)



The export options for saving files as extended reality files are moved from the XR Export Settings dialog box to the **Tools > Options > System Options > Export** dialog box.

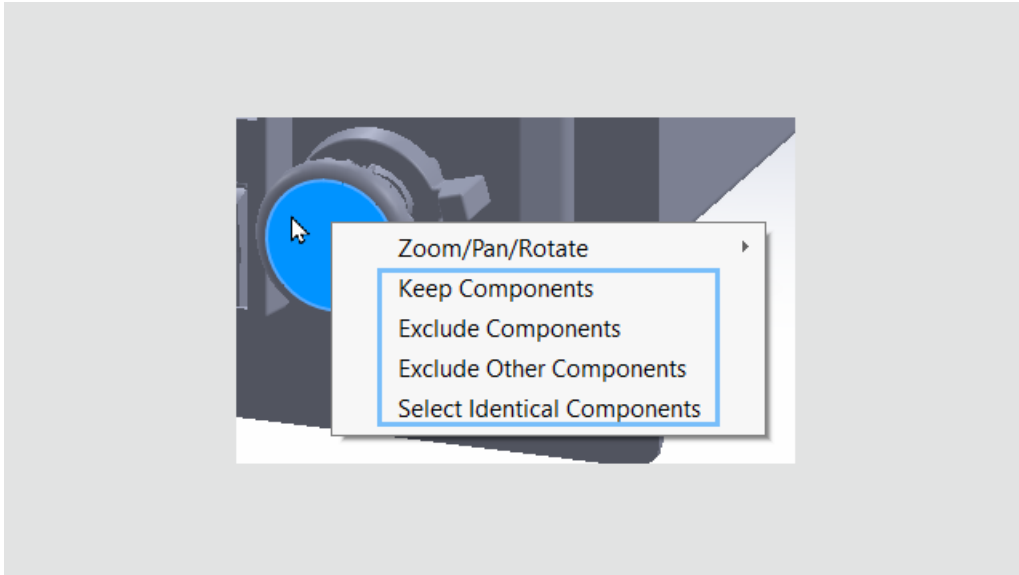
**Benefits:** This revised architecture permits future performance improvements.


To open the Export dialog box:

1. In a model, click **File > Save As**.
2. In the dialog box, for **Save as type**, select **Extended Reality (\*.glb)** or **Extended Reality (.gltf)**.
3. Click **Options** to open the System Options Export dialog box for **GLTF/GLB** files.

The export options are unchanged.

## Importing IFC and STEP Files (2025 SP2)



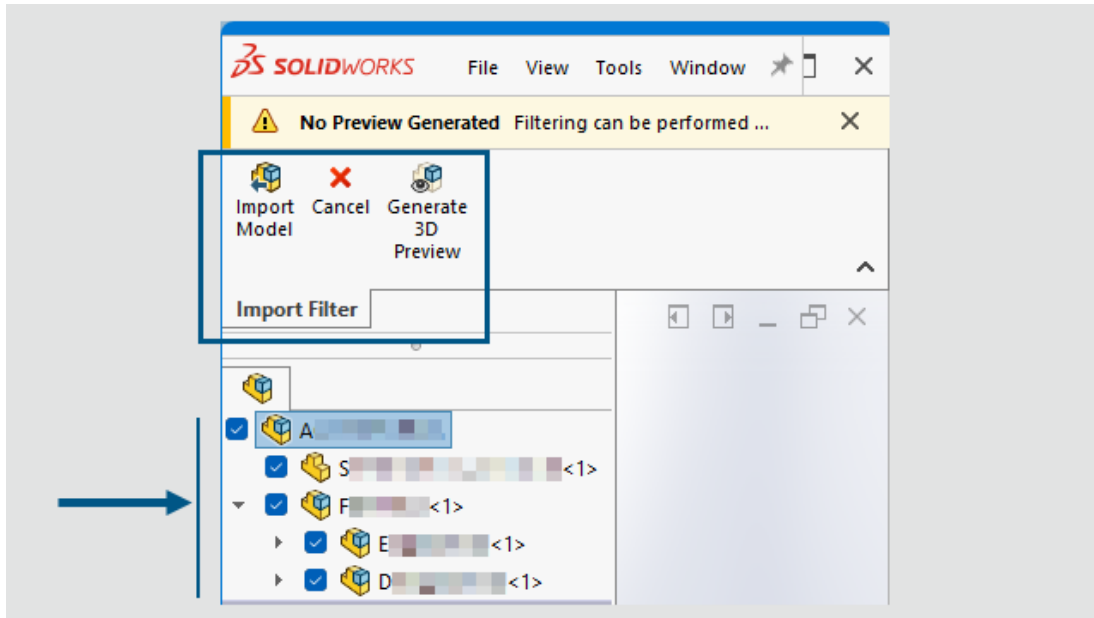
When you filter components as you import IFC or STEP files, all component selection options are available when you click **Generate 3D Preview**  and right-click components in the graphics area. Previously the options were available only in the FeatureManager design tree.

**Benefits:** Selecting components to filter is more efficient and uniform.

These options are available when you right-click components in the graphics area:

- **Keep Components**
- **Exclude Components**
- **Exclude Other Components**
- **Select Identical Components**

## Filtering Components When Importing IFC Files (2025 SP1)



When you import IFC files, you can filter which components to import.

**Benefits:** Filtering of components on import of IFC files lets you specify exactly the components you need, which saves you time and streamlines your work, especially for large IFC files.

### To filter components when importing IFC files:


1. In the Open dialog box, browse to select an IFC file, select **Enable Filter**, then click **Open**.

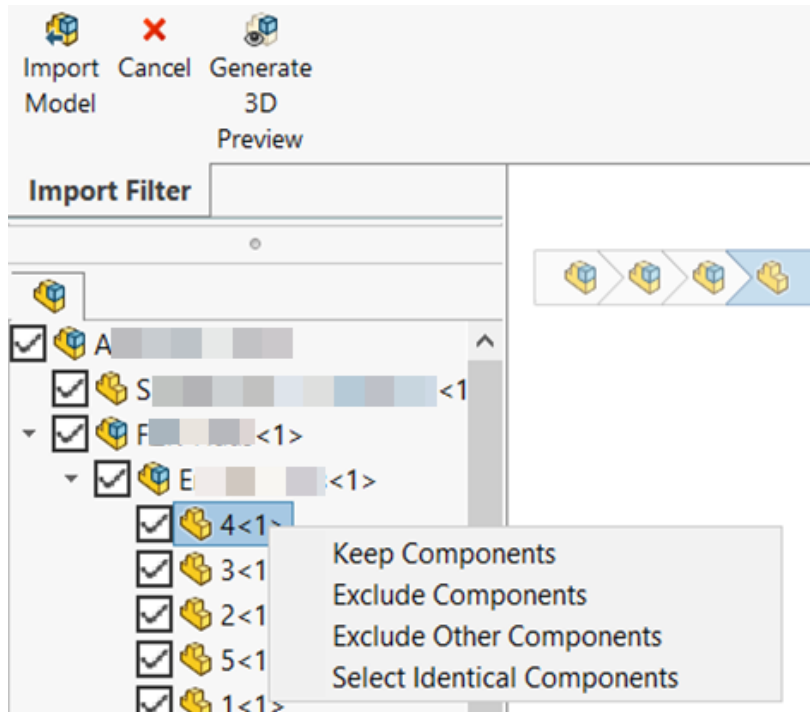
The software generates the product structure in the FeatureManager design tree, which displays the components that you can select to import. The graphics area is empty. The Import Filter CommandManager displays the available tools.


If you specified filter options in **Tools > Options > System Options > Import > File Format: IFC** under **Entities to Import**, SOLIDWORKS automatically applies those filter options. Specifying these system-level settings before filtering components saves you time, especially for large IFC files, because you can target the exact entities and components to open.

2. In the FeatureManager design tree, select components to import. You can select individual check components or box select multiple components.

To generate a preview, in the CommandManager, click **Generate 3D Preview** .

Subassemblies that contain a mix of selected and cleared components display a partially selected check box . To assist you with handling multiple selections, you can right-click the selected components and select **Keep Components** or **Exclude Components**. To invert the selections, select **Exclude Other Components**. If the components are identical, the **Select Identical Components** option also appears.



3. To import the IFC file with the selected components, in the CommandManager, click **Import Model** .

## Exporting Custom Properties to IFC Files



When you export SOLIDWORKS® models as IFC™ files, you can map SOLIDWORKS custom properties to IFC property sets.

### To export custom properties to IFC files:

1. In **Tools > Options > System Options > Export**, under **File Format**, select **IFC**.
2. Under **Output as**, select **Use Property Set mapping file**.
3. Then specify the XML Schema or .xsd mapping file that the software uses to validate the exported properties.

**Benefits:** BIM customers can export their custom properties data, which is important for the construction and operation of the building. This functionality is flexible. It lets you map SOLIDWORKS properties to IFC properties, potentially with a different name, and to define your own target property sets in the IFC file. In previous releases, you could export properties when you saved as IFC files, but only to a single hard-coded property set in the IFC file.

### To export custom properties to IFC property sets:

1. In the SOLIDWORKS file, click **File > Properties**.
2. On the Custom tab, add properties that you want to export to the IFC file and save the file.
3. Create an XML mapping file that maps the SOLIDWORKS custom properties to the IFC property set values.

SOLIDWORKS offers sample mapping files at *SOLIDWORKS install folder\lang\language\IFC*.


#### Sample mapping file:

```
<CustomPropertiesPSETMapping>
  <Schema Version="1.0"/>
  <PropertySet Name="Pset_DoorCommon">
    <AppliesTo ElementType="IFCDOOR"/>
    <PropertyMapping SOLIDWORKS="Reference" IFC="Reference"
Type="IfcIdentifier"/>
    <PropertyMapping SOLIDWORKS="FireRating" IFC="FireRating"
Type="IfcLabel"/>
    <PropertyMapping SOLIDWORKS="NoiseRating" IFC="AcousticRating"
Type="IfcLabel"/>
    <PropertyMapping SOLIDWORKS="Security" IFC="SecurityRating"
Type="IfcLabel"/>
    <PropertyMapping SOLIDWORKS="External" IFC="IsExternal"
Type="IfcBoolean"/>
    <PropertyMapping SOLIDWORKS="Infiltration" IFC="Infiltration"
Type="IfcVolumetricFlowRateMeasure"/>
    <PropertyMapping SOLIDWORKS="ThermalTransmit"
IFC="ThermalTransmittance" Type="IfcThermalTransmittanceMeasure"/>
    <PropertyMapping SOLIDWORKS="Glazing"
IFC="GlazingAreaFraction" Type="IfcPositiveRatioMeasure"/>
    <PropertyMapping SOLIDWORKS="Accessible"
IFC="HandicapAccessible" Type="IfcBoolean"/>
    <PropertyMapping SOLIDWORKS="FireDoor" IFC="FireExit"
Type="IfcBoolean"/>
    <PropertyMapping SOLIDWORKS="StarTrekDoor" IFC="SelfClosing"
Type="IfcBoolean"/>
    <PropertyMapping SOLIDWORKS="SmokeStop" IFC="SmokeStop"
Type="IfcBoolean"/>
```

```

    </PropertySet>
    <PropertySet Name="ACME_CageCodes">
        <AppliesTo ElementType="IFCDOOR"/>
        <AppliesTo ElementType="IFCWINDOW"/>
        <PropertyMapping SOLIDWORKS="RefCode" IFC="CageCode"
Type="IfcLabel"/>
    </PropertySet>
</CustomPropertiesPSETMapping>

```

4. In the SOLIDWORKS file, click **Save As**  (Standard toolbar) or **File > Save As**.
5. In the dialog box, for **Save as type**, select the IFC file type, then click **Options**.  
You can select any IFC file type.
6. In the System Options dialog box, under **Output as**, select **Use Property Set mapping file**, and select the mapping file from the list or browse to select it.

To include all the custom properties from the SOLIDWORKS file in the exported IFC file, under **Output as**, also select **Custom Properties**. To include these in the IFC property set, map all the custom properties in the XML Schema file.

SOLIDWORKS checks the validity of the XML IFC property set for these items:

- Proper tags, tag attributes, and tag structure.
- The Schema Version is equal to or lower than the version supported by the current version of SOLIDWORKS.
- SOLIDWORKS custom properties map one-to-one or one-to-many IFC properties. You cannot map multiple SOLIDWORKS custom properties to the same IFC property.

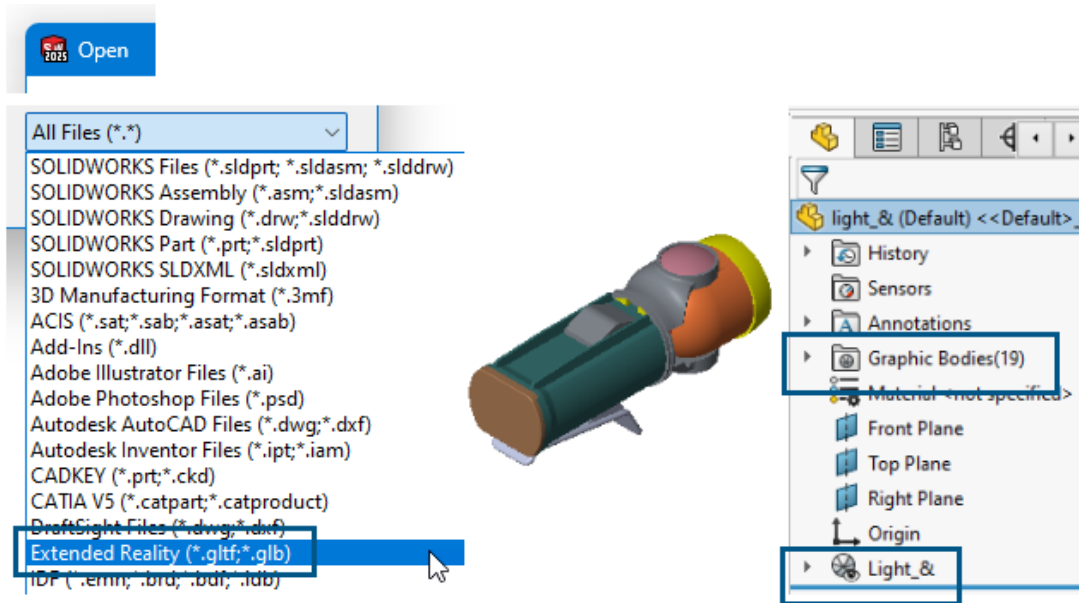
The software stores up to 10 property sets in the registry.

7. Click **OK** then click **Save** to export the file as an IFC file.

The IFC file contains the SOLIDWORKS custom properties in the IFC property set, based on the XML Schema mapping file.




## Importing Extended Reality Files



You can import the extended reality file types .glTF and .GLB.

### To import extended reality files:

1. Click **Open**  (Standard toolbar) or **File > Open**.
2. In the dialog box, for **Files of Type**, select **Extended Reality (\*.GLTF and .GLB)**.
3. Browse to select a file and click **Open**.

Importing glTF™ and GLB files includes:

- Geometry hierarchy of the imported glTF or GLB file.
- Draco™ compression.

This is a compression option for large-sized files. You do not specify any options on import. The file owner specifies the Draco compression on export of the glTF or GLB files from the source software.

- Noneditable textures. The software imports textures but not as proper SOLIDWORKS appearances.

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## SOLIDWORKS PDM

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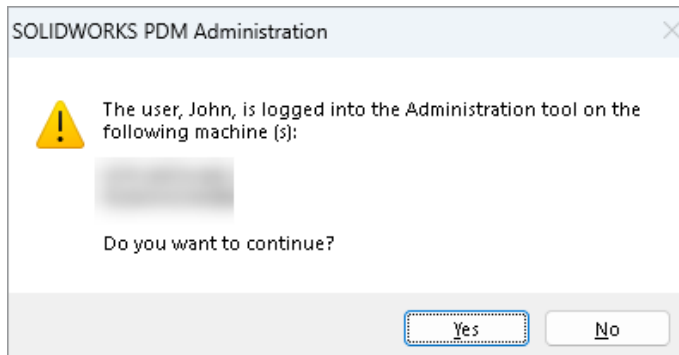
This chapter includes the following topics:

- **Display Warning for Multiple Authentication (2025 SP2)**
- **Bill of Materials for Electrical Assembly (2025 SP2)**
- **Display Options - Show Image Preview (2025 SP1)**
- **Card Controls Options (2025 SP1)**
- **Configuring the Convert Task (2025 SP1)**
- **Search Favorites (2025 SP1)**
- **Electrical Assembly Bill of Materials (2025 SP1)**
- **Default Settings for Computed BOM**
- **Checking Out Files During the Get Operation**
- **Logging Information for User Authentication**
- **Opening File Data in Microsoft Excel with Thumbnails**
- **Viewing the FeatureManager Design Tree Order of Assembly Structure in Computed BOMs**
- **Getting Information on Time Taken in Opening Files**
- **Getting Information on the Latest Revision**
- **Separate Add or Rename Permissions for Files and Folders**
- **SOLIDWORKS PDM to Electrical Connector**
- **File Check in Performance**
- **Availability of SOLIDWORKS PDM Toolbar and CommandManager Tab**
- **Additional Options in the Task Pane Shortcut Menu and Toolbar**
- **Support for SSL or TLS Authentication in SMTP Email Notification**

SOLIDWORKS® PDM is offered in two versions. SOLIDWORKS PDM Standard is included with SOLIDWORKS Professional, SOLIDWORKS Premium, and SOLIDWORKS Ultimate, 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.

## Display Warning for Multiple Authentication (2025 SP2)

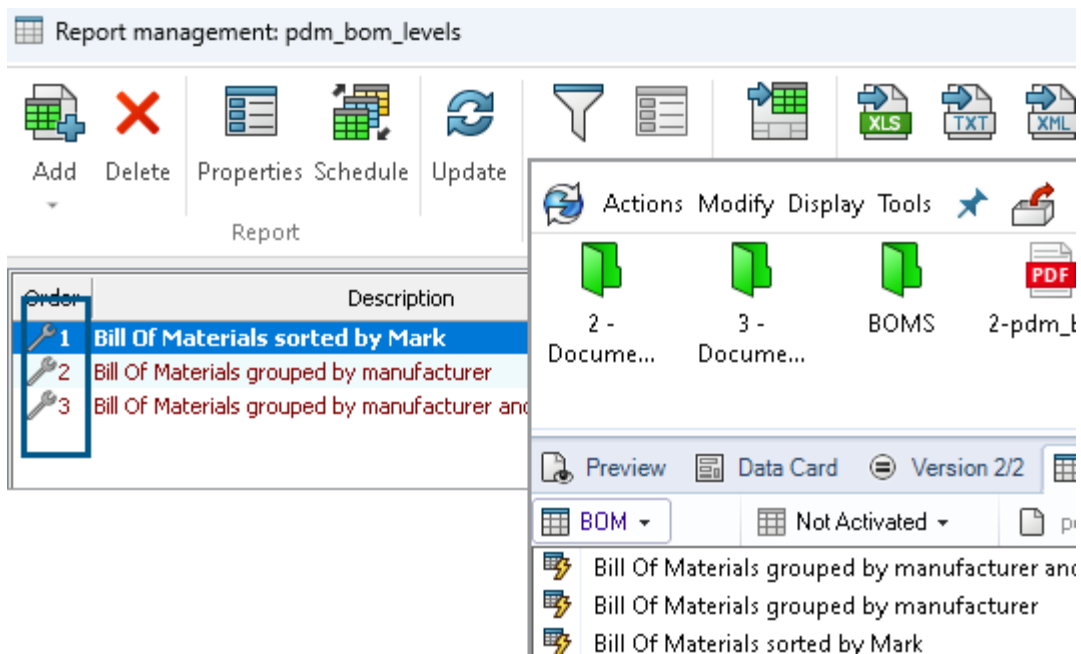


For SOLIDWORKS PDM Professional, when you attempt to log in to the SOLIDWORKS PDM Administration tool more than once from different computers with the same account, you receive a warning message reminding you about your previous sign-ins.

The warning message displays the names of the computers that you have already logged in to and asks you if you want to continue or cancel the log in. This lets you avoid accidentally overwriting prior updates that you have made from other computers.

**You get the warning message only if you select the option File Vault Properties > Logging operation > Log-in and Log-out.**

## Bill of Materials for Electrical Assembly (2025 SP2)

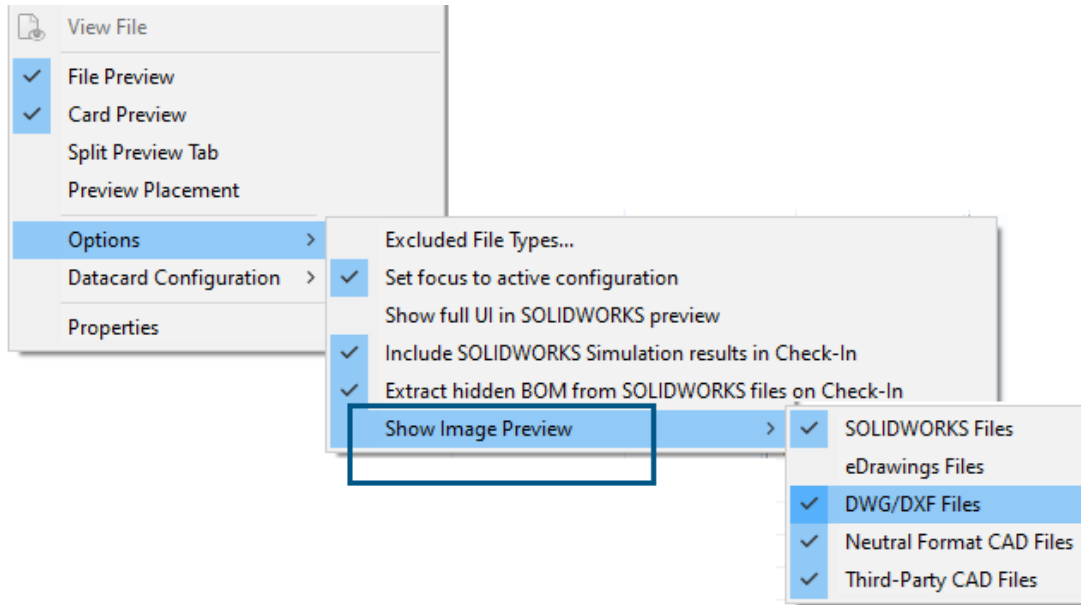


In the SOLIDWORKS PDM File Explorer, in the **BOM** view of the Bill of Materials tab, you can view all the Manufacturer Parts BOMs for the Electrical assemblies that you select in SOLIDWORKS Electrical.

For example,

- **Bill Of Materials by manufacturer**
- **Bill Of Materials by manufacturer and by book**

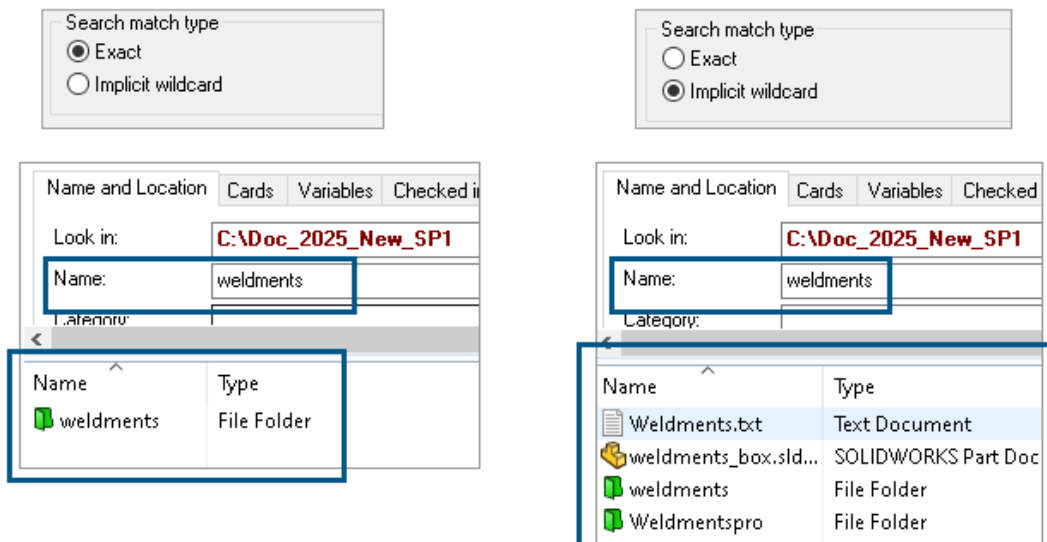
## Display Options - Show Image Preview (2025 SP1)



In the SOLIDWORKS PDM File Explorer, you can view a thumbnail or a full preview in the Preview tab based on the following file types using **Display > Options > Show Image Preview**:

- **SOLIDWORKS Files**
- **eDrawings Files**
- **DWG/DXF Files**
- **Neutral Format CAD Files**
- **Third-party CAD Files**

## Card Controls Options (2025 SP1)



In the SOLIDWORKS PDM Administration tool, you can select one of the following as a **Search match type** while editing or adding **List** and **Combobox** card controls to the search and file card:

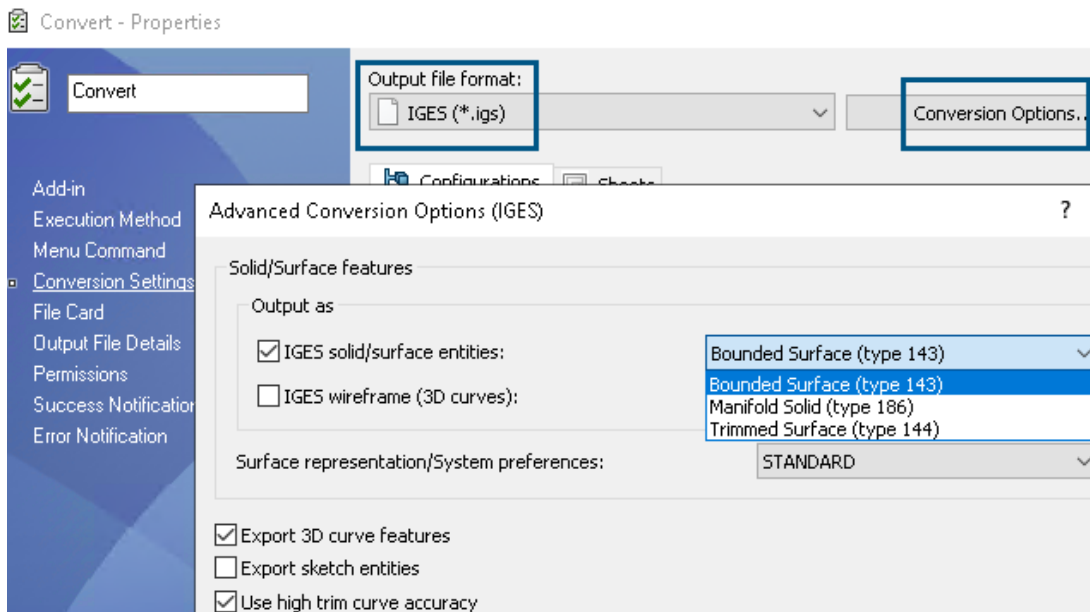
- **Exact:** You can search for files, folders, and variables in the SOLIDWORKS PDM File Explorer whose search results exactly match the search input.

For example, if you search for `weldments` in **Name**, the search results include only those files, folders, or variables with the exact name `weldments`. If you want all files that include **weldments** in the name, enter an asterisk (\*) as a wildcard, for example `weldments*` or `*weldments*`.

- **Implicit wildcard:** You can search for files, folders, and variables in the SOLIDWORKS PDM File Explorer whose search results include the search input.

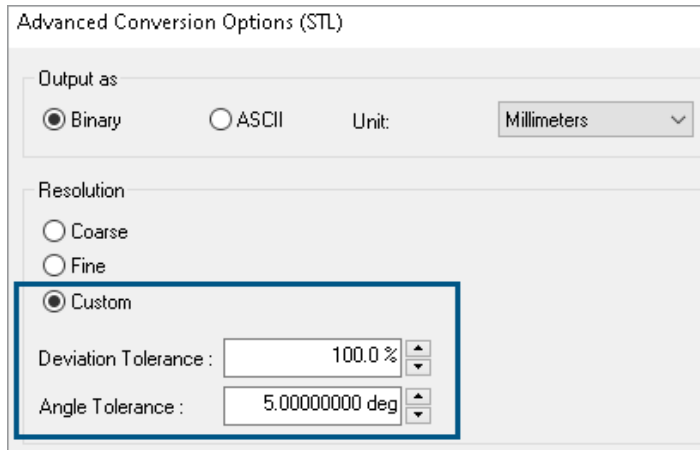
For example, if you search for `weldments` in **Name**, the search results include all files, folders, or variables whose names include `weldments` (for example, `weldments`, `weldments_box`, and `weldmentspro`).

## Configuring the Convert Task (2025 SP1)



In the SOLIDWORKS Administration tool, while configuring a convert task, you can use the following advanced conversion options for the .stl and .igs output file formats.

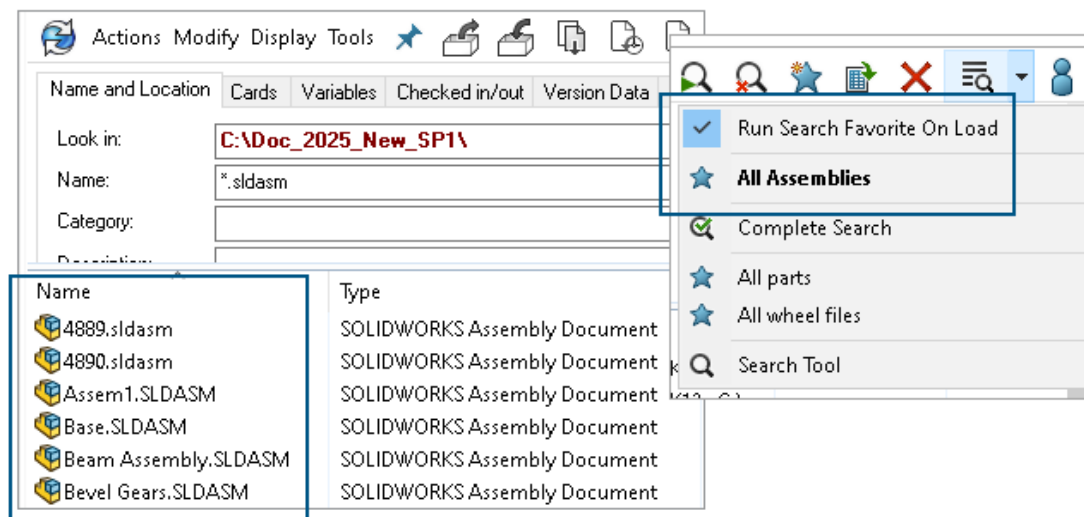
Output File Format	Advanced Conversion Options
<b>IGES (*.igs)</b>	<p><b>Bounded Surface (type 143):</b> Select to convert the faces of the part, assembly, or the selected surfaces and its boundaries defined by other IGES entities, for example, curves and edges.</p>
<b>STL (*.stl)</b>	<p><b>Custom</b> option under <b>Resolution</b> with the following sub options:</p> <ul style="list-style-type: none"> <li>• <b>Deviation Tolerance:</b> Controls whole-part tessellation. Lower numbers generate files with greater whole-part accuracy.</li> <li>• <b>Angle Tolerance:</b> Controls small-detail tessellation. Lower numbers generate files with greater small-detail accuracy, but those files take longer to generate.</li> </ul>



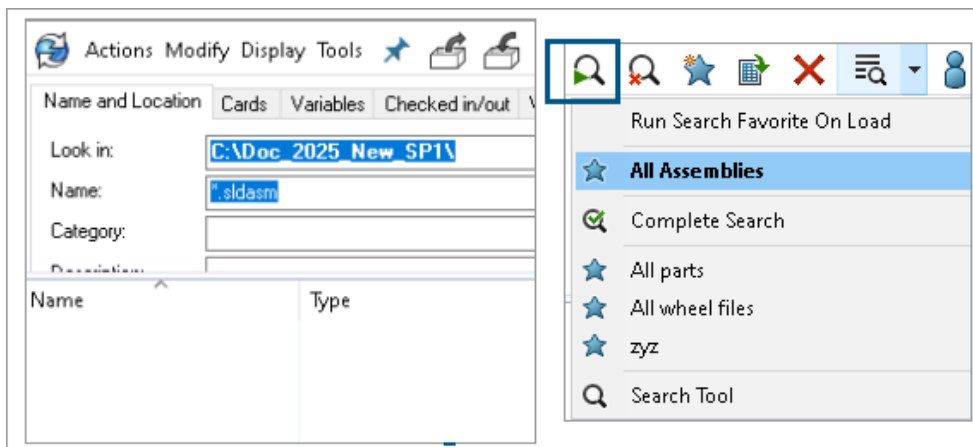
You can access these options under **Tasks > Convert > Open > Conversion Settings > Conversion Options**.


These options are similar to the SOLIDWORKS **Export** system options for the .stl and .igs file formats. For more information, see *SOLIDWORKS Help: IGES Export Options* and *SOLIDWORKS Help: STL, 3D Manufacturing Format, and Additive Manufacturing File Export Options*.

## Search Favorites (2025 SP1)

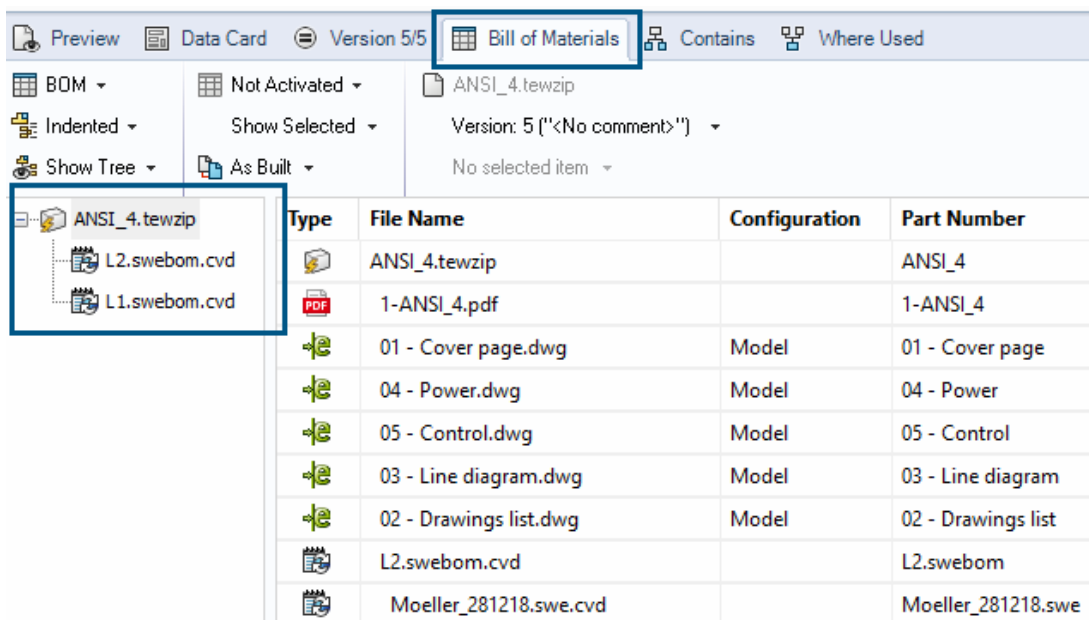


In the SOLIDWORKS File Explorer, you can use the **Run Search Favorite on Load** functionality to view Search Favorite results for files and folders by selecting the Search Favorite. The functionality is available with the integrated search and the **Search Tool**.



When this option is not selected, you can view the Search Favorite results by selecting the Favorite Search and clicking **Start Search** .

## Electrical Assembly Bill of Materials (2025 SP1)

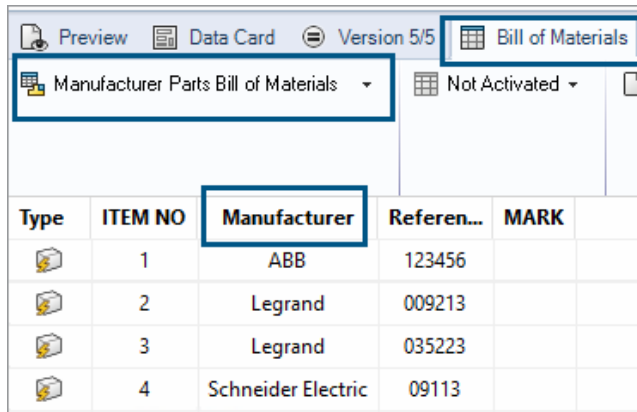


In the SOLIDWORKS PDM File Explorer, in the Bill of Materials tab you can view the Electrical assemblies BOM details.

For Electrical assemblies, you can view:

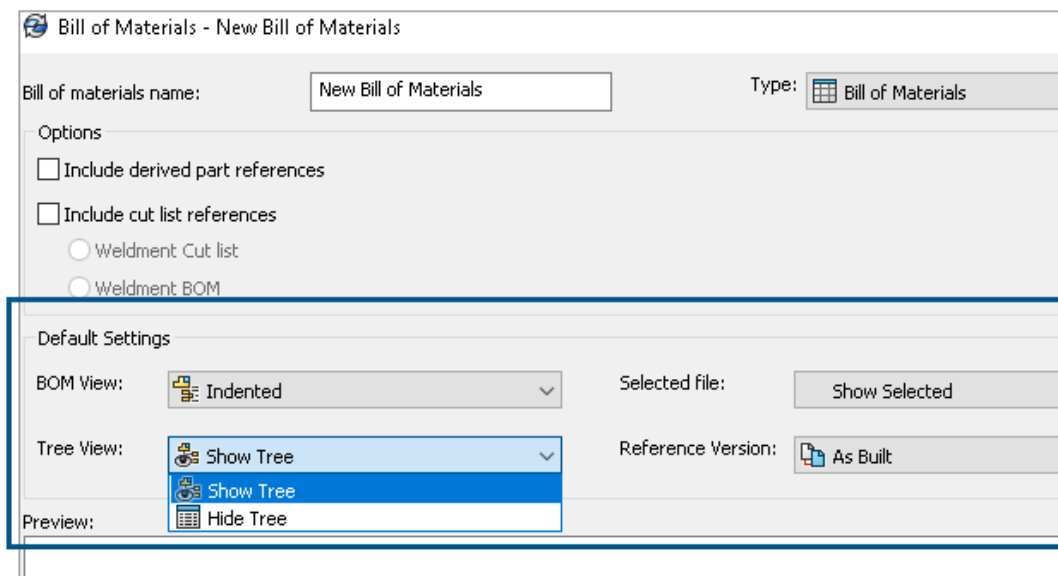
- The parent-child hierarchical and indented structure in the Computed BOM for **cvd** files.
- The **Manufacturer Parts Bill of Materials** view.





Type	ITEM NO	Manufacturer	Referen...	MARK
	1	ABB	123456	
	2	Legrand	009213	
	3	Legrand	035223	
	4	Schneider Electric	09113	

## Default Settings for Computed BOM



Bill of materials name:  Type: Bill of Materials

Options

☐ Include derived part references

☐ Include cut list references

☐ Weldment Cut list

☐ Weldment BOM

Default Settings

BOM View: Indented Selected file:

Tree View: Show Tree Reference Version: As Built

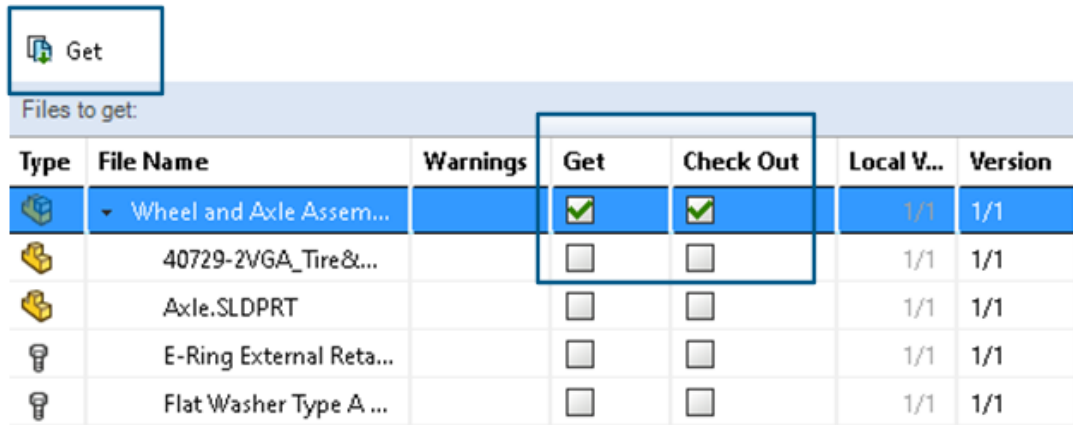
Preview: Hide Tree






Administrators can specify the default view and options settings for the computed BOM while creating the Bill of Materials (BOM) in the SOLIDWORKS PDM Administration tool.

The default settings that the administrators specify apply to the BOM **View** and **Options** sections under the Bill of Materials tab in the SOLIDWORKS PDM File Explorer. The default settings are applicable to both the desktop and the Web2 client.

In the Administration tool, right-click **Bill of Materials** > **New Bill of Materials**. In the Bill of Materials - New Bill of Materials dialog box, under **Default Settings**, specify the default settings for the computed BOM.

## Checking Out Files During the Get Operation



Type	File Name	Warnings	Get	Check Out	Local V...	Version
	Wheel and Axle Assem...		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1/1	1/1
	40729-2VGA_Tire&...		<input type="checkbox"/>	<input type="checkbox"/>	1/1	1/1
	Axle.SLDPRT		<input type="checkbox"/>	<input type="checkbox"/>	1/1	1/1
	E-Ring External Reta...		<input type="checkbox"/>	<input type="checkbox"/>	1/1	1/1
	Flat Washer Type A ...		<input type="checkbox"/>	<input type="checkbox"/>	1/1	1/1

In the SOLIDWORKS PDM File Explorer, you can check out files while performing a **Get** operation on them, for example, **Get Latest Version**, provided you have checkout permission granted.

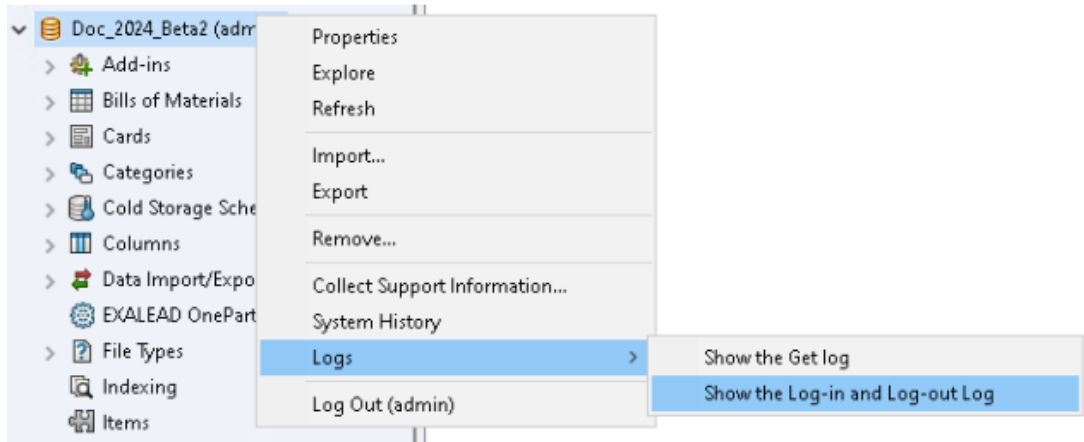
In the Get dialog box, when you select **Check Out** for single or multiple files, the **Get** option for these files is selected by default to perform both the operations at the same time. The combined **Get** and **Check out** operation simplifies your workflow.

