WHAT'S NEW

SOLIDWORKS 2025

Contents

1

Welcome to SOLIDWORKS 2025

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 **3D**EXPERIENCE® platform

This document covers all enhancements that affect how you interact with the **3D**EXPERIENCE platform. This includes both of the platform-connected versions of SOLIDWORKS - SOLIDWORKS Connected and SOLIDWORKS with the **3D**EXPERIENCE (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	• **Specifying a Z-Up Template** on page 25
Parts and Features	• **Defeature Silhouette Method for Parts** on page 43 • **Patterning Reference Geometry** on page 44 • **Repairing Dangling Relations** on page 39
Assemblies	• **Assembly Visualization** on page 65 • **SpeedPak Instances** on page 69 • **Interference Detection in Large Design Review Mode** on page 70
SOLIDWORKS MBD	• **Creating DimXpert Dimensions from Sketch Dimensions** on page 162

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 is improved when rendering for smooth zooming, panning, and rotating of complex sketches.

For More Information

Use the following resources to learn about SOLIDWORKS:

What's New in PDF and HTML

This guide is available in PDF and HTML formats. Click:

- > **What's New** > **PDF**
- > **What's New** > **HTML**

Interactive What's New

In SOLIDWORKS, appears next to new menu items and the titles of new or significantly changed PropertyManagers. Click to display the topic in this guide that describes the enhancement.

To enable Interactive What's New, click > **What's New** > **Interactive**.

Online Help

Contains complete coverage of our products, including details about the user interface and examples.

SOLIDWORKS User Forum	Contains posts from the SOLIDWORKS user community on the **3D**EXPERIENCE® platform (login required).
Release Notes	Provides information about late changes to our products, including changes to the *What's New* book, online help, and other documentation.
Legal Notices	SOLIDWORKS Legal Notices are available **online**.

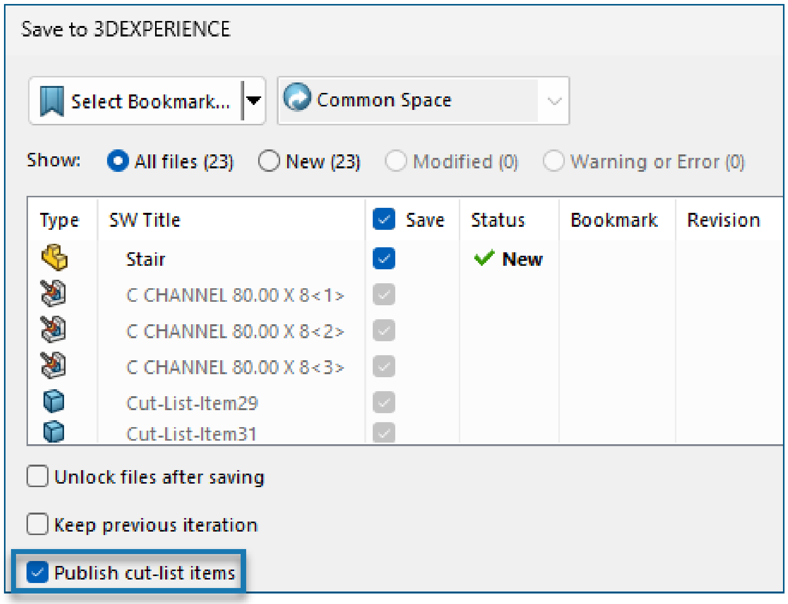
2

Using SOLIDWORKS on the 3DEXPERIENCE Platform

This chapter includes the following topics:

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

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



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

To publish the cut list items, save the SOLIDWORKS part as a weldment part to the **3D**EXPERIENCE 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 have already saved the part on the **3D**EXPERIENCE platform.
- The part must contain a weldment feature.
- The part must be a Single Physical Product.

Prerequisites to publish cut list items on the **3D**EXPERIENCE 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 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 **3D**EXPERIENCE platform.

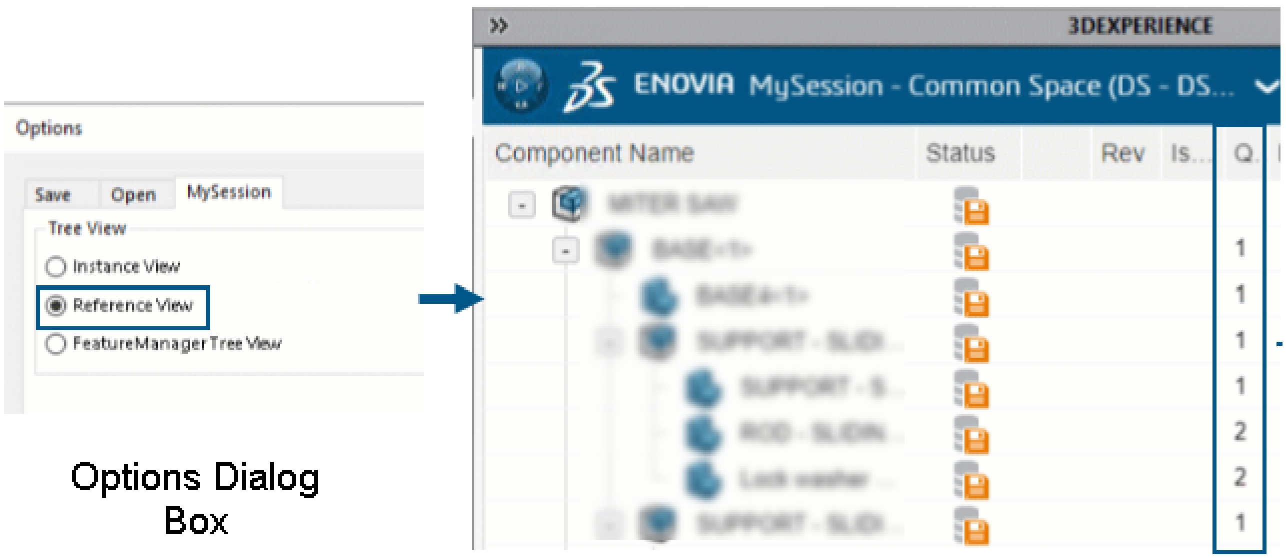
Removal of Option to Generate 3D Format

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.

Visibility of Quantity Column



Options Dialog Box

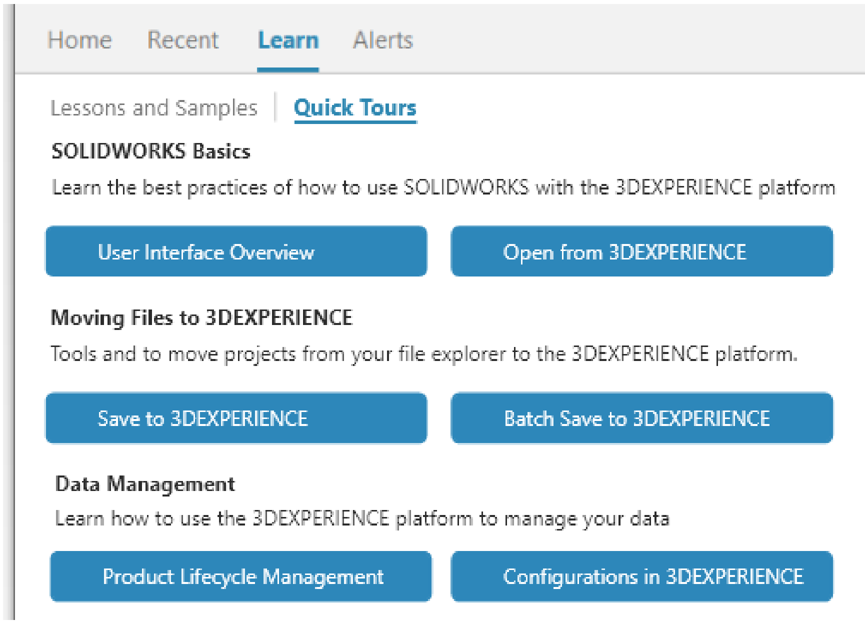
Quantity column in MySession

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.

Quick Tours



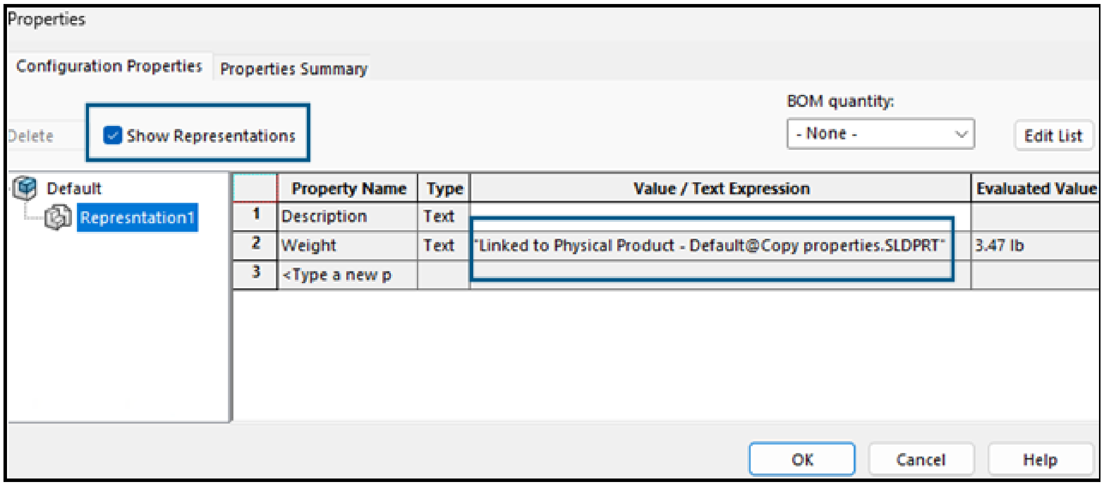
3DEXPERIENCE users can follow compact, integrated learning modules called Quick Tours. Each Quick Tour has a sequence of steps shown as interactive popups that point to elements in the user interface.

Benefits: You can interactively learn the **3D**EXPERIENCE 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.

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 **3D**EXPERIENCE platform and saved:

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

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.

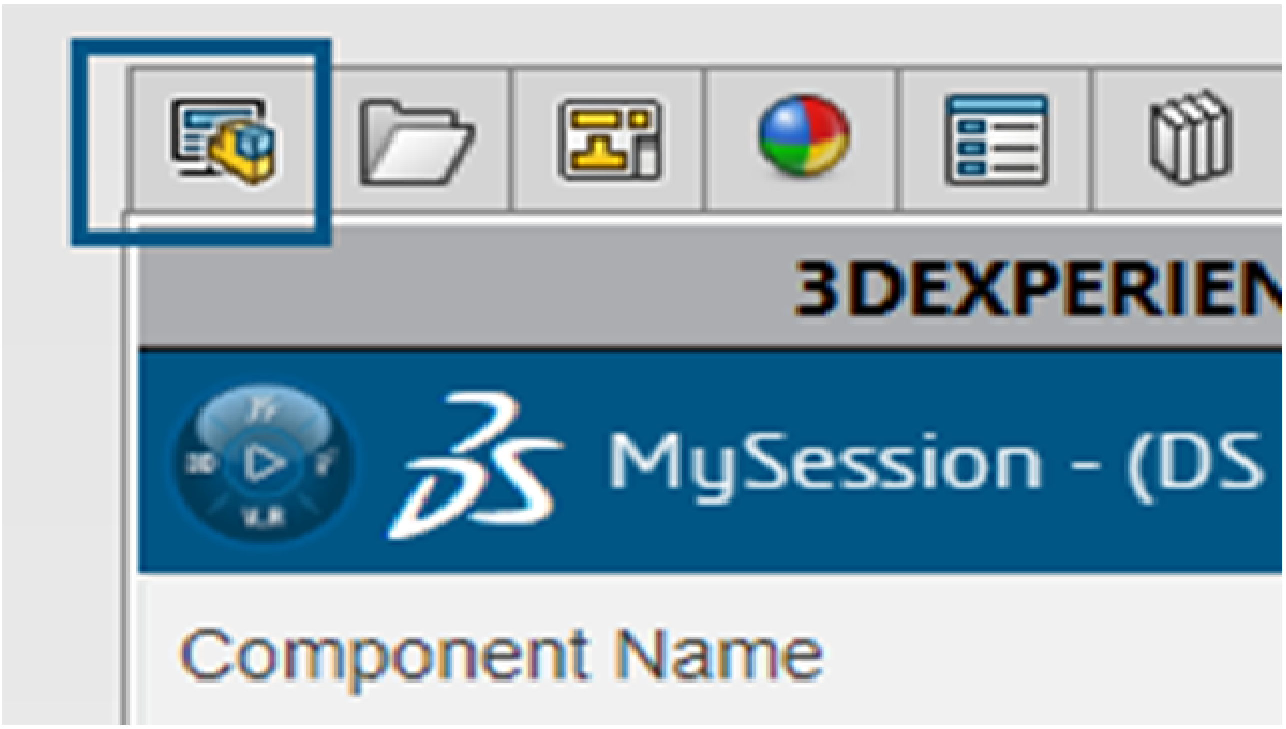
When installing SOLIDWORKS Connected, optionally select an add-in and enter your serial number. If you are using a network license, you must specify the address (`port@server`) of your SolidNetWork (SNL) License server.

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 **3D**EXPERIENCE Compass. The stand-alone app also supports the same activation methods and SolidNetWork (SNL) licensing.

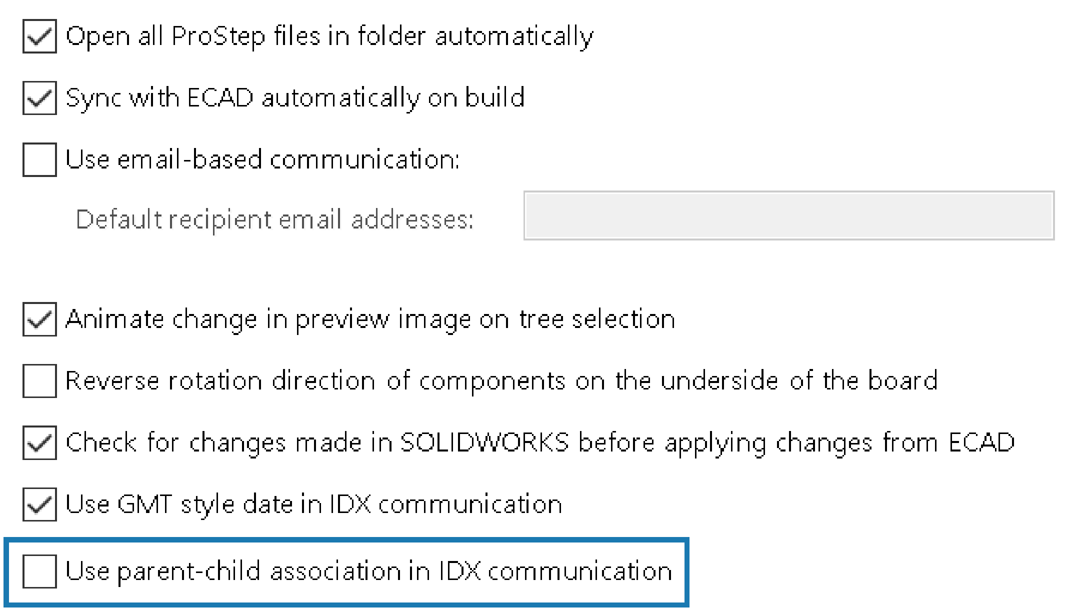
Task Pane



Enhancements to the user interface help improve productivity.

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

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



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.**

3

Installation

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.

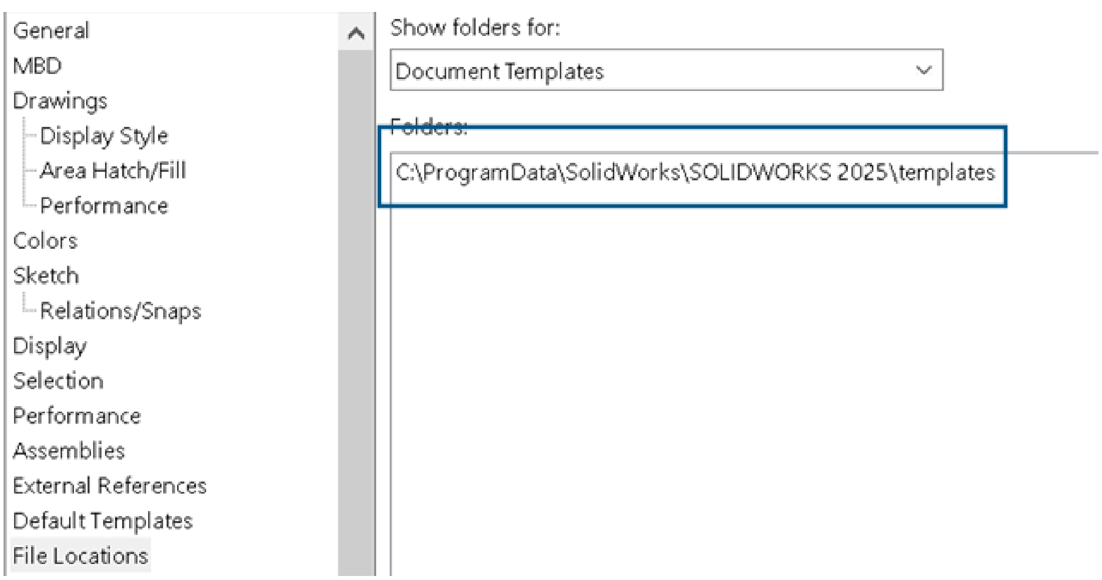
In addition, in the SOLIDWORKS Installation Manager, you can install the Manage Web API on the SOLIDWORKS Manage Server page and specify the Http port there as well.

4

Administration

This chapter includes the following topics:

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 **3D**EXPERIENCE Marketplace and **3D**EXPERIENCE apps.

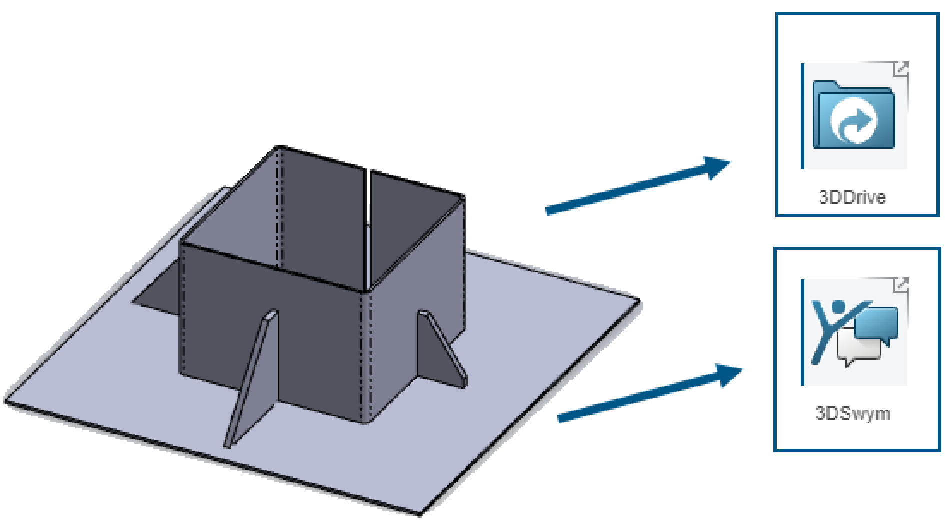
When installing an administrative image using the command line or through Microsoft Active Directory, you must include the SOLIDWORKS Login Manager file in the image. For example: `administrative_image_directory\swloginmgr\SOLIDWORKS Login Manager.msi`.

5

SOLIDWORKS Fundamentals

This chapter includes the following topics:

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 **3D**EXPERIENCE 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
Recognized mesh face **Unrecognized mesh face**	Specifies the colors shown for the mesh faces when you use the **Insert** > **Mesh** > **Segment Imported Mesh Body** or **Convert Mesh to Standard** tool. See **Colors** > **Color scheme settings**.	**Colors**
Use Property Set mapping file	Maps custom properties to IFC™ property sets. See **Export** > **File Format: IFC** > **Output as**.	**Export**
File Locations	The logic for inheriting file locations from previous installations has improved. See **Inheriting Default File Locations When Upgrading to SOLIDWORKS 2025** on page 20	**Installation**
Zoom to fit on open	When you open a drawing, you have the option to have it automatically zoom to fit your graphics area.	**Drawings**

Document Properties

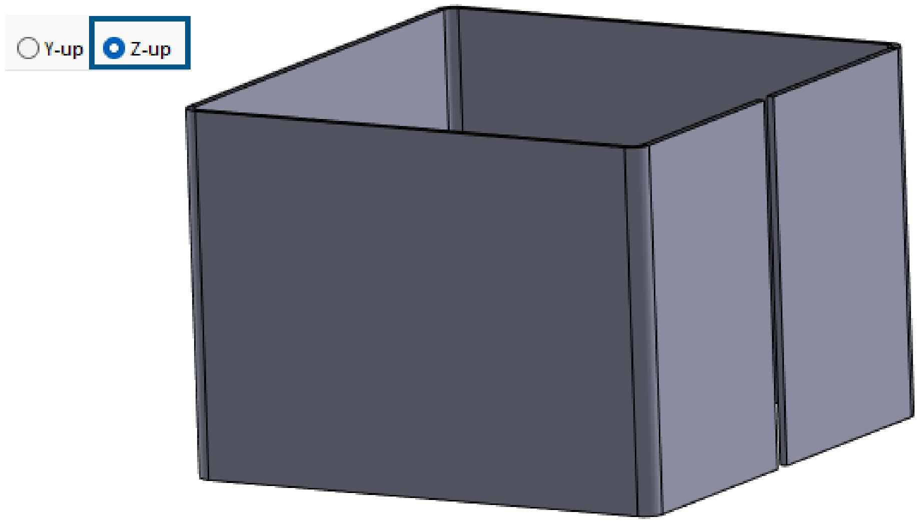
Option	Description	Access
Automatically add Flange Length dimension to flange profiles	SOLIDWORKS® automatically adds length dimensions to all edge flange profiles, where the sketch dimension (not the feature dimension) controls the flange length.	**Sheet Metal**
Surface symbol standard	Select a standard: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

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

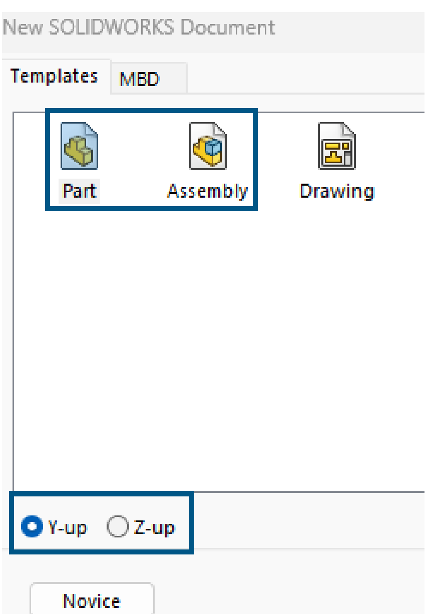


When you create a part or assembly, you can choose a template for Z-up orientation.

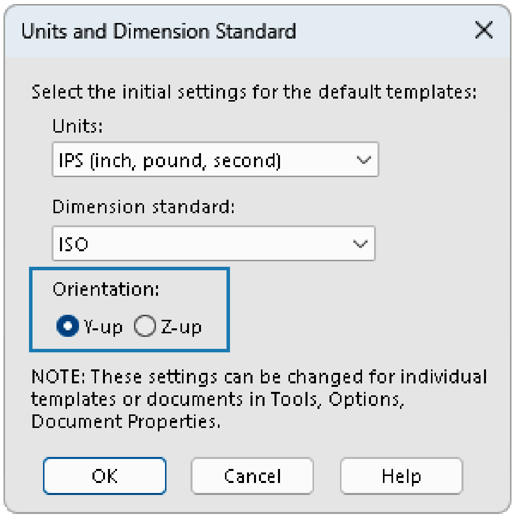
When you create a part, you can choose Y-up or Z-up and build on the template. In earlier releases, SOLIDWORKS had a default Y-up orientation only.

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

You can specify a SOLIDWORKS default Z-up template when you create a new SOLIDWORKS document.



After a new installation, you can specify the default orientation in the Units and Dimension Standard dialog box.



To specify a Z-up template when you create a new SOLIDWORKS document:

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

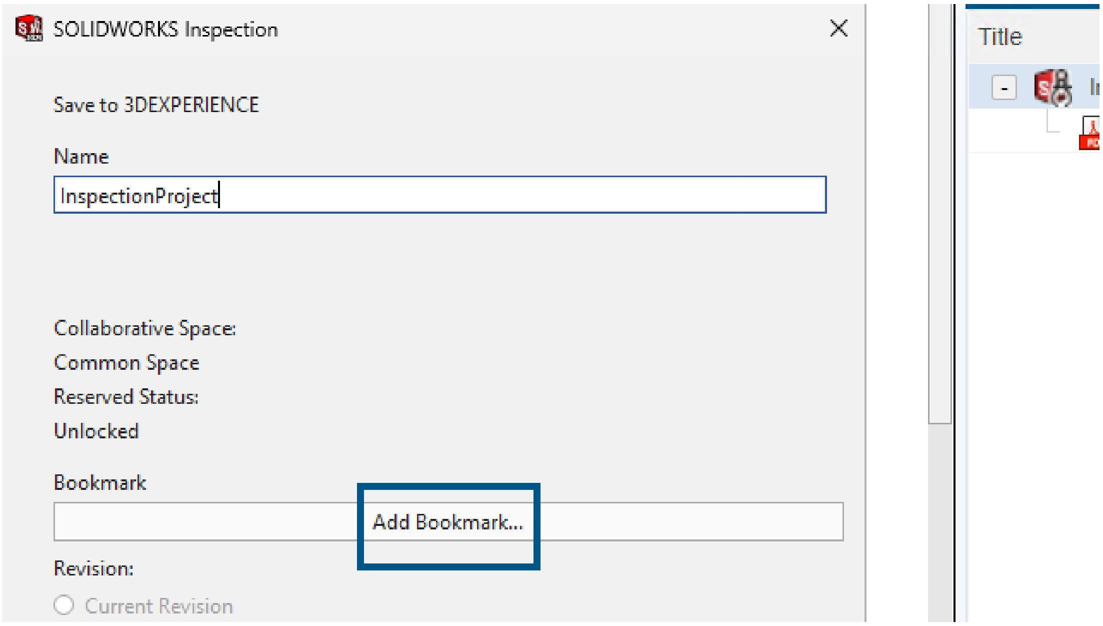
   c. Click **OK**.

To specify a Z-up template in the Units and Dimension Standard dialog box:

1. Access the Units and Dimension Standard dialog box:
   a. In the lower-right corner of the graphics window, in the task bar, click **IPS**.
   b. Specify units:
      - **MKS (meter, kilogram, second)**
      - **CGS (centimeter, gram, second)**
      - **MMG (millimeter, gram, second)**
      - **IPS (inch, pound, second)**

   c. Click **Edit Documents Units...**.
2. In the Units and Dimension dialog box, under **Orientation**, specify an option:
   - **Y-up**. The Y-axis points upward.
   - **Z-up**. The Z-axis points upward.
3. Click **OK**.

Saving SOLIDWORKS Inspection Files Using Bookmarks



You can save SOLIDWORKS Inspection files to the **3D**EXPERIENCE® 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 **3D**EXPERIENCE platform, click **Save**.

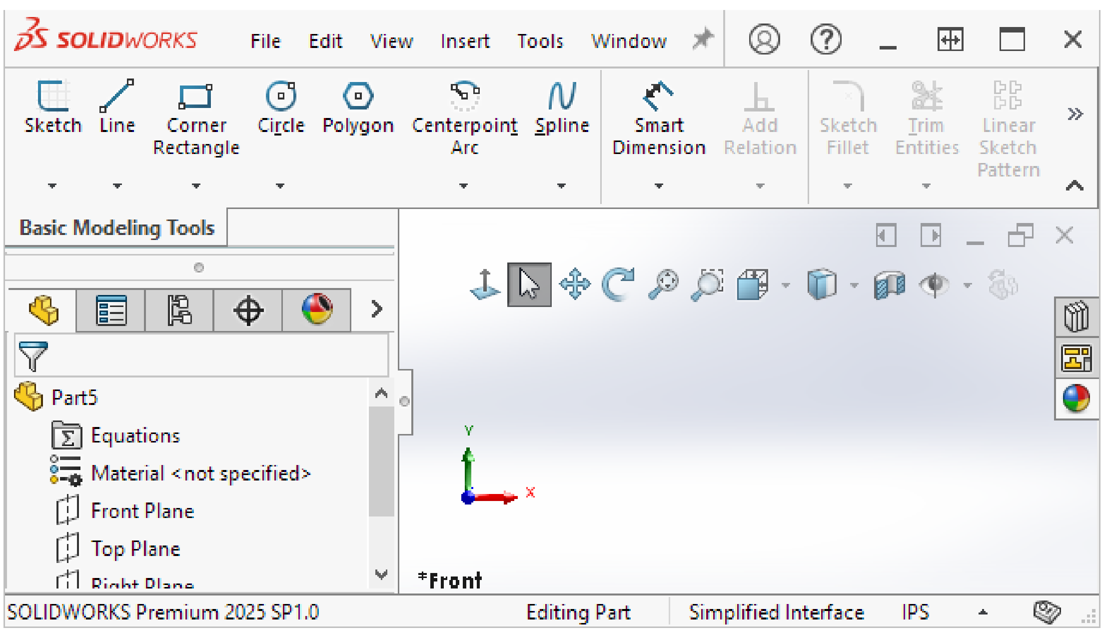
6

User Interface

This chapter includes the following topics:

- **Simplified Interface (2025 SP1)**
- **Command Predictor**
- **Reorganize Components**
- **Usability**
- **Hole Wizard**
- **Save and Auto Save Progress**
- **Create Document Group**

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

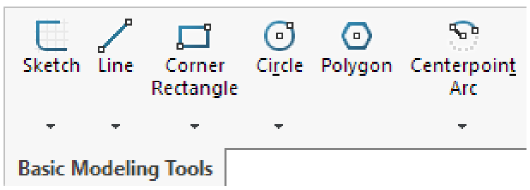
For parts and assemblies, the Heads-up View toolbar contains tools to manipulate views. It does not include appearances, scenes, or view settings.

Simplified interface

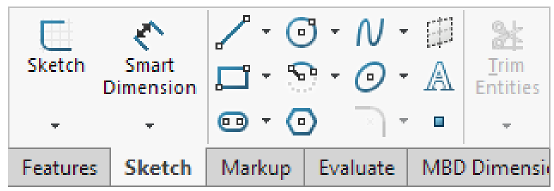
Default interface

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.



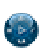
Simplified interface

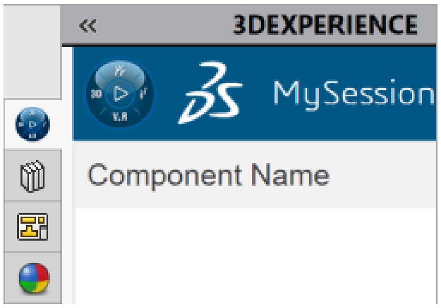


Default interface

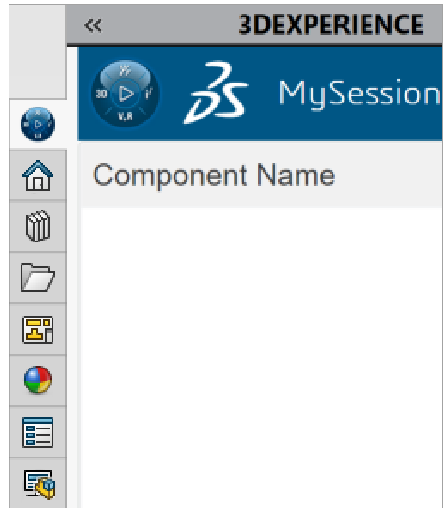
Task Pane

The Task Pane contains the following tabs:

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



Simplified interface



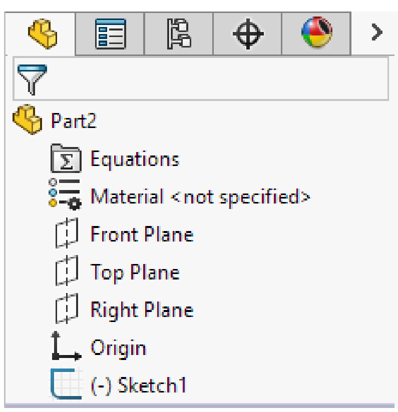
Default interface

You can specify tabs to include. On a Task Pane tab, right-click and select **Customize**.

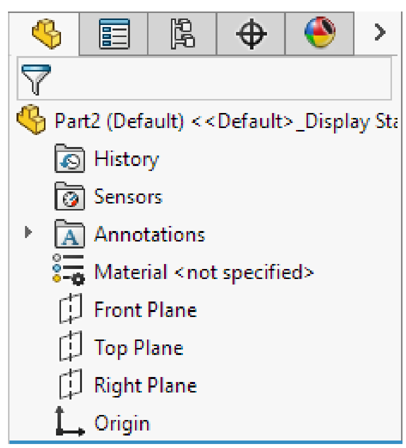
FeatureManager Design Tree

The FeatureManager design tree contains the following items:

- Equations
- Material
- Planes
- Origin
- Sketch



Simplified interface



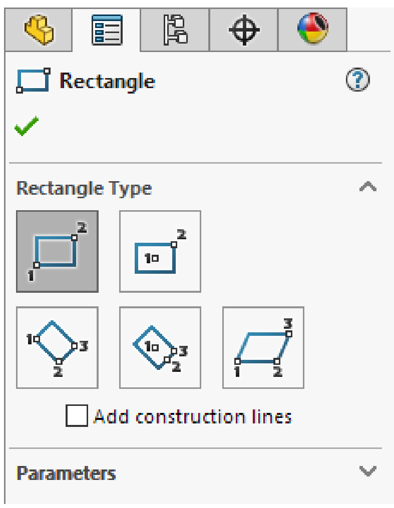
Default interface

You can specify items to include in **Tools** > **Options** > **System Options** > **FeatureManager** under **Hide/show tree items**.

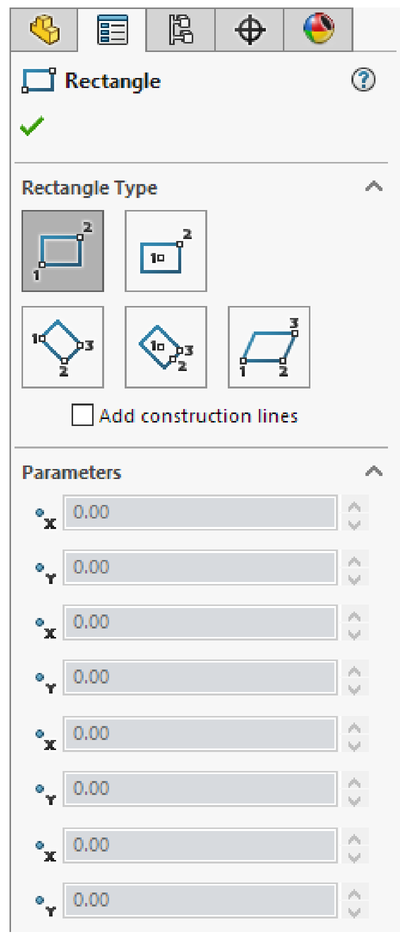
Items in the FeatureManager design tree do not include configuration or display state names if only one exists. You can turn this option on or off. In the FeatureManager design tree, right-click a part or assembly name and click **Tree Display** > **Component Name and Description**. In the dialog box, clear **Do not show Configuration or Display State name if only one exists**.

PropertyManagers

Several PropertyManagers contain collapsed sections:



Simplified interface



Default interface

Sketch Mode

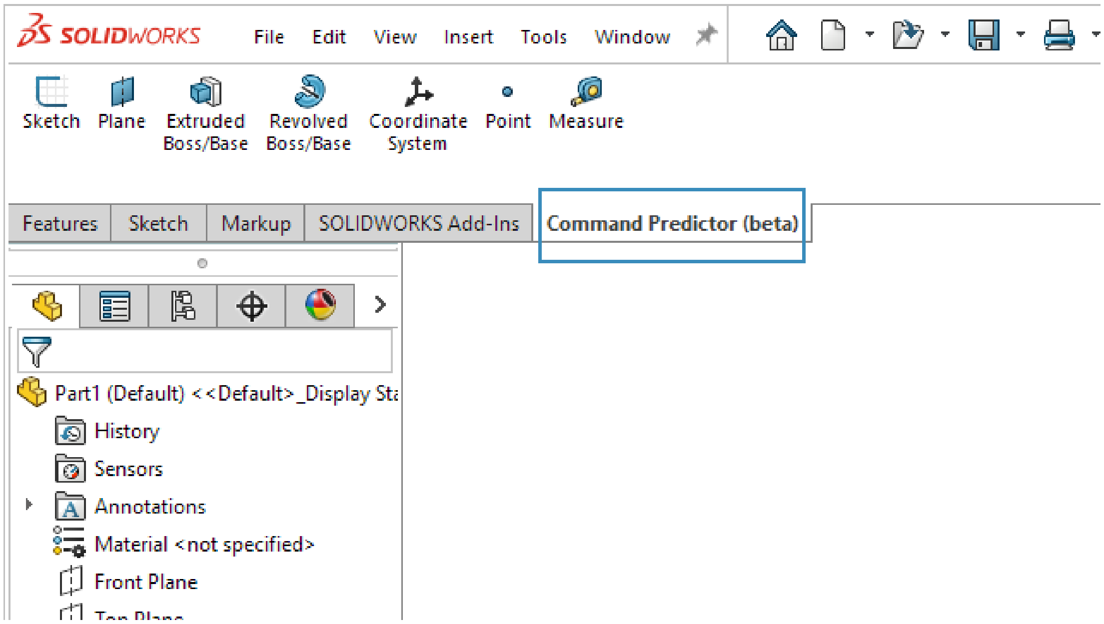
For parts, the **Simplified Interface** opens a new part with an active sketch on the Front plane.

You can turn this option on or off in **Tools** > **Options** > **System Options** > **Sketch** and clear **Create sketch on new part**.

MotionManager Tree

The MotionManager 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.

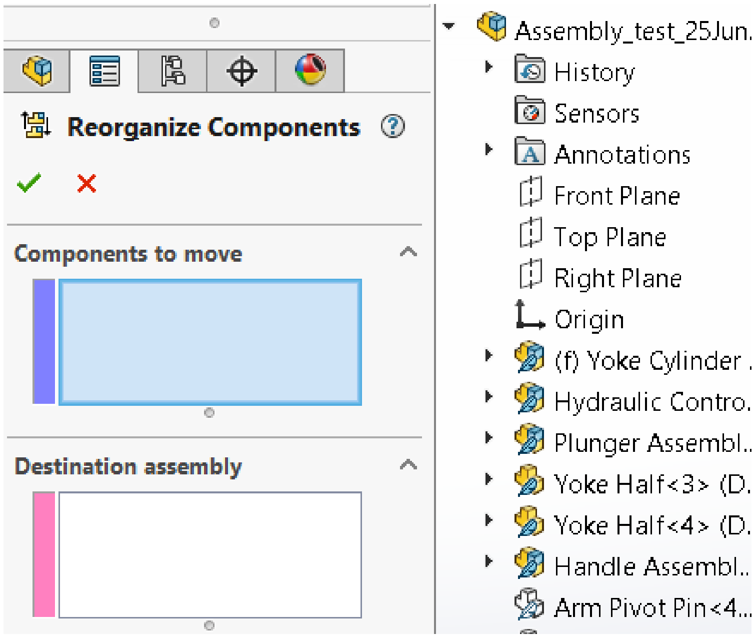
To use the Command Predictor:

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.

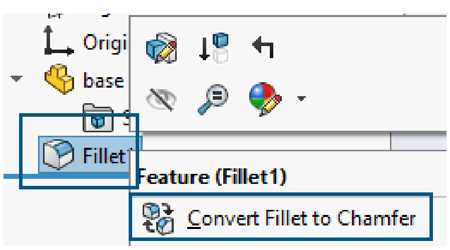


The Reorganize Components dialog box has moved to a PropertyManager. The dialog box no longer obscures the graphics area.

Usability

The user interface is enhanced to improve productivity.

