



# WHAT'S NEW

## SOLIDWORKS 2023



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# 1

## Welcome to SOLIDWORKS 2023

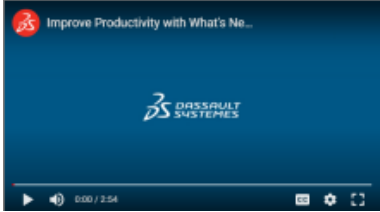
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This chapter includes the following topics:

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



Model courtesy of Philipp Burgstaller, SOLIDWORKS 2022 Beta Splash Screen Prize Winner.

	<p><b>Video: What's New in SOLIDWORKS 2023</b></p>
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At SOLIDWORKS®, we know that you create great designs, and that your great designs get built. To streamline and accelerate your product development process from concept through manufactured products, SOLIDWORKS 2023 contains new, user-driven enhancements focused on:

- **Working Smarter.** Reduce your workload when designing large assemblies and detailing drawings, and handle more complex electrical routing scenarios.
- **Working Faster.** Work with bills of materials (BOMs) more efficiently, improve the performance for searching and saving assemblies, and design large assemblies faster.
- **Working Together.** Help teams share and manage data more productively with a streamlined user interface, reduced latency, and improved upgrade performance in SOLIDWORKS PDM. Communicate changes to design data better with the ability to customize PDM notification templates.

## Top Enhancements

The top enhancements for SOLIDWORKS® 2023 provide improvements to existing products and innovative new functionality.

- Parts and Features** on page 27
- Coordinate Systems
  - Dimension Support for the Move/Copy Body Command
  - Single-Line Fonts for Wrap Features

- Sheet Metal** on page 35
- Sensors

- Structure System and Weldments** on page 39
- Corner Treatment to Similar Corners in Structure Systems
  - Configuration-Specific Size of Weldment Members

- Assemblies** on page 44
- Automatically Optimize Resolved Mode
  - Repairing a Missing Mate Reference
  - Additional End Conditions for Cut Features

- Detailing and Drawings** on page 64
- Overrides for Bills of Materials
  - Filters for Bill of Materials

### **SOLIDWORKS**

**Inspection** on page 140

- Autoballoons for PDF Files
- Balloon Sequence
- Balloon Drawing Sheets Separately
- Export Sheets as Separate 2D PDF Files
- Export FAI Reports to Separate Microsoft® Excel® Documents
- User Interface Redesign

### **SOLIDWORKS MBD**

on page 151

- Limiting Geometric Tolerances to a Standard
- Dimension Extension Lines

## Performance

SOLIDWORKS® 2023 improves the performance of specific tools and workflows.

Some of the highlights for performance and workflow improvements are:


### SOLIDWORKS Fundamentals


You can use the **Level of detail** system option to improve the performance when manipulating views.

To access this option, click **Tools > Options > System Options > Performance**.

Performance is improved for **Zoom In/Out**, **Pan**, and **Rotate** in draft quality HLR/HLV and wireframe modes of models.

### Section Views of Models

Starting and exiting the **Section View**  command is about five times faster than before.

While you are in the **Section View**  command, adding or removing section planes is instantaneous.

### Assemblies

- Improved performance when searching with **Include sub-folders** selected.
- For assemblies that contain parts with multiple imported bodies and body level appearances, performance is improved when:
  - Opening a file
  - Changing a configuration
  - Applying, modifying, or removing an appearance
  - Rolling back or rolling forward assembly items
- The save efficiency of assemblies is improved to avoid saving unchanged model data. This performance improvement is most noticeable in very large assemblies.



## Import/Export

Performance of importing third-party CAD files is improved using 3D Interconnect.

## SOLIDWORKS PDM

- SOLIDWORKS PDM 2023 makes significant performance improvements in the presence of high latency to SOLIDWORKS PDM servers.

The improvements might vary depending on the amount of latency, size of data sets, and operations.

- The following operations perform better when latency to the database server is high:
  - Check-in
  - Check-in with automatic transition
  - Undo Check-out
  - Interactive operations such as menus, shortcut menus, reference dialog boxes, and SOLIDWORKS PDM File Explorer tabs
- The following operations perform better when latency to the archive server is high:
  - Add files
  - Change state on file modifications
  - Get files
  - Get latest version of files
- With performance improvements in the File Version Upgrade tool, you can upgrade SOLIDWORKS files in a SOLIDWORKS PDM vault faster.

If you are upgrading assembly files and you select **Overwrite** and **Latest Version** in the Version Settings page, the software skips upgrading and uploading references that are not the latest in the assembly's as-built version. This improves the performance of the upgrade process.
- In SOLIDWORKS PDM 2023 SP04, the mechanism for loading of SOLIDWORKS PDM and custom add-ins is reworked. As a result, operations such as logging in to a vault and starting the SOLIDWORKS PDM Search tool perform significantly better.

## Structure System and Weldments

Performance is improved when you create or edit a corner management feature. A corner gets trimmed when you select it under **Corners** in the Corner Management PropertyManager or in the graphics area.

Previously, the corners were trimmed as soon as you created or edited the corner management feature.

Performance is improved when you insert a structural member or change its profile to the part that includes weld beads.

## Configuration Publisher (2023 SP3)

Performance of the Configuration Publisher dialog box is improved when you add lists and check boxes from the palette into the **Edit** panel.

### For More Information



Use the following resources to learn about SOLIDWORKS:

#### **What's New in PDF and HTML**

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

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

#### **Interactive What's New**

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

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

#### **Online Help**

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

#### **SOLIDWORKS User Forum**

Contains posts from the SOLIDWORKS user community on the 3DEXPERIENCE® platform (login required).

#### **Release Notes**

Provides information about late changes to our products, including changes to the *What's New* book, online help, and other documentation.

#### **Legal Notices**

SOLIDWORKS Legal Notices are available [online](#).

# 2

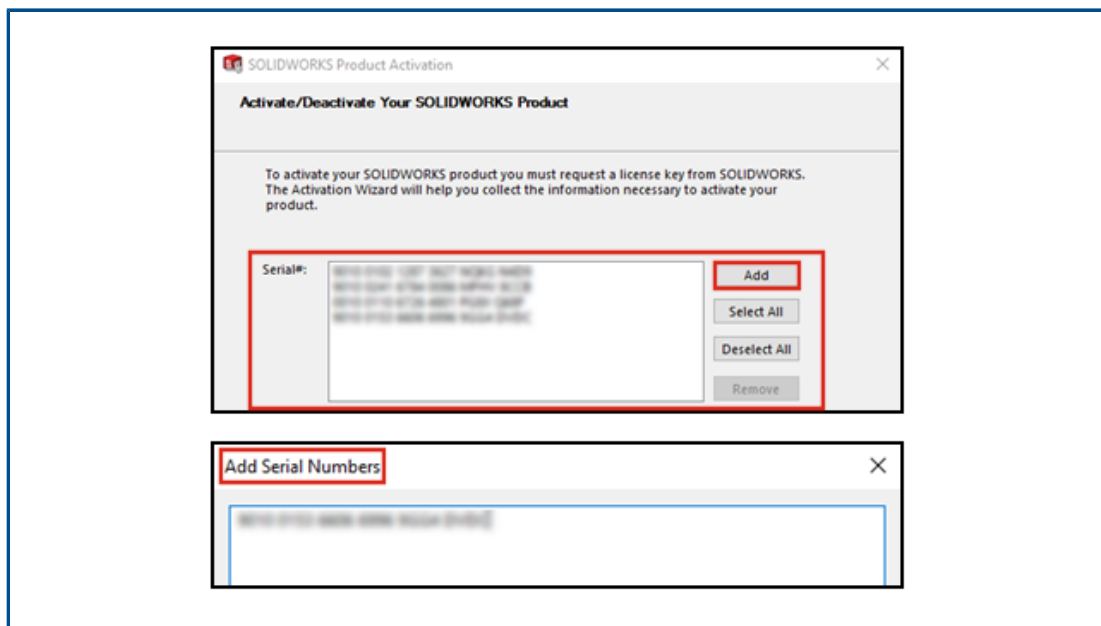
## Installation

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This chapter includes the following topics:

- **Adding Serial Numbers for SolidNetWork License Installations**
- **Renewing and Reactivating Term SNL Licenses**
- **Load SOLIDWORKS Flow Simulation Modules**
- **Faster Startup for Borrowed Licenses**

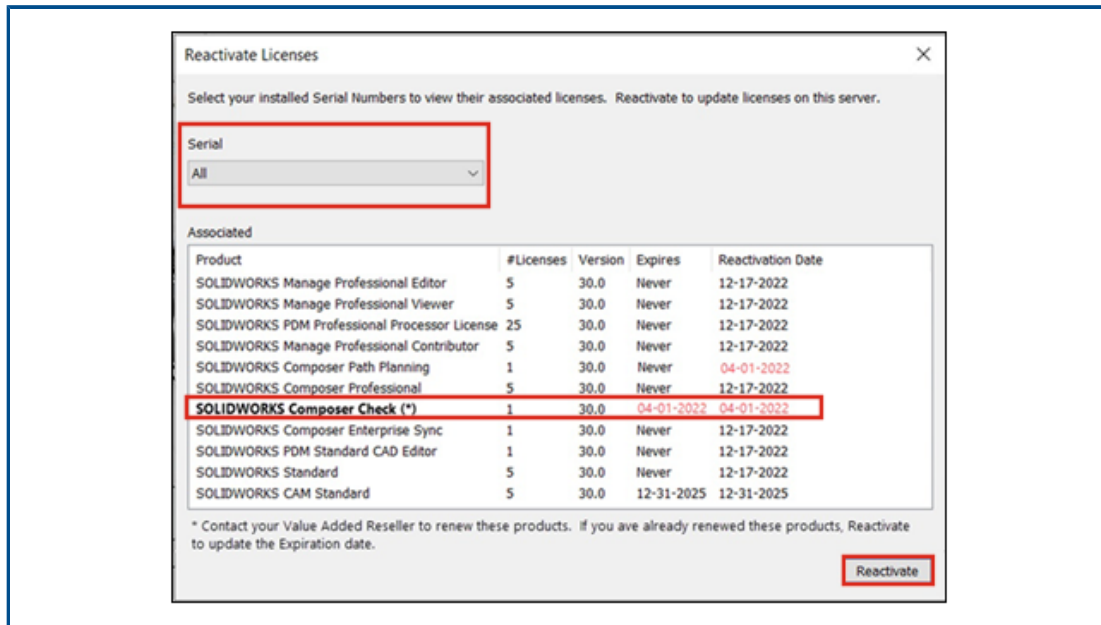
### Adding Serial Numbers for SolidNetWork License Installations



Adding serial numbers to licenses for all SOLIDWORKS applications and add-ins is much simpler for CAD administrators to handle.

You can use the Product Activation Wizard from the SOLIDWORKS SolidNetWork License Manager Server to add and activate serial numbers for perpetual and term licenses. Previously, serial numbers were entered when installing the License Manager Server. The new method makes it easier to add serial numbers to existing installations. You can also use the Activation Wizard to remove serial numbers that have not been activated.

## Renewing and Reactivating Term SNL Licenses

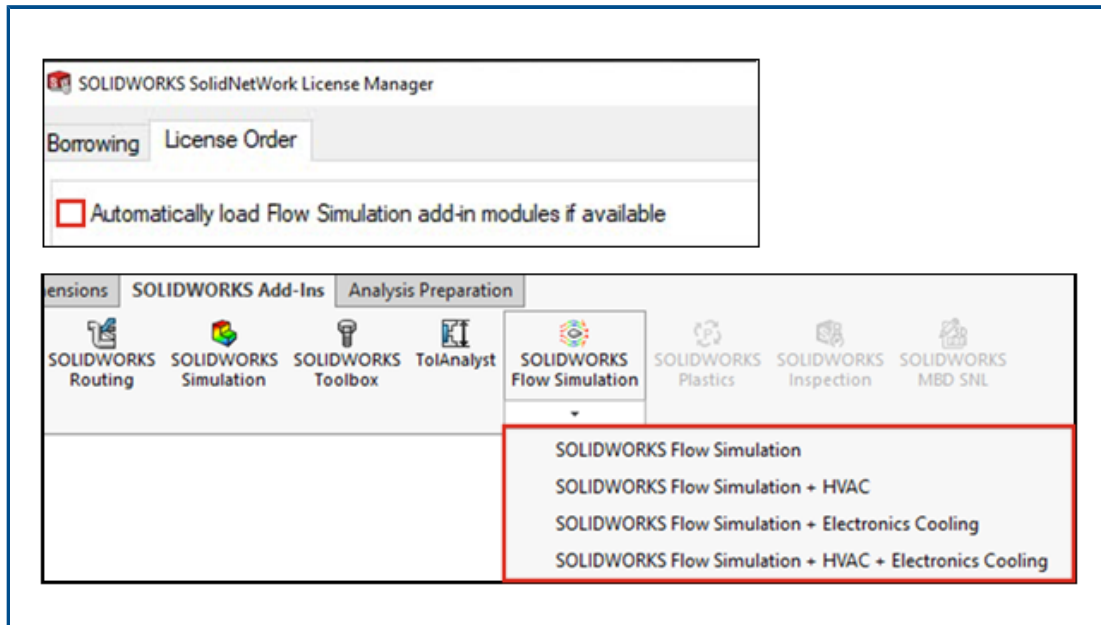


For term licenses, renewal and reactivation warnings display in all SOLIDWORKS add-ins, SOLIDWORKS PDM, and SOLIDWORKS Manage.

When users start these applications, if any term licenses are due to expire or require activation within 30 days, warnings notify the users and prompt them to contact their CAD administrator.

When you click **Reactivate** on the SolidNetWork License Manager **Server Administration** tab, the Reactivate Licenses dialog box highlights in red all license expiration and reactivation dates within 30 days. Asterisks mark licenses that require renewal. You can also filter and view licenses associated with a specific serial number.

## Load SOLIDWORKS Flow Simulation Modules



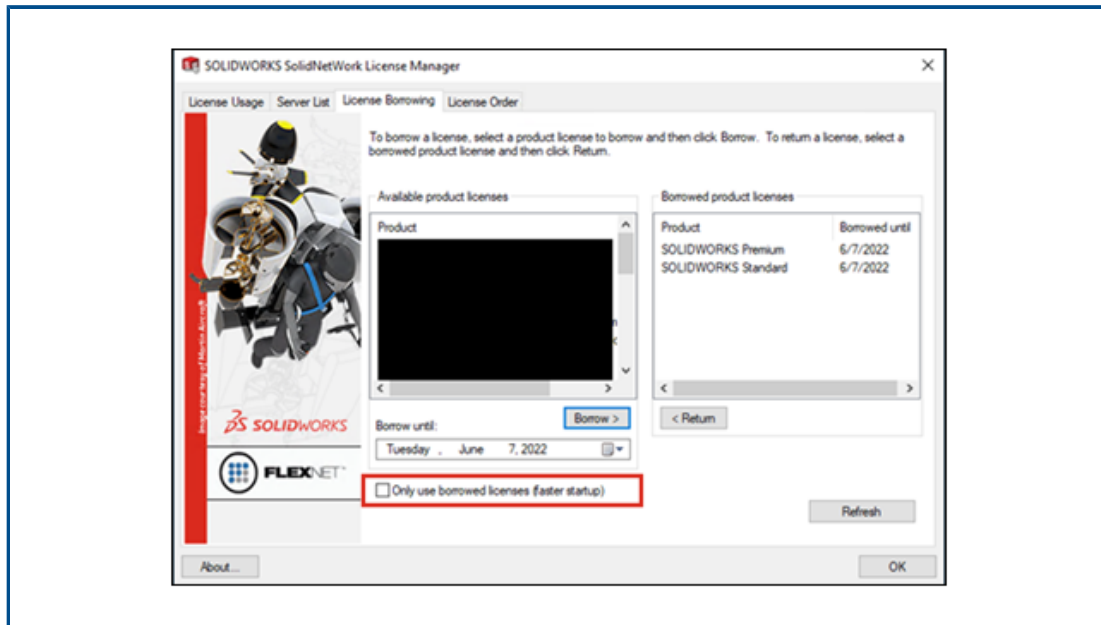
As a SolidNetWork license customer, you can control how to load the HVAC and Electronics Cooling add-in modules for Flow Simulation using two methods.

The first method lets you specify the default behavior for the add-ins when you load Flow Simulation. The option **Automatically load Flow Simulation add-in modules if available** is in the **SolidNetWork License Manager Client > License Order** tab. Clear this option if you do not have a need for the Flow Simulation add-ins. The HVAC and Electronics Cooling add-in modules do not load when Flow Simulation is added in with existing methods.

The second method lets you override default behavior and include the HVAC and Electronics Cooling add-in modules when you load Flow Simulation. Click the Down arrow ▼ on **SOLIDWORKS Flow Simulation** (SOLIDWORKS® Add-ins toolbar) or **SOLIDWORKS Flow Simulation** with one or both of the add-modules to load.

Without these methods, the licenses for the add-in modules are consumed whether or not you need them and may be denied to other users on your network.

## Faster Startup for Borrowed Licenses



If you are working remotely and your SOLIDWORKS apps are slow to open, you can make them faster by blocking license server queries and using only borrowed licenses.

License server queries are slow in some environments. Your apps will start faster if you block these queries, but will only use the licenses you have borrowed.

To block license server queries and use only borrowed licenses, click **Start > SOLIDWORKS Tools 2023 > SolidNetWork License Manager Client > License Borrowing > Only use borrowed licenses for faster startup** and restart your apps.



# 3

## SOLIDWORKS Fundamentals

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This chapter includes the following topics:

- **Application Programming Interface**
- **Changes to System Options and Document Properties**
- **Ellipses**
- **Performance Improvement When Manipulating Views**

### Application Programming Interface

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

### Support

There is API support for:

- Mate controller features in assemblies
- Simple, two-member, and complex corner treatments in structure systems

### Redesign

Sheet metal sketched bend features have been redesigned.