You can add the **Check out** column in the Get dialog box of SOLIDWORKS PDM File Explorer. The customization is done using the **Customizable Columns** view for **Get** file operations columns in the SOLIDWORKS PDM Administration tool.

The following conditions apply for the combined **Get** and **Check out** operation:

- If the get operation fails, checkout does not proceed.
- If checkout fails, the get operation still proceeds.
- When running the get operation for an older version, if you select **Check out**, you get the specified version with a checkout performed.

## Logging Information for User Authentication



You can view the user authentication details for a vault in the SOLIDWORKS PDM Professional Administration tool.

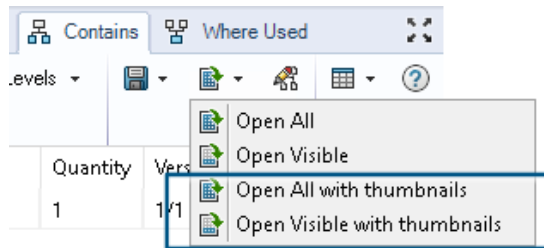
The authentication details include the user name, the date, and time when the user has logged in and out, and the SOLIDWORKS PDM client (desktop or Web2).

Type	Log-In...	Log-O...	Log-Out D...	Application	Process Name	Client Ma
Info...	2024-...	2024-...		Desktop Client	explorer.exe	DTP-DRT
Info...	2024-...	2024-...		Administration	ConisioAdmin.exe	DTP-DRT
Info...	2024-...	2024-...		Desktop Client	explorer.exe	DTP-DRT
Info...	2024-...	2024-...		Desktop Client	explorer.exe	DTP-DRT
Info...	2024-...	2024-...	Disconnected	WebAPI	PostmanRuntime/7.37.3	
Info...	2024-...	2024-...	Disconnected	Web2	w3wp.exe	

You can right-click the vault name and select **Logs > Show the Log-in and Log-out Log** to view the authentication details. To see this option, you must have:

- A SOLIDWORKS PDM Professional vault.
- **File Vault Management** permission.
- The **Log-in and Log-out** option selected in the file vault properties under **Logging Operations**.

## Opening File Data in Microsoft Excel with Thumbnails

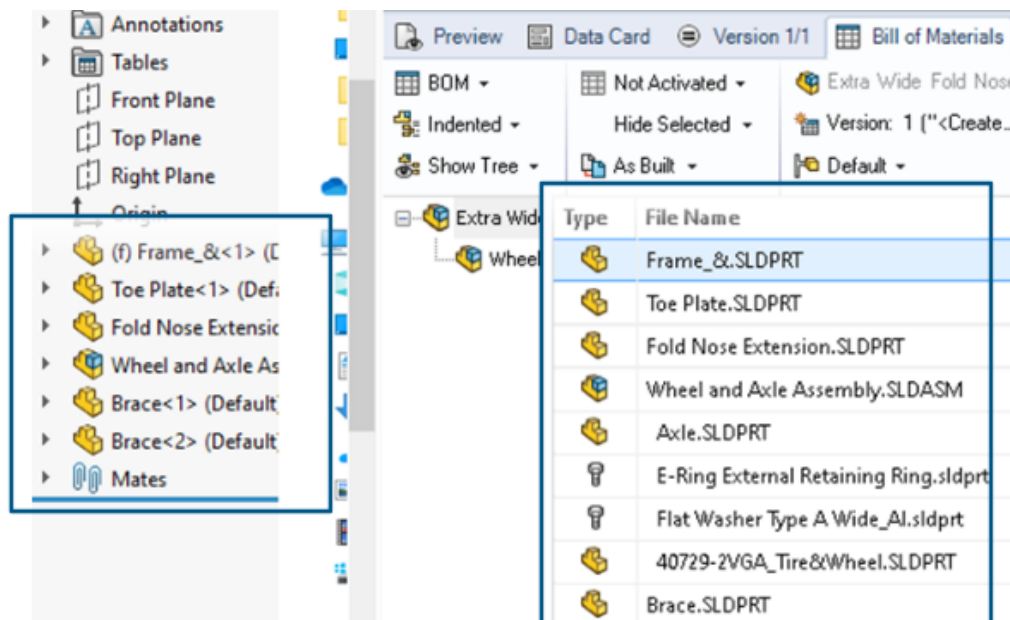


You can open the file data in the Microsoft® Excel® format along with a thumbnail preview in the Bill of Materials, Contains, and Where Used tabs of the SOLIDWORKS PDM File Explorer.

You can open file data with thumbnails using **Open All With Thumbnails**  and **Open Visible With Thumbnails**  under **Open as CSV** in the toolbar of the tabs.

With thumbnail preview, you can understand the data more clearly and you can effectively communicate the process outside the vault.

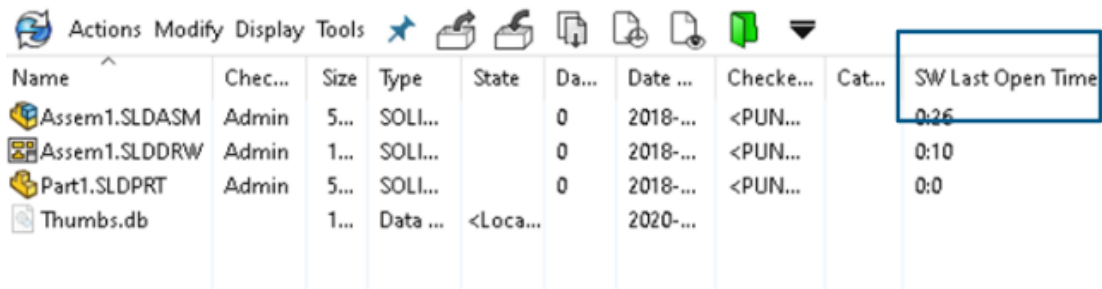
## Viewing the FeatureManager Design Tree Order of Assembly Structure in Computed BOMs



You can view the order of the assembly structure in the computed BOMs of the SOLIDWORKS PDM File Explorer for newly checked in files. The view is similar to that in the SOLIDWORKS FeatureManager® design tree.

The order of assembly components in the BOM for data already checked in to the vault does not change to match with the FeatureManager design tree.

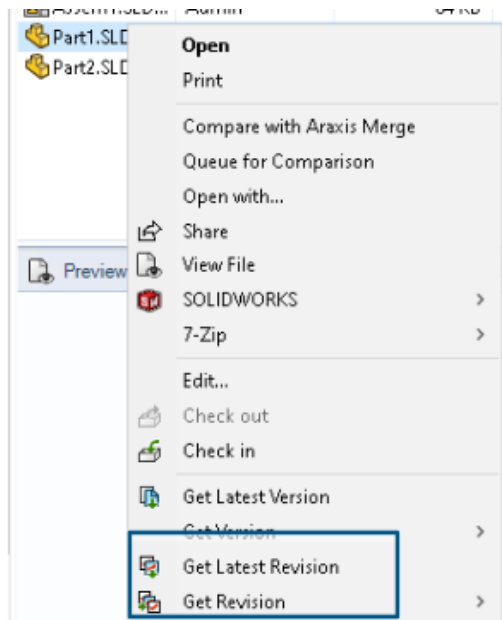
## Getting Information on Time Taken in Opening Files





Name	Chec...	Size	Type	State	Da...	Date ...	Checke...	Cat...	SW Last Open Time
Assem1.SLDASM	Admin	5...	SOLI...		0	2018-...	<PUN...		0:25
Assem1.SLDDRW	Admin	1...	SOLI...		0	2018-...	<PUN...		0:10
Part1.SLDPRT	Admin	5...	SOLI...		0	2018-...	<PUN...		0:0
Thumbs.db		1...	Data ...	<Loca...		2020-...			

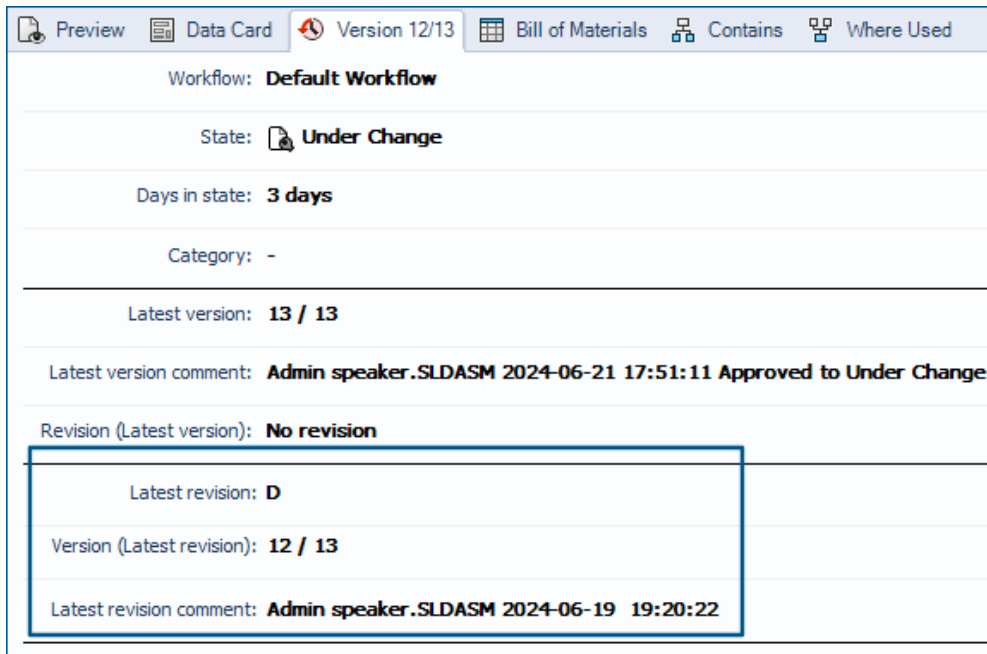
You can know the time taken to open a file when it was last opened in SOLIDWORKS 2023 and above. The time is measured in seconds. To know the file open time, a new variable **\_SW\_Last\_Open\_Time\_** is added to the SOLIDWORKS PDM variables.

## Getting Information on the Latest Revision

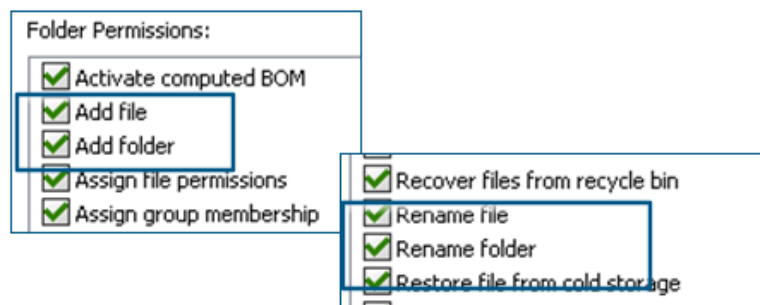


You can get the latest revision for a file in SOLIDWORKS PDM. For getting the latest revision, a **Latest Revision** variable is added to the existing system variables.

To retrieve the revision information for a file, you can use the **Get Latest Revision**  and **Get Revision**  commands in the SOLIDWORKS PDM File Explorer at different places such as during searching files, in right-click menu of a file view, the **Version** tab, and the Column Sets. You can also use these commands in the SOLIDWORKS PDM add-in toolbar and CommandManager.



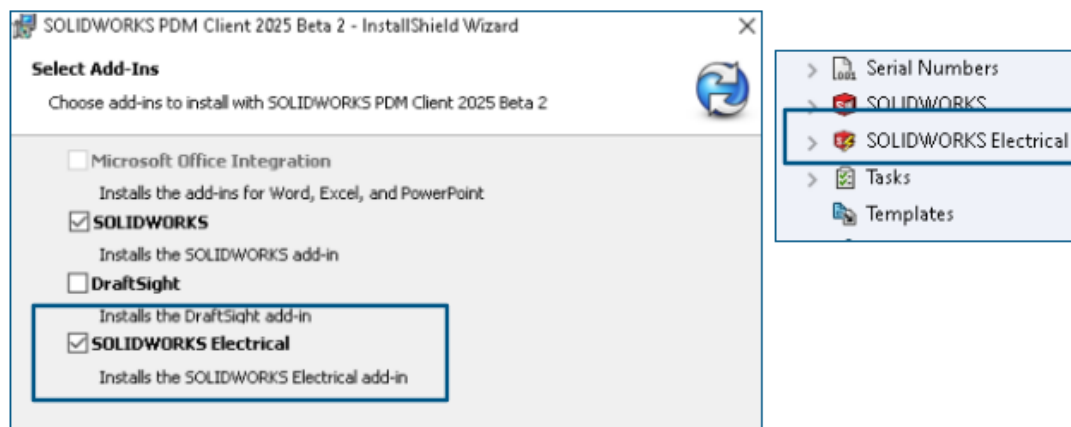
## Separate Add or Rename Permissions for Files and Folders



The existing **Add or rename file** and **Add or rename folder** permissions are split into separate permissions for add and rename.

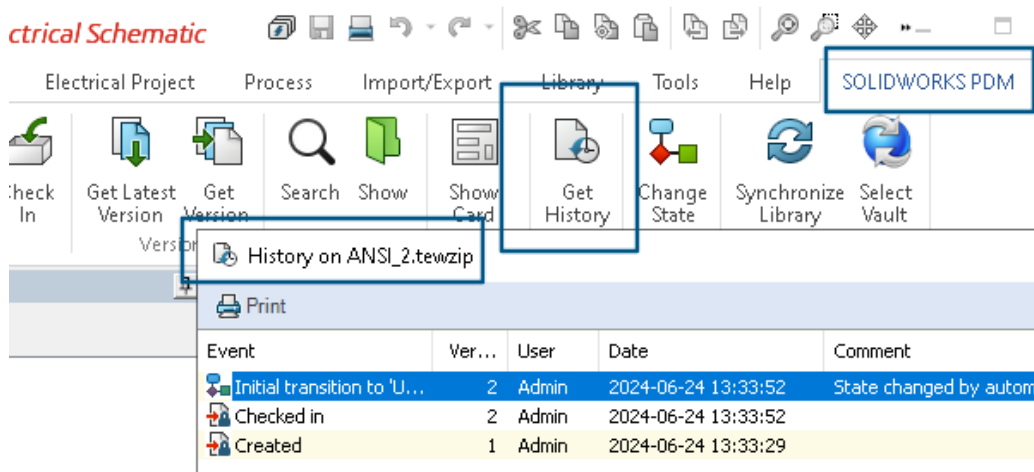
Administrators can use **Add file** and **Rename File** under **Folder Permissions** and **State Permissions** in the SOLIDWORKS PDM Administration tool.

## SOLIDWORKS PDM to Electrical Connector



The SOLIDWORKS Electrical to SOLIDWORKS PDM connector is available with the SOLIDWORKS PDM install. It is integrated with SOLIDWORKS PDM and is not available as a SOLIDWORKS PDM add-in.

You can configure the SOLIDWORKS Electrical connector from the SOLIDWORKS PDM Administration tool. A **SOLIDWORKS Electrical** node is added under the SOLIDWORKS PDM vault for the configuration.

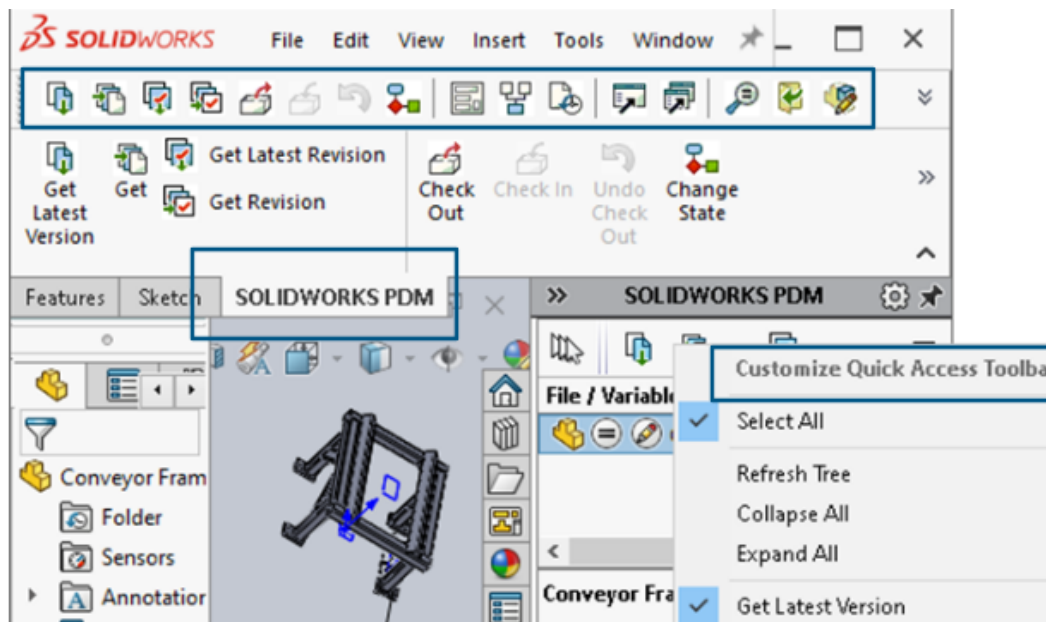


The **History** option is added to the SOLIDWORKS PDM CommandManager options. You can see the history of SOLIDWORKS Electrical projects using this option for better monitoring of the changes.

## File Check in Performance

SOLIDWORKS PDM performance is improved during the file check in to the SOLIDWORKS PDM database. The file check in operation is two times faster than before.

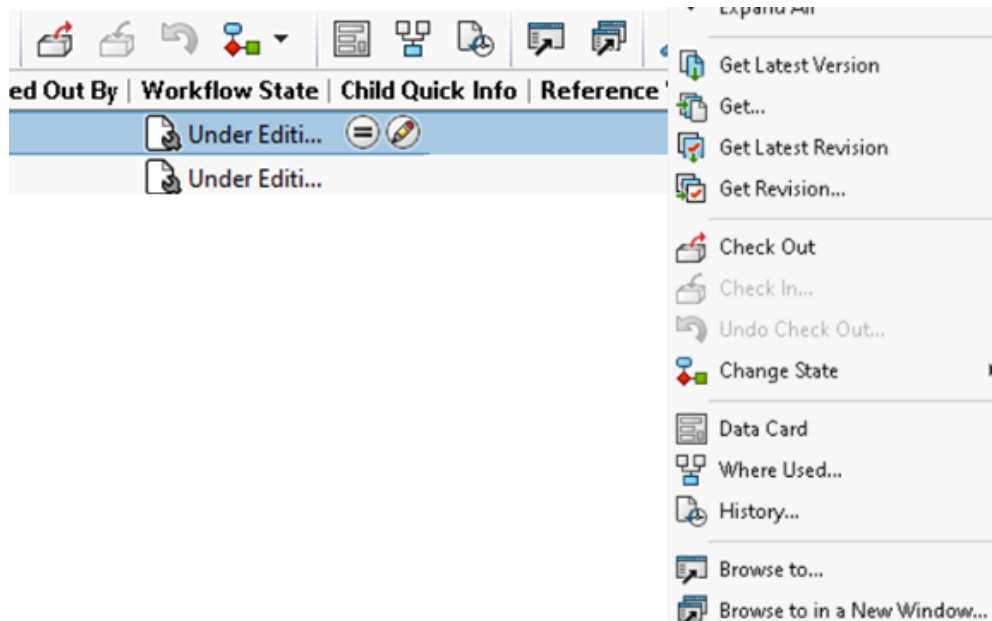
## Availability of SOLIDWORKS PDM Toolbar and CommandManager Tab







You can access SOLIDWORKS PDM and all its commands from a dedicated SOLIDWORKS PDM toolbar and the CommandManager tab in SOLIDWORKS when you select the SOLIDWORKS PDM add-in.

## Additional Options in the Task Pane Shortcut Menu and Toolbar



The Task Pane of the SOLIDWORKS PDM add-in has new options in the shortcut menu and toolbar. Also, some of the existing options are updated. All the options are organized in meaningful groups for better clarity.

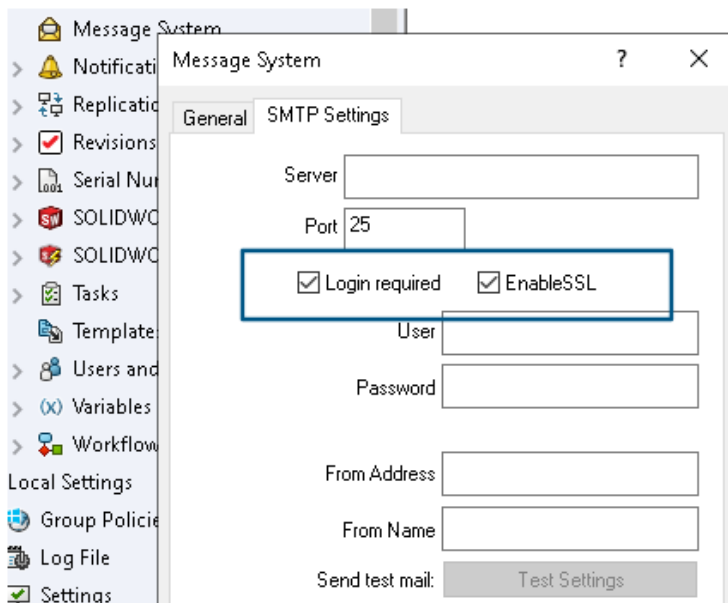
For example, the following are the options added:

- **Browse to** : Opens the selected file in the same SOLIDWORKS PDM File Explorer window.
- **Browse to in a New Window** : Opens the selected file in a new SOLIDWORKS PDM File Explorer window.
- **Data Card** and **Where Used**: Display information of the data card and where it is used. These options are grouped with the **History** option.

The **Edit** option is renamed as **Edit Component** .

You can customize the Task Pane toolbar to include options you use frequently.

## Support for SSL or TLS Authentication in SMTP Email Notification



You can enable Secured Socket Layer (SSL) or TLS (Transport Layer Security) authentication in the SMTP email notification.

In the SOLIDWORKS PDM Administration tool, you can select **EnableSSL** under **Message System > SMTP > SMTP Setting** to enable SSL or TLS authentication in the SMTP email notifications. It is a two-way authentication along with the login credentials.

The following SMTP servers are supported:

Mail Server	SMTP Server
<b>Gmail®</b>	smtp.gmail.com
<b>Outlook®</b>	smtp.outlook.com
<b>Microsoft 365®</b>	smtp.office365.com
<b>Yahoo®</b>	smtp.mail.yahoo.com

# 16

## SOLIDWORKS Manage

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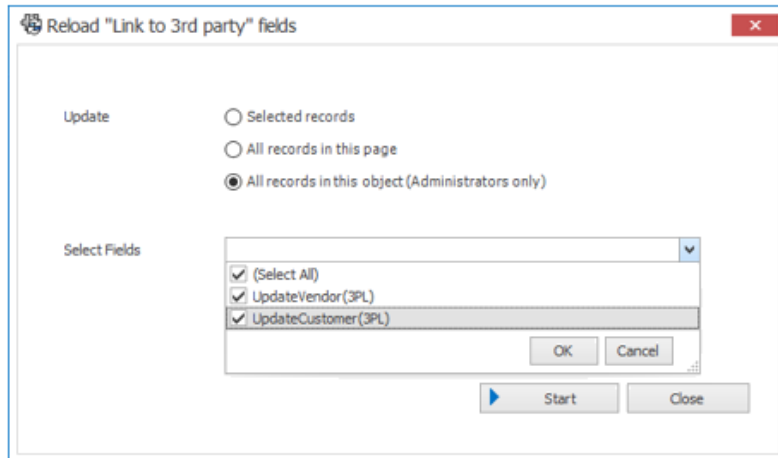
This chapter includes the following topics:

- **Batch Updates for Link to 3rd Party Fields**
- **Sync with SOLIDWORKS PDM**
- **Future Date Notifications**
- **Batch Updates for Process Fields**
- **Send Affected Items to New Processes**
- **Collaboration Comments in File Sharing**
- **Client Version Check**
- **Flat BOM Groupings**
- **Adding Automated Task Subject Information**
- **Project Snapshots**
- **Tasks from Cancelled Processes**
- **Application Programming Interface**
- **Creating New Process Records from Existing Process Records**
- **Send to Process for Affected Items**
- **Affected Items in Microsoft File Explorer**
- **Thumbnails for BOM Copy From**
- **Installing the SOLIDWORKS Manage Web API**

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.

## Batch Updates for Link to 3rd Party Fields



You can update **Link to 3rd party field** values for some or all records in an object.

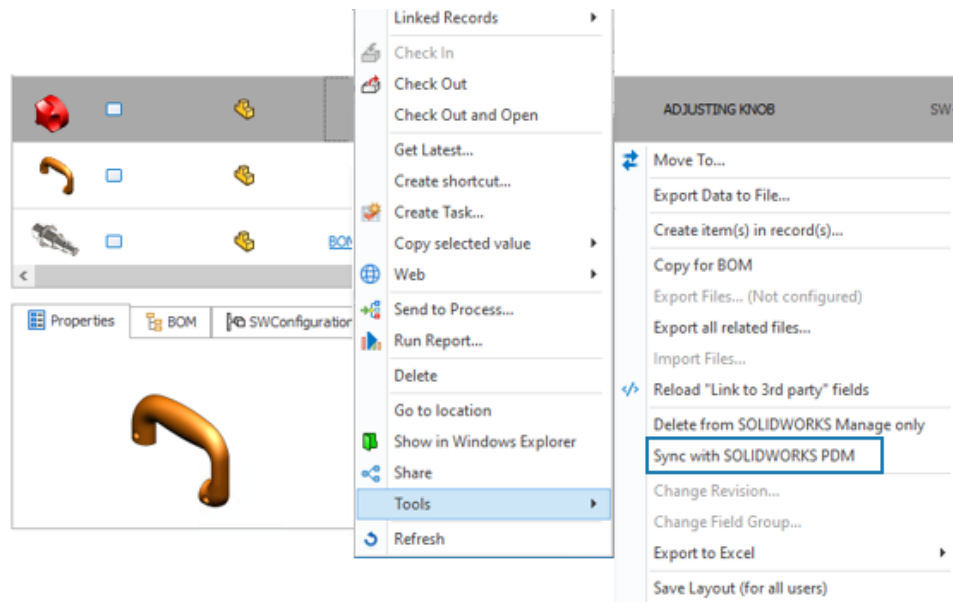
Nonadministrators can update the values for selected records in the Main Grid or for all records on a page. Administrators can update the values for all records in an object. This restricts users from affecting system performance if there are many fields or ones with complex queries.

This is a convenient way to populate a new **Link to 3rd party field** without writing a separate SQL query.

### Implementing Batch Updates to Link to 3rd Party Fields

1. Navigate to an object that has **Link to 3rd party** fields.
2. Select records, then right-click and click **Tools > Reload "Link to 3rd party" fields**.
3. In the dialog box:
  - a) Specify options.
  - b) Click **Start**.
  - c) After the fields update, click **Close**.

## Sync with SOLIDWORKS PDM



All users can sync selected records in the Main Grid of a SOLIDWORKS PDM object.

SOLIDWORKS Manage reads data from the SOLIDWORKS PDM database, then synchronizes the information in the SOLIDWORKS Manage database. Previously only administrators could sync records in the System Administration tool.

Right-click a record and click **Tools** > **Sync with SOLIDWORKS PDM**.

## Future Date Notifications

You can send a notification after a certain date and time for a process notification.

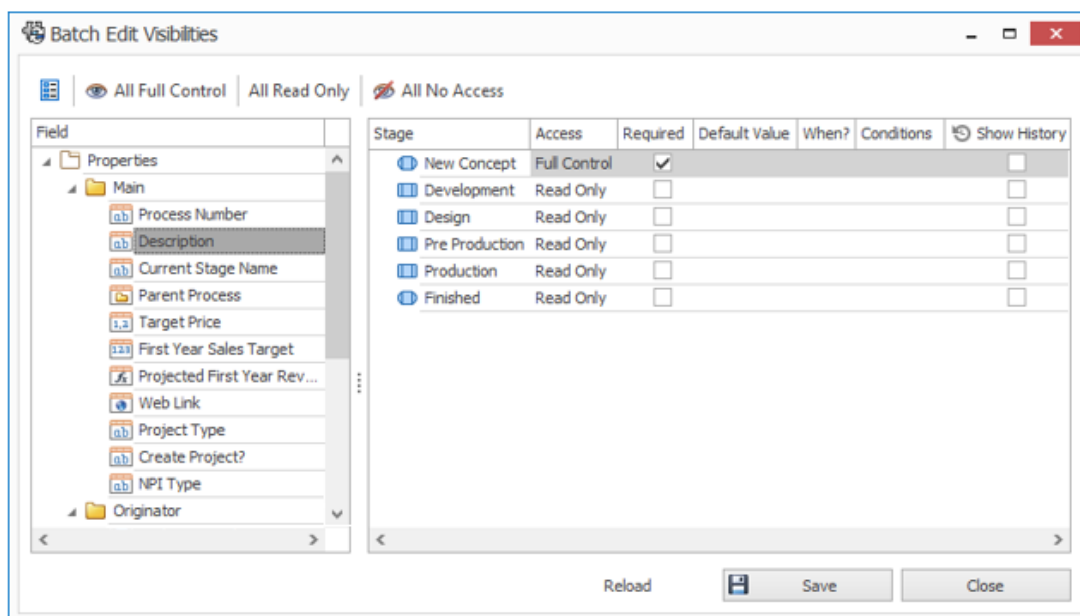
The setting remains active even after a process finishes unless you restrict the notification by a condition. This causes a notification such as a renewal or follow-up reminder to be sent out after a process finishes.

### Creating Future Date Notifications

1. In the System Administration tool, right-click a process and click **Administration**.
2. In the Process Wizard, on the Fields page, specify a **Date** field to contain the date used to send the notification.  
If the process already has a suitable **Date** field, you can skip this step.
3. On the Workflow Properties page, select:
  - a) A stage for when to send the notification.
  - b) The **Visibility** node.
4. Specify the **Date** field you defined in step 2 to specify the notification send date.  
For example, specify the **Default Value** as the *current date* and **When?** to **End**. This specifies the date for when the process stage goes to the next stage.

5. Select **Notifications** for the stage and edit an existing notification or create a new notification.
6. In the Stage Notifications dialog box, on the General tab in:
  - a) **When to send**, select **Custom**.
  - b) **Select Date Field**, select the **Date** field you defined in step 2.
  - c) (Optional) **Time**, specify the time of day on the selected date to send the notification.
  - d) (Optional) **Adjustment days**, add days to the **Select Date Field**.
  - e) Click **Save** then **Close**.

## Batch Updates for Process Fields



You can edit fields for multiple process stages with the **Batch Edit** tool.

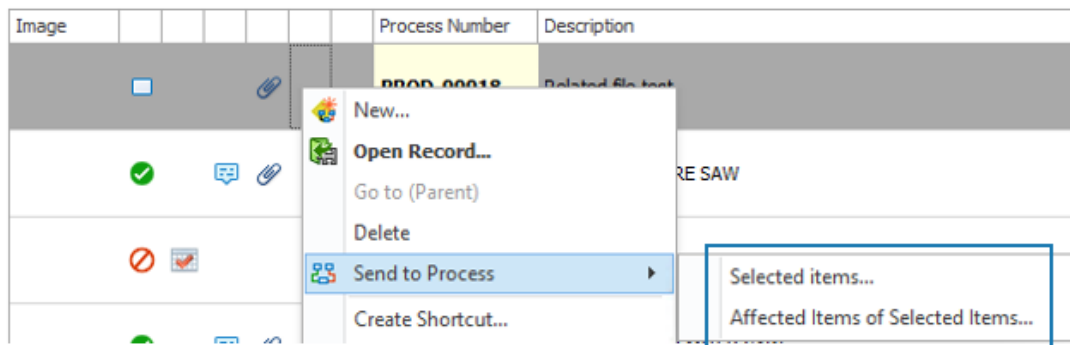
With the **Batch Edit** tool, you can change a field for all stages in one place. Previously, you had to select each stage in the workflow diagram, then save the edited field.

## Implementing Batch Updates to Process Fields

1. In the System Administration tool, right-click a process and click **Administration**.
2. On the Workflow Properties page:
  - a) Select a stage.
  - b) Select the **Visibility** node.
  - c) Click **Batch Edit**.

3. In the Batch Edit Visibilities dialog box:
  - a) In the left pane, select a **Field**.  
In the right pane, all the stages defined in the process appear under **Stage**.
  - b) Change the settings for each stage, then click **Save**.
  - c) Repeat steps 3a and 3b for additional fields.  
If you select another **Field** without clicking **Save**, the changes to the previously selected field do not save.
  - d) Click **Close**.

## Send Affected Items to New Processes








You can send affected items from selected processes to new processes.


You can send either the process itself or only its affected items to a new process. This makes it easier to resend the same affected items from one process to another. Previously, you had to add each item individually to a new process.


In the Main Grid of a process object, right-click a process and click **Send to Process** > **Selected items** or **Affected items of selected items**.

## Collaboration Comments in File Sharing

 Download

<input type="checkbox"/> File Name	File Size
<input checked="" type="checkbox"/>  SW-201765.SLDPRT	1.09 MB
<input type="checkbox"/>  SW-201807.SLDPRT	110.21 KB
<input type="checkbox"/>  SW-201822.SLDPRT	186.00 KB
<input type="checkbox"/>  SW-201781.SLDPRT	651.77 KB

 Add comment Click file to see comments

<input type="checkbox"/>	User	Date	Comments
	Dave Munder	28/Jan/2024 14:19	Design has undercuts and will be hard to manufacture as is

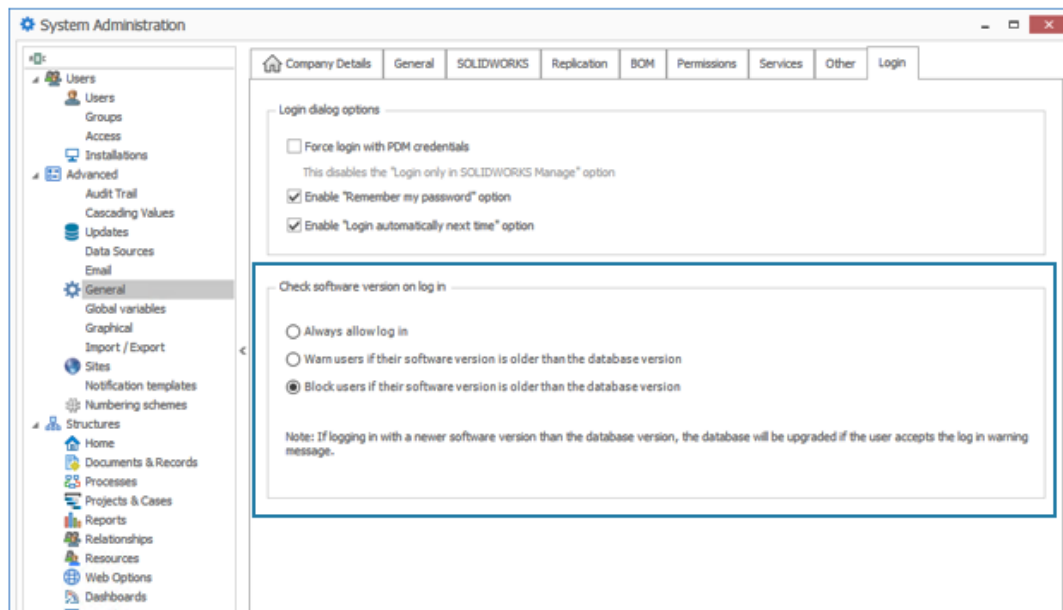
You can specify an option that allows the participants of file sharing to add comments to each file. This makes it easier to communicate with external users about the shared files.

### To enable collaboration comments in file sharing:

1. In the Main Grid of an object, select a record and create a new share or edit an existing file share in the right pane.
2. In the Share dialog box:
  - a. Select **Enable Collaboration Options**.
  - b. Click the **Enable Collaboration Options** link.
3. In the Collaboration Options dialog box:
  - a. Select **Show Comments section**.
  - b. (Optional) Select **External users can add comments**.
  - c. (Optional) Select **Overwrite internal user name in grids** to display a generic name in **Created by** on the file share web page.



## Client Version Check



You can specify an option to restrict users from signing in if they use an older client version than the database version.

### To perform client version checks:

1. In the System Administration tool, click **Advanced > General > Login**.
2. Under **Check software version on log in**, specify an option.

The default setting is **Block users if their software version is older than the database version**.

## Flat BOM Groupings

You can show multiple lines for the same part number for flat bill of materials (BOM) views based on a secondary BOM field value.

For example, consider that a part instance from one subassembly has a reference-specific value of *Spare Part*, and the same part exists elsewhere in the assembly without a value. The flat BOM rolls the quantities of the parts with blank values and the parts with the *Spare Part* value on two separate lines. This functionality is also available in the Plenary Web and you can access it in reports.

Previously, there was no way to separate the same part instances into different groups. All instances rolled into a single line.

### Grouping Instances in Flat BOMs

1. On the BOM property tab toolbar, click **Format > Flat View (advanced) > Group By**.

2. Select a field to use for grouping and click **Apply**.

The BOM shows a line item for the same part number for each value in the selected group by field.

## Adding Automated Task Subject Information

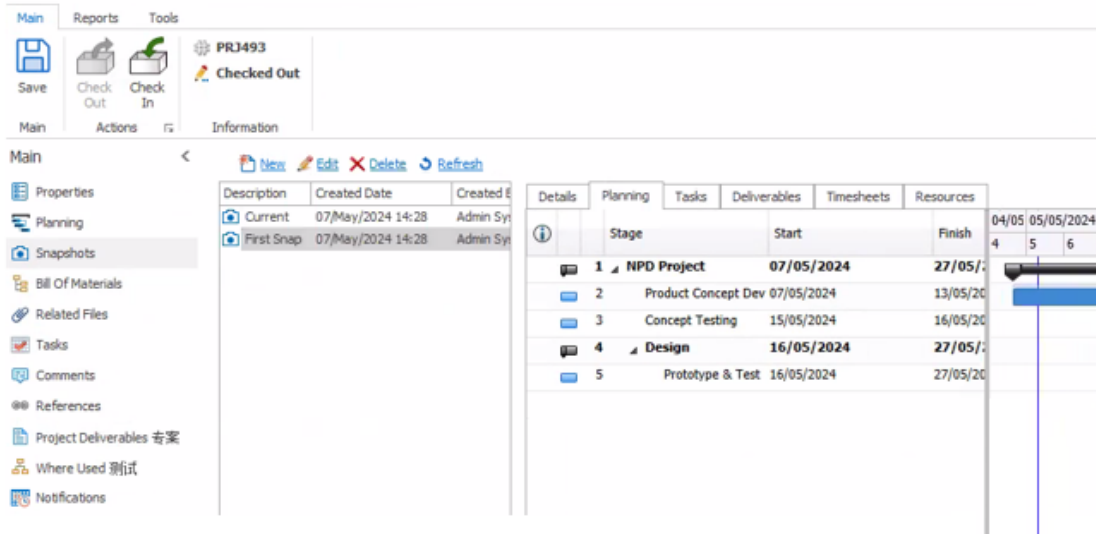
You can include field values from an associated object in the task subject. This makes task subjects associated with Project, Process, and Case objects more meaningful to users.

In earlier releases, you could only add the record part number and the current stage name.

### To add automated task subject information:

1. Edit and process an object.
2. In the System Administration tool, open the Process Wizard.
3. On the Workflow Properties page:
  - a. In the workflow view, select a stage.
  - b. Select **Tasks**.
  - c. Edit an existing task or create a new task.
  - d. In the Task Template dialog box:
    1. At the right end of **Subject**, click the right arrow icon and select a field.
    2. (Optional) Add static text or additional fields.
  - e. Click **Save** then **Close**.

## Project Snapshots



You can capture project record details at certain points in time to create a history of the changes made to a project record.

Snapshots are available in a property card tab named Snapshots. The tab's left pane displays the snapshots that you created in addition to the current record. You can compare the snapshot information to the current record and to other snapshots. The right pane displays information for the selected snapshot or for the current record. Information in the right pane includes:

- **Details.** Displays the record's field values.
- **Planning.** Shows the work breakdown structure and Gantt chart.
- **Tasks.** Lists the tasks as they were when you took the snapshot, including progress, status, and assignment information.
- **Deliverables.** Shows the deliverables and their lifecycle statuses.
- **Timesheets.** Displays the timesheets connected to the project.
- **Resources.** Lists the resources assigned to the project at the time of the snapshot.

### Creating Project Snapshots

1. Edit a project object.
2. In the System Administration tool, open the Process Wizard.
3. On the Property Tabs page:
  - a) Select **Snapshots**.
  - b) Select the users or groups to have access to the Snapshots tab.
  - c) Click **Next** and make any other changes to the project object.
4. Select the completed wizard page and click **Finish**.
5. Open a project record and check it out.
6. On the Snapshots tab, click **New**.

7. Enter a name and comment for the snapshot.  
The snapshot appears in the list with the **Current** record.
8. Make changes to the project record.  
For example, add a project stage and tasks for the new stage.
9. Click **Save**.
10. Select the Snapshots tab.
11. Select the snapshot and **Current** record, then compare the information on the Planning tab.

## Tasks from Cancelled Processes

You can control the status of associated tasks from cancelled processes. This eliminates leftover tasks that you can see after cancelled processes. You can leave edited, unedited, or completed tasks as unchanged, delete them, or change them to completed when the associated process is canceled.

### To specify what to do for tasks from cancelled processes:

1. In the System Administration tool, open the Process Wizard.
2. On the Options page, under **Task Options**, specify options for **When a process is cancelled**.

## Application Programming Interface

A web-based API is available. You can use the API to get data out of SOLIDWORKS Manage and update or add records.

You install the API through the SOLIDWORKS Manage Server installer in the SOLIDWORKS Installation Manager. You can access the documentation on the website included in Internet Information Services (IIS) with the **Browse Website** link.

## Creating New Process Records from Existing Process Records

You can create new process records from existing process records to capture the field values and other attributes from the source record.

1. In the Main Grid of a process object, right-click an existing process record and select **New From**.
2. Make changes in the properties area and select content to copy under **What do you want to copy**.
3. Click **OK**.

## Send to Process for Affected Items

You can send affected items from one process to a new process.

The new process can be any process that accepts the selected record types.

If the selected affected items are in a process that has not completed but the affected items have a **Change Status** output, you cannot add the affected items to a new process that also has a **Change Status** output.

1. Select an existing process record or open its property card.
2. On the Affected Items tab, right-click an affected item record and select **Send to Process**.

You can select multiple affected items.

3. In the Select dialog box, select a process object for the new process record.  
The new process record appears with the selected records added as affected items.

## Affected Items in Microsoft File Explorer

You can navigate to the Microsoft® File Explorer location for a SOLIDWORKS PDM file that is an affected item in a process.

1. Select an existing process record or open its property card.
2. On the Affected Items tab, right-click an affected item record and select **Show in Windows Explorer**.

File Explorer opens with the affected item selected.

## Thumbnails for BOM Copy From

When you copy data into a Bill of Materials (BOM) using **Copy From**, the Select Record dialog box includes thumbnail images in the search results area. Thumbnails make it easier to understand the data that you copy.

## Installing the SOLIDWORKS Manage Web API

You can install the Manage Web API in the SOLIDWORKS PDM InstallShield Wizard. During the installation, you can either use the default port or specify another value for the Http port.

In addition, in the SOLIDWORKS Installation Manager, you can install the Manage Web API on the SOLIDWORKS Manage Server page and specify the Http port there as well.