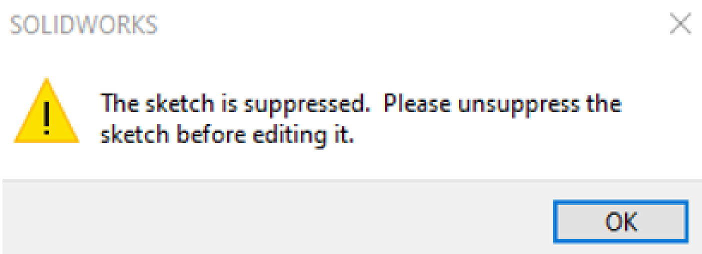
Fillet to Chamfer Naming



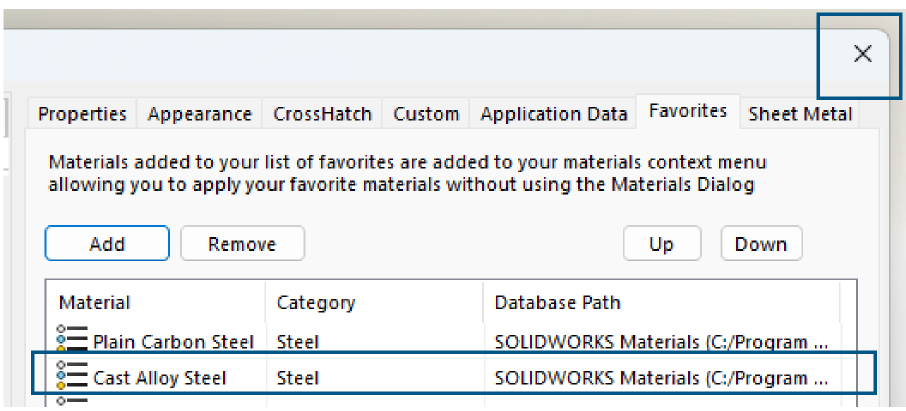
In the FeatureManager® design tree, when you right-click a fillet and select **Convert Fillet to Chamfer**, the FeatureManager design tree changes the fillet name to chamfer. You can use **Convert Chamfer to Fillet** as well and the software updates the name accordingly. In earlier releases, the fillet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

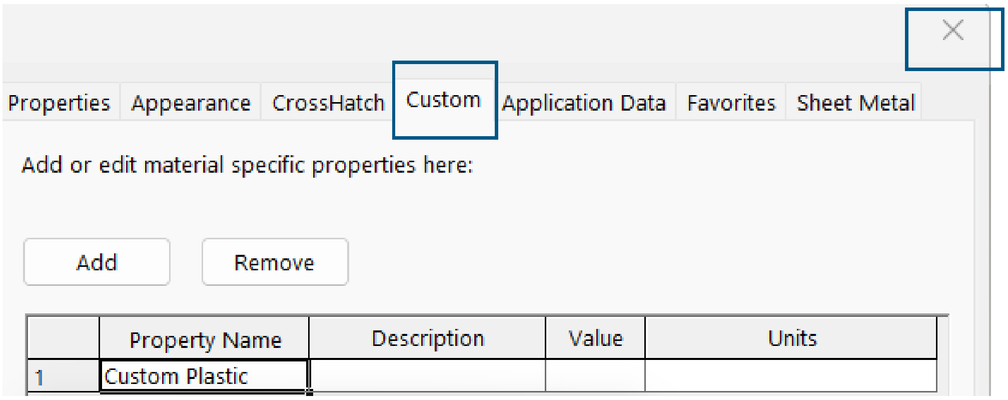


Material Dialog Box - Favorites Tab

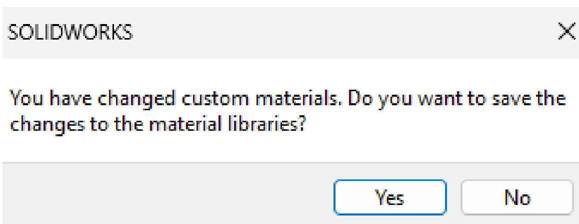


In the Material dialog box, after you add a new material to **Favorites**, you can click **Close** or **x** in the upper-right corner to save the changes and close the dialog box. In earlier releases, when you clicked **x**, the software did not save the changes.

Material Dialog Box - Custom Tab

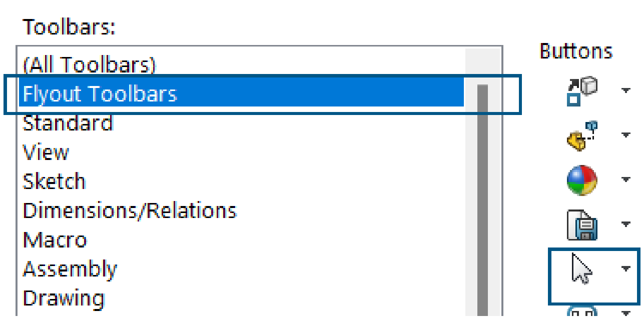


In the Material dialog box, after you add a new material from **Custom Materials** to **Custom**, and click **x**, you receive the following notification:



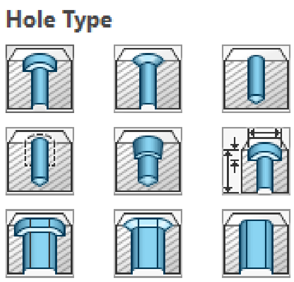
In earlier releases, you only received the notification when you clicked **Close**.

Flyout Menu in Customize Dialog Box



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

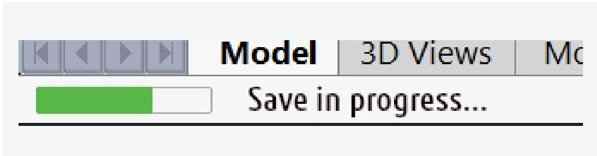
Hole Wizard

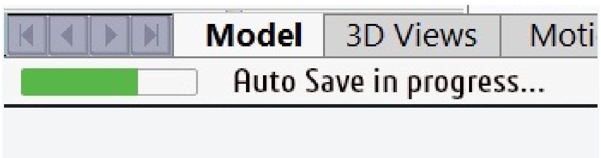


Enhancements to the user interface help improve productivity.

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

Save and Auto Save Progress

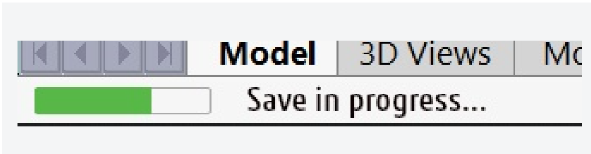




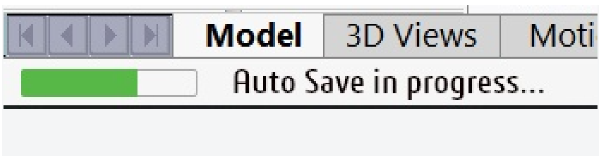
Enhancements to the user interface help improve productivity.

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

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



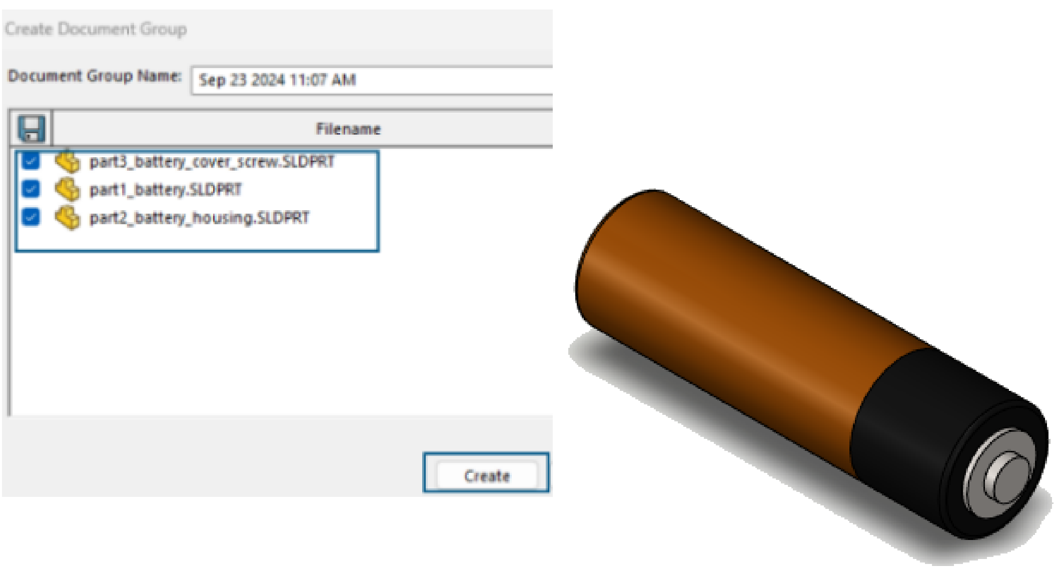
While a file autosaves on the **3D**EXPERIENCE platform, the software shows a progress bar and displays "Auto Save in progress..." in the status bar.



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.

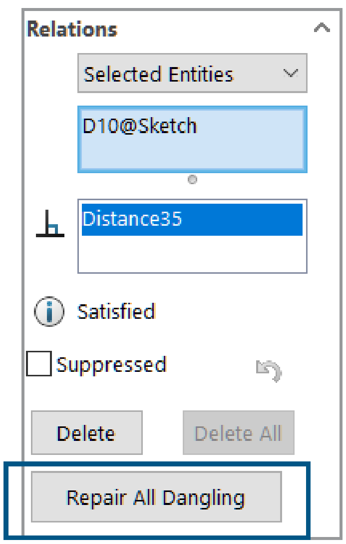
      The software populates the list with the open files and the files saved under the selected document group.
   b) Click **Create**.

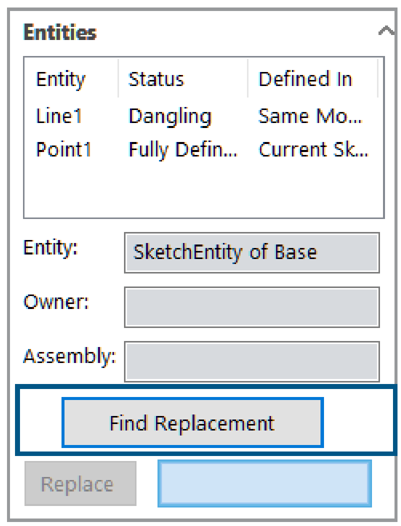
7

Sketching

This chapter includes the following topics:

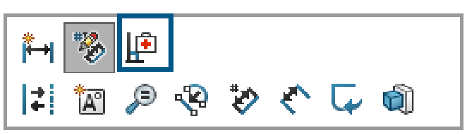
Repairing Dangling Relations





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:

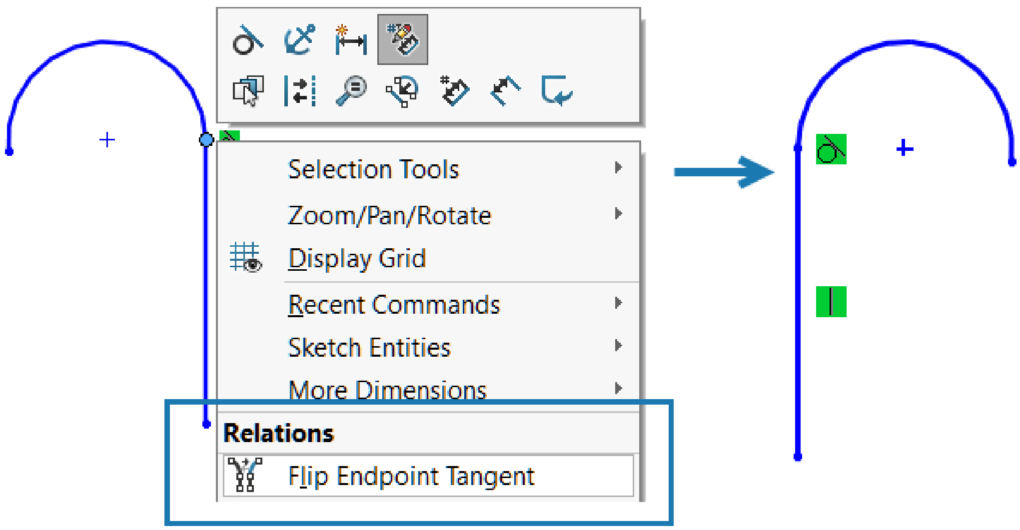
1. Open a model that has a dangling relation.
2. Click **Display/Delete Relations** (Dimensions/Relations toolbar) or **Tools** > **Relations** > **Display/Delete**.
3. In the PropertyManager, under **Relations**, select a dangling relation.
4. Under **Entities**, click **Find Replacement**.

   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**.

Flip Endpoint Tangent (2025 SP1)

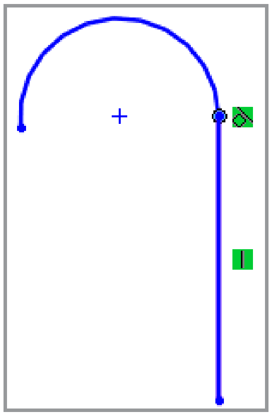


You can flip the endpoint of a tangent arc that is connected to a line. The radius of the arc does not change.

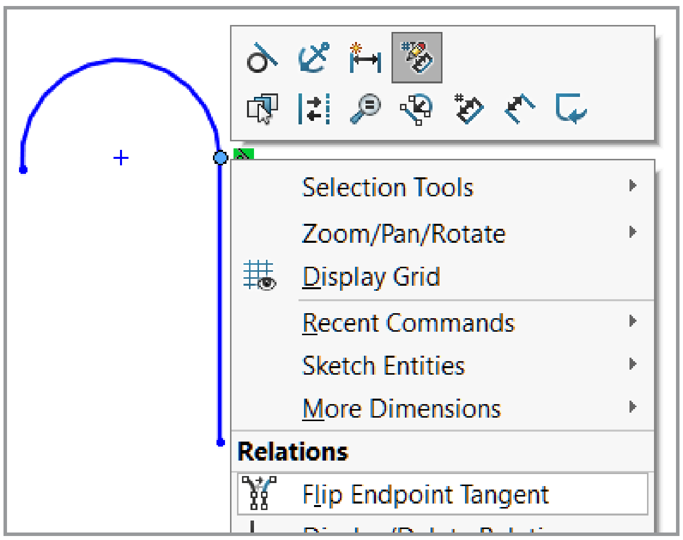
The functionality is not available for 3D sketches.

To flip the endpoint of a tangent arc:

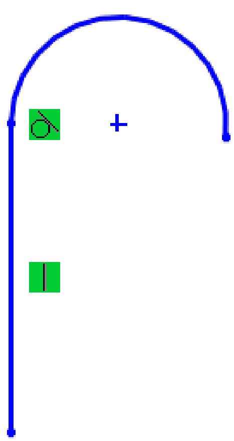
1. Open a new sketch and sketch a line.
2. Click **Tangent Arc** and create an arc from the end point of the line.



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



The tangent arc flips position:



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.

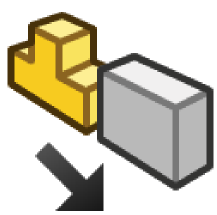
8

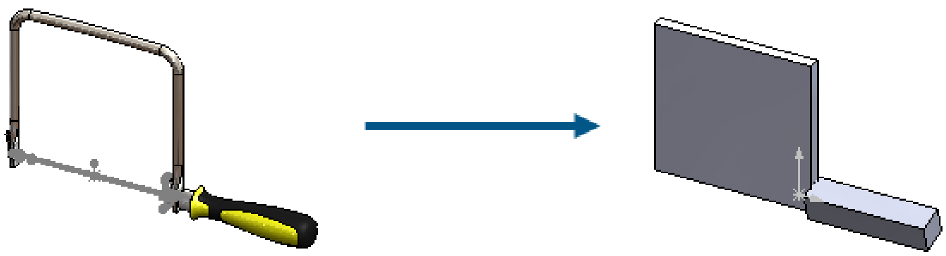
Parts and Features

This chapter includes the following topics:

- **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**

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**

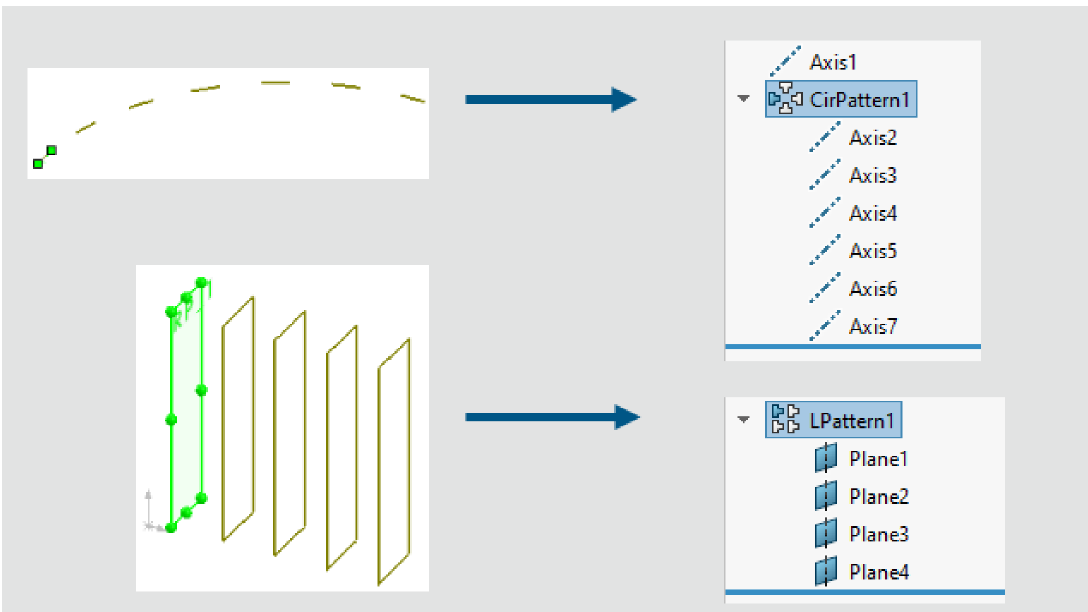
- **Cylinder**
- **Polygon Outline**
- **Tight Fit Outline**
- **None (Copy Geometry)**

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:

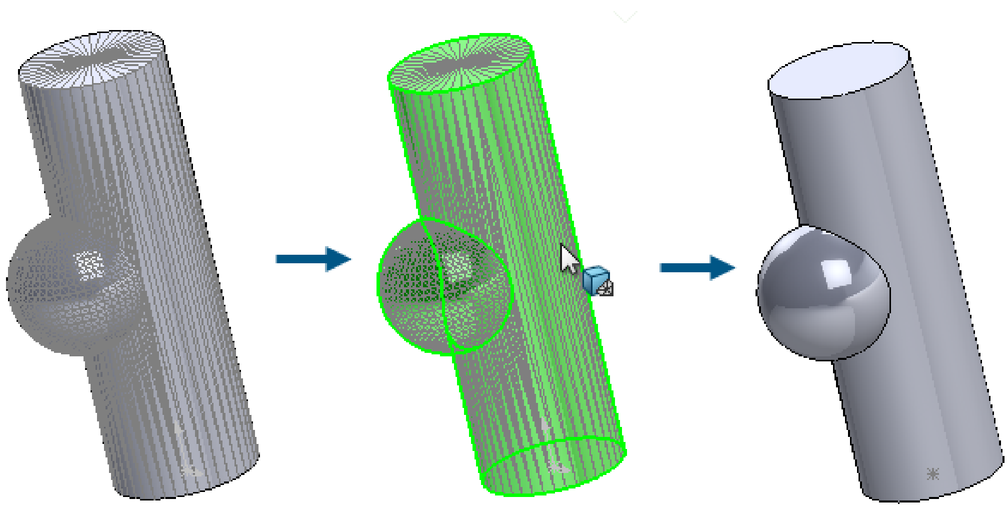
1. Open a part, click **Insert** > **Pattern/Mirror**, and select **Linear Pattern** or **Circular Pattern**.
2. In the PropertyManager, select **Reference Geometry**.
3. For **Reference Plane or Reference Axis to Pattern**, select the plane or axis to pattern.
4. Specify the parameters, then click .

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.

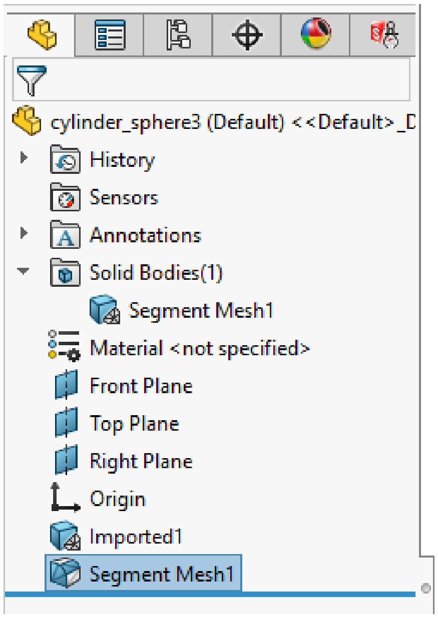
This functionality works for mesh BREP or hybrid mesh bodies that have recognized geometry. The functionality works best for meshes with well-defined planar, cylindrical, conical, and spherical geometry that do not have significant noise.

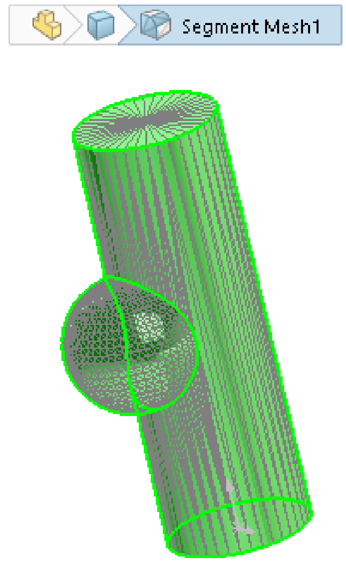
Benefits: Standard BREP geometry is more functionally complete than mesh or hybrid geometry.

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.





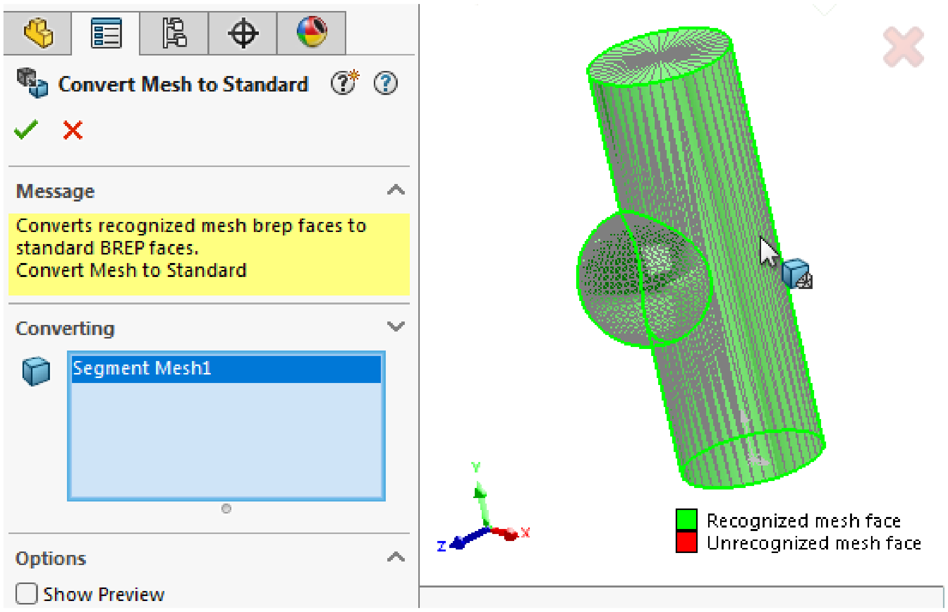
2. Do one of the following:

   - 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).

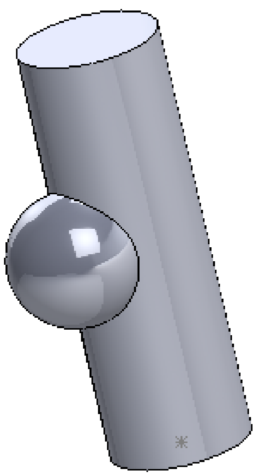
3. In the PropertyManager, under **Select Body**, select bodies to convert segmented, recognized mesh BREP faces to standard BREP faces.

   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.

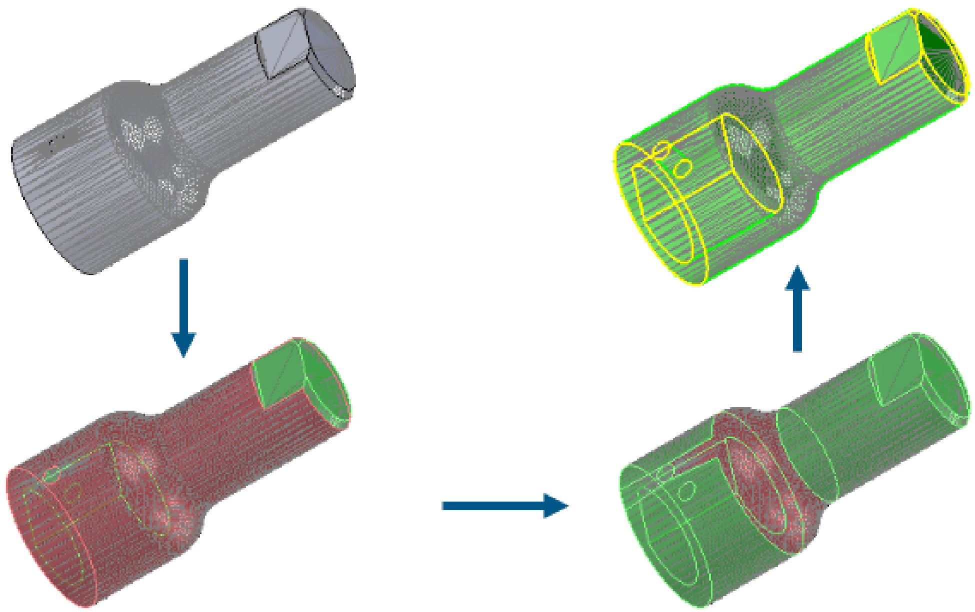


4. Click ✔ to convert the recognized segmented mesh BREP faces to standard BREP faces.



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.

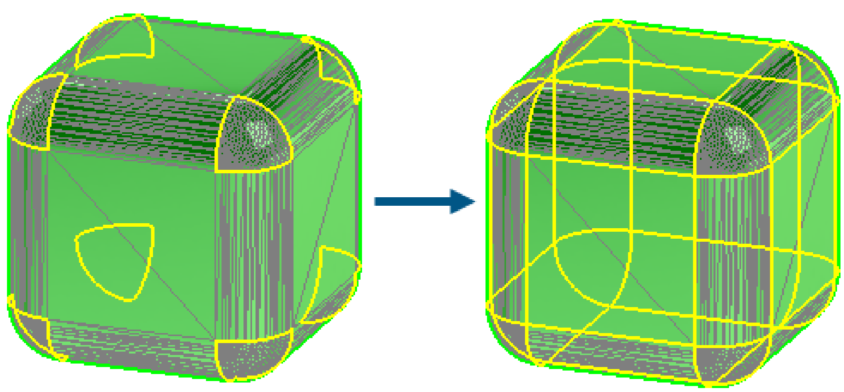
Additional Face Types Recognized

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.



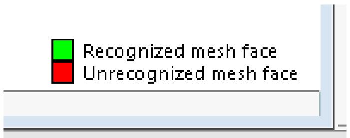
When you run the **Segment Imported Mesh Body** tool, improved graphical information helps you understand which faces have been recognized.

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**.

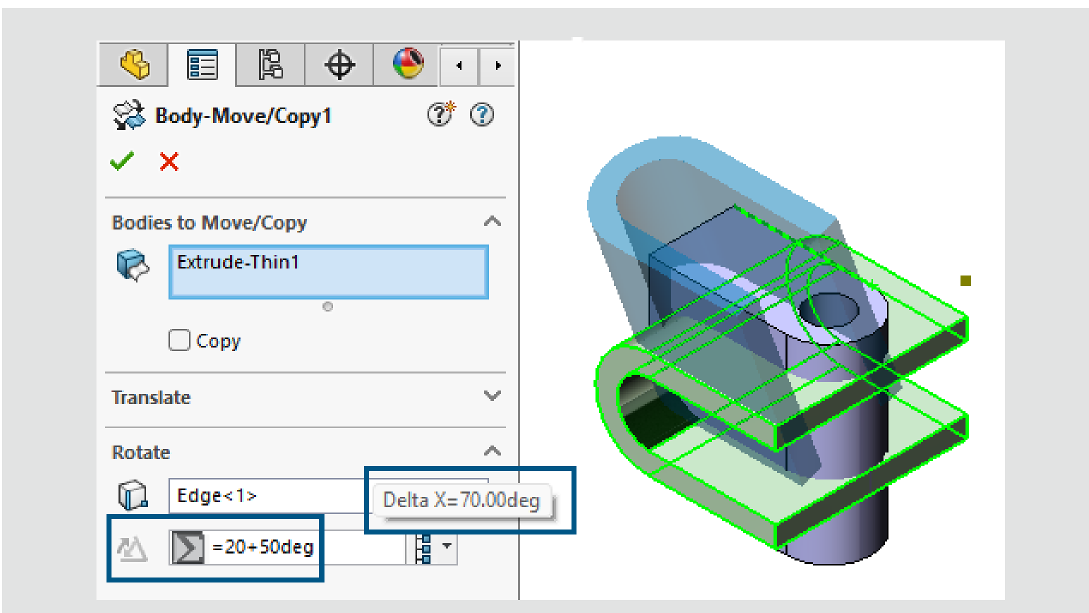
After the first round of segmenting the model, the following display changes apply:

- 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

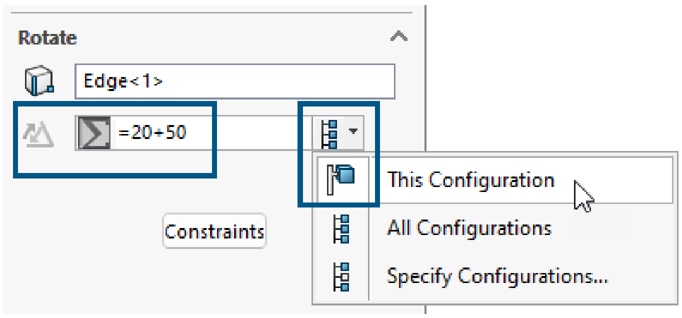


The **Move/Copy Body** feature offers enhanced support for equations and configurations.

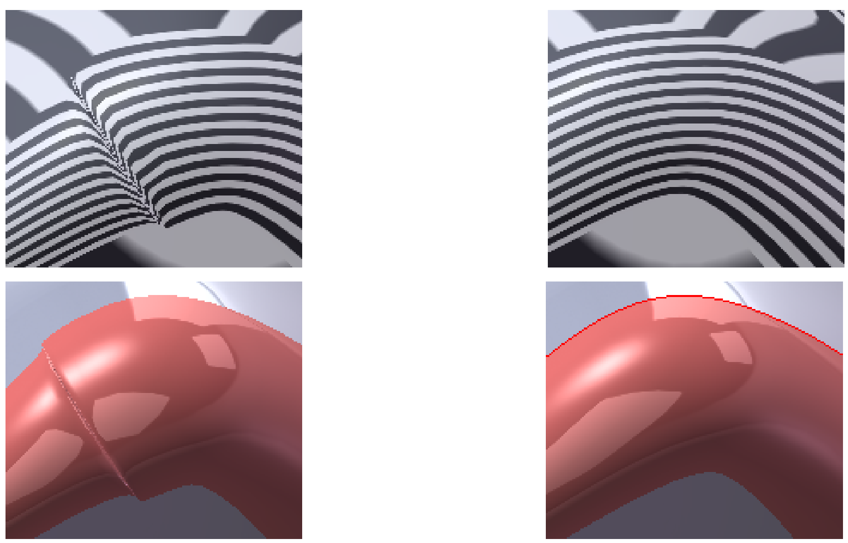
In the Move/Copy Body PropertyManager, you can use equations to specify values for the **Distance** dimension under **Translate** and for the **Angle** dimension under **Rotate**. In the PropertyManager, enter = and the equation. For example, enter `=20+50`. To access this equation in the Equations, 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

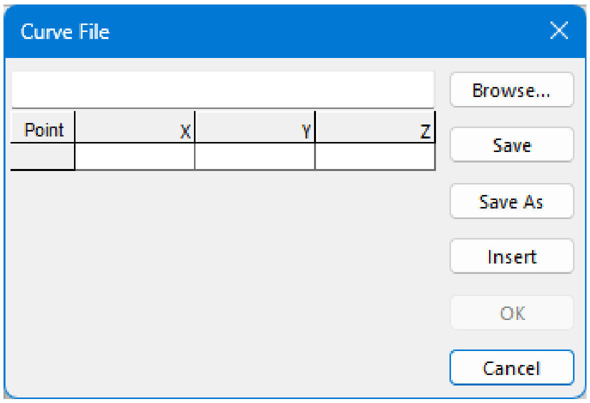
2025 Option

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

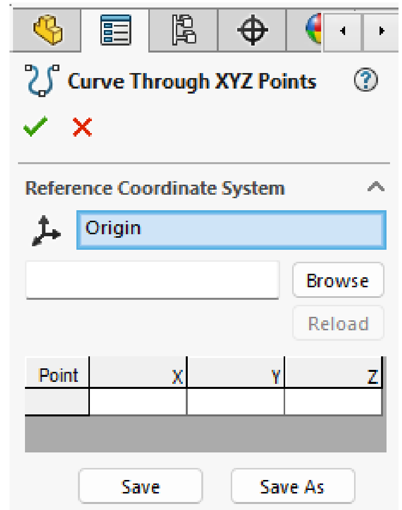
To access this option, in the Fillet PropertyManager, for **Fillet Type**, select **Variable Size Fillet** and under **Fillet Options**, select **Continuous edge blend**.

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)**.

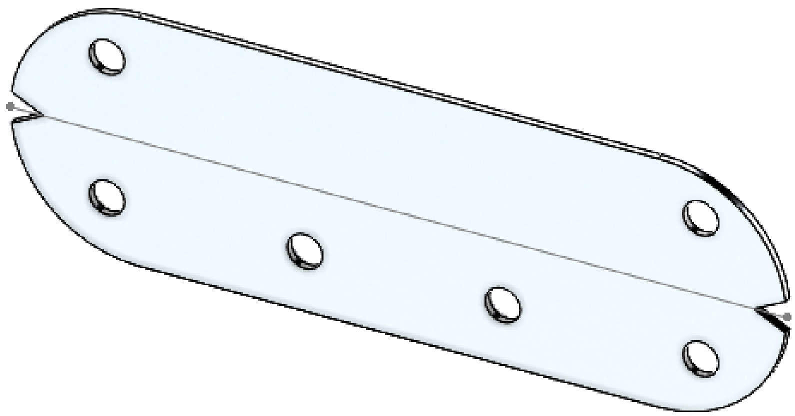
9

Sheet Metal

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



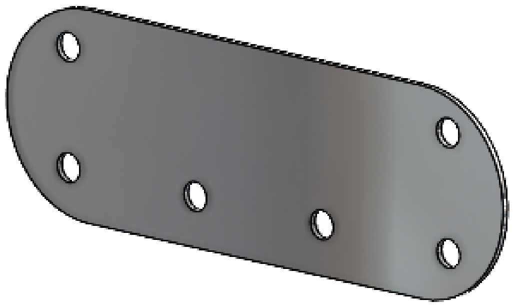
You can create notches across bends in flattened sheet metal parts. In manufacturing, bend notches help manufacturers determine where to put the press brake. You can use notch features on all bends so the bending operator can then use them to line up the bend with the tooling.

Creating Bend Notches

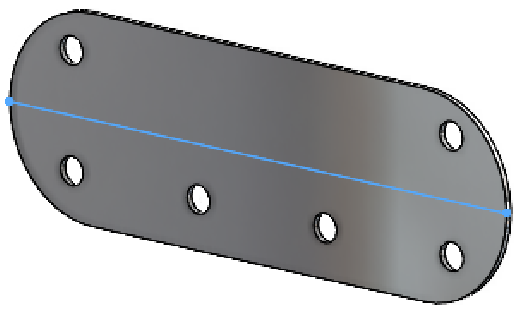
You can create bend notches on sheet metal parts in the flattened state.

To create bend notches:

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

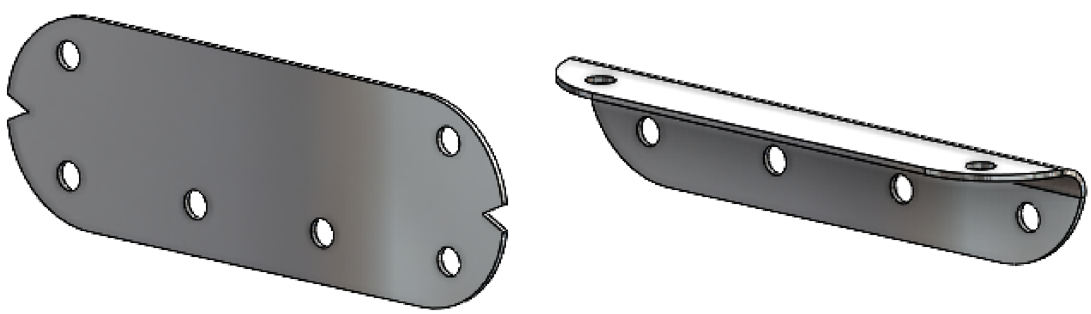


2. In the graphics area, select the bends where you want to add notches.



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.



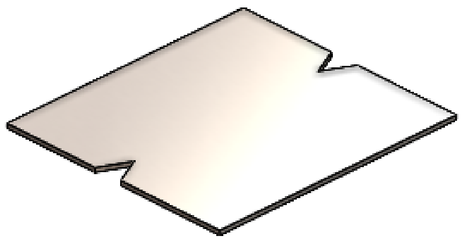
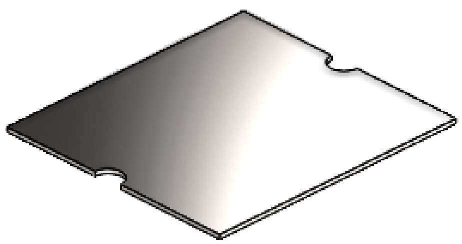
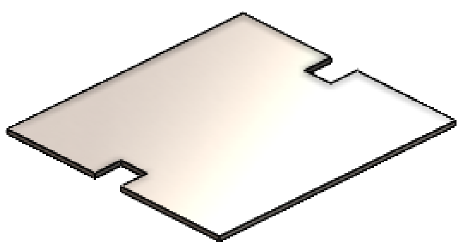
Flattened	Bent

Bend Notch PropertyManager

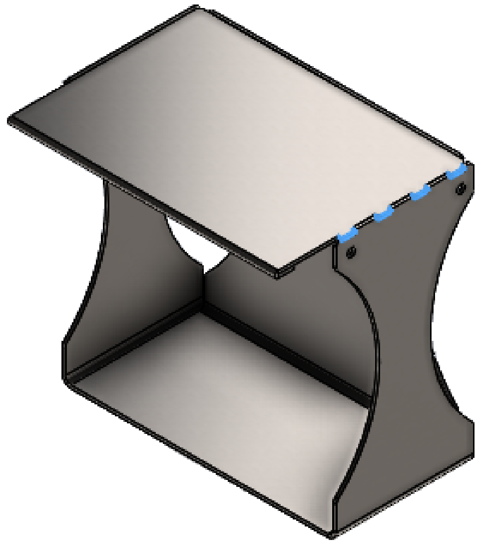
To open this PropertyManager:

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

Bend Notch

Bends	Lists the bends to apply notches to.
Collect all bends	Selects all bends in the part to apply notches to.
Notch type	Specifies the notch shape: • **Triangular**. Specify the **Width** and **Depth** .  • **Circular**. Specify the **Radius** .  • **Rectangular**. Specify the **Width** and **Depth** . 

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.

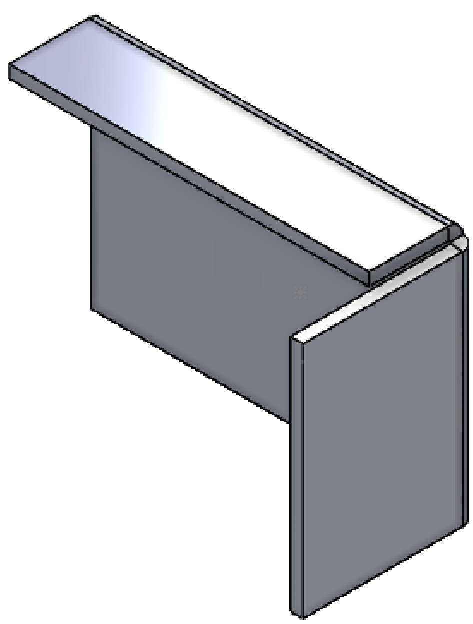
Tabs

	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

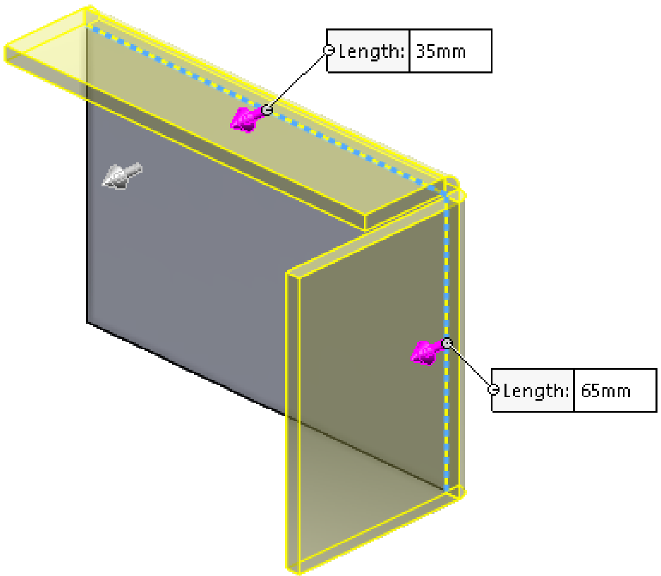
	Instances to Skip	Skips the tab and slot instances that you select in the graphics area. In the graphics area, pink selection orbs display on the tab and slot instances. The pointer changes to when you hover over each instance and the coordinates of the instance appear. Click a selection orb. To restore 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.

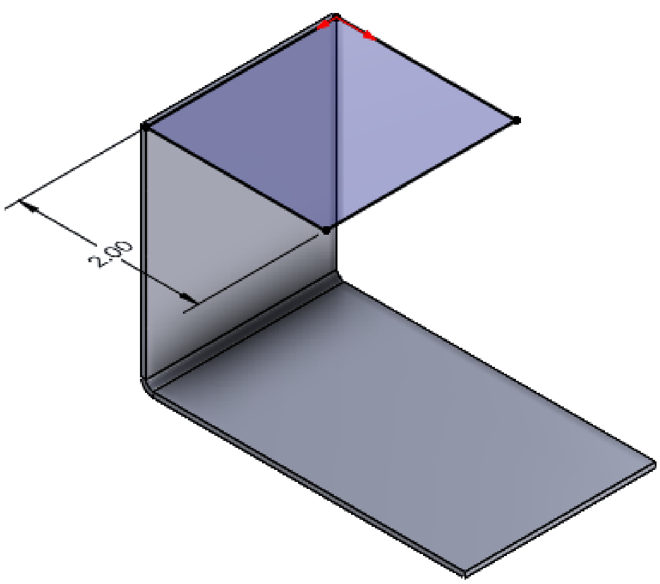
In the PropertyManager, you can select **Multi Length Flange** and specify the length of each flange in the feature. You can specify the **Length** in the PropertyManager or in the graphics area.