### Other Enhancements

- Get whether an assembly configuration is defeatured
- Get and set whether to use gauge table values in a sheet metal sketched bend feature
- Get whether to override a preset bend angle in a sheet metal sketched bend feature
- Get and set whether to symmetrically thicken a bidirectional base flange feature
- Get and set whether to symmetrically thicken a bidirectional lofted bends feature
- Convert a solid body to a mesh body
- Create a surface body from a mesh body
- Create a surface body from a graphics body

## Changes to System Options and Document Properties

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

### System Options

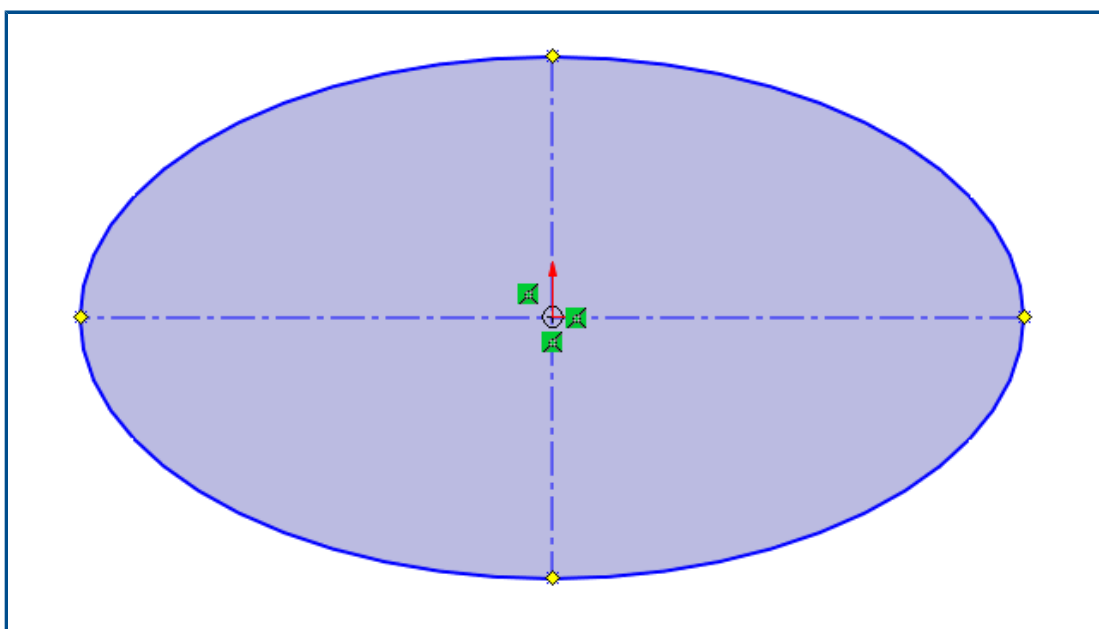
Option	Description	Access
<b>Drawings, Modified Cells (BOM)</b>	Changes the default color of overridden cells.	<b>Colors</b>
<b>Sketch, Exploded</b>	Specifies a color for an explode line.	<b>Colors</b>
<b>Automatically optimize resolved mode, hide lightweight mode</b>	Loads component data as resolved when you open an assembly.	<b>Performance</b>
<b>Manually manage resolved and lightweight modes</b>	Controls when a component loads in lightweight or resolved mode.	<b>Performance</b>
<b>Load component lightweight</b>	Renamed from <b>Automatically load components lightweight</b> .	<b>Performance</b>
<b>Export assembly components as separate STEP files (recommended for large assemblies)</b>	Exports assemblies as atomic STEP files.	<b>Export</b>
<b>Export appearances</b>	Exports file's appearances with reduced performance. Clear the option to skip exporting appearances but make the performance faster.	<b>Export</b>

### Document Properties

Option	Description	Access
<b>Allow tolerance symbols for all drafting standards</b>	Includes all geometric tolerance symbols regardless of the drafting standard. Clear the option to limit symbols to the document's drafting standard.	<b>Annotations &gt; Geometric Tolerances</b>
<b>Connect dimension extension lines to model</b>	Completes connection of dimension extension lines to model.	<b>DimXpert &gt; Display Options</b>

Option	Description	Access
<b>See through transparent components in HLR/HLV</b>	(Available for <b>High quality</b> drawing views only). Shows edges behind transparent components with solid lines.	<b>Drawings &gt; Detailing</b>
<b>Default Bill of Materials part number for new configurations</b>	Specifies a value for the default BOM part number. Options are <b>Document Name</b> or <b>Configuration Name</b> .	<b>Configurations</b>

## Ellipses



When you sketch an ellipse, you can apply construction geometry to the major and minor axes.

In the Ellipse PropertyManager, under **Options**, select **Add construction lines**.

## Performance Improvement When Manipulating Views

You can use the **Level of detail** system option to improve the performance when manipulating views.

To access this option, click **Tools > Options > System Options > Performance**.

Performance is improved for **Zoom In/Out**, **Pan**, and **Rotate** in draft quality HLR/HLV and wireframe modes of models.

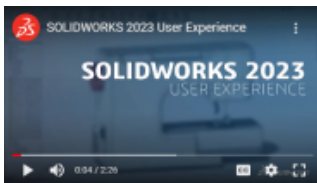
# 4

## User Interface

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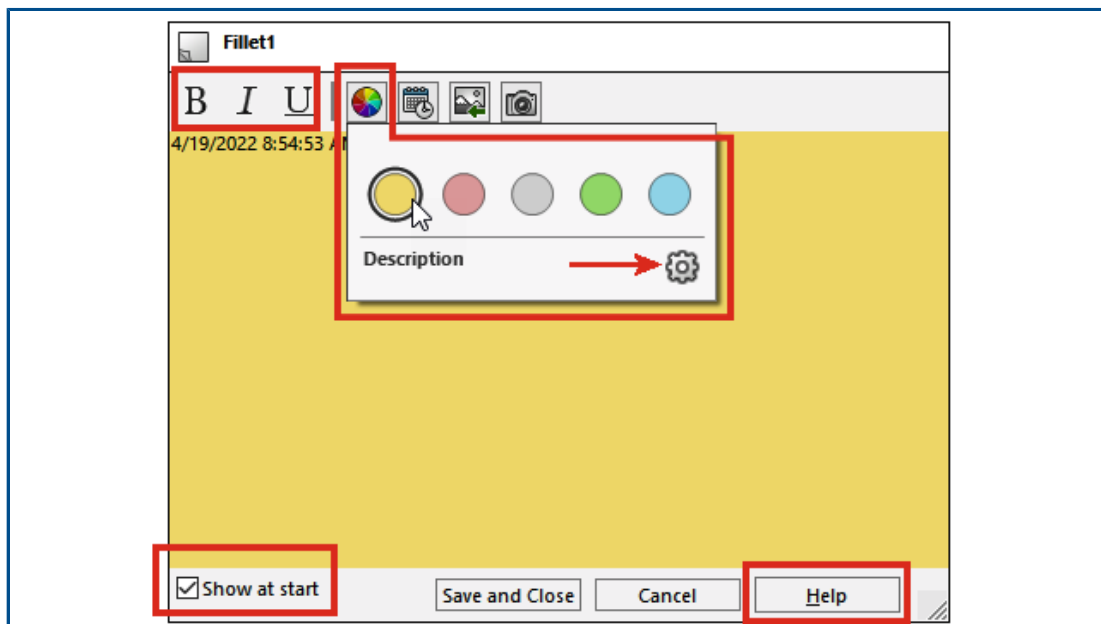
This chapter includes the following topics:

- **Comments Enhancements**
- **Enforcing Legal File Names**
- **Restoring Default Settings**
- **Component Name Display**



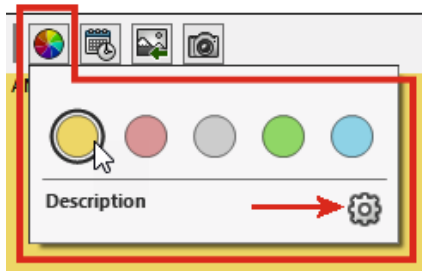
**Video: What's New in SOLIDWORKS 2023 - User Experience**

### Comments Enhancements

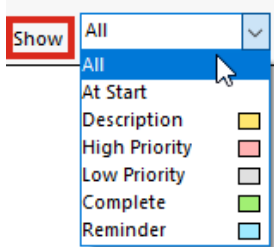


Comments are enhanced to improve usability.

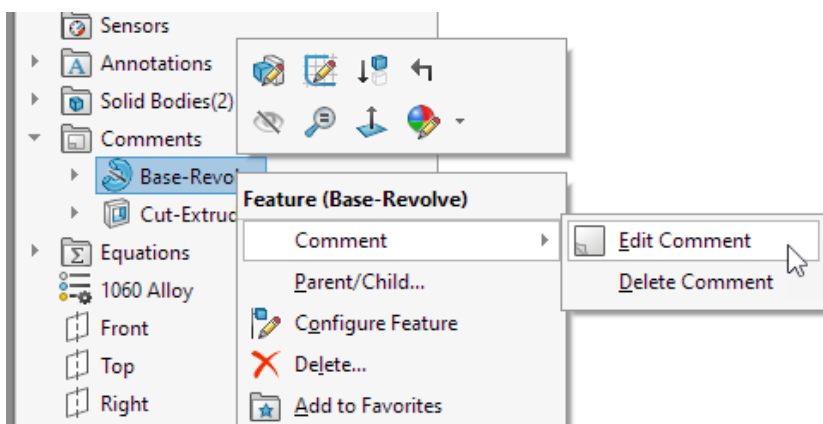
- You can choose different background colors and add bold **B**, italics *I*, and underline U to text.
- Click **Options** in the Background Colors tab to open the **Tools > Options > Document Properties > Model Display** dialog box where you can specify the **Comment background colors**. You can customize the names or reset all names back to the default. You cannot customize the colors.



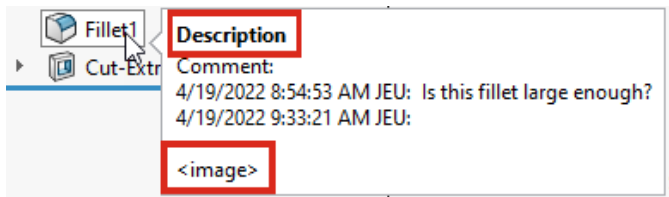
- Select **Show at start** in various places to show the comments in the View All Comments dialog box when you reopen the file.
- In the View All Comments dialog box, in the **Show at start** column, you can select the comments to show when you open the file. In the **Show** menu, you can specify the comments to show at the start or based on a color. Click **Apply** to save changes to the **Show at start** status of comments.



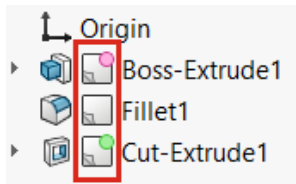
- When you right-click a comment, a context toolbar and shortcut menu appear so you can directly edit features when you read the comments.



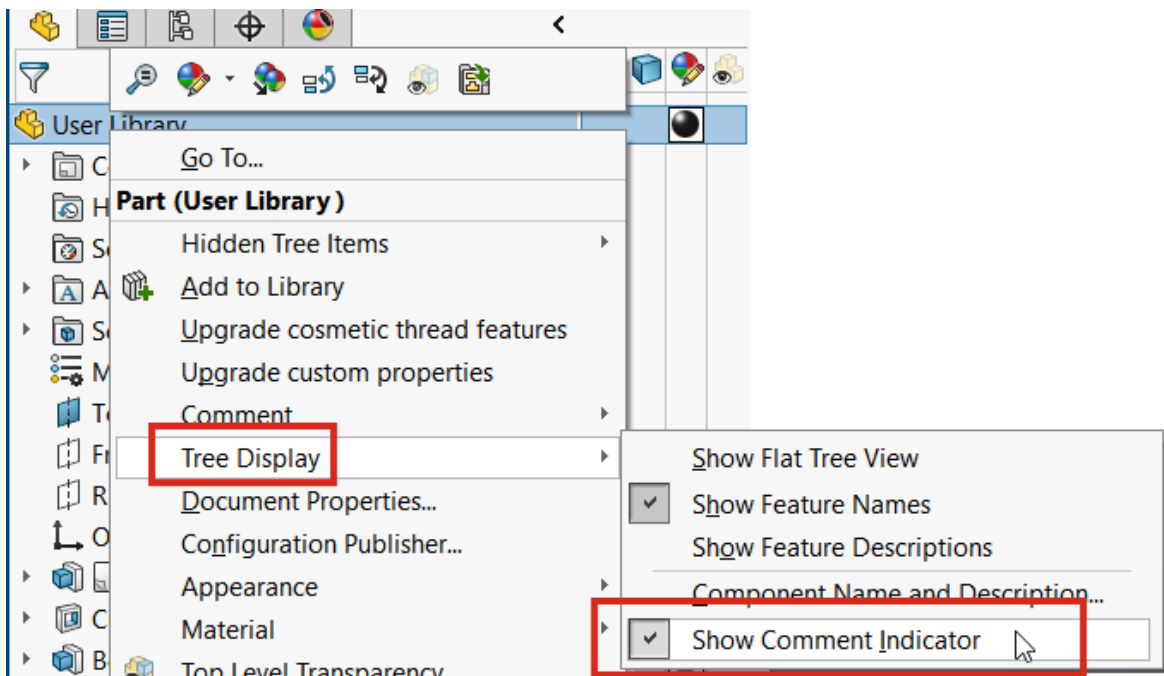
- Comment tooltips show the text `<image>` to indicate that the full comment contains an image. The tooltip name matches the name from the **Comment background colors** section in the Model Display dialog box.



- Images in comments retain their current aspect ratio and the display quality is improved.
- When you import images into comments, in the Open dialog box, the **All files (\*.bmp,\*.jpg,\*.tif,\*.tiff,\*.png)** option is the default to display all acceptable image types.
- In the FeatureManager® design tree, comment indicators appear by default next to features that have comments. The color circle matches the color of the comment.

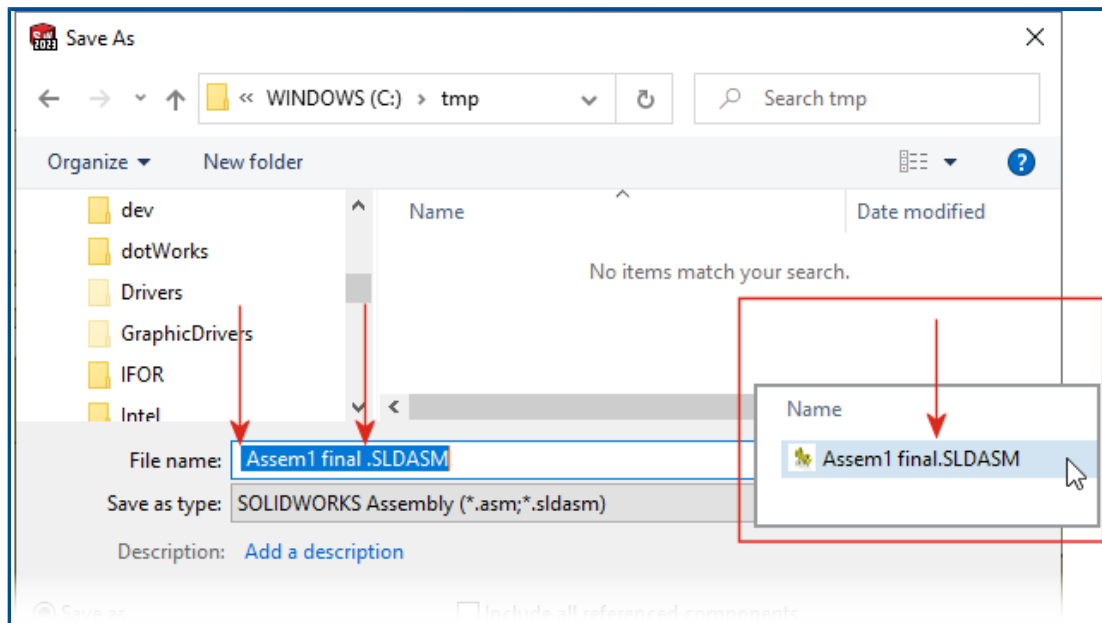


To control the display of the comment indicators, in the FeatureManager design tree, right-click the part node and click **Tree Display > Show Comment Indicator**.





## Enforcing Legal File Names



To ensure compatibility with the **3DEXPERIENCE** platform, you cannot save new SOLIDWORKS documents with preceding or trailing spaces in their names. The software automatically removes the preceding or trailing spaces when you save the file. Spaces in the middle of the file name are supported.

This enforcement applies in these cases:

- Save As dialog box, including the **Advanced** option
- Renaming parts and assemblies in the FeatureManager design tree
- Mirror Components PropertyManager
- Split PropertyManager and Save Bodies PropertyManager
- Pack and Go

## Restoring Default Settings



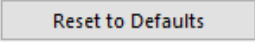
You can more easily restore the factory default settings in SOLIDWORKS®.

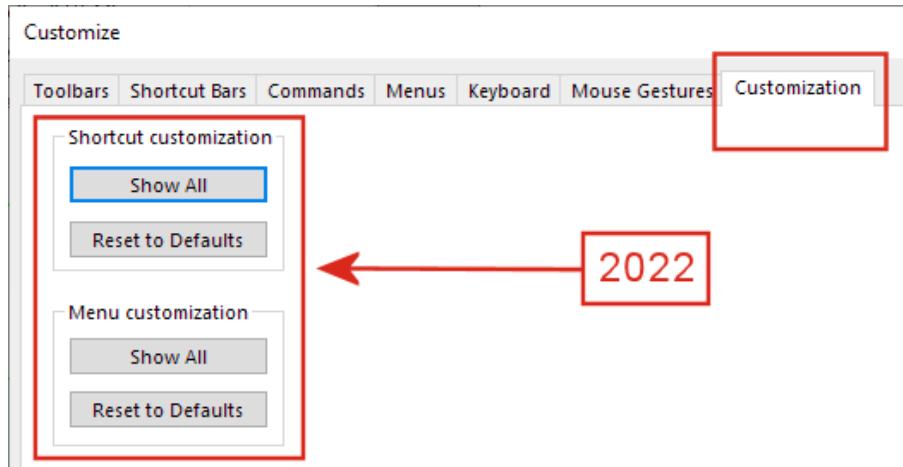
### SOLIDWORKS Settings Wizard

The Copy Settings Wizard is renamed to the SOLIDWORKS Settings Wizard that has an improved and simplified user interface with three options:

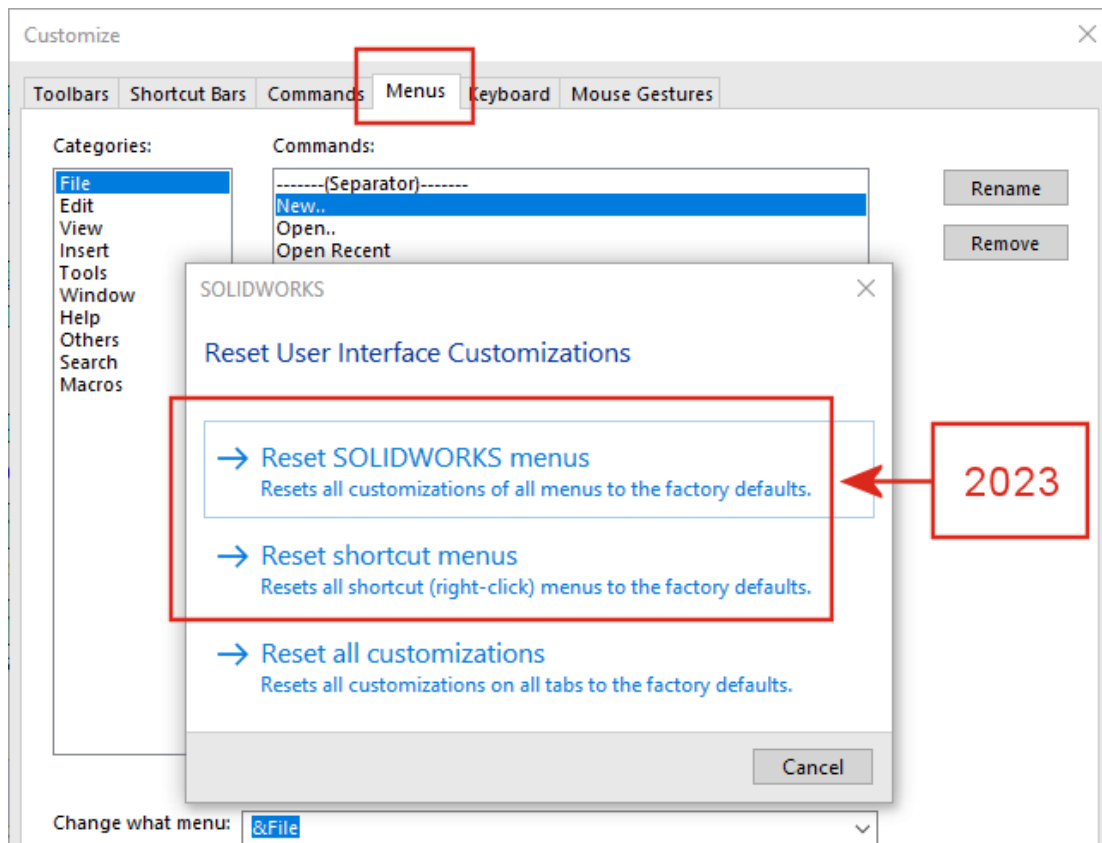
- |                         |   |
|-------------------------|---|
| <b>Save Settings</b>    | Saves SOLIDWORKS settings to a file. You choose the location for the settings file and the settings to save.  |
| <b>Restore Settings</b> | Restores SOLIDWORKS settings from a file.   |
| <b>Reset Settings</b>   | Resets all custom settings and returns SOLIDWORKS to the factory defaults. You have the option to save all custom settings to a file before resetting SOLIDWORKS to the factory defaults. |

### Customize Dialog Box

- The  button appears at the bottom of each tab in the Customize dialog box so you can customize the settings to reset.
- The Customization tab is removed.