## SOLIDWORKS Simulation

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This chapter includes the following topics:

- **Automatic Detection of Underconstrained Bodies**
- **Bonding Interactions with Offset**
- **Contact Penalty Stiffness for Shells**
- **Contact Penalty Stiffness Control for Nonlinear Studies**
- **Edge Weld Connector**
- **Enhanced Pin Connector**
- **Exclude Bodies from Analysis**
- **General Spring Connector**
- **Geometry Correction for Surface-to-Surface Bonding**
- **Mesh**

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

### Automatic Detection of Underconstrained Bodies

System Options   Default Options

General  
Default Library  
Messages/Errors/Warnings  
Email Notification Settings  
Simulation sensors

What's Wrong messages

- ☒ Show errors
- ☒ Show warnings

Load/Fixture symbol quality

- ☒ Load all simulation studies when opening a model (requires to open a model)
- ☒ Automatically update beam joints when study is activated
- ☐ Save file after meshing and after the analysis completes
- ☐ Automatically detect underconstrained bodies

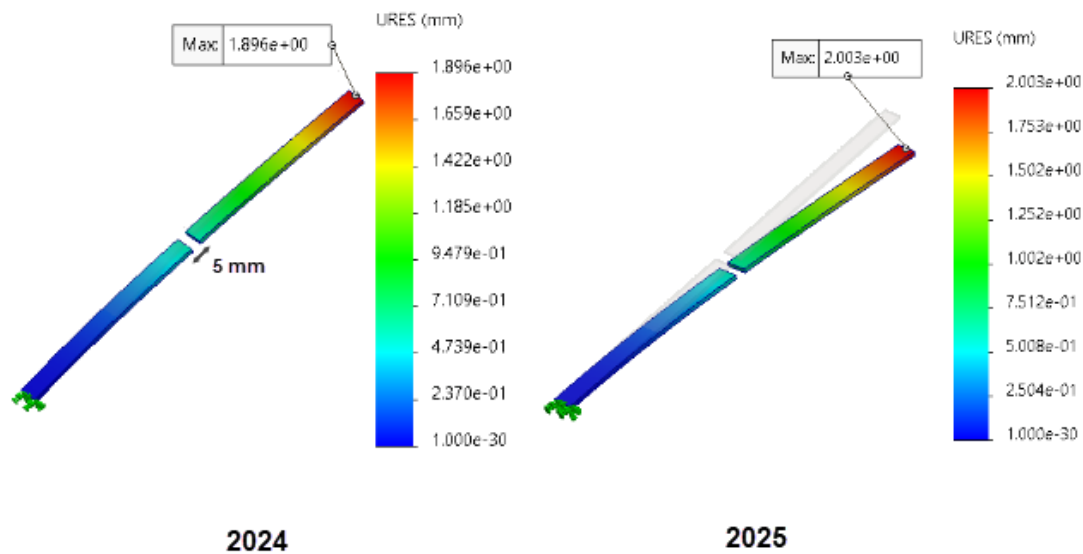
You can detect rigid body modes at the early stage of a linear static study's solution.

The option **Automatically detect underconstrained bodies** is available from the **System Options - General** dialog box. This option detects bodies that are not sufficiently constrained during simulation and can exhibit translational or rotational rigid body modes.

When the solver detects rigid body modes, you have the option to continue with the solution, or stop the solution and review the rigid modes using the **Underconstrained Bodies** tool.

The automatic detection of rigid bodies is available for linear static studies.

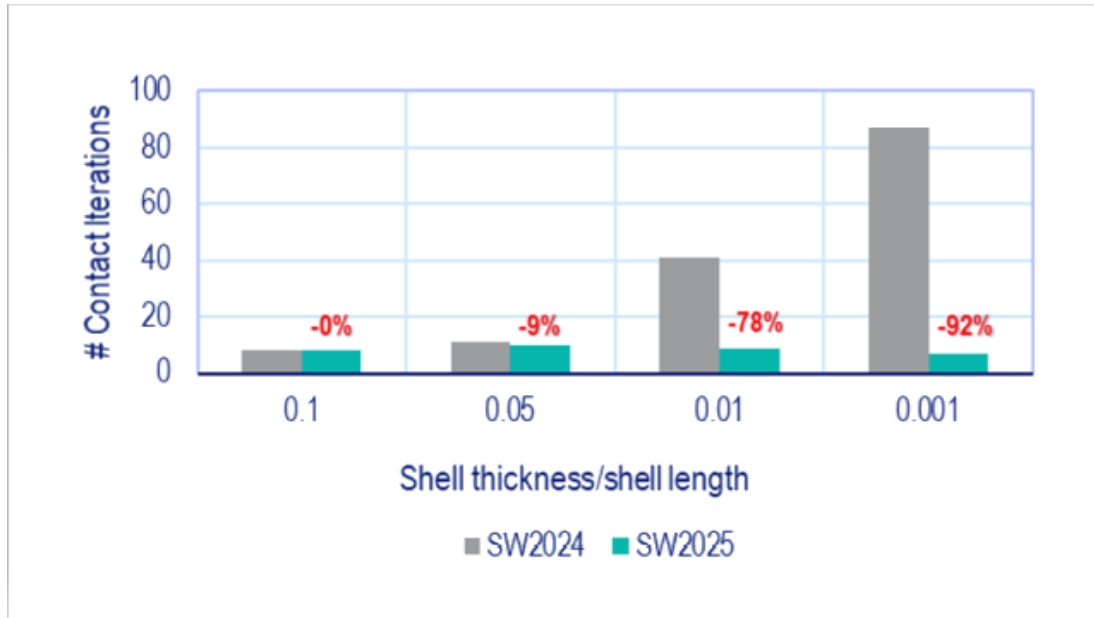
## Bonding Interactions with Offset



The enforcement of node-to-surface bonding interactions between geometries within a user-defined gap is improved.

This enhancement improves the accuracy for bonding offset defined by a user-defined **Gap range for bonding**. You can expect to see improved solution accuracy for all bonding interactions (solid-solid, shell-shell, and solid-shell) that are based on either a draft-quality or high-quality mesh. The studies that support this enhancement include Linear Static, Frequency, Buckling, Linear Dynamics, Fatigue, Design Scenario, and Pressure Vessel.

## Contact Penalty Stiffness for Shells

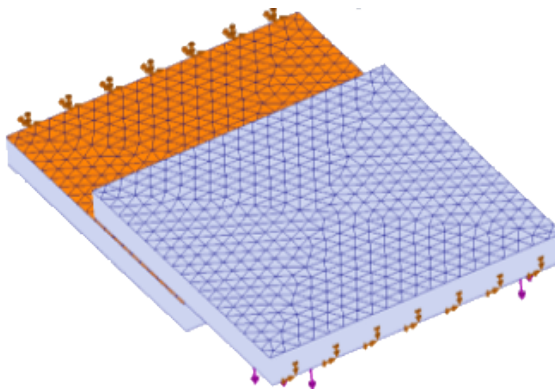


A new algorithm is introduced to apply penalty stiffness for contact interactions of shells. This enhancement improves performance and accuracy for a large range of shell thickness ratios.

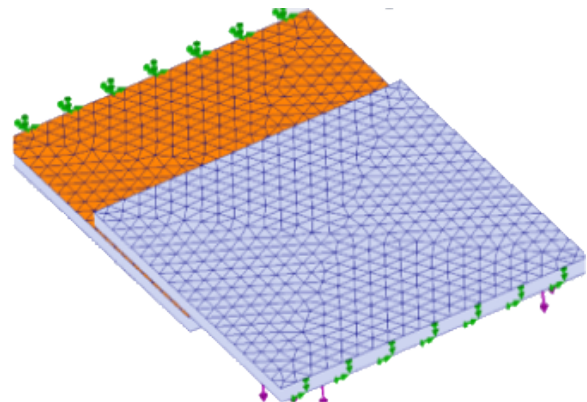
Shell thickness ratio = shell thickness / characteristic length of shell

The image shows the performance improvement for contact interactions depending on several shell thickness ratios.

The force magnitude applied for each test case was adjusted for the different shell thickness/shell length ratios to maintain a small displacement range and similar maximum displacements across all test cases.

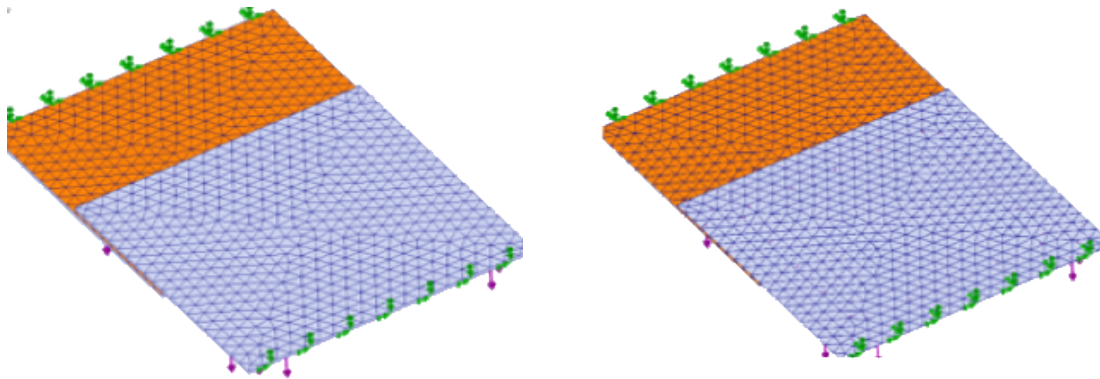


shell thickness / shell length = 0.1



shell thickness / shell length = 0.05



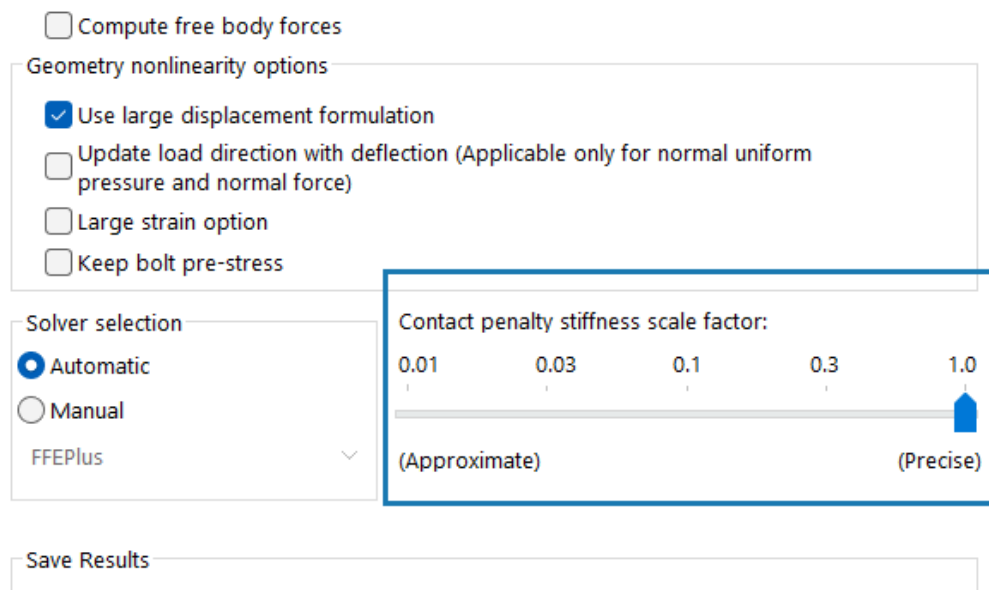


shell thickness / shell length = 0.01

shell thickness / shell length = 0.001

The penalty stiffness applies to shell-to-shell, solid-to-shell, surface-to-surface, and edge-to-surface contact in linear static studies.

## Contact Penalty Stiffness Control for Nonlinear Studies

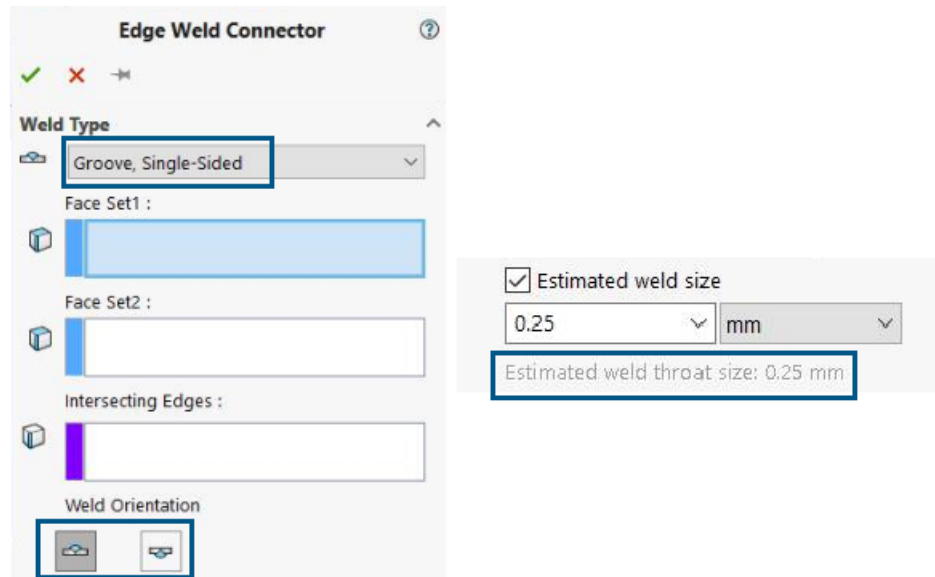


You can adjust a scale factor for the penalty stiffness applied to surface-to-surface contact interactions when solving nonlinear studies.

The default value for the contact penalty stiffness factor is 1.0, which yields the most accurate solution. To obtain an approximate solution and assess design iterations faster, you can specify a value lower than 1.0.

You can set the study-level scale factor for the penalty stiffness in the Nonlinear- Static dialog box.

## Edge Weld Connector



Several enhancements for the Edge Weld connector improve its usability.

- The program calculates the **Estimated weld throat size** when you define edge weld connectors in the Edge Weld Connector PropertyManager. The formulas for the calculation of the **Estimated weld throat size** are given in the table.

Weld Type	Estimated Weld Throat Size
Fillet	<b>Estimated weld size</b> * square root (2)
Groove	<b>Estimated weld size</b>

- The icons for **Weld Orientation** in the Edge Weld Connector PropertyManager for the **Groove, Single-sided** type of connectors are updated to show an accurate representation of the edge weld type.
- The **Weld Check Plot** annotation also lists the **Calculated weld throat size** and the **Estimated weld throat size** for each edge weld connector.

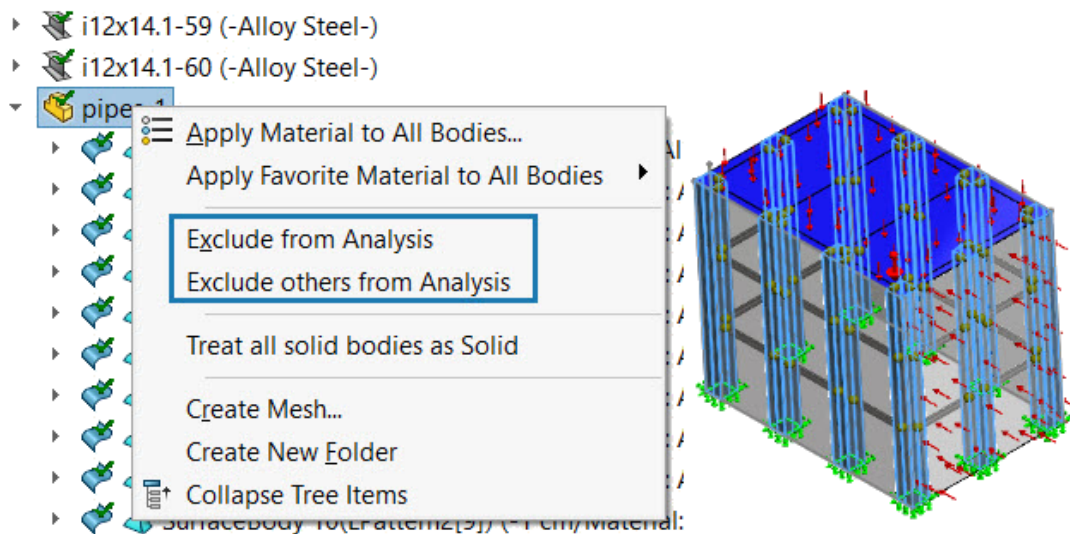


In previous releases, when the number of nodes were very large, only a subset of the nodes participated in the distributed coupling constraints. In SOLIDWORKS Simulation 2025, the distributed coupling constraints for pin connectors include all the nodes on the cylindrical surfaces.

The solution time with the FFEPlus iterative solver for similar studies is unchanged in SOLIDWORKS Simulation 2025. However, the stress results are more accurate because all nodes are considered in the distributed coupling formulation.

This enhancement is available for Linear Static studies, along with the associated Fatigue, Design, and Pressure Vessel Design studies.

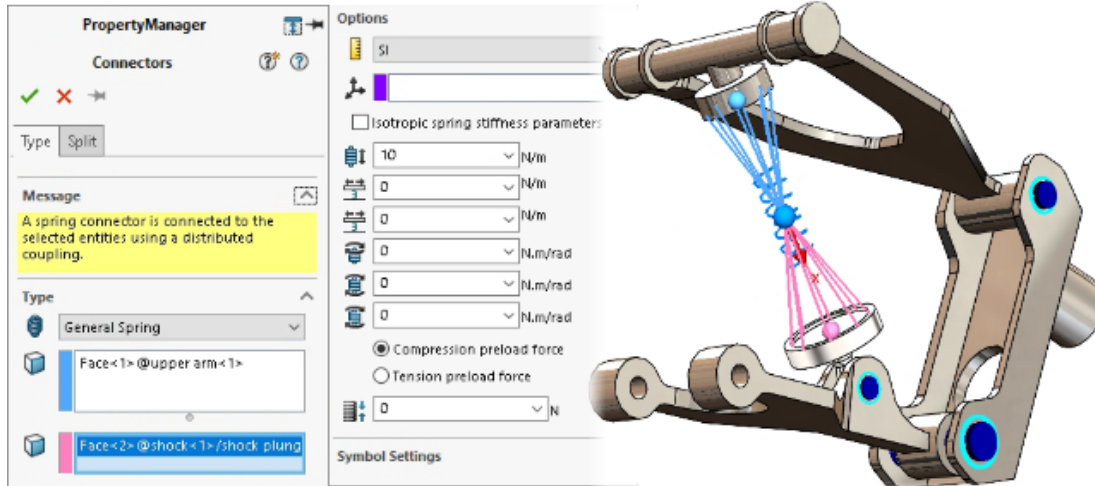
## Exclude Bodies from Analysis



You can exclude multiple bodies from an analysis.

From a Simulation study tree, select a folder under the **Parts** node and use the shortcut menu to exclude all bodies in the selected folder from the analysis.

## General Spring Connector



You can specify a general spring connector between flat, nonflat, and concentric cylindrical surfaces.

The general spring connector uses distributed coupling to establish an enhanced spring connector formulation that improves the performance and accuracy of the simulation studies.

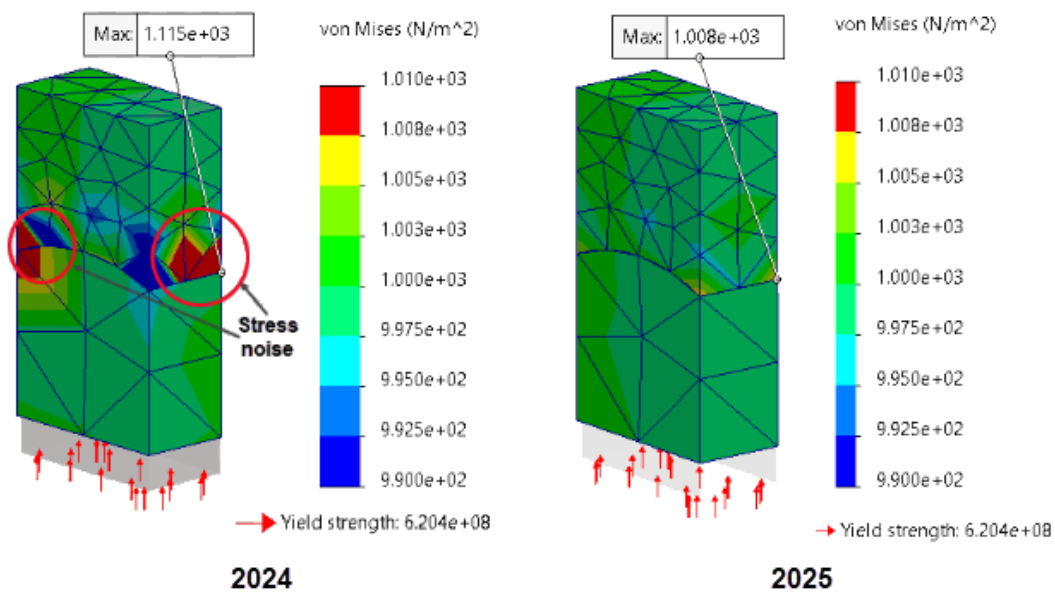
To accurately represent the general spring connector, you can define up to six stiffness parameters using a local coordinate system.

The general spring connector is available with SOLIDWORKS Simulation Professional and SOLIDWORKS Simulation Premium.

### To open the General Spring PropertyManager:

In the Simulation study tree, right-click **Connections**  and click **General Spring** .

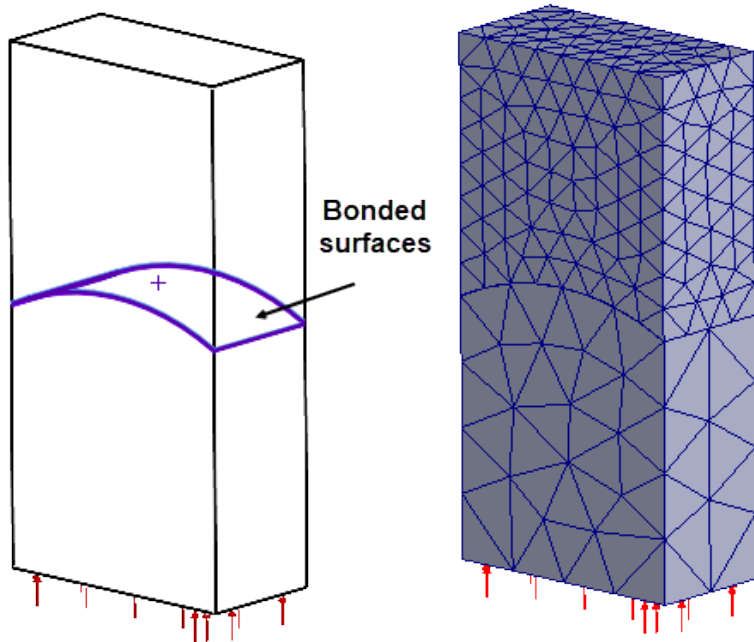
## Geometry Correction for Surface-to-Surface Bonding



Simulation accuracy is improved for studies with bonded curved surfaces (surface-to-surface bonding formulation) when the mesh sizes of the source and target surfaces differ.

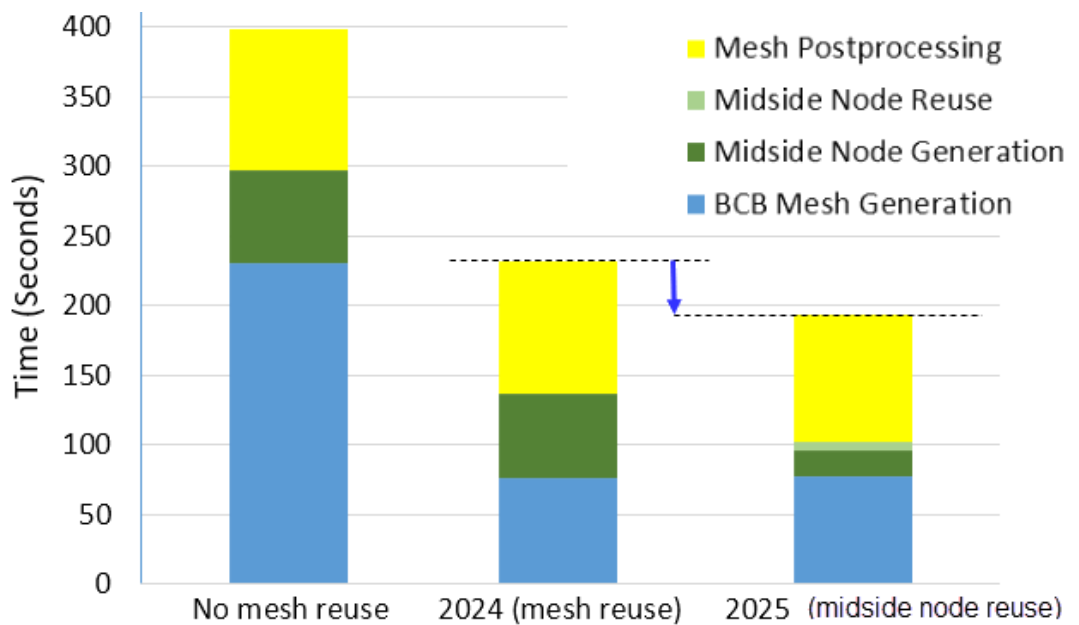
The algorithm that enforces surface-to-surface bonding integrates geometry correction factors that improve the representation of curved surfaces of cylindrical, spherical, and conical geometries. The integration of surface geometry correction reduces the stress noise at the vicinity of bonded curved surfaces, thus improving the solution accuracy.

The image above shows the stress noise reduction at the boundary where solid-to-solid bonding with geometry correction is applied between two curved surfaces. The geometry of the bonded surfaces is shown in the image below.



The studies that support this enhancement include Linear Static, Frequency, Buckling, Linear Dynamics, Fatigue, Design Scenario, and Pressure Vessel.

## Mesh



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

The Blended curvature-based mesher creates the midside nodes of the higher-order elements once, and reuses the midside node positions across repeated identical parts, thus saving meshing time. The mesh performance improvement is more prominent for

assemblies with many repeated parts that have curved surfaces and are meshed with a high-quality mesh.

The image shows the total mesh time reduction for an assembly with 450 parts.



# 18

## SOLIDWORKS Visualize

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This chapter includes the following topics:

- **Temporary Offline Mode Support for SOLIDWORKS VISUALIZE Connected (2025 FD03)**
- **Splitting Parts (2025 SP3)**
- **Improved Import of PBR Appearance Information for glTF and USDZ Format and Support for SketchUp 2024 (2025 SP3)**
- **Updated System Info Checks and Removal of OpenCL Version Requirement (2025 SP3)**
- **Denoiser Support for CPU Rendering with Stellar Engine (2025 SP2)**
- **Random Position, Rotation, and Scale for Objects (2025 SP2)**
- **Enhancing Images with the Camera Bokeh Effect (2025 SP1)**
- **Fast Mode Updates for Stellar Render Engine (2025 SP1)**
- **Import Improvements (2025 SP1)**
- **Updates for DSPBR Shading Model Appearances (2025 SP1)**
- **Support for Distributed Rendering in SOLIDWORKS Visualize Connected (2025 SP1)**
- **Fading the Ground Floor**
- **Added Fast Rendering Mode for Stellar**
- **Render Engine Selection**
- **Photorealistic Rendering in SOLIDWORKS with the SOLIDWORKS Visualize API**
- **Visualize Boost Redesign**

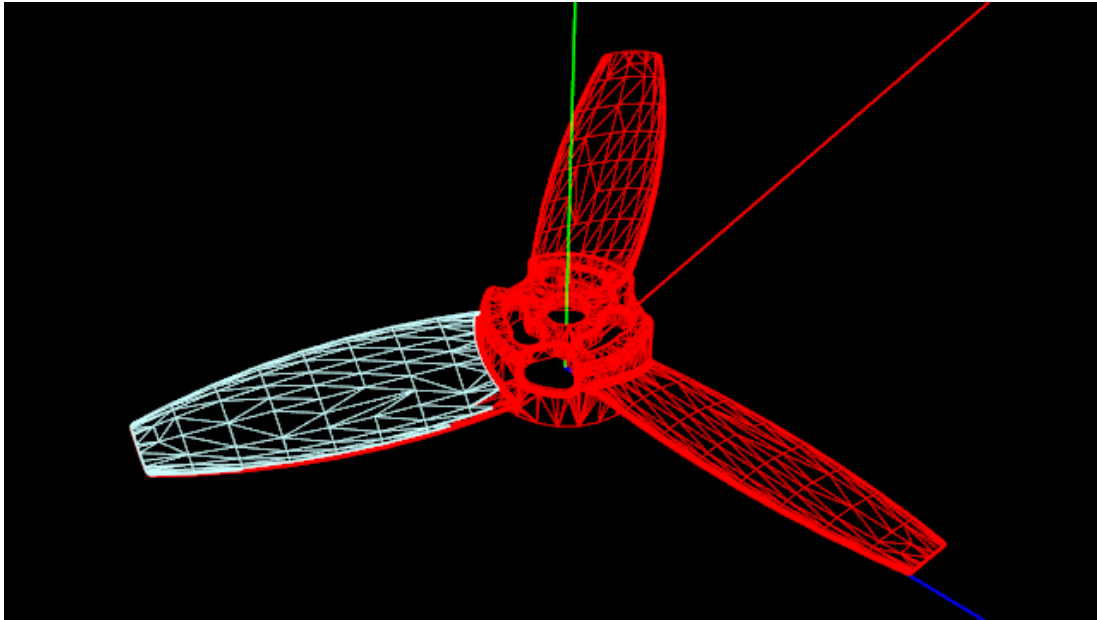
SOLIDWORKS® Visualize is a separately purchased product that you can use with SOLIDWORKS Standard, SOLIDWORKS Professional, SOLIDWORKS Premium, and SOLIDWORKS Ultimate, or as a completely separate application.

### Temporary Offline Mode Support for SOLIDWORKS VISUALIZE Connected (2025 FD03)

SOLIDWORKS Visualize Connected supports temporary offline mode. If you lose connection during a session, you can continue working offline with local files. The app attempts to reconnect and prompts you to restart when the connection returns.

See **Work Offline When a Connection is Unavailable**.

## Splitting Parts (2025 SP3)



Additional options let you customize the part splitting functionality when selecting facets in large parts.

Splitting is essential because you cannot assign appearances to faces (or to anything in the hierarchy lower than the part level). If you want to assign multiple appearances to a single part, splitting is required.

When you use part splitting, you can:

- See geometry more easily. SOLIDWORKS Visualize renders all parts in the scene in wireframe against a dark background for a better understanding of the underlying mesh structure.
- Specify parts to split. The software shows a preview of the split mesh in blue wireframe and the part to split in red wireframe. Any part that is not in the part splitting display shows as faded wireframes.
- Use specified tools for more control over face selection. You can specify **Facet** or **Paint Brush Mode** to select faces.

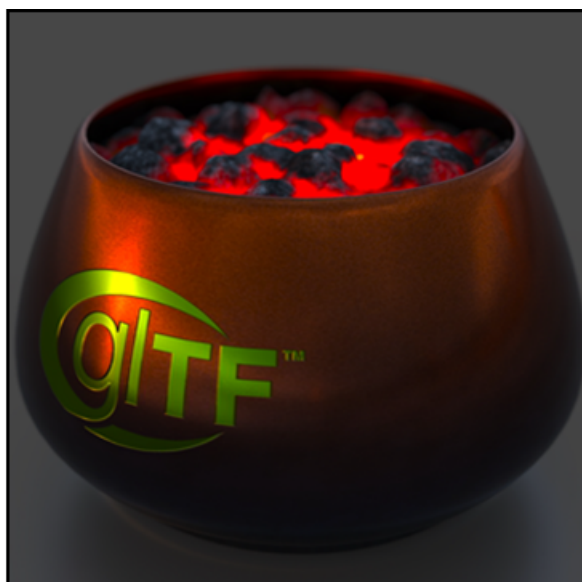
### To split parts:

1. Do one of the following:
  - Click **Tools > Split Part**.
  - In the viewport or Palette, right-click a part and click **Edit > Split Part**.
  - Press **CTRL+ALT+P**.
2. In the dialog box, specify the **Selection Mode**:

Option	Description
<b>Facet</b>	Lets you click part facets to split. Adjust the <b>Facet Angle Tolerance</b> to specify the part tolerance. You can also use box selection in <b>Facet</b> mode.
<b>Paint Brush</b>	Lets you click and drag to select multiple part facets to split. Adjust the <b>Paint Brush Radius</b> to fine tune the selection area.
<b>Show Only Active Part</b>	Displays only the split mesh and the part undergoing the split instead of the entire part. Remaining geometry is hidden or rendered in gray wireframe.

3. In the viewport, click a face or drag a selection area to see a preview of the split part.  
The split mesh changes to a blue wireframe and the part undergoing the split turns to a red wireframe.
4. Optional: Press **SHIFT** to remove portions from the mesh selection or **CTRL** to add portions to the mesh selection.
5. Click **Execute Split**.

## Improved Import of PBR Appearance Information for glTF and USDZ Format and Support for SketchUp 2024 (2025 SP3)



You can achieve better results when importing Physically Based Rendering (PBR) materials from glTF and USDZ files to SOLIDWORKS Visualize. This support provides more detailed and realistic appearances.

**Benefits:** Imported models look more realistic with improved material details. You can also open SketchUp 2024 files directly in SOLIDWORKS Visualize.

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- © 2017, Khronos Group. Khronos Trademark or Logo
  - Noncopyrightable logo for glTF logo

## Updated System Info Checks and Removal of OpenCL Version Requirement (2025 SP3)

Hardware requirements have been streamlined and brought up to date with the 3DS Stellar Fast and Accurate modes, as well as the AMD ProRender Accurate mode.

**Benefits:** These updates help ensure that your machine meets current rendering requirements and avoid potential issues.

The System Information dialog box includes the following updates:

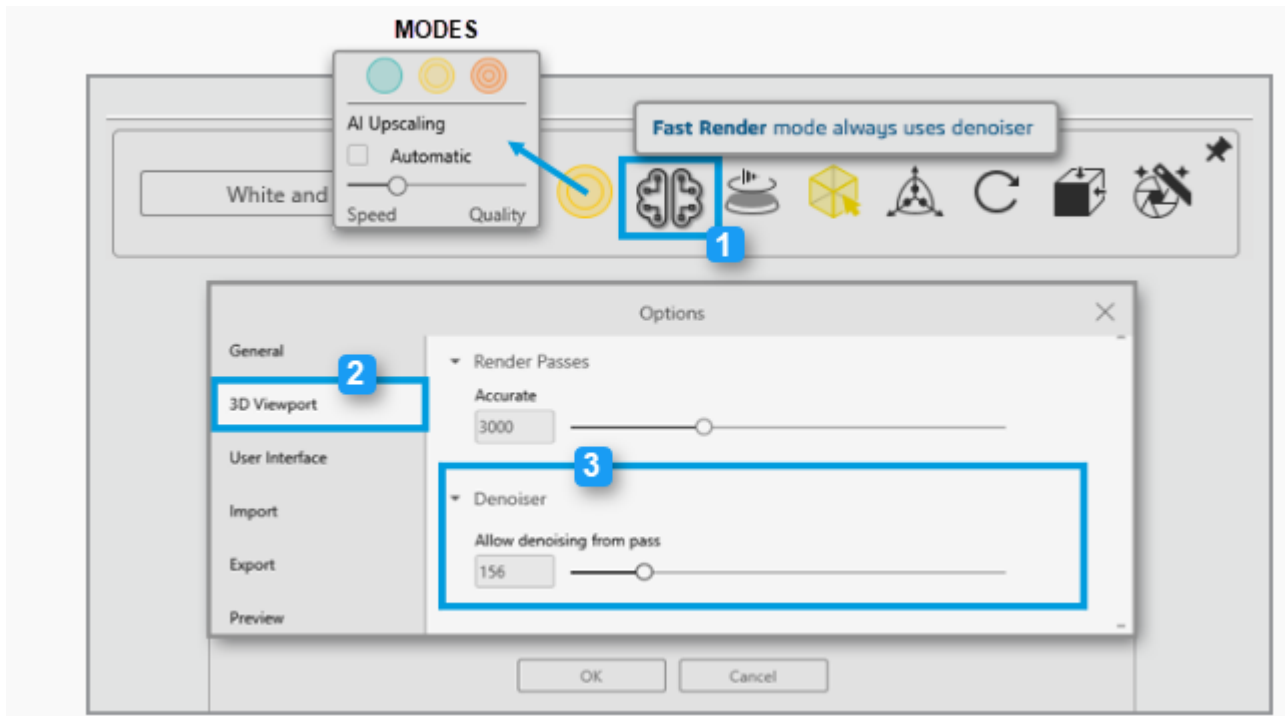
Component	Previous Requirement	Updated Requirement
System Memory	8 GB	<b>16 GB</b>
Free Disk Space	2 GB	<b>10 GB</b>
Graphics Memory	1 GB	<b>4 GB</b>
Vulkan Version		<b>1.3</b> required for 3DS Stellar Fast
		<b>1.2</b> required for AMD ProRender
OpenCL Version	1.2 or higher	No longer required

Refer to [the latest SOLIDWORKS Visualize system requirements](#) for a detailed breakdown.

Raytracing Initialization Notice: If you have not used raytracing before, a warning appears under **Tools > Options > 3D Viewport** in the **Render Device** option. You must initialize the render engine before selecting a GPU device.

The render engine initializes automatically the first time you switch to a raytraced render mode such as 3DS Stellar Accurate or AMD ProRender.

## Denoiser Support for CPU Rendering with Stellar Engine (2025 SP2)



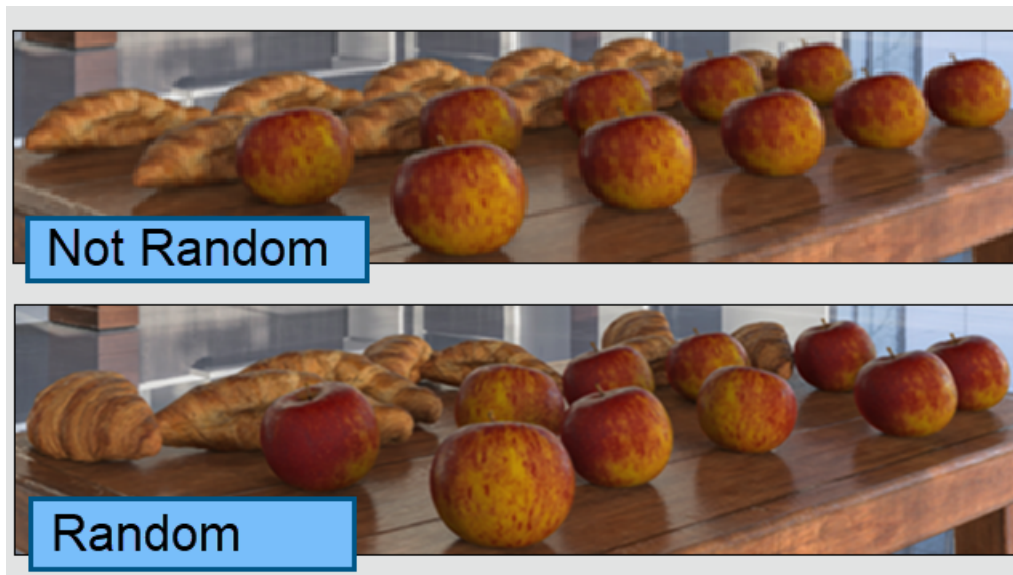
SOLIDWORKS Visualize supports CPU denoising with the 3DS Stellar Physically Correct engine.

**Benefits:** Denoising reduces noise and improves render quality with fewer passes, helping CPU users achieve cleaner results faster. Previously, denoising was only available for GPU rendering.

Key changes include:

- You can turn the denoiser on and off in CPU mode.
- The **Initialize Denoiser** and **Show Button in the Main Toolbar** options are removed from **Tools > Options > 3D Viewport > Denoiser** section.
  - The denoiser is always available in the Main Toolbar when using:
    - 3DS Stellar Physically Correct (CPU or GPU)
    - AMD Radeon™ ProRender (GPU)
- You can adjust the denoiser starting pass in **Tools > Options > 3D Viewport > Denoiser**.
- When using 3DS Stellar Physically Correct in Fast mode, denoising is always enabled and cannot be turned off. The Denoiser button in the Main Toolbar stays toggled on but is disabled, with a tooltip explaining its status.

## Random Position, Rotation, and Scale for Objects (2025 SP2)



You can easily apply a random amount of position, rotation, and scale values to a group of patterned instances and other selected objects.

**Benefits:** This feature helps create more realistic renders when dealing with collections of the same object by randomly adjusting their position, rotation, or scale.

You can randomize:

- **Position (X, Y, Z):** Adjusts the position of objects randomly in the selected axis.
- **Rotation (X, Y, Z):** Adjusts the rotation of objects randomly about the selected axis.
- **Scale (X, Y, Z):** Adjusts the scale of objects randomly in the selected axis.
- **Scale All:** Adjusts the scale of objects in all axes by a random amount.

You can randomize transformations, when selecting multiple parts, groups, or models. When you enable **Randomize** in the **Relative Transform** tool, each object gets a different random value within the chosen range. For example:

- **Position:** Objects move randomly +/- the value entered.
- **Rotation:** Objects rotate randomly +/- about the selected axis.
- **Scale:** Objects resize randomly within the given range. If the value is below 1.0, objects scale between that value and 1.0. If the value is above 1.0, the scale of the object is between 1.0 and the entered value.
- **Scale All:** Objects scale a random amount in all axes. If the value is below 1.0, objects scale between that value and 1.0. If the value is above 1.0, the scale of the object is between 1.0 and the entered value.
- **Random Seed:** Each random seed generates a unique set of random values. Using the same random seed always produces the same random values. This is useful when you find a seed that gives you a result you like. You can reuse it to achieve the same outcome for a given input

## Enhancing Images with the Camera Bokeh Effect (2025 SP1)

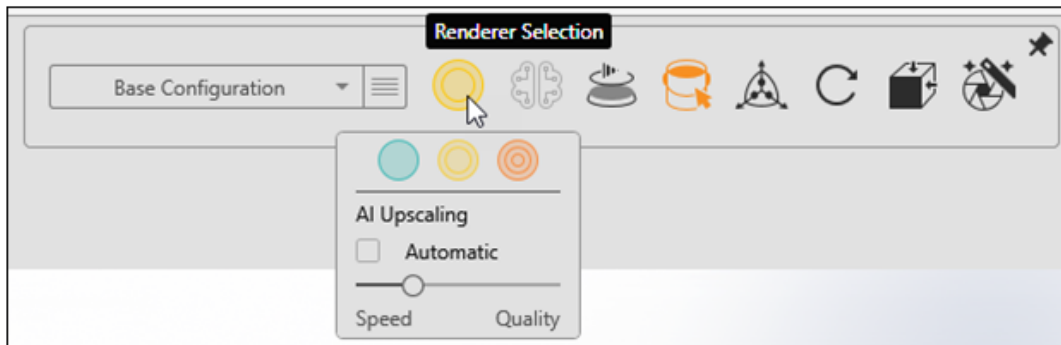


The **Depth of Field (DOF) Bokeh Effect**, seen in photography and 3D rendering, creates blur in out-of-focus areas, giving highlights a soft and circular or polygonal shape. You can think of blurred streetlights in a night scene as an example. In traditional photography, lens aperture blades shape these highlights.

With SOLIDWORKS Visualize, you can fine-tune this effect by adjusting **Blade Count** and **Blade Angle**, allowing you to customize the shape of Bokeh highlights. To access these parameters, go to **Palette > Camera > General > Depth of Field** and select both **Depth of Field** and **Bokeh Effect**.