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

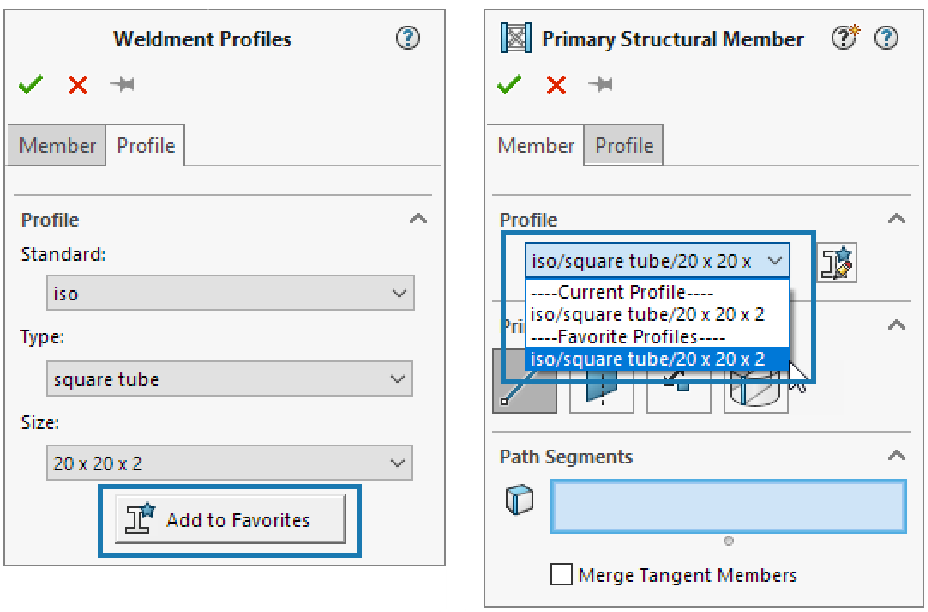
10

Structure System and Weldments

This chapter includes the following topics:

- **Accessing and Working with Favorite Profiles**
- **Complex Corner PropertyManager and Structure System**
- **Trimming Attached Members**
- **Groove Beads**

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.

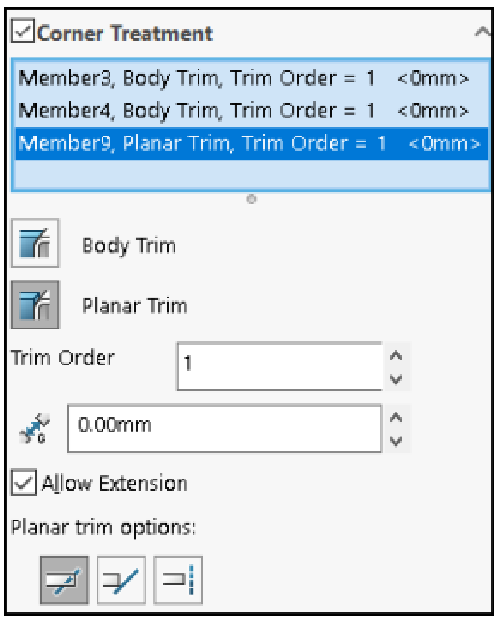
   ★ as a suffix to the size indicates a favorite profile.

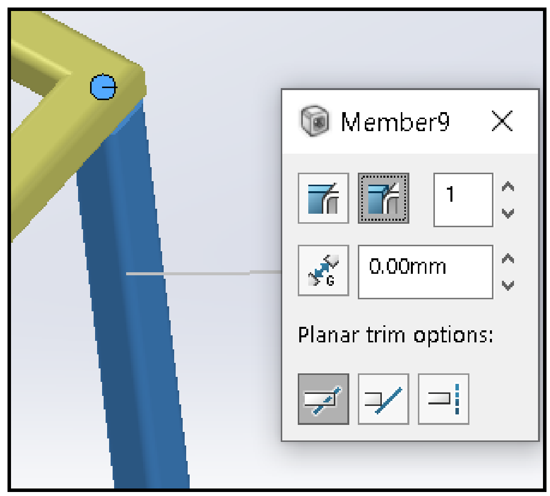
5. On the Member tab, under **Profile**, select the profile in **Favorite Profiles**.
6. Click to modify the list of favorite profiles.
7. In the Favorite Profile List dialog box, select a profile and click the following:
   - **OK**. Accepts the changes, if any.
   - **Delete**. Deletes the selected 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:

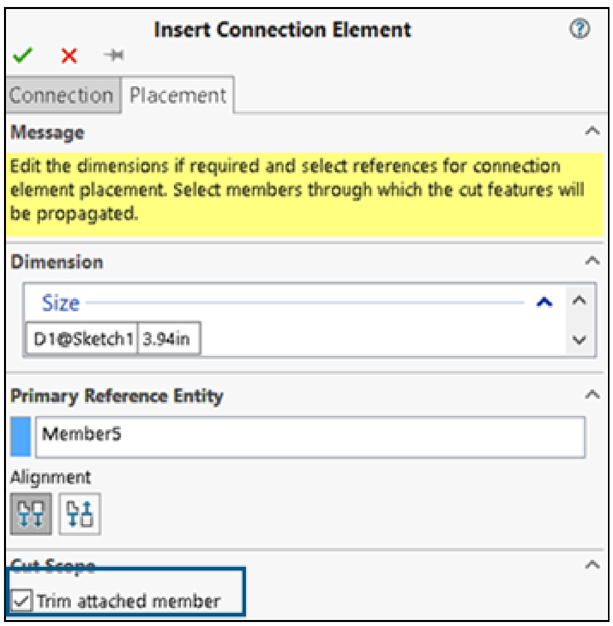
- Under **Corner Treatment**, the members box displays body trim members and planar trim members. You can select a member and click **Body Trim** or **Planar Trim** to change its trim type.
- Details of the selected member, such as body trim, planar trim, and trim order appear as callouts in the graphics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

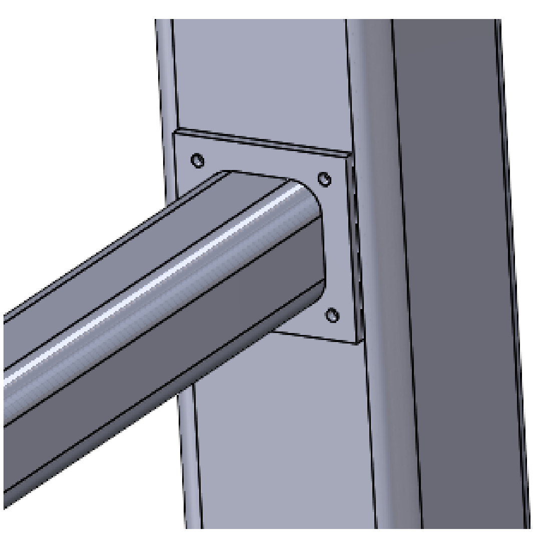


You can trim an attached member when you insert a connection element.

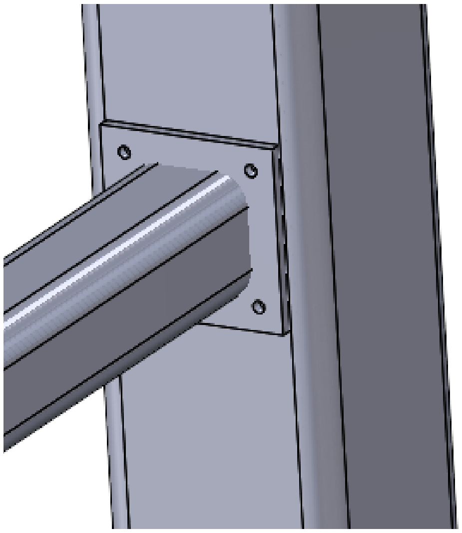
In the Insert Connection Element PropertyManager, **Trim attached member** trims the body member from its intersection point with the connection element.

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 ✓.

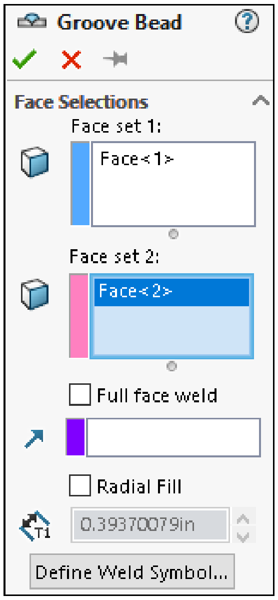


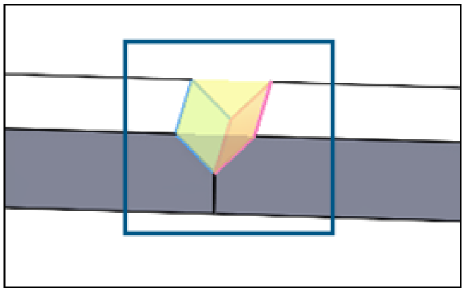
Trim attached member selected

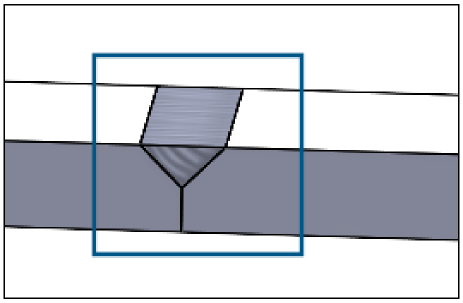


Trim attached member cleared

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:

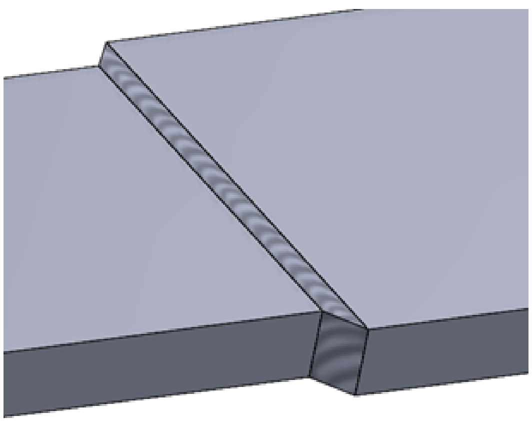
1. Open a multibody part and click **Insert** > **Weldments** > **Groove Bead**.

Face Selections

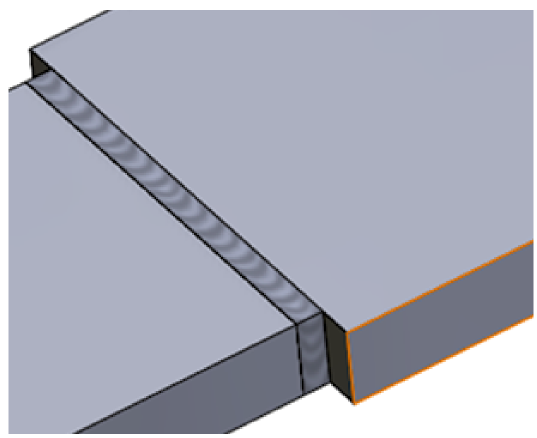
Face set 1 and **Face set 2**. Specifies the faces of solid bodies to connect from the graphics area.

Full face weld

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



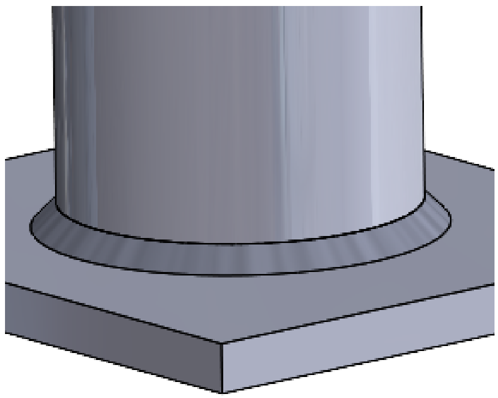
Full face weld selected



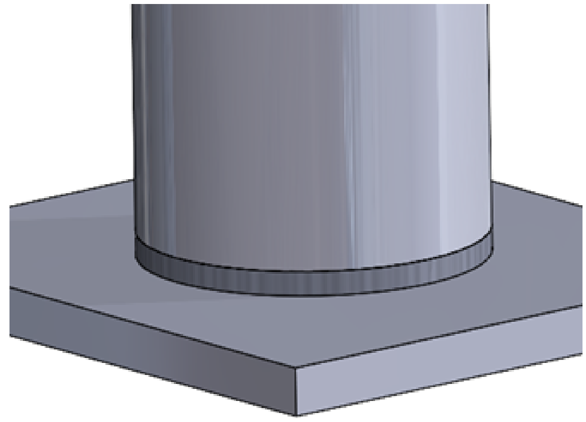
Full face weld cleared

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**.

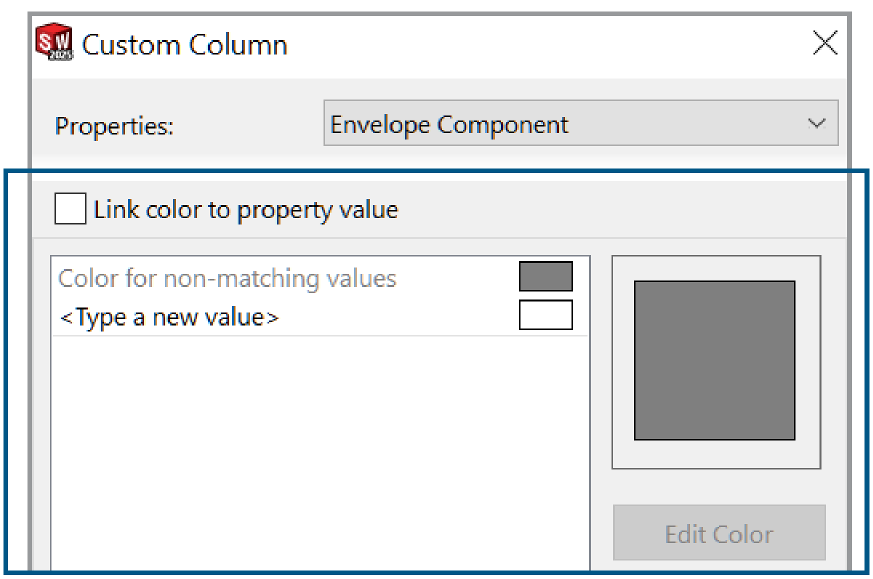
11

Assemblies

This chapter includes the following topics:

- **Assembly Visualization**
- **SpeedPak Instances**
- **Interference Detection in Large Design Review Mode**
- **Canceling Interference Detection Calculations (2025 SP1)**
- **Performance Evaluation**
- **Linking Display State to the Patterned Seed Component**
- **Inserting Assemblies with Rolled-Back Features**
- **Copy with Mates**
- **Maintaining External References to Derived Sketches (2025 SP1)**
- **Warning When Moving Components (2025 SP1)**
- **Performance When Calculating Mass Properties**
- **Controlling the Visibility of Part Sketches in Assemblies**

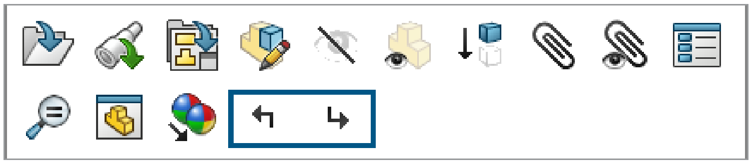
Assembly Visualization



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:

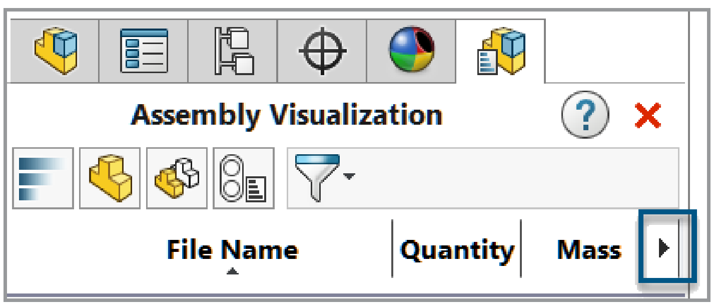
The **3D**EXPERIENCE properties are available on the **3D**EXPERIENCE platform. For these properties, **Link color to property value** is always selected.

Property	Description
Envelope Component	Reports whether the component has an envelope component.
Overridden Mass Properties	Reports whether the component has overridden mass properties.
3DEXPERIENCE - CAD Format	Reports the CAD format of the component. Examples of CAD formats: • **3D**EXPERIENCE® • CATIAV5 • X-CAD • SOLIDWORKS®
3DEXPERIENCE - Collaborative Space	Reports the collaborative spaces where the component is saved.
3DEXPERIENCE - Latest Revision	Reports whether this is the latest revision of the component.
3DEXPERIENCE - Lock status	Reports the lock status of the component: • Locked by me • Locked by other user • Not locked

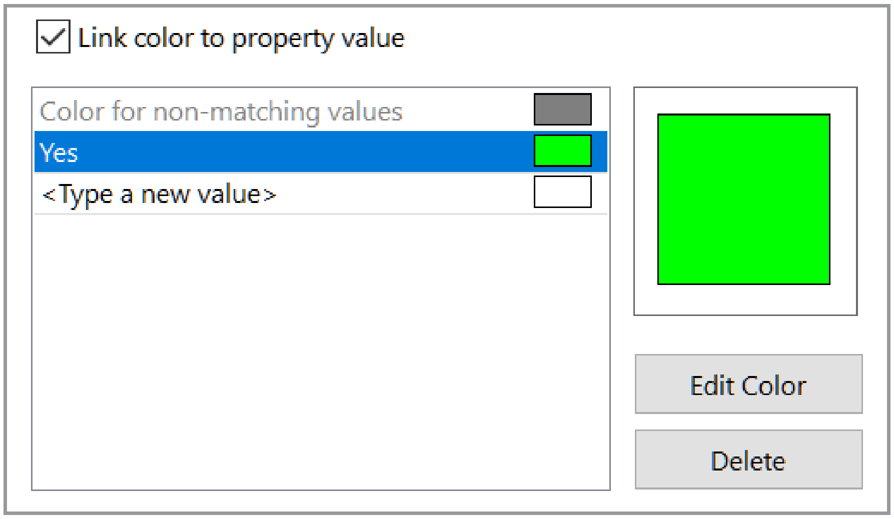
Property	Description
3DEXPERIENCE - Maturity	Reports the maturity level of the component: • Frozen • In Work • Obsolete • Private • Released
3DEXPERIENCE - Updated for compatibility	Reports whether the component is updated for compatibility with the **3D**EXPERIENCE platform.

To link a color to a property value:

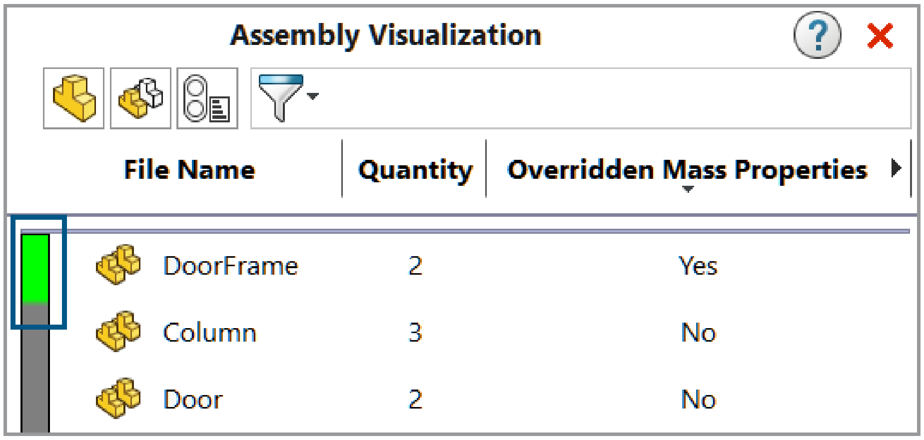
1. Open a model that contains components with overridden mass properties.
2. Click **Assembly Visualization** (Tools toolbar or Evaluate tab on the CommandManager) or **Tools** > **Evaluate** > **Assembly Visualization**.
3. On the Assembly Visualization tab, click the arrow to the right of the column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



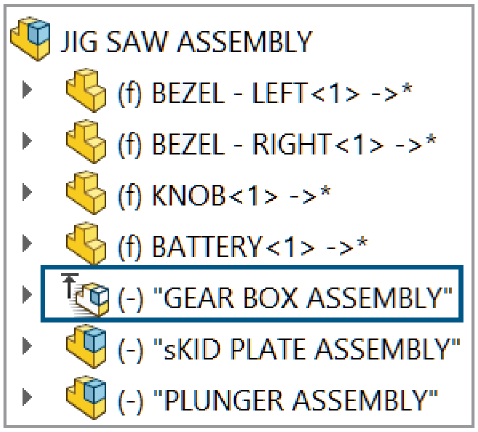
4. Click **More**.
5. In the Custom Column dialog box, under **Properties**, select a property like **Overridden Mass Properties**.
6. Select **Link color to property value**.
7. Double-click **Type a new value** and enter a value.
8. Click **Edit Color** and select a color for the value.



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



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**.

4. When prompted, select **Create a SpeedPak Instance in the top level assembly**.

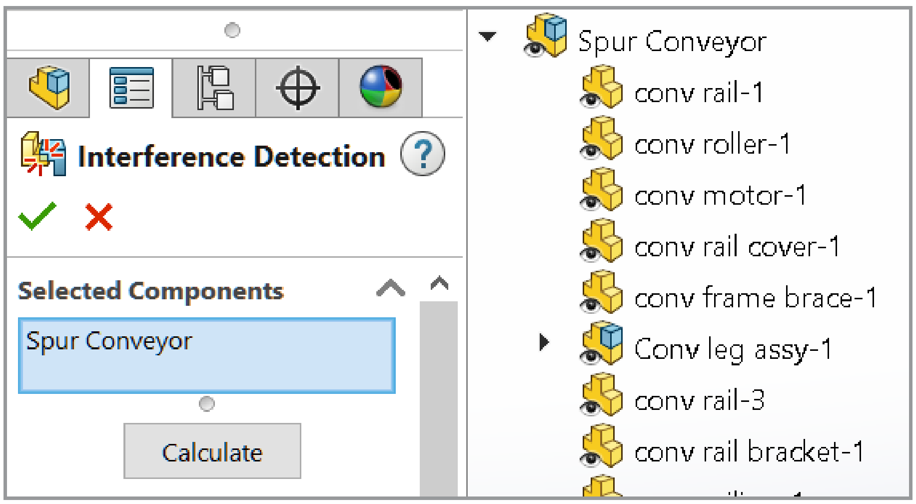
   The SpeedPak instance appears in the FeatureManager design tree.

Switching Between a SpeedPak Instance and a Parent Subassembly

To switch between a SpeedPak instance and a parent subassembly:

1. In the FeatureManager design tree, right-click the SpeedPak instance and click **SpeedPak Options** > **Set SpeedPak to Parent**.
2. 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.

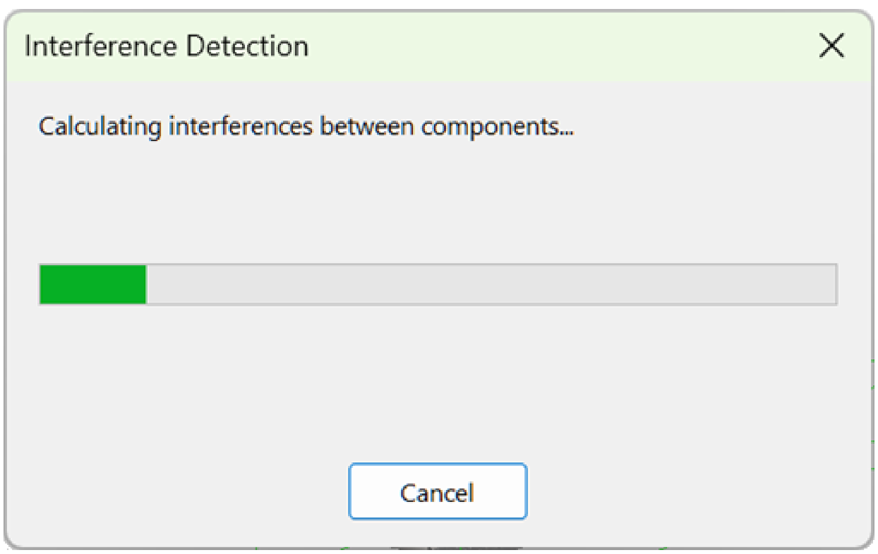
To use interference detection in Large Design Review mode:

1. Open an assembly in Large Design Review mode.
2. Click **Interference Detection** (Large Design Review tab) or **Tools** > **Evaluate** > **Interference Detection**.
3. 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**

Canceling Interference Detection Calculations (2025 SP1)

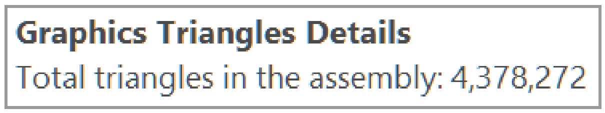


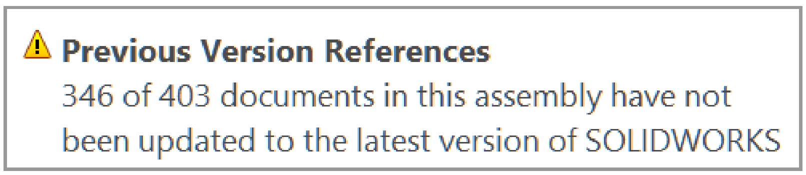
You can cancel calculations for interference detection.

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**.

Performance Evaluation





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 **Open Summary**, displays the time taken to open the assembly.	Open Performance
Searching for Referenced Documents	Lists the number of documents found in the **Referenced Documents** folders and the time taken to perform the search.	Open Performance
Total number of triangles in the assembly	Under **Graphics Triangles Details**, 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

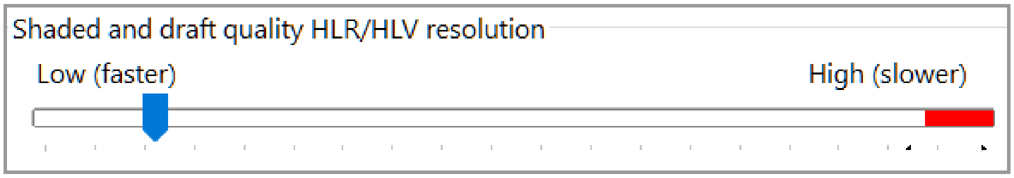
Options and Information	Description	Section
Reduce Image Quality	Under **Shaded Image Quality**, reduces the shaded image quality to 50% for the parts with higher image quality. This option does not apply to subassemblies. Not available for assemblies opened in lightweight mode except when the assembly has a flexible subassembly. Clicking **Reduce Image Quality** moves the **Low (faster) - High (slower)** slider closer to the **Low (faster)** side. To view the slider, click **Tools** > **Options** > **Document Properties** > **Image Quality**. The slider is under **Shaded and draft quality HLR/HLV resolution**.	Display Performance
Time to solve mates	Under **Mate**, displays the time required to solve the mates when you rebuild the assembly.	Rebuild Performance
Open and **Isolate Components**	You can use **Open** and **Isolate Components** in the Mates dialog box. Under **Mate**, click **Show These Files** 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 **Rebuild on Save** mark .	Rebuild Performance
Statistics	Under **Assemblies**, the statistics do not include suppressed mates.	Statistics

To use performance evaluation:

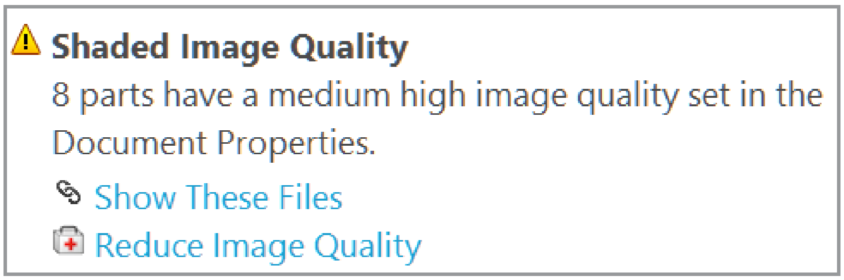
1. Open an assembly.
2. Click **Performance Evaluation** (Evaluate toolbar) or **Tools** > **Evaluate** > **Performance Evaluation**.

To reduce the image quality:

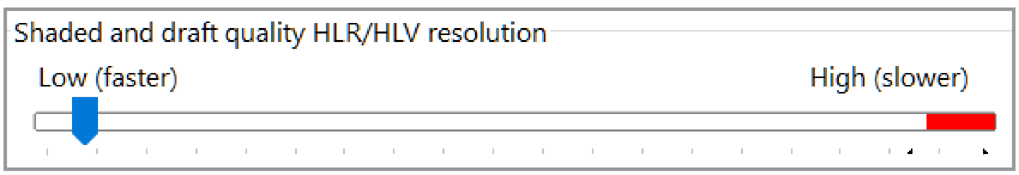
1. Open a model and click **Tools** > **Options** > **Document Properties** > **Image Quality**.
2. Review the slider position under **Shaded and draft quality HLR/HLV resolution**.



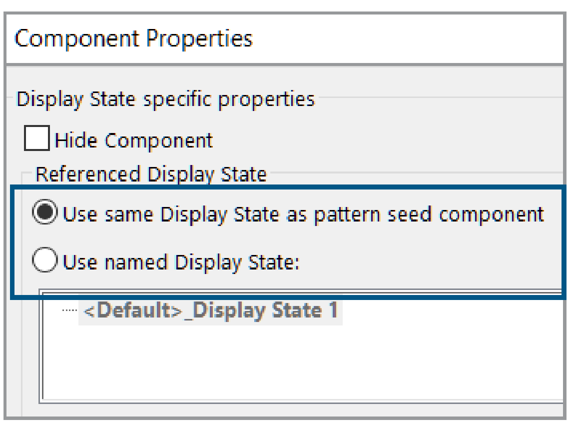
3. Click **Tools** > **Evaluate** > **Performance Evaluation**.
4. Under **Shaded Image Quality** in the **Display Performance** section, click **Reduce Image Quality** .

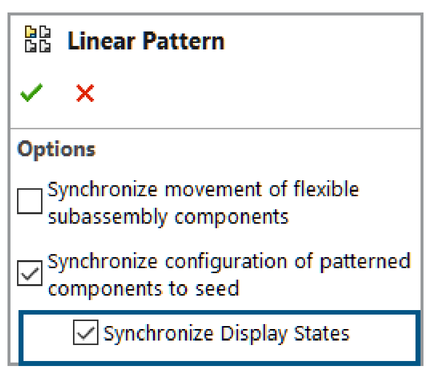


5. After the Performance Evaluation results update, check the slider position under **Shaded and draft quality HLR/HLV resolution**.



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:

Use same Display State as pattern seed component	Links the display state of the patterned components to the patterned seed component. Disables the list of display states.
Use named Display State	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.

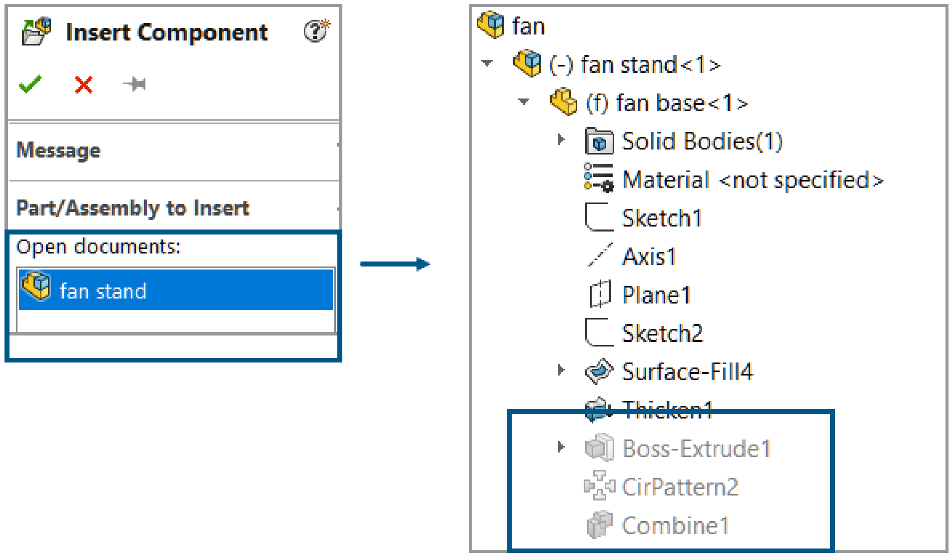
You can link the display state in any Component Pattern PropertyManager. In the PropertyManager, under **Synchronize configuration of patterned components to seed**, select **Synchronize display states**.

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



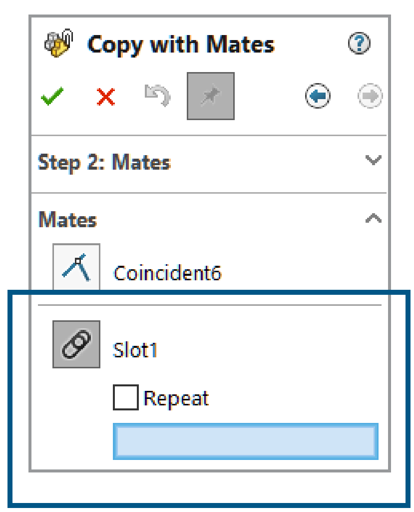
In a model, you can insert an assembly with a part reference that has rolled-back features.

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.

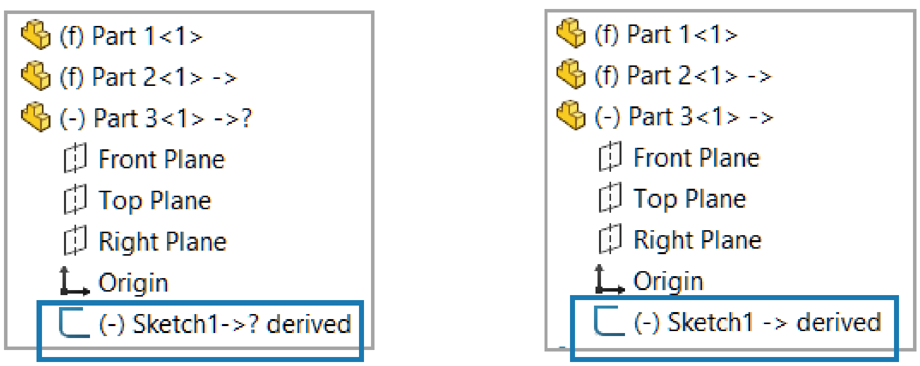
> For hinge mates, you can copy a maximum of six hinge mates at the same time.

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.

Maintaining External References to Derived Sketches (2025 SP1)



2024 2025

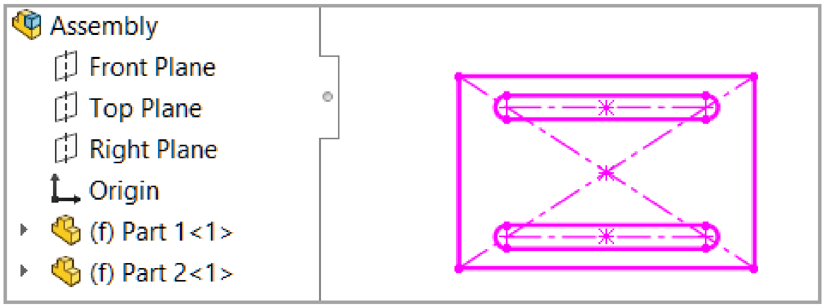
When you use **Save As Copy and Continue** to copy a part, external references to a derived sketch in the copied part are maintained.

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:

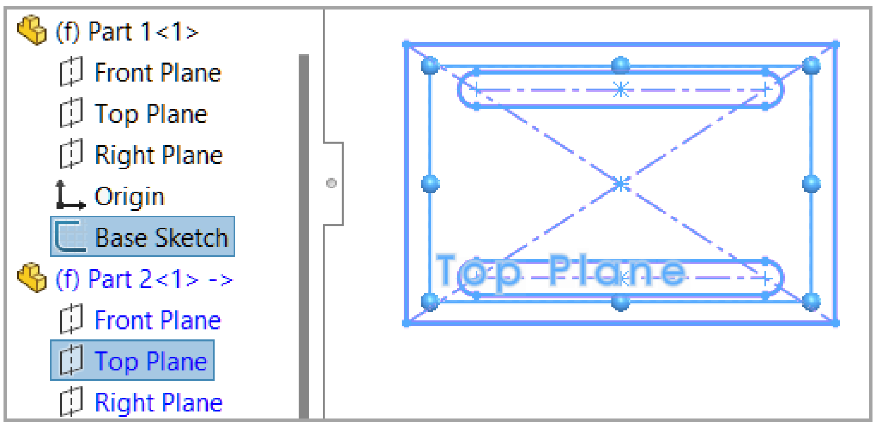
1. Open a model that contains two parts.

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



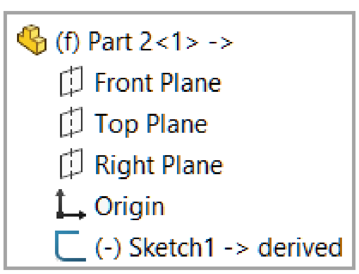
2. Create a derived sketch.

   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.

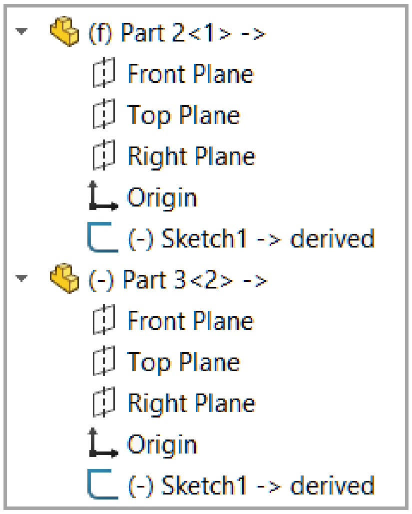
   a. Right-click Part 2 and click **Edit Part**.
   b. Click **File** > **Save As** > **Save as Copy and Continue**.
   c. Save the new part as Part 3.
   d. Exit the edit-in-context mode.

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.

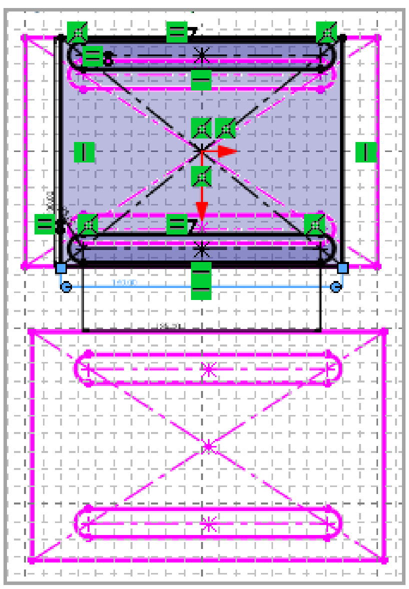


5. Update the first part.

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

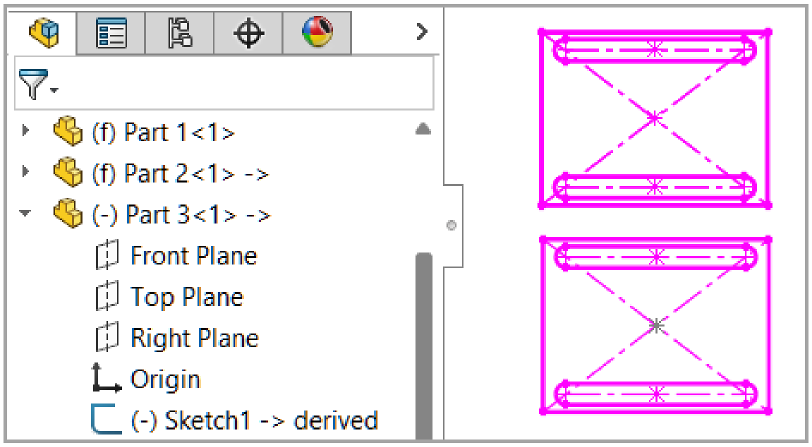
b. Modify a dimension.

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

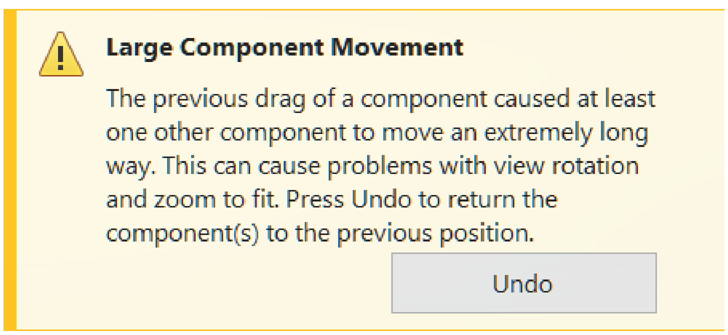


c. Exit the edit-in-context mode.

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** .

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

This chapter includes the following topics:

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		Ø40	Ø20	70	80	80	118.79	30	3	80	15	22	36
2	B01001		Ø40	Ø20	70	80	80	118.79	30	3	80	15	22	36
3	B02001		Ø41	Ø20.5	71	80	80	120.59	30	3	80	15	22	37
4	B03001		Ø42	Ø21	72	80	80	122.39	30	3	80	15	22	38
5	B04001		Ø43	Ø21.5	73	80	80	124.19	30	4	80	15	22	39
6	B05001		Ø44	Ø22	74	80	80	126	30	4	80	15	22	40
7	B06001		Ø45	Ø22.5	75	80	80	127.81	30	4	80	15	22	41
8	B07001		Ø46	Ø23	76	80	80	129.62	30	5	80	15	22	42
9	B08001		Ø47	Ø23.5	77	80	80	131.44	30	5	80	15	22	43
10	B09001		Ø48	Ø24	78	80	80	133.25	30	5	80	15	22	44
11	B10001		Ø49	Ø24.5	79	80	80	135.07	30	5	80	15	22	45

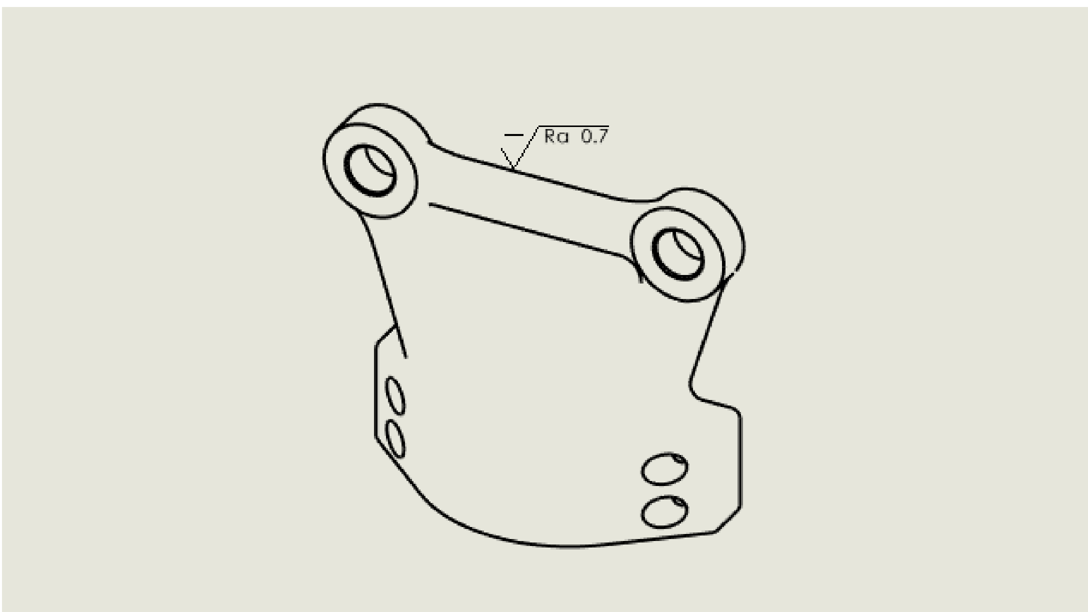
You can use the **Family Table** tool to insert configuration data in drawings.