The options from the Customization tab are reorganized as follows:



- **Shortcut customization:**

- **Show All**

Removed because this was rarely used. The preferred method is to keep the menu small and click  $\nabla$  to expand the list

- **Reset to Default**

Moved to **Menus tab > Reset shortcut menus.**

- **Menu customization:**

**Show All**

Removed because there was very little difference between this command and **Reset to Default** for this customization.

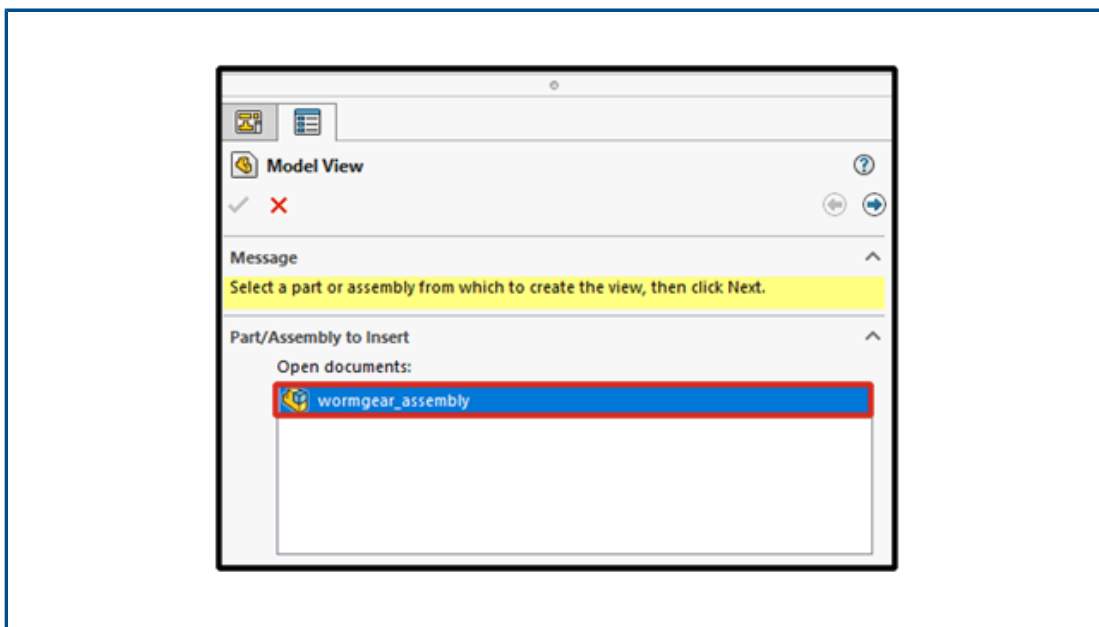
**Reset to Default**

Moved to **Menus tab > Reset SOLIDWORKS menus.**

## Customizations and Upgrades

When you upgrade to a new version of SOLIDWORKS, the software keeps all the customizations and also shows any new tabs, menus, toolbars, etc. that were added to the software. Previously, warning dialogs prompted you to make choices that might have led to lost settings.

## Component Name Display



When you specify display options for component names in a drawing, the names in the PropertyManager and View Palette match those in the FeatureManager design tree.

**To specify display options for component names:**

1. In the FeatureManager design tree, right-click the drawing name and click **Tree Display > Component Name and Description.**
2. (Optional.) Under **Primary**, specify an option:
  - **Component Name**
  - **Component Description**

Based on the selection under **Primary** display options:

- The Model View and Standard 3 View PropertyManagers update the **Open documents** field.
- The View Palette updates. The View Palette displays the part or assembly icon and the file name.


# 5

## Parts and Features

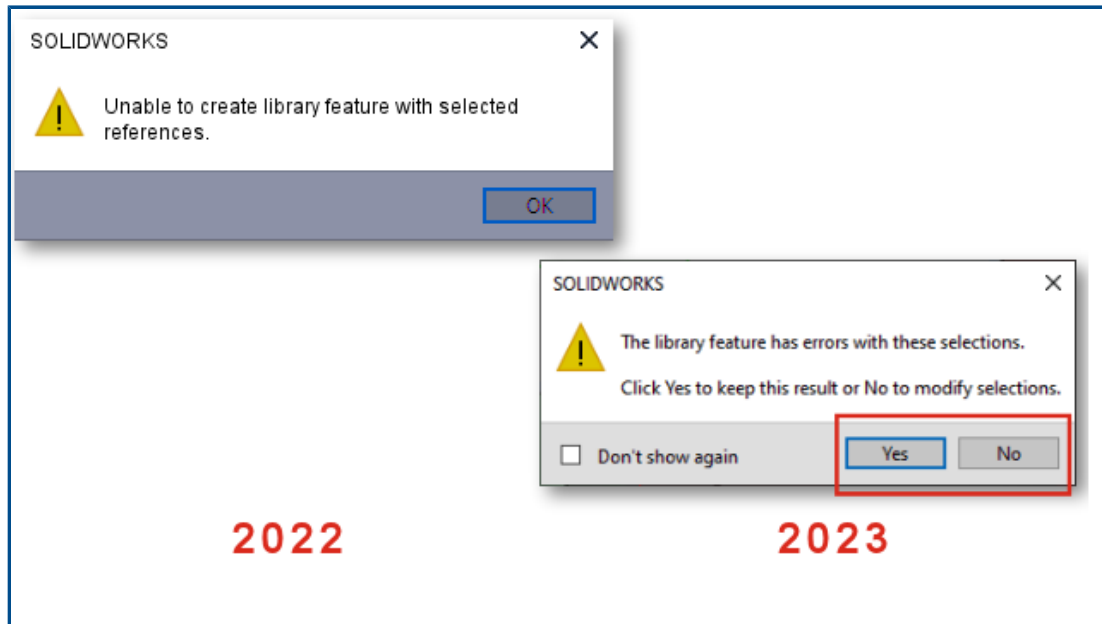
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This chapter includes the following topics:



- **Allowance of Library Feature Errors**
- **Coordinate Systems**
- **Defeatured Parts as Derived Configurations (2023 SP1)**
- **Dimension Support for the Move/Copy Body Command**
- **Hybrid Mesh BREP Modeling Improvements**
- **Mesh BREP Display Improvements (2023 SP2)**
- **Section View Performance Improvements**
- **Section View through Axis**
- **Single-Line Fonts for Wrap Features**

	<p><b>Video: What's New in SOLIDWORKS 2023 - Parts and Features</b></p>
---	---

## Allowance of Library Feature Errors

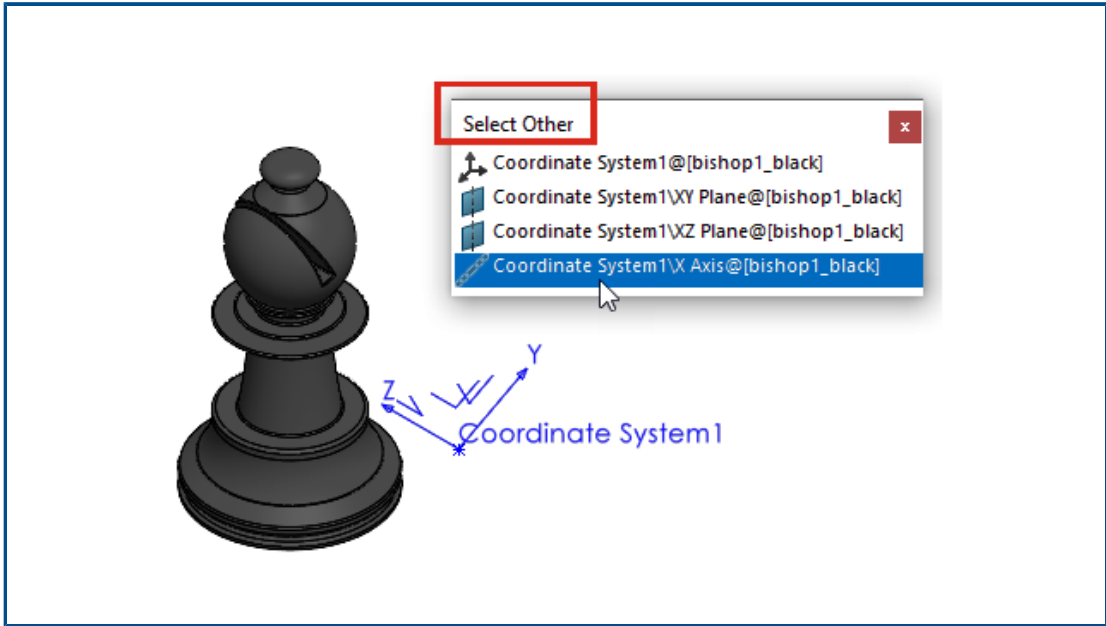


When you insert a library feature into a model and get error messages, you can keep the results to view and manually repair the errors.





In the warning message, click **Yes** to keep the results. In the FeatureManager® design tree, you can manually repair the library feature or click **Undo**  to remove it. If you click **No** in the warning message, in the Library Feature PropertyManager, you can modify the selections or click  to restore the FeatureManager design tree without the library feature.

Previously, the warning message blocked you from inserting library features that generated errors.

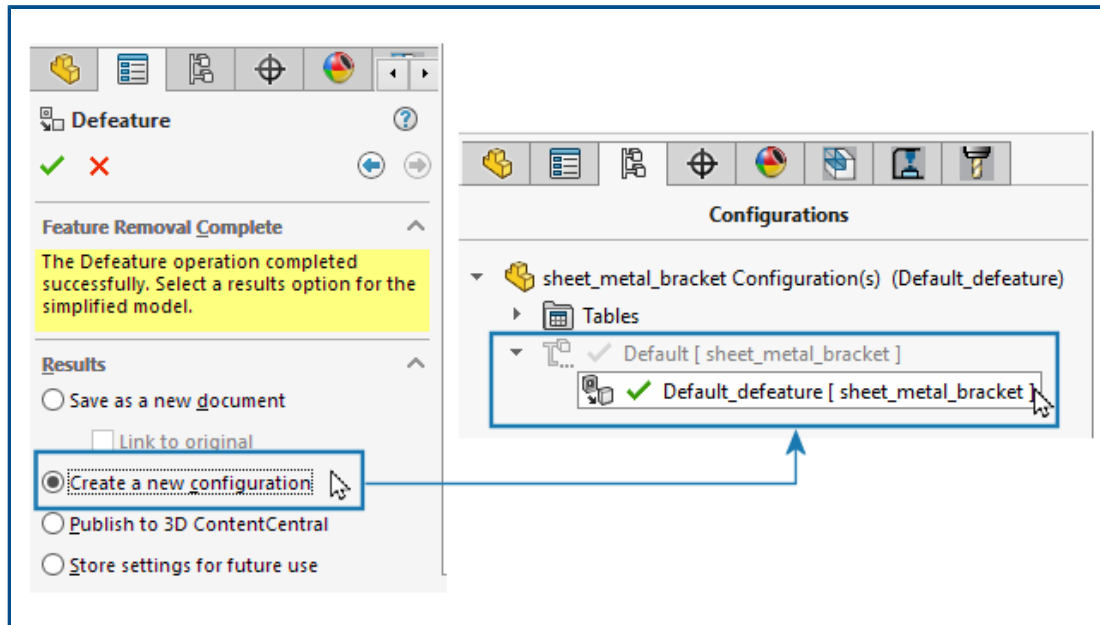
## Coordinate Systems



You can reference coordinate system elements from additional entities, features, and commands, including the **Select Other** command.

Item	Description
3D sketches	Create sketch relations between 3D sketch entities and coordinate system axes, planes, and origin.
2D sketch dimensions	Create 2D sketch dimensions that reference coordinate system axes, planes, and origin.
<ul style="list-style-type: none"><li>• <b>Mirror</b>  features and bodies</li><li>• <b>Mirror Components</b> </li><li>• <b>Insert &gt; Mirror Part</b></li></ul>	Select a coordinate system plane as the mirror plane.
Reference <b>Plane</b> 	Select a coordinate system axis to create a new plane perpendicular to that axis. Use a vertex or point from the model to position the plane.
<b>Select Other</b> 	Select coordinate system axes, planes, and origin.

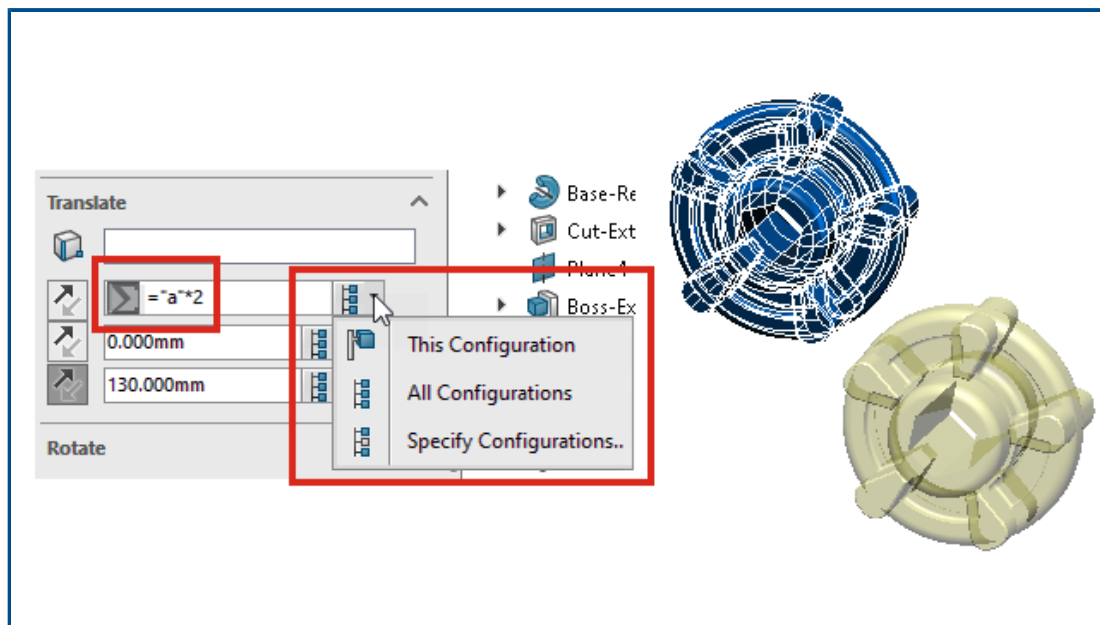
## Deeatured Parts as Derived Configurations (2023 SP1)



You can save a defeatured version of a part as a derived configuration.

In the Defeature PropertyManager, on the Feature Removal Complete tab, under **Results**, select **Create a new configuration**. You can quickly switch between the parent configuration and the defeatured configuration. You also have fewer files to manage. Previously, you could create only a separate external defeatured file.

## Dimension Support for the Move/Copy Body Command



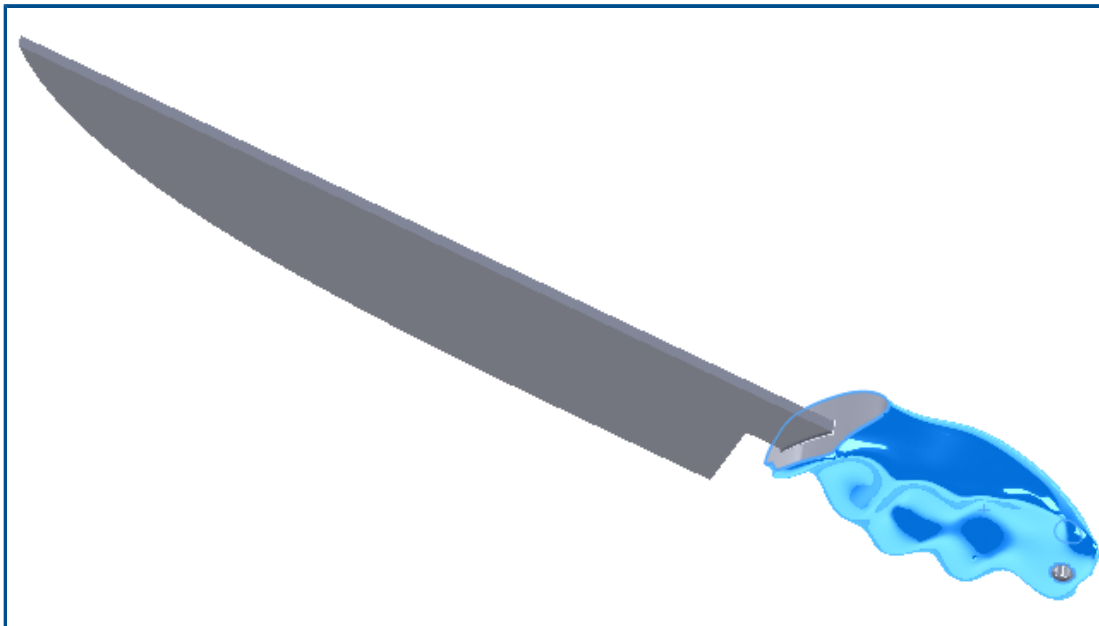


When you use the **Move/Copy Body** command, you can control the **Translate** and **Rotate** values with equations. You can add these equations to a design table and control them there. You can double-click the dimensions in the graphics area and specify them using the **Modify** dialog box.

Under **Rotate**, you can select **Euler's angular rotation** and specify values for **Yaw** (side-to-side), **Pitch** (up-down), and **Roll** (twist). This method delivers the same resulting rotation regardless of the order in which you specify these values.

If models have configurations, you can specify the configurations to which the dimensions apply for **Translate** and for **Rotate** when you select **Euler method of angular dimensions**.

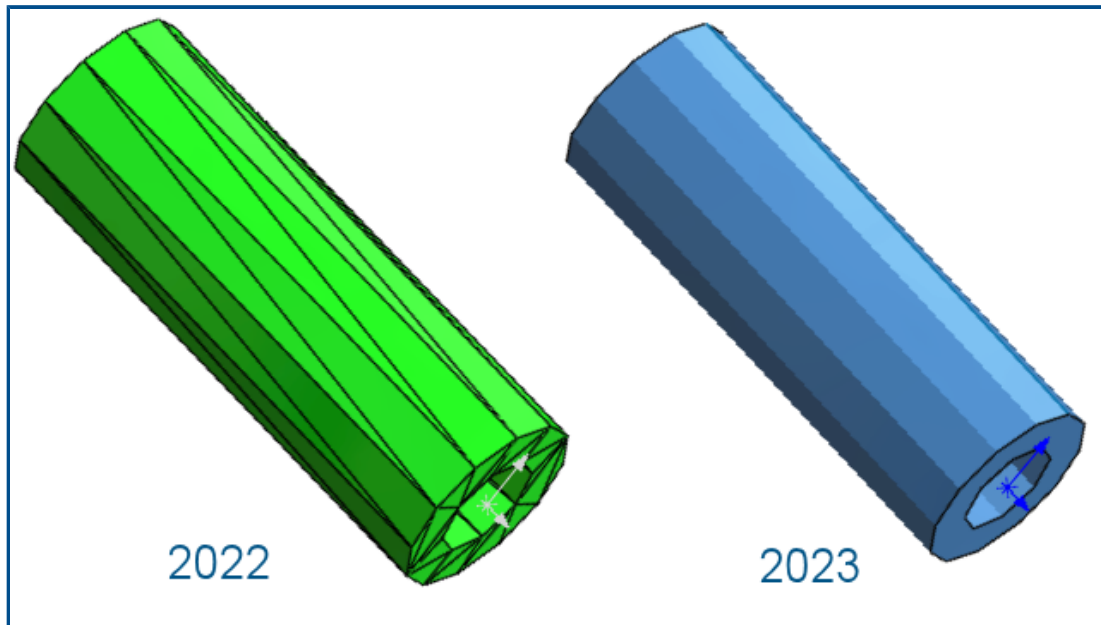
## Hybrid Mesh BREP Modeling Improvements




Hybrid mesh BREP modeling is improved to support more SOLIDWORKS® BREP features that you can combine with mesh BREP geometry.