- **Blade Count:** Specifies the number of aperture blades that shape the Bokeh. Higher values result in a smoother, more circular effect.
- **Blade Angle:** Adjusts the orientation of the **Bokeh Effect** from 0° to 360°.

## Fast Mode Updates for Stellar Render Engine (2025 SP1)



The latest updates to **Fast** mode for the Stellar render engine enhance performance, usability, and access to critical settings.

- **AI Upscaling.**

- **Fast** mode helps you balance performance and visual quality. Depending on your hardware, the option does not appear.
- **Automatic** adjusts the **AI Upscaling** mode based on your viewport resolution. This option is helpful if you frequently change the viewport size.
- **Speed** maximizes responsiveness with lower detail. **Quality** provides the sharpest visuals with reduced performance. Moving the slider in between these options offers a balanced middle ground between interactivity and image clarity.

- **Camera Motion Blur.**

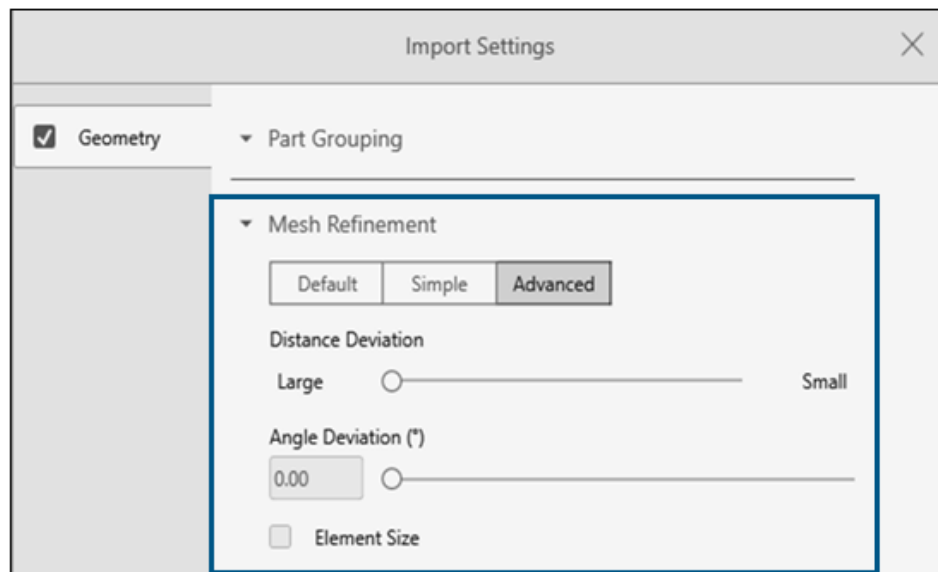
By adding natural blur to moving objects, **Fast** mode creates smoother visuals while maintaining performance.

- **Simplified Controls.**

You no longer need to specify pass limits or time in the Render Wizard for final renders. This removal ensures high-quality results and lets you focus more on creativity.



## Import Improvements (2025 SP1)



Import improvements in SOLIDWORKS Visualize improve format loading, and give you more control over mesh refinement quality.

SOLIDWORKS Visualize uses a new format-loading component, replacing older import methods. This update improves mesh refinement quality for better detail and accuracy during import. It also handles materials, textures, and specific file types more efficiently, speeding up visualization loading. In addition, the mesh refinement experience is designed to align more closely with SOLIDWORKS, providing a more consistent workflow.

The Geometry tab in the Import Settings dialog box offers the following **Mesh Refinement** modes:

- **Default**

Delivers the fastest import speed while preserving full material properties, including textures. This mode uses either existing tessellation data or default settings.

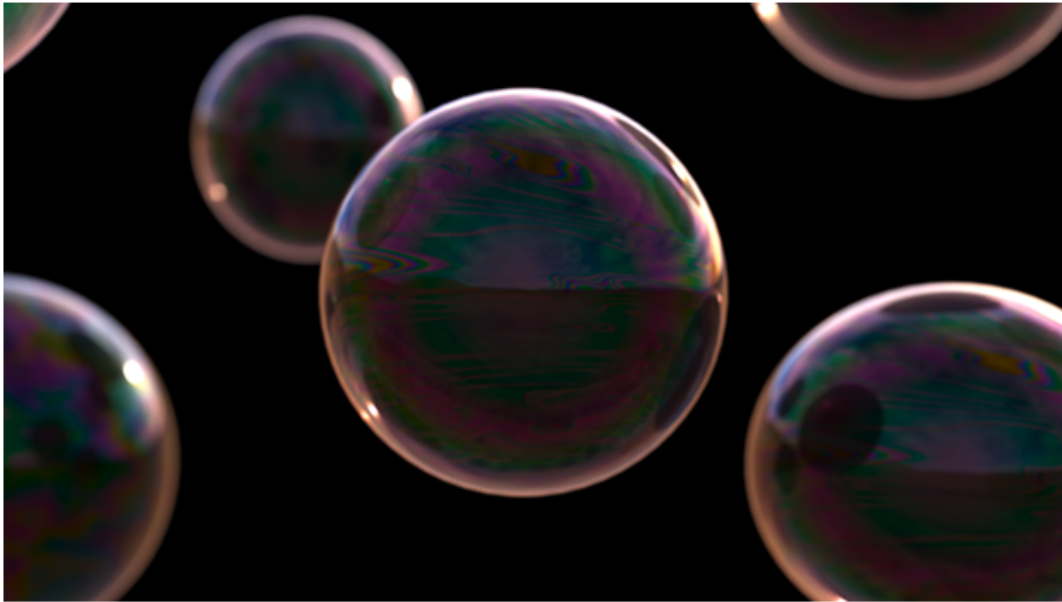
- **Simple**

Supports basic mesh refinement with limited material properties (color only). You can adjust mesh refinement using a single slider and later retessellate part of the model using the Models tab in the **Palette**, which provides the same **Mesh Refinement** controls.

- **Advanced**

Offers more flexibility to customize mesh refinement settings, although material properties are limited to color only. Similar to **Simple** mode, this mode lets you retessellate parts of the model after import using the Models tab in the **Palette**.

## Updates for DSPBR Shading Model Appearances (2025 SP1)



SOLIDWORKS Visualize enhances appearance workflows for the DSPBR Shading Model with the **Translucency Color** and **Thin Film** parameters.

These parameters offer more control over how light interacts with materials:

- **Translucency Color** lets you add a diffuse color to transparent materials, similar to the **Subsurface Color**. It is helpful for simulating objects like translucent curtains.
- **Thin Film** effect simulates light diffraction, creating colorful patterns on materials. This is ideal for effects like soap bubbles or oil on water.

For older DSPBR appearances, click **Convert** next to the **Appearance Type** to update them and access the latest features and controls. A tooltip shows the current version and the version to which it will be converted. New appearances automatically include these parameters in the user interface.

## Support for Distributed Rendering in SOLIDWORKS Visualize Connected (2025 SP1)

SOLIDWORKS Visualize Connected supports distributed rendering through Visualize Boost.

To facilitate this functionality, the SOLIDWORKS Visualize Connected interface includes Boost controls that are identical to those in the SOLIDWORKS Visualize desktop app.

- **Tools > Options > Boost**

The Boost tab displays the **Coordinator IP Address**, **Boost Port**, and **Boost Status** for easy access and management.

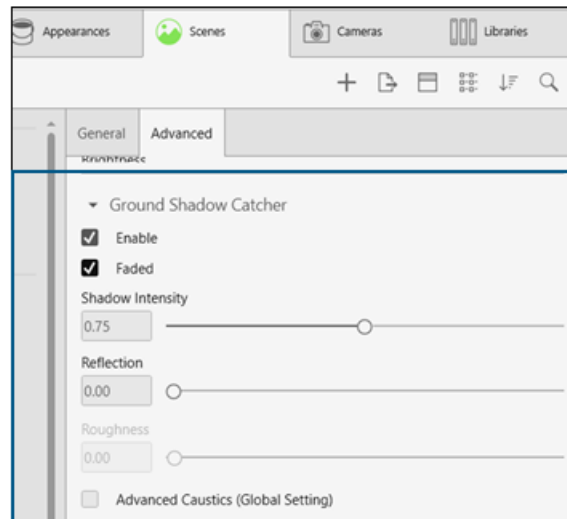
- **Heads-Up Display (HUD) and Render Wizard > Quality**

These areas include **Boost Status** controls, giving you visibility over Boost's activity and status during rendering.

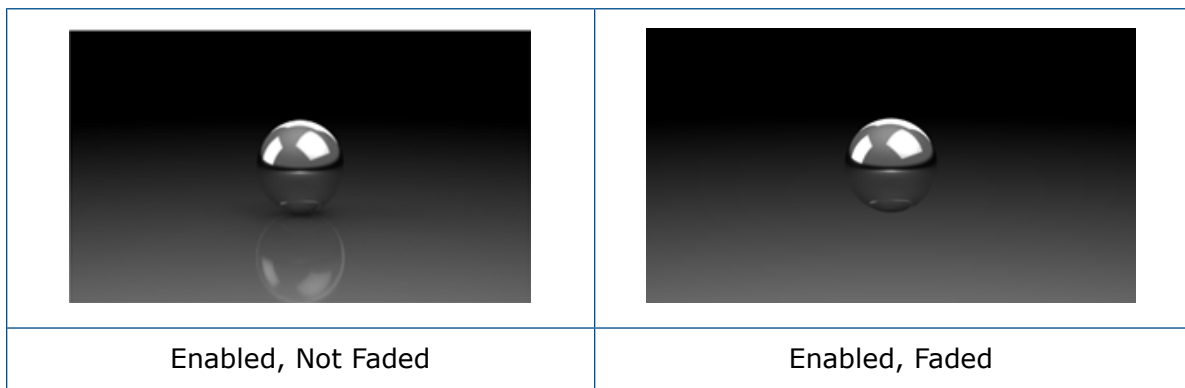
Avoid running Visualize Boost on the same machine as SOLIDWORKS Visualize Connected.

For details on installing and configuring Visualize Boost, see [Visualize Boost Redesign](#) and SOLIDWORKS Visualize Help.

## Fading the Ground Floor



In SOLIDWORKS Visualize, you can fade the ground floor similar to fading parts. This makes the ground invisible while still affecting reflections and shading of nearby parts.




During editing and post-processing, there are occasions where hiding the ground floor becomes necessary. Doing so may alter the visual representation of parts due to the absence of interactions between the floor and parts.

You can access the **Faded** property **Palette > Scenes > Advanced > Ground Shadow Catcher**.

This feature is supported exclusively in **Accurate** mode and is not accessible in **Preview** or **Fast** mode.

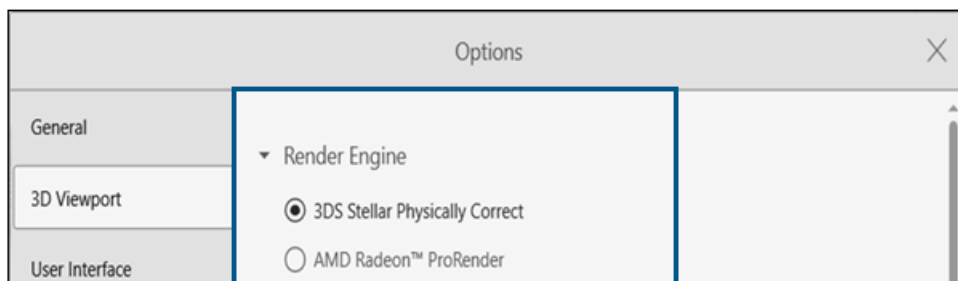
## Added Fast Rendering Mode for Stellar



SOLIDWORKS Visualize offers **Fast**  mode rendering with the Stellar render engine, providing real-time interactive rendering for both the Visualize viewport and offline renders.

It uses the Vulkan ray-tracing API and Deep Learning AI technology to achieve real-time ray-tracing performance, making it ideal for next generation video cards and high resolutions.

## Render Engine Selection




With the completion of the implementation of the Stellar Physically Correct rendering engine, SOLIDWORKS Visualize has discontinued support for NVIDIA Iray.

Consequently, the option to choose NVIDIA Iray as the rendering engine has been removed from the **Tools > Options** menu, so users can no longer select it.

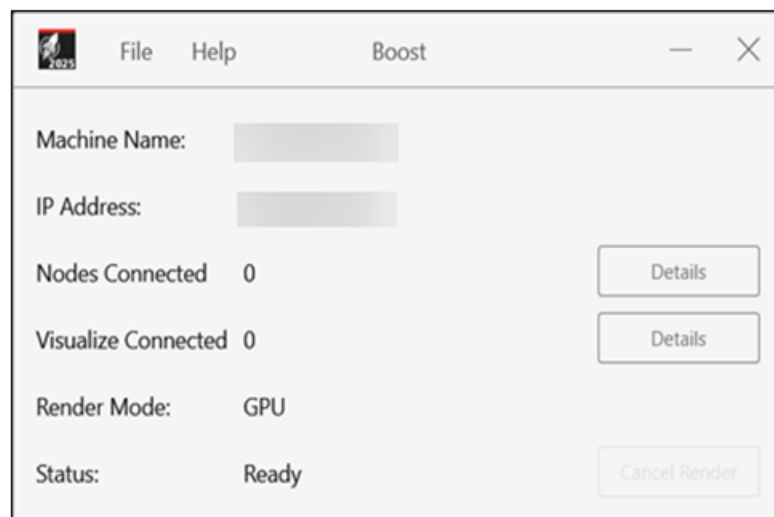
## Photorealistic Rendering in SOLIDWORKS with the SOLIDWORKS Visualize API

Using the SOLIDWORKS Visualize API, you can create functionality for photorealistic renderings of SOLIDWORKS models.

This API, available through the SOLIDWORKS Visualize Add-In, allows you to either render SOLIDWORKS documents directly or convert them into Visualize project files.

For API assistance, click the  **Help > API Help**.

## Visualize Boost Redesign



Visualize Boost has undergone a significant redesign, introducing enhanced capabilities tailored for managing SOLIDWORKS Visualize render tasks across multiple machines.

With a simplified and intuitive setup process, configuring render jobs across a network is more efficient than ever.

The latest iteration of Visualize Boost boasts a user-friendly setup interface, streamlined machine discovery, and heightened stability.

### To install and set up Visualize Boost:

1. Use the SOLIDWORKS Installation Manager to install Boost on one or multiple machines that are accessible in the network.
2. On each Boost machine, do the following:
  - a. Start **SOLIDWORKS Visualize Boost 2025**.

- b. Go to **File > Settings**.
  - c. For one machine, select **Coordinator** to make it the coordinator node. Leave the Coordinator check box clear for all other Boost nodes.
  - d. For non-coordinator Boost nodes, enter the **Coordinator IP Address**.
  - e. Click **Apply**.
3. In SOLIDWORKS Visualize, go to **Tools > Options > Boost**, and enter the **Coordinator IP Address**.
4. Click **Connect**.

Once connected, you can choose the Boost Renderer in the Render Wizard/Quality page to start a render that is distributed over the network.

# 19

## SOLIDWORKS CAM

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This chapter includes the following topics:

- **Contour Mill Toolpaths That Machine from Bottom to Top**
- **Automatic Feature Recognition of Turn Features**
- **Dockable Legends for Toolpath Simulations**

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, SOLIDWORKS Premium, and SOLIDWORKS Ultimate.

### Contour Mill Toolpaths That Machine from Bottom to Top

You can specify an option to generate Contour Mill toolpaths that machine from bottom to top of 2.5 Axis Mill features.

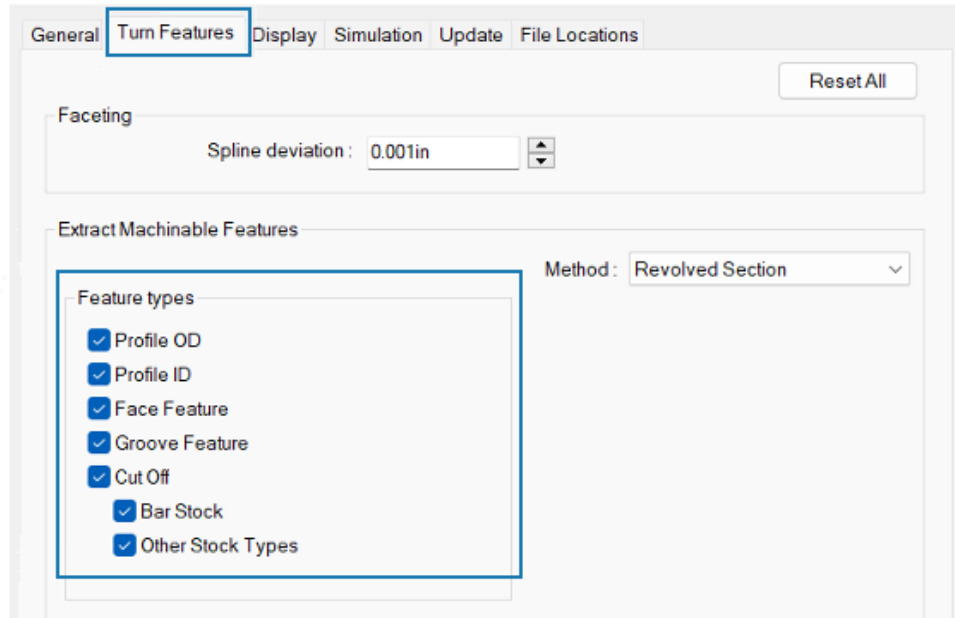
This option helps when machining:

- Tapered features
- Keyway slot features (Recommended tools for such features include the lollipop and keyway tools.)

#### **To specify this option:**

1. In the Operation Parameters dialog box, on the Contour tab, under **Depth processing**, select **Bottom to top**.

## Automatic Feature Recognition of Turn Features



Options are available for recognizing Turn features using Automatic Feature Recognition (AFR).

In previous releases, when you used AFR with the **Extract Machinable Features** (EMF) tool, SOLIDWORKS CAM recognized all Turn features in the model. You could not control which feature types to recognize.

### To specify these options:

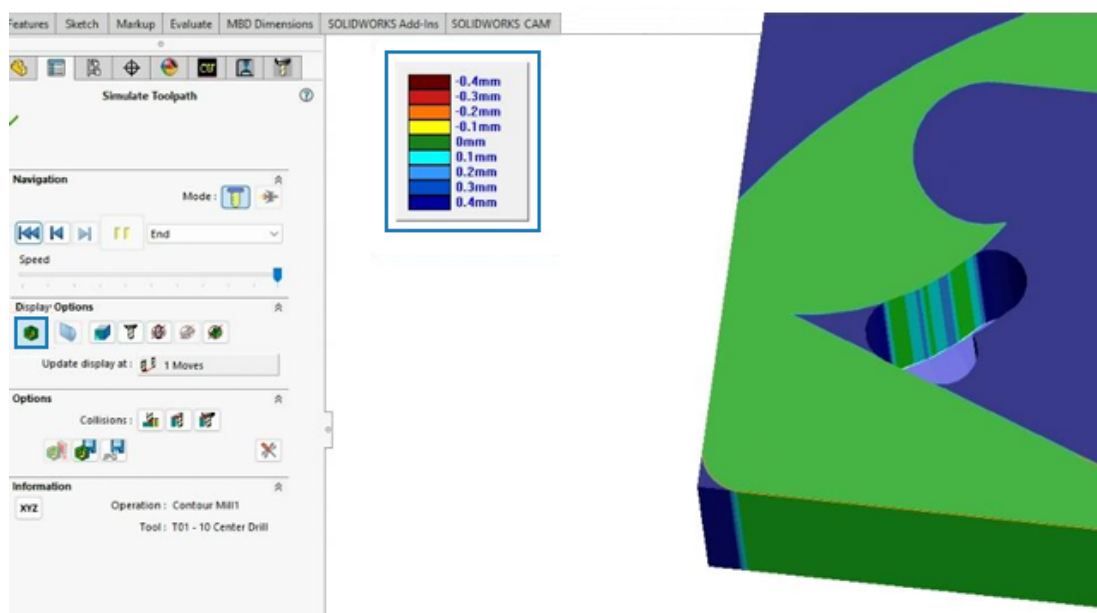
1. Click **Tools > SOLIDWORKS CAM > Options**.
2. In the dialog box, on the Turn Features tab, under **Extract Machinable Features**, specify **Feature types** options.

Option	Description
<b>Profile OD</b>	Recognizes profile ODs in the active part through the <b>Extract Machinable Features</b> tool.
<b>Profile ID</b>	Recognizes profile IDs in the active part through the <b>Extract Machinable Features</b> tool.




Option	Description
<b>Face Feature</b>	<p>Recognizes face features depending on the stock type:</p> <ul style="list-style-type: none"> <li>Round bar stock. Recognizes a single face feature at the start of the part model.</li> <li>Any stock type other than round bar stock. Recognizes: <ul style="list-style-type: none"> <li>Face features at the start of the part model. (These features appear under the same <b>Turn Setup</b> as other recognized Turn features.)</li> <li>Face features at the end of the part model. (These features appear under the reversed <b>Turn Setup</b>.)</li> </ul> </li> </ul> <p>When cleared, the software does not create a face feature under the <b>Turn Setup</b>. You can add face features using Interactive Feature Recognition.</p>
<b>Groove Feature</b>	Recognizes groove features in the active part through the <b>Extract Machinable Features</b> tool.
<b>Cut Off</b>	<p>Recognizes the specified type of cut off features:</p> <ul style="list-style-type: none"> <li><b>Bar Stock.</b> If the stock type is a bar stock, recognizes Cut Off features under the same <b>Turn Setup</b> as the other recognized features.</li> <li><b>Other Stock Types.</b> If the stock type is anything except a round bar, recognizes Cut Off features under the same <b>Turn Setup</b> as the other recognized features.</li> </ul>

## Dockable Legends for Toolpath Simulations



During toolpath simulations, you can move the legend that shows the graphical comparison of the machined part and the design part.

In the Simulate Toolpath PropertyManager, under **Display Options**, click **Show Difference** . In the graphics area, you can move the legend.

# 20

## CircuitWorks

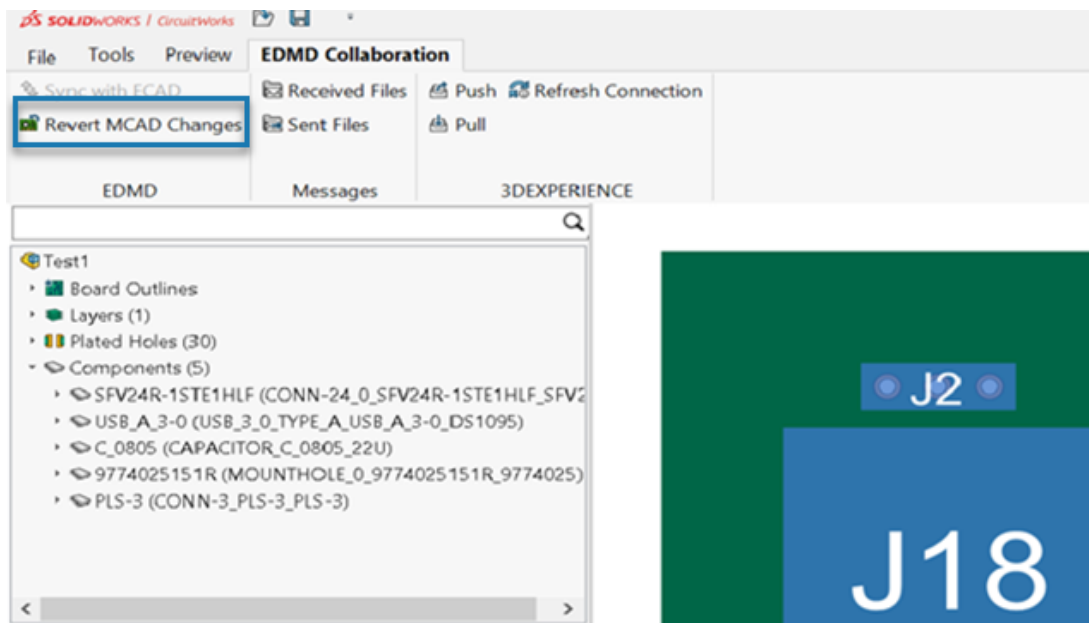
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This chapter includes the following topics:

- **Undo Latest MCAD Changes in CircuitWorks (2025 SP1)**
- **Restore Collaboration State after SOLIDWORKS Restarts or Crashes (2025 SP1)**

CircuitWorks™ is available in SOLIDWORKS® Standard, SOLIDWORKS Professional, SOLIDWORKS Premium, and SOLIDWORKS Ultimate.

### Undo Latest MCAD Changes in CircuitWorks (2025 SP1)



You can now undo the latest MCAD changes if the ECAD has not started working on them.

When MCAD proposes a change, you can reverse it in the **EDMD Collaboration** section if ECAD has not processed the task. This restores both SOLIDWORKS and CircuitWorks to the last synchronized state.

#### Benefits:

- Undo unnecessary or incorrect MCAD changes to keep CircuitWorks and SOLIDWORKS in sync.
- Easily revert to a previous state without affecting other ongoing tasks.

**To revert MCAD changes:**

1. In CircuitWorks, select the **EDMD collaboration** section.
2. Click **Revert MCAD Changes**.

**Revert MCAD Changes** is available only if MCAD made the last change.

## Restore Collaboration State after SOLIDWORKS Restarts or Crashes (2025 SP1)

CircuitWorks now includes a collaboration recovery feature that lets you resume your ECAD and MCAD collaboration smoothly in the event of a SOLIDWORKS restart or crash.

After a restart or crash, open the first backup file in the EDMD Collaboration folder (identify the baseline `.idx` file by its timestamp) to resume collaboration. This preserves your workflow and minimizes disruption.

# 21

## SOLIDWORKS Composer

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This chapter includes the following topics:

- **Composer Plug-In for Adobe Acrobat**
- **Prevent Outline Generation for Hidden Geometry**

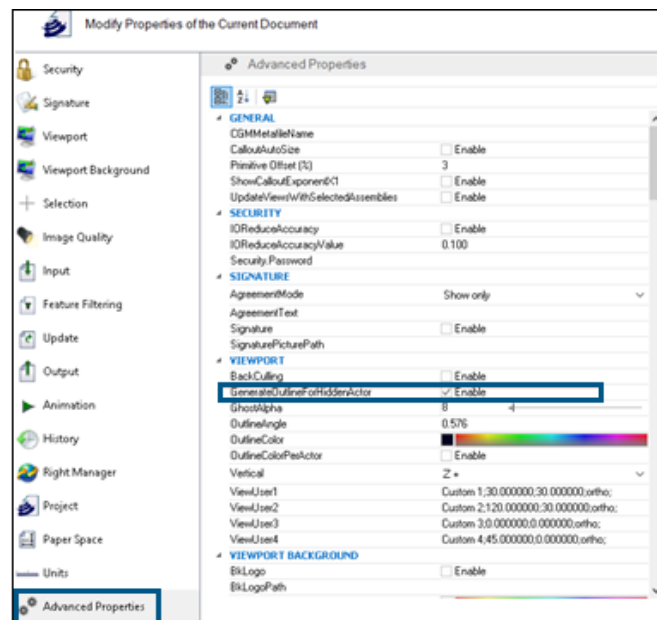
SOLIDWORKS® Composer™ software streamlines the creation of 2D and 3D graphical contents for product communication and technical illustrations.

### Composer Plug-In for Adobe Acrobat

The Composer plug-in for Adobe® Acrobat® is no longer supported by 64-bit Adobe configurations.

It is still supported by 32-bit Adobe configurations.

### Prevent Outline Generation for Hidden Geometry



The **GenerateOutlineForHiddenActor** property available in the **Viewport** category of the Advanced Properties page specifies whether or not hidden actors are outlined in render mode.

To prevent outlines from being generated by hidden actors, clear this option. This saves time when using render mode for large assemblies.

# 22

## SOLIDWORKS Electrical

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This chapter includes the following topics:

- **Exporting Manufacturer Parts and Cable References (2025 FD03)**
- **Temporary Offline Mode for Electrical Schematic Designer (2025 FD03)**
- **Allowing Non-Repeated Column Values for Circuits, Terminals, and Cable Cores (2025 SP2)**
- **Export PDF Files (2025 SP2)**
- **Filter Options for Configuration Dialog Boxes (2025 SP2)**
- **3D Tab (2025 SP1)**
- **Accessory Association for Complex Components and Electrical Assemblies**
- **Cable Management**
- **Distribute Terminals**
- **New Variables in Formula Management**
- **Update Data and Replace Data in SOLIDWORKS Electrical 3D**
- **Wire Termination Types**

SOLIDWORKS® Electrical is a separately purchased product.

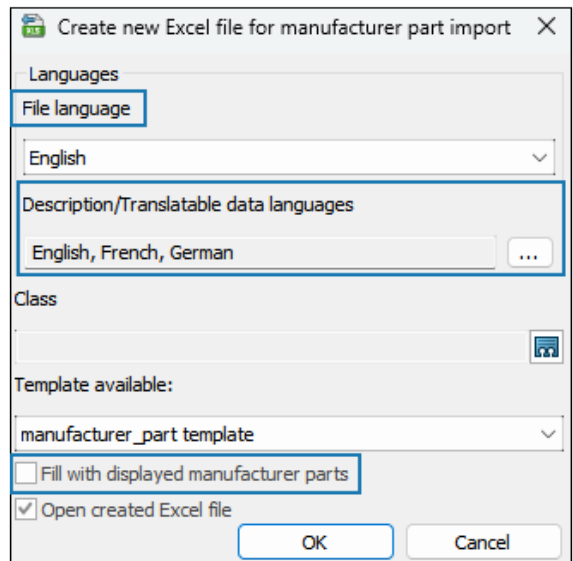
### Exporting Manufacturer Parts and Cable References (2025 FD03)

You can export data from the displayed manufacturer parts or cable references into a generated Excel file. You can also import new data and overwrite existing data in the library.


**Benefits:** This reduces manual entry and improves efficiency in data management and export.

The updates below support manufacturer parts and cable references. They do not support electrical assemblies.

Creating a New Excel File for Import (2025 FD03)

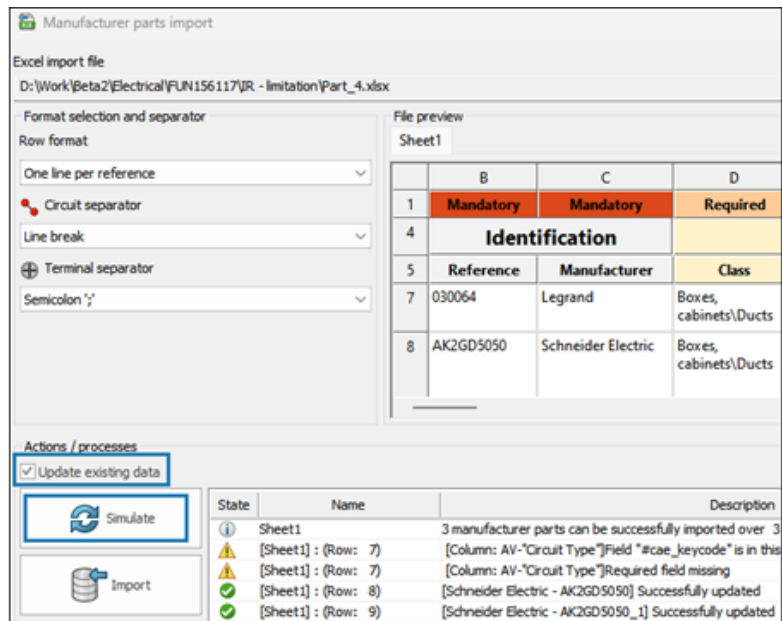


The Create new excel file for manufacturer part import dialog box has updated options. The updated options are as follows:

Option	Description
<b>File languages</b>	Lets you specify the language for exporting an Excel file.
<b>Description/Translatable data languages</b>	Lets you add a description in multiple languages. More options  lets you open the Language selector dialog box to select multiple languages.
<b>Fill with displayed manufacturer parts</b>	Lets you fill the Excel file with the manufacturer parts displayed in the Manufacturer parts management dialog box. <div><b>Class</b> is disabled when this option is selected, as a file can contain multiple classes.</div>



## Updating Existing Data While Importing (2025 FD03)



You can overwrite the existing data in the library with content from an Excel file while importing new data.

In the **Action/Processes** section of the Manufacturer parts import dialog box, a new option, **Update Existing Data** is available.

Previously, you could only import new data and the existing data remained unchanged.

**Benefits:** You can quickly complete or update existing information.

To update the existing data while importing, do the following:

1. Click to browse for the Excel file that contains the data.
2. Specify the appropriate **Format selection and separator** options.
3. Select **Update existing data**.
4. Click **Simulate** to run a preview of the import and update process without applying changes.

The **Compare** command is renamed to **Simulate** to better reflect its functionality.

5. Click **Import** .

During the import process:

- Existing data in the library updates with the new data from the Excel file.
- Empty or deleted columns in the Excel file do not change existing data in the library.

To explicitly remove data, you must clear values in the Excel file before importing.

# Temporary Offline Mode for Electrical Schematic Designer (2025 FD03)

Electrical Schematic Designer supports temporary offline mode. If you lose connection during a session, you can continue working offline with local files. The app attempts to reconnect and prompts you to restart when the connection returns.

See **Work Offline When a Connection is Unavailable.**

## Allowing Non-Repeated Column Values for Circuits, Terminals, and Cable Cores (2025 SP2)

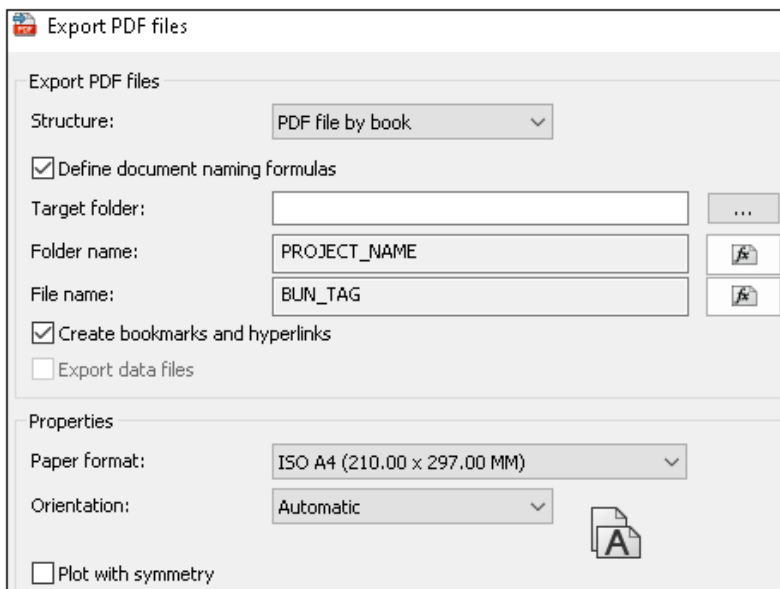
[illegible]

You can import data more efficiently by applying shared values to multiple circuits, terminals, or cable cores. While importing, if the values entered lack separators, the single value applies to all the circuits, terminals, or cable cores.

**Benefits:** This saves time and reduces manual input errors.

While importing the template in Manufacturer Part Management, if you enter a single value for a terminal column, it applies to all terminals of the circuit. For example, if for all the terminals, **Max Wire Section** is **6**, you can enter only the value **6** without repeating the information. Previously, you had to enter the value **6;6|6;6**. This is applicable when you select **One line per reference** for manufacturer parts and cable references and **One line per circuit** for manufacturer parts.

## Export PDF Files (2025 SP2)

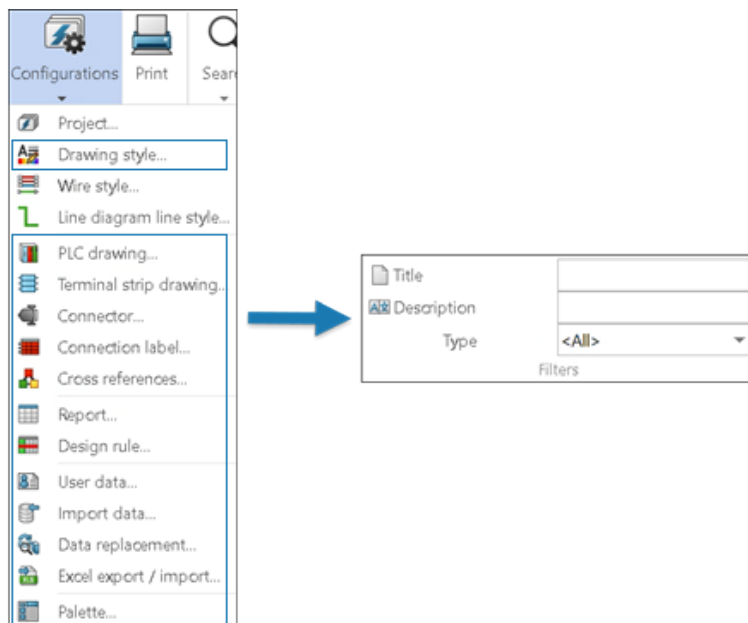


You can export a PDF by project, book, or page, and automate the orientation and size of each page of the PDF file based on the drawing format. In the Export PDF files dialog box, you can also define the formula for naming the documents.

Under **Properties**, for **Paper format**, select **Match drawing size** to automatically scale the paper format to match your drawing dimensions. The options in the Print drawings dialog box are reorganized to match the changes in the Export PDF files dialog box.

**Benefit:** This improves organization, and makes the process more efficient and intuitive. This enhances the user experience with a clearer dialog box structure.

## Filter Options for Configuration Dialog Boxes (2025 SP2)



You can use the filter options to filter and refresh the configuration list across multiple configuration files.

**Benefits:** This reduces the time required to search a specific configuration.

The configuration dialog boxes include a new group of options under **Filters**.

You can filter the configuration files by entering relevant text and selecting the configuration type in the following fields:

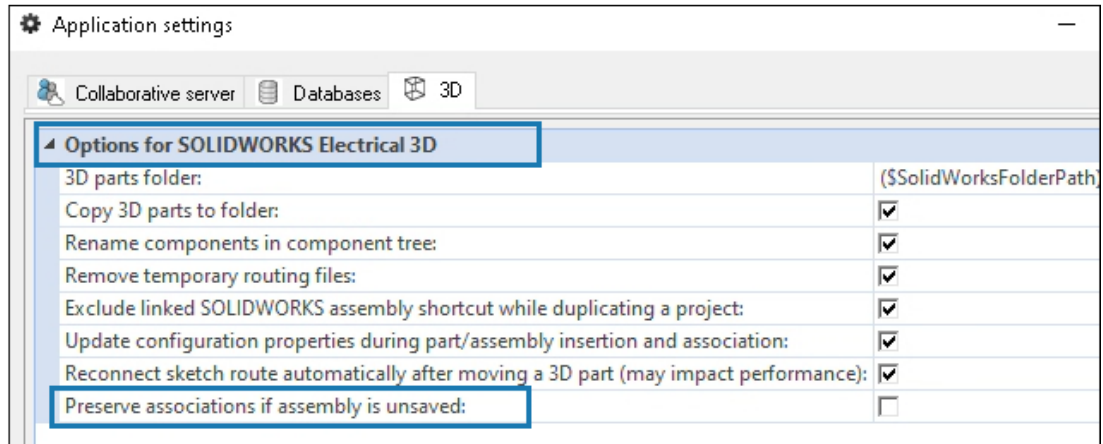
- **Title**
- **Description**
- **Type**

The availability of filters depends on the content of the configuration dialog box.

You can also use the **Title**, **Description**, and **Type** options together to filter configuration files.

The filtering options apply to both **Application configurations** and **Project configurations**.

## 3D Tab (2025 SP1)



The user interface of the **3D** tab in the **Application Settings** dialog box has been updated.

### User Interface Update

A dynamic property list replaces static check boxes.

The title **Options for SOLIDWORKS Electrical 3D** is added to improve the organization of the options.

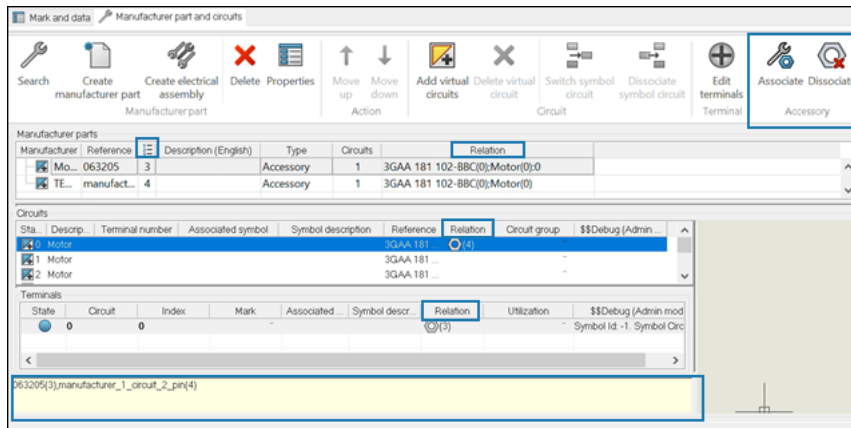
### Preserve Association Option

The **Preserve associations if assembly is unsaved** option lets you preserve associations between 3D components and electrical parts, even if you do not save the SOLIDWORKS assembly.

**Benefits:** This option enhances workflow flexibility and prevents data loss.



To access this option, click **Tools > SOLIDWORKS Electrical > Tools > Application Settings > 3D**.

## Accessory Association for Complex Components and Electrical Assemblies




You can simplify the assembly process by linking accessory parts to specific circuits or terminals on a component. This is particularly useful for the manufacturing of custom or complex connectors.

**Benefits:** You can ensure that only valid selections are associated, and update associations without removing existing ones, also streamlines the configuration of accessories.

The **Component properties** and **Electrical assembly properties** dialog boxes now contain **Associate**  and **Dissociate**  commands for accessories. You can access these commands through the shortcut menu, and you can also select the accessory and drop it onto the parts you want to associate it with.

- **Associate accessories:** Enables users to associate one or more accessories with a specific circuit or terminal.
- **Dissociate accessories:** Removes the association between accessories and the selected base part or terminal.

### User Interface Updates to View the Association

- The Component Properties and Electrical assembly properties dialog boxes now contain new columns, as follows:
  - **Order number** : Displays the order number for parts of the same category, to distinguish between multiple instances of the same part in the assembly.
  - **Relation:** Displays the relationships between base parts, component parts, circuits, and terminals along with their associated accessories.

You can also view the relationship in the text field at the bottom of the Component Properties dialog box.









## Associating and Dissociating Accessories with Electrical Assemblies

You can manage accessory associations and dissociations within complex electrical assemblies, save accessory relationships in the database, and apply them to components.

A complex assembly consists of multiple interconnected electrical components, subassemblies, wiring, circuits, and terminals all working together to perform a specific function.

You can attach and remove accessory parts to specific circuits or terminals on a complex assembly, which helps to simplify the assembly process. These links are saved in the library.

### To associate the accessory with assembly:

1. Click **Library > Manufacturer part management**.
2. In the Manufacturer part management dialog box, under **Classification**, select a valid class.
3. Do one of the following:
  - Click **Add manufacturer part > Add electrical assembly** .
  - Click **Multiple insertion > Add electrical assemblies** .
4. In the Electrical assembly properties  dialog box, click **Manufacturer parts** .
5. From a manufacturer part list, circuit list, or terminal list, select any component and accessory that you need to associate.
6. Do one of the following:
  - To associate:
    - Click **Associate**  or right-click and select **Associate** .
    - Drag the selected accessory to the part you want to associate.
  - To remove an association, do one of the following:
    - Select the associated part, click **Dissociate** .
    - Right-click the associated part and select **Dissociate** .

The app checks if the selection is valid. For example, if the selected parts do not include accessories, the app displays a warning message and cancels the command.