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:

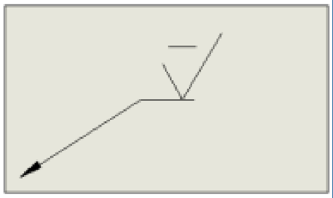
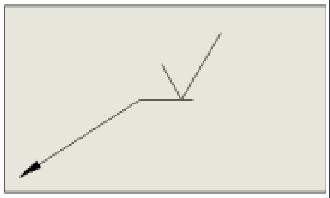
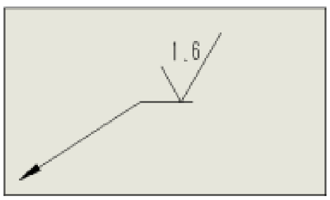
1. In a SOLIDWORKS drawing, click **Insert** > **Tables** > **Family Table**.
2. In the PropertyManager:
   a. Select the file from which to create the family table and click **Next**.
   b. Specify options to define the family table.
   c. Click .
3. Click in the drawing file to place the table.

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.

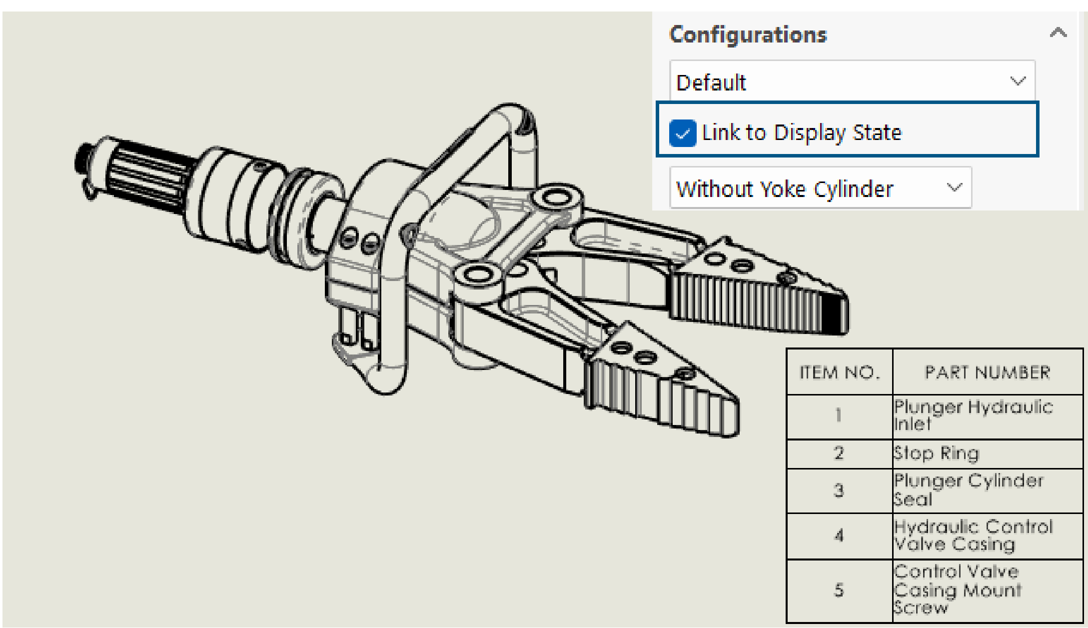
You can use the following symbols.

	**21920-1**
	**1302 (2002)**
	**1302 (1992)**

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)



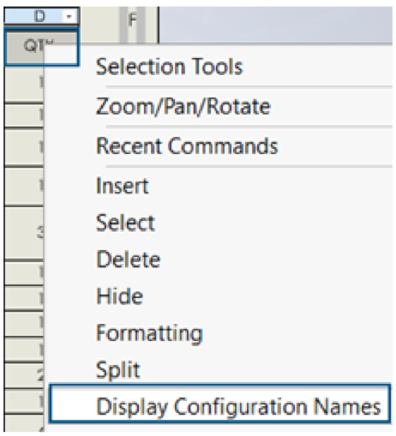
In the Bill of Materials (BOM) PropertyManager, you can link a BOM to display states.

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 .

Bill of Materials Configuration Names (2025 SP1)



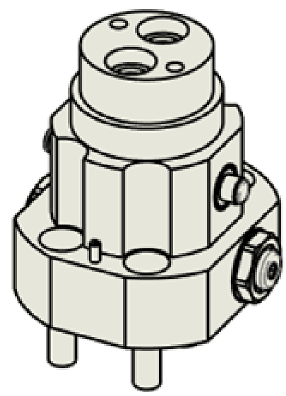
ITEM NO.	PART NUMBER	Default/ QTY.
1	Plunger Hydraulic Inlet	1
2	Stop Ring	1
3	Plunger Cylinder Seal	1

The option **Show all configurations** is renamed to **Display configuration names**. You can access this option by right-clicking the quantity column (**QTY.**) in a bill of materials (BOM).

When you link a display state to a BOM, the software displays the configuration name in the **QTY.** column as **Configuration name / QTY.**

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.

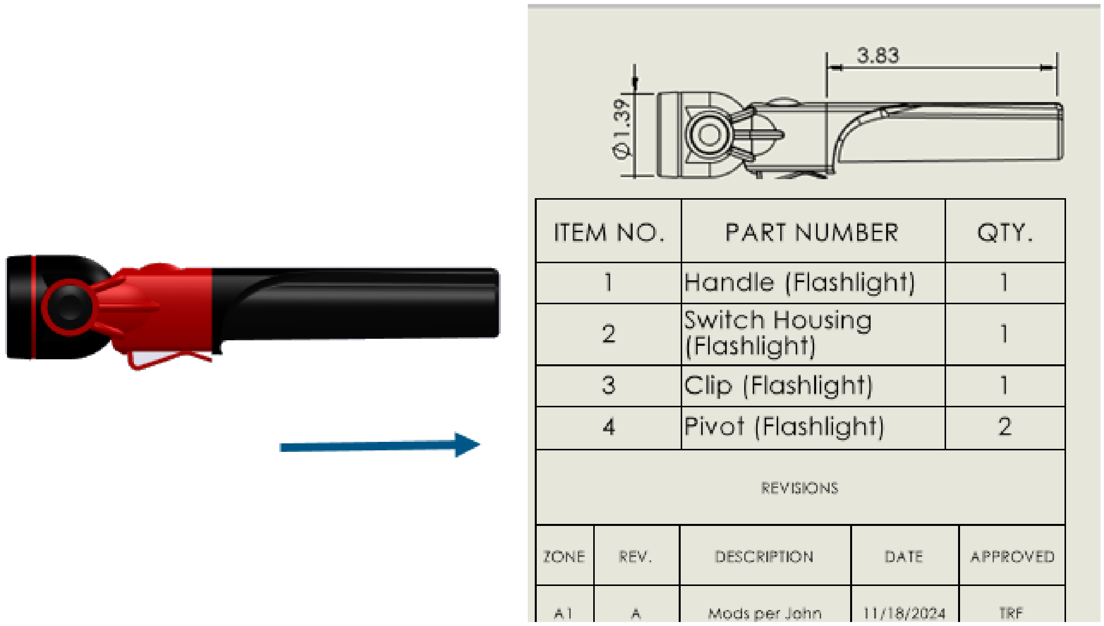
The flattened BOM displays the:

- 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 (2025 SP1)



You can auto generate drawings of parts and assemblies.

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

Auto Generating Drawings

You can automatically generate drawings of parts and assemblies.

To auto generate drawings:

1. Do one of the following:
   - Click **File** > **Auto-Generate Drawing**.
   - In the FeatureManager design tree or graphics area, right-click a part, subassembly, or assembly, and click **Auto-Generate Drawing**.
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**.
   - In the Auto-Generate Drawings task pane, click **Edit**.
3. In the PropertyManager, specify options and click .

Auto-Generate Drawing PropertyManager

In the Auto-Generate Drawing PropertyManager, you can select components from parts or assemblies to automatically generate a drawing.

To open this PropertyManager:

In a part or assembly, click **File** > **Auto-Generate Drawing**.

	Selected components	Specifies the components to include in the auto-generated drawing.
	Title	Specifies a title for the auto-generated drawing.
	**Reset to filename**	Resets the title of the drawing to the part or assembly file name.
	Save location	Specifies a folder to save the auto-generated drawing.
	Same as parent part/assembly	Saves the auto-generated drawing in the same folder as the component selected for the drawing generation.

Tasks (Auto-Generate Drawings) Tab

The Tasks (Auto-Generate Drawings) tab shows a list of generated drawings and their progress. You can monitor the progress of these drawings tasks and take actions.

To open this tab:

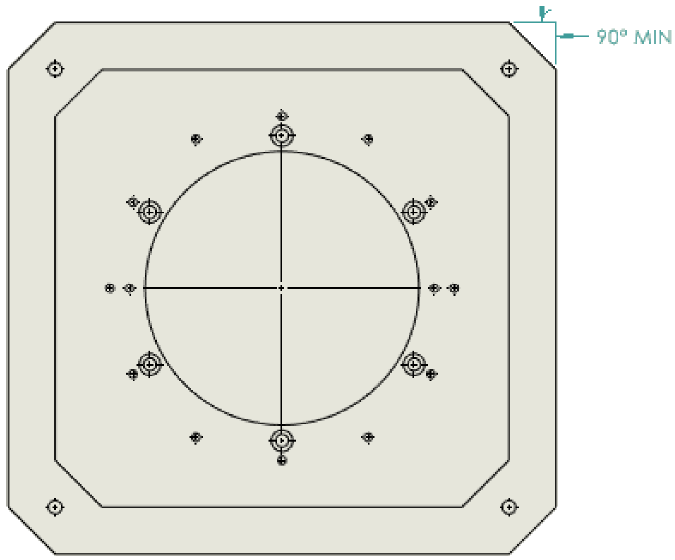
In a part or assembly, select the **Tasks (Auto-generate drawings)** tool from the task pane tab.

	Title	Displays the name of the generated drawing.

	Status	Displays the status of the drawing generation. The status includes one of these icons: • In-progress  • Completed •  • Failed • 

Actions	Displays actions that you can perform:
	• **Cancel**. (Available during drawing creation.) Cancels the auto-drawing generation for the selected item.
	• **Open**. (Available when the software completes the drawing creation.) Opens the selected drawing in Detailed Mode.
	• **View Details**. (Available when the drawing creation fails.) Opens the report to show why the auto-generated drawing failed.
	• Right-click any row in the task tab to:
	  • **Clear**. Clears the selected row from the list.
	  • **Clear All**. Clears all the rows from the task tab, except for the rows in progress. This includes rows in which the status is complete or failed.

Additional Tolerance Types for Chamfer Dimensions

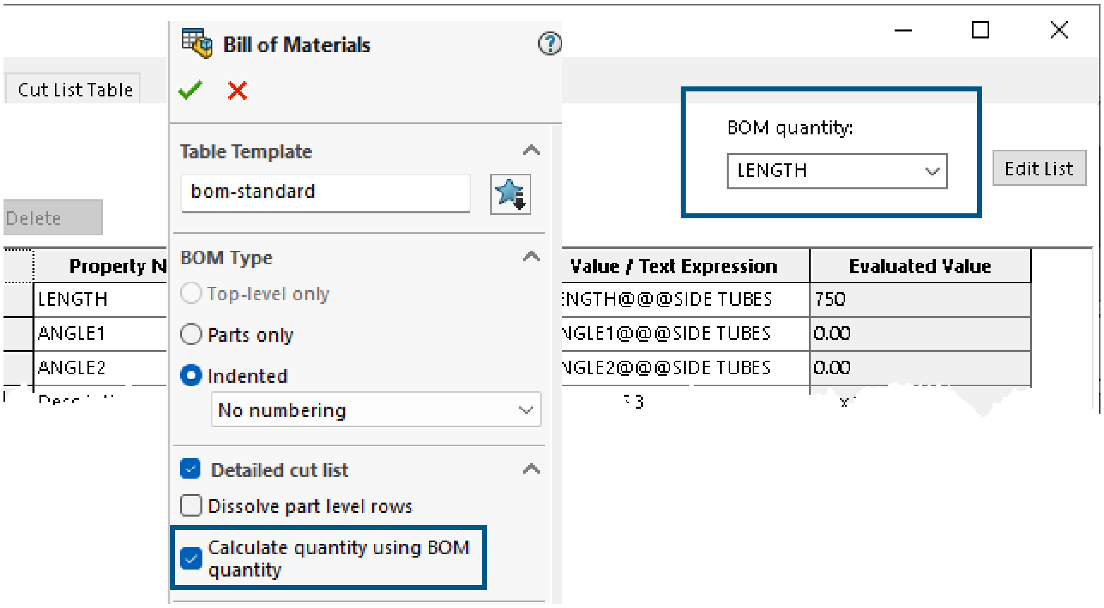


You can specify **MIN**, **MAX**, **Limit**, **Fit**, and **Fit with Tolerance** tolerance types for chamfer dimensions in drawings.

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.

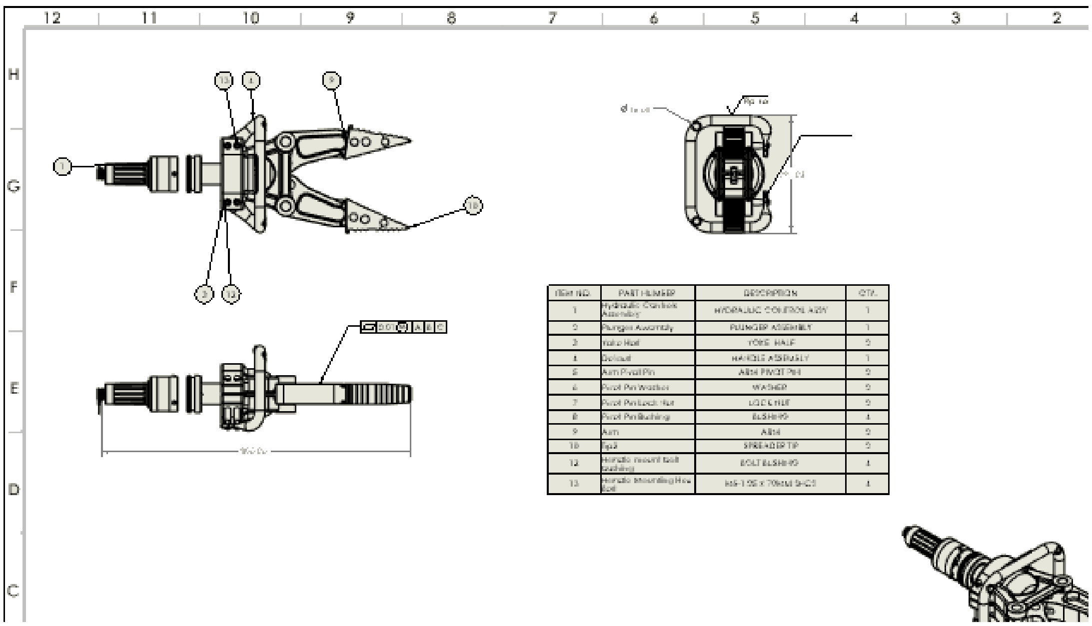
To use the BOM quantity override for detailed cut lists:

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 .

Performance Improvements in Drawings

Drawings performance has improved with panning and zooming.

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 share the latest version of files with other users.

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.

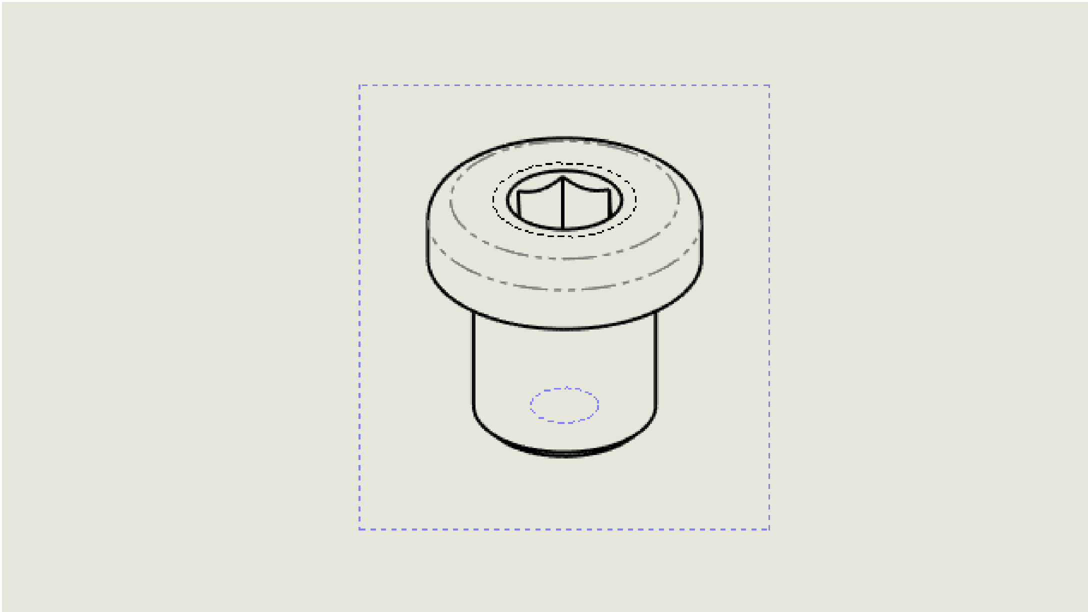
Benefits: Grouping related geometry into blocks helps organize drawings and makes it easier to navigate and manage complex designs.

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 assembly drawings, you did not see the cosmetic threads. You had to click **Insert** > **Model Items** every time in the assembly drawing to display 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**.

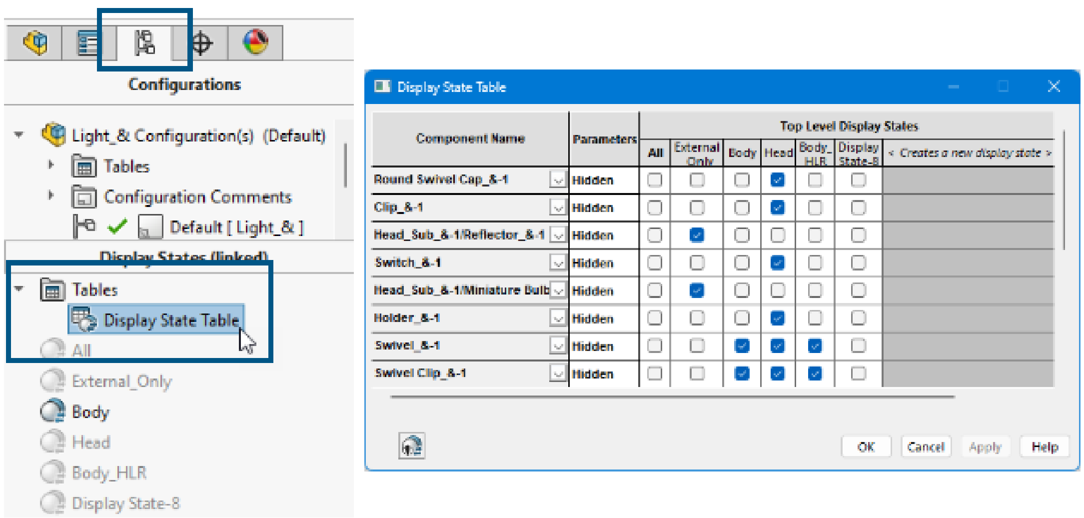
To view cosmetic threads in assembly drawings:

1. In the Drawing View PropertyManager, under **Import options**, select **Import annotations** and **Cosmetic threads**.
2. Click ✓.

13

Configurations

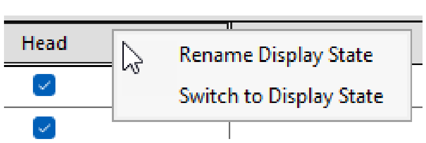
Display State Tables



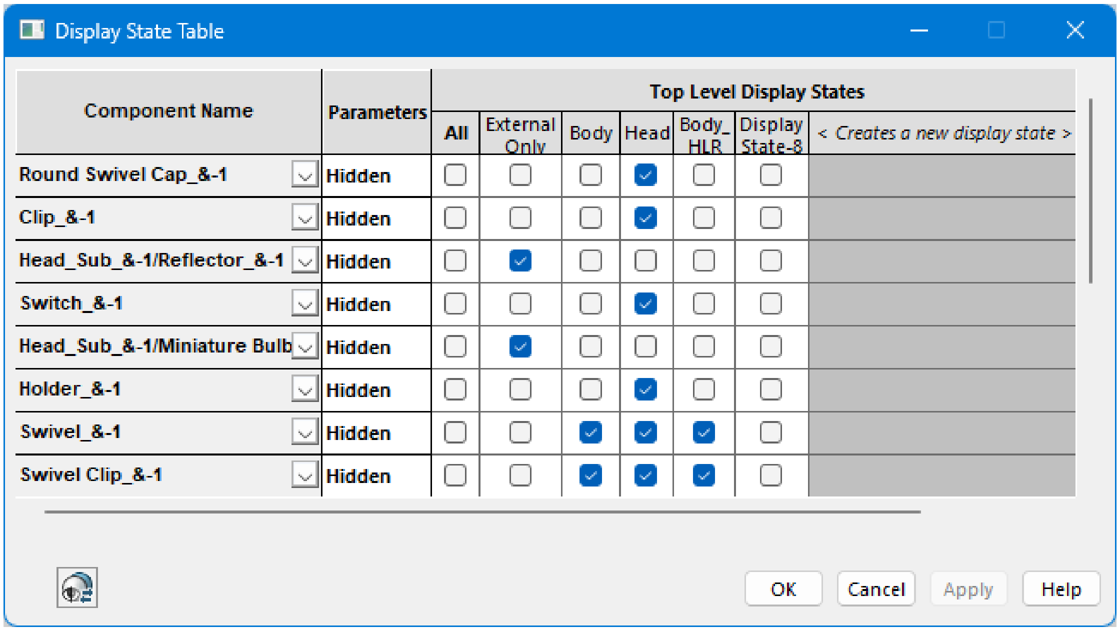
In assemblies with multiple display states, you can use the **Display State Table** to control the display states.

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**.



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.

14

Import/Export

This chapter includes the following topics:

- **Exporting Custom Properties to IFC Files**
- **Importing Extended Reality Files**

Exporting Custom Properties to IFC Files

```xml
<CustomPropertiesPSETMapping xmlns="http://www.solidworks.com/ifcpropertysets">
    <Schema Version="1" />
    <PropertySet Name="SOLIDWORKS Common Properties">
        <PropertyMapping SOLIDWORKS="SWProp01" IFC="SWProp01" Type="IfcLabel" />
    </PropertySet>
    <PropertySet Name="Weather Proofing">
        <AppliesTo ElementType="IFCROOF" />
        <PropertyMapping SOLIDWORKS="Cladding" IFC="Roof Covering Material" Type="IfcLabel" />
    </PropertySet>
    <PropertySet Name="Structural Cladding">
        <AppliesTo ElementType="IFCBEAM" />
        <AppliesTo ElementType="IFCCOLUMN" />
        <PropertyMapping SOLIDWORKS="Cladding" IFC="Structural Cladding Material" Type="IfcLabel" />
    </PropertySet>
</CustomPropertiesPSETMapping>
```

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:

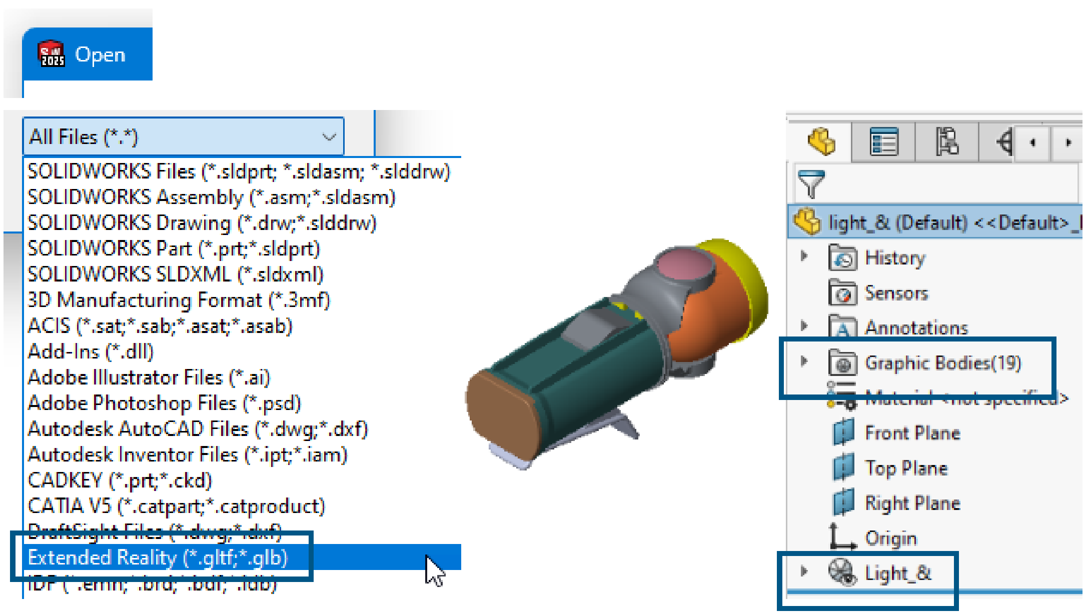
    - Proper tags, tag attributes, and tag structure.
    - The Schema Version is equal to or lower than the version supported by the current version of SOLIDWORKS.
    - SOLIDWORKS custom properties map one-to one or one-to-many IFC properties. You cannot map multiple SOLIDWORKS custom properties to the 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.

15

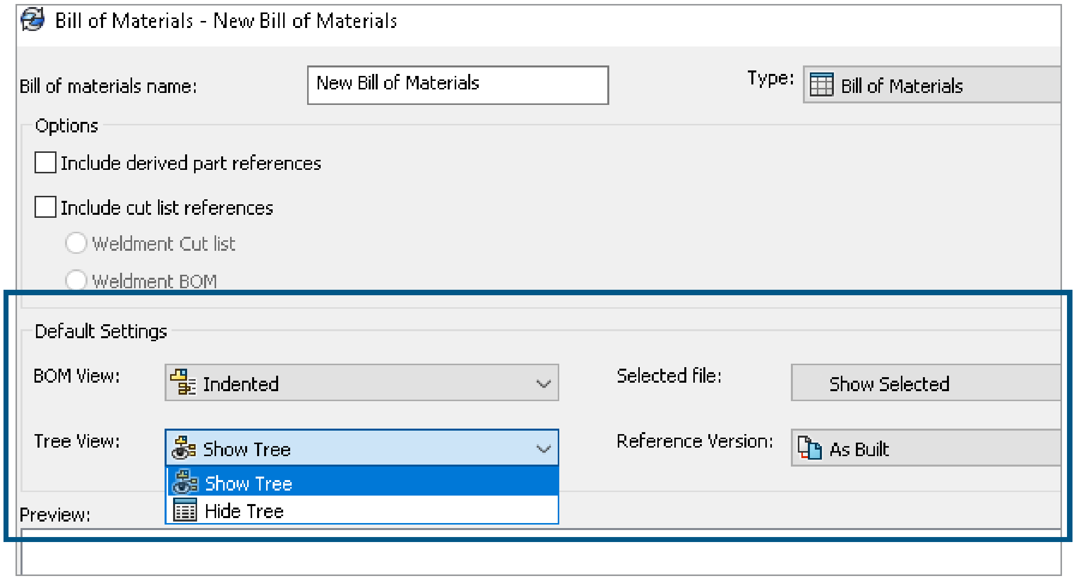
SOLIDWORKS PDM

This chapter includes the following topics:

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.

Default Settings for Computed BOM

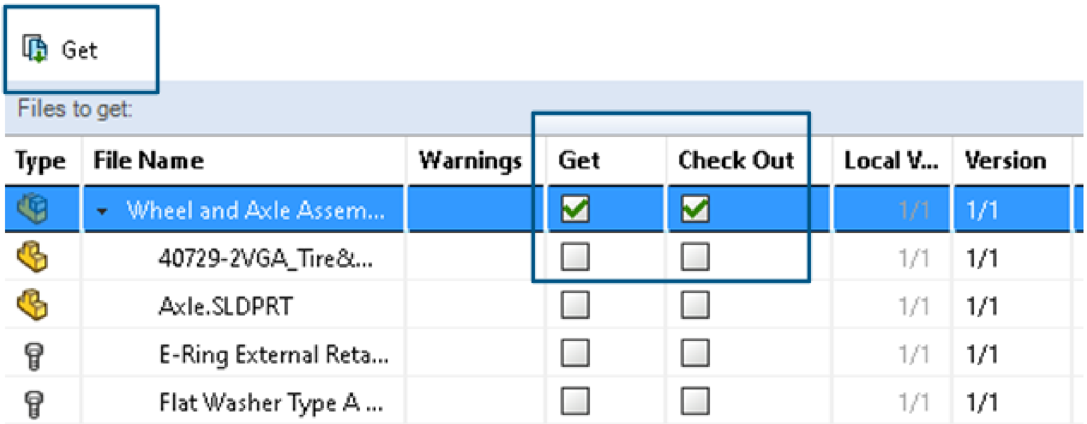


Administrators can specify the default view and options settings for the computed BOM while creating the Bill of Materials (BOM) in the SOLIDWORKS PDM Administration tool.

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



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.

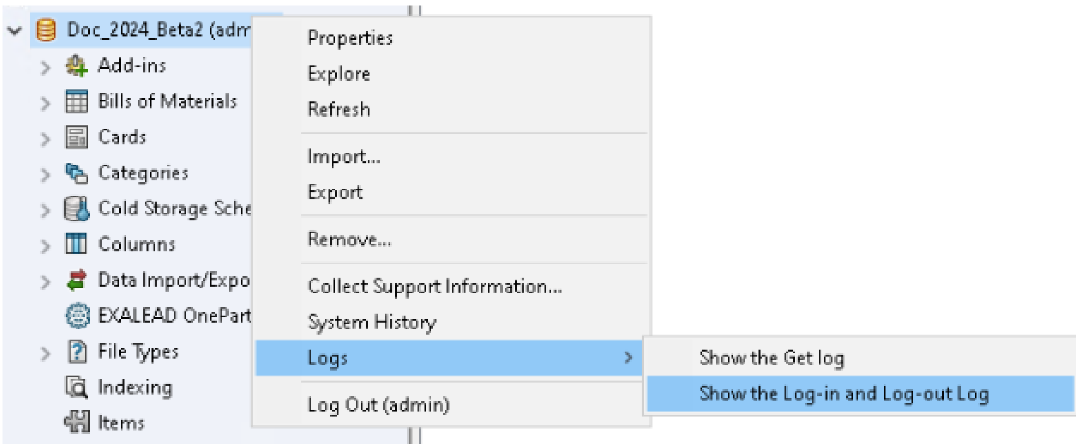
In the Get dialog box, when you select **Check Out** for single or multiple files, the **Get** option for these files is selected by default to perform both the operations at the same time. The combined **Get** and **Check out** operation simplifies your workflow.

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.

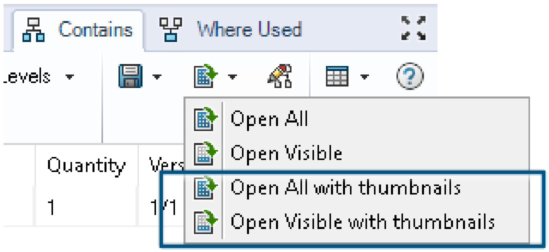
The authentication details include the user name, the date, and time when the user has logged in and out, and the SOLIDWORKS PDM client (desktop or Web2).

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

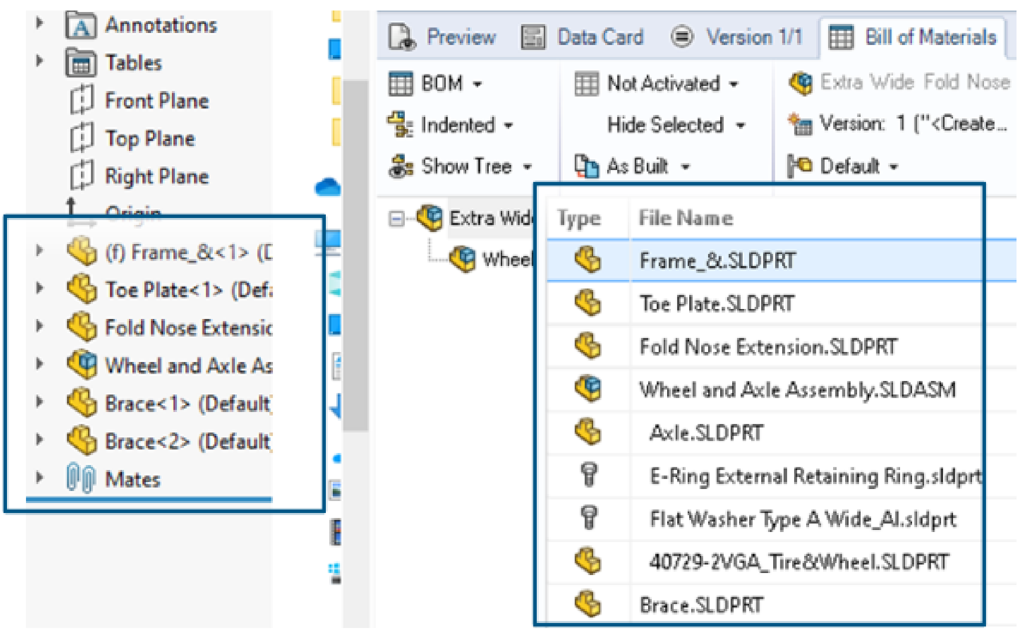


You can open the file data in the Microsoft® Excel® format along with a thumbnail preview in the Bill of Materials, Contains, and Where Used tabs of the SOLIDWORKS PDM File Explorer.

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.

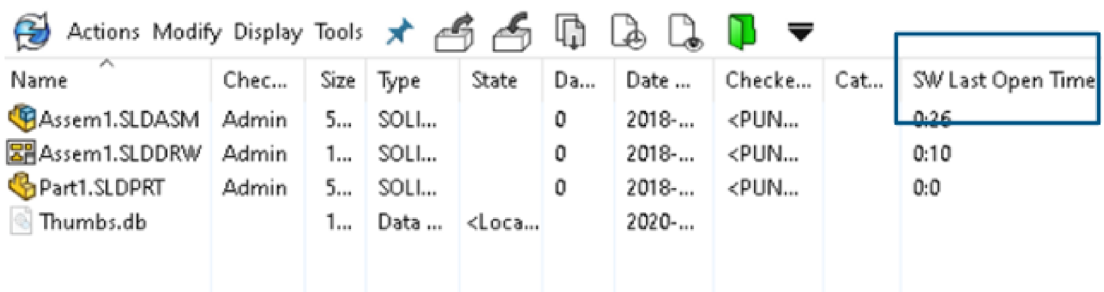
Viewing the FeatureManager Design Tree Order of Assembly Structure in Computed BOMs



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.

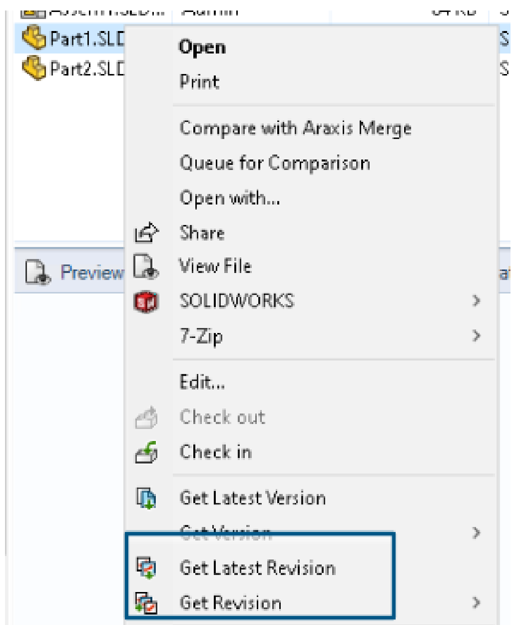
> The order of assembly components in the BOM for data already checked in to the vault does not change to match with the FeatureManager design tree.

Getting Information on Time Taken in Opening Files



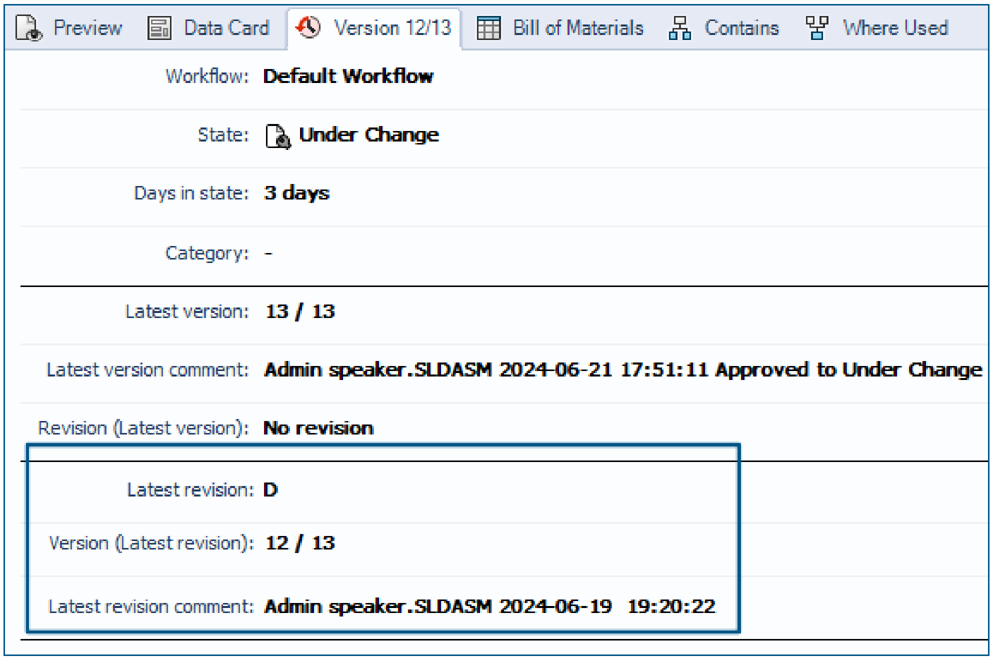
You can know the time taken to open a file when it was last opened in SOLIDWORKS 2023 and above. The time is measured in seconds. To know the file open time, a new variable **_SW_Last_Open_Time_** is added to the SOLIDWORKS PDM variables.

Getting Information on the Latest Revision

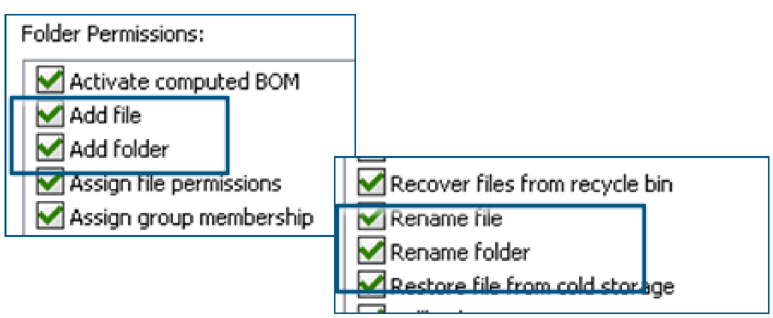


You can get the latest revision for a file in SOLIDWORKS PDM. For getting the latest revision, a **Latest Revision** variable is added to the existing system variables.

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.



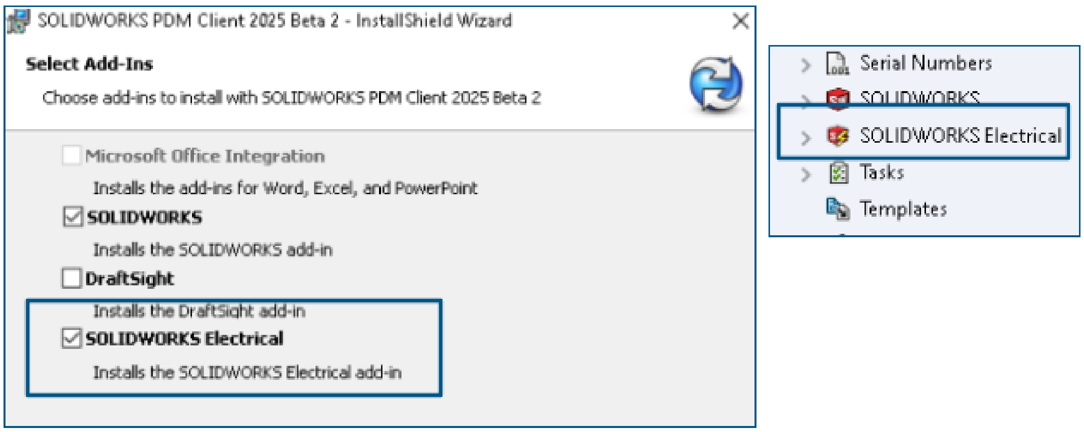
Separate Add or Rename Permissions for Files and Folders



The existing **Add or rename file** and **Add or rename folder** permissions are split into separate permissions for add and rename.