Hybrid mesh BREP modeling now supports features such as boss and cut sweeps, lofts, and boundaries. For a complete list of supported features, see *SOLIDWORKS Online Help: List of Features Supporting Hybrid BREP Bodies*.

## Mesh BREP Display Improvements (2023 SP2)



For mesh BREP bodies, you have the option to display or hide facet fins. In **Tools > Options > System Options > Display**, select or clear the option **Display facet fins in mesh BREP bodies**.


This includes bodies you create with the **Convert to Mesh Body**  tool. It also includes bodies imported from \*.stl and \*.3mf files with the **Graphics Body** import option selected and that you later convert to mesh BREP bodies.


This option does not affect the display of regular edges on mesh BREP bodies, which includes edges created directly on import or created later with the **Segment Imported Mesh Body**  tool.

Previously, there was no option to control the display of facet fins.

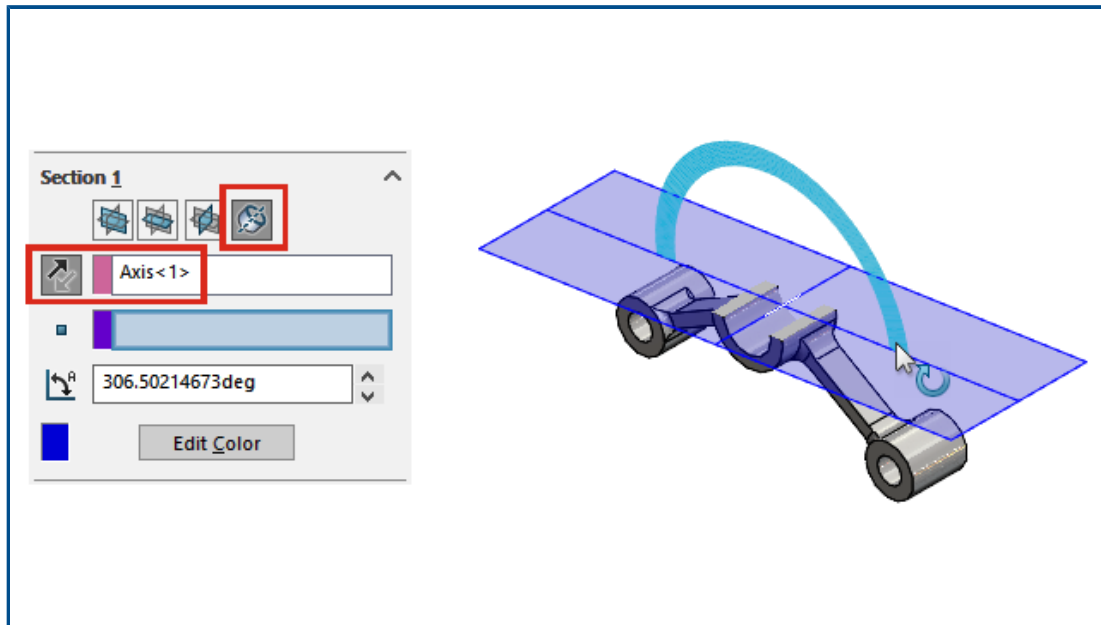
## Section View Performance Improvements



Performance is improved when you work with section views in models.

Starting and exiting the **Section View**  command is about five times faster than before.

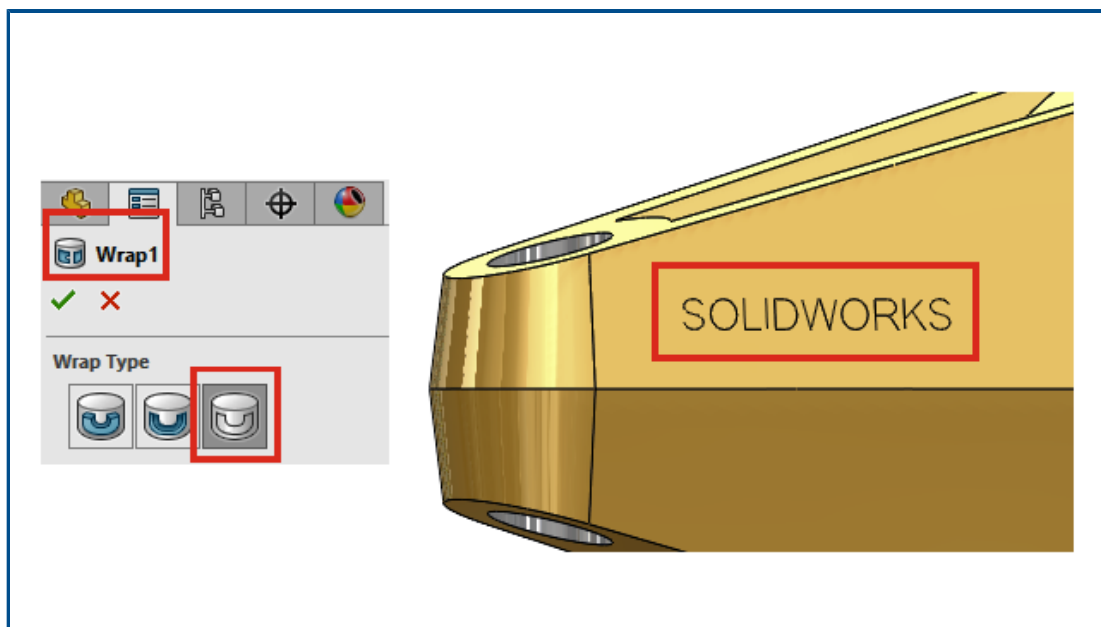
While you are in the **Section View**  command, adding or removing section planes is instantaneous.


## Section View through Axis



You can define a section view through an axis without constraining it to a secondary point, such as a vertex. Previously, a secondary selection was mandatory. When you click **Plane about axis** , you can click **Reverse Section Direction**  to flip the section view across the created plane.

## Single-Line Fonts for Wrap Features



You can create wrap features from sketches that use single-line fonts when you select **Scribe**  for **Wrap Type** in the Wrap PropertyManager. Previously, wrap features did not support single-line fonts.

Single-line fonts are also called stick fonts. This functionality is particularly helpful for users who want to engrave stick fonts on parts.

If text or characters do not display properly, you may be using an inauthentic single-line font. To check the font, dissolve the sketch text and review the text entities.

# 6

## Sheet Metal

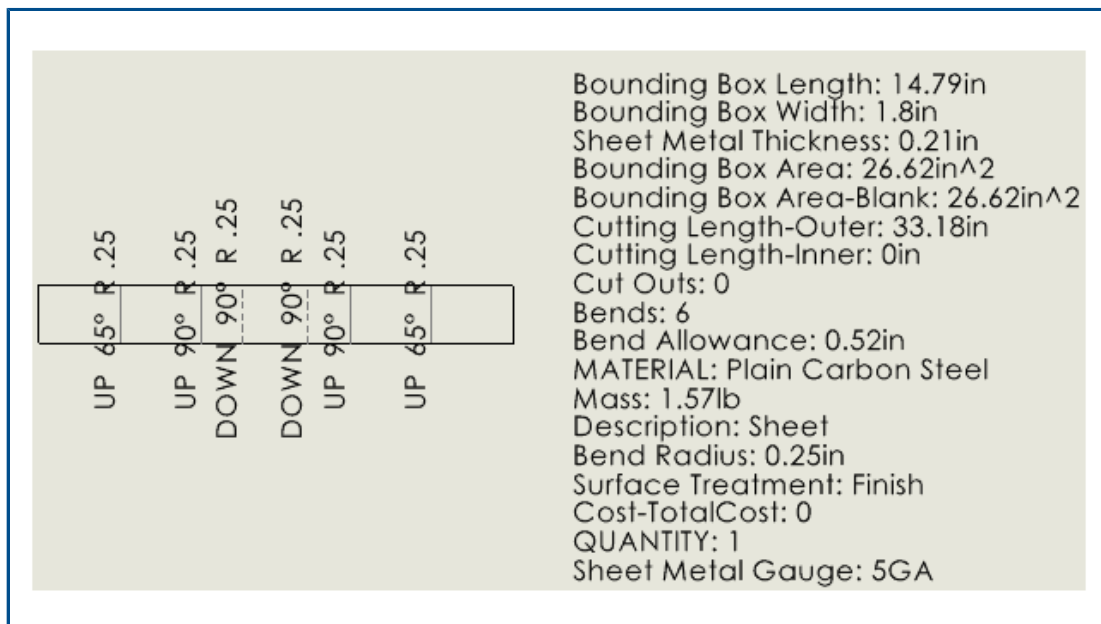
This chapter includes the following topics:

- **Gauge Values in Cut List Properties**
- **Sensors**
- **Sensor PropertyManager**
- **Symmetrical Thickness**



**Video: What's New in SOLIDWORKS 2023 - Sheet Metal**

### Gauge Values in Cut List Properties



If you use gauge tables to define the thickness of sheet metal parts, the gauge number appears in the Cut-List Properties dialog box. In the associated drawing, the gauge value is available for annotations and cut lists.

In the Cut-List Properties dialog box, the **Evaluated Value** for **Sheet Metal Gauge** is **Gauge Number GA**. For example, if you use a 3 Gauge in a sheet metal part, the **Evaluated Value** is **3 GA**.

In drawings, you can link an annotation to the **Sheet Metal Gauge** property and you include the property in a cut list.

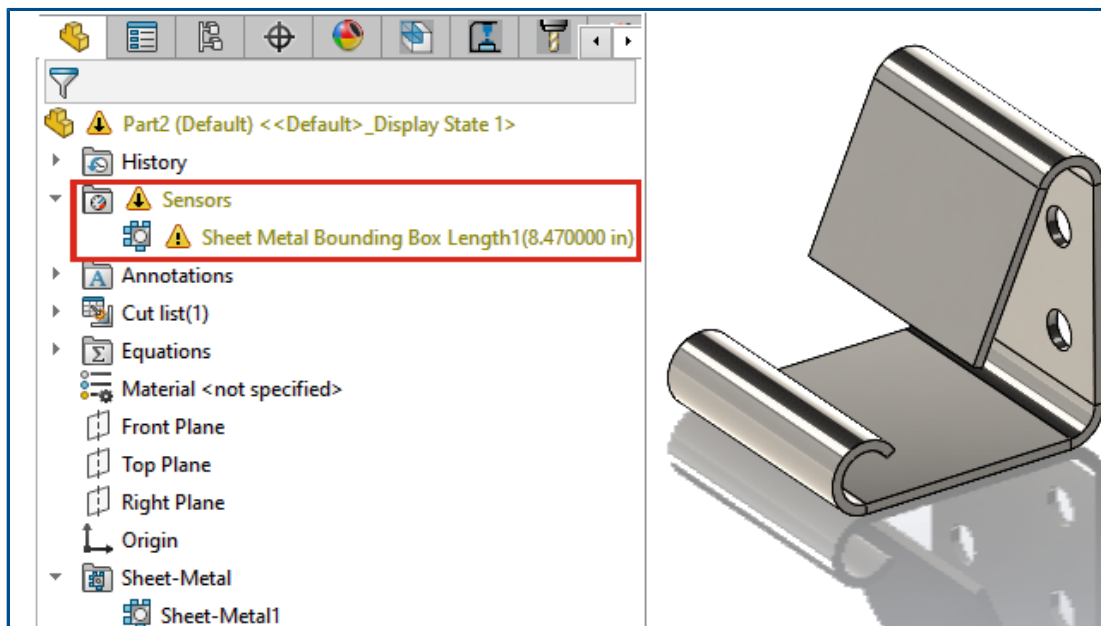
#### To link annotations to gauge values:

1. In a drawing of a sheet metal part, click an annotation, such as **Note A**.
2. In the PropertyManager, click **Link to Property**.
3. In the Link to property dialog box, in **Property name**, select **Sheet Metal Gauge**.

#### To include gauge value properties in cut lists:

1. Right-click a flat pattern view and select **Annotations > Cut List Properties**.
2. Click in the sheet to place the cut list properties.

## Sensors




You can use sensors to alert you when sheet metal models deviate from the limits specified for a bounding box. In multibody parts, you can create sensors for individual bodies.

During the design process, if the model exceeds the bounding box parameters, an alert appears in the FeatureManager® design tree. You can double-click an alert to see the values in the Cut-List Properties dialog box.


## Sensor PropertyManager

Updates to the Sensor PropertyManager include support for sheet metal models.


### Message


Displays a message if you do not select **Create Cut Lists Automatically** and **Update Automatically**. Sheet metal bounding box sensors depend on up-to-date cut list properties. In the FeatureManager design tree, right-click **Cut list**  and select **Create Cut Lists Automatically** and **Update Automatically** to generate automatic alerts.

### Sensor Type

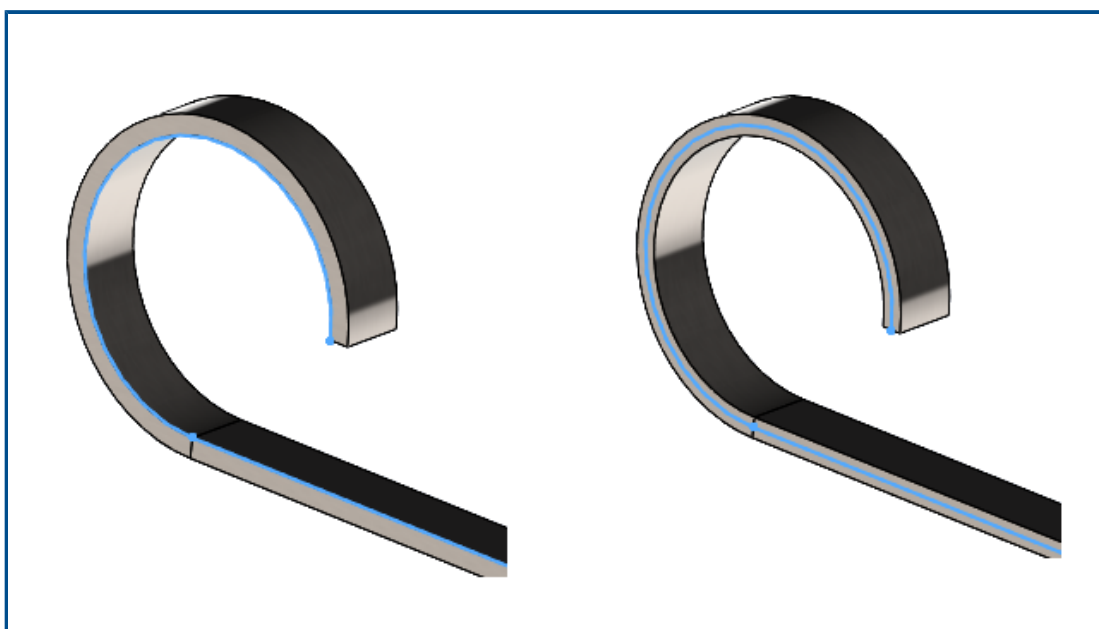
	<b>Sheet Metal Bounding Box Properties</b>	Specifies the smallest rectangle in which the flat pattern can fit.
---	--	---

### Properties

	<b>Sheet Metal Bounding Box Length</b>	<p>Specifies the longest side of the bounding box. For multibody parts, this is the sum of the bounding box length for all bodies in the part.</p> <p>For assemblies, this is the sum of the bounding box length for all parts in the assembly.</p>
	<b>Sheet Metal Bounding Box Width</b>	<p>Specifies the shortest side of the bounding box. For multibody parts, this is the sum of the bounding box width for all bodies in the part.</p> <p>For assemblies, this is the sum of the bounding box width for all parts in the assembly.</p>
	<b>Sheet Metal Bounding Box Area</b>	<p><b>Sheet Metal Bounding Box Length</b> × <b>Sheet Metal Bounding Box Width</b></p> <p>For multibody parts, this is the sum of the bounding box area for all bodies in the part.</p> <p>For assemblies, this is the sum of the bounding box area for all parts in the assembly.</p>

<b>Sheet Metal Bounding Box Area Blank</b>	Specifies the area of the flat pattern excluding through cut-outs. For multibody parts, this is the sum of the bounding box area blank for all bodies in the part.  For assemblies, this is the sum of the bounding box area blank for all parts in the assembly.
	<b>Sheet Metal Bodies to Monitor</b> Specifies the individual body or bodies to which the sensor applies.

## Symmetrical Thickness



When you create a sheet metal part as a base flange or lofted bend (with the **Bent Manufacturing Method**), you can specify a symmetrical thickness that adds an equal amount of material to both sides of the sketch.

Symmetrical thickness helps you create sheet metal parts from sketches to help achieve equal bend radii for upward and downward bends. In the image above, **Symmetric** is cleared for the example on the left and selected for the example on the right.

In the Base Flange or Lofted Bends PropertyManager, under **Sheet Metal Parameters**, select **Symmetric**.



# 7

## Structure System and Weldments

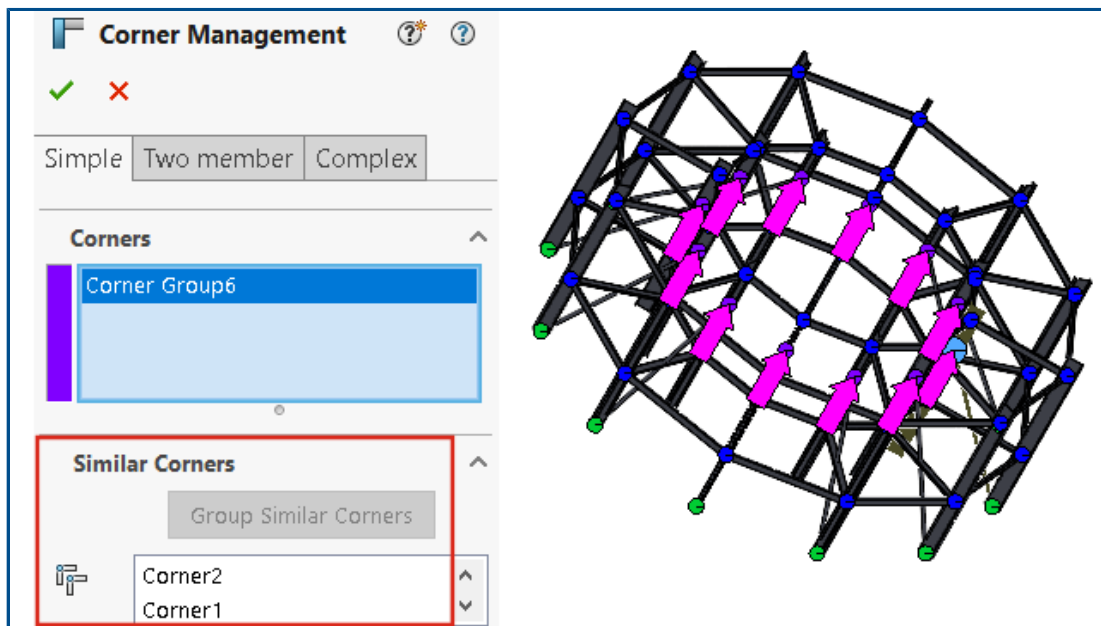
This chapter includes the following topics:

- **Corner Treatment to Similar Corners in Structure System**
- **Pattern Connection Element PropertyManager**
- **Configuration-Specific Size of Weldment Members**
- **Structure System and Weldments Performance Improvement (2023 SP4)**



**Video: What's New in SOLIDWORKS 2023 - Structure Systems**

### Corner Treatment to Similar Corners in Structure System



You can use **Group Similar Corners** to group similar corners, apply a corner treatment to the group, and edit the groups as required.