You can see the associated relation between base manufacturer part, circuit, and terminal with their associated accessory in the **Relation** column. If there is no association, the column remains empty.







You can also see the relationship in the text field at the bottom of the dialog box.

7. Click **OK**.

## Associating and Dissociating Accessories with Components

You can add accessories to a base part, circuit, or terminal while working with components. The new commands and dialog improvements allow better association, dissociation, and visualization of accessory relationships, providing more detail for the manufacturing process.

### To associate the accessory with a component:

1. Right-click a component of an electrical project, and select **Component** .
2. In the Component properties dialog box, click **Manufacturer part and circuits** .
3. From a manufacturer part list, circuit list, or terminal list, select any component and accessory that you need to associate.
4. Do one of the following:
  - To associate:
    - Click **Associate** , or right-click and select **Associate** .
    - Drag the selected accessory on the component that you want to associate.
  - To remove an association, do one of the following:
    - Select the associated part, click **Dissociate** .
    - Right-click the associated part and select **Dissociate** .

The app checks if the selection is valid. For example, if the selected components do not include accessories, the app raises a warning message and cancels the command.

You can see the associated relation between component parts, circuits, and terminals with their associated accessory in the **Relation** column. If there is no association, the column remains empty.

You can also see the relationship in the text field at the bottom of the dialog box.

5. Click **OK**.



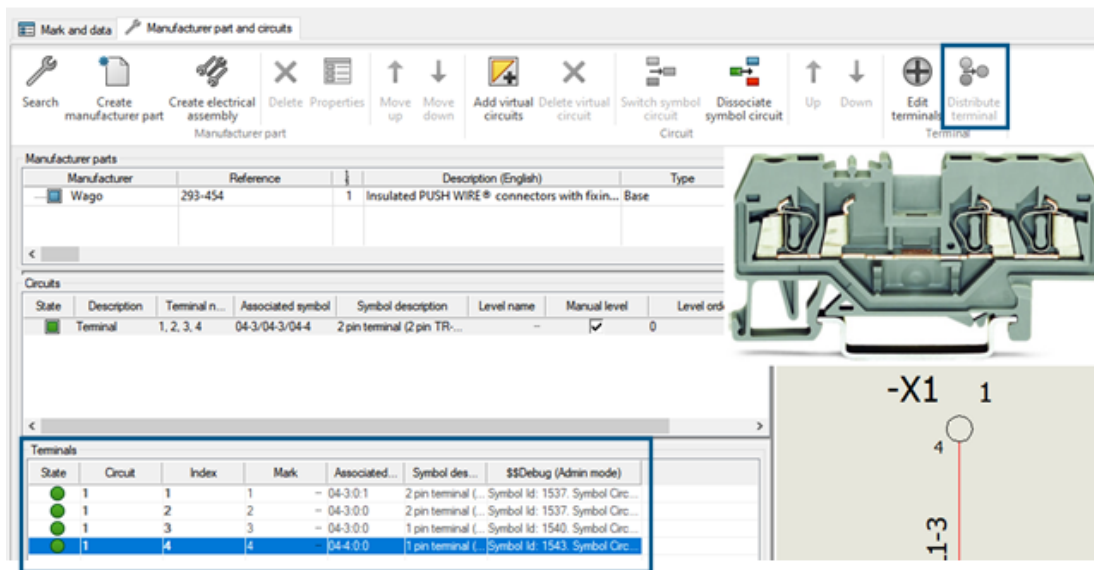
## Cable Management

Cable reference properties	
<span>Properties</span> <span>User data</span> <span>Cable cores</span>	
<b>General</b>	
Reference:	Alsecure PI
Manufacturer:	Nexans
Class	**** Unclas
Article number:	
External ID:	
Library:	MM2_INDU
Family:	SmXGB-F2
Standard:	0,6/1kV NB
Series:	
Mark root:	
Description (English):	
<b>Supplier</b>	
Supplier name:	
Stock number:	

Cable management and configuration is enhanced for a better user experience.

- In the Cable reference properties dialog box, you can specify the **Mark root** option in the Properties tab. When you add a cable to a project from the cable reference, it automatically copies the **Mark root** from the cable reference to the cable's mark root. This value is also accessible for filters.
- There are new variables for efficient cable organization:
  - **Position**
  - **Component Origin/Destination**

## Distribute Terminals



The **Distribute terminal** tool lets you link symbols to specific circuits and pins, simplifying the depiction of complex terminal arrangements in electrical schematics. It offers an intuitive interface for dynamic circuit and pin selection, ensures precise symbol-to-terminal mapping, and improves design accuracy.

You can select a specific terminal when adding a new terminal strip, in addition to selecting the circuit. It distributes a single circuit over multiple schematic symbols.

This functionality is available for terminal components only.

The **Distribute terminal** tool also lets you change the mapping between symbol connection points and component circuit terminals. This command is enabled when two terminals are selected. You can switch component connections between different circuits.

The Component Properties dialog box contains a Terminal section that has a list of terminals with columns for **Circuit**, **Index**, **Mark**, and **Relation**.

### Distributing Terminal Components


You can use the **Distribute terminal** tool to manage and switch component connections.

#### To distribute a terminal:

1. Click **Insert Terminal** .

In the Terminal Mark tab, a node for terminals appears in the right pane.

- The software groups terminals of the same circuit together and displays available circuits for multilevel terminal components.
- Partially used circuits appear as with a half-color/half-grey icon in the Component tree, showing only free terminals.

2. Select a component to associate with the circuit terminal.
3. In the Manufacturer part and circuits tab, click **Distribute terminal**  to manage and switch component connections.

## New Variables in Formula Management

Formula management: Origin - destination mark	
<div>  Predefined formulas            Recent formulas            Variables and simple formulas            Functions         </div>	
Simple formula	Description
BOOK_TAG	Book mark, empty when same book.
BOOK_TAG_ALWAYS	Book mark, always visible.
STRZ(VAL(BOOK_ORDERNO), 2, 0)	Book order number on 2 characters, empty when same book.
STRZ(VAL(BOOK_ORDERNO_ALWAYS), 2, 0)	Book order number on 2 characters, always visible.
LOCATION_TAG	Location mark
FOLDER_TAG	Folder mark
FOLDER_ORDERNO	Order number
STRZ(VAL(FOLDER_ORDERNO), 2, 0)	Folder order number on 2 characters.
STRZ(VAL(FOLDER_ORDERNO), 3, 0)	Folder order number on 3 characters.
FILE_TAG	File mark

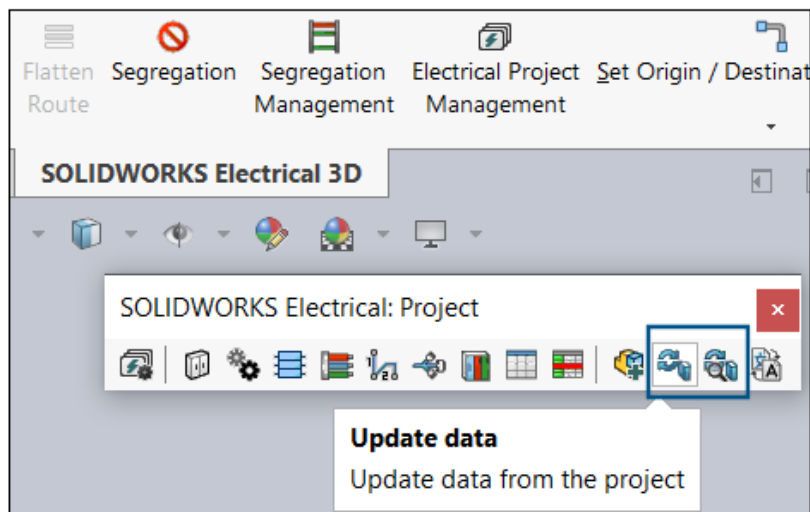
New variables are available in the Formula Management dialog box that let you label origin and destination arrows more effectively. This makes it easier to find and understand bookmarks, especially when the arrows are in the same book.

In the Formula management: Origin - destination mark dialog box, on the Variables and simple formulas tab:

- **BOOK\_TAG\_ALWAYS** variable appears under **BOOK\_TAG**.
- **STRZ(VAL(BOOK\_ORDERNO\_ALWAYS), 2, 0)** appears under **STRZ(VAL(BOOK\_ORDERNO), 2, 0)**.

In the Attribute management dialog box, **#BUN\_TAG\_ALWAYS** appears under **#BUN\_TAG**.

## Update Data and Replace Data in SOLIDWORKS Electrical 3D



**Update data** and **Replace data** tools are available in the SOLIDWORKS Electrical 3D Project toolbar.

You can also access these tools from **Tools > SOLIDWORKS Electrical > Process**.

In earlier releases, these tools were only available in SOLIDWORKS Electrical Schematic. With these tools in SOLIDWORKS Electrical 3D, you can update project data such as manufacturer part properties, cable references, symbols, and title blocks. You need not switch back to the SOLIDWORKS Electrical Schematic application each time to update or refresh the changes.

## Wire Termination Types

You can add user data and customize details about wire termination types in your electrical designs.

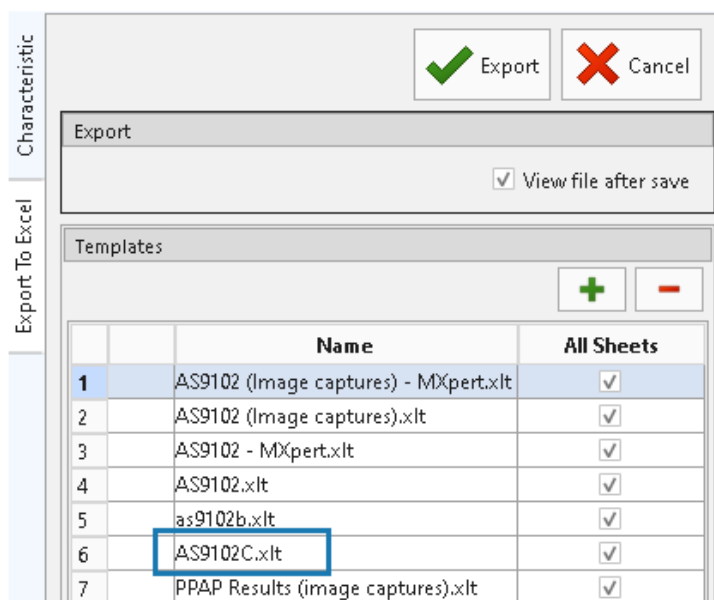
- **User data** and **Translatable data** are added in the Wire termination type properties dialog box.
- New attributes are available for user data and termination types.

# 23

## SOLIDWORKS Inspection

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

### Exporting FAI Reports to AS9102 Revision C Template (2025 SP2)



You can export data of your inspection project to the standard AS9102 revision C report format.

This feature is available in both SOLIDWORKS Inspection Standalone and SOLIDWORKS Inspection Add-in.

## SOLIDWORKS MBD

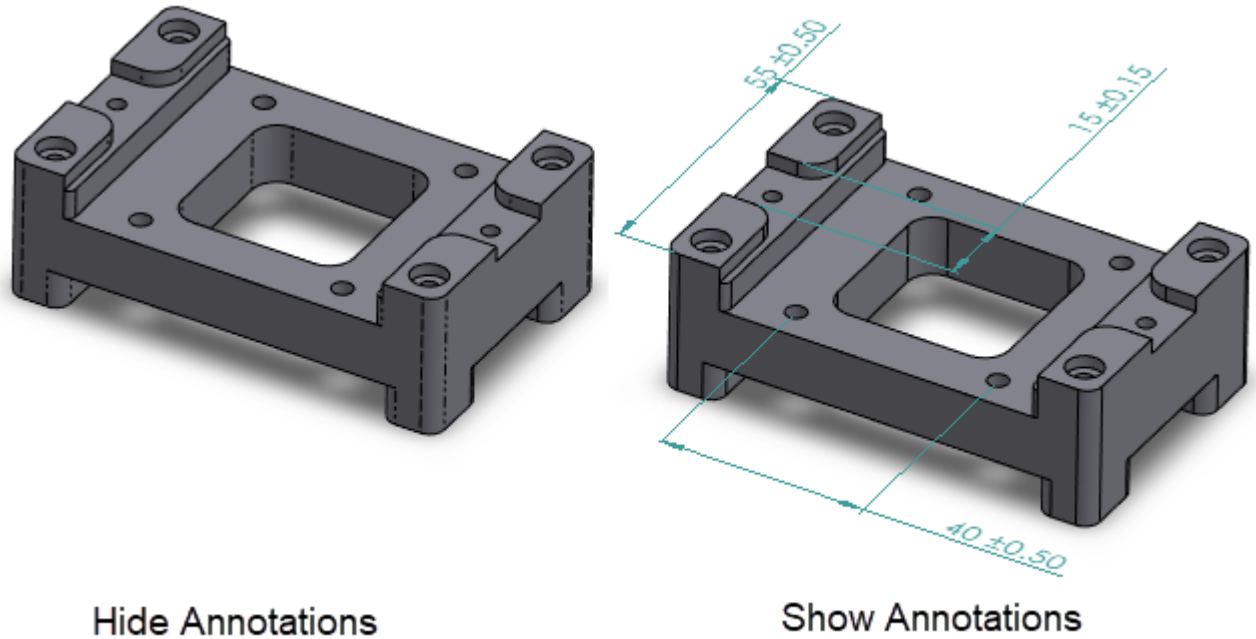
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This chapter includes the following topics:

- **Hiding and Showing Annotations in Parts and Assemblies (2025 FD03)**
- **Specifying STEP 242 Editions (2025 SP2)**
- **Aligning DimXpert Dimensions (2025 SP2)**
- **Creating DimXpert Dimensions from Feature and Reference Dimensions (2025 SP2)**
- **Saving DimXpert Dimensions to Library Features (2025 SP1)**
- **Creating DimXpert Dimensions from Sketch Dimensions**
- **Using the SOLIDWORKS MBD Add-In with SolidNetWork License**
- **Delete General Profile Tolerance**
- **Creating Length Dimensions in Drafted Features**
- **Creating Two Separate Positional Tolerances for Slots**

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

## Hiding and Showing Annotations in Parts and Assemblies (2025 FD03)

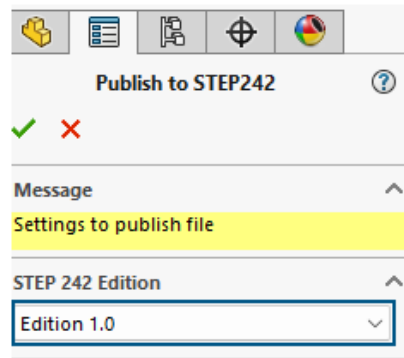


You can hide and show annotations in parts and assemblies and turn on and off the visibility from the same tool.

**To hide and show annotations in parts and assemblies:**

1. Click **View > Hide/Show > Annotations**.

## Specifying STEP 242 Editions (2025 SP2)



In the Publish to STEP242 PropertyManager, when you publish to STEP 242, you can specify Edition 1.0, 2.0, or 3.0.

### To specify STEP 242 editions:

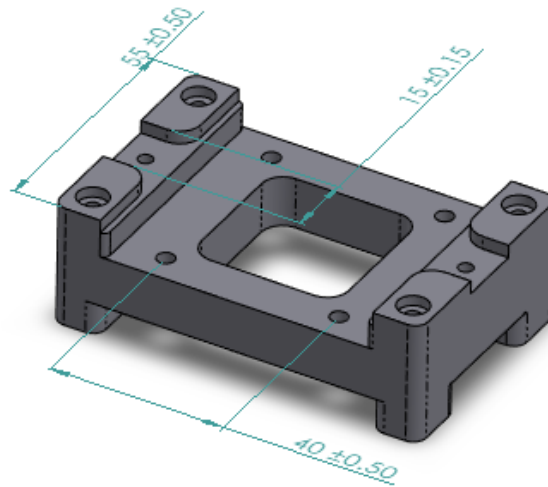
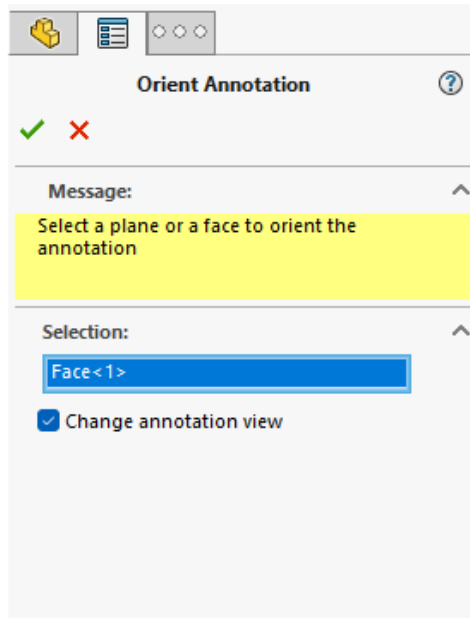
1. Click **Publish STEP 242 File**  (MBD toolbar).
2. In the PropertyManager, under **STEP 242 Edition**, click  and specify an option:
  - **Edition 1.0**
  - **Edition 2.0**
  - **Edition 3.0**

The default is Edition 1.0.

3. Click .



## Aligning DimXpert Dimensions (2025 SP2)



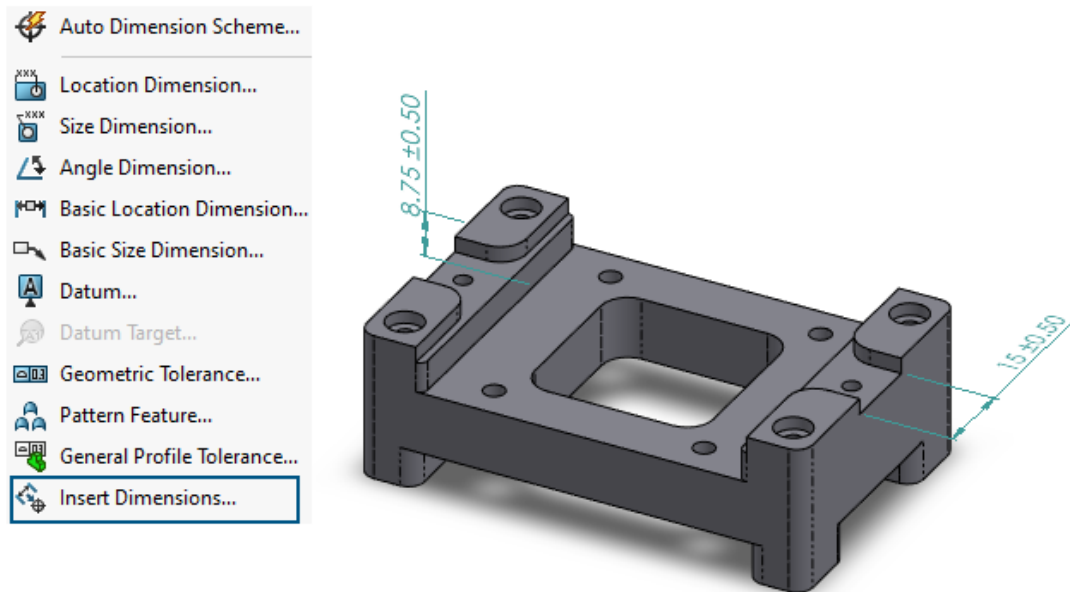
You can align DimXpert annotations to a user-defined plane.

DimXpert dimensions can become obscured when you apply them to contoured geometry. You can align the DimXpert annotations by moving them to a selected plane or planar face.

### To align DimXpert annotations to a user-defined plane:



1. Right-click a DimXpert annotation and click **Select Annotation View > By Selection**.
2. In the graphics area, select a plane or planar face to define the new orientation.
3. In the PropertyManager, select **Change annotation view** to move the annotation to the orientation view that corresponds to the new orientation.
4. Click .

## Creating DimXpert Dimensions from Feature and Reference Dimensions (2025 SP2)

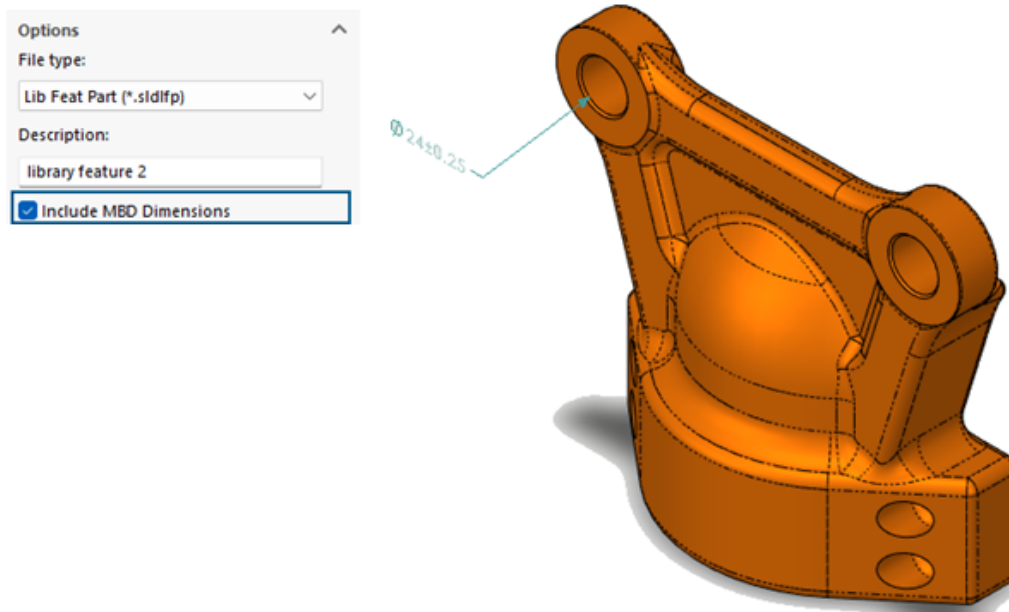


You can create DimXpert dimensions from feature and reference dimensions.

**To create DimXpert dimensions from feature and reference dimensions:**

1. Click **Insert Dimensions**  (MBD Dimension toolbar) or **Tools > MBD Dimension > Insert Dimensions**.
2. In the PropertyManager:
  - a. For **Features**, select features in the graphics area or the FeatureManager® design tree.
  - b. For **Feature Dimensions** or **Reference Dimensions**, select dimensions in the graphics area.
  - c. Click .



## Saving DimXpert Dimensions to Library Features (2025 SP1)



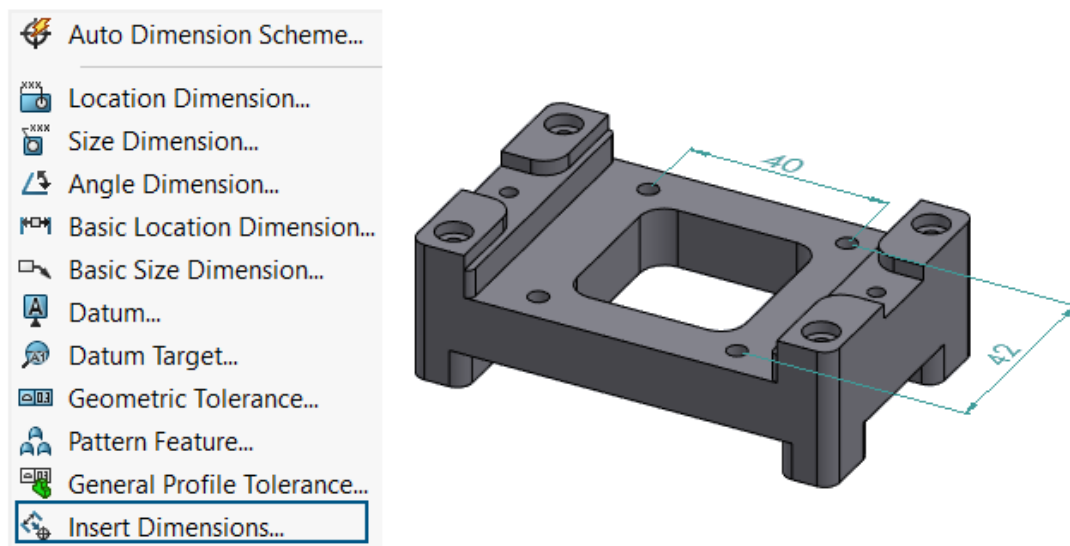
You can save DimXpert dimensions to library features.

**Benefits:** You can save the DimXpert dimensions to library feature parts to reuse them when you use a library feature on a model.

### To save DimXpert dimensions to library features:



1. Click **Add to Library**  on the Task Pane Design Library tab.
2. In the PropertyManager:
  - a. For **Items to Add**, select features from the graphics area or FeatureManager design tree.
  - b. For **File name**, type a file name (the default is document name.)
  - c. For the **Design Library folder**, select a subfolder to add the library feature.
  - d. For **Description**, type a description to be displayed in the item's tooltip.
  - e. Select **Include MBD Dimensions** and click .

## Creating DimXpert Dimensions from Sketch Dimensions

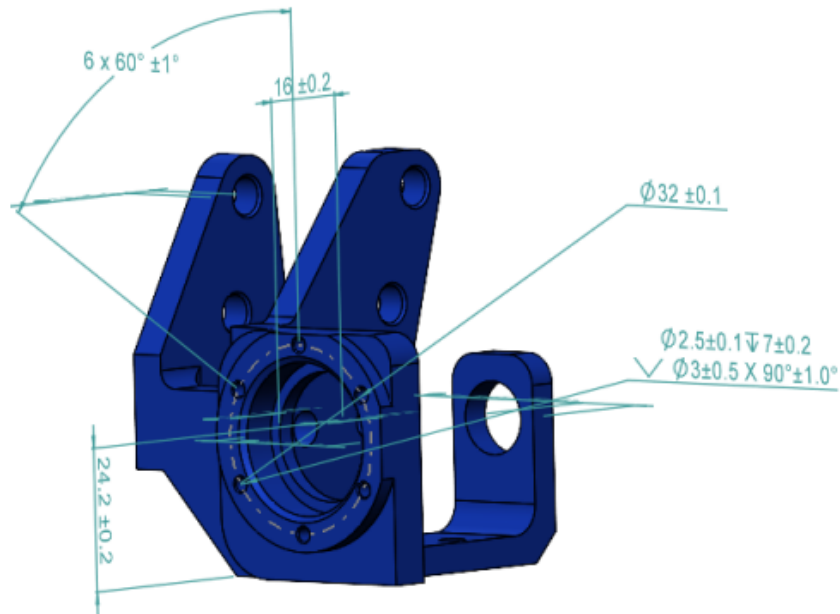


You can create DimXpert dimensions from sketch dimensions.

**To create DimXpert dimensions from sketch dimensions:**

1. Click **Insert Dimensions**  (MBD Dimension toolbar) or **Tools > MBD Dimension > Insert Dimensions**.
2. In the PropertyManager:
  - a. For **Features**, select features from the graphics area or FeatureManager® design tree.
  - b. For **Sketch Dimensions**, select the dimensions in the graphics area to create DimXpert dimensions.
  - c. Click .

## Using the SOLIDWORKS MBD Add-In with SolidNetWork License

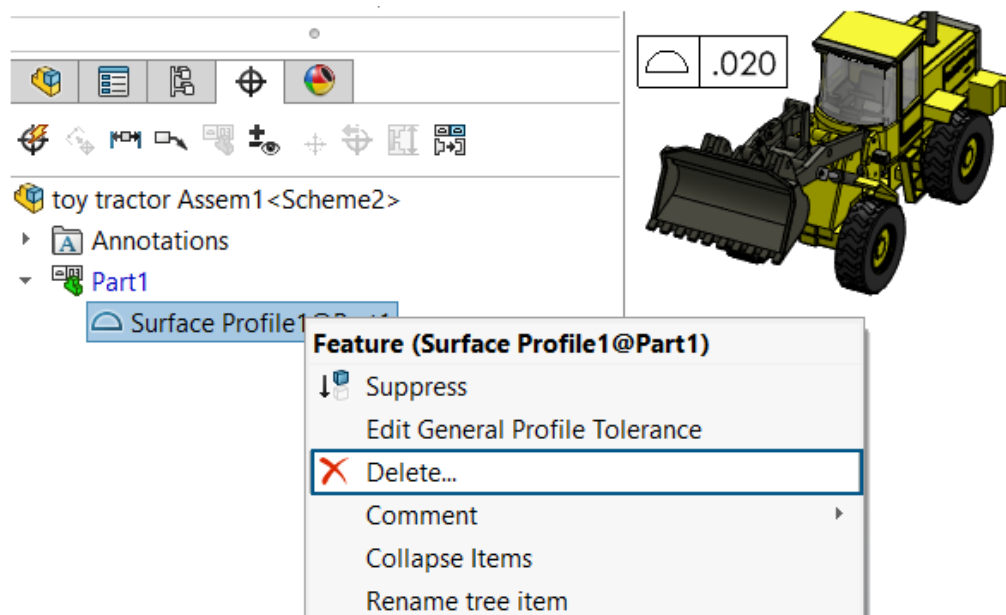


SolidNetWork License (SNL) customers can use the SOLIDWORKS MBD add-in.


### To use the SOLIDWORKS MBD add-in with SNL:

1. In SOLIDWORKS, click **Tools** > **Add-Ins**.
2. In the dialog box, under **SOLIDWORKS Add-ins**, select **SOLIDWORKS MBD** and click **OK**.

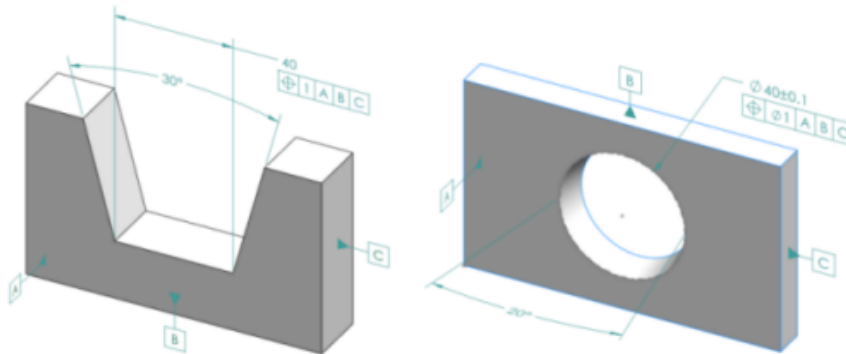
## Delete General Profile Tolerance




In Design with SOLIDWORKS, you can delete a general profile tolerance.

To delete a general profile tolerance, in the DimXpertManager , right-click a general profile tolerance and click **Delete**.

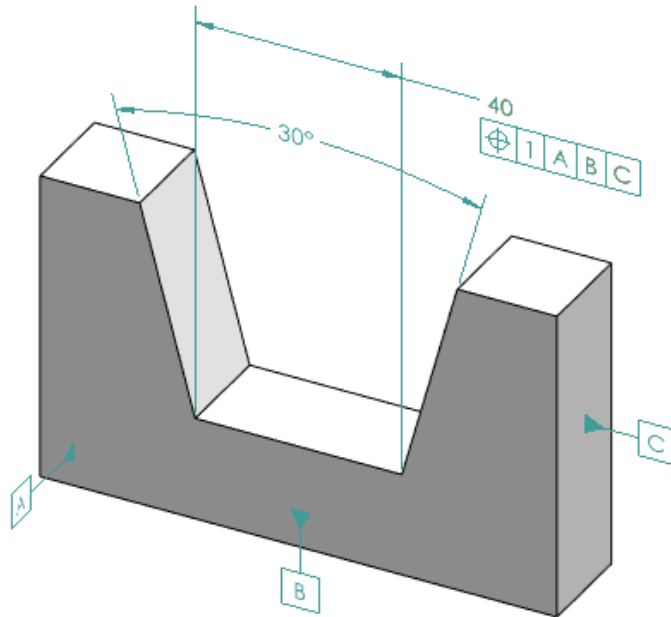
## Creating Length Dimensions in Drafted Features







You can create length dimensions in drafted features.

You can use the DimXpert **Size Dimension**  tool to create dimensions for drafted features, such as wedges and cones. The dimension is typically a distance dimension with tolerances. The dimension can be between two edges of draft ends or circular edges of a cylinder.

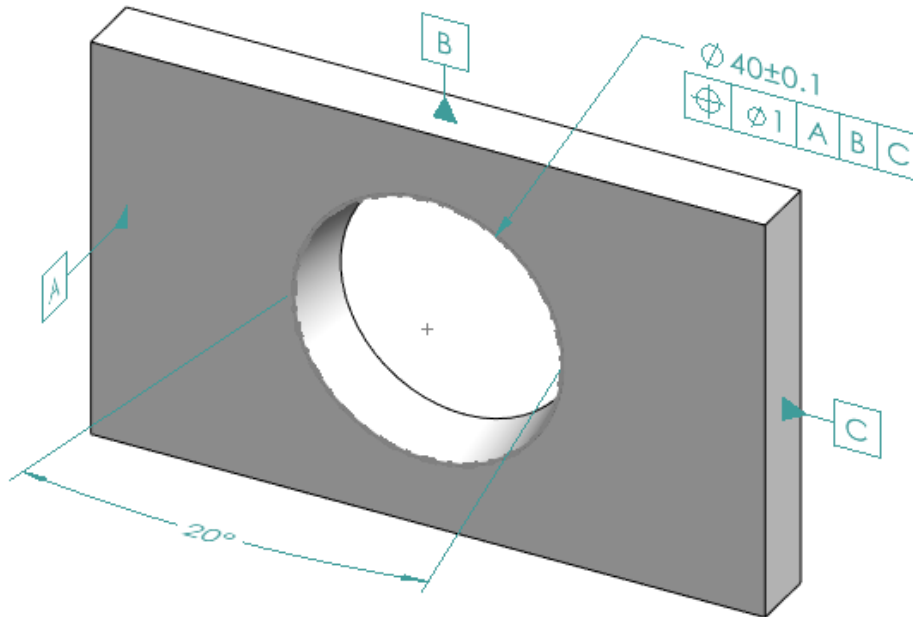
## Creating Length Dimensions in Wedges




### To create length dimensions in wedges:

1. Click **Size Dimension**  (MBD Dimension toolbar) or **Tools > MBD Dimension > Size Dimension**.
2. Click the face of one of the side planes.
3. In the feature selector, click **Create Width/Wedge Feature** .
4. Click the face of the second side.
5. Click the face of the end plane, which is the plane that intersects the two sides, and click .
6. Place the angle dimension.
7. Click **Size Dimension**  (MBD Dimension toolbar) or **Tools > MBD Dimension > Size Dimension**.
8. Click the end plane.
9. Click in the graphics area to place the width dimension.
10. Apply a geometric tolerance to the width dimension to create the position callout.

## Creating Length Dimensions in Cones

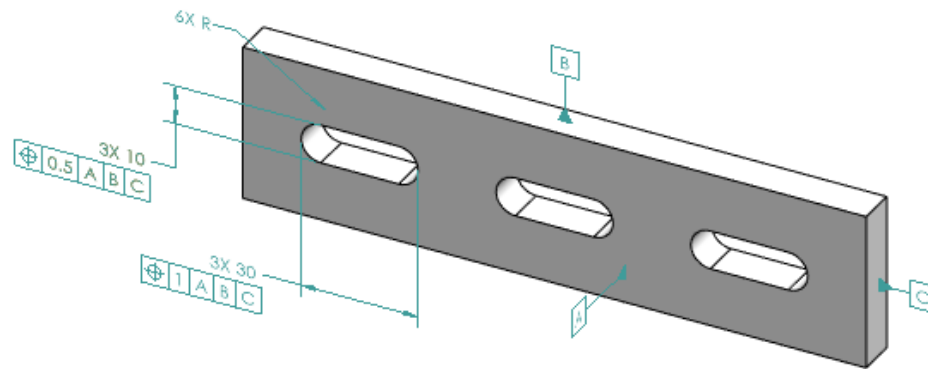


### To create length dimensions in cones:

1. Click **Size Dimension**  (MBD Dimension toolbar) or **Tools > MBD Dimension > Size Dimension**.
2. Click the conical face to create the cone feature.
3. Place the angle dimension.
4. Click the top edge to create the intersect circle feature and diameter dimension.  
*See SOLIDWORKS Help: DimXpert Features.*
5. Apply a geometric tolerance to the intersect circle feature to create the position callout.








## Creating Two Separate Positional Tolerances for Slots



You can create two separate positional tolerances for slots.

### To create two separate positional tolerances for slots:

1. Click **Size Dimension**  (MBD Dimension toolbar) or **Tools > MBD Dimension > Size Dimension**.
2. Click the edge of the length of a slot and click in the graphics area to place the dimension.
3. Click .
4. Apply a geometric tolerance and a position tolerance and click in the graphics area to place the tolerance.
5. Click .
6. Click **Size Dimension**  (MBD Dimension toolbar) or **Tools > MBD Dimension > Size Dimension**.
7. Click the edge of the width of a slot and click in the graphics area to place the dimension.
8. Click .
9. To choose the type of dimension to apply to a feature, such as the "6XR" in the illustration, see *SOLIDWORKS Help: Using the Dimension PropertyManager*.

## DraftSight

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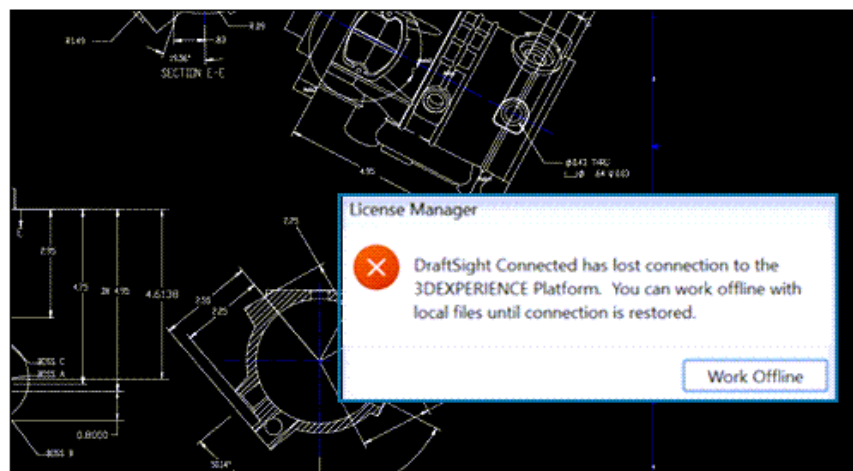
This chapter includes the following topics:

- **Temporary Offline Mode Support for DraftSight Connected (2025 FD03)**
- **Batch Print for 3DEXPERIENCE Drawings (DraftSight Connected Only) (2025 FD03)**
- **Data Grid View in MySession (2025 FD03)**
- **Welding Symbols (2025 SP3)**
- **Adding Fit to Dimension (2025 SP3)**
- **Adding Tolerance to the Dimension (2025 SP3)**
- **Weld Representation (2025 SP3)**
- **Construction Lines (2025 SP3)**
- **Importing a PDF File as a Block from the 3DEXPERIENCE Platform (DraftSight Connected Only) (2025 FD02)**
- **Sheet Set Manager on the 3DEXPERIENCE Platform (DraftSight Connected Only) (2025 FD02)**
- **Design Resources Palette Compatibility with the 3DEXPERIENCE Platform (2025 FD01)**
- **Attaching Files from the 3DEXPERIENCE Platform (DraftSight Connected Only) (2025 FD01)**
- **Bookmarks for Batch Save to 3DEXPERIENCE (DraftSight Connected Only)**
- **Open Dialog Box (DraftSight Connected Only)**
- **Managed DS License Server**
- **DGN File Export**
- **Auto-Fill Table Cells**
- **Accessing Tables and Creating Table Breaks**
- **Libraries of Dynamic Blocks**
- **Dynamic Search in an Options Dialog Box**
- **Dimension Styles Dialog Box**
- **Block Structure Palette**
- **Editing Clipped External References and Blocks**
- **Drawing Order**
- **Managing Spacing Between Dimensions**
- **Menu Bar Visibility**
- **Dimensional Constraints for Custom Blocks**
- **FLATTEN Command**
- **Visual Styles**
- **Printing in MacOS**
- **AMUSERHATCH Command (DraftSight Mechanical Only)**

- **Table Edits**
- **Import STEP Files**
- **DWGUNITS Command**
- **PDF Export and Batch Print Usability**
- **Blocks in the Design Resource Palette**
- **Multiple Visibility Elements**
- **Lasso Selection**

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. **3DEXPERIENCE®** DraftSight is a combined solution of DraftSight with the power of the **3DEXPERIENCE** platform.

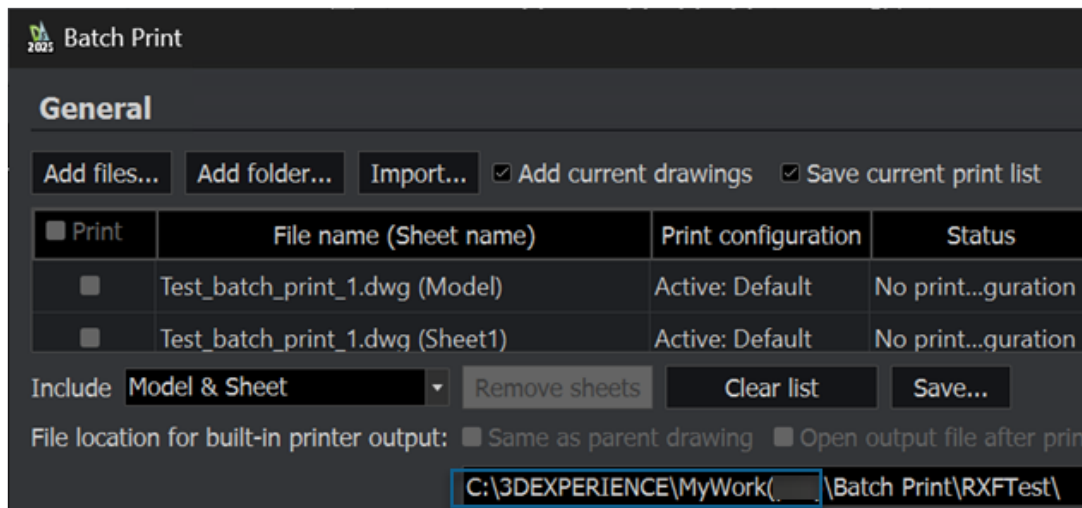
## Temporary Offline Mode Support for DraftSight Connected (2025 FD03)



DraftSight Connected supports temporary offline mode. If you lose connection during a session, you can continue working offline with local files. The app attempts to reconnect and prompts you to restart when the connection returns.

See **Work Offline When a Connection is Unavailable**.

## Batch Print for 3DEXPERIENCE Drawings (DraftSight Connected Only) (2025 FD03)



You can add files from the **3DEXPERIENCE** platform and files from bookmarks to a batch print list. You can also save the batch print output of PDF files to the **3DEXPERIENCE** platform.

### To add files from the 3DEXPERIENCE platform to a batch print list:

1. Type `BATCHPRINT` in the command window.
2. In the Batch Print dialog box, click **Add files**.
3. In the Specify File Names dialog box, click **Open from 3DEXPERIENCE**.
4. In the Open dialog box, select files and click **Open**.

### To add files from bookmarks to a batch print list:

1. Type `BATCHPRINT` in the command window.
2. In the Batch Print dialog box, click **Add folder**.
3. In the Specify Folder dialog box, click **Select from 3DEXPERIENCE**.
4. In Select a Bookmark dialog box, select bookmarks and click **Select**.