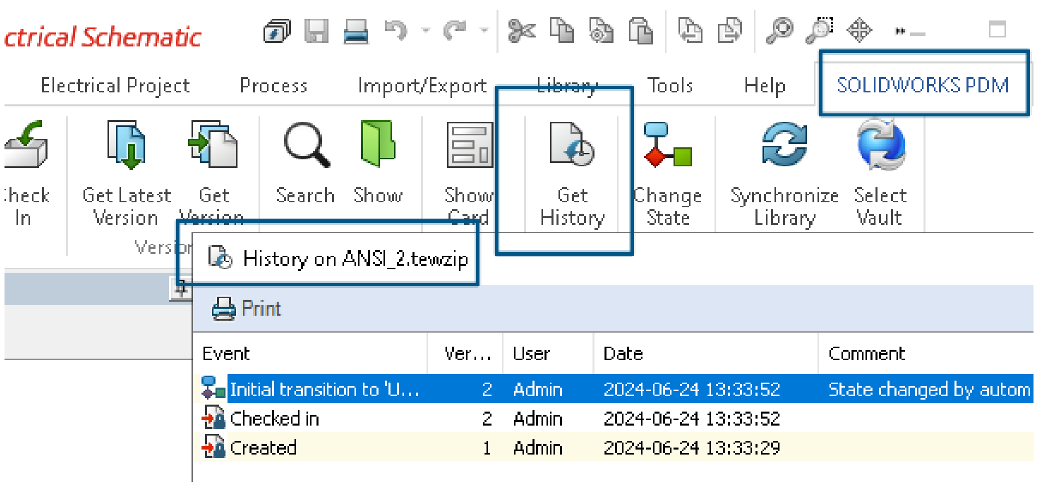
Administrators can use **Add file** and **Rename File** under **Folder Permissions** and **State Permissions** in the SOLIDWORKS PDM Administration tool.

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.

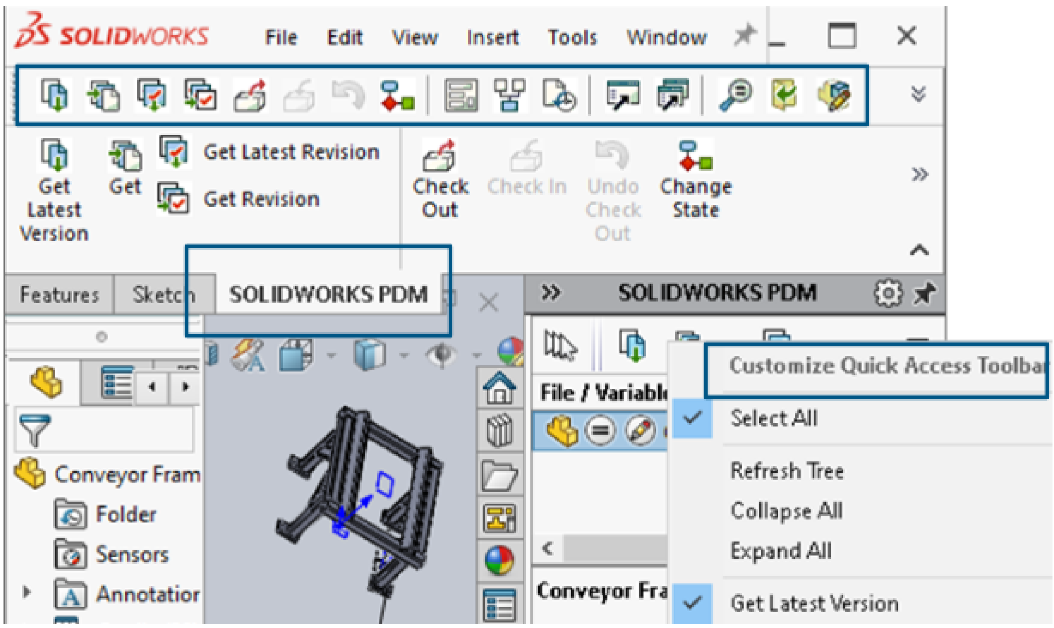


The **History** option is added to the SOLIDWORKS PDM CommandManager options. You can see the history of SOLIDWORKS Electrical projects using this option for better monitoring of the changes.

File Check in Performance

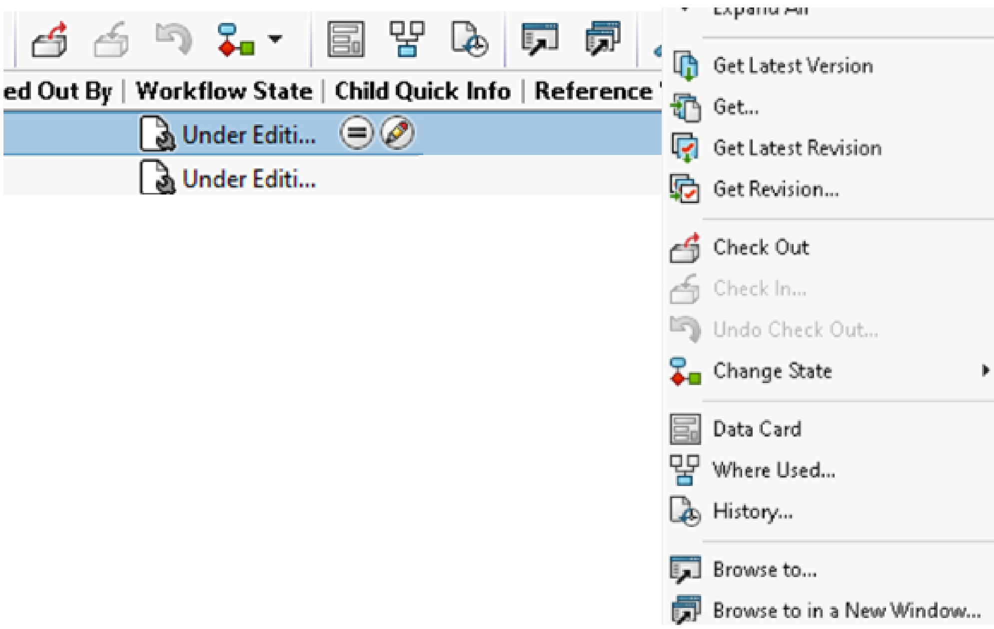
SOLIDWORKS PDM performance is improved during the file check in to the SOLIDWORKS PDM database. The file check in operation is two times faster than before.

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.

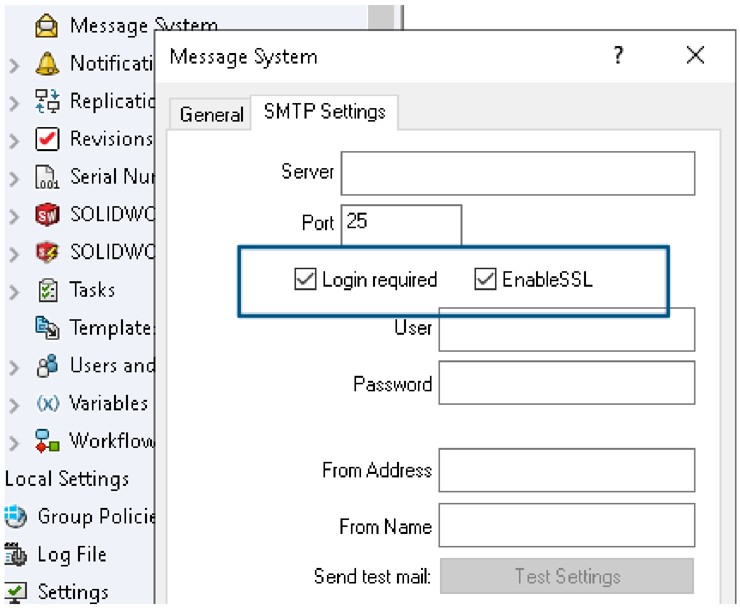
For example, the following are the options added:

- **Browse to** : Opens the selected file in the same SOLDWORKS PDM File Explorer window.
- **Browse to in a New Window** : Opens the selected file in a new SOLDWORKS 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



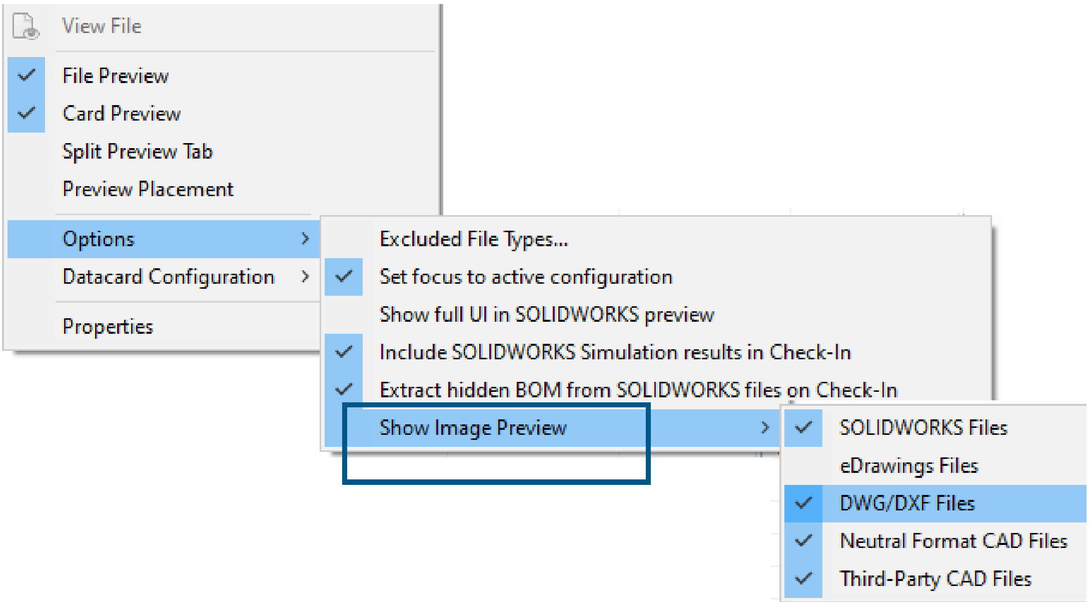
You can enable Secured Socket Layer (SSL) or TLS (Transport Layer Security) authentication in the SMTP email notification.

In the SOLIDWORKS PDM Administration tool, you can select **EnableSSL** under **Message System** > **SMPT** > **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
Gmail®	smtp.gmail.com
Outlook®	smtp.outlook.com
Microsoft 365®	smtp.office365.com
Yahoo®	smtp.mail.yahoo.com

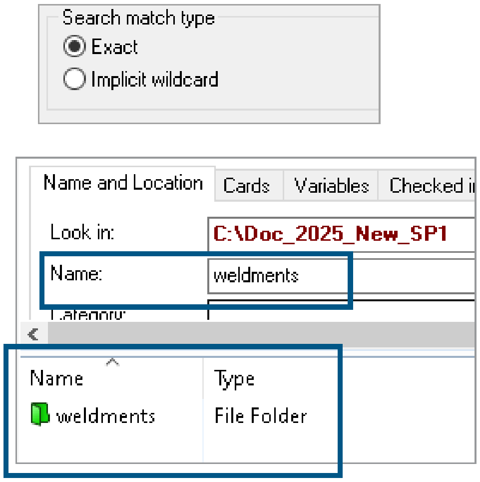
Display Options - Show Image Preview (2025 SP1)

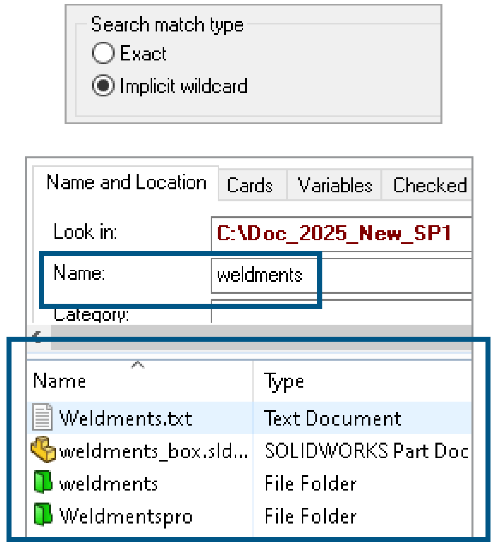


In the SOLIDWORKS PDM File Explorer, you can view a thumbnail or a full preview in the Preview tab based on the following file types using **Display** > **Options** > **Show Image Preview**:

- **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:

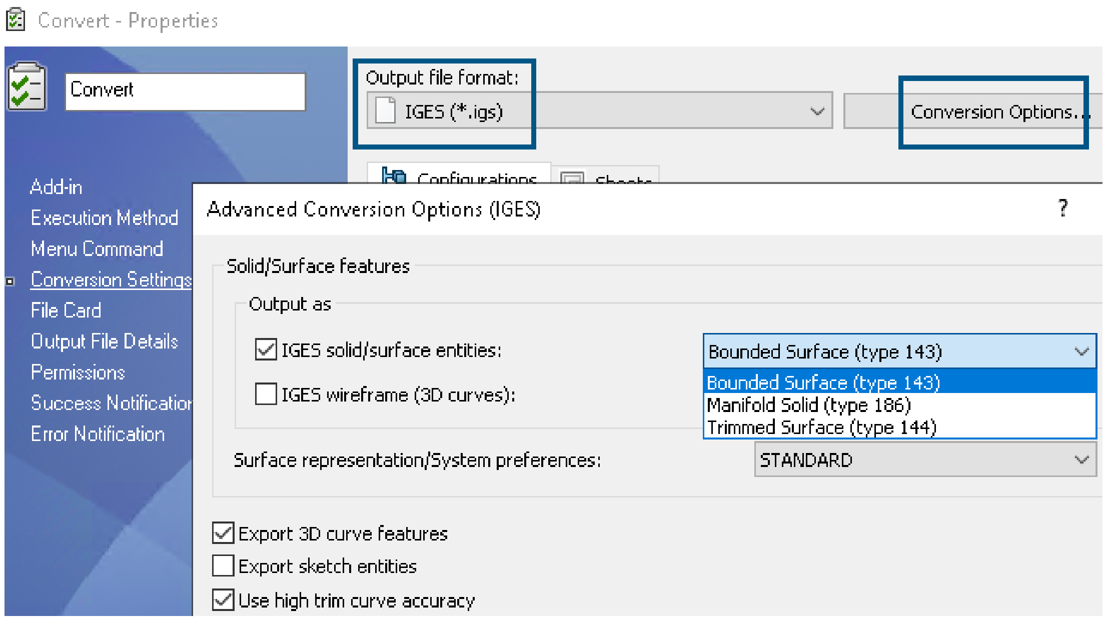
- **Exact**: You can search for files, folders, and variables in the SOLDWORKS PDM File Explorer whose search results exactly match the search input.

  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 SOLDWORKS PDM File Explorer whose search results include the search input.

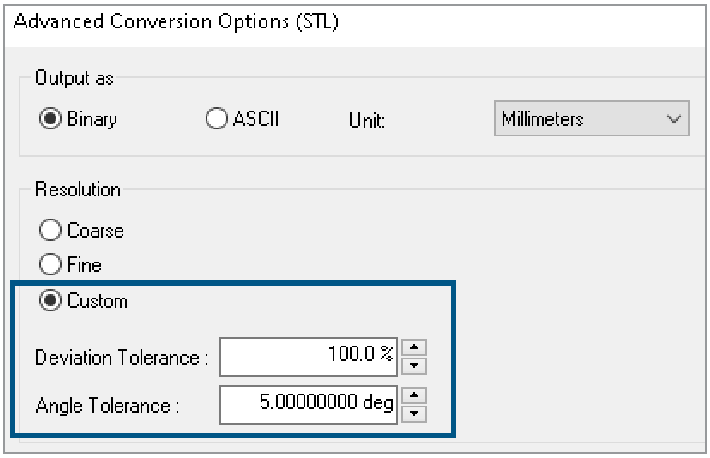
  For example, if you search for `weldments` in **Name**, the search results include all files, folders, or variables whose names include `weldments` (for example, `weldments`, `weldments_box`, and `weldmentspro`).

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.

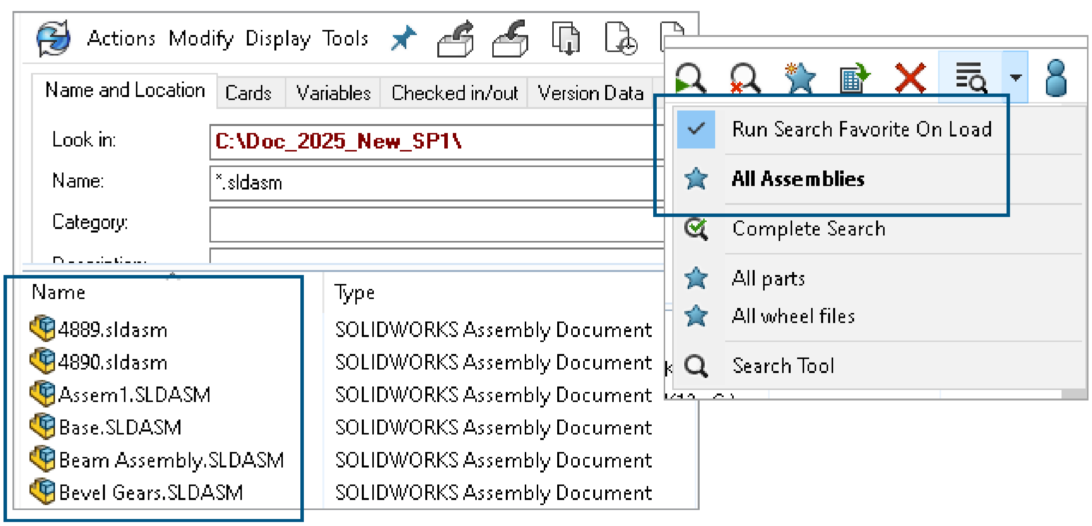
Output File Format	Advanced Conversion Options
IGES (*.igs)	**Bounded Surface (type 143)**: Select to convert the faces of the part, assembly, or the selected surfaces and its boundaries defined by other IGES entities, for example, curves and edges.
STL (*.stl)	**Custom** option under **Resolution** with the following sub options: • **Deviation Tolerance**: Controls whole-part tessellation. Lower numbers generate files with greater whole-part accuracy. • **Angle Tolerance**: Controls small-detail tessellation. Lower numbers generate files with greater small-detail accuracy, but those files take longer to generate.



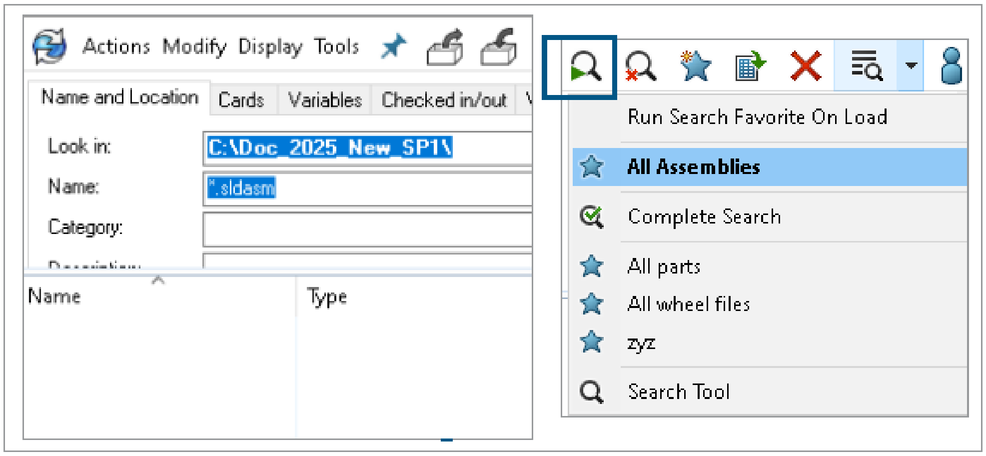
You can access these options under **Tasks** > **Convert** > **Open** > **Conversion Settings** > **Conversion Options**.

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)

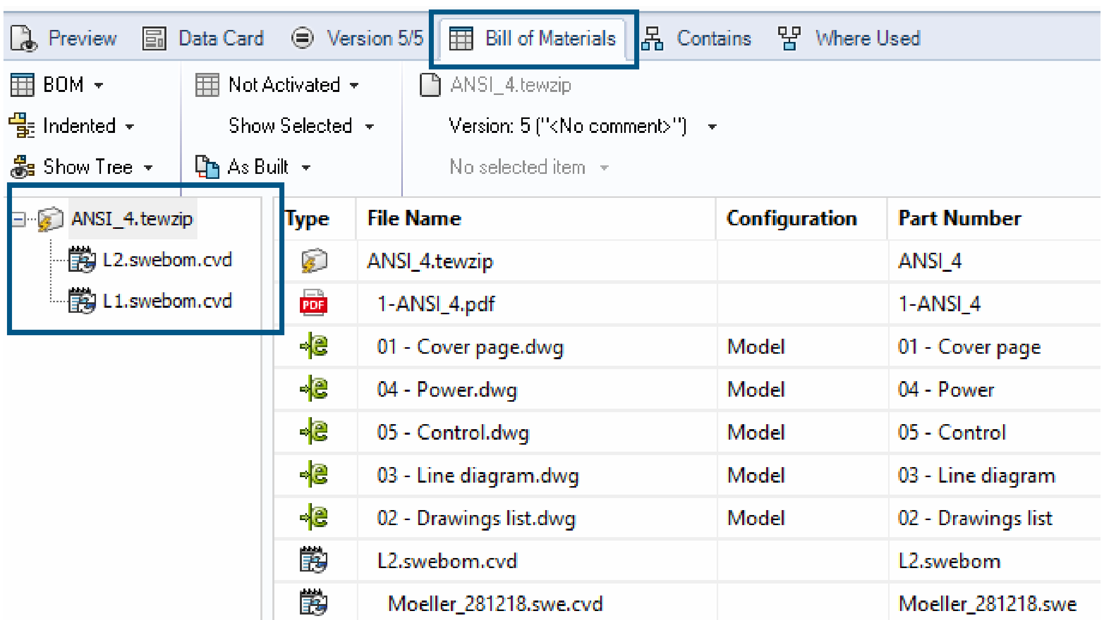


In the SOLIDWORKS File Explorer, you can use the **Run Search Favorite on Load** functionality to view Search Favorite results for files and folders by selecting the Search Favorite. The functionality is available with the integrated search and the **Search Tool**.



When this option is not selected, you can view the Search Favorite results by selecting the Favorite Search and clicking **Start Search**.

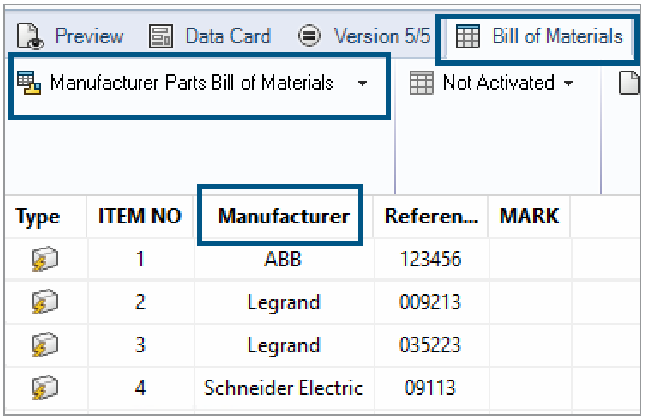
Electrical Assembly Bill of Materials (2025 SP1)



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.


Preview
Data Card
Version 5/5
Bill of Materials
Manufacturer Parts Bill of Materials
Not Activated
Type
ITEM NO
Manufacturer
Referen...
MARK
1
ABB
123456
2
Legrand
009213
3
Legrand
035223
4
Schneider Electric
09113

16

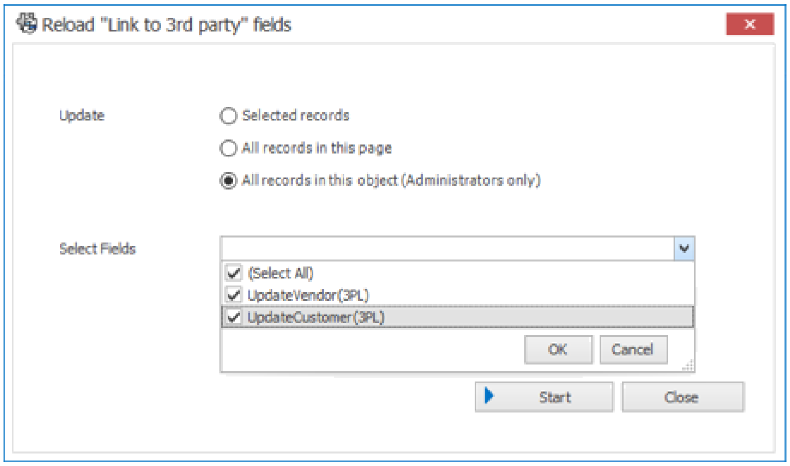
SOLIDWORKS Manage

This chapter includes the following topics:

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

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

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.

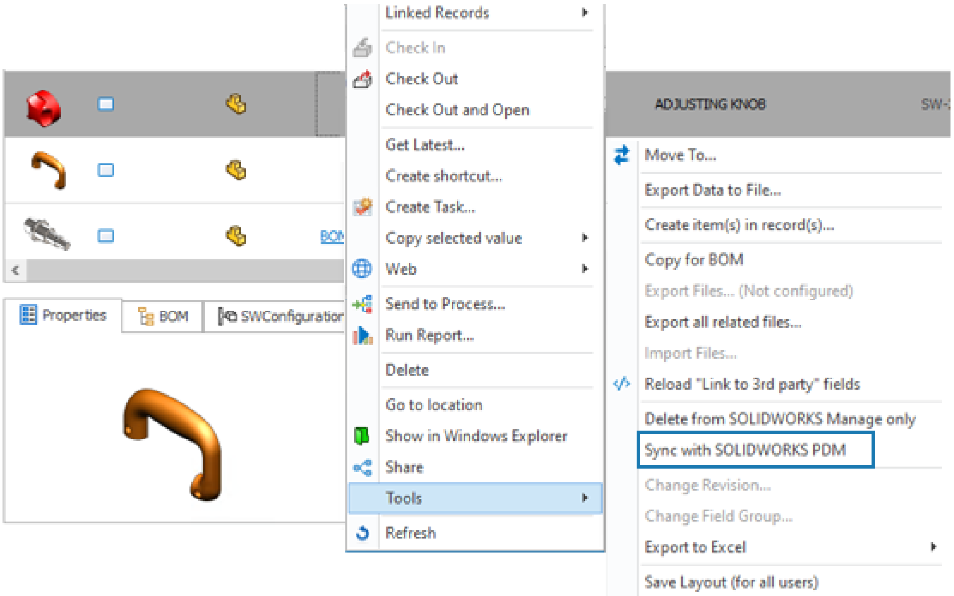
Nonadministrators can update the values for selected records in the Main Grid or for all records on a page. Administrators can update the values for all records in an object. This restricts users from affecting system performance if there are many fields or ones with complex queries.

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

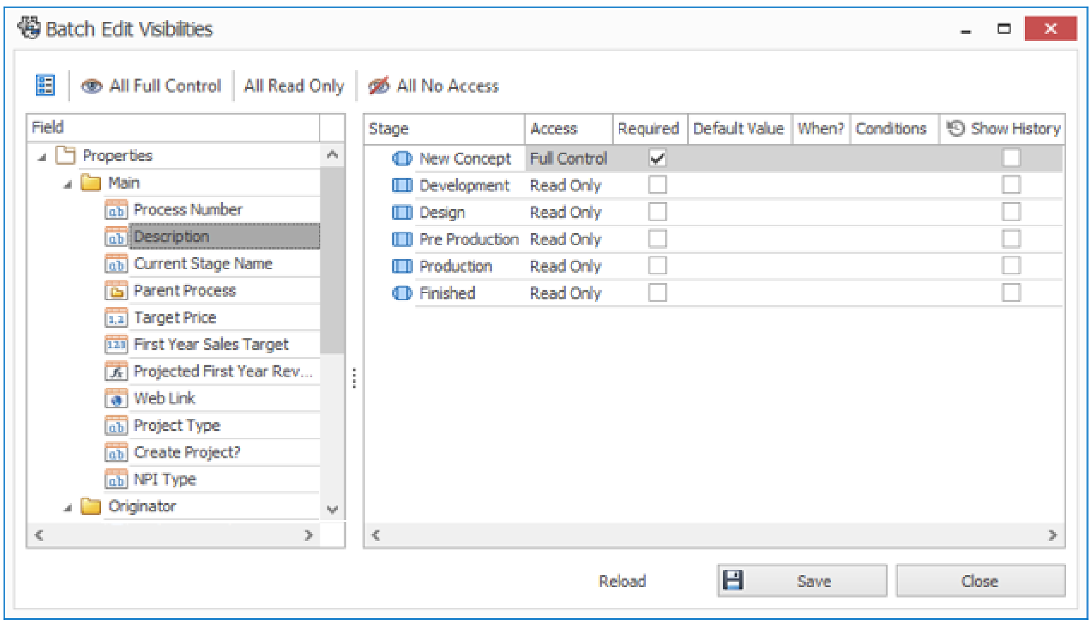
1. In the System Administration tool, right-click a process and click **Administration**.
2. In the Process Wizard, on the Fields page, specify a `Date` field to contain the date used to send the notification.

   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

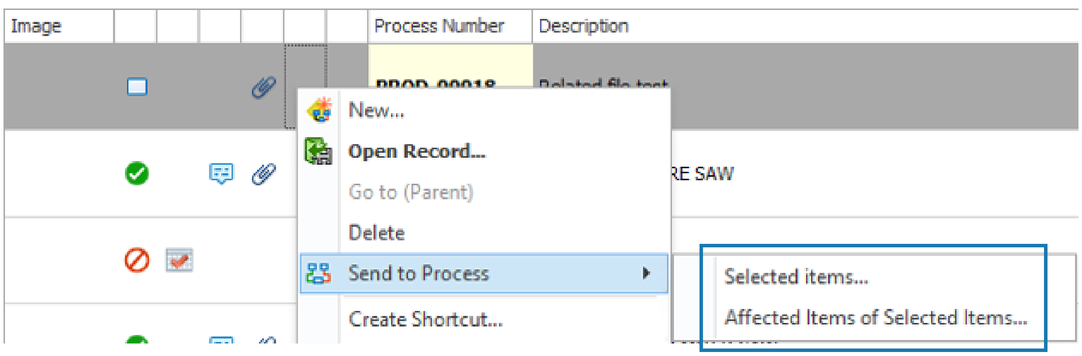
1. In the System Administration tool, right-click a process and click **Administration**.
2. On the Workflow Properties page:
   a) Select a stage.
   b) Select the **Visibility** node.
   c) Click **Batch Edit**.

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

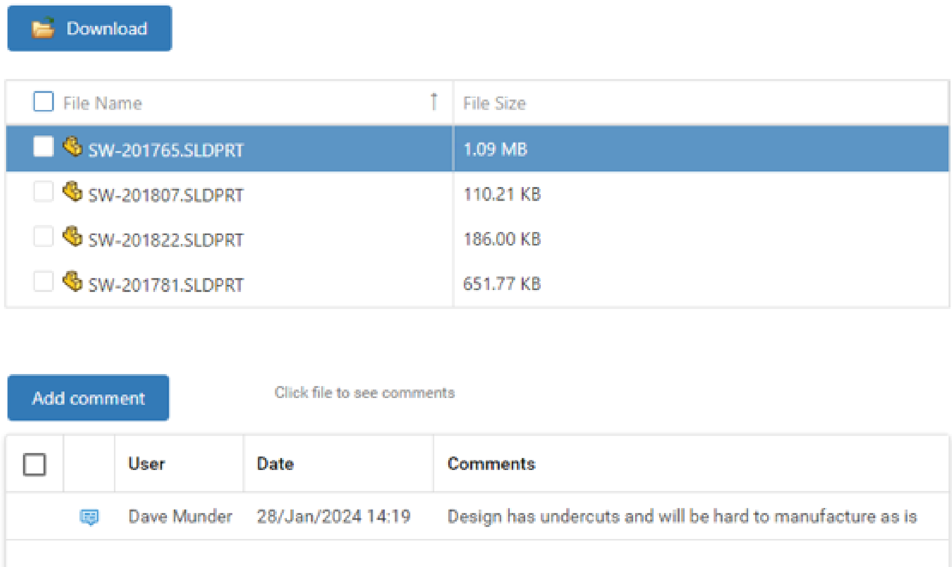


You can send affected items from selected processes to new processes.

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

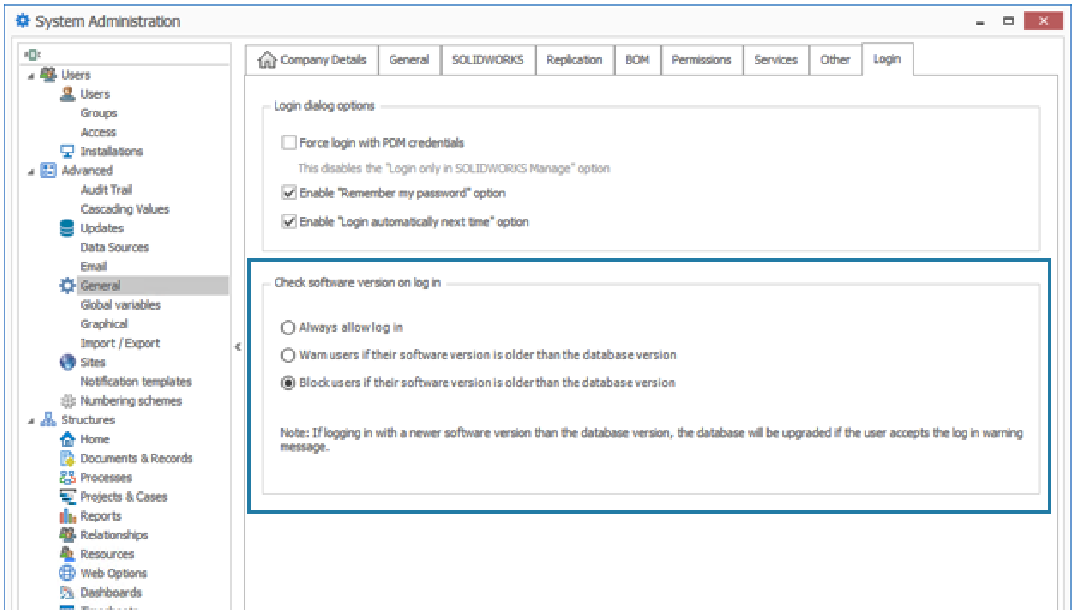


You can specify an option that allows the participants of file sharing to add comments to each file. This makes it easier to communicate with external users about the shared files.

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.

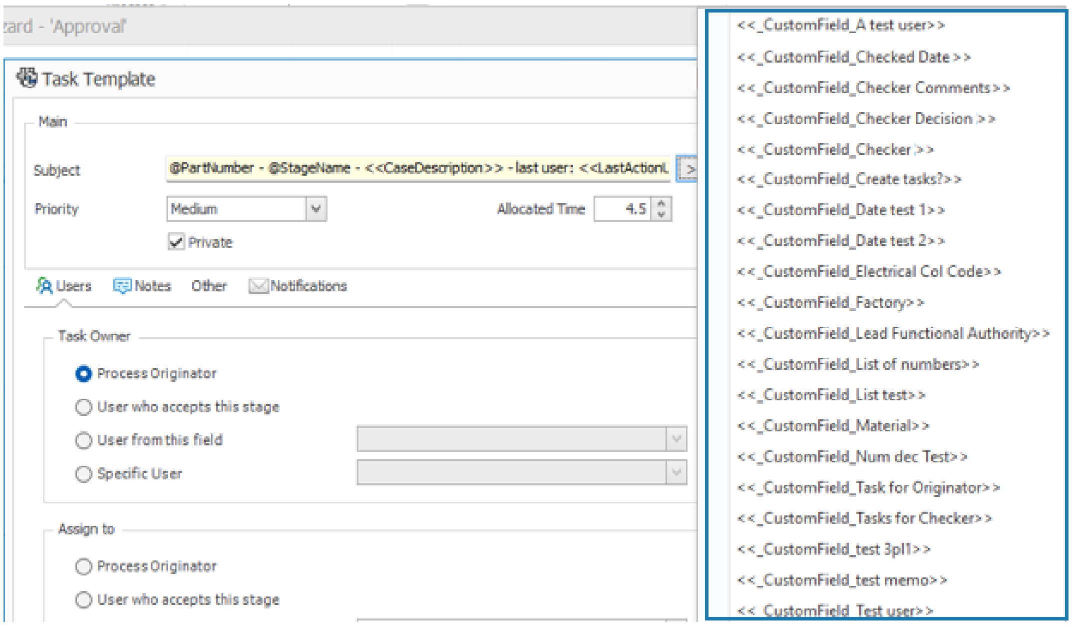
Grouping Instances in Flat BOMs

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



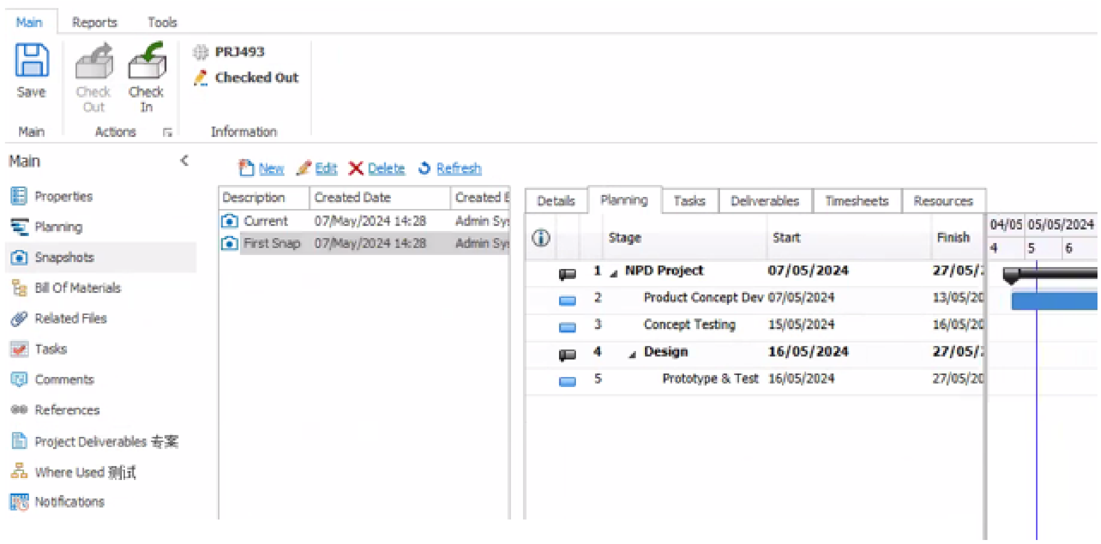
You can include field values from an associated object in the task subject. This makes task subjects associated with Project, Process, and Case objects more meaningful to users.

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.

17

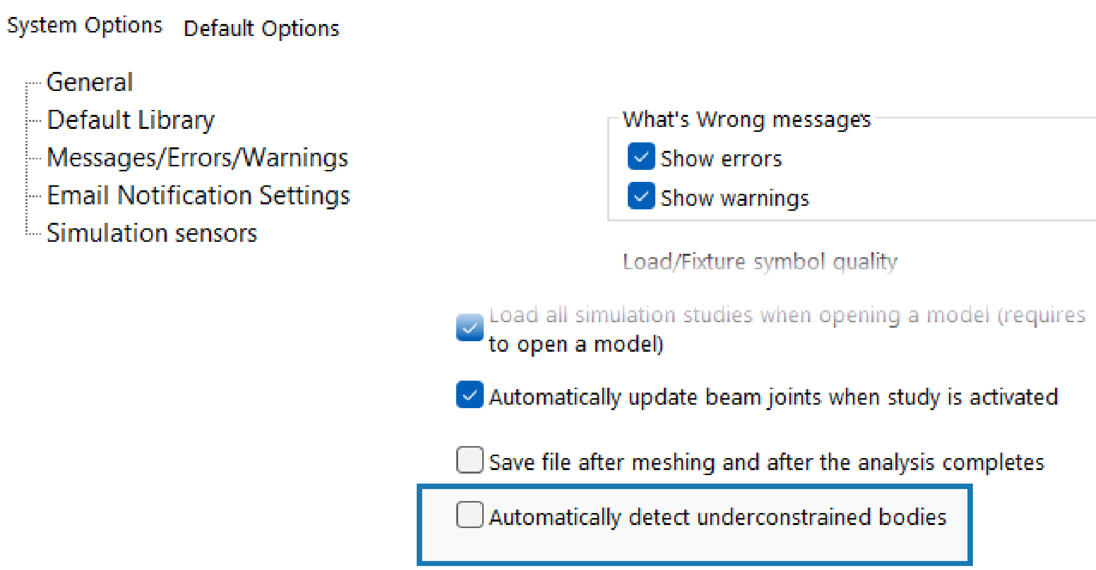
SOLIDWORKS Simulation

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



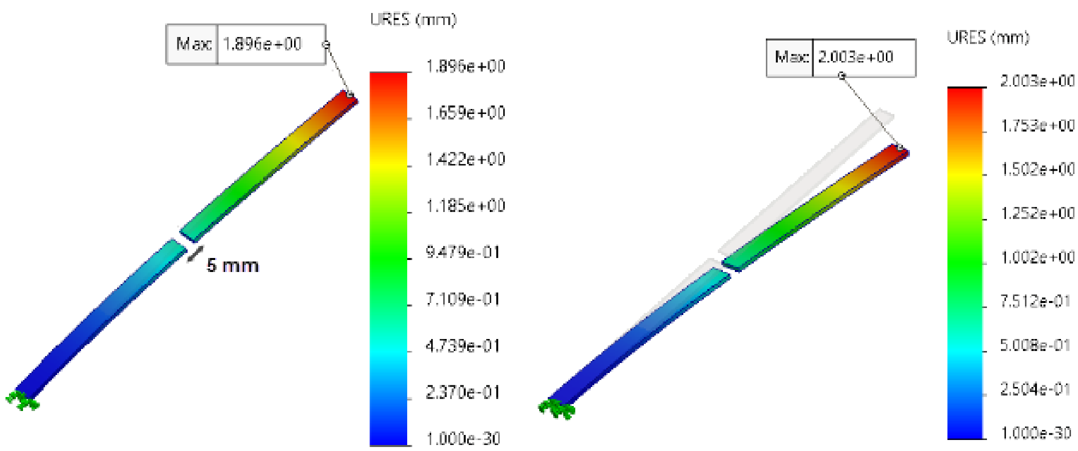
You can detect rigid body modes at the early stage of a linear static study's solution.