This is helpful for large structure system files that have multiple corners.

1. Open the structure system file.
2. In the FeatureManager® design tree, click **Corner Management** and select **Edit Feature**.
3. In the Corner Management PropertyManager, under **Corners**, select a corner.
4. Click **Group Similar Corners**.

The similar corners are grouped together, appear as **Corner Group xx** under **Corners** in the PropertyManager, and highlighted with arrows in the graphics area.

5. Apply corner treatment to the group.

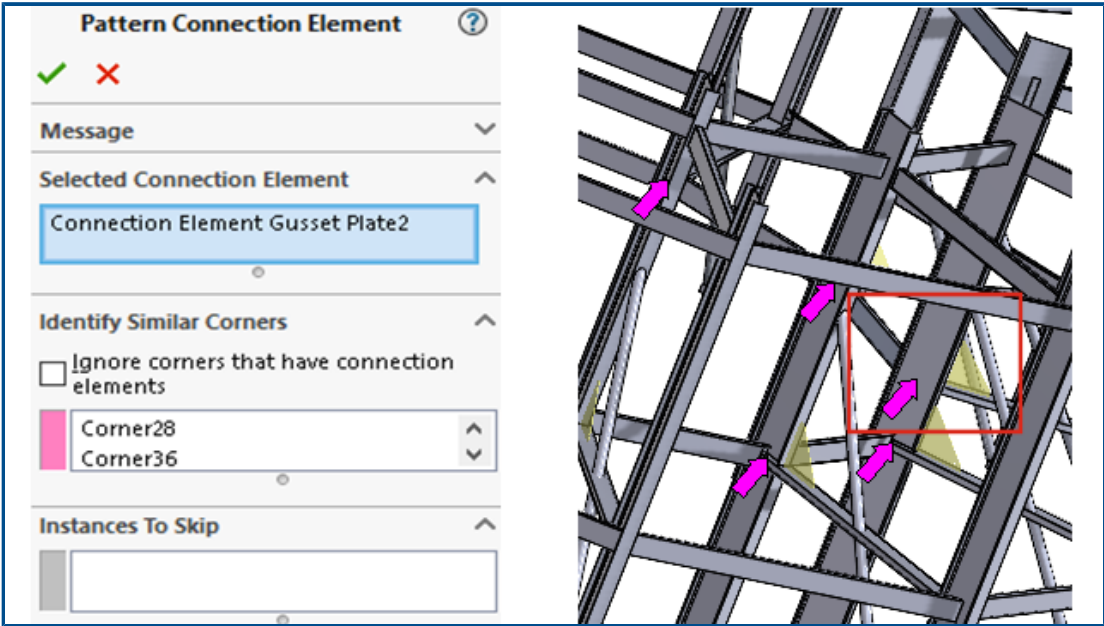
The FeatureManager design tree groups the similar corners as **Corner Group XX**. In the Corner Management PropertyManager, you can edit a group of similar corners and remove a corner from the group. You can add this corner back to the group of similar corners, if required.

- Right-click a corner of the group and select one of the following:

<b>Remove from Corner group</b>	Removes the selected corner from the group of similar corners
	Right-click the removed corner and select <b>Include back to Corner group</b> .
<b>Zoom to Selection</b>	Zooms in the selected corner
<b>Clear Selections</b>	Degroups the corners
	To degroup the corners, you can also right-click the group and select <b>Dissolve Corner group</b> .
<b>Customize Menu</b>	Lets you select the context menu

You can use similar corner treatment for simple, two member, and complex corners.

## Pattern Connection Element PropertyManager



In the Pattern Connection Element PropertyManager, you can insert a connection element that is at a corner to similar corners.

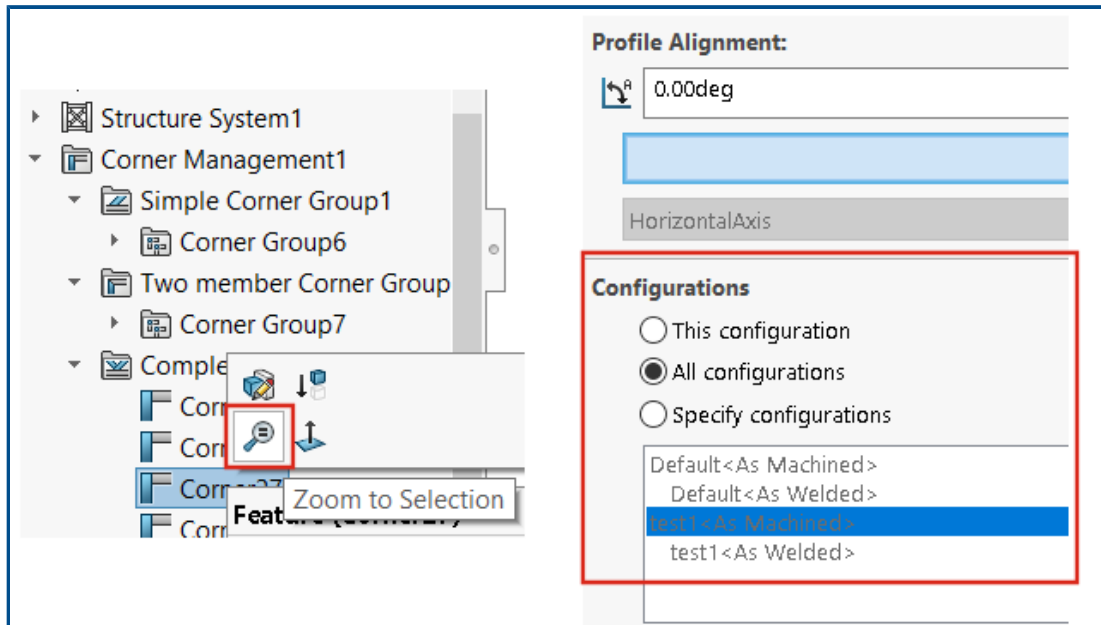
To access this PropertyManager, click **Insert** > **Structure System** > **Pattern Connection**.

## Pattern Connection Element PropertyManager

Select the connection element in the graphics area. Similar corners are highlighted with arrows and previews of connection elements appear at the corners.

<b>Selected Connection Element</b>	Displays the name of the selected connection element.
<b>Identify Similar Corners</b>	Lists similar corners on which you can insert the connection element. You can right-click a corner and select <b>Skip Instance</b> to exclude a connection element.
<b>Ignore corners that have connection elements</b>	Excludes corners with connection elements.
<b>Instances to Skip</b>	Lists the corners that you excluded using <b>Skip Instance</b> .

## Configuration-Specific Size of Weldment Members



If you select a set of weldment members of the same size and type, in the Profile tab, you can change the size of the members for specific configurations.

### Using Zoom to Selection for Corners

If the structural member has multiple corners, you can zoom to a corner from the FeatureManager design tree or the Corner Management PropertyManager using **Zoom to Selection**.

### Configuring the Size for Configurations in Structure System

#### To configure the size for configurations of structure system:

1. Edit the structure system from the FeatureManager design tree.
2. Select the members in the graphics area.
3. Optional: To edit a specific member, in the graphics area, right-click the member, and select **Edit Feature**.
4. In Profile tab, change the size.
5. Under **Configurations**, select specific configuration and click **OK**.

## Structure System and Weldments Performance Improvement (2023 SP4)

Performance is improved when you create or edit a corner management feature.

A corner gets trimmed when you select it from the **Corners** group in the Corner Management PropertyManager or in the graphics area.

Previously, corners were trimmed when you created or edited the corner management feature.

Performance is improved when you insert a structural member or change its profile to the part that includes weld beads.

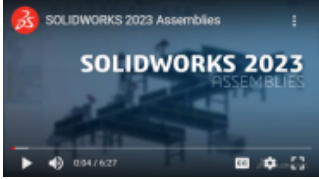
# 8

## Assemblies

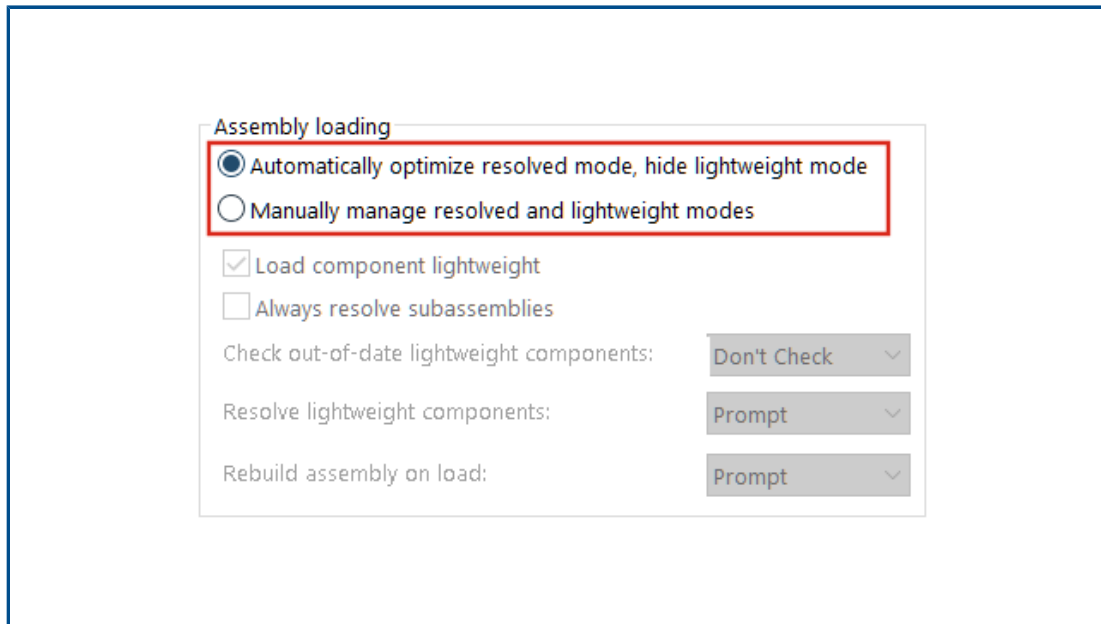
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This chapter includes the following topics:

- **Automatically Optimize Resolved Mode**
- **Repairing a Missing Mate Reference**
- **Delete Missing Components from a Model (2023 SP1)**
- **Additional End Conditions for Cut Features**
- **Previewing a Replacement Component**
- **Specifying a Color for an Explode Line**
- **Selecting a Configuration for Skipped Instances**
- **Specifying a Default BOM Part Number**
- **Magnetic Mates**
- **Overriding Calculated Values in a Design Table**
- **Assembly Visualization**
- **Configuration Publisher: Updating a Multiple-Configuration Model**
- **Fix and Float Options When Inserting a Component (2023 SP2)**
- **Selecting Flexible Subassemblies and Components (2023 SP2)**
- **Width Mate Selections (2023 SP2)**
- **Displaying Component Descriptions and Configuration Descriptions (2023 SP3)**
- **Propagating Assembly Features to Parts Blocked for Toolbox Parts**
- **Assembly Performance Improvements**

	<b>Video: What's New in SOLIDWORKS 2023 - Assemblies</b>
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## Automatically Optimize Resolved Mode



You can optimize resolved mode automatically. Using this system option improves performance by selectively using lightweight technology when you load components in resolved mode.

Not available when your environment includes SOLIDWORKS PDM.

Options for lightweight and resolved modes are hidden as the components load as resolved. Lightweight states do not appear in the FeatureManager® design tree.

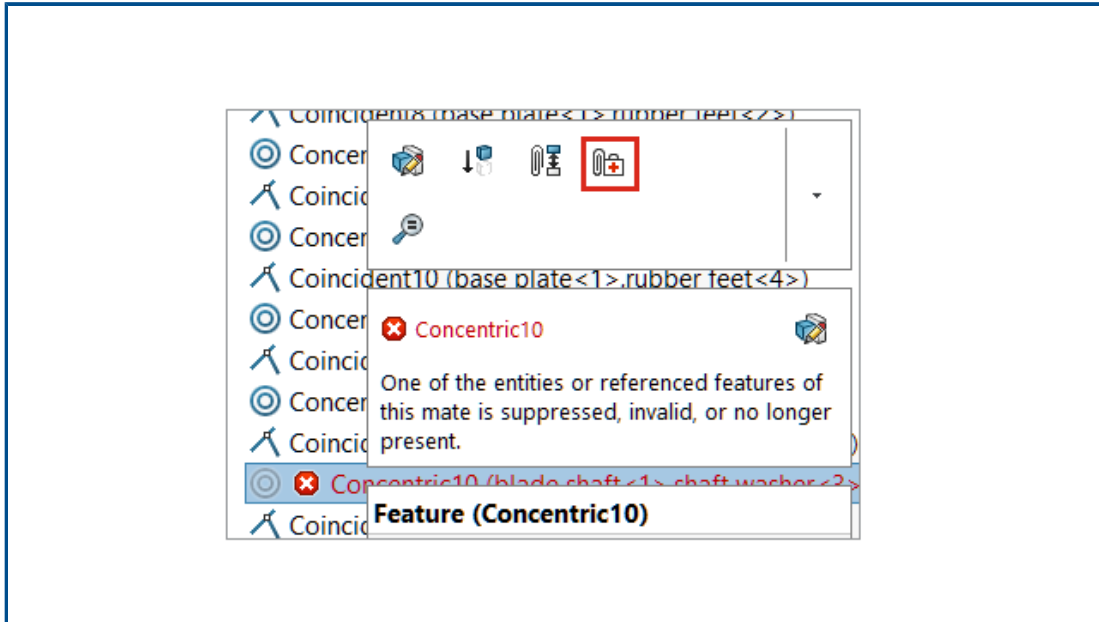
You can manually manage when to resolve lightweight components by selecting **Manually manage resolved and lightweight modes**.

**Automatically load components lightweight** is renamed to **Load component lightweight**.

**To automatically optimize resolved mode:**



1. Click **Tools > Options > System Options > Performance**.
2. Under **Assembly loading**, select **Automatically optimize resolved mode, hide lightweight mode**.

## Repairing a Missing Mate Reference




You can repair missing mate references for references to faces, edges, planes, axes, and points.

SOLIDWORKS® repairs the missing reference for a component by selecting a reference on the component that is in the same location and orientation to the missing reference.

For models that contain multiple missing mate references, right-click the **Mates**  folder and click **Auto Repair** .

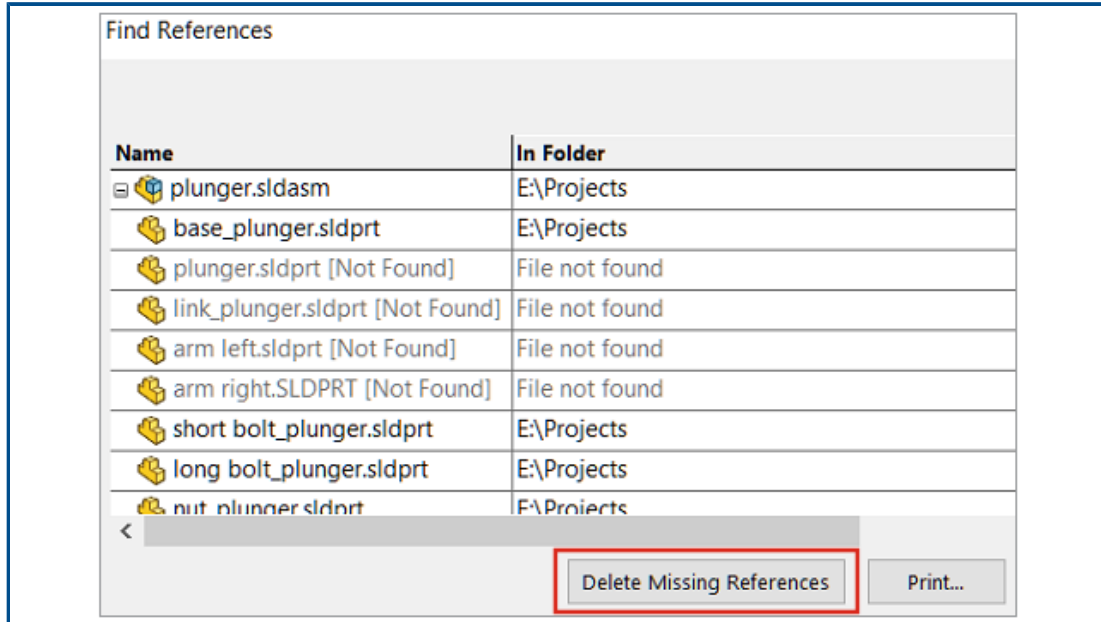
### To repair a missing mate reference:

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

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



## Delete Missing Components from a Model (2023 SP1)



You can delete all missing components from a model in the Find References dialog box.

In the dialog box, **Delete Missing References** displays when the model has missing components.

**Delete Missing References** does not delete missing external references like derived parts, mirror parts, or split parts.

### To delete missing components:

1. Open a model that has missing components.
2. Click **File > Find References**.

In the dialog box, under **In Folder**, **File not found** appears for the missing components.

3. Click **Delete Missing References**.

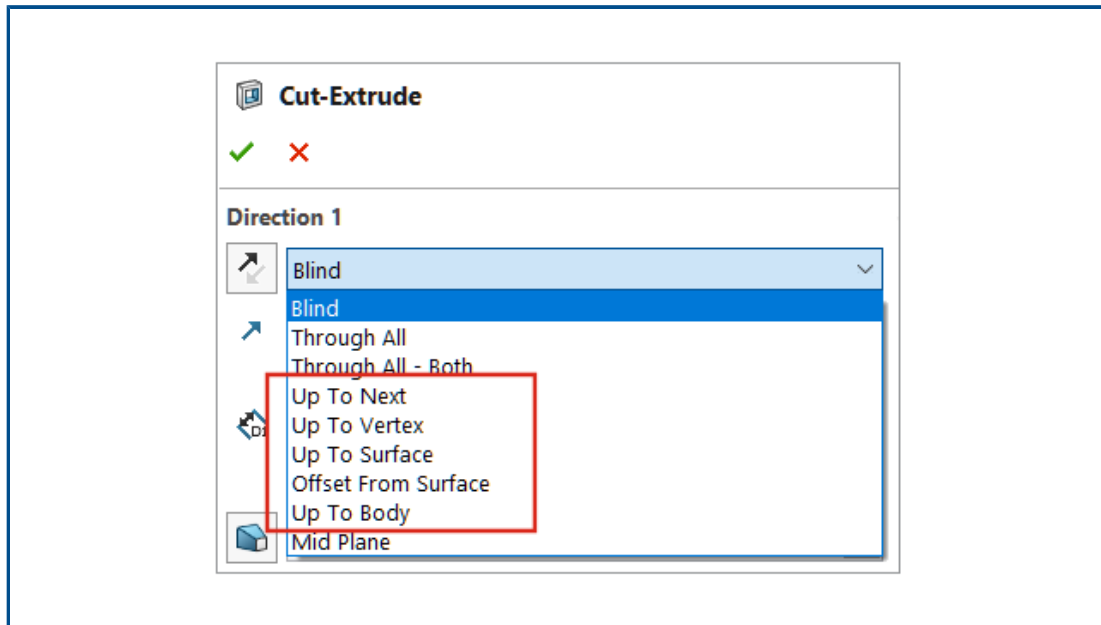
The Confirm Delete dialog box opens.