### To save the batch print output of PDF files to the 3DEXPERIENCE platform:

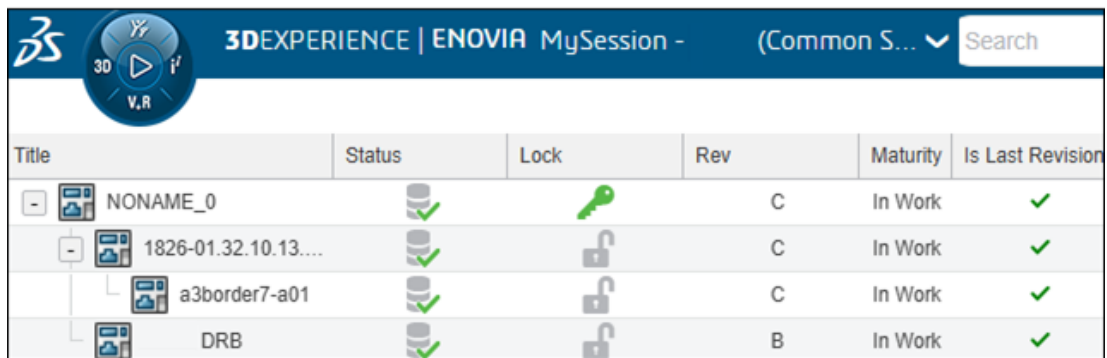
You can save the batch print output of PDF files only.

1. Type `BATCHPRINT` in the command window.
2. In the Batch Print dialog box, for **File location for built-in printer output**, click **Browse**.
3. In the Select a Bookmark dialog box, select a bookmark and click **Select**.

**Add current drawings** lets you add all the current drawings that you have opened from the **3DEXPERIENCE** platform to the batch print list.

For details, see [Processing Print Output in a Batch](#).

## Data Grid View in MySession (2025 FD03)



The screenshot shows the 3DEXPERIENCE | ENOVIA MySession interface. At the top, there is a header bar with the Dassault Systèmes logo, a circular navigation menu with icons for 3D, V.R, and i?, and a search bar. Below the header, a data grid displays file details. The grid has columns for Title, Status, Lock, Rev, Maturity, and Is Last Revision. The data is organized into a tree structure, with a root node 'NONAME\_0' and three child nodes: '1826-01.32.10.13....', 'a3border7-a01', and 'DRB'. Each row shows the file's status as 'In Work' with a green checkmark, and the revision number (C or B). The 'Lock' column shows a green key icon for the root node and a grey lock icon for the child nodes.

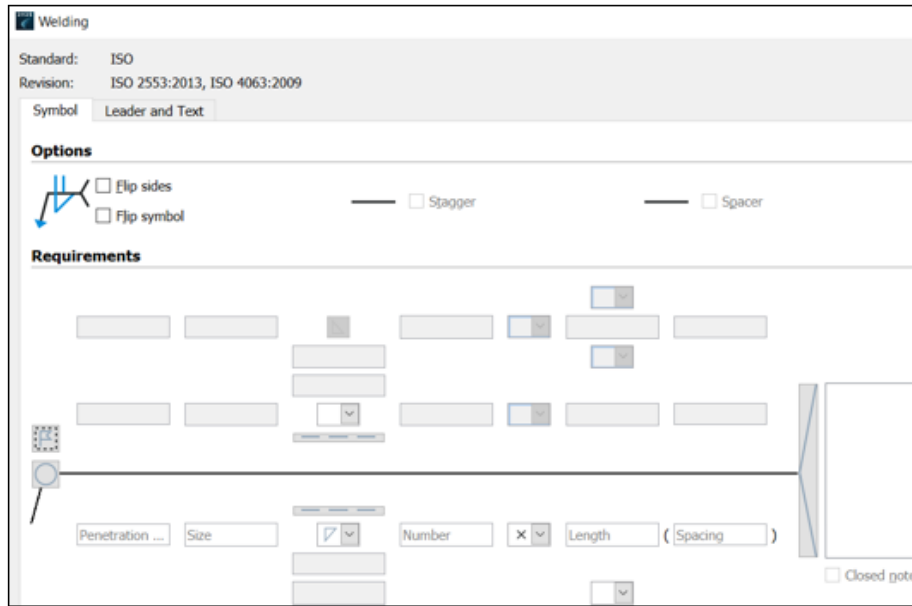
Title	Status	Lock	Rev	Maturity	Is Last Revision
NONAME_0	In Work	Locked	C	In Work	✓
1826-01.32.10.13....	In Work	Unlocked	C	In Work	✓
a3border7-a01	In Work	Unlocked	C	In Work	✓
DRB	In Work	Unlocked	B	In Work	✓

The MySession widget displays file details in a data grid view.

Previously, the MySession widget displayed the file details in a tree list view. The data grid view helps you to see the file details easily.

For details, see [Data Grid View](#) in Dassault Systèmes User Assistance. Access to Dassault Systèmes User Assistance requires 3DEXPERIENCE credentials.

## Welding Symbols (2025 SP3)



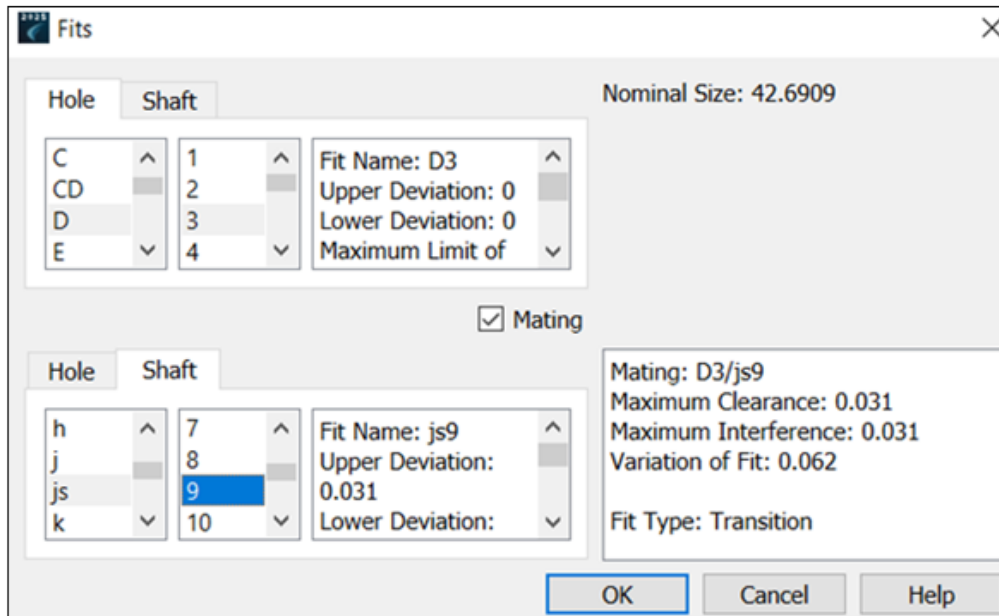
You can use the `AM_WELDINGSYMBOL` command to add welding symbols to drawings.

The welding symbols ensure clear communication of welding specifications and adherence to industry standards. They enhance the accuracy and quality of technical documentation.

The `AMWELDSYM` command offers multiple options for flexible and precise placement of welding symbols:

- Supports a range of standard welding symbols that represent various weld types.
- Lets you customize symbol type, size, angle, and placement to meet project requirements.
- Provides options for adding symbols like all around, field weld, and staggered fillet welds at the arrow's junction with the reference line.
- Lets you include multiple reference lines and arrows to indicate weld sequences and identical weld locations.
- Attaches symbols to objects and ensures they move with the object when repositioned.
- Places symbols as stand-alone annotations.
- Includes ways to add details for specific welding processes.

## Adding Fit to Dimension (2025 SP3)



You can add precise fit information to dimensions in drawings. This enhances the design process by automatically retrieving hole and shaft fit values. It retrieves values from a data table based on the selected nominal dimension.

By incorporating fit data directly into dimensions, you can ensure that mated parts achieve the required degree of tightness or looseness, adhere to industry standards, and improve assembly accuracy. Adding fit values to dimensions has the following benefits:

- Reduces manual calculations and errors by automatically fetching hole and shaft fit values from a data table.
- Lets you specify the precision of fit.
- Ensures compliance with industry-standard fit classes and representations for accurate manufacturing compatibility.
- Provides easy access to **Fit** options through the Power Dimensioning contextual ribbon tab and **Properties Palette**.
- Lets you compare shaft and hole fits, select appropriate notations, and view values in the Fit dialog box.

## Adding Tolerance to the Dimension (2025 SP3)

<b>Precision</b>		<b>X.XX Tolerance</b>	
<small>+0.00 -0.00</small> Primary	3	<small>+X -X</small> Upper	0.1
<small>[+0.00] [-0.00]</small> Alternate	4	<small>+X -X</small> Lower	-0.1
Precision		Tolerance	

60

+0.0100  
-0.0010

Method

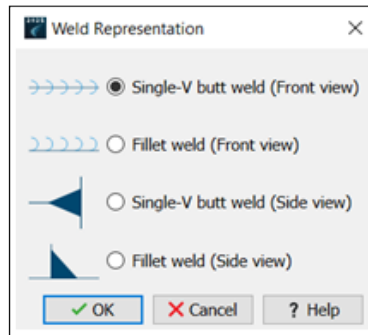
You can add tolerance information directly to dimensions in drawings.

This ensures clarity by specifying permissible variations in dimensions, and supports accurate manufacturing and assembly processes. Adding tolerance has the following benefits:

- Lets you define upper and lower limits for dimensions directly.
- Provides various tolerance methods, such as symmetrical, deviation, and limit-based representations.
- Lets you control the precision of tolerances independently.
- Provides easy access to tolerance options through the Power Dimensioning Ribbon tab and **Properties Palette**.



## Weld Representation (2025 SP3)

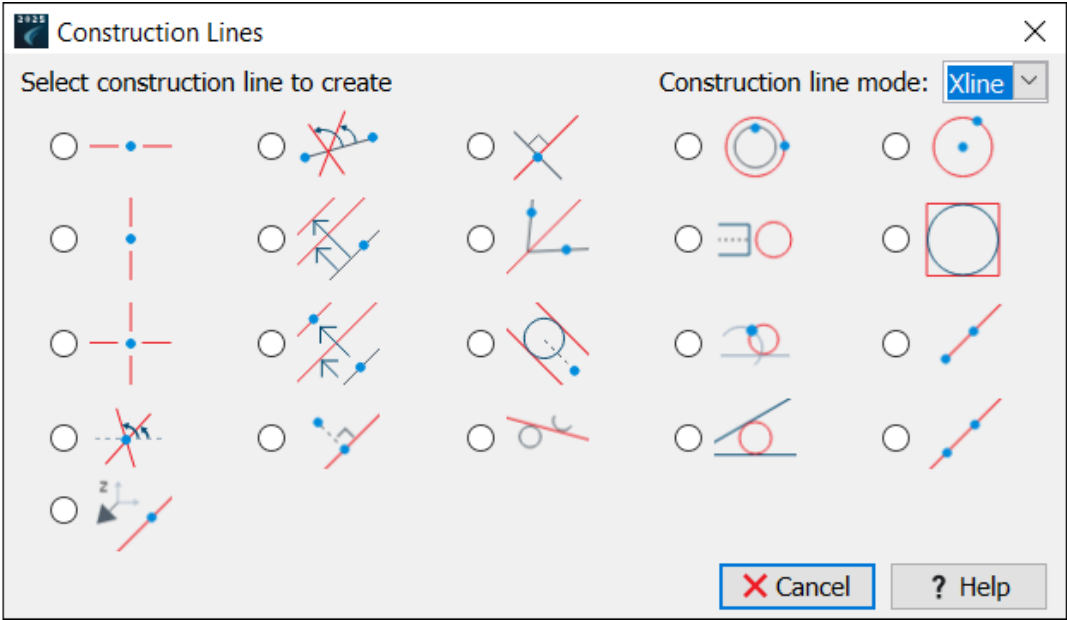


You can use the `AM_SIMPLEWELD` command to create and customize fillet welds and single-V butt welds on various entities, including ellipses, circles, arcs, lines, and polylines.

This command supports front-view and side-view weld representations, offering precise control and flexibility in defining welding details. It simplifies the process of creating weld symbols, enhances productivity, and ensures compliance with industry standards. The command:

- Offers precise control over weld dimensions, leg widths, and symbol placement.
- Lets you edit weld properties with the `AM_SIMPLEWELDEDIT` command or directly through **Properties Palette**.

## Construction Lines (2025 SP3)



You can use the `AM_CONSTLINES` command to provide a comprehensive solution for creating construction lines in drawings.

Construction lines are reference guides that help align, position, and lay out objects during the design process. This simplifies complex design tasks, improves accuracy, and enhances workflow efficiency. It offers multiple types of construction lines including rays, xlines, and circular lines.

Construction lines let you:

- Create lines extending to infinity in one or both directions or circular construction lines for referencing curved geometries.
- Snap to key points (for example, intersections or midpoints) to ensure accurate placement of objects.
- Automatically place construction lines on a dedicated layer (`AM_CL`) that you can customize, lock, and freeze for better management.
- Reduce manual calculations and drawing adjustments by providing advanced configurations like bisectors, perpendicular lines, and concentric circles.

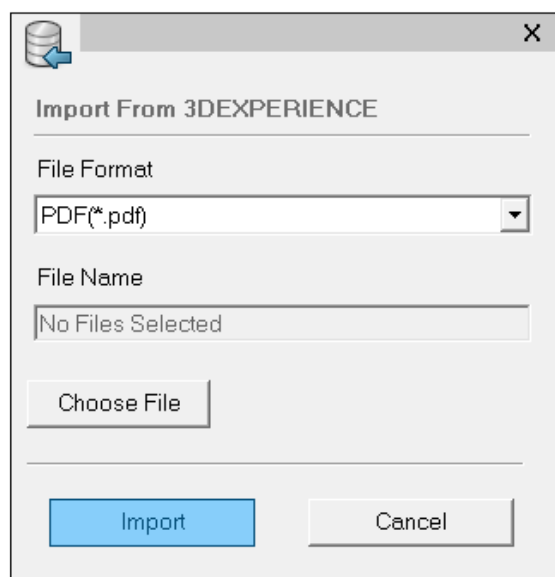
The commands used with the construction lines are:

Command	Description
<code>AM_CONSTLINES</code>	Lets you select the type of construction line.
<code>AM_CONSTSWI</code>	Turns ray mode on or off.
<code>AM_CONSTSWI_XLINE</code>	Turns ray mode off. Construction lines extend to infinity in both directions.

Command	Description
AM_CONSTSWI_RAY	Turns ray mode on. Construction lines extend to infinity in one direction only.
AM_ERASECL	Erases selected construction lines.
AM_ERASEALLCL	Erases all construction lines.
AM_CLINEL	Locks or unlocks the construction line layer.
AM_CLINEO	Freezes or thaws construction line layers.
AM_CONSTHOR	Creates a horizontal construction line.
AM_CONSTHW	Creates a construction line through a point by specifying an angle relative to an apparent line.
AM_CONSTLOT	Creates a construction line that is perpendicular to a specified line.
AM_CONSTCC	Creates a circular construction line that is concentric to the selected circle or arc.
AM_CONST_CIRCLE	Creates a circular construction line.
AM_CONSTVER	Creates a vertical construction line.
AM_CONSTPAR	Creates a construction line parallel to the existing line at a specified distance from the selected line.
AM_CONSTHM	Creates a construction line that bisects an angle.
AM_CONSTCCREA	Creates a circular construction line to represent the top view of a shaft or hole.
AM_CONSTCIRCLI	Creates a rectangular construction line around a circle.
AM_CONSTCRS	Creates a construction line cross.
AM_CONSTPAR2	Creates a construction line parallel to an existing line and bisects the distance between the selected line and a specified point.
AM_CONSTTAN	Creates two parallel construction lines tangent to a specified circle.
AM_CONSTC2	Creates a circular construction line that uses a specified line as a tangent.

Command	Description
AM_CONSTXRAY	Creates construction lines starting from one point and extending to infinity in one direction.
AM_CONSTHB	Creates a construction line by specifying two points or a point and an angle.
AM_CONSTLOT2	Creates a construction line through a specified point that is perpendicular to a direction.
AM_CONSTTC	Creates construction lines that are tangent to two specified circles.
AM_CONSTK	Creates a construction line that is tangent to two specified lines or circles.
AM_CONSTXLINE	Creates a construction line through a point, which extends to infinity in both directions.
AM_CONSTZ	Creates a construction line in the Z-direction.

## Importing a PDF File as a Block from the 3DEXPERIENCE Platform (DraftSight Connected Only) (2025 FD02)



You can use the **IMPORTPDFFROM3DEXPERIENCE** command to import a PDF file as a block from the 3DEXPERIENCE platform.

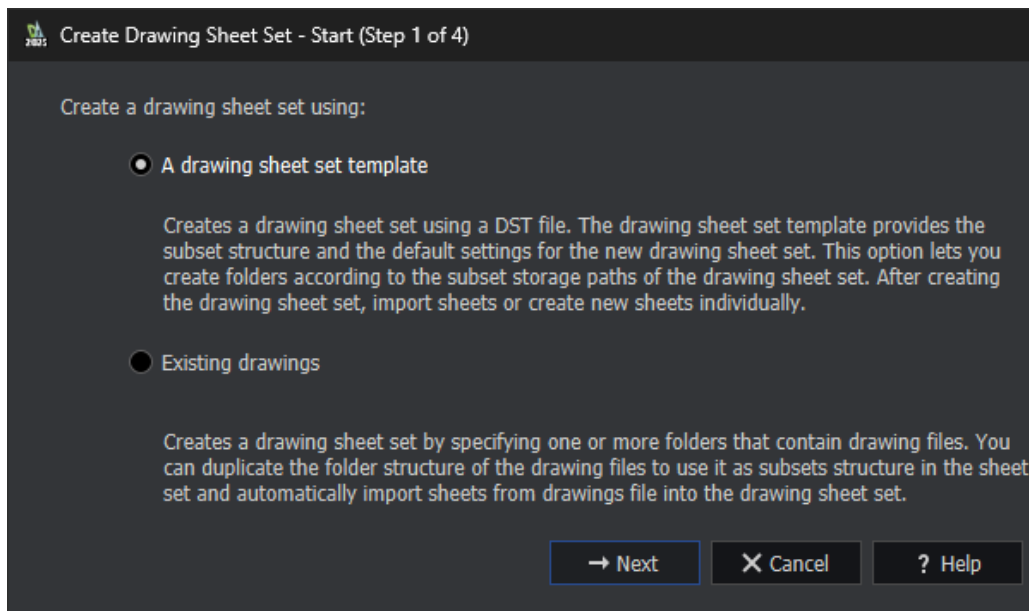
### To import a PDF file as a block from the 3DEXPERIENCE platform:

1. Do one of the following:

- Click **Import > Import from 3DEXPERIENCE**.
  - Click **File > Import > Import from 3DEXPERIENCE**.
  - Type `IMPORTPDFFROM3DEXPERIENCE` in the command window.
2. In the Import from 3DEXPERIENCE dialog box:
    - a. In **File Format**, select **PDF (\*.pdf)**.
    - b. Click **Choose File**.
  3. In the Open dialog box:
    - a. Select a PDF file.
    - b. Click **Open**.

In the Import from 3DEXPERIENCE dialog box, **File Name** displays the selected file.
  4. Click **Import**.
  5. In the PDF Import dialog box, click **OK**.

## Sheet Set Manager on the 3DEXPERIENCE Platform (DraftSight Connected Only) (2025 FD02)




**3DEXPERIENCE** DraftSight lets you create the Sheet Set Data (DST) files and save them to bookmarks. You can open the saved DST files from bookmarks.

You can also define the properties of the Sheet Set Manager. See [Working with Drawing Sheet Sets](#). You can create DST files using an existing drawing or drawing sheet set template. **3DEXPERIENCE** DraftSight creates DST files as PLM objects.

## Creating Drawing Sheet Sets Using an Existing Drawing

You can use the Create Drawing Sheet Set wizard to create drawing sheet sets from an existing drawing.


### To create drawing sheet sets using an existing drawing:

1. In the **Sheet Set Manager** palette, click **New Drawing Sheet Set** .
2. In the Create Drawing Sheet Set - Start wizard, select **Existing drawings** and click **Next**.
3. Click **Browse** for **Location for drawing sheet set data file (\*.dst)**.
4. In Browse for Drawing Sheet Set Folder dialog box, click **Select from 3DEXPERIENCE**.
5. In the Select a Bookmark dialog box:
  - a) Select an existing bookmark or create a bookmark to save the DST file to.
  - b) Click **Select**.Alternatively, you can select a folder from **This PC**.
6. Click **Drawing Sheet Set Properties** to select a bookmark for the **Model view** from the **3DEXPERIENCE** platform.  
You can select a bookmark for **Label block for views** and **Callout blocks**.
7. In the Create Drawing Sheet Set - Drawing Sheet Set Details wizard, click **Next**.
8. In the Create Drawing Sheet Set - Choose Sheets wizard, click **Browse**.
  - a) In the Browse for Folders dialog box, select a folder from your computer or a bookmark that contains drawings.
  - b) Click **Specify Folder**.
9. In the Create Drawing Sheet Set - Choose Sheets wizard, click **Next**.
10. In the Create Drawing Sheet Set - Finalize wizard, click **Finish**.

## Creating Drawing Sheet Sets Using a Drawing Sheet Set Template

You can use the Create Drawing Sheet Set wizard to create drawing sheet sets using a drawing sheet set template.

### To create drawing sheet sets using a drawing sheet set template:


1. In the **Sheet Set Manager** palette, click **New Drawing Sheet Set** .
2. In the Create Drawing Sheet Set - Start wizard, select **A drawing sheet set template** and click **Next**.
3. In the Create Drawing Sheet Set- Drawing Sheet Set template wizard:
  - a) Select **Browse to another drawing sheet set to use as a template**.
  - b) Click **Browse**.
4. In the Browse for Drawing Sheet Set dialog box, click **Open from 3DEXPERIENCE**.
5. In the Open dialog box:
  - a) Select a drawing sheet set template (.DST) from **3DSearch** or **Bookmarks**.
  - b) Click **Open**.

In the Create Drawing Sheet Set - Drawing Sheet Set Template wizard, the name of the drawing sheet set template (DST) appears.

6. In the Create Drawing Sheet Set - Drawing Sheet Set Template wizard, click **Next**.
7. In the Create Drawing Sheet Set - Drawing Sheet Set Details wizard, click **Next**.
8. In the Create Drawing Sheet Set - Finalize wizard, click **Finish**.

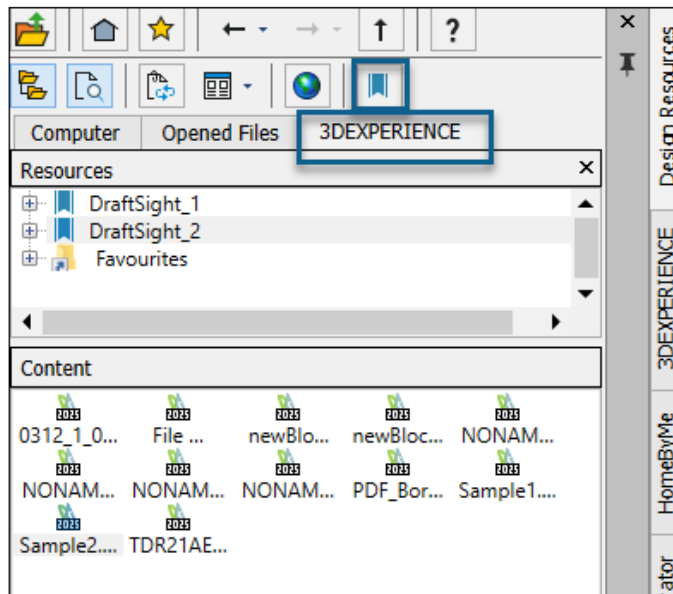
## Opening Drawing Sheet Sets

### To open drawing sheet sets:

1. In the **Sheet Set Manager** palette, click **Open Drawing Sheet Set** .
2. In the dialog box, do one of the following:
  - Select the drawing sheet set (DST) and click **OK**.
  - From **Bookmarks** or **3DSearch**, click **Open from 3DEXPERIENCE**, select the sheet set manager file, and click **Open**.


The **Sheet Set Manager** palette displays the DST file's references.

## Design Resources Palette Compatibility with the 3DEXPERIENCE Platform (2025 FD01)



The **Design Resources** palette lets you access resources and contents of drawing files available on the **3DEXPERIENCE** platform.

The compatibility is applicable for DraftSight Connected and Design with DraftSight.


**Add Bookmark**  lets you add bookmarks from the **3DEXPERIENCE** platform. In **Content**, you can view the files of the bookmark and categories of drawing files.


**Open Resources** lets you open the files from the **3DEXPERIENCE** platform.

See *DraftSight Help: Design Resources Palette*.

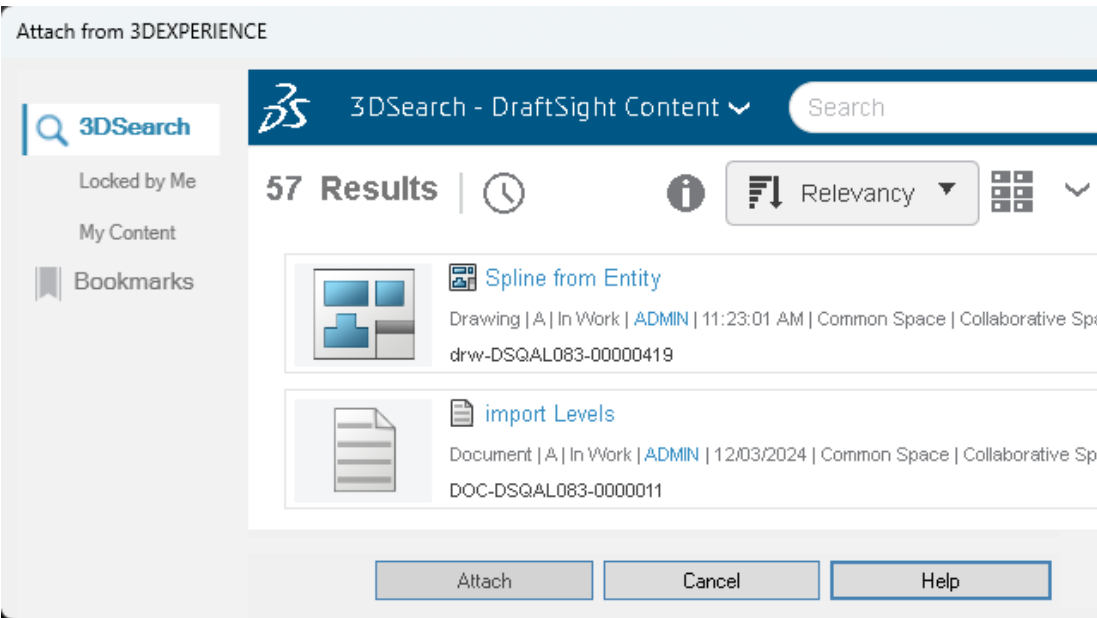
Adding Bookmarks from the 3DEXPERIENCE Platform

To add bookmarks from the 3DEXPERIENCE platform:

- 1. In the **Design Resources** palette, on the 3DEXPERIENCE tab, click **Add Bookmark** .
- 2. In the Select a Bookmark dialog box, select a bookmark and click **Select**.  
The selected bookmark appears in the list.
- 3. Right-click a bookmark and specify an option:

Option	Description
Check Status	Checks the status of the bookmark.  appears to indicate if the bookmark is not up to date.
Update	Updates the bookmark with the latest files.
Remove	Removes the bookmark from the list.

Attaching Files from the 3DEXPERIENCE Platform (DraftSight Connected Only) (2025 FD01)



You can attach drawing, image, and PDF files from the **3DEXPERIENCE** platform as external references to the current drawing.

To attach files from the 3DEXPERIENCE platform:

- 1. Do one of the following:



- In the Drafting and Annotation workspace, click **Attach** > **Attach From 3DEXPERIENCE**.
  - In the **References** palette, select **Attach From 3DEXPERIENCE**.
  - Type `ATTACHFROM3DEXPERIENCE` in the command window.
2. In the Attach From 3DEXPERIENCE dialog box, select one of the following:
    - **3DSearch**
    - **Locked by Me**
    - **My Content**
    - **Bookmarks**
  3. Select a file to attach.

You can use **6WTags** to search for the specific file type.

Based on the file type you select, the corresponding dialog box opens:

File Type	Dialog Box
DWG file	Attach Reference: Drawing
PDF	Attach Reference: PDF Underlay
DGN	Attach Reference: DGN Underlay
PNG	Attach Reference: Image Underlay

4. Select a file to attach and click **Attach**.  
The selected file gets attached to the drawing file.

## Attach from 3DEXPERIENCE Dialog Box

The Attach From 3DEXPERIENCE dialog box lets you attach files from **3DSearch**, **Bookmarks**, **My Content**, and files locked by you.

### To open the dialog box:

Do one of the following:

- In the Drafting and Annotation workspace, click **Attach** > **Attach From 3DEXPERIENCE**.
- In the **References** palette, select **Attach From 3DEXPERIENCE**.
- Type `ATTACHFROM3DEXPERIENCE` in the command window.

## 3DSearch

Displays files saved on the **3DEXPERIENCE** platform.

## Locked by Me

Displays files locked by you. Click **Clear Filter** to clear the results and display all files.

## My Content

Displays files created by you. Click **Clear Filter** to clear the results and display files created by all users.

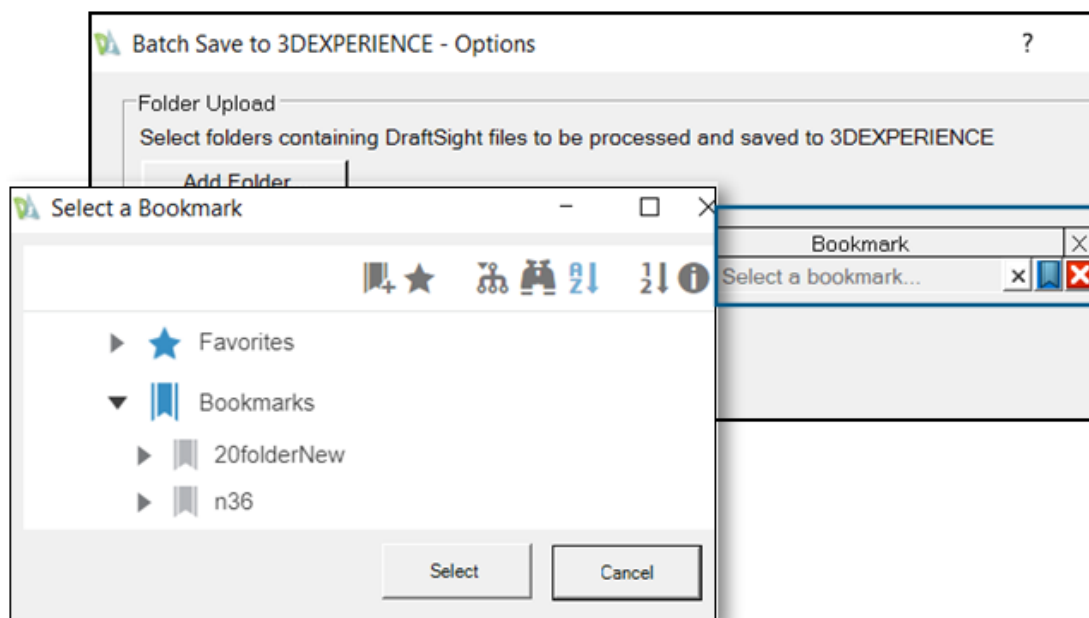
## Bookmarks

Displays bookmarks and files saved to the bookmarks.

## Attach

Attaches the selected file to the drawing.

## Bookmarks for Batch Save to 3DEXPERIENCE (DraftSight Connected Only)




You can batch upload files to bookmarks on the **3DEXPERIENCE** platform.

To open the Batch Save to **3DEXPERIENCE** - Options dialog box, on the ribbon, click **DraftSight > Batch Save to 3DEXPERIENCE**.

### Select a Bookmark Dialog Box

You can use this dialog box to select an existing bookmark or create new bookmarks.

To access the Select a Bookmark dialog box, in the Batch Save to 3DEXPERIENCE - Options

dialog box, click  .

## Toolbar

Tool	Description
<b>New Bookmark</b>	Creates a new bookmark.
<b>Favorite</b>	Marks bookmarks as favorites.
<b>Expand All</b>	Expands the folder structure.
<b>Find in Tree</b>	Searches for the file in the selected bookmark.
<b>Alphabetical Order</b>	Sorts the bookmarks in alphabetical order.
<b>Date Order</b>	Sorts the bookmarks based on the creation date.

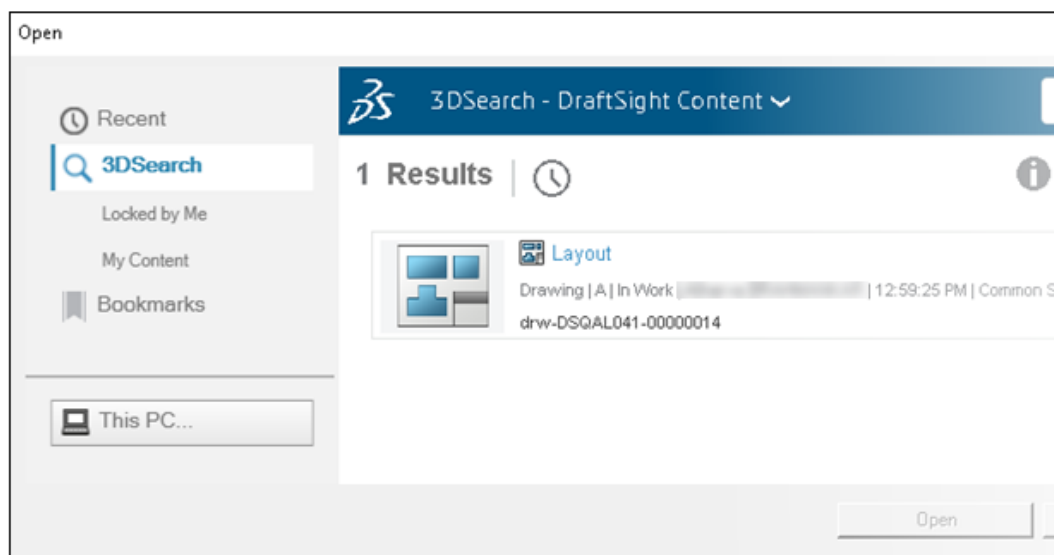
## Favorites

Lists the favorite bookmarks.

## Bookmarks

Lists the bookmarks available on the **3DEXPERIENCE** platform and newly created bookmarks.

## Open Dialog Box (DraftSight Connected Only)



You can use the Open dialog box to open recently opened drawing files and files on 3DSearch, locked by you, in My Content, and in bookmarks.

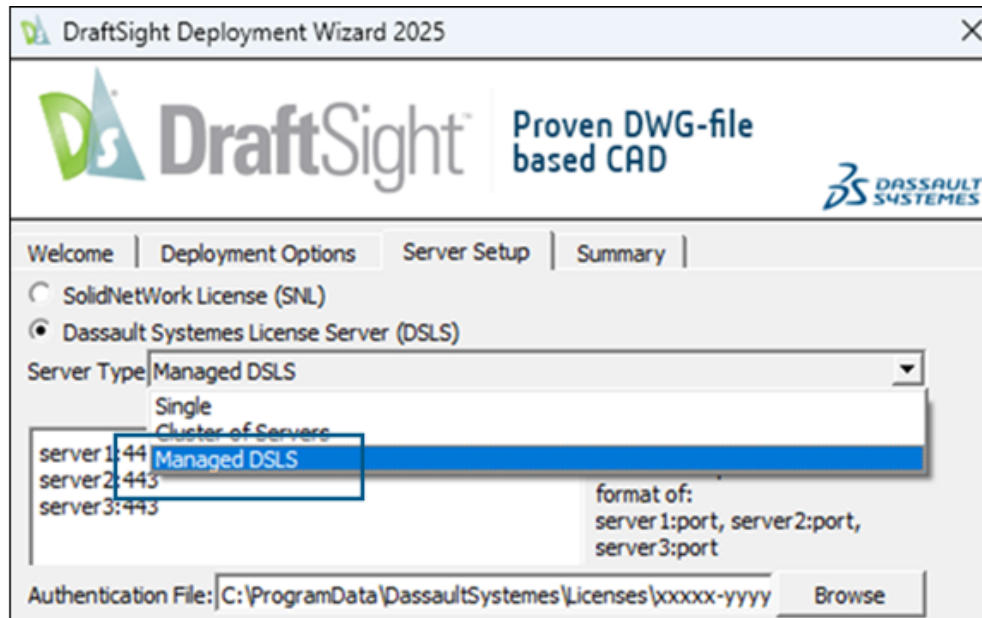
The dialog box contains various options that let you display files in the Results panel.

To open the dialog box, do one of the following:

- Click **Open** (Quick Access toolbar).
- Click **File > Open**.
- Type `Open` in the command window.

Option	Description
<b>Recent</b>	Displays the recently opened files. The cloud symbol denotes the file that you have opened on the <b>3DEXPERIENCE</b> platform. Select the file and click <b>Open</b> to open it.
<b>3DSearch</b>	Displays the files saved on the <b>3DEXPERIENCE</b> platform.
<b>Locked by Me</b>	Displays the files locked by you. Click <b>Clear Filter</b> to clear the results and display all files.
<b>My Content</b>	Displays the files created by you. Click <b>Clear Filter</b> to clear the results and display the files created by all users.
<b>Bookmarks</b>	Displays the bookmarks and files saved to the bookmarks.
<b>This PC</b>	Opens the locally saved files.
<b>Open</b>	Opens the file that you selected from results. <div>If you are working in the offline mode, you can open only recently opened and locally saved files.</div>

## Managed DS License Server



DraftSight supports Managed DS License Server.

Managed DS License Server (DSLS) is also known as the Managed Licensing Service. With Managed DSLS, on-premises customers do not require a physical computer to install the DSLS.

See [Managed Licensing Service](#).

### Setting up Managed DSLS in the Deployment Wizard

You can use the **Managed DSLS** server type when you set up the server in the DraftSight deployment wizard.

#### To set up Managed DSLS in the deployment wizard:

1. In DraftSight Deployment Wizard, select **Dassault Systemes License Server (DSLS)**.
2. For **Server Type**, select **Managed DSLS**.
3. Enter the server details that you received when you selected managed licensing service mode.

### Setting up Managed DSLS in DraftSight

You can set up the **Managed DSLS** server type when you install DraftSight.

When you install DraftSight, select **Dassault Systemes License Server (DSLS)** as the license type.

#### To set up a Managed DSLS in DraftSight:

1. In the DraftSight License Administrator, select **Add Server**.
2. For **Server Type**, select **Managed DSLS**.

## DGN File Export

You can use the `EXPORTDGN` or `DGNEXPORT` commands to export DGN files.

### To use DGN file export:

Do one of the following:

- On the ribbon, click **Menu > Export > DGN Export**.
- On the menu, click **Menu > Export > DGN Export**.
- Enter `EXPORTDGN` or `DGNEXPORT` in the command window.

## Auto-Fill Table Cells

	A	B	C	D	E	F
1	ITEM	DAY	MONTH	YEAR	DATE	VALUE
2	1	12	January	2023	24-10-2027	1,250
3	2	13	February	2024	25-10-2027	2,250
4	3	14	2025	26-10-2027	3,250	
5	4	15	April	2026	27-10-2027	4,250

Auto-fill is useful when you want the data in a logical or repetitive order in the adjacent cells of a table. The data includes dates, sequential numbers, days of the week, months etc.

This feature is also useful when you want to repeat the formula of one cell to others.

### To use Auto-fill feature:

1. Select a cell.

The fill handle appears in the lower right corner of the selected cell.

2. Drag the fill handle in the direction that you want to fill the data.

The cells in the row are auto-filled if you drag the handle horizontally. The cells in the column are auto-filled if you drag the handle vertically.

## Accessing Tables and Creating Table Breaks

Part Number	Description	Length (mm)	Width (mm)	Height (mm)	Weight (kg)
P001	Gear Assembly	120	50	30	0.75
P002	Bearing Housing	80	80	40	0.45
P003	Piston Rod	200	25	25	1.2
P004	Valve Body	90	60	35	0.6
P005	Cylinder Head	150	70	50	1.8
P006	Shaft	180	20	20	1
P007	Spring	60	10	10	0.15
P008	Bearing	30	30	15	0.25
P009	Flange	120	80	30	1.5
P010	Bolt	10	5	5	0.05
P011	Nut	10	10	5	0.03
P012	Washer	15	15	1	0.02
P013	Gasket	40	40	2	0.08

P014	Pin	25	3	3	0.01
P015	Bracket	70	40	20	0.7
P016	Connector	50	30	15	0.4
P017	Plate	100	60	5	0.3
P018	Rod	130	10	10	0.5
P019	Sleeve	40	40	30	0.9
P020	Bushing	35	20	15	0.2
P021	Hinge	50	15	10	0.25
P022	Cam	75	25	25	0.6
P023	Spacer	15	15	3	0.05
P024	Bracket	60	30	10	0.4
P025	Lever	90	10	5	0.2
P026	Plug	20	20	10	0.1
P027	Seal	25	25	2	0.08
P028	Screw	8	4	4	0.02
P029	Key	12	6	6	0.03
P030	O-Ring	18	18	2	0.02

You can use the `TABLE` command to create tables and break large tables into multiple tables so they fit in a drawing area or sheet.

For tables with many rows, you can break the table so the table displays the rows side by side.

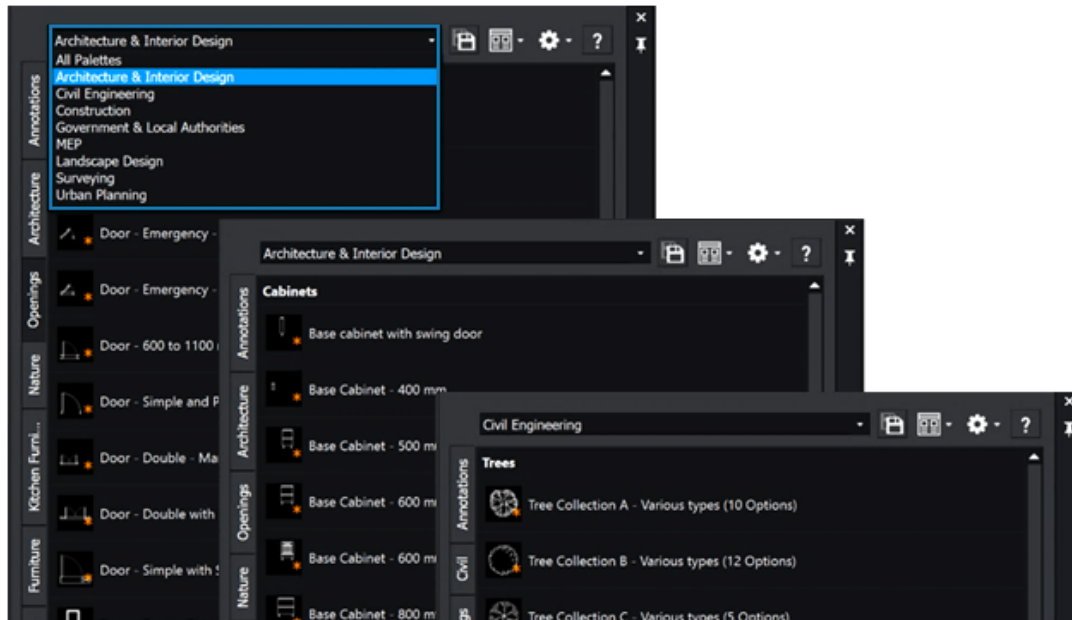
You can define the table height by dragging the grip point or you can enter the height in a drawing unit in the Properties palette.

### To access tables:

Do one of the following:

- On the ribbon, click **Annotate** > **Table** > **Insert**.
- On the menu, click **Draw** > **Table**.
- Enter `TABLE` in the command window.