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

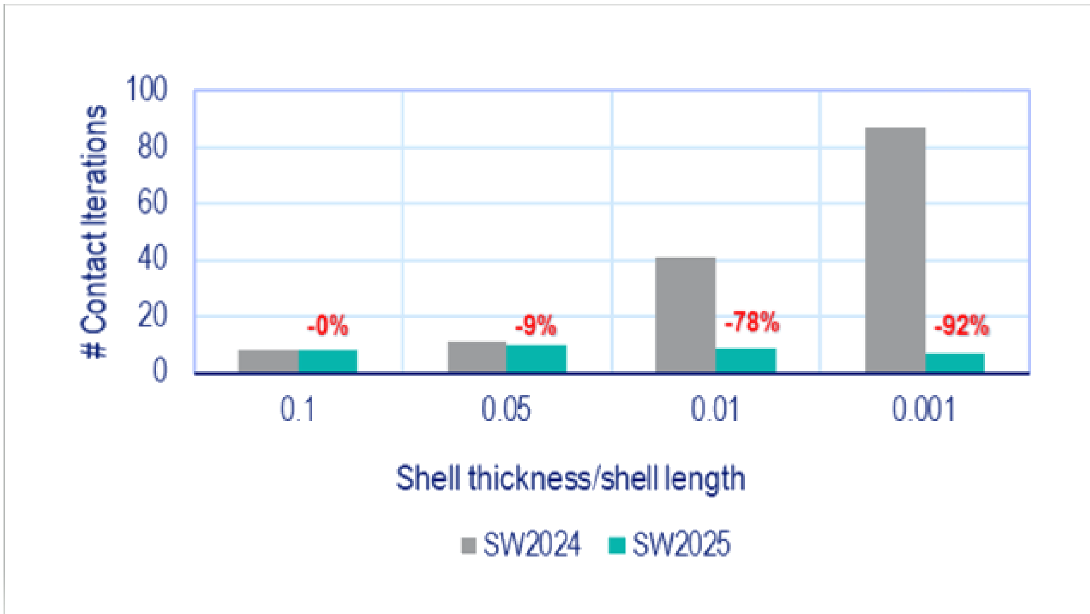


2024 2025

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

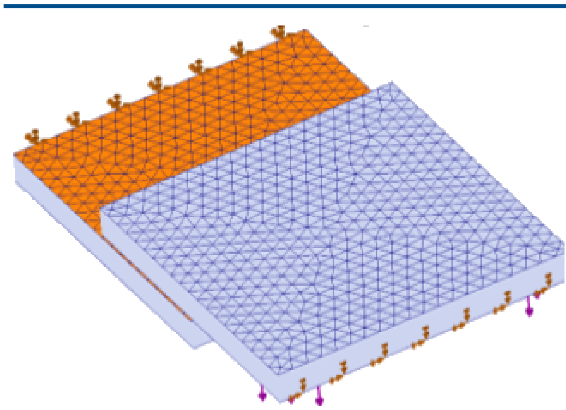


A new algorithm is introduced to apply penalty stiffness for contact interactions of shells. This enhancement improves performance and accuracy for a large range of shell thickness ratios.

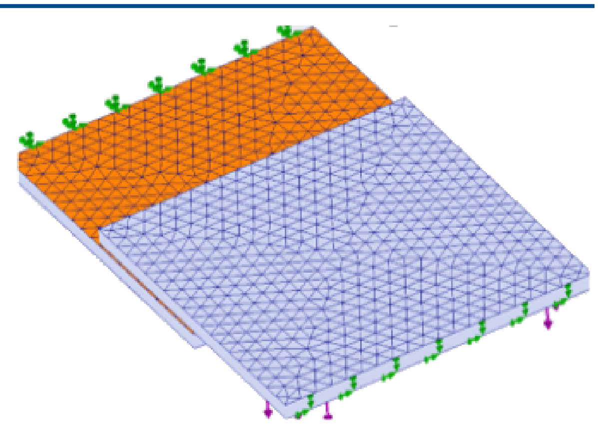
> Shell thickness ratio = shell thickness / characteristic length of shell

The image shows the performance improvement for contact interactions depending on several shell thickness ratios.

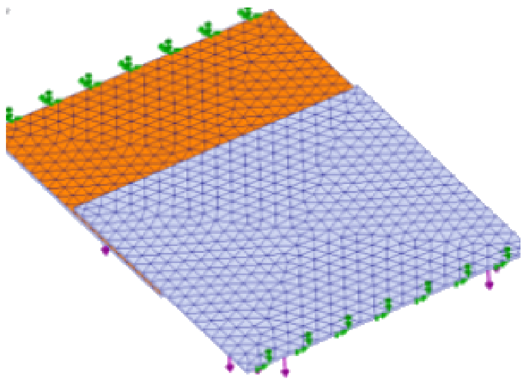
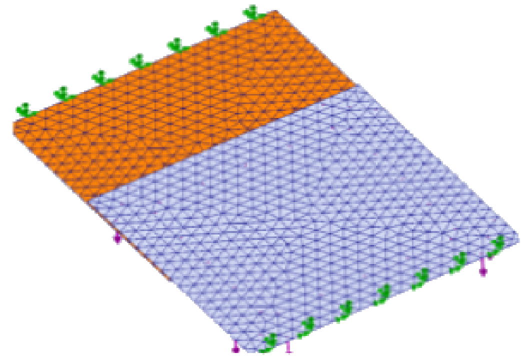
> The force magnitude applied for each test case was adjusted for the different shell thickness/shell length ratios to maintain a small displacement range and similar maximum displacements across all test cases.



shell thickness / shell length = 0.1

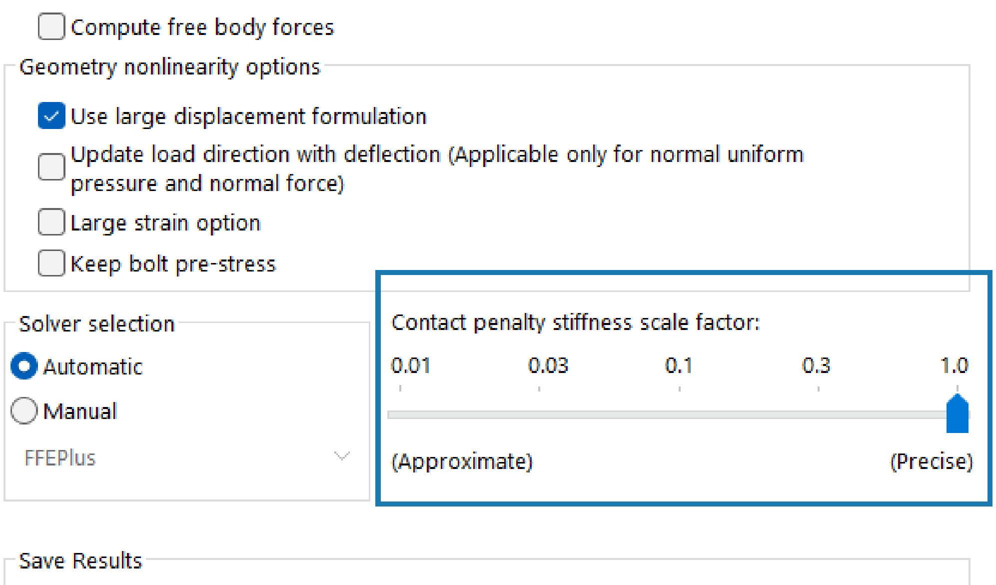


shell thickness / shell length = 0.05

	
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

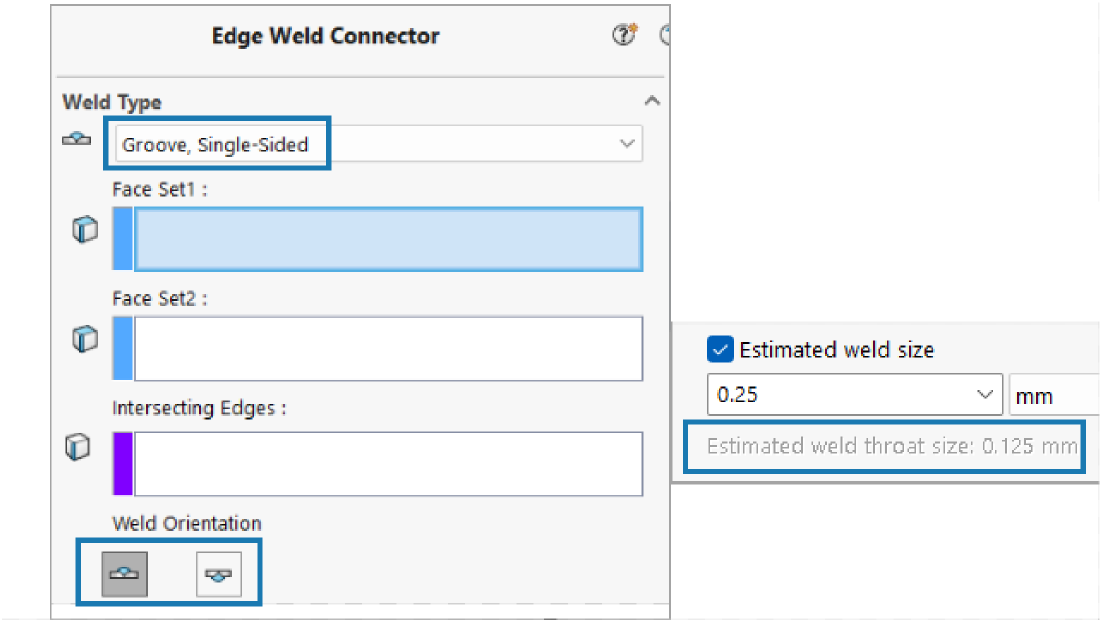


You can adjust a scale factor for the penalty stiffness applied to surface-to-surface contact interactions when solving nonlinear studies.

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

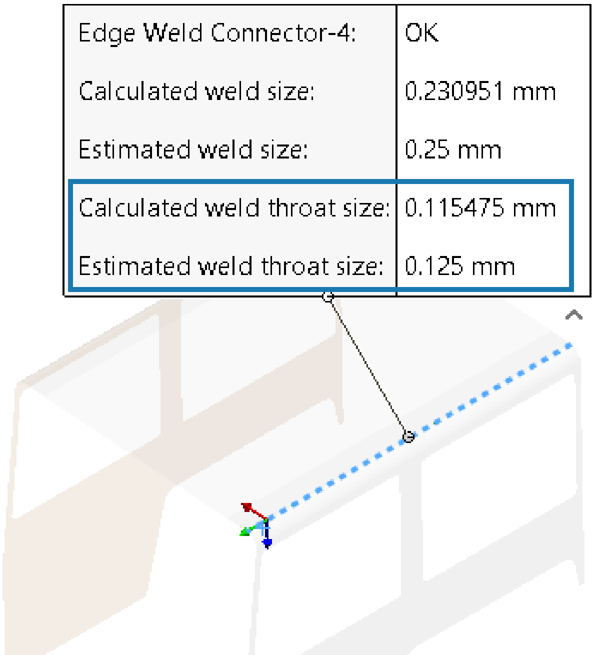


Several enhancements for the Edge Weld connector improve its usability.

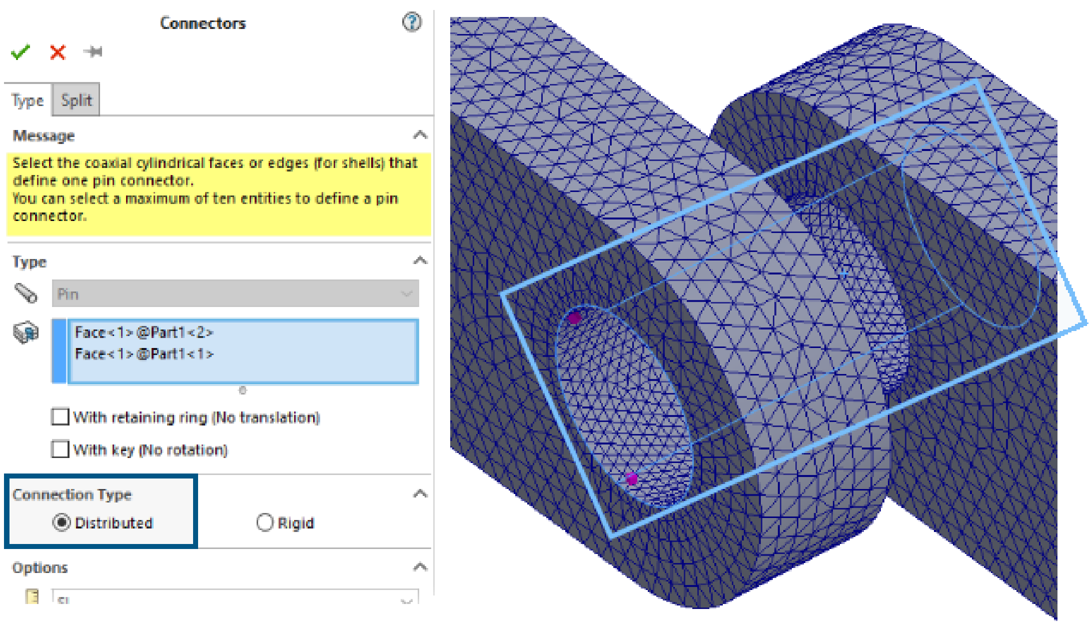
- The program calculates the **Estimated weld throat size** when you define edge weld connectors in the Edge Weld Connector PropertyManager. The formulas for the calculation of the **Estimated weld throat size** are given in the table.

Weld Type	Estimated Weld Throat Size
Fillet	**Estimated weld size** * square root (2) / 2
Groove	**Estimated weld size** / 2

- 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.



Enhanced Pin Connector



The introduction of a distributed coupling algorithm enhances the performance of studies that use pin connectors.

Results from studies with pin connectors that you apply to cylindrical surfaces with large number of nodes and use the **Distributed** connection are more accurate.

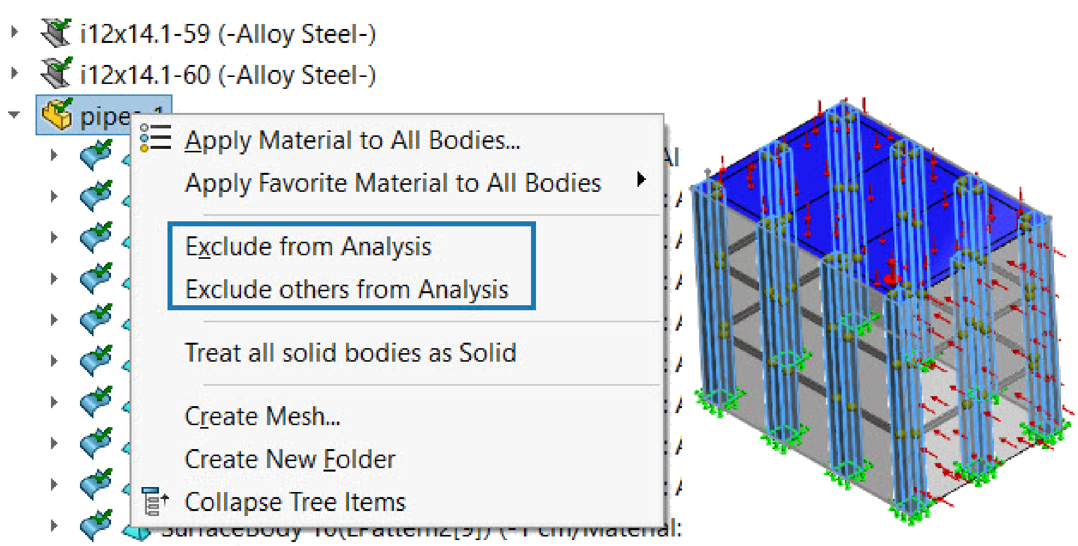
The solution time for these studies is improved for the Intel Direct Sparse solver.

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.

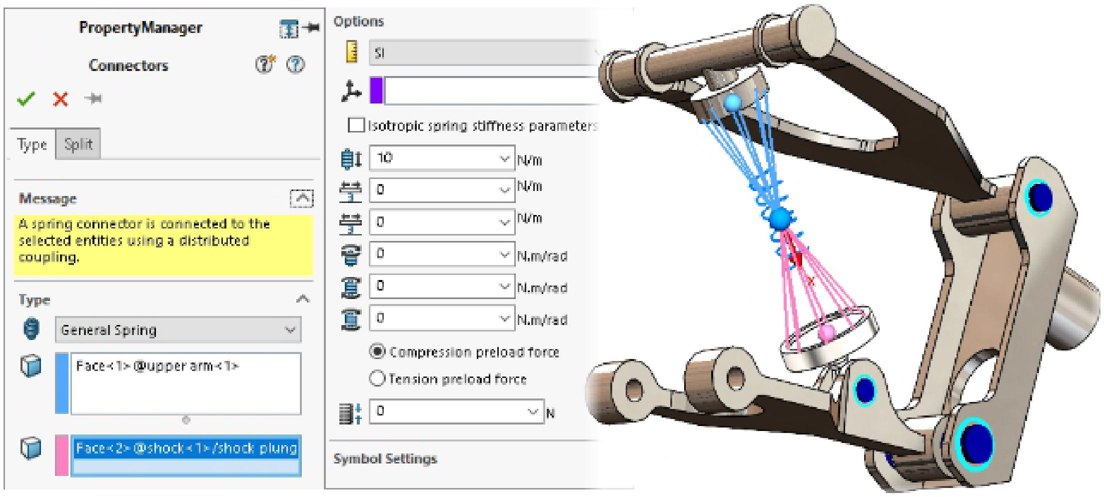
Exclude Bodies from Analysis



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.

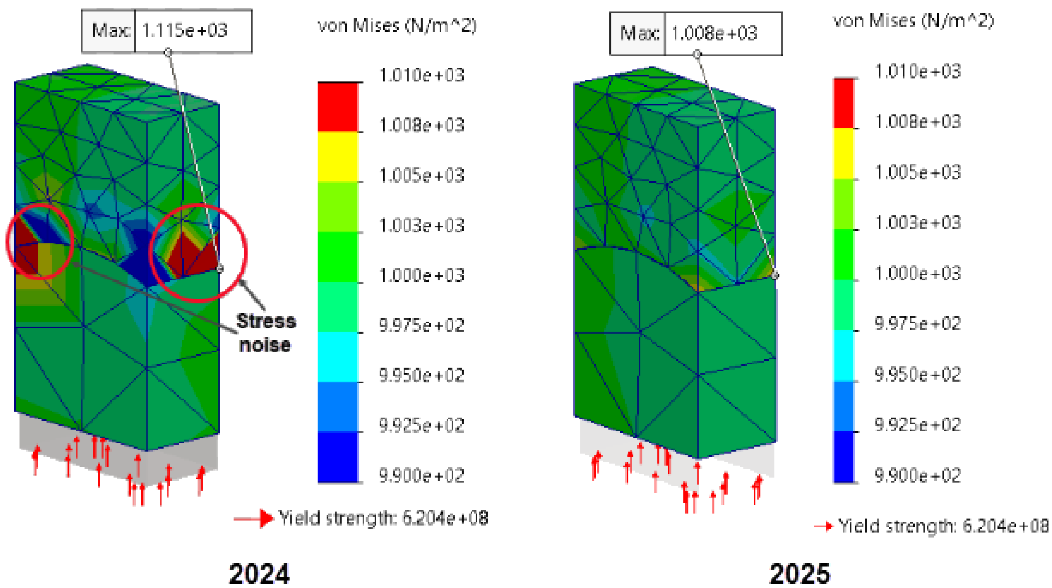
To accurately represent the general spring connector, you can define up to six stiffness parameters using a local coordinate system.

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** .

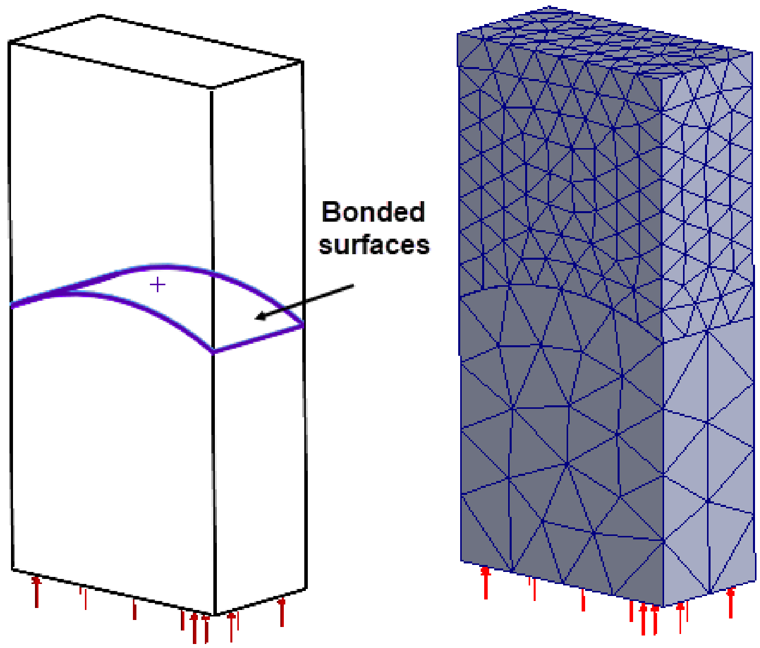
Geometry Correction for Surface-to-Surface Bonding



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.

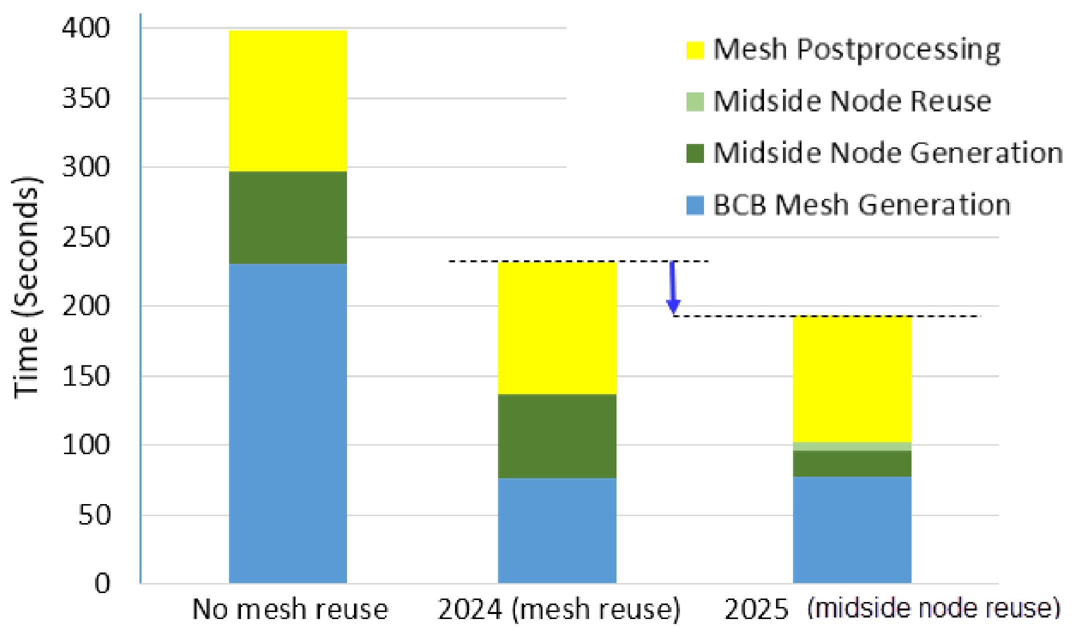
The algorithm that enforces surface-to-surface bonding integrates geometry correction factors that improve the representation of curved surfaces of cylindrical, spherical, and conical geometries. The integration of surface geometry correction reduces the stress noise at the vicinity of bonded curved 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.

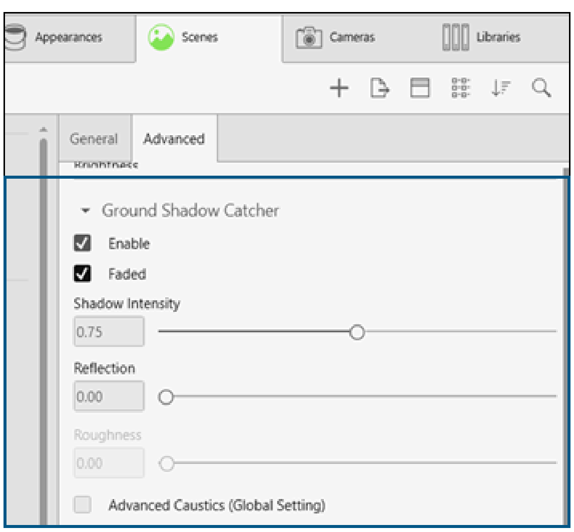
18

SOLIDWORKS Visualize

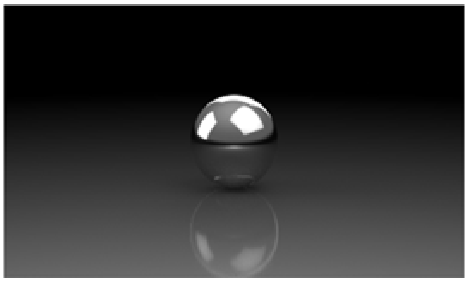
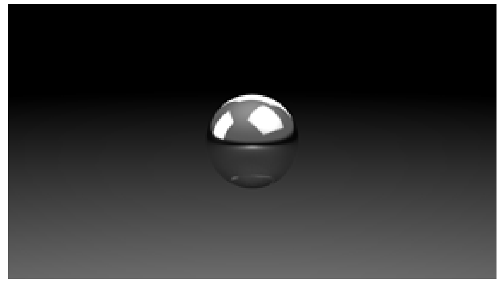
This chapter includes the following topics:

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.

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.

	
Enabled, Not Faded	Enabled, Faded

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**.

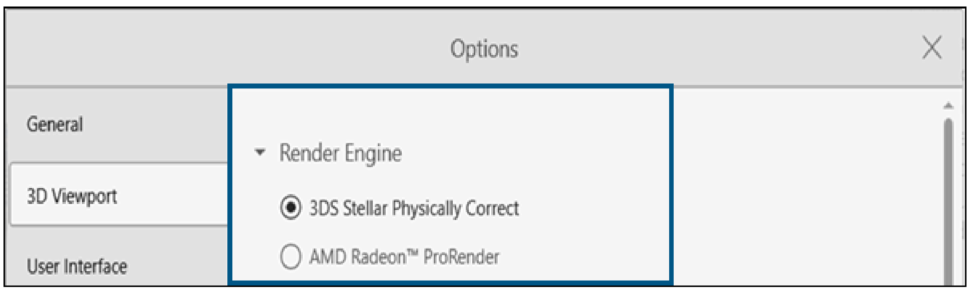
> This feature is supported exclusively in **Accurate** mode and is not accessible in **Preview** or **Fast** mode.

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.

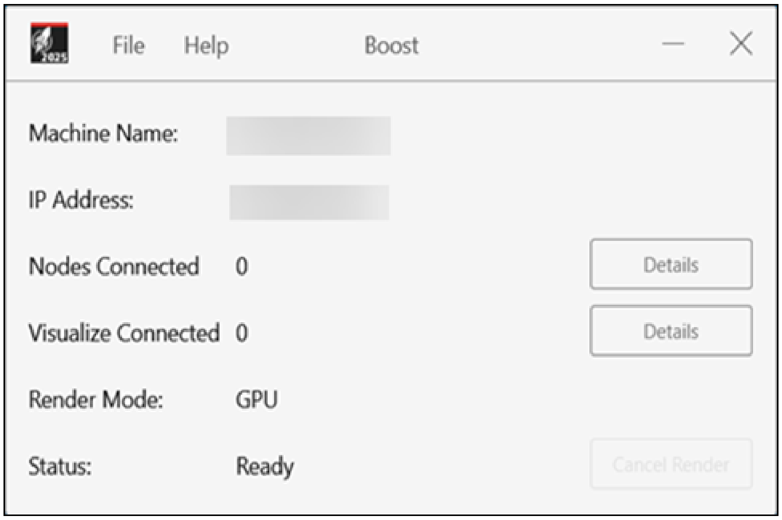
Photorealistic Rendering in SOLIDWORKS with the SOLIDWORKS Visualize API

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.

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.

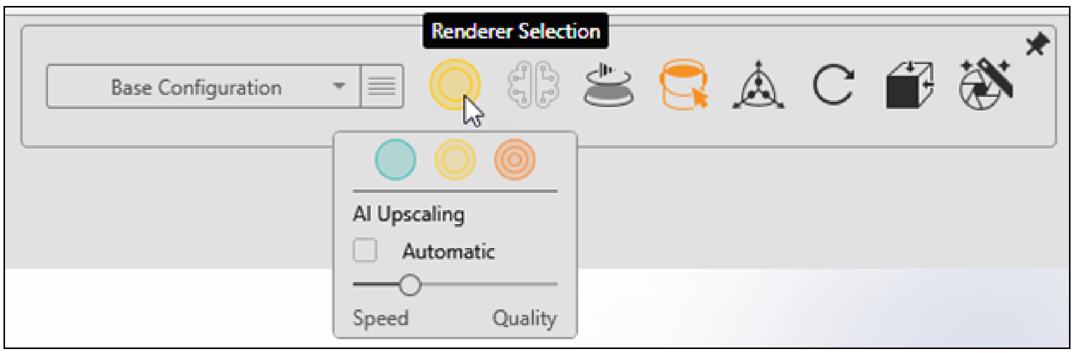
Enhancing Images with the Camera Bokeh Effect (2025 SP1)

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)



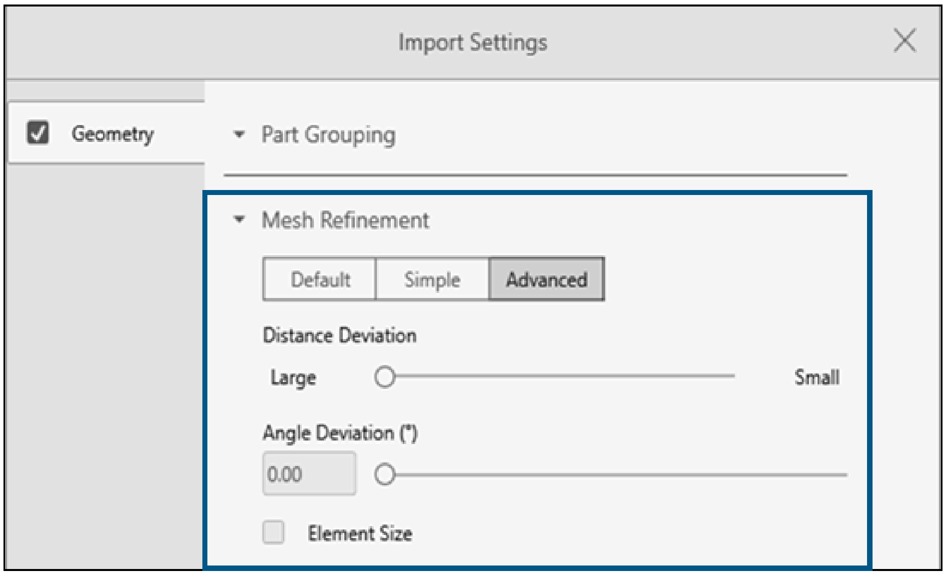
The latest updates to **Fast** mode for the Stellar render engine enhance performance, usability, and access to critical settings.

- **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 retessallate 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 retessallate 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.

19

SOLIDWORKS CAM

This chapter includes the following topics:

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

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

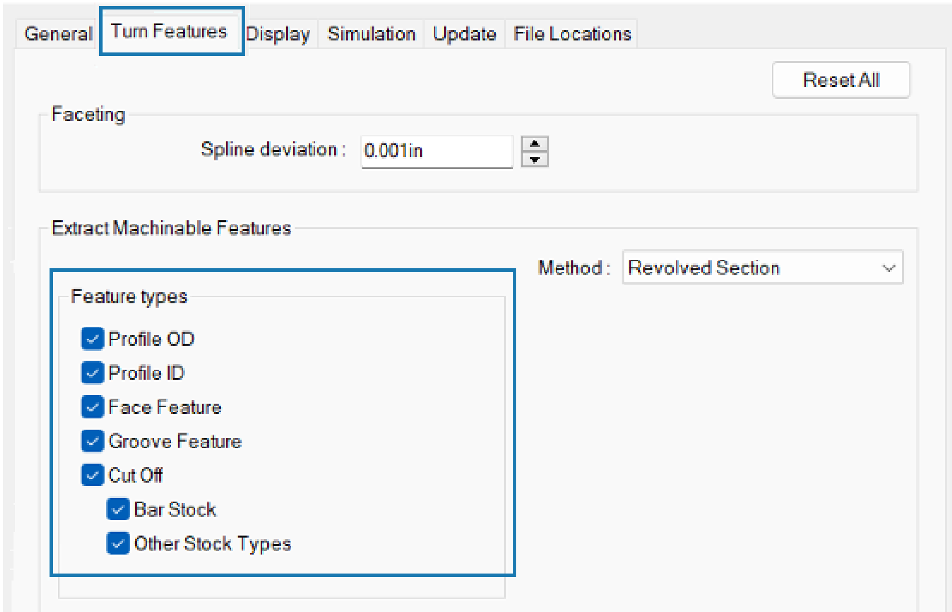
This option helps when machining:

- 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.

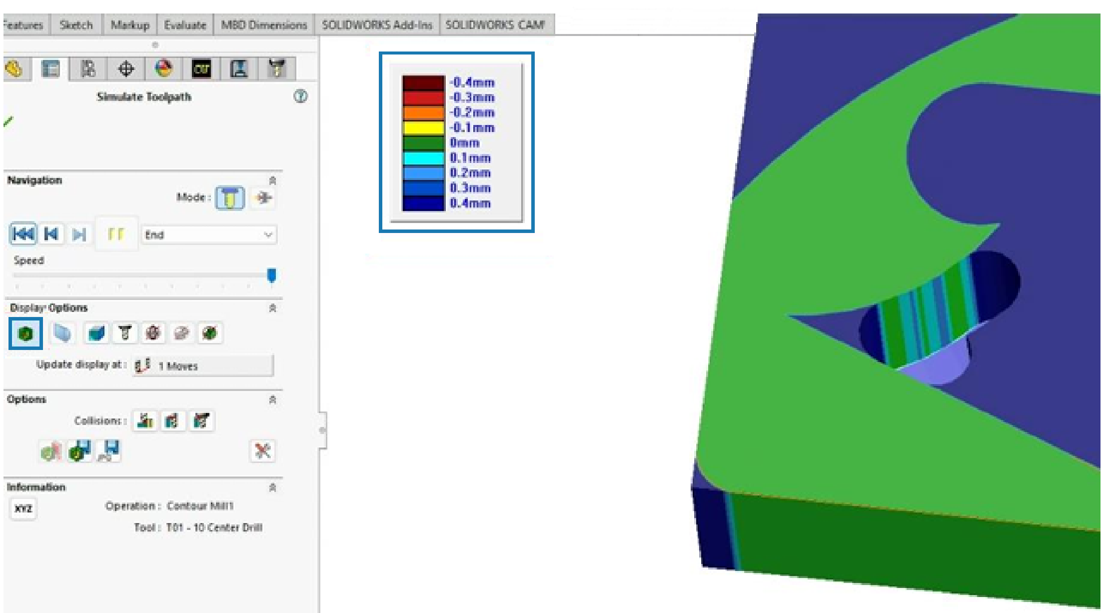
Option	Description
Profile OD	Recognizes profile ODs in the active part through the **Extract Machinable Features** tool.
Profile ID	Recognizes profile IDs in the active part through the **Extract Machinable Features** tool.

Option	Description

Face Feature	Recognizes face features depending on the stock type:
	• Round bar stock. Recognizes a single face feature at the start of the part model.
	• Any stock type other than round bar stock. Recognizes:
	    • Face features at the start of the part model. (These features appear under the same **Turn Setup** as other recognized Turn features.)
	    • Face features at the end of the part model. (These features appear under the reversed **Turn Setup**.)
	
	When cleared, the software does not create a face feature under the **Turn Setup**. You can add face features using Interactive Feature Recognition.

Groove Feature	Recognizes groove features in the active part through the **Extract Machinable Features** tool.
Cut Off	Recognizes the specified type of cut off features: • **Bar Stock**. If the stock type is a bar stock, recognizes Cut Off features under the same **Turn Setup** as the other recognized features. • **Other Stock Types**. If the stock type is anything except a round bar, recognizes Cut Off features under the same **Turn Setup** as the other recognized features.

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

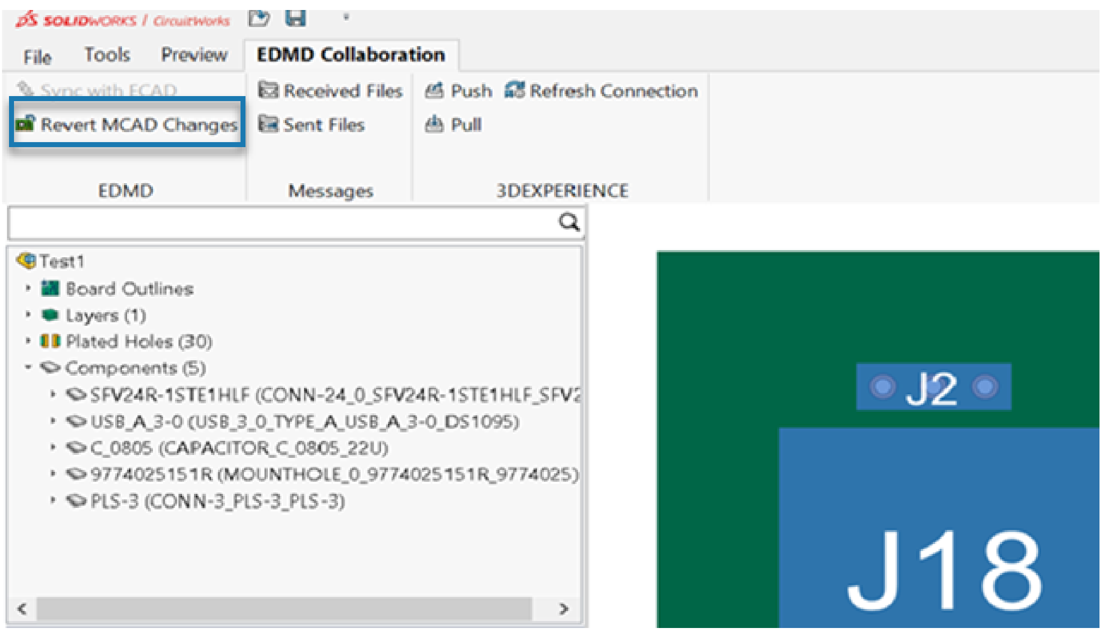
CircuitWorks

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 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

This chapter includes the following topics:

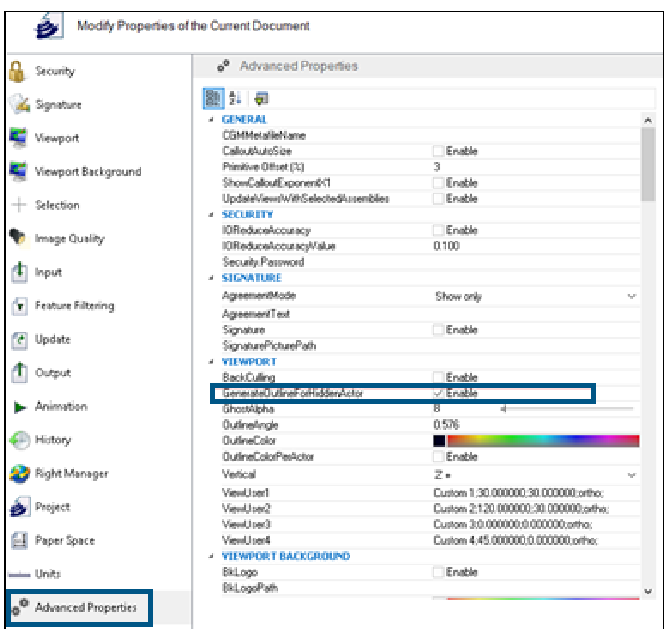
SOLIDWORKS® Composer™ software streamlines the creation of 2D and 3D graphical contents for product communication and technical illustrations.

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.

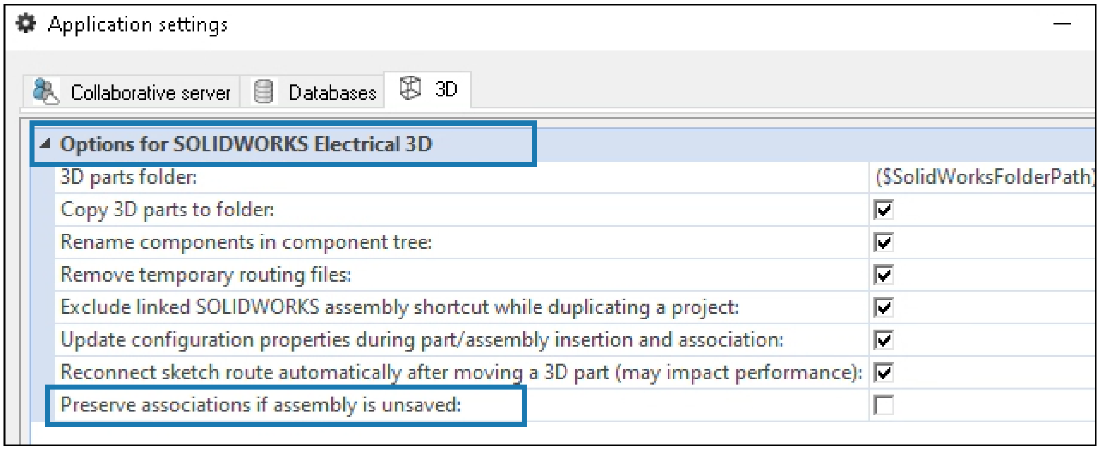
22

SOLIDWORKS Electrical

This chapter includes the following topics:

SOLIDWORKS® Electrical is a separately purchased product.