If you previously selected **Don't show again** in the Confirm Delete dialog box, the dialog box does not open, and the missing components are deleted when you click **Delete Missing References**.

To show the dialog box, click **Tools > Options** and under **Messages/Errors/Warnings**, click **Dismissed messages**. Clear **Do you really want to delete the following item(s)?**.

4. In the Confirm Delete dialog box, click **Yes to All** to delete all the missing components.

## Additional End Conditions for Cut Features



More end conditions are available for cut features in assemblies.

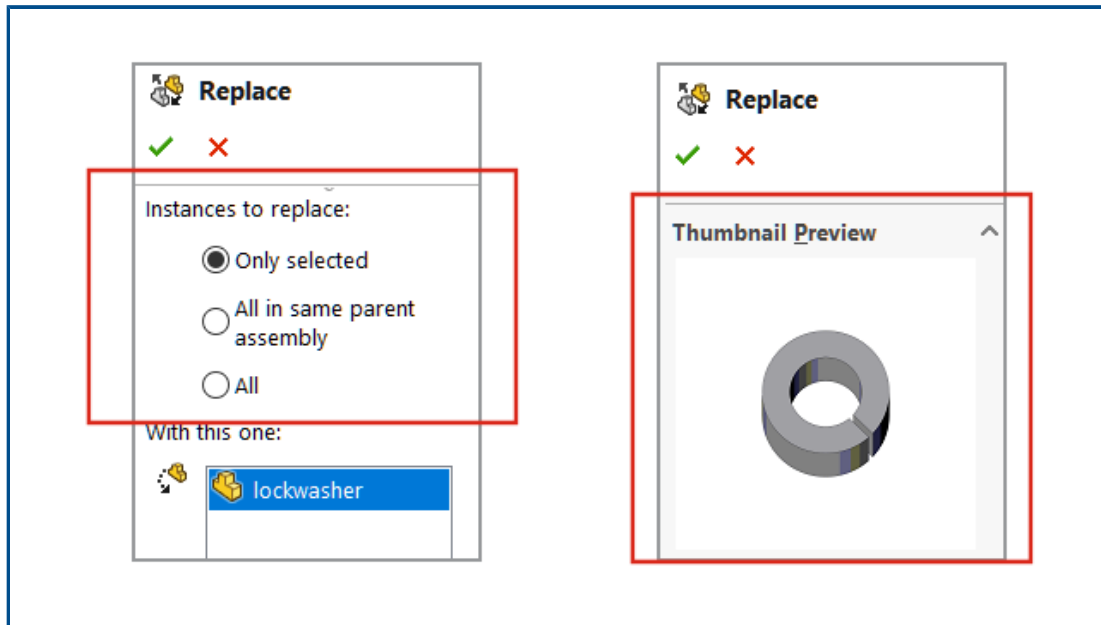
You can select these options for **Direction 1** and **Direction 2**:

- **Up To Next**
- **Up To Vertex**
- **Up To Surface**
- **Offset From Surface**
- **Up To Body**

**To select an end condition for a cut feature:**

1. Open a model and create a sketch.
2. Click **Insert > Assembly Feature > Cut > Extrude**.
3. In **Direction 1**, select an end condition.

## Previewing a Replacement Component





You can preview a replacement component in the Replace PropertyManager. You can specify the scope of the replacement.

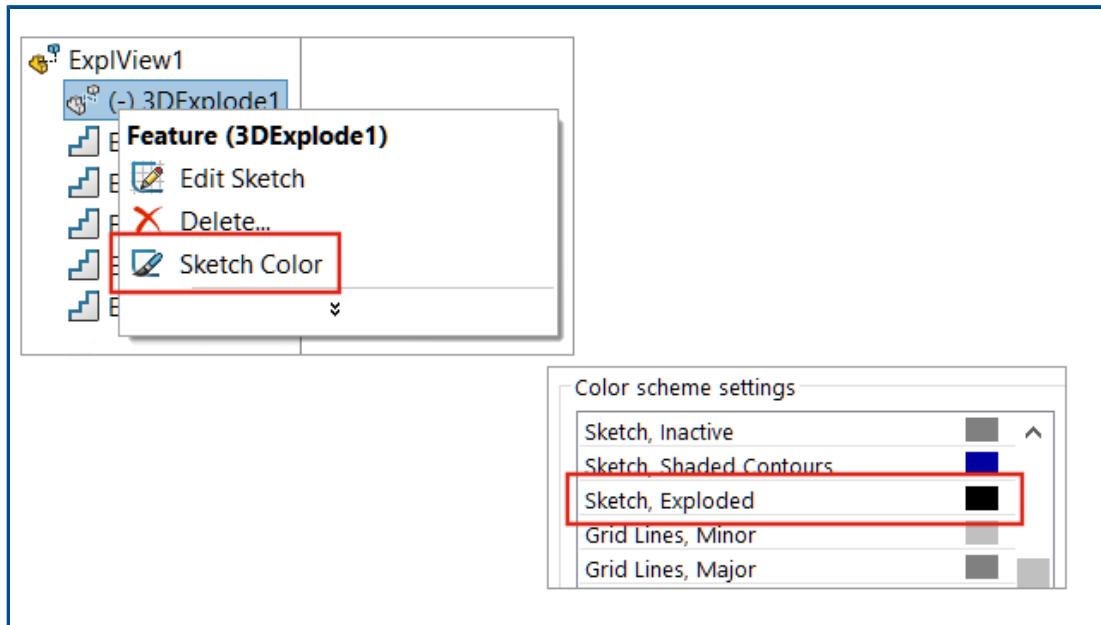
In the PropertyManager, **Instances to replace** replaces **All instances**. To specify the scope of the replacement, select an option for **Instances to replace**:

<b>Only selected</b>	Replaces only the selected instance.
<b>All in same parent assembly</b>	Replaces all instances of the selected component that are in the same assembly level. If a selected component is in the top-level assembly, only instances in the top level are replaced. If a selected component is in a subassembly, only instances in the subassembly are replaced.
<b>All</b>	All instances of the component are replaced in the assembly and all subassemblies.

### To preview a replacement component:

1. Open a model and click **File > Replace**.
2. In **Replace these component(s)** , select a component.
3. For **Instances to replace**, select an option.
4. In the PropertyManager, select a component for **With this one** .  
The selected component appears under **Thumbnail Preview**.

## Specifying a Color for an Explode Line






You can use a system option to specify an explode color. You can change the color of explode lines in an exploded view.

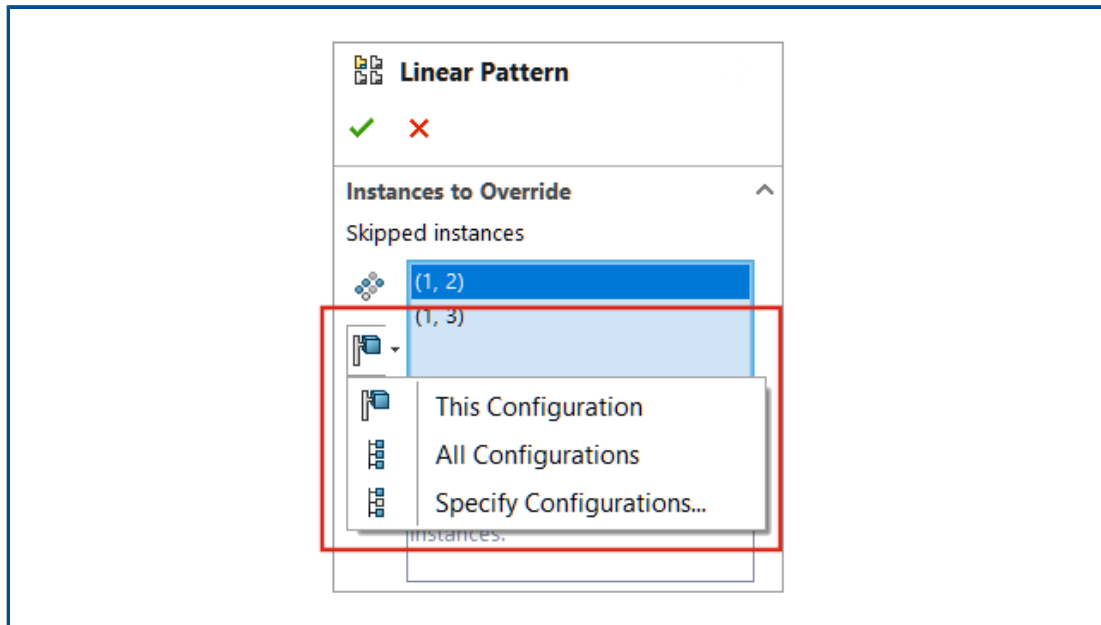
### To specify a color for an explode line:




1. Click **Options > System Options > Colors**.
2. Under **Color scheme settings**, select **Sketch, Exploded**.

### To change the sketch color in an exploded view:

1. Open a model that has an exploded view.
2. In the ConfigurationManager  tab, click  to expand **Exploded View** .
3. Right-click **3DExplode**  and click **Sketch Color** .
4. In the Sketch/Curve Color PropertyManager, select a color.

## Selecting a Configuration for Skipped Instances






You can select **This Configuration** , **All Configurations** , or **Specify Configurations**  for skipped instances in assembly component patterns.

The configuration option is available for these assembly component patterns:

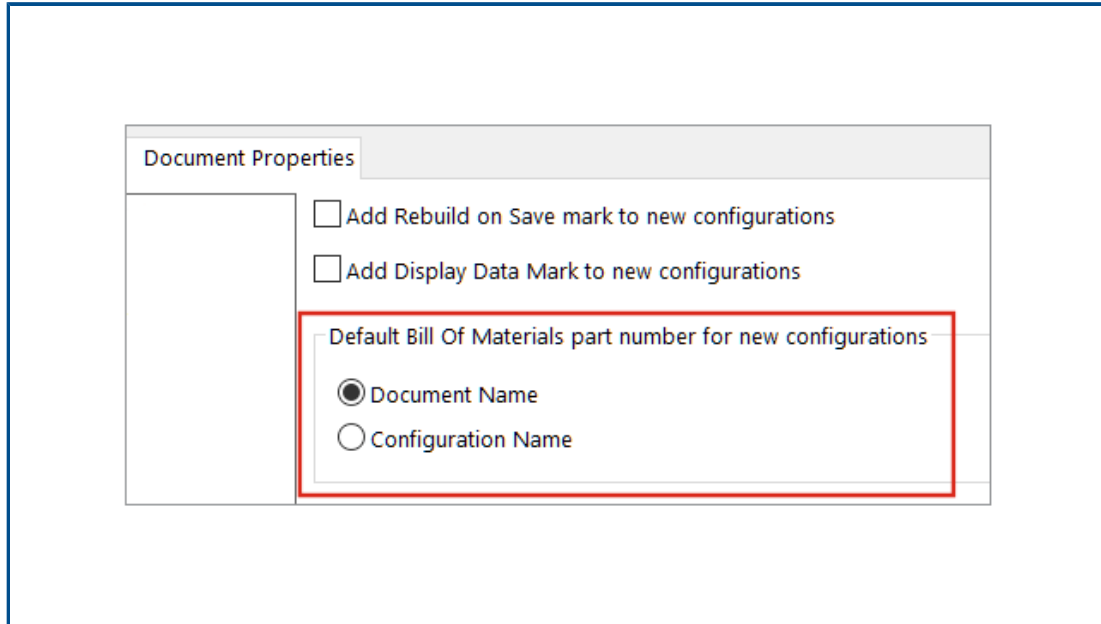
- Linear Component Pattern
- Circular Component Pattern
- Pattern Driven Component Pattern
- Sketch Driven Component Pattern
- Curve Driven Component Pattern

In a design table, you can use `$skip@<type of pattern><number>` to specify skipped instances. For example, `$skip@Lpattern1`. If there is more than one skipped instance, use a semicolon to separate values. For example: `10,1;10,2;`.

### To select a configuration for skipped instances:

1. Open a model with multiple configurations and edit a pattern feature.
2. In the PropertyManager, select an instance to skip and select **This Configuration** , **All Configurations** , or **Specify Configurations** .

## Specifying a Default BOM Part Number



In a model, you can select **Document Name** or **Configuration Name** as the default Bill of Materials (BOM) part number for new configurations.

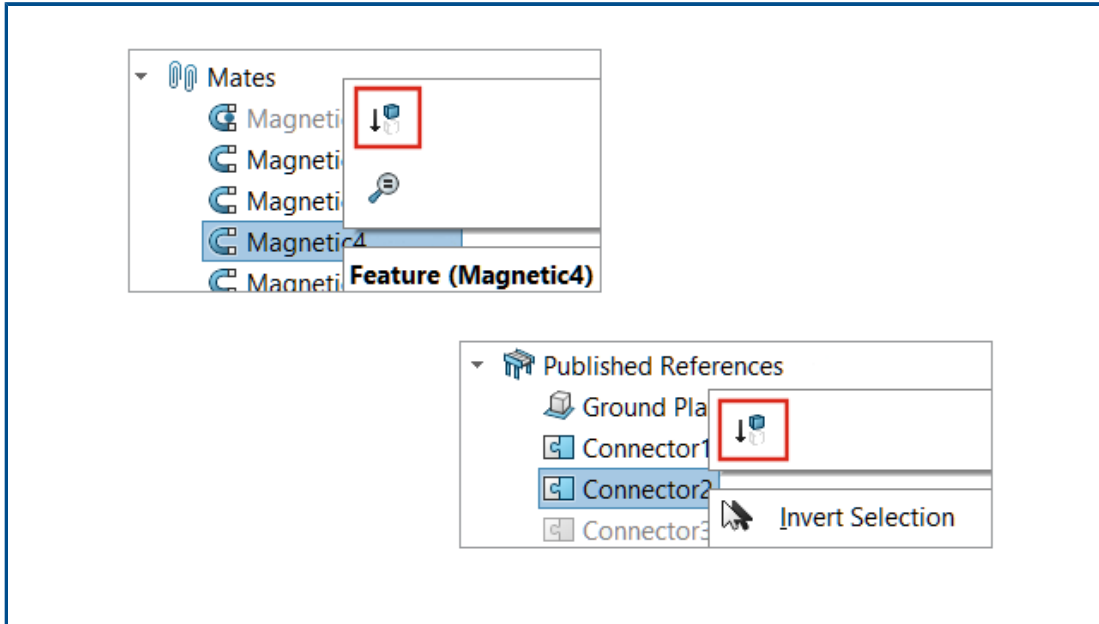
This option does not apply to derived configurations.

The selected default value shows in the Configuration Properties PropertyManager under **Bill of Materials Options** for **Part number displayed when used in a bill of materials**.

### To specify a default BOM part number:

1. Open a model and click **Tools > Options > Document Properties**.
2. Click **Configurations**.
3. Under **Default Bill of Materials part number for new configurations**, select **Document Name** or **Configuration Name**.

## Magnetic Mates






You can suppress magnetic mates and connection points.




When you suppress a component, magnetic mates associated with the component are suppressed. Suppressed connection points are not available when you create a magnetic mate.

When you have unused connection points and move a component chain, magnetic mates are no longer created between components within the chain. When you move a component, the color for all connection points is magenta.

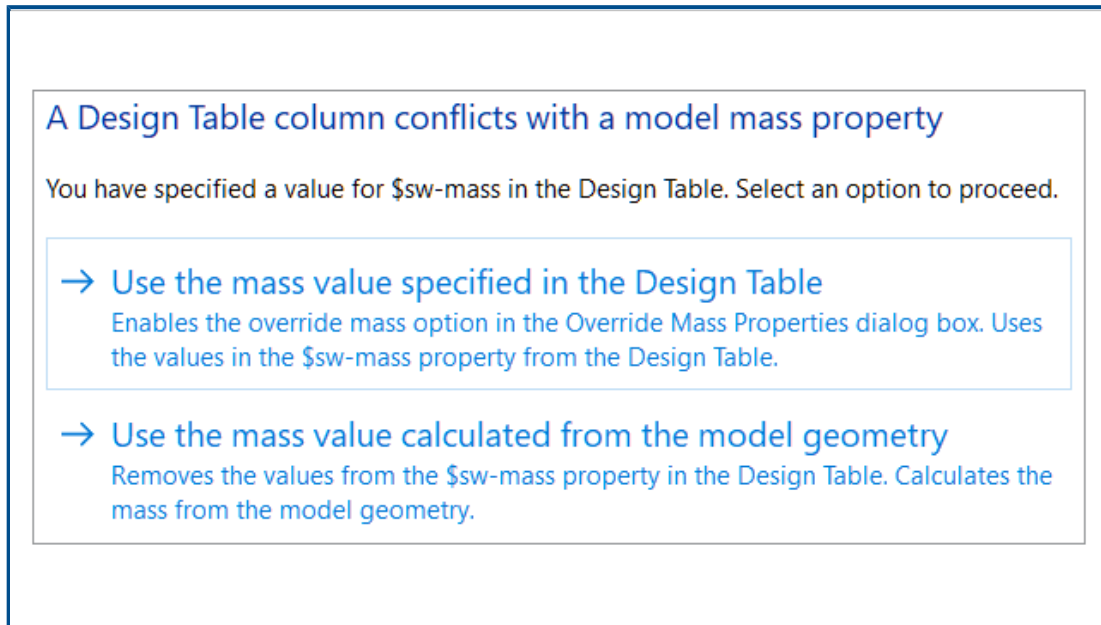
### To suppress a magnetic mate:

1. Open a model that contains magnetic mates.
2. In the FeatureManager design tree, expand the **Mates**  folder.
3. Right-click **Magnetic Mate**  and click **Suppress** .

### To suppress a connection point:

1. Open a model that contains magnetic mates.
2. In the FeatureManager design tree, expand a component and then expand **Published References** .
3. Right-click **Connector**  and click **Suppress** .

## Overriding Calculated Values in a Design Table



When you close a design table that contains manually entered values for \$SW-MASS and \$SW-COG, SOLIDWORKS prompts you to use either the manually entered values or the values calculated from the model geometry.

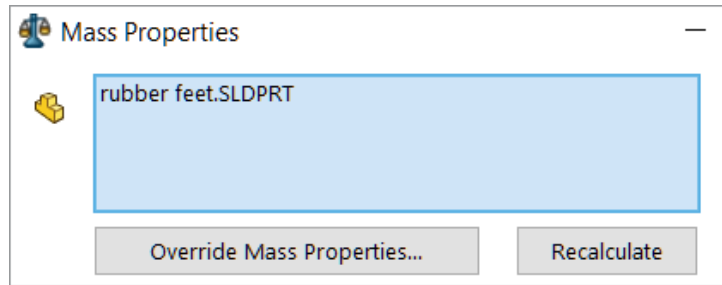
The prompt provides these options:

<b>Use the mass value specified in the Design Table</b>	Enables <b>Override mass</b> or <b>Override center of mass</b> and assigns the manually entered values to the mass properties.
<b>Use the mass value calculated from the model geometry</b>	Deletes the \$SW-MASS column or the \$SW-COG column that has the manually entered values. The model uses values calculated from the model geometry.

### To override calculated values in a design table:

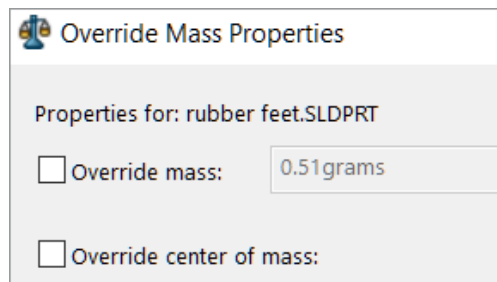
1. Open a model that has a design table with a \$SW-MASS column or a \$SW-COG column.
2. Clear the override properties:
  - a. Click **Tools > Evaluate > Mass Properties**.
  - b. Click **Override Mass Properties**.