## Libraries of Dynamic Blocks



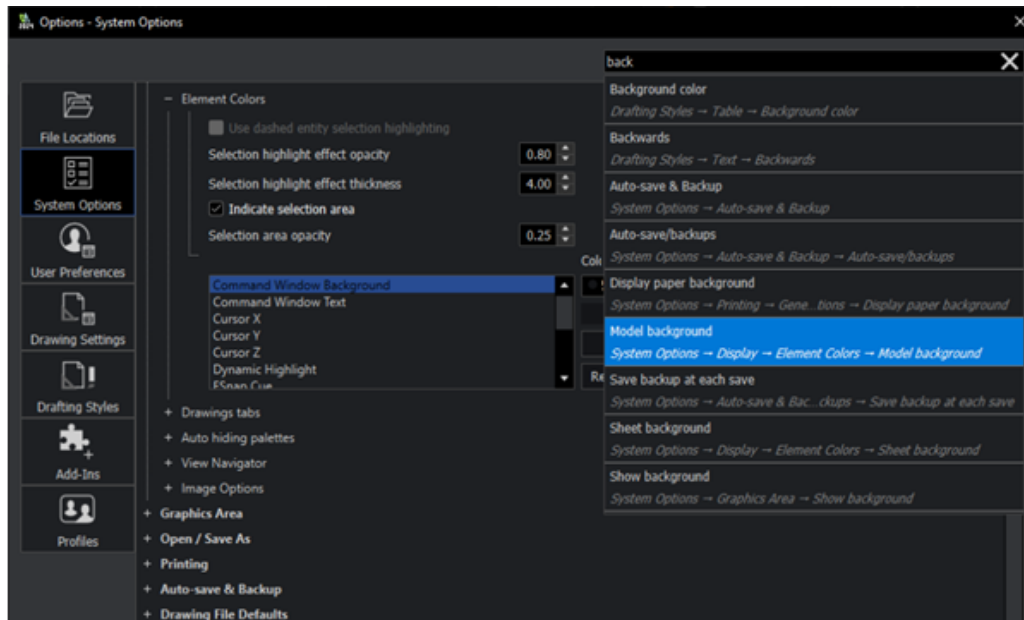
The tool palettes include more than 400 dynamic blocks. The blocks are parametric and compatible with AutoCAD®.

Instead of creating new blocks to adapt or update a design, you can adjust the size, shape, and configuration of the dynamic blocks. This can simplify the drawing process and reduce repetitive tasks.

The dynamic blocks include symbols for architecture, interior design, HVAC, electricity, plumbing, civil engineering, and urban planning. They are grouped into palettes according to the industry focus.



## Dynamic Search in an Options Dialog Box



The search functionality in the Options dialog box is more intuitive and user friendly, ensuring that you find options quickly.

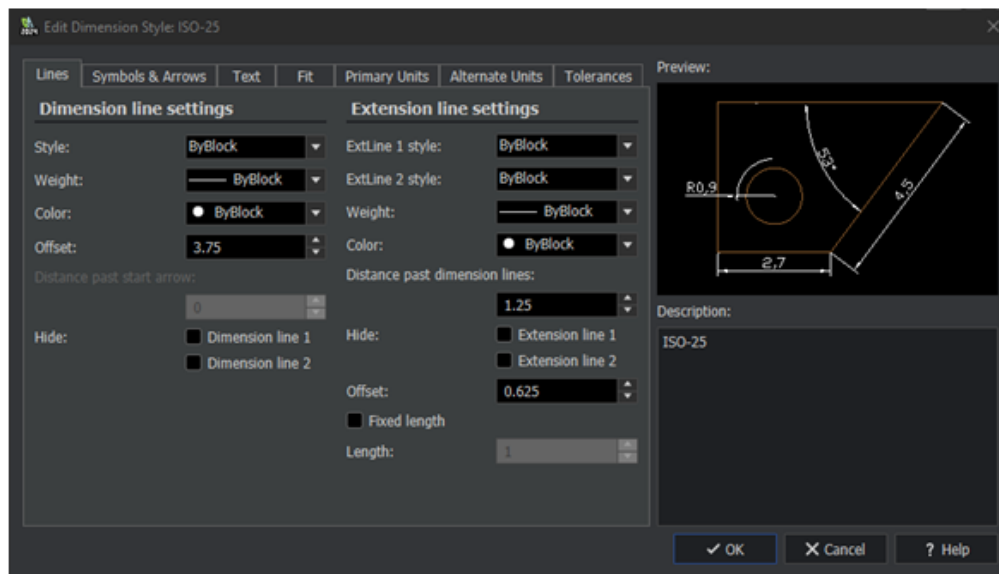
In the Search box of the Options dialog box, start to enter a term or system variable name to see a list of options containing the string that you entered. Relevant options appear in a list. You can click an option to go directly to the specified option.

### To use the dynamic search in the Options dialog box:

Do one of the following:

- On the ribbon, click **Manage** > **Customization** > **Options**.
- On the menu, click **Tools** > **Options**.
- Enter `OPTIONS` in the command window.

## Dimension Styles Dialog Box



The Dimension Styles dialog box is simplified for editing Dimension Styles.

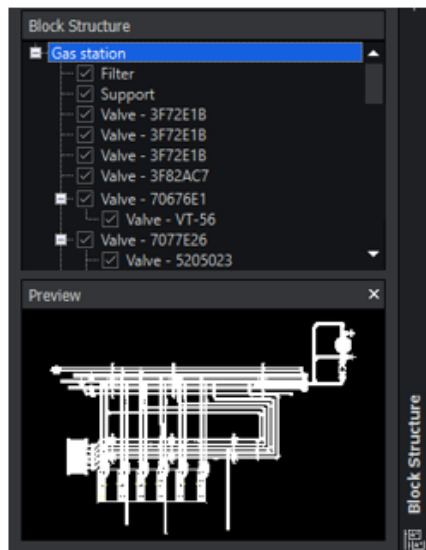
When you edit Dimension Styles, the user interface more closely resembles the AutoCAD interface. This ensures a smoother transition for users migrating from AutoCAD to DraftSight.

### To access the Dimension Styles dialog box:

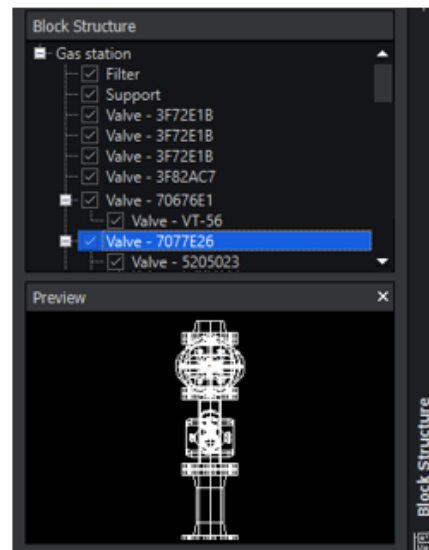
Do one of the following:

- On the ribbon, click **Annotate** > **Dimension** > **Dimension Style**.
- On the menu, click **Format** > **Dimension Style**.
- Enter `DIMSTYLE` / `DIMENSIONSTYLE` in the command window.

## Block Structure Palette



Main drawing selected



Nested block selected

The Block Structure palette helps you visualize, manage, and navigate complex block hierarchies. It enhances the efficiency and organization of design and drafting tasks.

A block structure is an arrangement of nested blocks that create a hierarchy. The Block Structure palette displays the nested block structures and provides a way for you to manage the blocks.

The palette provides the following benefits:

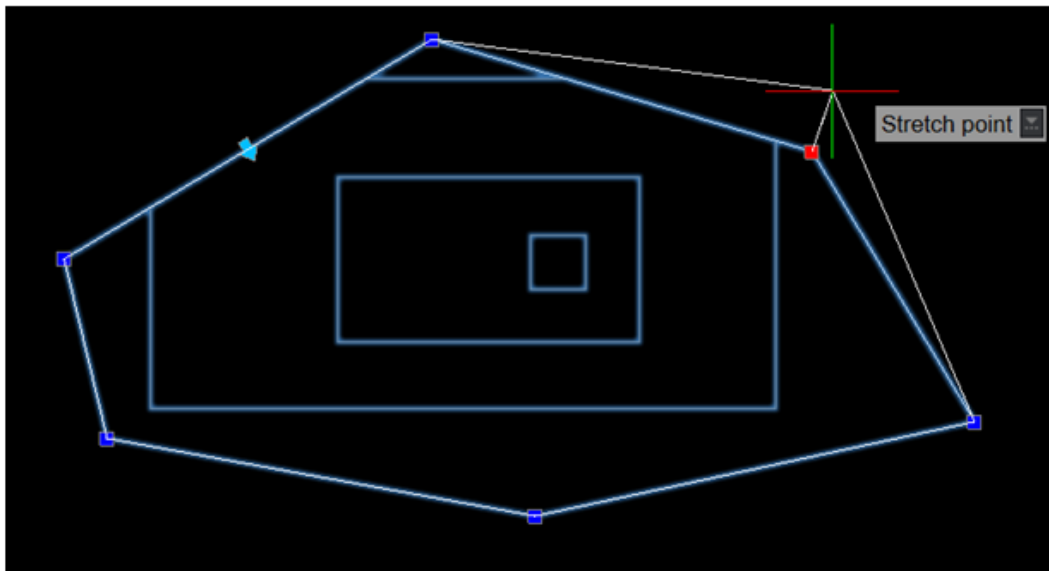
- Hierarchy visualization. This helps when dealing with large and intricate designs that have numerous nested blocks.
  - Get a structured view of block organization within a drawing.
  - Display a visual representation of the block structure for a selected block instance.
  - Facilitate the creation and management of hierarchical block structures.
  - Highlight nested blocks within the main block or parent block. A block may serve as a nested block within several parent blocks. The palette displays the block as a nested element within all relevant parent block structures.
  - Support for nested-inside-nested block structures.
  - Collapse or expand the block structure.
  - Control the level of detail displayed.
  - Show or hide individual block instances in the graphics area.
  - Manage the visibility of specific blocks within the structure.
- Block management. Enhanced organization ensures that the CAD drawing remains coherent and easier to work with.
  - Access and edit nested blocks directly from the palette, streamlining the editing process when blocks contain other nested blocks. For example, a window block nested within a wall block.

- Copy blocks from one area of the drawing and paste them elsewhere, maintaining the hierarchical structure. This simplifies the process of reusing design elements and maintaining consistency in the drawing.
- Rename, group, organize, or delete blocks within the palette.
- Nest a block within another block in the drawing.
- Navigation. This makes it easier to locate and edit specific elements within the design, saving time and effort.
  - Navigate through the drawing by selecting blocks in the palette.
  - Locate and focus on particular elements within complex block structures.
  - Zoom in on individual block instances in the graphics area.

#### To open the Block Structure palette:

- On the ribbon, click **Insert Tab > Palettes Section > Block Structure**.
- On the menu, click **Tools > Sheet Set Manager > Block Structure**.
- Enter `BLOCKSTRUCTURE` in the command window.

## Editing Clipped External References and Blocks



When you clip a block or an externally referenced (xref) drawing, you can resize or edit their boundaries with grips. In earlier releases, you had to recreate the clip each time you resized or edited the boundaries.

This makes it easier to isolate a specific entity or area from the block or xref drawing to display in the graphics area.

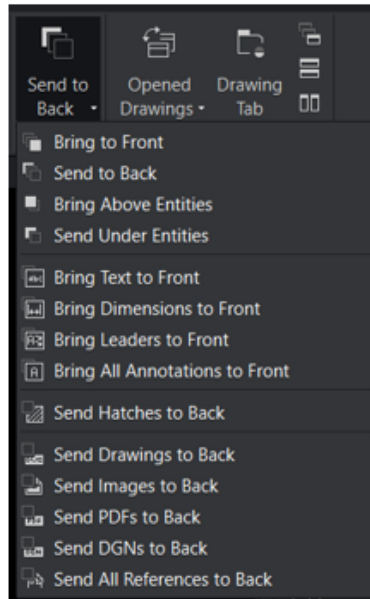
#### To edit clipped external references and blocks:

Do one of the following:

- On the ribbon, click **Insert > Reference > Clip > Reference**.
- On the menu, click **Modify > Clip > Reference**.

- Enter `CLIPREFERENCE` (XCLIP) in the command window.

## Drawing Order



The **Drawing Order** command has options tailored to specific entity types. The options offer more control over the visibility of annotations, further streamlining the design process.

The options provide:

- Improved clarity. Ensures that crucial design elements, such as dimensions and annotations, are visible in the visual hierarchy.
- Efficient workflow. Allows granular control over layering to efficiently manage the visibility of different elements, reducing the time spent on manual adjustments.
- Enhanced precision. Provides more precision in CAD designs by bringing specific elements to the front and sending others to the back.

Option	Description
<b>Bring Annotations to Front</b>	<p>Brings all annotation entities, including text, dimensions, and leaders, to the forefront of the design.</p> <p>By consolidating annotations in the foreground, you enhance the communication of critical information, improve the legibility of design annotations, facilitate better comprehension of measurements, and simplify the review and presentation process.</p> <p>You can create more precise, visually appealing, and impactful drawings while promoting efficient collaboration and communication.</p>
<b>Send Hatches to Back</b>	<p>Relegates hatches to the background, ensuring unobstructed visibility of underlying entities.</p> <p>This is useful when hatch patterns interfere with the clarity and comprehensibility of the design, providing clearer drawing views.</p>
<b>Send References to Back</b>	<p>Relegates references to the background, optimizing the visibility of primary design elements.</p> <p>This lets you focus on the key components, resulting in improved efficiency and accuracy during the design process.</p>

Consider a user has a detailed floor plan for a commercial building. The project involves several dimensions, annotations, and graphical elements, making layers and visibility crucial for clarity and precision.

By using the Bring to Front and Send to Back options, you have more control over layering. You can bring dimensions, leaders, text, and annotations to the front, while sending hatches, drawings, and images in DGN and PDF formats to the back.

**To access the TEXTTOFRONT, HATCHTOBACK, or REPERENCETOBACK commands:**

Do the following:

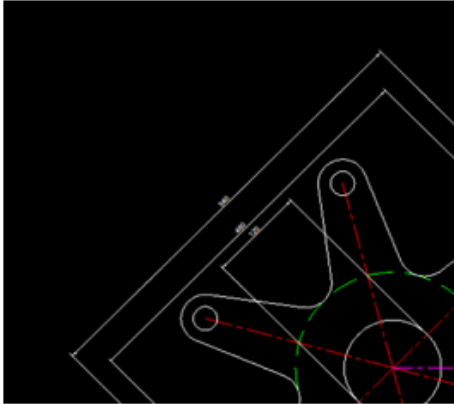
Ribbon	Menu
<b>View &gt; Order &gt; Bring Text to Front</b>	<b>Tools &gt; Display Order &gt; Bring Annotations to Front &gt; Text Only</b>
<b>View &gt; Order &gt; Bring Dimensions to Front</b>	<b>Tools &gt; Display Order &gt; Bring Annotations to Front &gt; Dimensions Only</b>

<b>Ribbon</b>	<b>Menu</b>
<b>View &gt; Order &gt; Bring Leaders to Front</b>	<b>Tools &gt; Display Order &gt; Bring Annotations to Front &gt; Leaders Only</b>
<b>View &gt; Order &gt; Bring All Annotations to Front</b>	<b>Tools &gt; Display Order &gt; Bring Annotations to Front &gt; All Annotation Entities</b>
<b>View &gt; Order &gt; Send Hatches to Back</b>	<b>Tools &gt; Display Order &gt; Send Hatches to Back</b>
<b>View &gt; Order &gt; Send Drawings to Back</b>	<b>Tools &gt; Display Order &gt; Send References to Back &gt; Drawings Only</b>
<b>View &gt; Order &gt; Send Images to Back</b>	<b>Tools &gt; Display Order &gt; Send References to Back &gt; Images Only</b>
<b>View &gt; Order &gt; Send PDFs to Back</b>	<b>Tools &gt; Display Order &gt; Send References to Back &gt; PDFs Only</b>
<b>View &gt; Order &gt; Send DGNs to Back</b>	<b>Tools &gt; Display Order &gt; Send References to Back &gt; DGNs Only</b>
<b>View &gt; Order &gt; Send All References to Back</b>	<b>Tools &gt; Display Order &gt; Send References to Back &gt; All Referenced Entities</b>

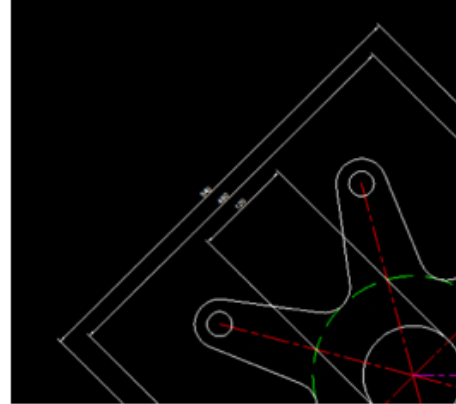
Or

Enter `TEXTTOFRONT`, `HATCHTOBACK`, or `REFERENCETOBACK` in the command window.

## Managing Spacing Between Dimensions



Before DIMSPACE



After DIMSPACE

You can use the `DIMSPACE` command to manage the spacing between dimensions in DWG files. This ensures precision, clarity, and design consistency in drawings.

With the `DIMSPACE` command, you have greater precision and can spend less time on manual adjustments. The `DIMSPACE` command is similar to AutoCAD functionality for drawing dimensions, so it is easy to learn if you are familiar with AutoCAD.

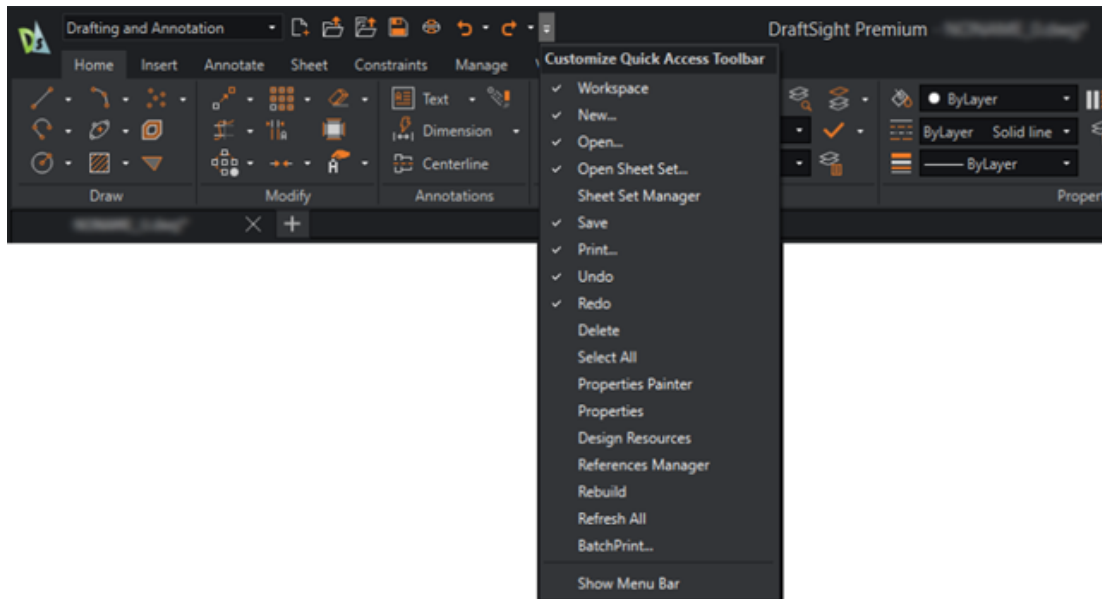
### To manage spacing between dimensions:

Do one of the following:

- On the ribbon, click **Annotate** > **Dimensions** > **Adjust Space**.
- On the menu, click **Dimension** > **Adjust Space**.
- Enter `DIMSPACE` in the command window.



## Menu Bar Visibility



You can use the ribbon and menu bar simultaneously in the user interface.

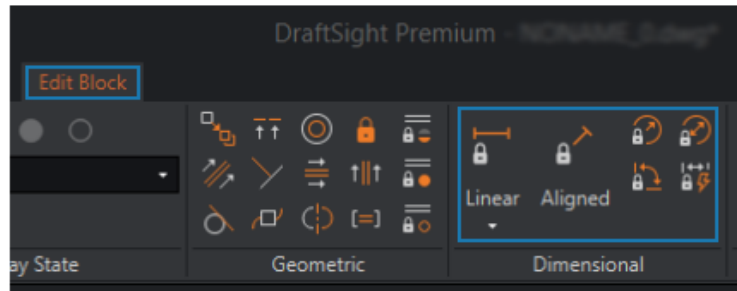
The **Customize Quick Access Toolbar** functionality switches the menu bar visibility.

**To specify the MENUBAR visibility, do one of the following:**

- On the ribbon, click **Customize Quick Access Toolbar** > **Show Menu Bar / Hide Menu Bar**.
- On the menu, click **Customize Quick Access Toolbar** > **Show Menu Bar / Hide Menu Bar**.
- In the command window, enter `MENUBAR`.

The System variable 0 is Off and 1 is On.

## Dimensional Constraints for Custom Blocks



When you edit the CustomBlocks, you can use Dimensional Constraints. This lets you control the distance, length, angle, and radius of entities. Dimensional Constraints can also constrain the distances and angles between geometric entities or points on entities.

For example, if you design a layout for a circuit board, you must position electronic components at specific locations. It is important to maintain precise distances and proportions between components, while allowing for flexibility in their individual sizes. You can replicate it in different parts of the drawing using it inside a CustomBlock.

You can edit dynamic blocks created in AutoCAD that use Dimensional Constraints. This transforms the Block into a CustomBlock in DraftSight. The conversion process recognizes Dimensional Constraints for precise editing within CustomBlocks.

### To use Dimensional Constraints for CustomBlocks:

Do the following:

- On the ribbon, click **Insert** > **Block** > **Edit Block**.
- On the menu, click **Modify** > **Entity** > **Edit Block**.
- Enter `EDITBLOCK` in the command window.

## FLATTEN Command

With the `FLATTEN` command, you can automatically specify the elevation (Z value) of certain commands as 0.

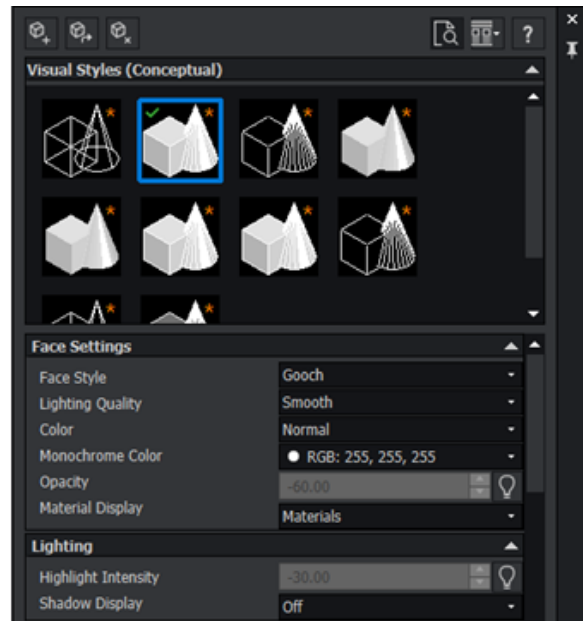
With certain commands (such as `TRIM`, `FILLET`, and `JOIN`) and other tools (snap, measure, and dimension), you need to specify the elevation (Z value) as 0. Otherwise, the commands and tools do not work as expected. The `FLATTEN` command ensures that the elevation is 0.

**To access the FLATTEN command:**

Do the following:

- On the ribbon, click **XtraTools** > **Modify** > **Flatten**.
- On the menu, click **XtraTools** > **Modify** > **Flatten**.
- Enter `FLATTEN` in the command window.

## Visual Styles



You can represent 3D models with an specified appearances. For example, if the model is in the schematic design stage, you can show the model to a design team in a “sketch appearance” and present it to customers in a “realistic appearance.”

The different appearances, called Visual Styles, depend on the settings that change the edge, color, and shading display.

The following table lists the benefits of Visual Styles:

Compatibility with AutoCAD	Ensure visual consistency between applications. If you create visual styles in AutoCAD such as transparency or wood textures, you can apply the same styles to models in DraftSight.
Enhanced visualization	Use diverse rendering options to choose the most suitable style for projects. This enhances the visual representation of designs, which improves communication and understanding.
Improved communication	Create more realistic and visually compelling drawings. This helps when you share designs with clients, stakeholders, or team members who may not be familiar with technical drawings.

Efficient analysis	Analyze designs more efficiently. For instance, use a hidden-line Visual Style to identify obscured or overlapping elements in complex drawings.
High-quality presentations	Improve the quality of presentations and design proposals. You can showcase designs as polished and professional, enhancing the overall impact.
Customization options	Customize Visual Styles to meet specific needs. You can tailor the visual representation of designs to match project requirements or personal preferences.
3D modeling capabilities	View and manipulate 3D models from different perspectives. This helps you to understand the spatial relationships within the design.

#### To access the VISUALSTYLES command:

Do the following:

- On the ribbon, click **View** > **Visual Styles** panel **Visual Styles** > **Visual Styles Manager**.
- On the menu, click **View** > **Visual Styles**.
- Enter VISUALSTYLES in the command window.

### Preset Visual Styles

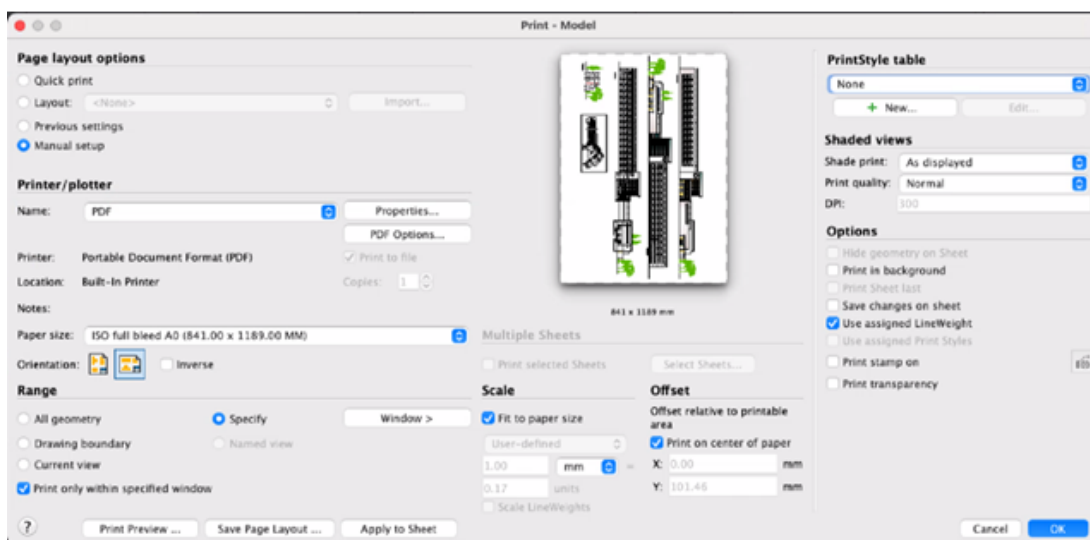
DraftSight provides preset Visual Styles that you can edit to create customized Visual Styles.

You can adjust lighting for realism, refine edge visibility, or choose a specific face style to shape the design environment according to project requirements.

Visual Style	Description
<b>2D Wireframe</b>	Uses only lines and curves without shading or rendering.
<b>Wireframe</b>	Suitable for viewing and editing 3D models with lines and curves.
<b>Hidden</b>	Uses hidden lines removed to provide a clear view of visible lines.
<b>Realistic</b>	Adds realistic lighting and shading to the model, providing a lifelike representation of materials and textures.
<b>Conceptual</b>	Applies a stylized rendering to the model, emphasizing contours and shapes. Useful for conceptual design and artistic presentations.

Visual Style	Description
<b>Shaded</b>	Displays the model with flat shading.
<b>Shaded with Edges</b>	Combines shaded surfaces with visible edges to define the boundaries of objects in the model.
<b>Shades of Gray</b>	Displays the drawing in varying shades of gray to differentiate between different objects and their elevations. This provides a monochromatic, effective representation.
<b>X-Ray</b>	Makes all objects transparent so you can see through the model. Helpful for analyzing complex assemblies.
<b>Sketchy</b>	Applies a hand-drawn, sketch-like appearance to the model, giving it a more artistic and informal look.

## Printing in MacOS



If you run DraftSight on macOS®, the Print dialog box uses a similar interface to that in Windows®. The dialog box is more versatile and user friendly.

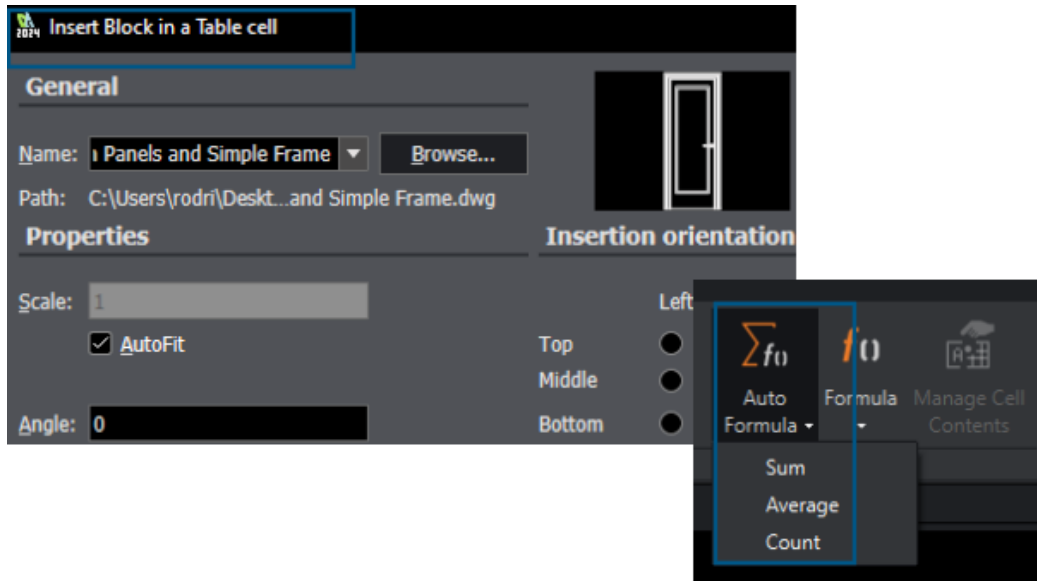
Unlike the system Print dialog box, this dialog box provides a broader range of options, giving you greater control over printing preferences. Printing is simpler and more efficient, ensuring that drawings print the way you want them.

Users can also switch between Windows and Mac without changing their habits, as the Windows and Mac versions share the same ribbon user interface.

## AMUSERHATCH Command (DraftSight Mechanical Only)

You can use the `AMUSERHATCH` command to insert user-defined, predefined, and nonassociative hatches into object areas. You can modify the properties of a selected hatch before inserting it into an object area.

## Table Edits



You can use advanced features when editing tables.

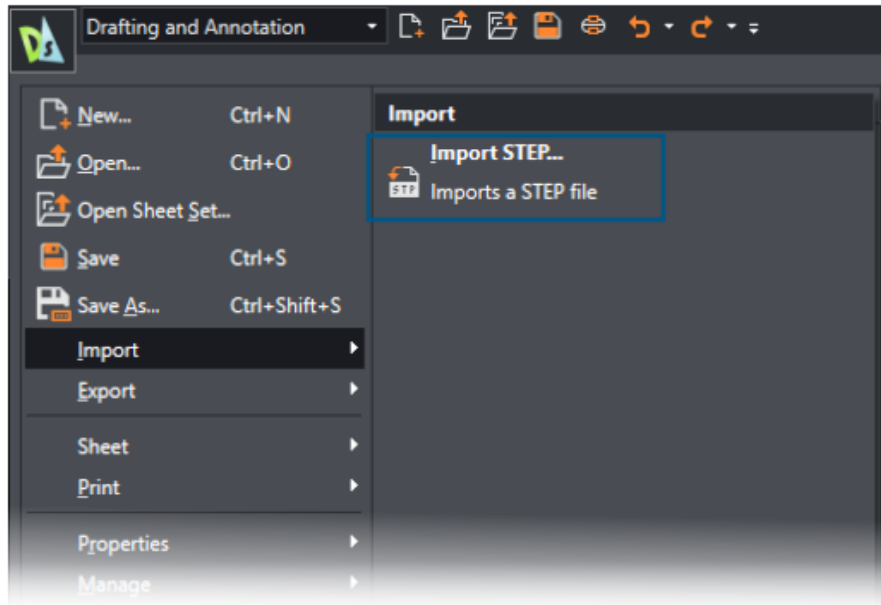
To make tables more useful, you can:

- Insert and manage blocks in table cells
- Match cell properties
- Repeat features

Improved table functionality:

- **Formula** options such as **AutoSum**
- **Add** rows and columns
- Grips
- **Cell** shortcut menu and **Table** contextual ribbon

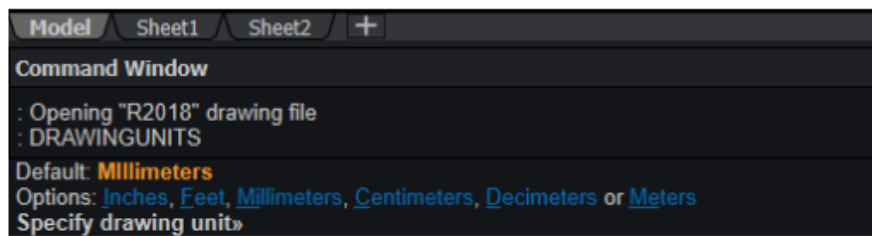
## Import STEP Files



You can use the `IMPORTSTEP` command to import 3D models from `STEP` files.

You can incorporate `STEP` file models into drawings.

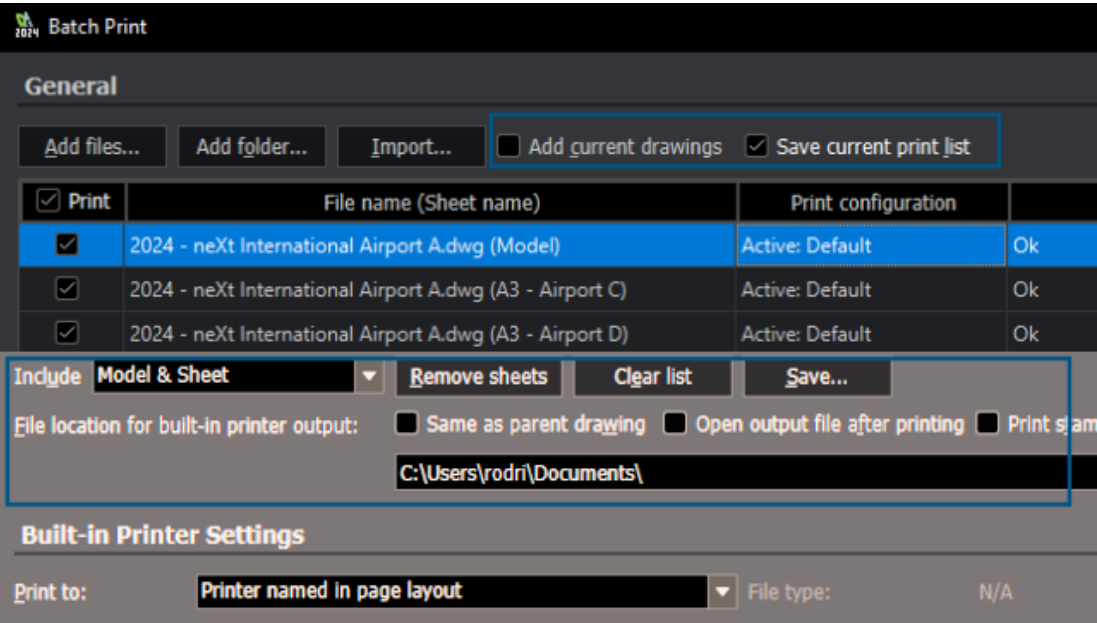
## DWGUNITS Command



The `DWGUNITS` command converts drawings to other unit systems.

For imperial and metric units, the `DWGUNITS` command lets you maintain precision and consistency in various projects. This command enhances the workflow efficiency and ensures that the drawing adheres to project requirements and industry standards.

PDF Export and Batch Print Usability

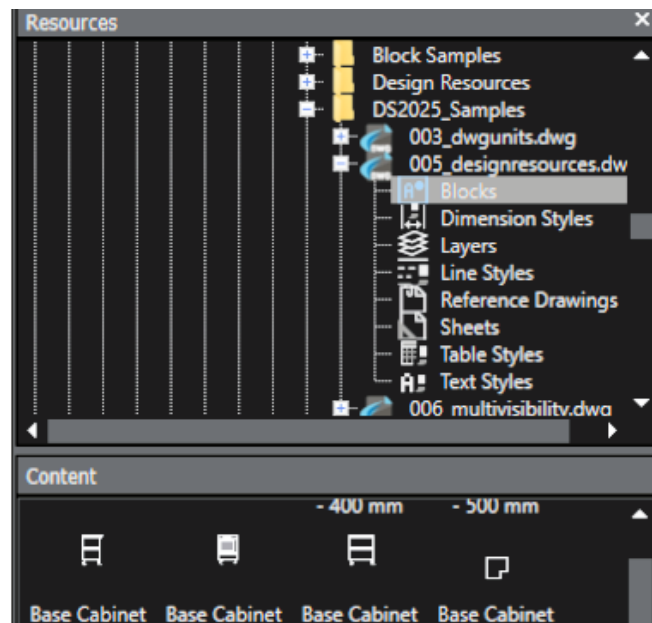


You can retain the settings for exporting to PDF and batch printing for the next session.

You can export the PDF and print batch files with the same settings. For printing batch files, you can retain the same name for PDFs and the same location of the source .dwg files, then open the PDF files after printing them.



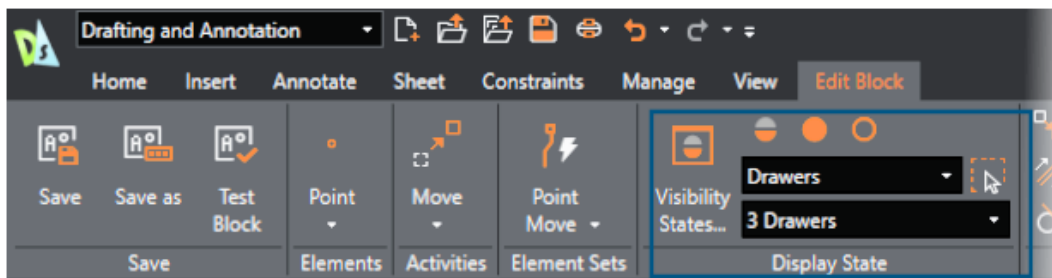
## Blocks in the Design Resource Palette



The Design Resource Palette has improved usability for blocks.

The block names of design resources are fully visible. The block thumbnails are larger so you can identify the blocks quickly.

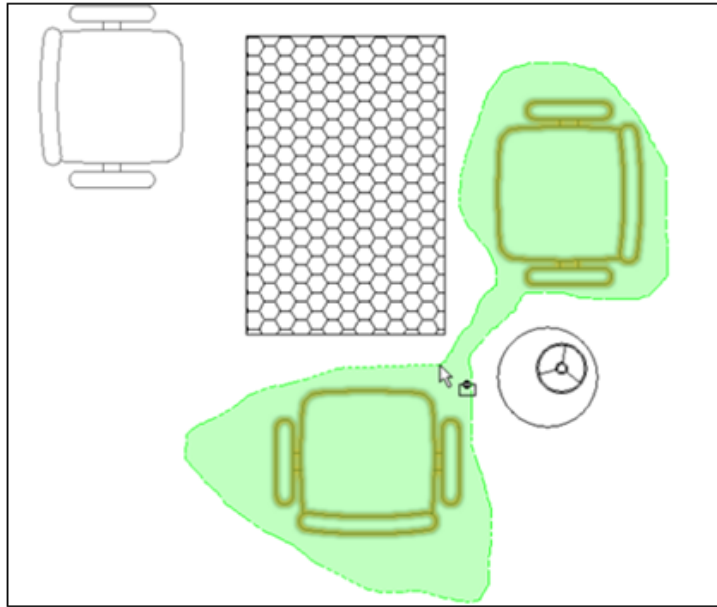
## Multiple Visibility Elements



You can use CustomBlocks to attach multiple visibility elements to a single block.

You can efficiently control the visibility of individual entities and without creating multiple visibility states. Previously, you could attach only one visibility element per block.

## Lasso Selection



You can use lasso selection to enhance efficiency and save time.

With lasso selection, you can move the pointer around an area to specify and select entities in an irregularly shaped contour. This method helps you select complex groups of entities that do not have standard rectangular boundaries. It simplifies the workflow and improves productivity.

# 26

## eDrawings

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This chapter includes the following topics:

- **Supported File Types (2025 FD04)**
- **Viewing Component References**
- **eDrawings ActiveX HTML File Format**
- **Assembly Envelopes**

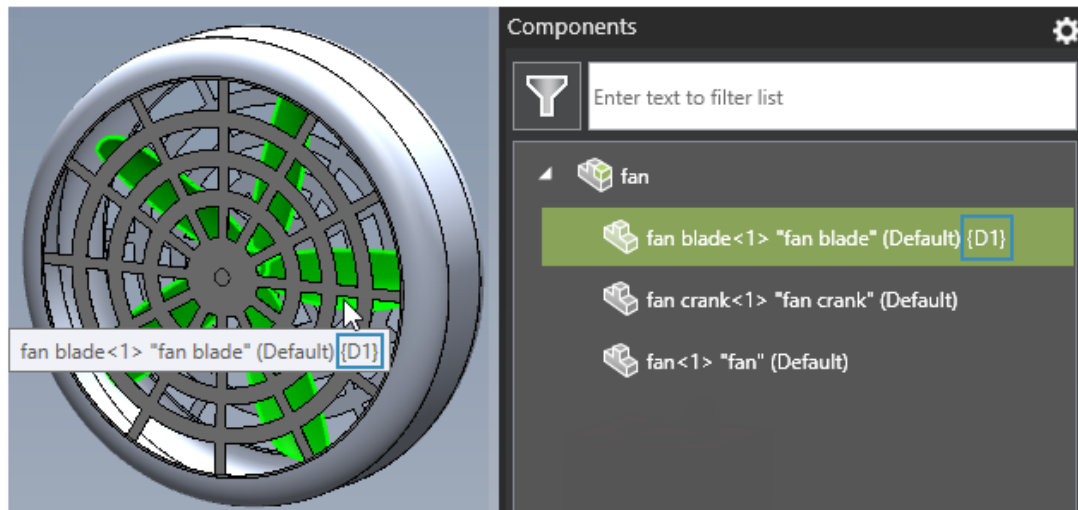
eDrawings® Professional is available in SOLIDWORKS® Professional, SOLIDWORKS Premium, and SOLIDWORKS Ultimate.

### Supported File Types (2025 FD04)

eDrawings has updated the supported versions for several file types.

Format	Version
ACIS® (.sat, .sab)	Up to 2023
Autodesk® Inventor® (.ipt, .iam)	Up to 2025
CATIA® V5 (.CATPart, .CATProduct)	Up to V5-6R2024
CATIA V6 / 3DEXPERIENCE® .3DXML	Up to V5-6R2024
Creo® - Pro/Engineer® (.ASM, .NEU, .PRT, .XAS, .XPR)	Pro/Engineer 19.0 to Creo 11.0
JT (.jt)	Up to v10.9
NX™ (Unigraphics®) (.prt)	UG11 to UG18, UG NX, NX5 to NX12, NX1847 to NX2412
Parasolid™ (.x_b, .x_t, .xmt, .xmt_txt)	Up to 37.1
Solid Edge® (.asm, .par, .pwd, .psm)	1 to 20, ST1 - ST10, 2019 to 2025
STEP (.step, .stp, .stpz)	AP 203 E1/E2, AP 214, AP 242 E1/E2/E3


## Viewing Component References



If a SOLIDWORKS or eDrawings assembly file has components with component references, you can specify an option in eDrawings to show the component references in the Components pane.