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.

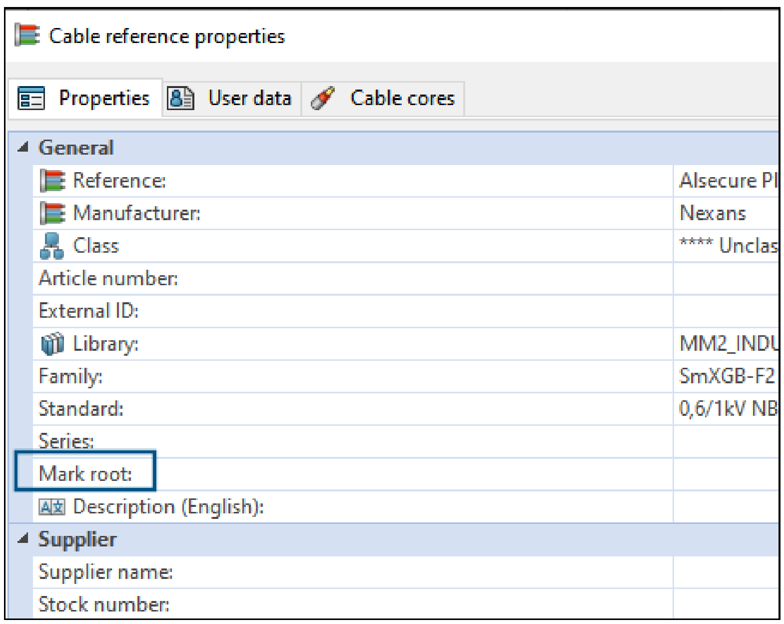
Preserve Association Option

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**.

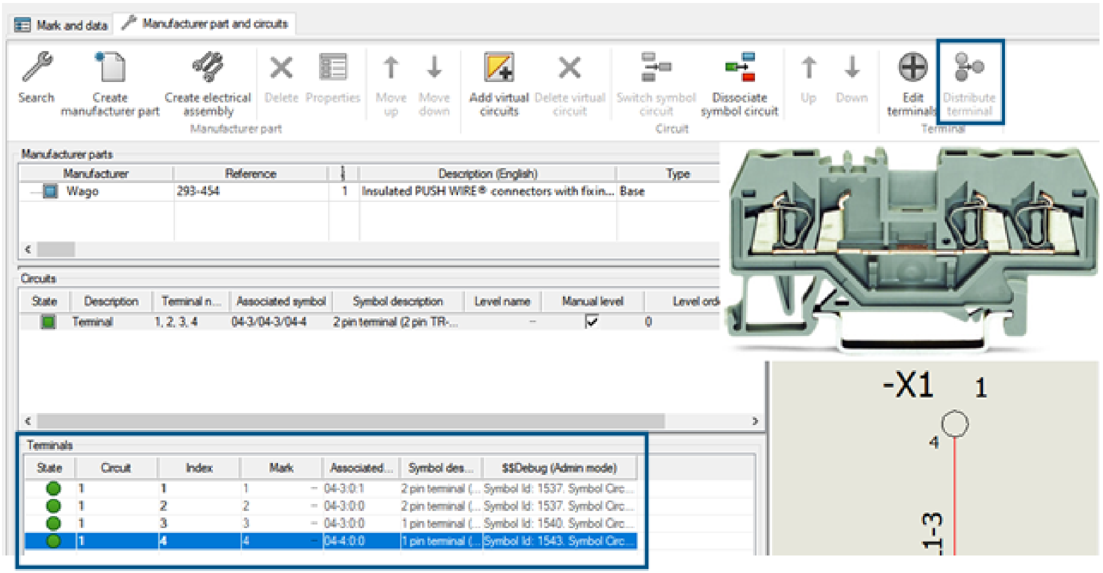
Cable Management



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:

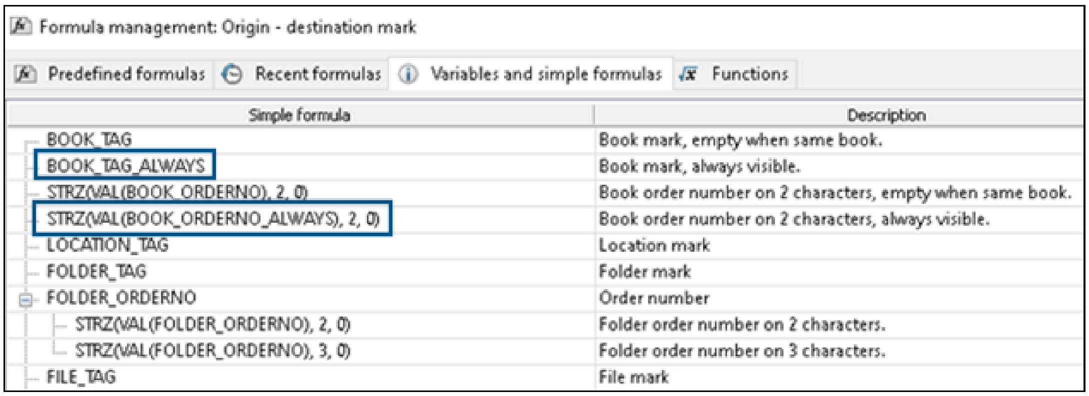
1. Click **Insert Terminal** .

   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



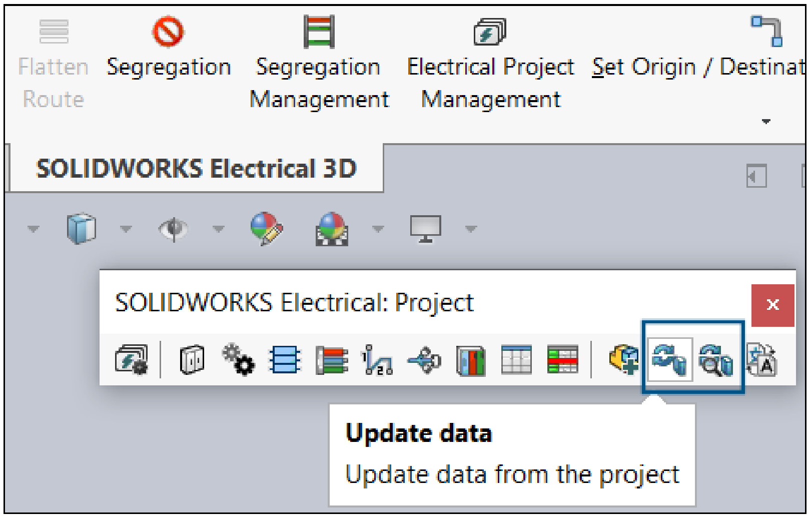
New variables are available in the Formula Management dialog box that let you label origin and destination arrows more effectively. This makes it easier to find and understand bookmarks, especially when the arrows are in the same book.

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.

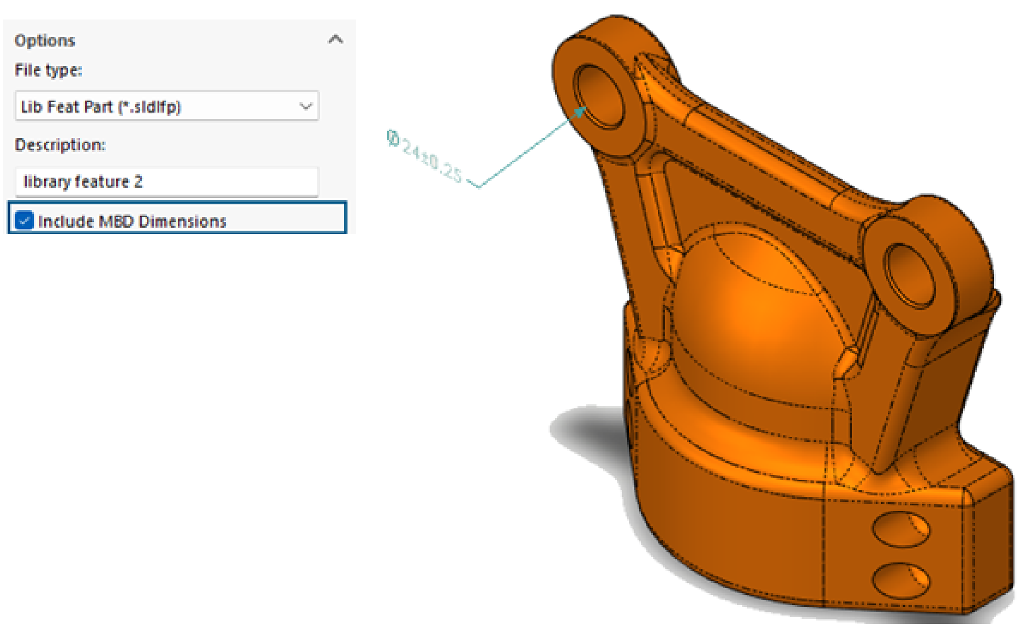
23

SOLIDWORKS MBD

This chapter includes the following topics:

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

Saving DimXpert Dimensions to Library Features (2025 SP1)



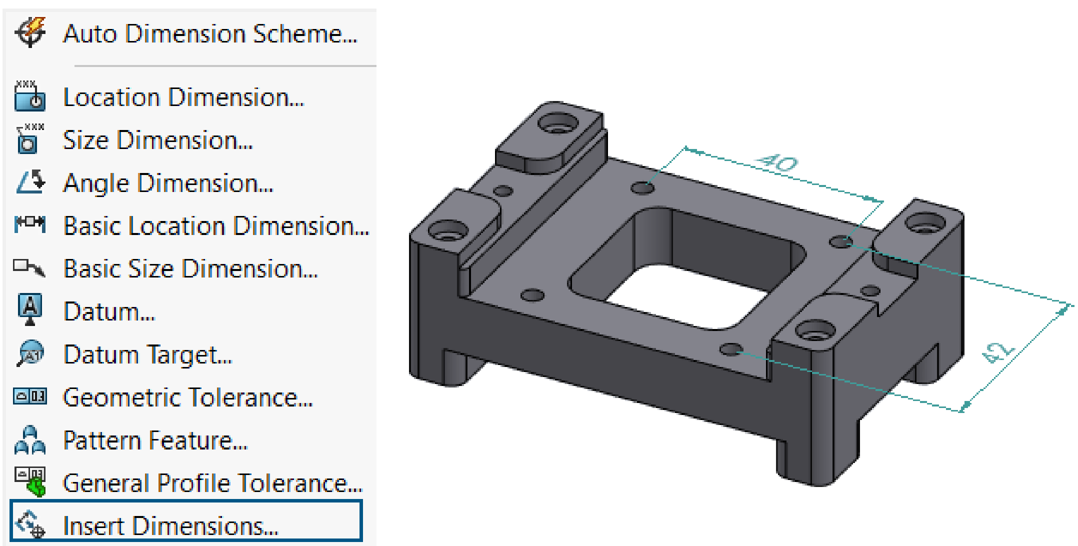
You can save DimXpert dimensions to library features.

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

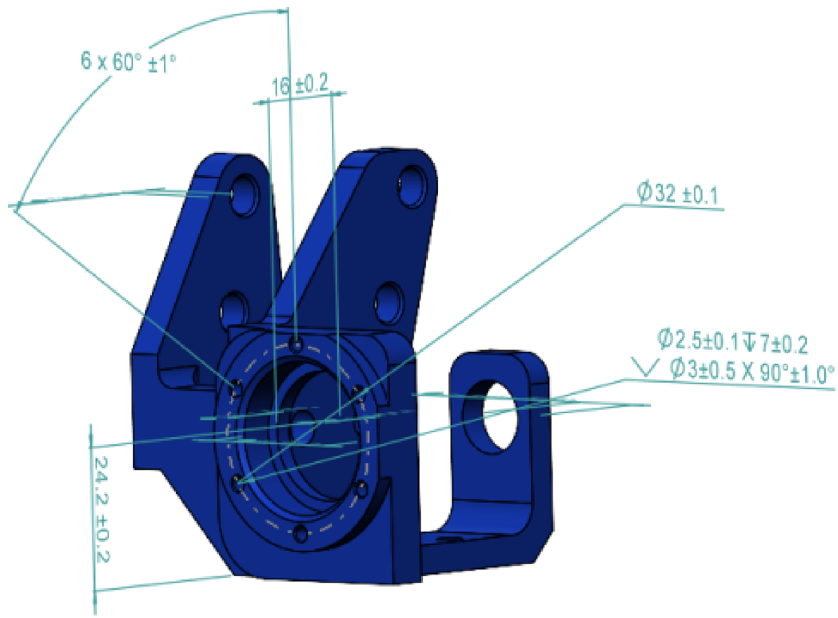


You can create DimXpert dimensions from sketch dimensions.

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

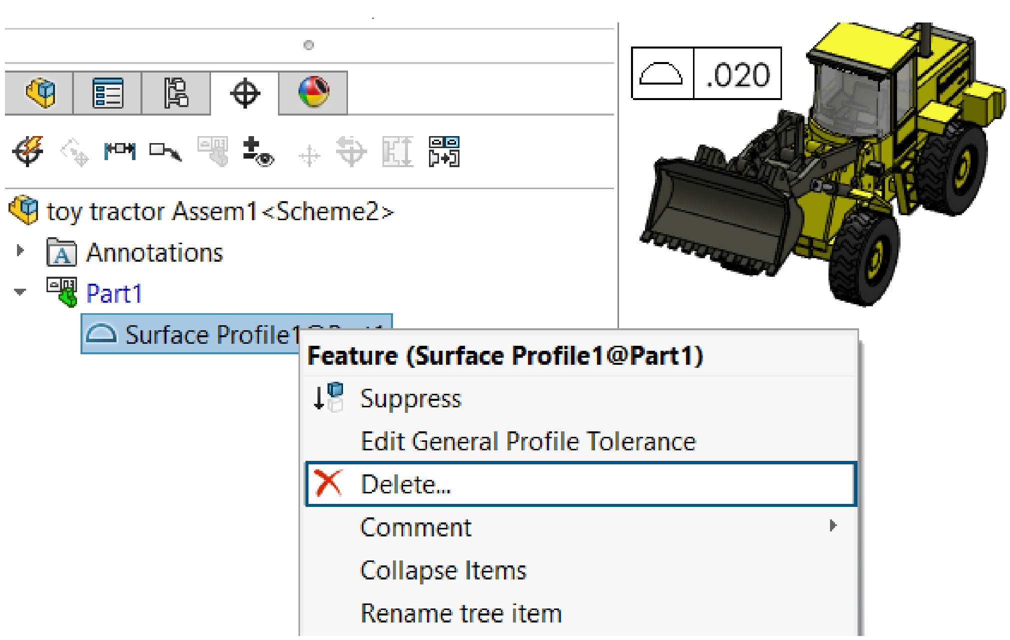


SolidNetWork License (SNL) customers can use the SOLIDWORKS MBD add-in.

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**.

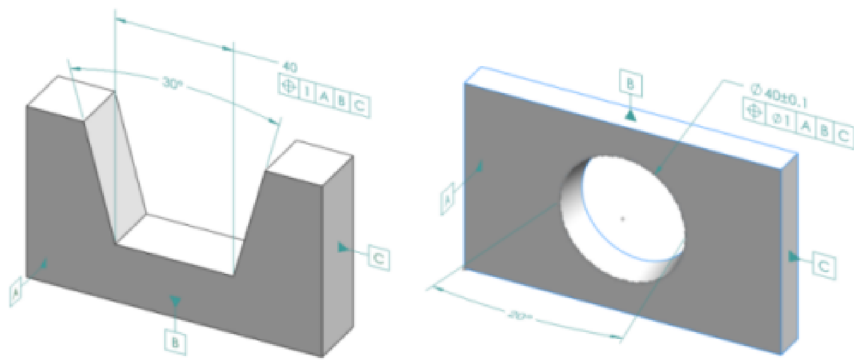
Delete General Profile Tolerance



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**.

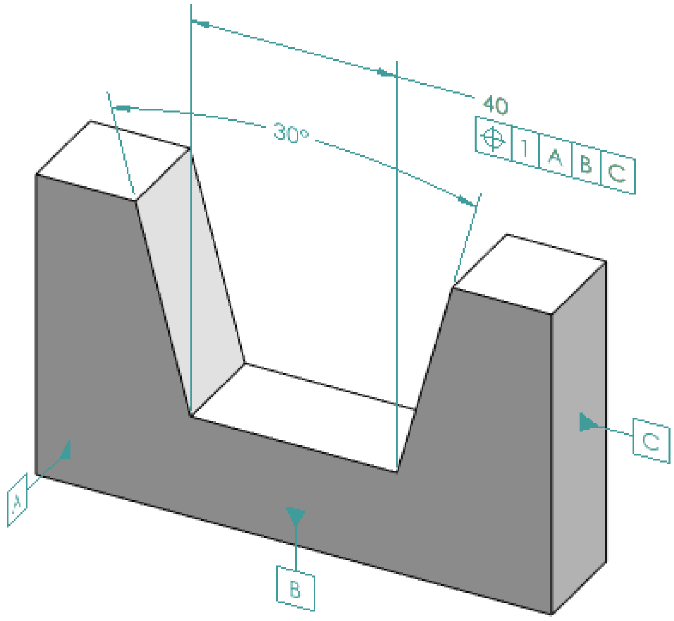
Creating Length Dimensions in Drafted Features



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.

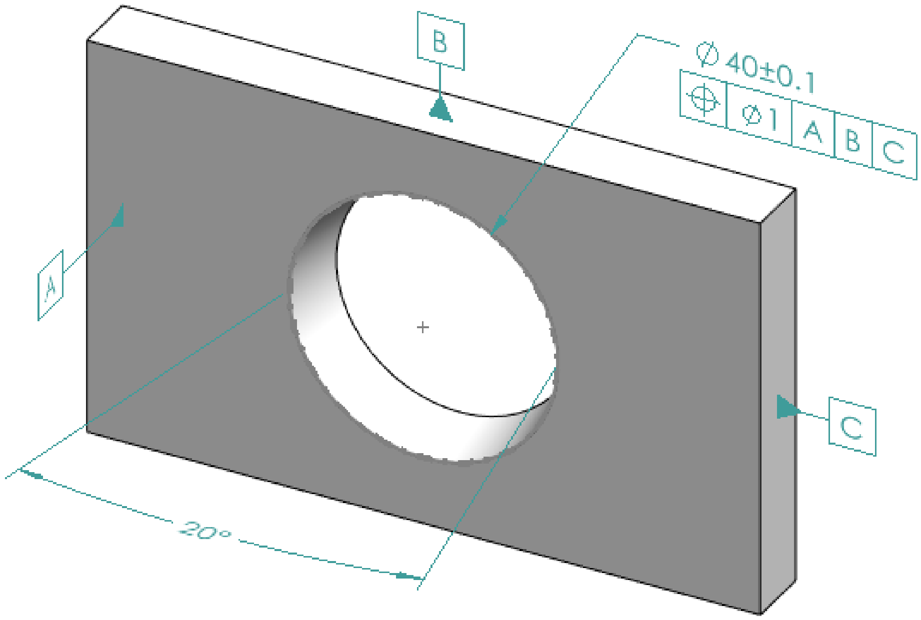
Creating Length Dimensions in Wedges



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



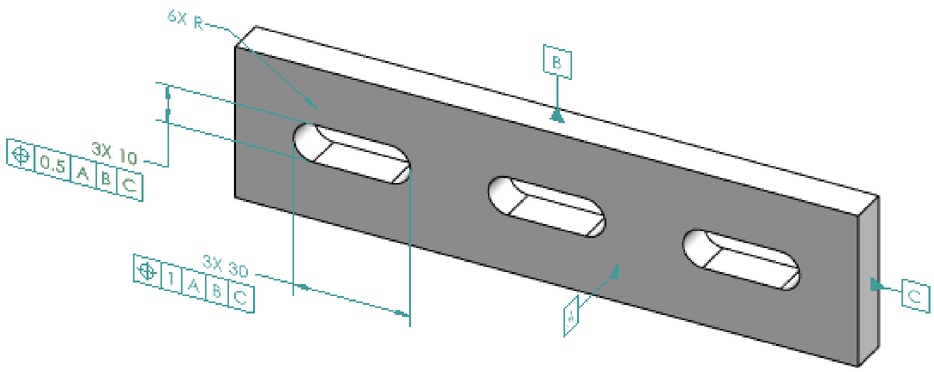
To create length dimensions in cones:

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*.

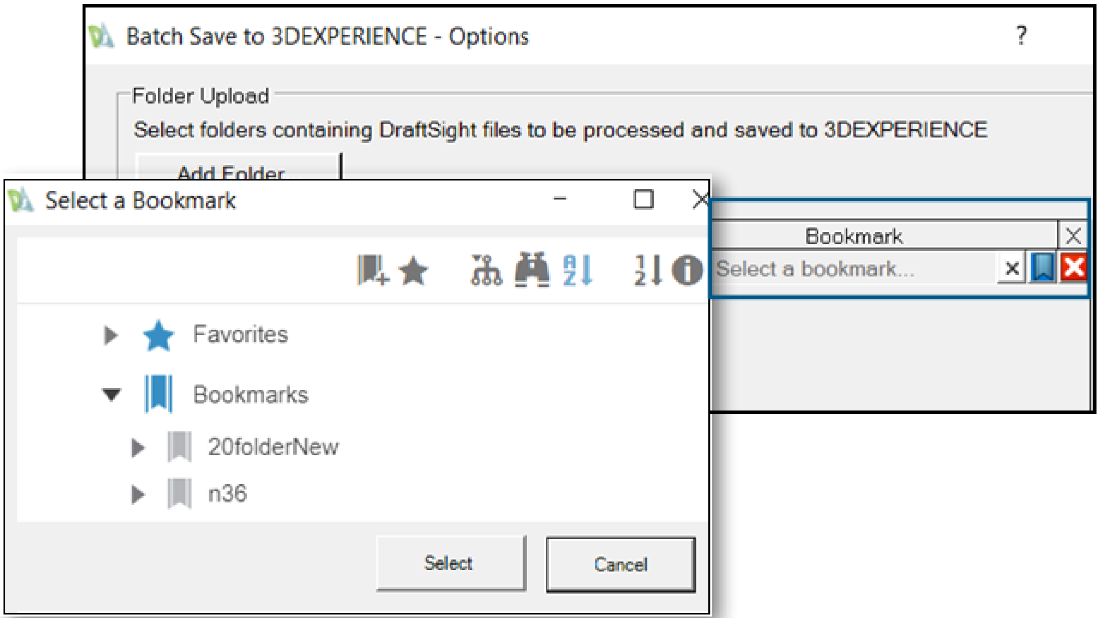
24

DraftSight

This chapter includes the following topics:

DraftSight® is a separately purchased product that you can use to create professional CAD drawings. It is available as DraftSight Professional, DraftSight Premium, and DraftSight Mechanical. In addition, DraftSight Enterprise and Enterprise Plus are available on network license. **3D**EXPERIENCE® DraftSight is a combined solution of DraftSight with the power of the **3D**EXPERIENCE platform.

Bookmarks for Batch Save to 3DEXPERIENCE (DraftSight Connected Only)

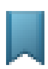


You can batch upload files to bookmarks on the **3D**EXPERIENCE platform.

To open the Batch Save to **3D**EXPERIENCE - 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
New Bookmark	Creates a new bookmark.
Favorite	Marks bookmarks as favorites.
Expand All	Expands the folder structure.
Find in Tree	Searches for the file in the selected bookmark.
Alphabetical Order	Sorts the bookmarks in alphabetical order.
Date Order	Sorts the bookmarks based on the creation date.

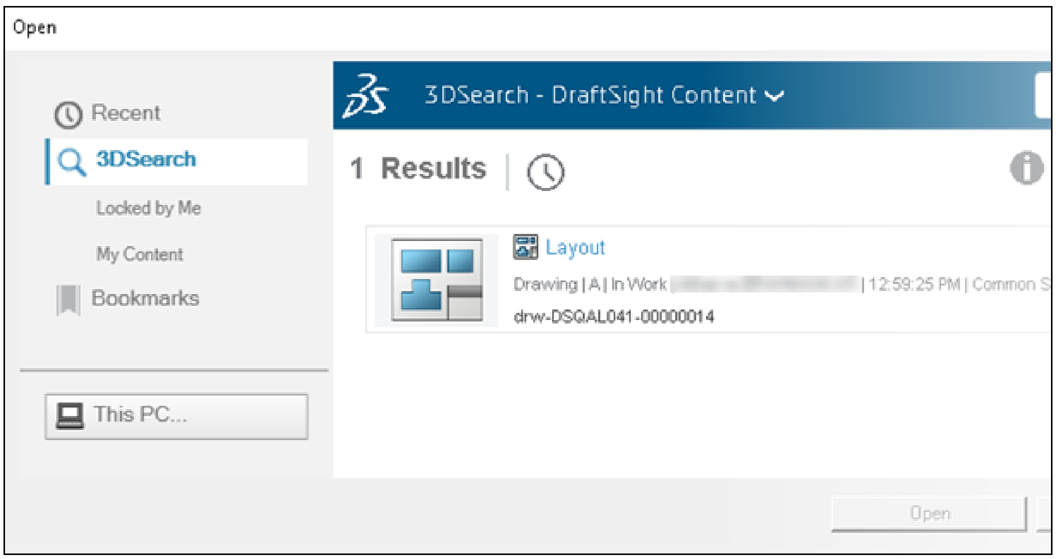
Favorites

Lists the favorite bookmarks.

Bookmarks

Lists the bookmarks available on the **3D**EXPERIENCE 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.

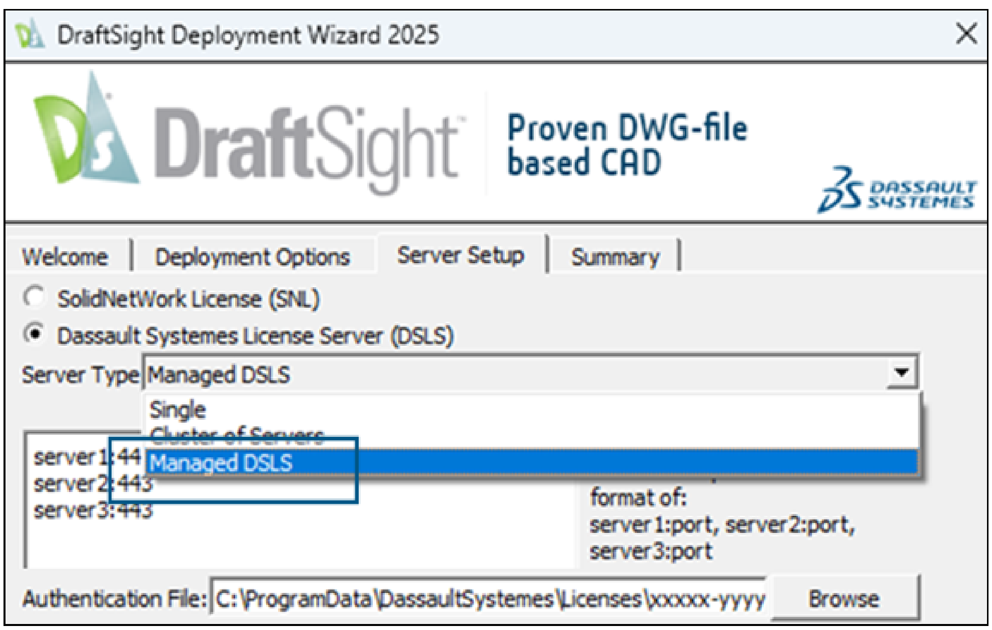
To open the dialog box, do one of the following:

- Click **Open** (Quick Access toolbar).
- Click **File** > **Open**.
- Type `Open` in the command window.

Option	Description
Recent	Displays the recently opened files. The cloud symbol denotes the file that you have opened on the **3D**EXPERIENCE platform. Select the file and click **Open** to open it.
3DSearch	Displays the files saved on the **3D**EXPERIENCE platform.

Option	Description
Locked by Me	Displays the files locked by you. Click **Clear Filter** to clear the results and display all files.
My Content	Displays the files created by you. Click **Clear Filter** to clear the results and display the files created by all users.
Bookmarks	Displays the bookmarks and files saved to the bookmarks.
This PC	Opens the locally saved files.
Open	Opens the file that you selected from results. If you are working in the offline mode, you can open only recently opened and locally saved files.

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	[illegible]	2025	26-10-2027	3,250
5	4	15	April	2026	27-10-2027	4,250

You can use the TABLE command to automatically enter data in the adjacent cells of a table.

This feature is useful where data follows a logical or repetitive order, such as filling in dates, sequential numbers, days of the week, months, etc. Auto-fill is an efficient tool to handle large data sets and ensure consistency.

To access the TABLE command:

Do one of the following:

- On the ribbon, click **Annotate** > **Table** > **Insert**.
- On the menu, click **Draw** > **Table**.
- Enter TABLE in the command window.

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.

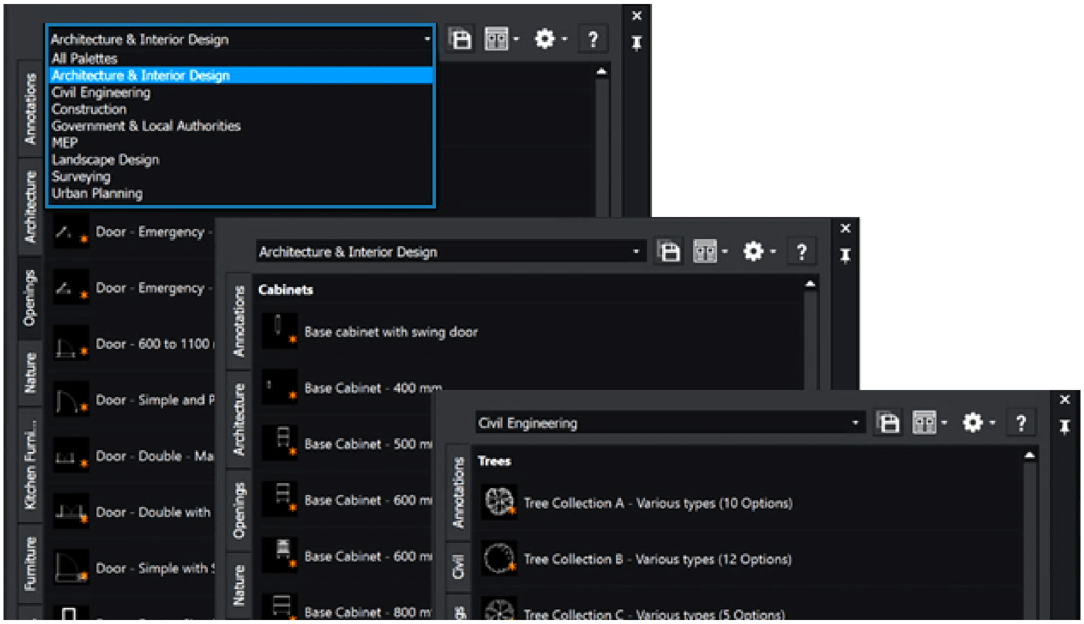
You can define the table height by dragging the grip point or you can enter the height in a drawing unit in the Properties palette.

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

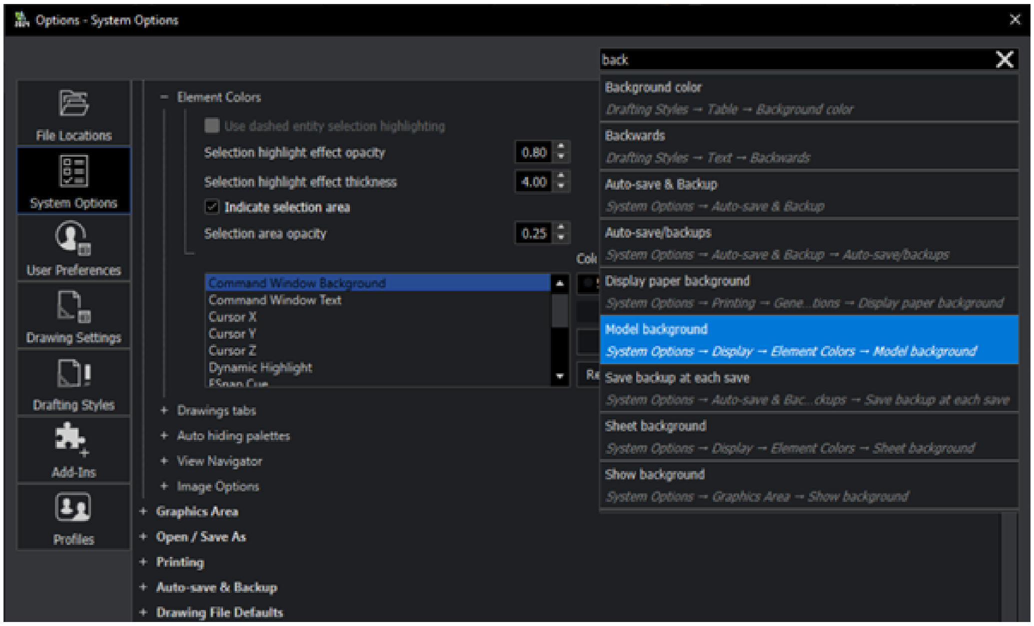


The tool palettes include more than 400 dynamic blocks. The blocks are parametric and compatible with AutoCAD®.

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.

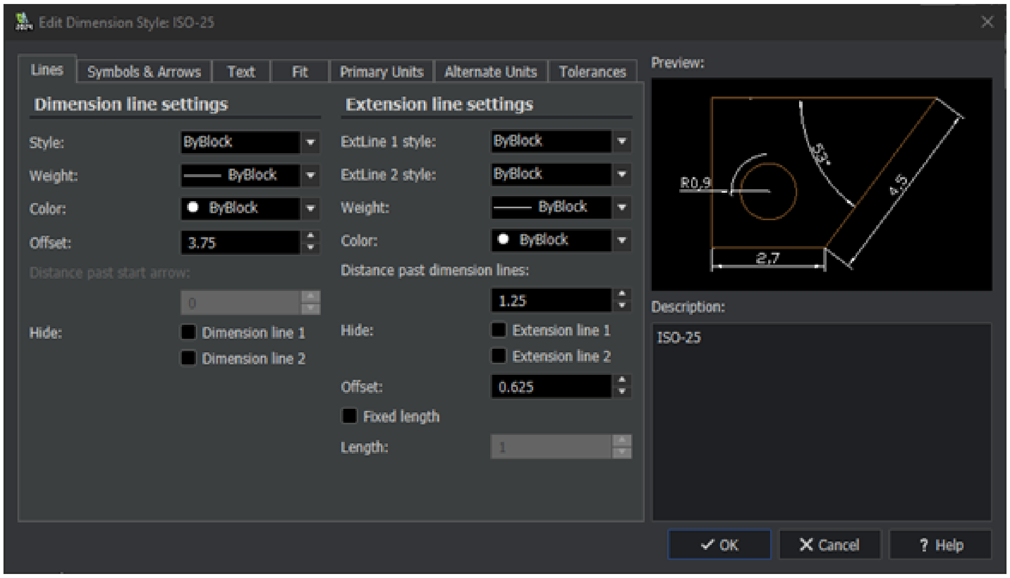
In the Search box of the Options dialog box, start to enter a term or system variable name to see a list of options containing the string that you entered. Relevant options appear in a list. You can click an option to go directly to the specified option.

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.

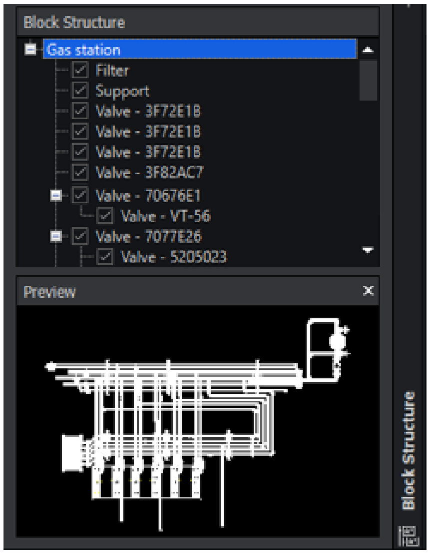
When you edit Dimension Styles, the user interface more closely resembles the AutoCAD interface. This ensures a smoother transition for users migrating from AutoCAD to DraftSight.

To access the Dimension Styles dialog box:

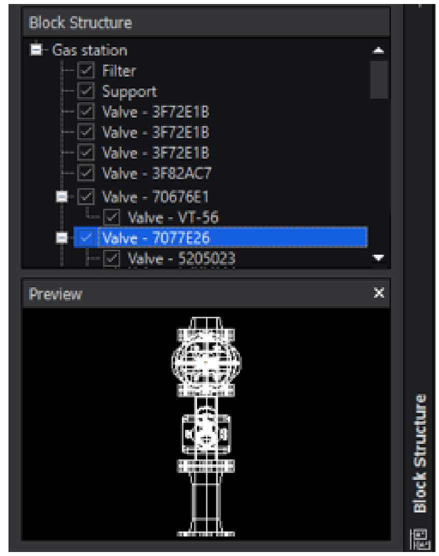
Do one of the following:

- 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.

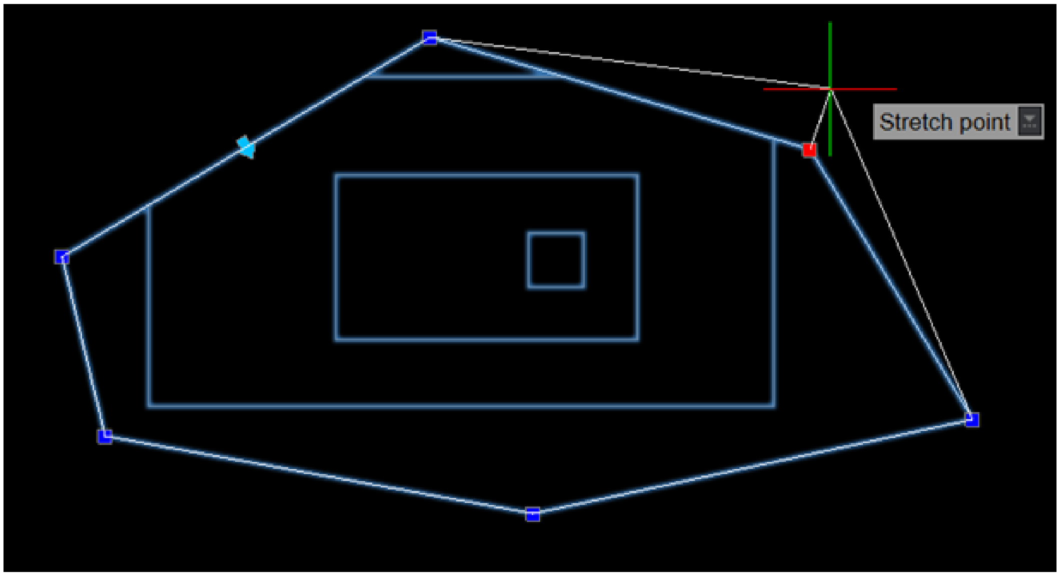
- Copy blocks from one area of the drawing and paste them elsewhere, maintaining the hierarchical structure. This simplifies the process of reusing design elements and maintaining consistency in the drawing.
- Rename, group, organize, or delete blocks within the palette.
- Nest a block within another 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.

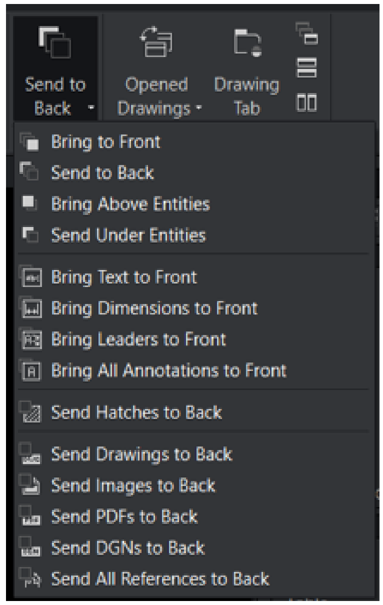
To edit clipped external references and blocks:

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
Bring Annotations to Front	Brings all annotation entities, including text, dimensions, and leaders, to the forefront of the design. 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. You can create more precise, visually appealing, and impactful drawings while promoting efficient collaboration and communication.
Send Hatches to Back	Relegates hatches to the background, ensuring unobstructed visibility of underlying entities. This is useful when hatch patterns interfere with the clarity and comprehensibility of the design, providing clearer drawing views.
Send References to Back	Relegates references to the background, optimizing the visibility of primary design elements. This lets you focus on the key components, resulting in improved efficiency and accuracy during the design process.

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 `REFERENCETOBACK` commands:

Do the following:

Ribbon	Menu
View > **Order** > **Bring Text to Front**	**Tools** > **Display Order** > **Bring Annotations to Front** > **Text Only**
View > **Order** > **Bring Dimensions to Front**	**Tools** > **Display Order** > **Bring Annotations to Front** > **Dimensions Only**

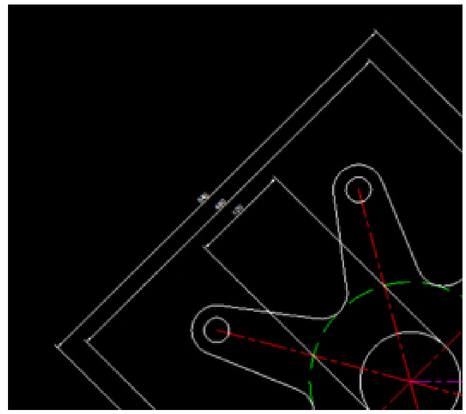
Ribbon	Menu
View > **Order** > **Bring Leaders to Front**	**Tools** > **Display Order** > **Bring Annotations to Front** > **Leaders Only**
View > **Order** > **Bring All Annotations to Front**	**Tools** > **Display Order** > **Bring Annotations to Front** > **All Annotation Entities**
View > **Order** > **Send Hatches to Back**	**Tools** > **Display Order** > **Send Hatches to Back**
View > **Order** > **Send Drawings to Back**	**Tools** > **Display Order** > **Send References to Back** > **Drawings Only**
View > **Order** > **Send Images to Back**	**Tools** > **Display Order** > **Send References to Back** > **Images Only**
View > **Order** > **Send PDFs to Back**	**Tools** > **Display Order** > **Send References to Back** > **PDFs Only**
View > **Order** > **Send DGNs to Back**	**Tools** > **Display Order** > **Send References to Back** > **DGNs Only**
View > **Order** > **Send All References to Back**	**Tools** > **Display Order** > **Send References to Back** > **All Referenced Entities**

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.