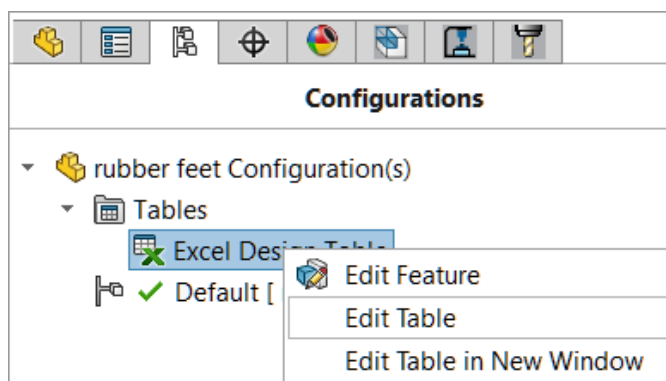
c. In the Override Mass Properties dialog box:

- For \$SW-MASS, clear **Override mass**.
- For \$SW-COG, clear **Override center of mass**.



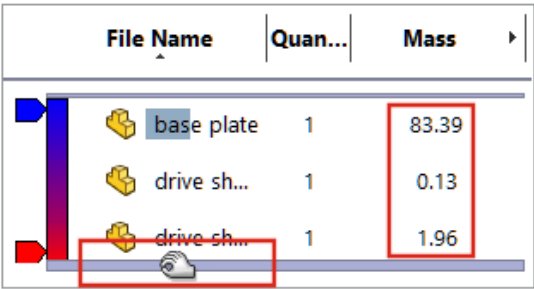
If these override options are enabled, the prompt does not show and the model is saved with the manually entered values.

3. In the ConfigurationManager  tab, expand **Tables** .
4. Right-click **Excel Design Table**  and click **Edit Table**.



5. In the table, manually enter values for \$SW-MASS or \$SW-COG.
6. Close the design table.
7. When prompted, select **Use the mass value specified in the Design Table** to override the calculated values with the manually entered values.

## Assembly Visualization



File Name	Quan...	Mass
base plate	1	83.39
drive sh...	1	0.13
drive sh...	1	1.96

In lightweight mode, SOLIDWORKS calculates more Assembly Visualization columns without setting the lightweight components to resolved.

If SOLIDWORKS cannot calculate a column value for a component and you select **Manually manage resolved and lightweight modes** and **Prompt** for **Resolve lightweight components**, SOLIDWORKS prompts you to resolve the component.

To modify these options, click **Options > System Options > Performance**.

Assembly loading

☐ Automatically optimize resolved mode, hide lightweight mode

☒ Manually manage resolved and lightweight modes

☐ Load component lightweight

☐ Always resolve subassemblies

Check out-of-date lightweight components:

Resolve lightweight components:

In the Custom Column dialog box, the columns display in alphabetical order. The Assembly Visualization rollback bar increases in size when hovering over the bar. The font color is black for all column values. You can add more than seven columns.

### To view the Assembly Visualization improvements:

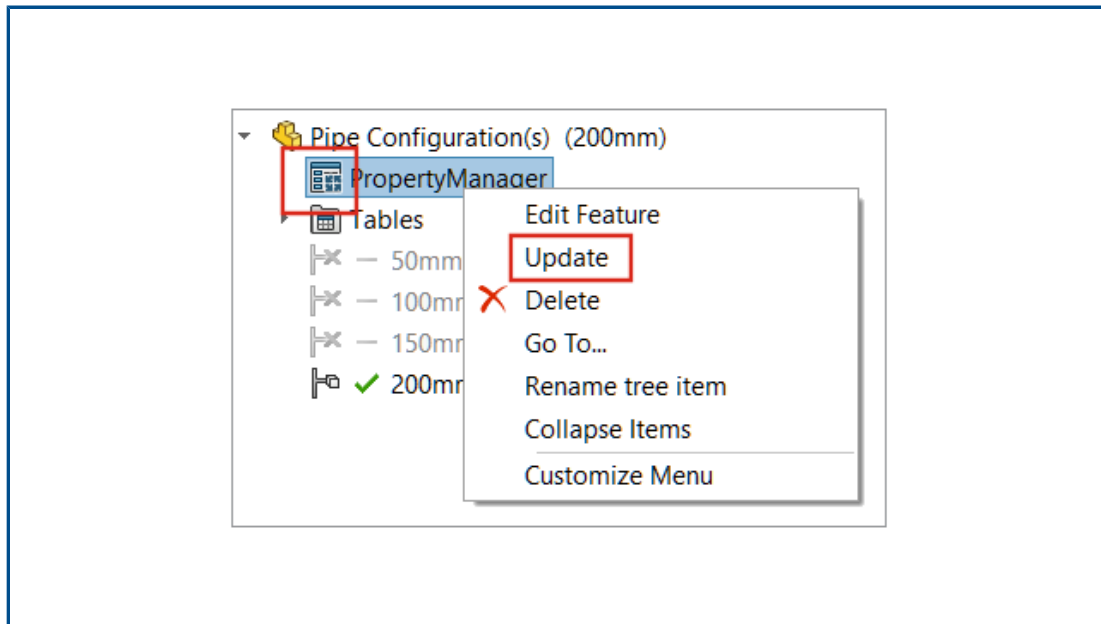
1. Open a model in lightweight mode.
2. Click **Tools > Evaluate > Assembly Visualization**.


All column values display in a black font color and the rollback bar is wider when you hover over the bar. For calculated properties like **Mass**, the calculated values display instead of 0.00.

3. To open the Custom Column dialog box, in the Assembly Visualization panel, click the arrow ▶ to the right of the column headers and click **More**.
4. In the dialog box, click **Properties**.

The columns display in alphabetical order.

## Configuration Publisher: Updating a Multiple-Configuration Model



For models that have multiple configurations created with a custom PropertyManager, the **PropertyManager** icon changes to  after you manually add, delete, or rename a configuration. **Update** appears when you right-click **PropertyManager**.

### To update a multiple-configuration model:

1. Open a model with multiple configurations created with a custom PropertyManager.

Use Configuration Publisher to create a custom PropertyManager.

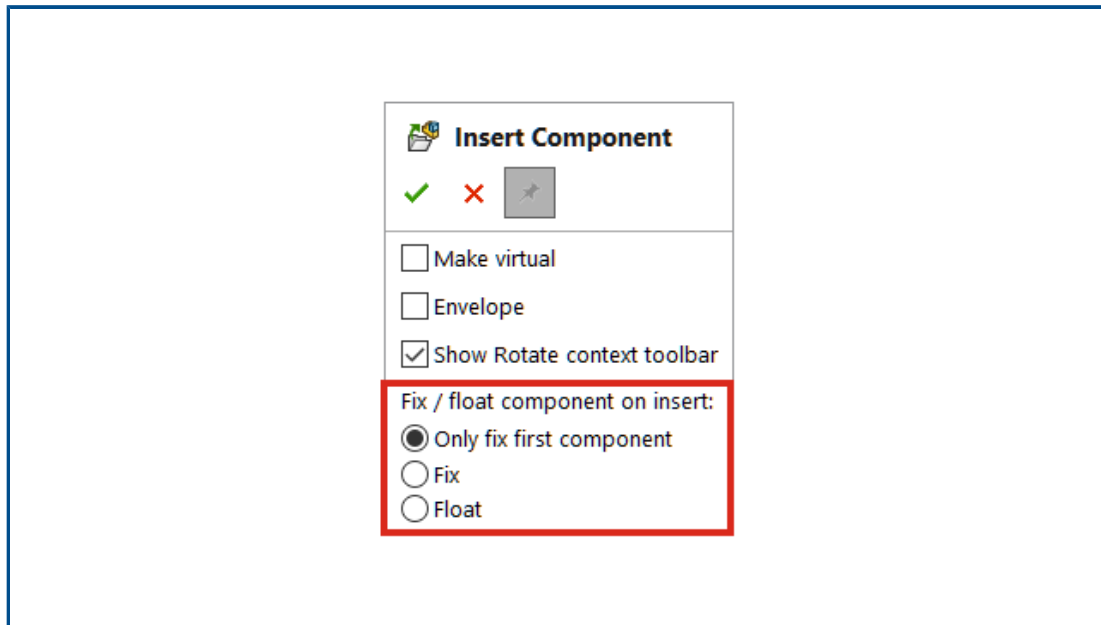
2. In the ConfigurationManager  tab, add a configuration.

The PropertyManager icon changes to .

3. Right-click **PropertyManager**  and click **Update**.

After clicking **Update**, the **Excel design table**  opens, updates, and closes. The PropertyManager icon changes to .


## Fix and Float Options When Inserting a Component (2023 SP2)



In the Insert Component PropertyManager, you can specify the position of a component as **Only fix first component**, **Fix**, or **Float**.

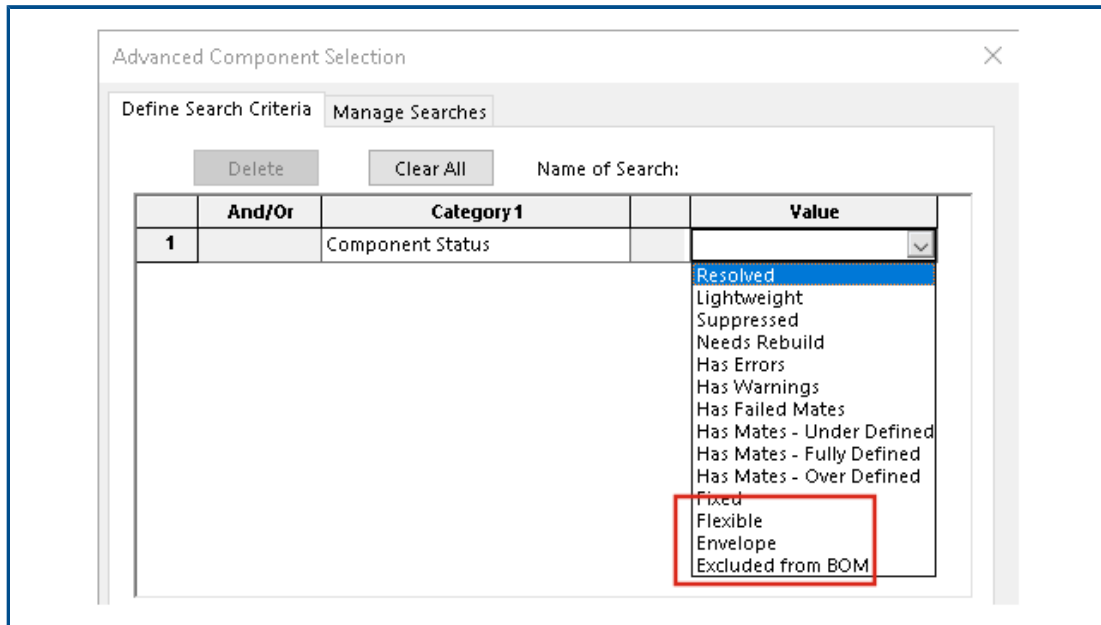
You can select **Float** when inserting the first component. Previously, the default behavior positioned the first component as fixed. You can use **Only fix first component** to position only the first component as fixed in the model.

### To specify a component position:

1. Create a new assembly.
2. Click **Insert** > **Component** > **Existing Part/Assembly**.
3. Select a part.
4. In the PropertyManager, click **Keep Visible**  to keep the PropertyManager open.
5. Under **Fix /float component on insert**, select **Only fix first component**.
6. Insert the component twice.


The first component has a fixed position. The second component has a floating position.

## Selecting Flexible Subassemblies and Components (2023 SP2)




In the Advanced Component Selection dialog box, you can specify **Flexible**, **Envelope**, and **Excluded from BOM** for **Component Status**. From the **Select** flyout, you can use **Select Flexible Subassembly** to select flexible subassemblies in the model.

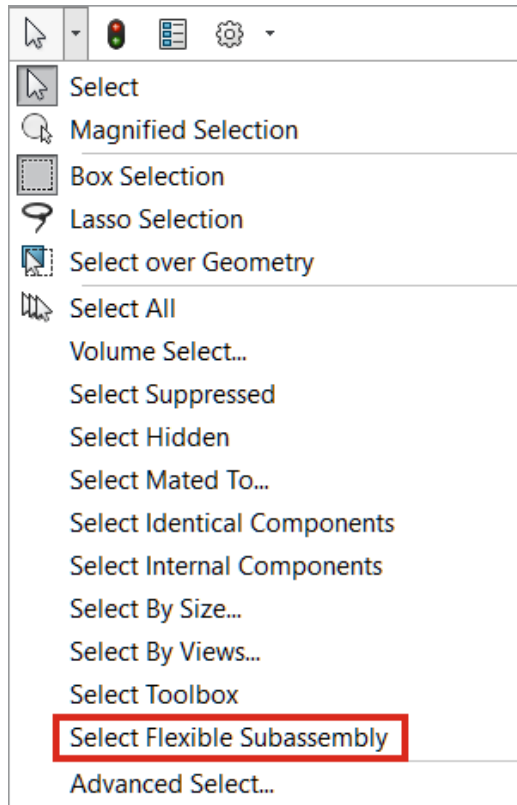
### To select a value for Component Status:

1. Open a model.
2. On the Standard toolbar, click the Select  flyout and click **Advanced Select**.
3. For Category1, select **Component Status**.
4. For Value, click the down arrow.

**Flexible**, **Envelope**, and **Excluded from BOM** are available as options.

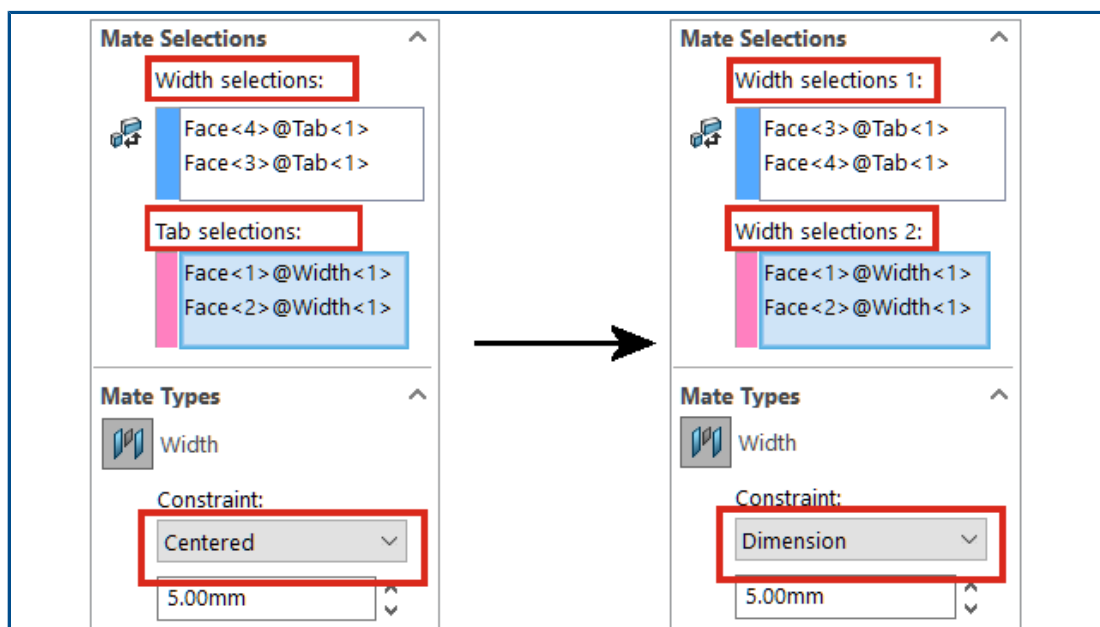
### To select flexible subassemblies:

1. Open a model that has a flexible subassembly.
2. On the Standard toolbar, expand the Select  flyout and click **Select Flexible Subassembly**.



In the FeatureManager design tree, the flexible subassembly is highlighted.

## Width Mate Selections (2023 SP2)



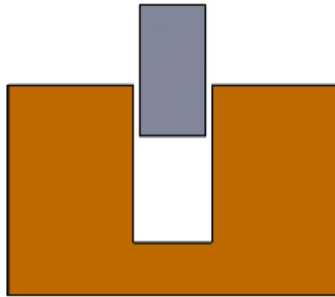
In the Mates PropertyManager, the width mate options **Width selections** and **Tab selections** are renamed to **Width selections 1** and **Width selections 2**.

You can select the smaller selection set in either **Width selections 1** or **Width selections 2**, and all constraint types are available. Previously, the tab selections had to have the same or a smaller distance as the distance between the width selections. If the distances are equal, only the **Centered** constraint type is available.

For **Width selections 1**, you can select a cylindrical face or an axis. Previously, only tab selections allowed a cylindrical face or an axis selection.

#### To select a constraint type for width mates:

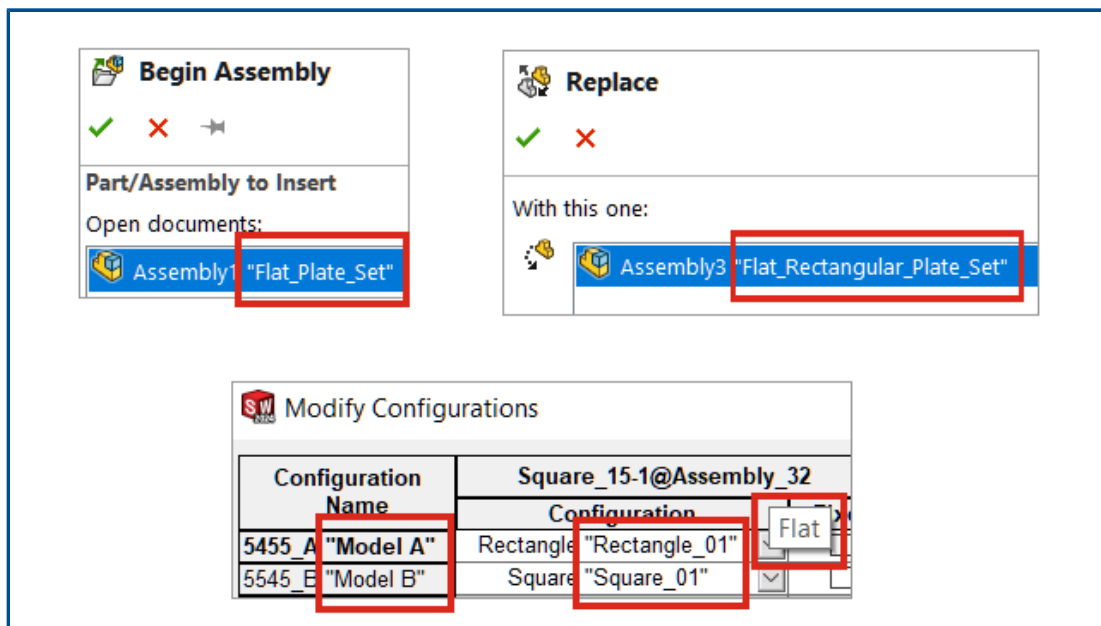
1. Open a model with a width mate where the selection sets are different sizes.



For example, this model has an existing width mate between two parts. The selection set for **Width selections 1** has a distance that is smaller than the selection set in **Width selections 2**.