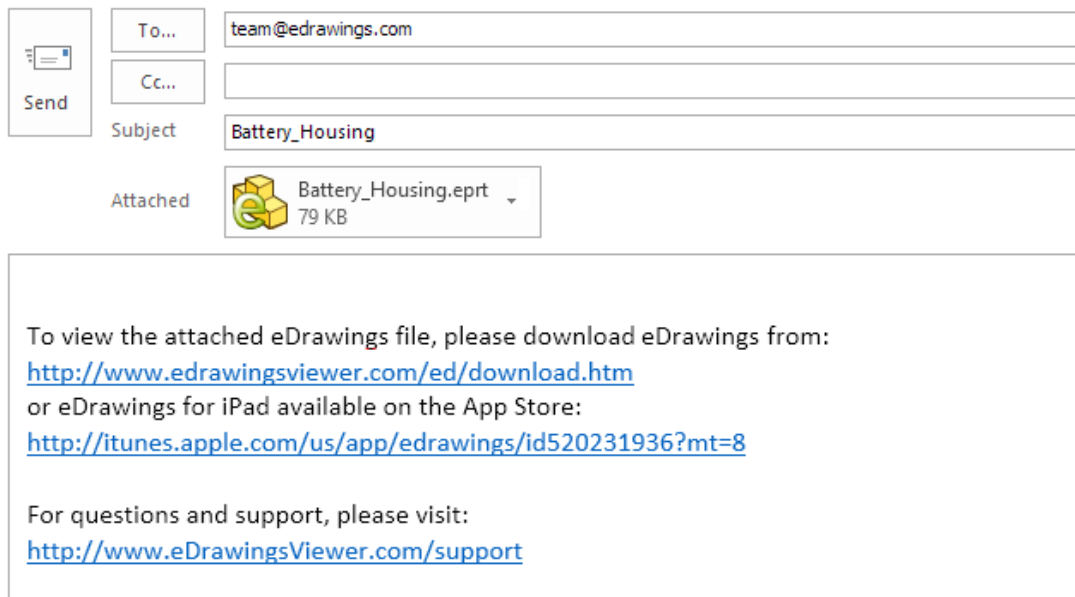
### To view component references:

1. In eDrawings, open a SOLIDWORKS or eDrawings assembly file that has component references.

2. In the Components pane, click **Options** .
3. In the dialog box, select **Show component reference**.

The component references appear in the Components pane.

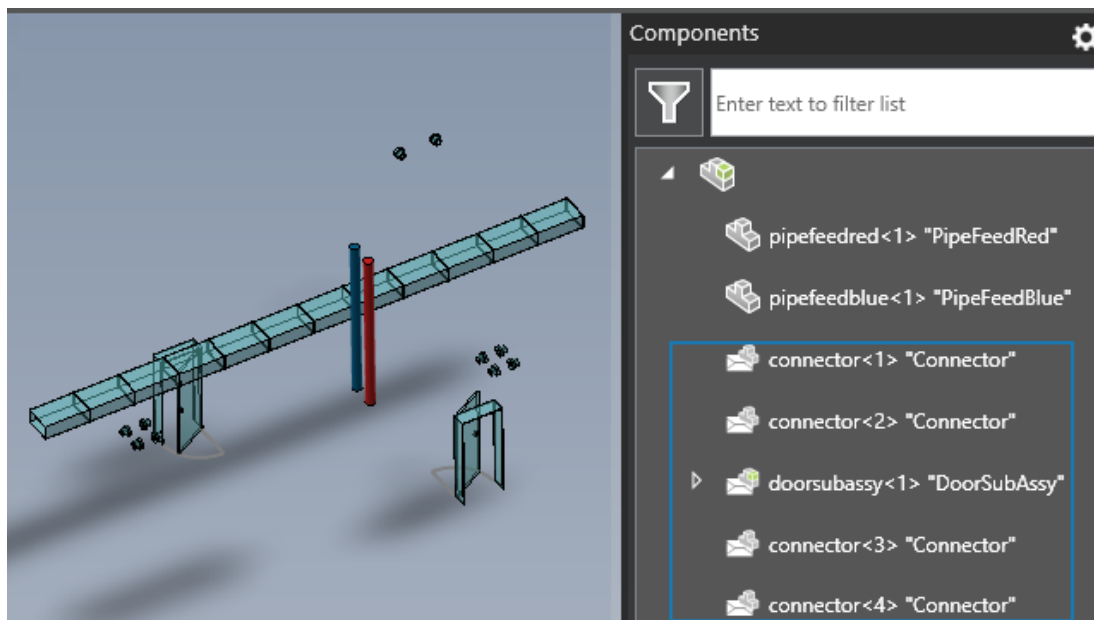
## eDrawings ActiveX HTML File Format



You can no longer save files as eDrawings ActiveX HTML files .htm files.

If you click **File > Send**, the Send As dialog box does not appear. Instead, eDrawings generates an email with the file attached as an .eprt, .easm, or .edrw file for streamlined functionality.

## Assembly Envelopes



If you open an assembly or assembly drawing that has envelopes, eDrawings displays the envelope contents with the same appearance as in SOLIDWORKS.

The Components pane displays icons that indicate envelope components.

## SOLIDWORKS Plastics

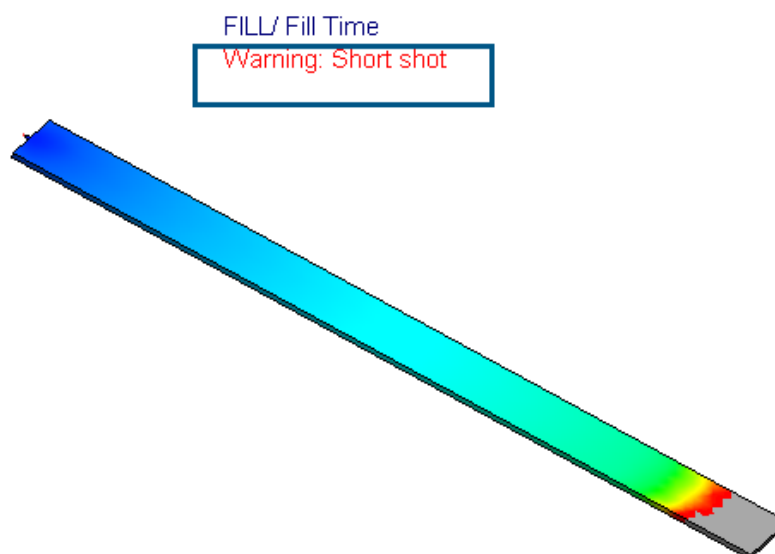
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This chapter includes the following topics:

- **Short Shot Detection (2025 SP2)**
- **Fill Analysis**
- **Improved Sink Marks Prediction**
- **Isolate the Cause of Warpage**
- **Materials Database**
- **Meshing**
- **Performance**
- **Renamed Warp Analysis Results**

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

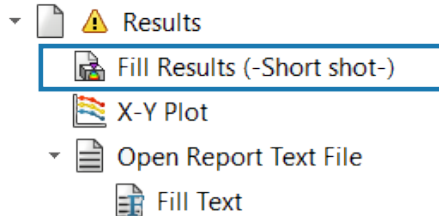
### Short Shot Detection (2025 SP2)



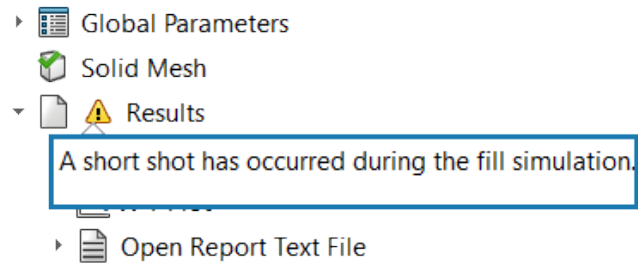
Several user interface enhancements make it easier to identify the presence of short shots for plastic injection simulations.

The following user interface enhancements help you detect the presence of short shots that might occur during filling.

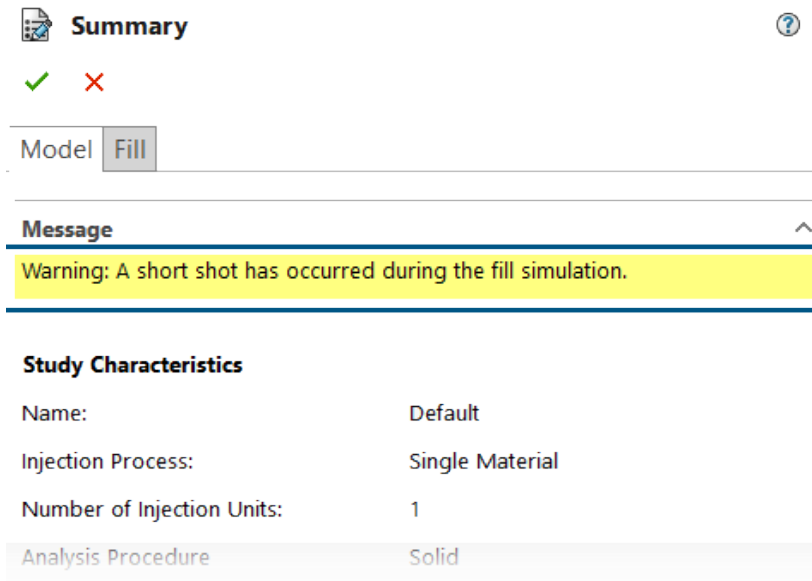
- Added the string **Warning: Short Shot** under the plot title of the **Fill Time** plot.
- Added the string **Short shot** next to the **Fill Results** node.



- Added the tooltip **A short shot has occurred during the fill simulation** under the **Results** node.

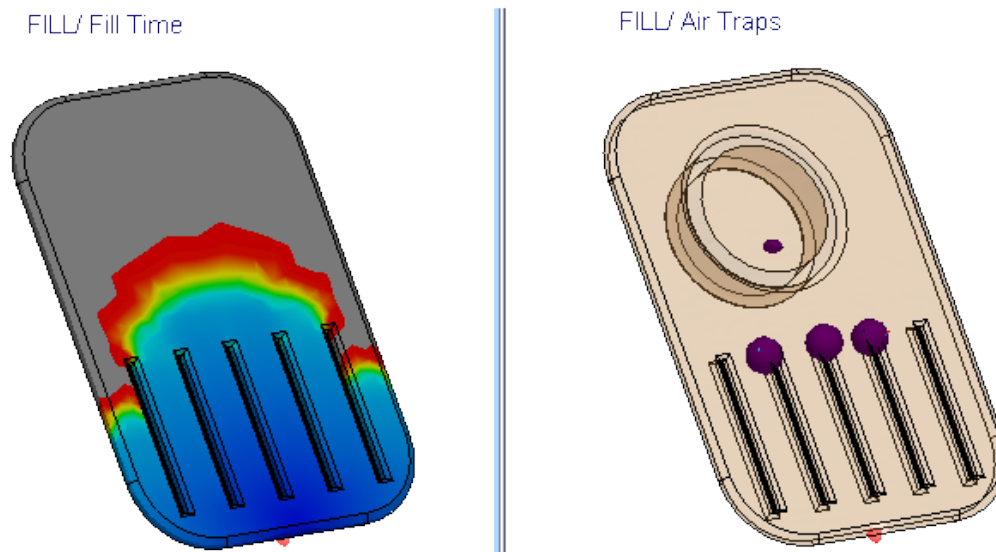


- Added a warning about the occurrence of short shots in the Summary PropertyManager.





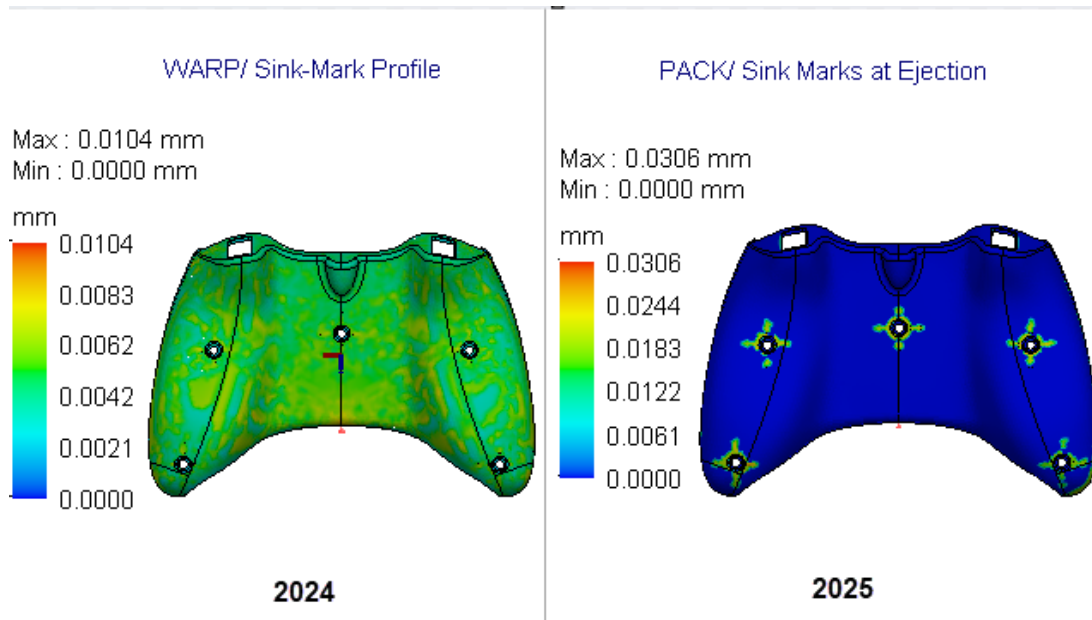
## Fill Analysis



There are several enhancements for the Fill analysis.

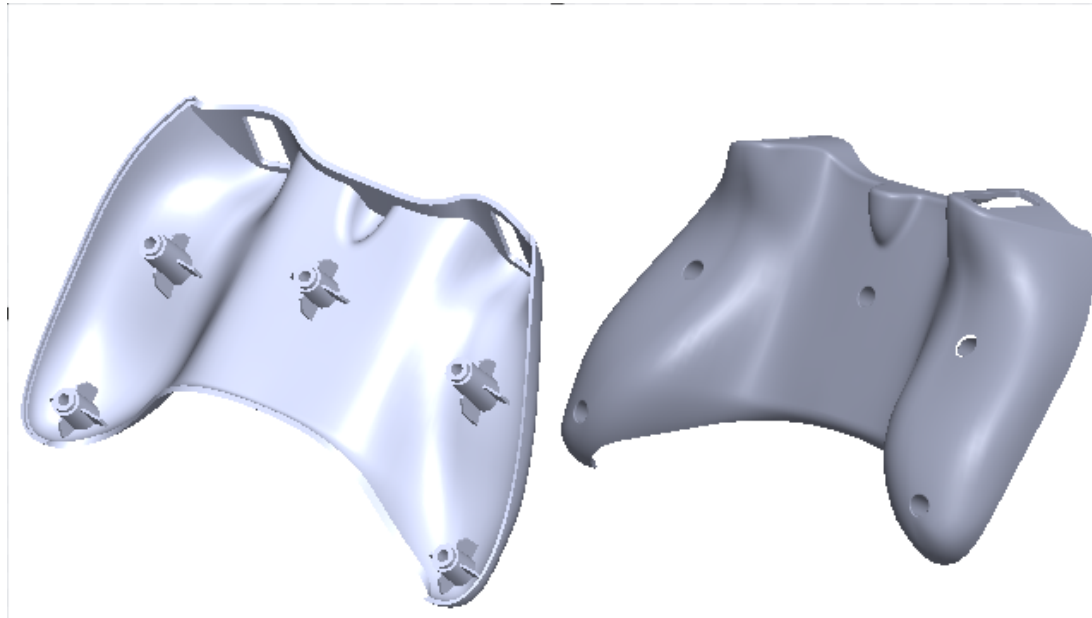
- The Fill analysis is 25% faster when using fiber-filled materials for plastic parts.
- The Fill analysis predicts weld lines and air traps even in instances of short shots. For example, the image above shows a short shot (left) and the predicted air traps (right) for a fill analysis of a part.
- Rendering of fill-time animations in isosurface mode has been significantly accelerated (up to 75%) for large models with a high number of elements. The memory required to generate the fill-time animations has also increased, as SOLIDWORKS Plastics uses all available memory resources for animation generation.
- The isosurface animation of fill-time plots saved in **AVI** format has a smoother appearance with significantly reduced lag because the delay time between successive result frames decreased.

## Improved Sink Marks Prediction



A new solver predicts with improved accuracy the location and depth of sink marks.

The new sink mark solver analyzes geometric features that are likely to induce sink marks, for example, ribs, bosses, gussets, and internal fillets. The solver then uses this geometric information to perform a localized analysis to predict the sink marks' depths. For example, the image above shows improved sink mark predictions at the surface of a game controller part that has internal boss and rib features.



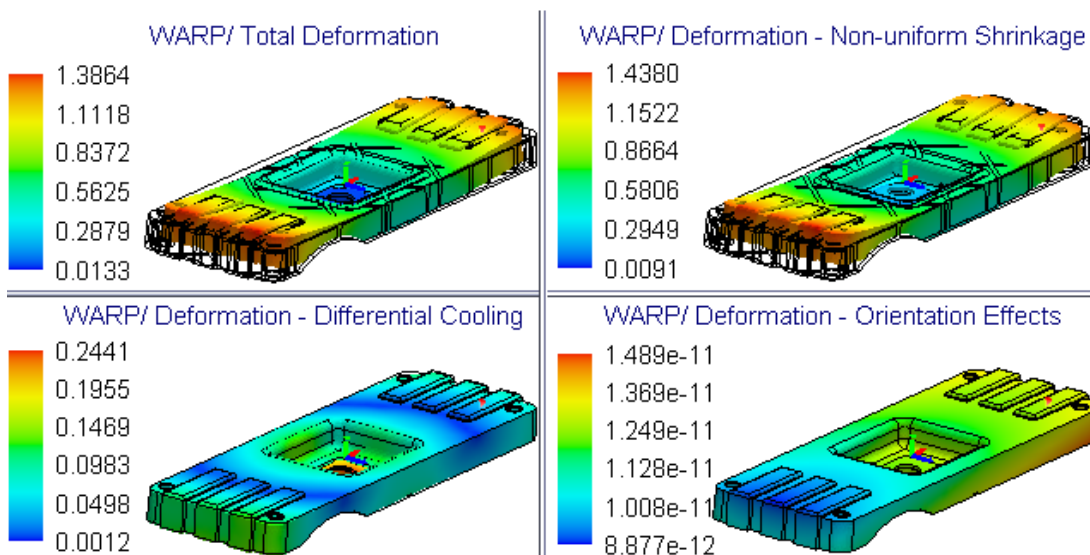
The sink mark results are updated as follows:

- The **Sink Marks** plot from the Fill results is renamed to **Sink Marks Estimate at End of Fill**.

- A new plot, **Sink Marks at Ejection**, is available with the Pack results.
- The **Sink-Mark Profile** plot from the Warp results is removed, as the prediction of sink marks based on the conditions at the end of filling is not accurate. Instead, you can refer to the **Sink Marks at Ejection** plot to review the location and depth of sink marks.

The new sink mark solver is available only for solid-hybrid and solid-hexahedral mesh procedures. The shell mesh procedure continues to use the current sink mark solver.

## Isolate the Cause of Warpage



New result plots for Warp analysis help you isolate the cause of warpage when designing plastic parts.

Warpage occurs to plastic molded parts because of three main causes: nonuniform shrinkage, differential cooling, and molecular or fiber orientation. The image shows result plots of the total deformation and the component deformation because of warpage. Understanding the dominant cause of warpage helps you make appropriate changes to the part or mold design, material, and manufacturing process to minimize design defects.

The Warp analysis in SOLIDWORKS Plastics 2025 isolates the cause of warpage by calculating, at each node, the component of total deformation attributed to each source. The following result plots are available, along with the Total Deformation plot, to assist you in identifying the cause of warpage.

Result Plot - Warp Analysis	Description
<b>Deformation – Nonuniform Shrinkage</b>	Shows the deformation that can be attributed to nonuniform mold temperatures, differential cooling rates between thin and thick sections of a part, and shrinkage variations between the

Result Plot - Warp Analysis	Description
	direction of melt flow and transverse to the direction of melt flow. (In general, these deformations occur because of nonuniform pressure, temperature, and shear stress distributions across the surface area or throughout the volume of a molded part.)
<b>Deformation – Differential Cooling</b>	Shows the deformation that can be attributed to nonuniform cooling arising from temperature variations across the injection mold's core and cavity surfaces. Nonuniform part cooling generally leads to nonuniform shrinkage and in-mold stresses, which both contribute to warpage.
<b>Deformation – Orientation Effects</b>	Shows the deformation that can be attributed to anisotropy from the orientation of fillers in the material, such as short glass fibers or carbon fibers. For materials without any fillers, this deformation is negligible.

You might notice slightly longer Warp analysis solve times because of the additional computation time required to calculate the components of the total warp deformations. The result plots that isolate the cause of warpage are available only for the **Solid Mesh** procedure.

## Materials Database

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

365 new material grades are added, 142 grades are updated, and 370 obsolete grades are removed from the database.

Manufacturer	Number of New Material Grades
DOMO®	123
Envalior™	97
SABIC Specialties®	77
Covestro®	42
MOCOM®	12
EMS-GRIVORY®	8
CHIMEI®	2

Manufacturer	Number of New Material Grades
Lehmann&Voss&Co.	2
Trinseo®	1
Solvay Specialty Polymers®	1

Manufacturer	Number of Updated Material Grades
Covestro®	37
LyondellBasell™	19
EMS-GRIVORY®	18
ARLANXEO®	14
BASELL	13
CWH, Chemwerk Huls	10
MOCOM®	9
SABIC Specialties®	7
Victrex®	6
Mueller Kunststoffe	3
Autotech-Sirmax	1
Teknor Apex®	1
TOTAL®	1
Asahi Kasei®	1
MILES	1
ENICHEM	1

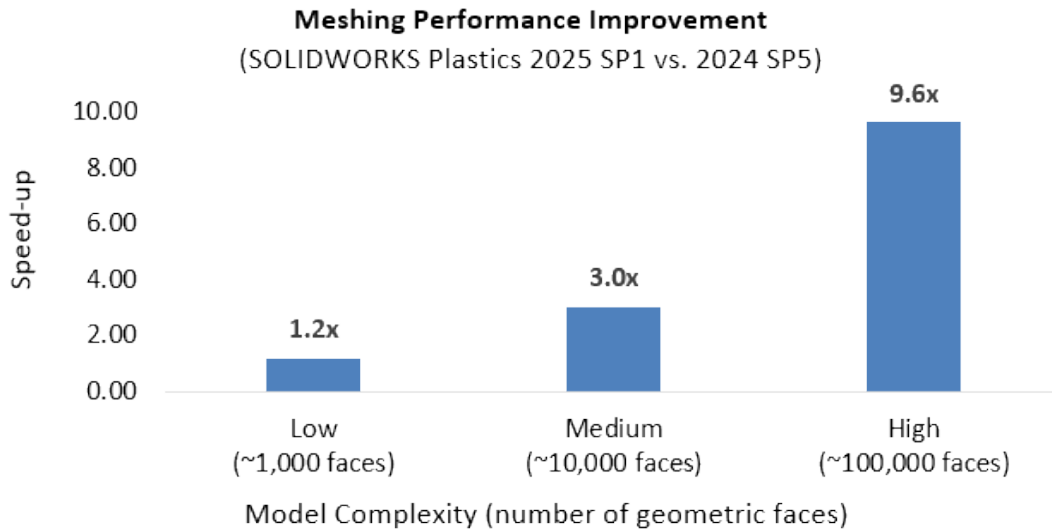
Manufacturer	Number of Removed Material Grades
DSM Engineering Plastics	151
Rhodia Engineering Plastics	94
LNP Engineering Plastics®	68

Manufacturer	Number of Removed Material Grades
Covestro®	26
Rhone-Poulenc	14
SABIC Specialties®	7
Monsanto Japan	5
Lehmann and Voss	2
Trinseo®	1
Mitsubishi Chemical Japan®	1
Mitsubishi Rayon	1

The following updates are implemented for the 2025 FD01 release.

Manufacturer	Material Grades
SABIC Specialties®	29 new grades added
SABIC Specialties®	10 grades updated
ICI	3 grades removed
Mitsubishi Chemical Japan®	1 grade removed

## Meshing

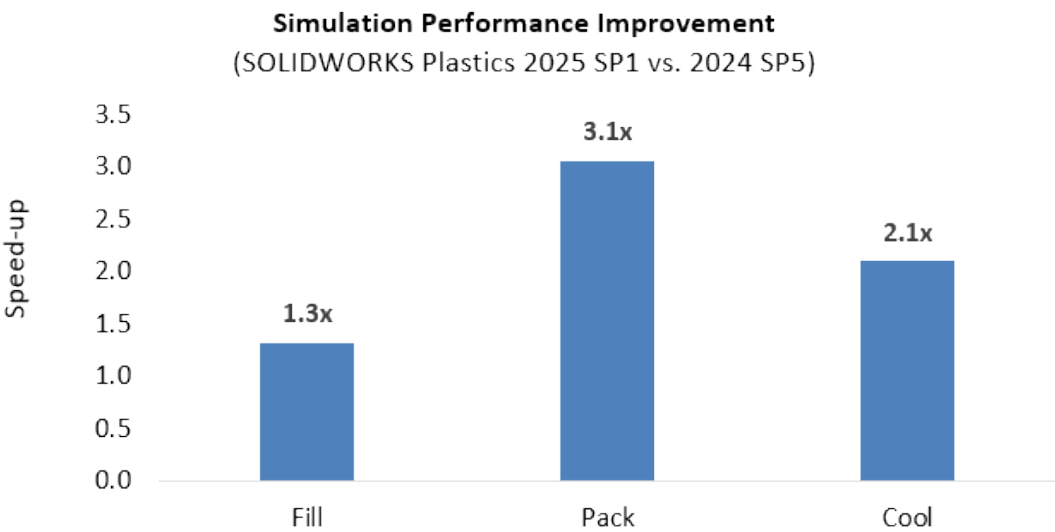


Meshing times for complex models have been significantly improved.

For meshing, a model's complexity is determined by the number of geometric faces and its curvature. Generally, models that have a higher number of faces and curvature require longer meshing times.

Highly complex models with more than 100,000 geometric faces showed the most improved meshing performance with up to 9.6 times faster meshing times. Medium-complexity models with more than 10,000 geometric faces showed up to 3 times faster meshing times, whereas simpler models with less than 1,000 faces did not show any significant meshing improvement.

Performance



Improved efficiency in solving the underlying systems of equations improves the solution times of plastics simulations without affecting robustness and accuracy.

- Up to 1.3 times faster solution for Fill simulations
- Up to 3.1 times faster solution for Pack simulations
- Up to 2.1 times faster solution for Cool simulations

Renamed Warp Analysis Results

Warp Analysis Results - 2024	Warp Analysis Results - 2025
Total Stress Displacement	Total Deformation
In-mold Residual Stress Displacement	In-mold Deformation
Quenching Thermal Stress Displacement	Quenching Thermal Deformation
Total Stress Displacement (orientation effect)	Deformation - Orientation Effects



The Warp analysis results are renamed to ensure consistent terminology.

The image shows the previous and current titles of the Warp analysis results.

# 28

## Routing

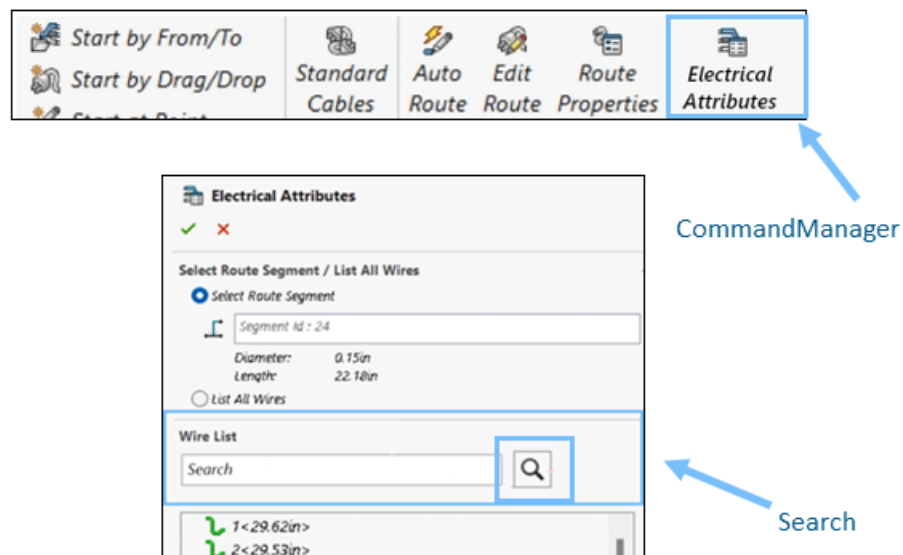
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This chapter includes the following topics:

- **Faster Access and Easier Search in Electrical Attributes (2025 SP3)**
- **BOM Entry Shows Total Cable Length across Subassemblies (2025 SP3)**
- **Splice Highlighting for Improved Visualization (2025\_SP3)**
- **Redesigned Routing Tooltips (2025 SP2)**
- **Support for Clip Assemblies and Clip Parts in the Routing Component Wizard (2025 SP2)**
- **Improving Performance in Flattened Harness Assembly Edits (2025 SP1)**
- **Create a Flattened Drawing with Cleaner Output**
- **Customizing Slack Percentages in the Route Properties and Route Segment PropertyManagers**
- **Enhancing Pipe and Tube Modifications**

Routing is available in SOLIDWORKS® Premium and SOLIDWORKS Ultimate.

### Faster Access and Easier Search in Electrical Attributes (2025 SP3)



You can find and use **Electrical Attributes** in SOLIDWORKS Routing more efficiently. Improved access and a search bar, let you quickly filter wires and cables by property values.

**Benefits:** These improvements make Electrical Attributes easier to find, faster to use, and more consistent with the rest of SOLIDWORKS Routing.

Enhancements for Electrical Attributes include:

- Quick search inside the Electrical Attributes PropertyManager

You can filter the wire list by entering into a new search bar. As you type, SOLIDWORKS Routing narrows the list to match your search based on property values. It supports partial matches and is not case-sensitive. A message appears if the search does not find any matches.

- Easy access from the CommandManager and Menu

You can open Electrical Attributes directly from the Routing tab in the CommandManager or by navigating to **Tools > Routing > Electrical > Electrical Attributes**. You can also right-click a route segment or the **Route1** feature in a subassembly to start it.

- Tooltip guidance

When you hover over the Electrical Attributes icon, a tooltip explains what you are viewing in the Electrical Attributes of the selected route segment. This information helps users understand the tool at a glance.

- Improved context awareness

When you are working in a top-level assembly with multiple subassemblies, SOLIDWORKS Routing prompts you to choose which route assembly to view Electrical Attributes for. To access the tool, you can right-click the route feature in a subassembly. The Electrical Attributes PropertyManager is accessible from both the 3D environment and flattening routes.

- Smarter Cable Search Results

When you search and the result matches a cable core, SOLIDWORKS Routing shows only the cable and its matching cores, not the entire core list.

## BOM Entry Shows Total Cable Length across Subassemblies (2025 SP3)

Before: Identical cables from different subassemblies are listed separately in lines 2 and 3.

	A	B	C	D	E
1		BOM Table			
2		ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
3		1	6 Way M APEX_33341373	6 Way M APEX 1.5 HETS 6.3 Sealed Connector Black	6
4		2	Scipio Electric - ATX_PSU_06P BK	Cable for 6 pin ATX motherboard power connector	1
5		3	Scipio Electric - ATX_PSU_06P BK	Cable for 6 pin ATX motherboard power connector	1

After: Identical cables are grouped into line 2, showing the total length across subassemblies.

	A	B	C	D	E
1		BOM Table			
2		ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
3		1	6 Way M APEX_33341373	6 Way M APEX 1.5 HETS 6.3 Sealed Connector Black	6
4		2	Scipio Electric - ATX_PSU_06P BK	Cable for 6 pin ATX motherboard power connector	2066.93 mm

You can display the total length and mass of identical cables used across multiple routing subassemblies as a single line in a Bill of Materials (BOM).

**Benefits:** In large assemblies with multiple harness subassemblies, identical cables often appear separately, making the BOM harder to read and analyze. With this enhancement, SOLIDWORKS Routing automatically detects identical cables across subassemblies and combines their total length and mass into one row.

This behavior applies to the following BOM types:

- **Parts Only BOM**

Cables that were previously listed separately appear as a combined entry with total values.

- **Indented BOM**

You can select the **Combine Identical Components** option to merge cable entries from other route subassemblies. The selected row remains, and the total length and mass display.

- **Flattened BOM**

Identical cables from multiple subassemblies merge into a single row with total values.

Top-Level BOMs are not affected by this change.

An example workflow is:

1. Create an assembly with multiple cabling subassemblies using the same cable.
2. Insert a BOM table and select a BOM type, such as **Parts Only**, **Indented**, or **Flattened**.

## Splice Highlighting for Improved Visualization (2025\_SP3)

SOLIDWORKS Routing helps you better understand splices in electrical routes by adding automatic visual highlights for related connectors and route segments.

**Benefits:** You can find and understand splices without searching through the design. It is also easier to follow wire paths and check connections, helping reduce design errors.

The changes in visual cues include:

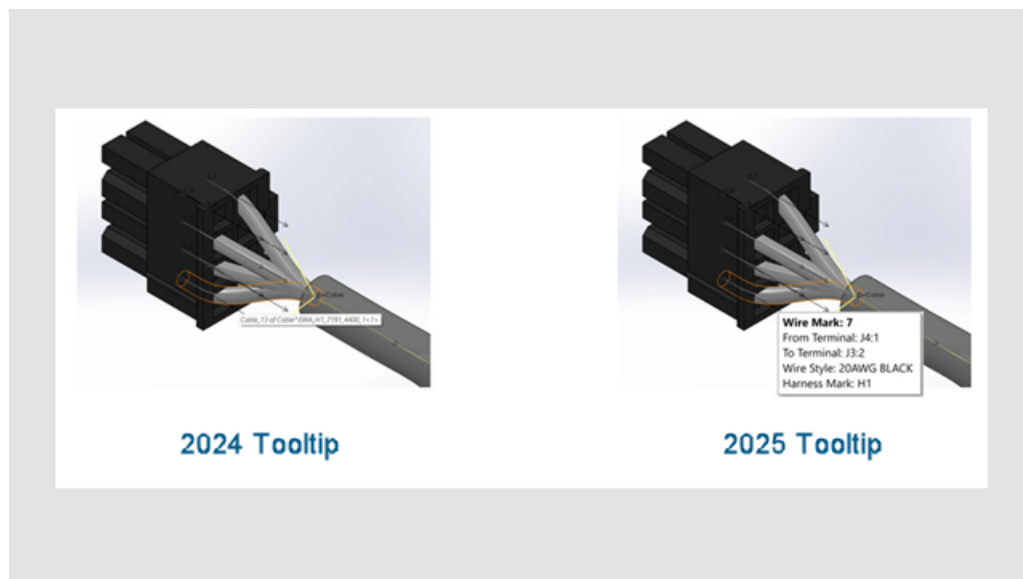
- Highlighted End Connectors

When you select or add a splice, the connectors linked to the spliced wires highlight automatically. A unique color helps you tell them apart from other connectors in the design.

- Highlighted Route Segments

The route segment connected to the splice appears in a different color or style. This distinction helps you see how the splice fits into the overall wiring path.

## Redesigned Routing Tooltips (2025 SP2)



The tooltips in the SOLIDWORKS Routing interface are redesigned to improve clarity and usability. When you hover over a wire, cable, or harness, the updated tooltips display key details in a logical order.

**Benefits:** This update makes it easier to quickly interpret connection information.

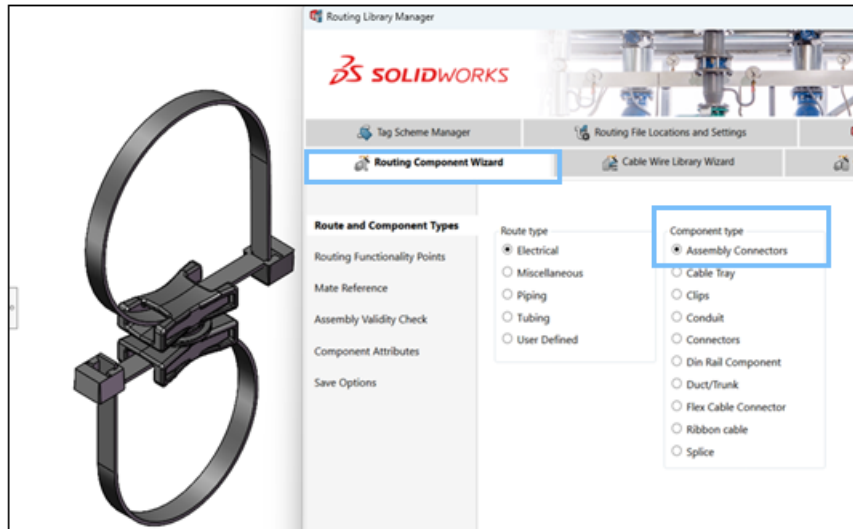
For wires, cables, and harnesses, the tooltips display:

- Wire/Cable Mark
- From Terminal
- To Terminal
- Wire Style or Cable Core

- Harness Mark (if applicable)

For harness bundles with multiple wires and cables, the tooltip provides a concise summary of key attributes.

## Support for Clip Assemblies and Clip Parts in the Routing Component Wizard (2025 SP2)



The Routing Component Wizard supports both clip assemblies and clip parts, allowing you to define and configure assemblies as routing components. The basic requirements for both types are the same and the steps to create them in the Routing Library Manager follow the same process.

**Benefits:** This update provides better flexibility in designing and integrating complex clips into routing workflows.

The enhancements in the Routing Library Manager are:

- **Clip Assemblies:** Users can select and configure an assembly file (.SLDASM) as a routing clip.
- **Routing Points:** Users can define routing points to align wires, cables, or hoses.
- **Seamless Integration:** Clip assemblies work with existing routing workflows and are stored in the Routing Library.

Follow these steps to define a clip assembly in the Routing Library Manager:

1. Select the clip assembly.
  - a. Open the Routing Library Manager and navigate to the Routing Component Wizard.
  - b. Choose the **Route Type** and **Component Type**, then click **Next**.
2. Add route points (**ARPoints**) to the clip assembly to define wire, cable, or hose alignment.

**Note:** Connection points (**CPoints**) are disabled. They are not required for clips.

3. Add routing geometry.
  - a. Define the **Clip Axis** to specify the routing direction.
  - b. If the clip assembly requires rotational placement, add the **Axis of Rotation**.
4. Add **Mate References** to define the proper alignment of the clip assembly.
5. Validate the clip assembly and confirm that it meets routing requirements.
6. Configure the **Design Table**.
  - a. If the clip assembly has multiple configurations, open the existing Design Table to make adjustments.
  - b. If none exists, create a new Design Table.
  - c. Validate standard and custom entries in the table using an embedded Excel worksheet.
7. Verify **Component Attributes**. Modify component attributes as required.
8. Save the clip assembly.
  - a. Save the configured clip assembly to the Routing Library.
  - b. Specify a library folder location and filename.
  - c. Save the component as an **.XML** file.

## Improving Performance in Flattened Harness Assembly Edits (2025 SP1)

The editing tools in the Edit Flattened Route PropertyManager perform faster, enhancing your experience for editing flattened harness configurations.

You can make multiple edits and preview them as temporary changes before finalizing, giving you more control over the design process.

While you edit, SOLIDWORKS Routing temporarily pauses updates to flattened features. Updates only occur when you confirm or cancel, ensuring efficient resource use and a smoother workflow.

For example, after you finish editing, SOLIDWORKS Routing prompts you to confirm. Clicking **OK** in the PropertyManager applies the updates to the flattened features, saving resources by preventing repeated updates with each change. Clicking **Cancel** removes the temporary changes.

Previously, each edit triggered a full update, slowing down your workflow. With this enhancement, only temporary graphics display with each change, without updating the underlying flattened features.

This functionality does not apply to annotation flatten route edits, flatten routes with discrete wires, and flatten routes with **Maintain 3D Orientation** segments.

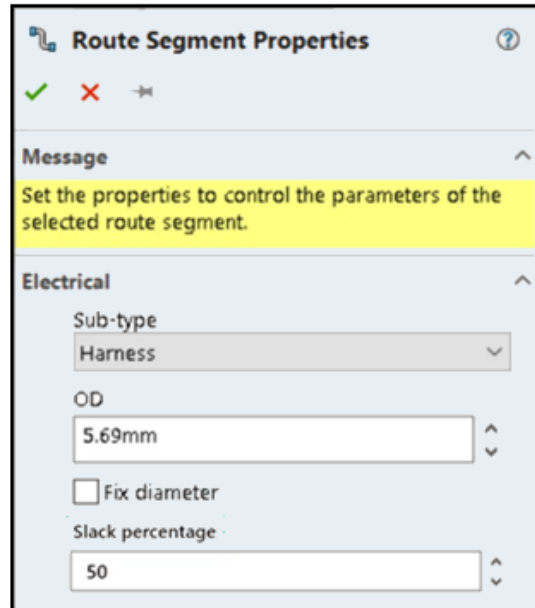
## Create a Flattened Drawing with Cleaner Output

The following updates in flattened drawings provide cleaner output and improved workflow:

- **Show/Hide Flatten Route Items:** An option to show/hide **Leader** lines in connector tables.
- **Quantity Display in Balloons:** Display quantity in connector balloons, similar to wire balloons.

- **Column and Row Formatting:** Prompt users to apply formatting changes for columns and rows in other tables.
- **Table Updates:** Prompt users to apply updates to all tables in the drawing.
- **Formboard Frame Visibility:** In the Flattened Items PropertyManager, an option to **show/hide** the formboard frame.

## Customizing Slack Percentages in the Route Properties and Route Segment PropertyManagers

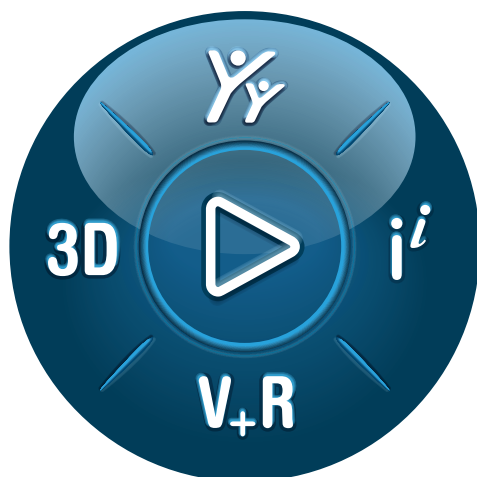


In the Route Properties and the Route Segment Properties PropertyManagers, you can define a custom value for the **Slack Percentage** for individual route segments. This value overrides the slack percentage specified in **Tools > Options > System Options > Routing**.

## Enhancing Pipe and Tube Modifications

When you edit a route assembly containing pipes and tubes, the SOLIDWORKS Routing software modifies existing components instead of creating new virtual components.





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