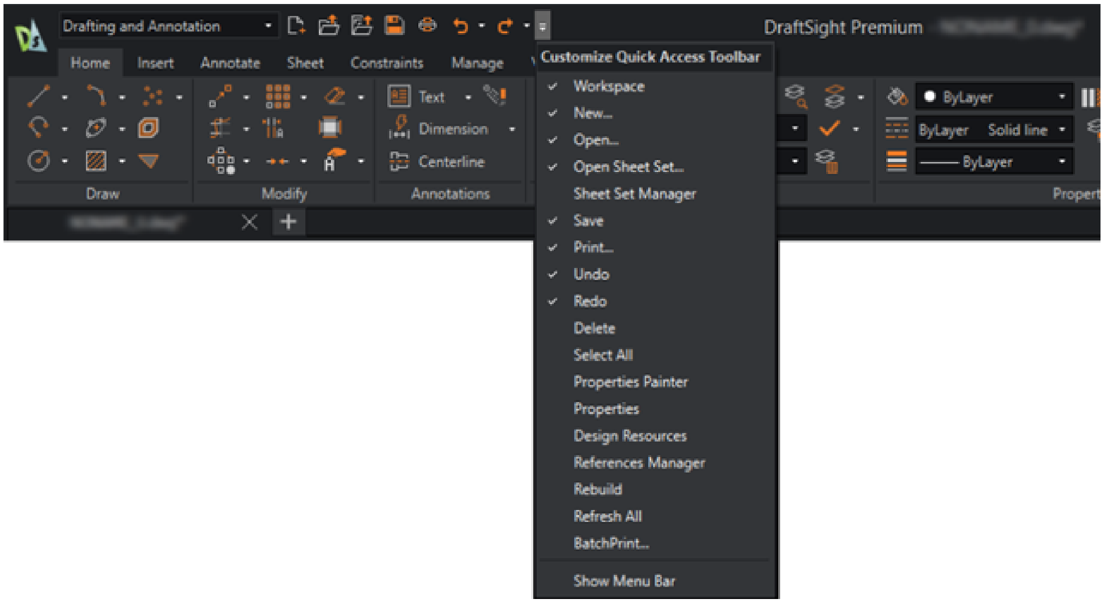
With the DIMSPACE command, you have greater precision and can spend less time on manual adjustments. The DIMSPACE command is similar to AutoCAD functionality for drawing dimensions, so it is easy to learn if you are familiar with AutoCAD.

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.

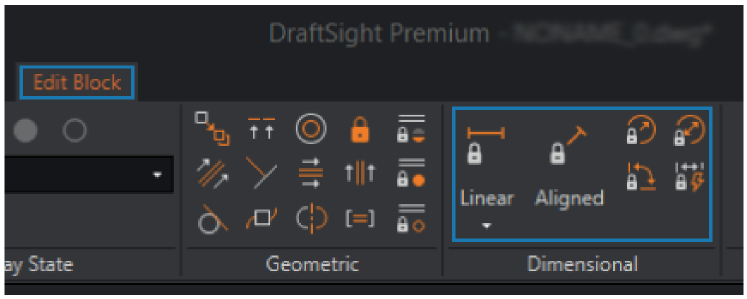
The **Customize Quick Access Toolbar** functionality switches the menu bar visibility.

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`.

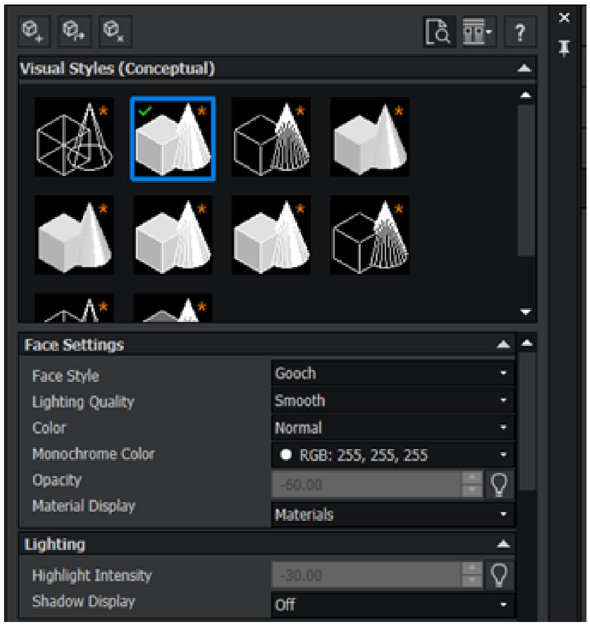
With certain commands (such as `TRIM`, `FILLET`, and `JOIN`) and other tools (snap, measure, and dimension), you need to specify the elevation (Z value) as `0`. Otherwise, the commands and tools do not work as expected. The FLATTEN command ensures that the elevation is `0`.

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
2D Wireframe	Uses only lines and curves without shading or rendering.
Wireframe	Suitable for viewing and editing 3D models with lines and curves.
Hidden	Uses hidden lines removed to provide a clear view of visible lines.
Realistic	Adds realistic lighting and shading to the model, providing a lifelike representation of materials and textures.
Conceptual	Applies a stylized rendering to the model, emphasizing contours and shapes. Useful for conceptual design and artistic presentations.

Visual Style	Description
Shaded	Displays the model with flat shading.
Shaded with Edges	Combines shaded surfaces with visible edges to define the boundaries of objects in the model.
Shades of Gray	Displays the drawing in varying shades of gray to differentiate between different objects and their elevations. This provides a monochromatic, effective representation.
X-Ray	Makes all objects transparent so you can see through the model. Helpful for analyzing complex assemblies.
Sketchy	Applies a hand-drawn, sketch-like appearance to the model, giving it a more artistic and informal look.

Export Models to Unreal Engine

You can export `DWG` file content to the Unreal Engine environment, which allows you real-time rendering and visualization.

When you export a CAD model, the export preserves materials, lighting, and other scene elements. In Unreal Engine, you can create a visualization of the model. This includes realistic lighting, shadows, and materials that provide a more immersive experience compared with traditional static renders.

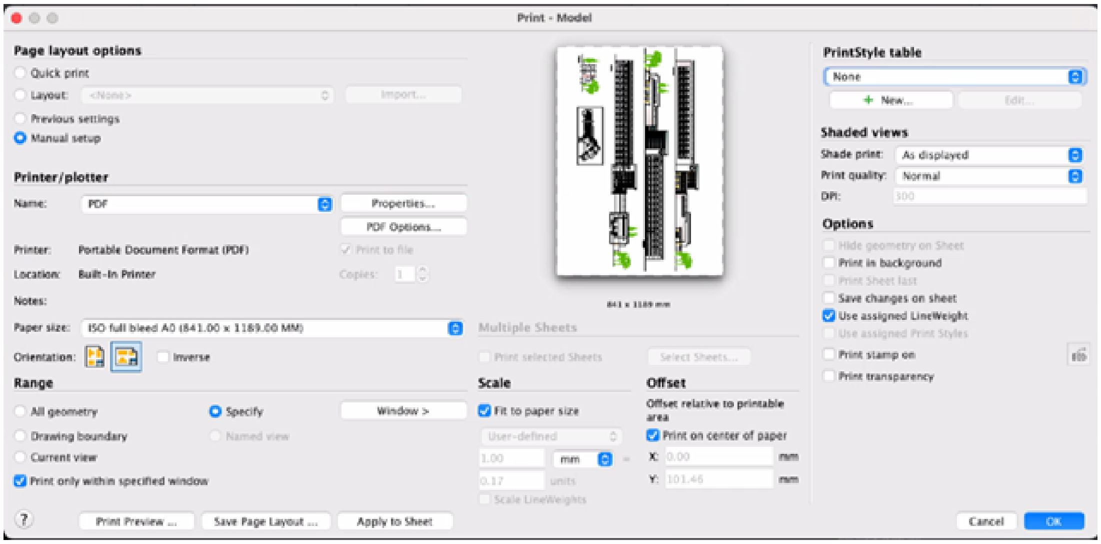
Unreal Engine offers real-time rendering capabilities for you to validate designs more effectively. By exploring a 3D model during client presentations or collaborative sessions, you gain clearer communication and understanding. Collaboration is more intuitive for you and your clients to interact with the design simultaneously.

To export models to Unreal Engine:

Do one of the following:

- On the ribbon, click **Application** > **Export** > **Export Datasmith**.
- On the menu, click **File** > **Export** > **Export Datasmith**.
- Enter `EXPORTDATASMITH` or `EXPORT` in the command window.

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.

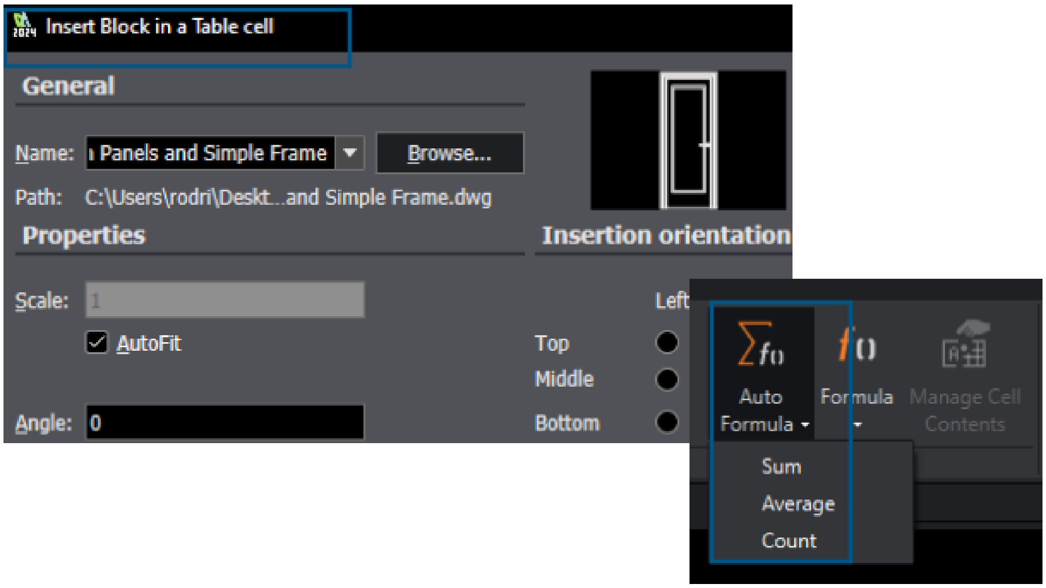
Unlike the system Print dialog box, this dialog box provides a broader range of options, giving you greater control over printing preferences. Printing is simpler and more efficient, ensuring that drawings print the way you want them.

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.

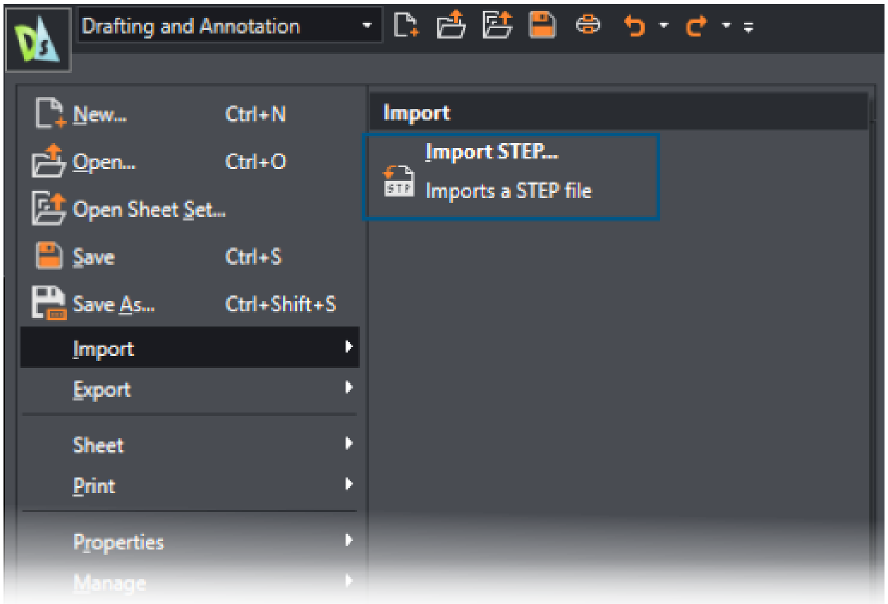
To make tables more useful, you can:

- 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

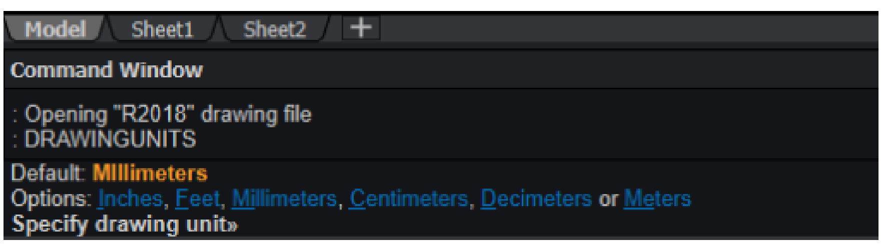
Import STEP Files



You can use the `IMPORTSTEP` command to import 3D models from `STEP` files.

You can incorporate `STEP` file models into drawings.

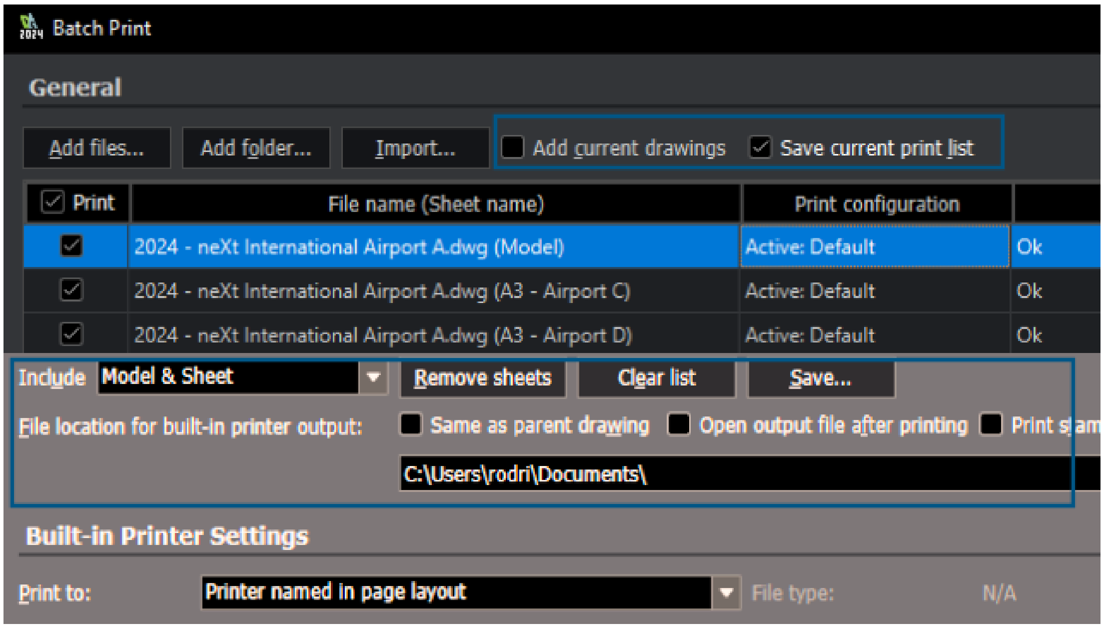
DWGUNITS Command



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.

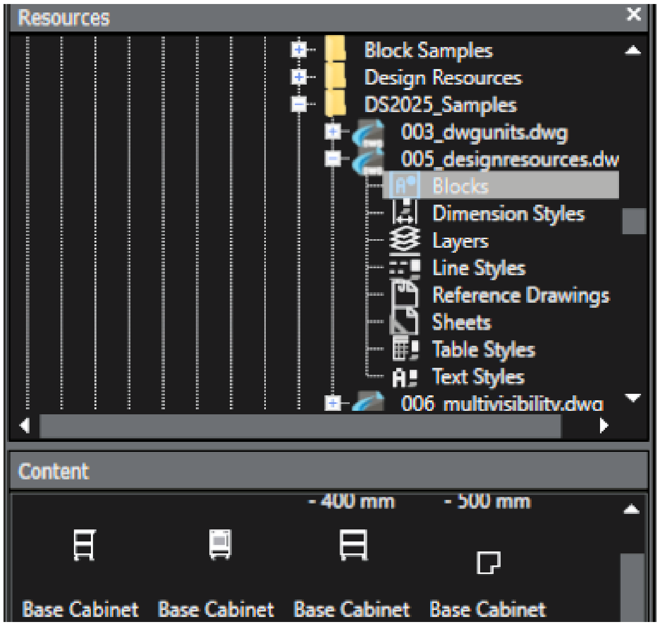
PDF Export and Batch Print Usability



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.

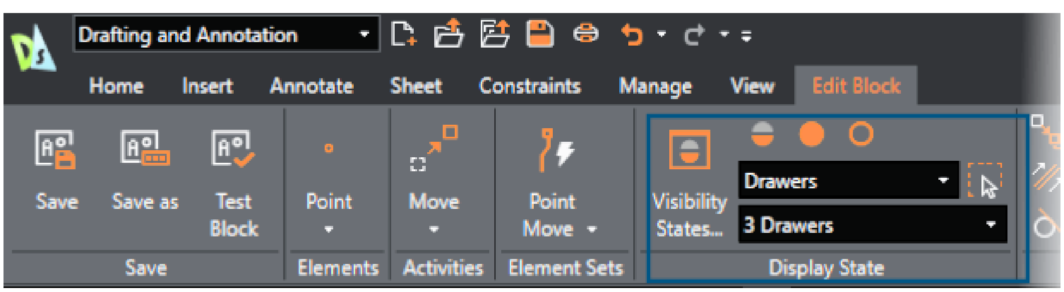
Blocks in the Design Resource Palette



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.

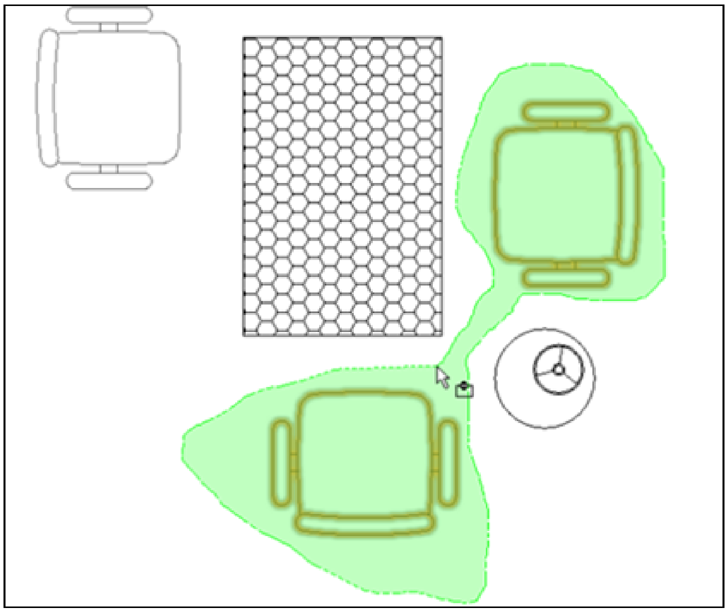
Multiple Visibility Elements



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.

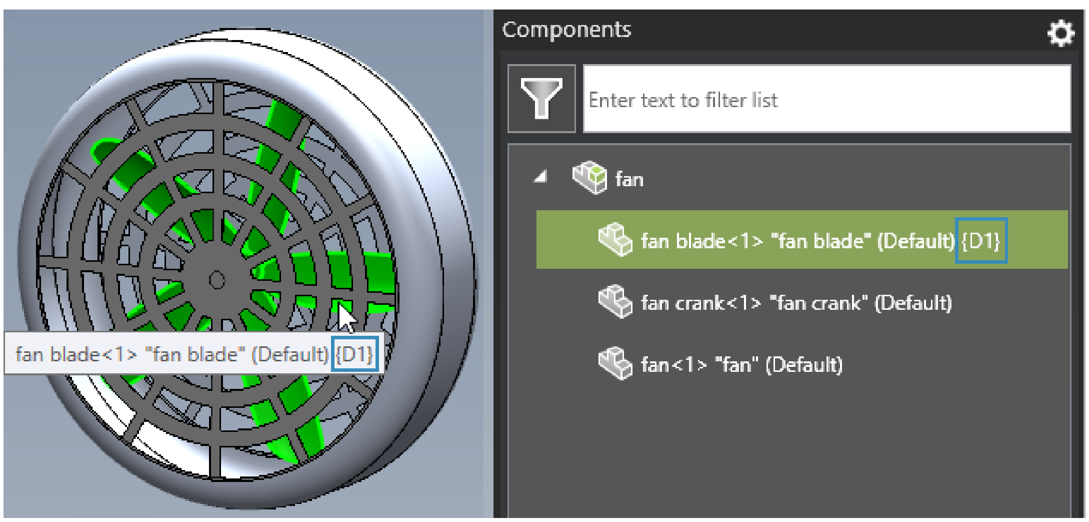
25

eDrawings

This chapter includes the following topics:

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

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.

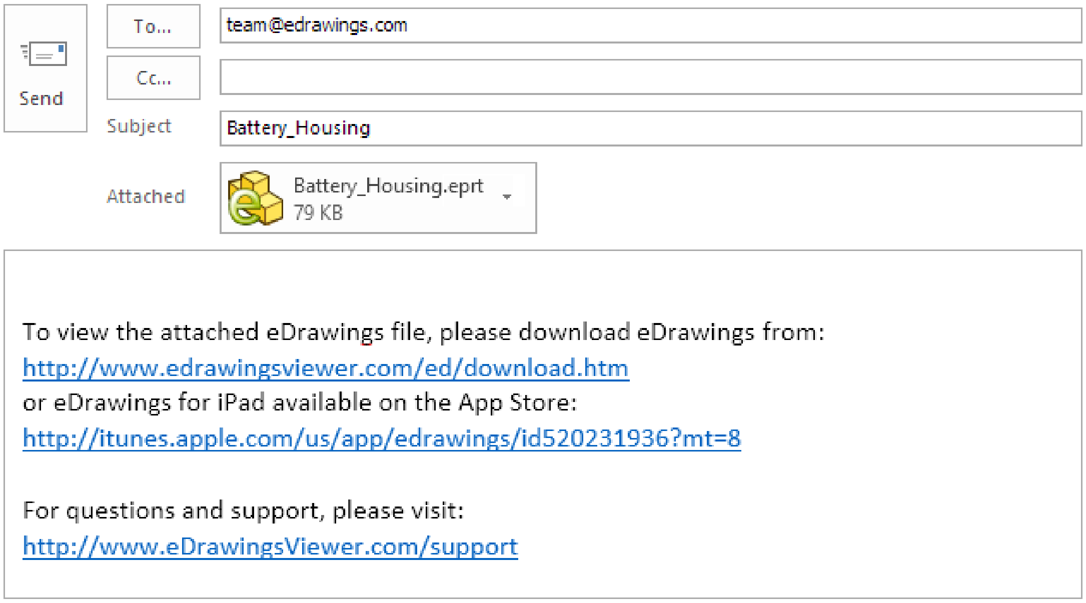
To view component references:

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.

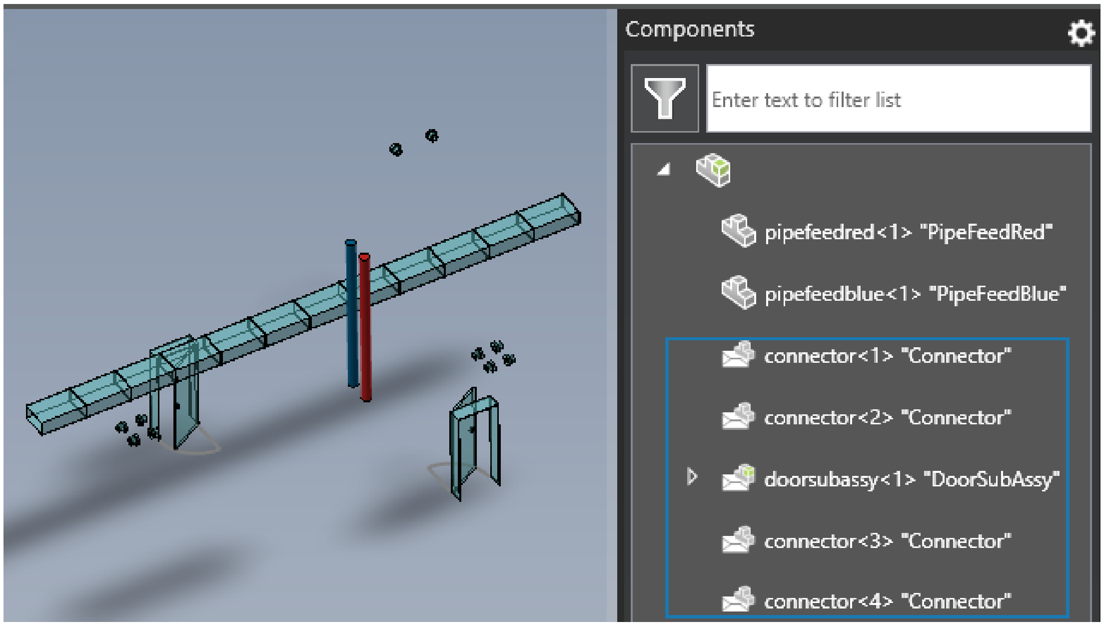
eDrawings ActiveX HTML File Format



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.

Supported File Types

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 / **3D**EXPERIENCE®	Up to V5-6R2024
Creo® - Pro/Engineer® (.ASM, .NEU, .PRT, .XAS, .XPR)	Pro/Engineer 19.0 to Creo 10.0
JT (.jt)	Up to v10.9
NX™ (Unigraphics®) (.prt)	UG11 to UG18, UG NX, NX5 to NX12, NX1847 to NX2312

Format	Version
Parasolid™ (.x_b, .x_t, .xmt, .xmt_txt)	Up to 36.1
Solid Edge® (.asm, .par, .pwd, .psm)	1 to 20, ST1 - ST10, 2019 to 2024

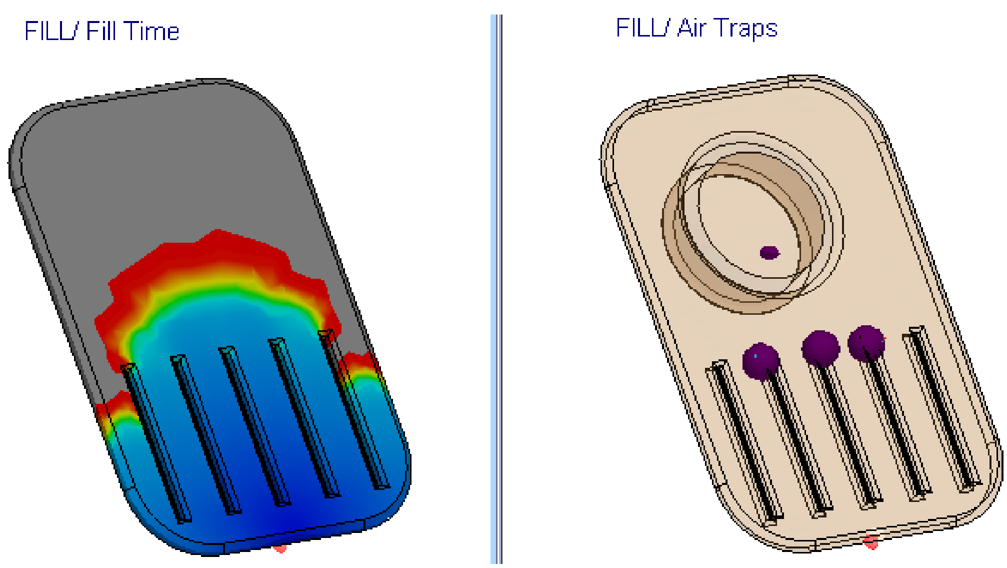
26

SOLIDWORKS Plastics

This chapter includes the following topics:

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

Fill Analysis

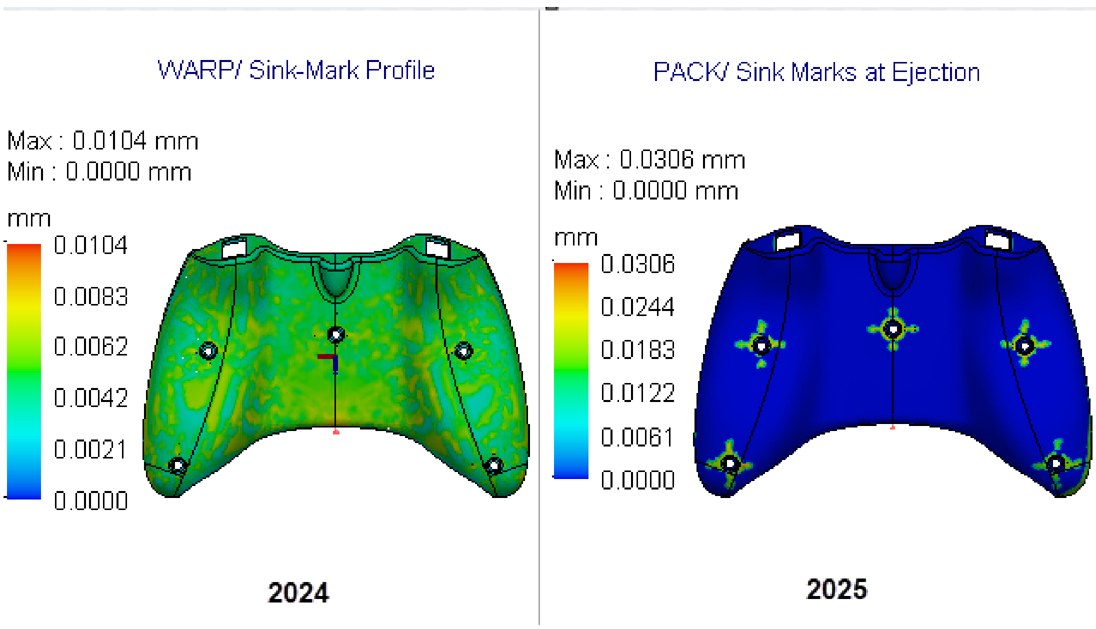


There are several enhancements for the Fill analysis.

- 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.

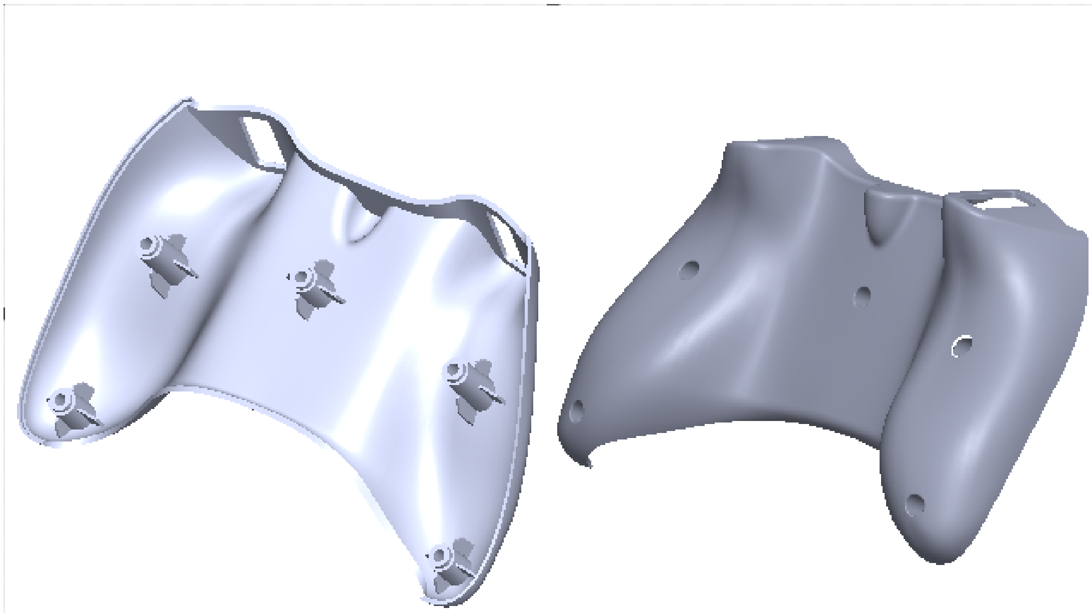
- Rendering of fill-time animations in isosurface mode has been significantly accelerated (up to 75%) for large models with a high number of elements. The memory required to generate the fill-time animations has also increased, as SOLIDWORKS Plastics uses all available memory resources for animation 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.

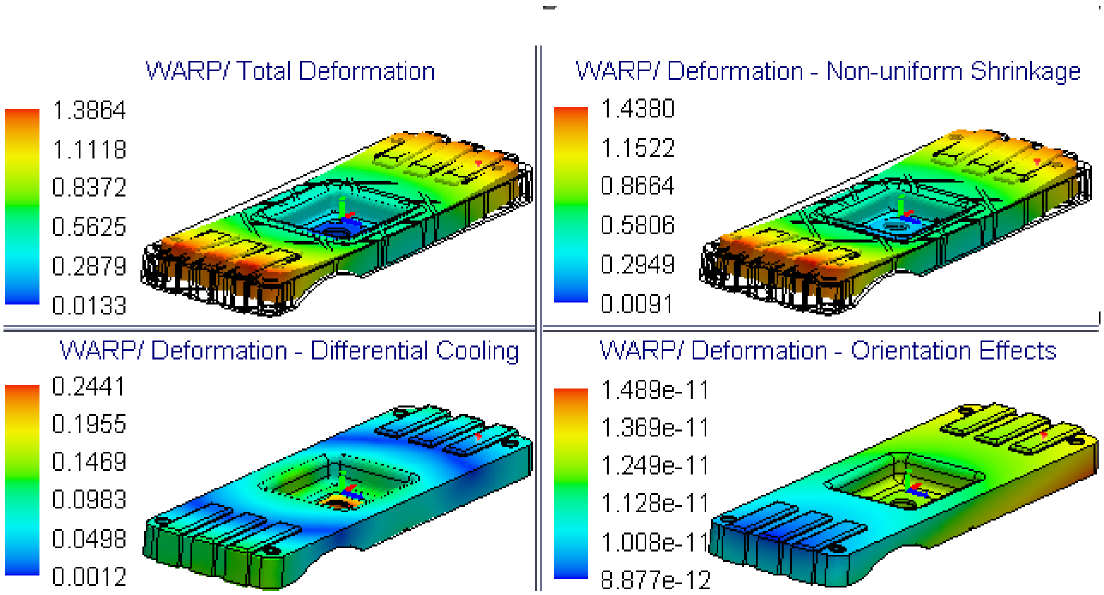


The sink mark results are updated as follows:

- 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
Deformation – Nonuniform Shrinkage	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 direction of melt flow and transverse to the direction of melt flow.

Result Plot - Warp Analysis	Description
	(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.)
Deformation – Differential Cooling	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.
Deformation – Orientation Effects	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
Lehmann&Voss&Co.	2

Manufacturer	Number of New Material Grades
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
Covestro®	26

Manufacturer	Number of Removed Material Grades
Rhone-Poulenc	14
SABIC Specialties®	7
Monsanto Japan	5
Lehmann and Voss	2
Trinseo®	1
Mitsubishi Chemical Japan®	1
Mitsubishi Rayon	1

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.

27

Routing

This chapter includes the following topics:

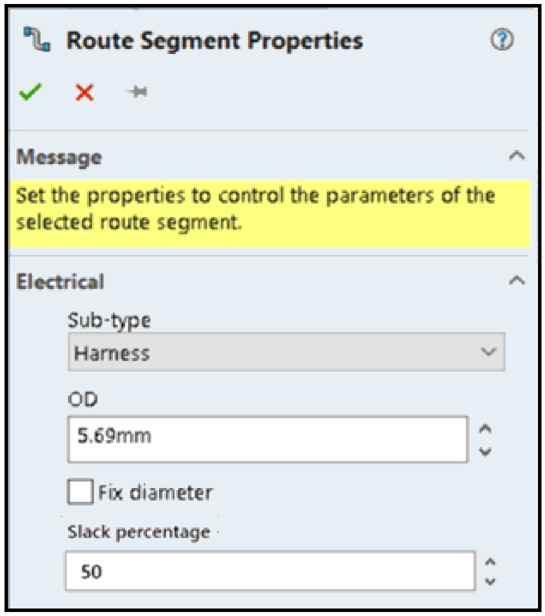
Routing is available in SOLIDWORKS® Premium and SOLIDWORKS Ultimate.

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

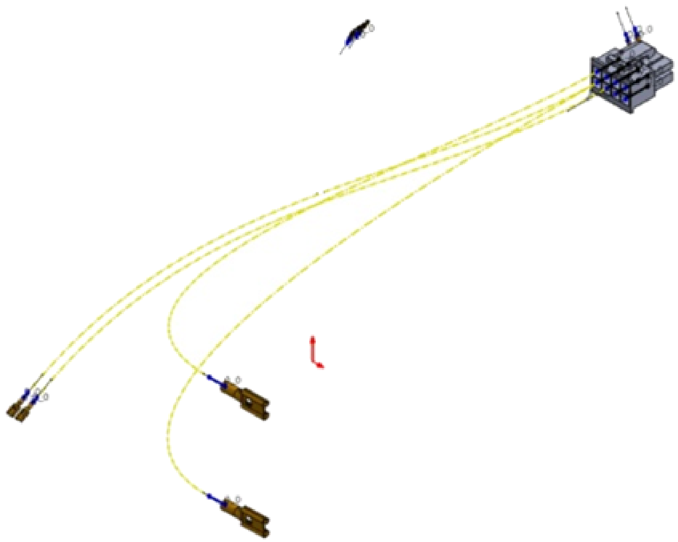


In the Route Properties and the Route Segment Properties PropertyManagers, you can define a custom value for the **Slack Percentage** for individual route segments. This value overrides the slack percentage specified in **Tools** > **Options** > **System Options** > **Routing**.

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.

Generate Guidelines to Follow a Route Path



In the Auto Route PropertyManager, you can generate guidelines to follow a route path. The guidelines identify the nearest sketch segment that leads to its corresponding end connector and follow that path.

The guidelines help simplify the process of manual harnessing.

To generate guidelines that follow a route path:

1. In the Auto Route PropertyManager, under **Routing Mode**, select **Guidelines**.
2. In **Routing Path**, click **Follow Routing Path**.
3. To identify sketches in the graphics area:
   - Select each sketch individually.
   - Box select to choose multiple sketches.
   - Select sketch features in the flyout FeatureManager design tree.
4. Click **Apply** to preview the route path. The sketches representing the route path are named **Routing_path** followed by an order number, such as **Routing_path 1**.
5. Click **Done**.

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.

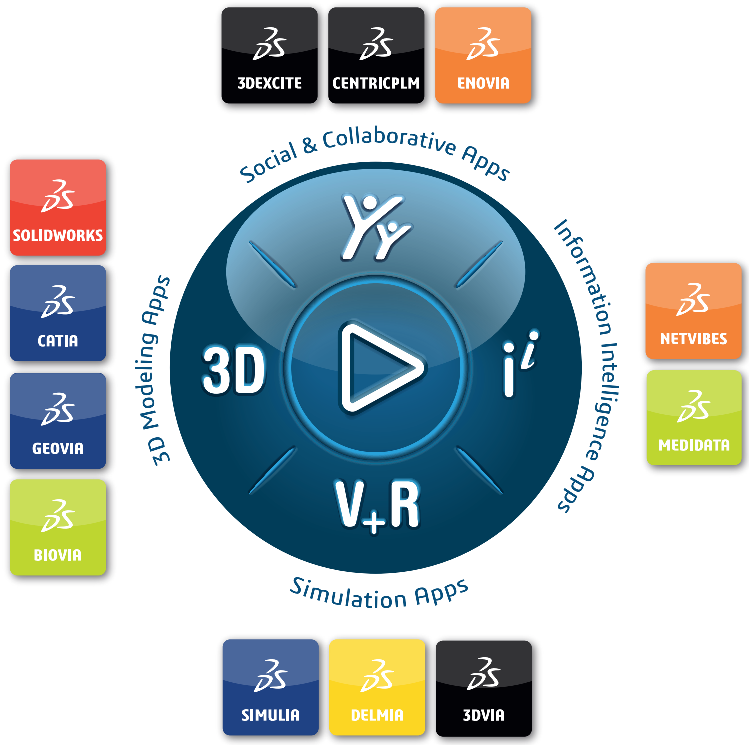
You can make multiple edits and preview them as temporary changes before finalizing, givingyou more control over the design process.

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.



Our **3D**EXPERIENCE® platform powers our brand applications, serving 12 industries, and provides a rich portfolio of industry solution experiences.

Dassault Systèmes is a catalyst for human progress. We provide business and people with collaborative virtual environments to imagine sustainable innovations. By creating virtual twin experiences of the real world with our **3D**EXPERIENCE platform and applications, our customers can redefine the creation, production and life-cycle-management processes of their offer and thus have a meaningful impact to make the world more sustainable. The beauty of the Experience Economy is that it is a human-centered economy for the benefit of all –consumers, patients and citizens.

Dassault Systèmes brings value to more than 300,000 customers of all sizes, in all industries, in more than 150 countries. For more information, visit **www.3ds.com**.

Europe/Middle East/Africa
Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

Asia-Pacific
Dassault Systèmes
17F, Foxconn Building,
No. 1366, Lujiazui Ring Road
Pilot Free Trade Zone, Shanghai 200120
China

Americas
Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
02451-1223
USA

DASSAULT SYSTEMES