2. Edit the width mate and select **Dimension** for the constraint type.

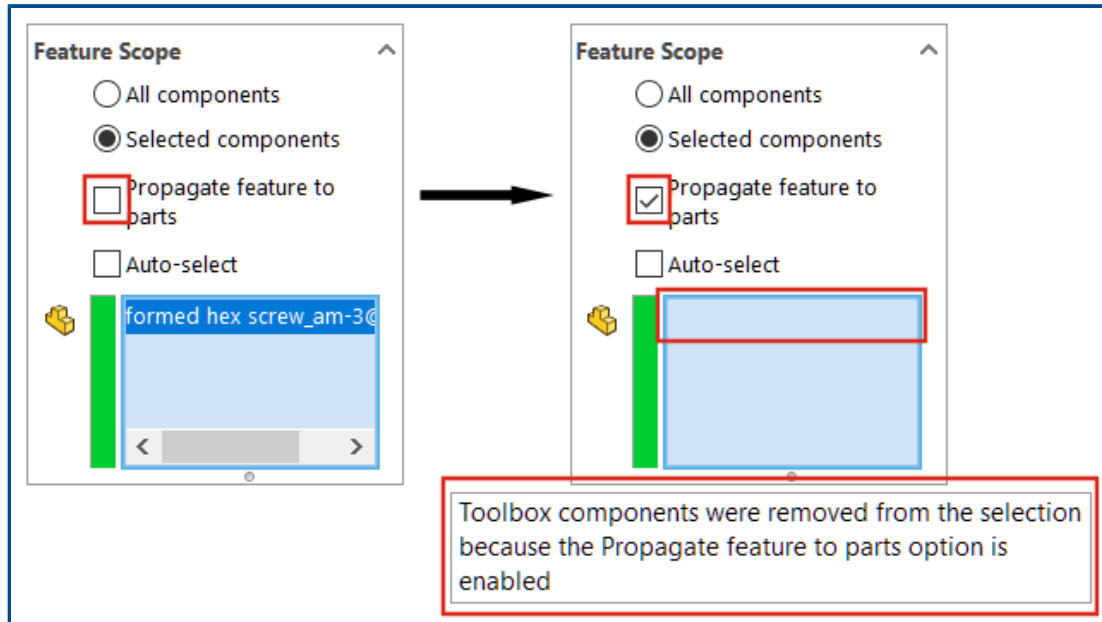
## Displaying Component Descriptions and Configuration Descriptions (2023 SP3)




In the Insert Component / Begin Assembly PropertyManager, under **Open documents**, the component name includes the component description. In the Replace PropertyManager, under **With this one**, the component name includes the component description.

In the Modify Configurations dialog box and the Configuration Table dialog box, under **Configuration Name** and **Configuration**, the configuration name includes the configuration description. You can hover over the header of a parameter column to view the component description.

## Propagating Assembly Features to Parts Blocked for Toolbox Parts



Assembly features cannot be propagated to Toolbox parts.


In an assembly feature PropertyManager, SOLIDWORKS displays a warning and removes any Toolbox parts from **Components to Affect**  when you have **Selected components** and **Propagate features to parts** selected.

When you select **All components** and **Propagate features to parts** under **Feature Scope** and the model contains Toolbox parts, the warning does not appear. The assembly features are not propagated to the Toolbox parts.

When **Propagate features to parts** is selected, you cannot select a Toolbox part.


For assembly features like fillets and chamfers that do not have the **All components** and **Selected components** options, the warning does not appear and the assembly features are not propagated to Toolbox parts. You cannot select a Toolbox part for a **Hole Series** as this assembly feature is always propagated to parts.

### To view the warning:

1. Open a model that contains a Toolbox part.
2. Select a sketch and click **Insert > Assembly Feature > Cut > Extrude**.
3. In the PropertyManager, under **Feature Scope**, clear **Auto-select**.
4. For **Components to Affect** , select a Toolbox part.



5. Select **Propagate features to parts**.

SOLIDWORKS removes the Toolbox part from **Components to Affect**  and the warning appears in the graphics area.

## Assembly Performance Improvements

Assembly performance is improved when:

- Searching with **Include sub-folders** selected.

To select this option, click **Tools > Options > External References > Reference Documents specified in File Locations > Include sub-folders**.

- For assemblies that contain parts with multiple imported bodies and body level appearances:
  - Opening a file
  - Changing a configuration
  - Applying, modifying, or removing an appearance
  - Rolling back or rolling forward assembly items
- Saving assemblies is faster as only changed model data is saved. This improvement is noticeable in very large assemblies. Previously, SOLIDWORKS saved all model data when an assembly or a component changed.

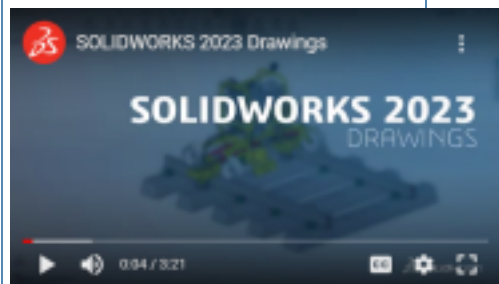
# 9

## Detailing and Drawings

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This chapter includes the following topics:

- **Overrides for Bills of Materials**
- **Displaying Transparent Models**
- **Filtering Bill of Materials Columns**
- **Component Name Display**
- **Limiting Geometric Tolerances to a Standard**
- **Detached Drawings**





Video: What's New in SOLIDWORKS 2023 - Drawings

## Overrides for Bills of Materials

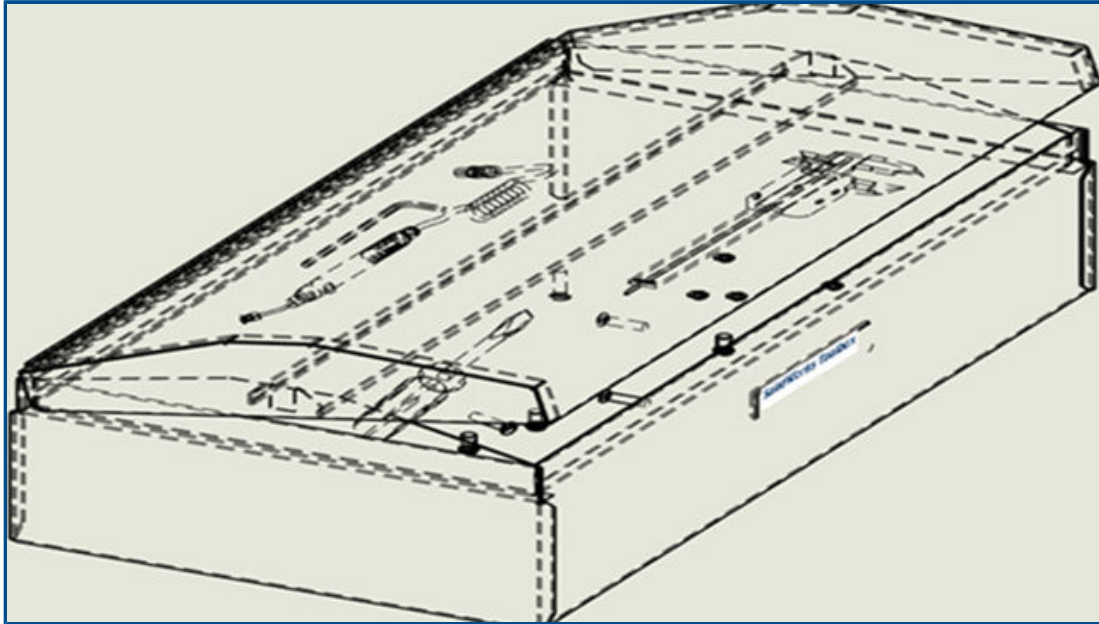
ITEM NO.	PART NUMBER	QTY.
1	emitter	1
2	<b>grip</b>	1
3	grip-focuser-lens-spinring	1
4	<b>grip-focuser-lens-lens</b>	1
5	grip-focuser-lens-innerring	1



When you override a value in a bill of materials (BOM) using **Break Link**, the value turns blue.

You can:

- Change the highlight color (default is blue) when you override values using **Break Link**. Click **Tools > Options > System Options > Colors**, and specify the color for **Drawings, Modified Cells (BOM)**.
- Restore overridden values to their original values. Right-click a row, column, or the entire BOM and click **Restore Original Value**.
- Prevent unwanted changes to BOMs. Click a table cell, row, or column, and click the lock  in the context toolbar. To unlock, click the lock  in the context toolbar.

## Displaying Transparent Models



You can display transparent models in drawings with **Hidden Lines Removed** (HLR)  and **Hidden Lines Visible** (HLV)  modes.

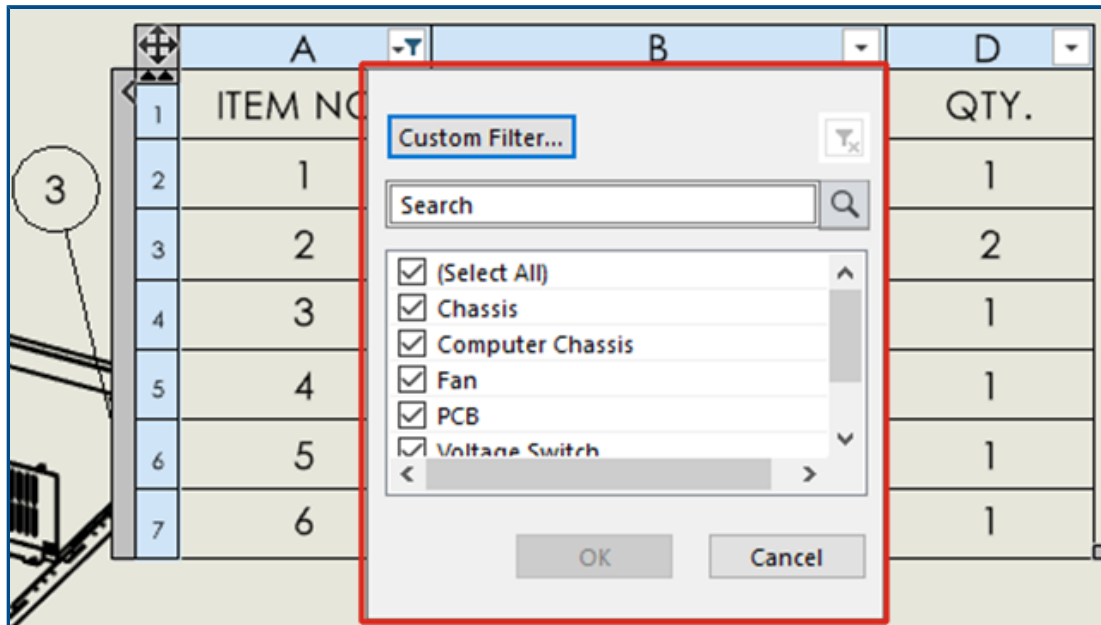
You can see through transparent bodies in HLR and HLV display states including components in assemblies and bodies in multibody parts.

Available for **High quality** drawing views only.

### To display transparent models:

1. Click **Tools > Options > Document Properties > Detailing** .
2. Select **See through transparent components in HLR/HLV**.


## Filtering Bill of Materials Columns





Filtering a bill of materials (BOM) lets you focus on what you need to see.

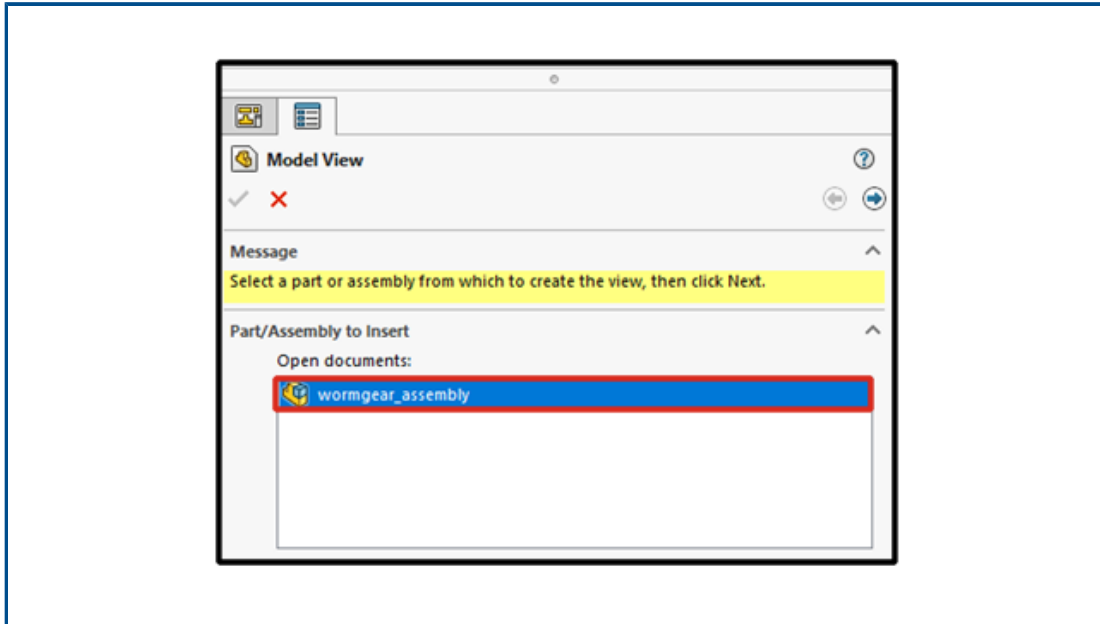
When you filter a BOM:

- You can filter a range of data based on your criteria. When you hover over the BOM column headers, there is an arrow in each column.
- Only the balloons for the filtered items appear in the view.
- **To filter BOM columns:**

1. Click  in the column to filter.
2. In the dialog box, select filter criteria.

The  changes to  indicating that the filter applies to the column.

## Component Name Display



When you specify display options for component names in a drawing, the names in the PropertyManager and View Palette match those in the FeatureManager design tree.

### To specify display options for component names:

1. In the FeatureManager design tree, right-click the drawing name and click **Tree Display > Component Name and Description**.
2. (Optional.) Under **Primary**, specify an option:
  - **Component Name**
  - **Component Description**

Based on the selection under **Primary** display options:

- The Model View and Standard 3 View PropertyManagers update the **Open documents** field.
- The View Palette updates. The View Palette displays the part or assembly icon and the file name.

## Limiting Geometric Tolerances to a Standard

You can choose all symbols for geometric tolerances or limit symbols to a standard with the **Allow tolerance symbols for all drafting standards** option.

For example, if you select the ISO standard and clear **Allow tolerance symbols for all drafting standards**, you limit the symbols and values to ISO standards.

If you select **Allow tolerance symbols for all drafting standards**, all symbols and values are available regardless of the tolerancing standard.

**To limit geometric tolerances to a standard:**

1. In a drawing, click **Tools > Options > Document Properties > Annotations > Geometric Tolerances**.
2. Under **Base geometric tolerance standard**, select:
  - a. A standard.
  - b. Clear **Allow tolerance symbols for all drafting standards**.

## Detached Drawings

Detached drawings are no longer supported.

Detached drawings are no longer supported as of SOLIDWORKS 2023. Detailing mode is available. Existing detached drawings continue to work as they do today. Once you save a detached drawing back to a regular drawing, you cannot save it as a detached drawing in SOLIDWORKS 2023.

# 10

## Import/Export

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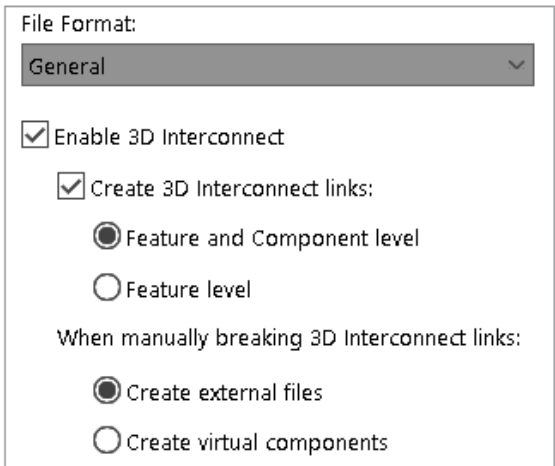
This chapter includes the following topics:

- **Import of Third-Party CAD Files (2023 SP2)**
- **Import System Options (2023 SP2)**
- **Exporting Large Assemblies as STEP Files**
- **Improved Display of CGR Files (2023 SP2)**
- **Installing the 3DEXPERIENCE Exchange Add-In**
- **OBJ Mesh Import Improvements**

### Import of Third-Party CAD Files (2023 SP2)

Performance of importing third-party CAD files is improved with 3D Interconnect. You can import third-party CAD files without creating 3D Interconnect links faster.

### Import System Options (2023 SP2)



The screenshot shows a dialog box titled "Import System Options" with a "File Format:" dropdown menu set to "General". Below the dropdown, there are several options:

- ☒ Enable 3D Interconnect
  - ☒ Create 3D Interconnect links:
    - ☒ Feature and Component level
    - ☐ Feature level
  - When manually breaking 3D Interconnect links:
    - ☒ Create external files
    - ☐ Create virtual components



The system options for importing third-party CAD files with links using 3D Interconnect are improved for usability.

**To access the import system options:**

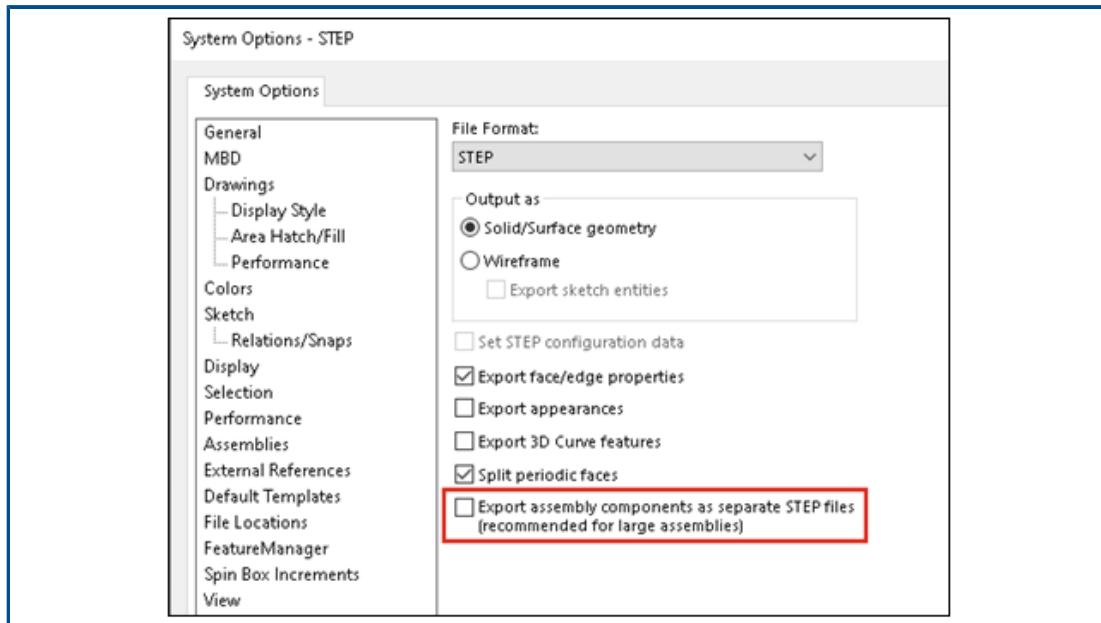
1. Click **Tools > Options > System Options > Import**.
2. Select **Enable 3D Interconnect**. When selected, you can specify additional options:
  - **Create 3D Interconnect links**. Imports the 3D Interconnect feature and component links. (Third-party files consist of multiple components (for assemblies) and each component has features.)
  - **Feature and Component Level** or **Feature Level**. Displays the component links and feature links with an arrow symbol on each component and feature in the FeatureManager design tree after import.

When you import, the component and feature links have an arrow on each component and feature in the FeatureManager design tree.

**After importing the file with links, you can manually break the links by right-clicking and selecting Break Link in the FeatureManager design tree. The output of the link depends on the following options:**

- **Create external files**. Breaks the 3D Interconnect component link and creates the resulting part or assembly as an external file.
- **Create virtual components**. Creates the resulting part or assembly as an internal file (virtual component).

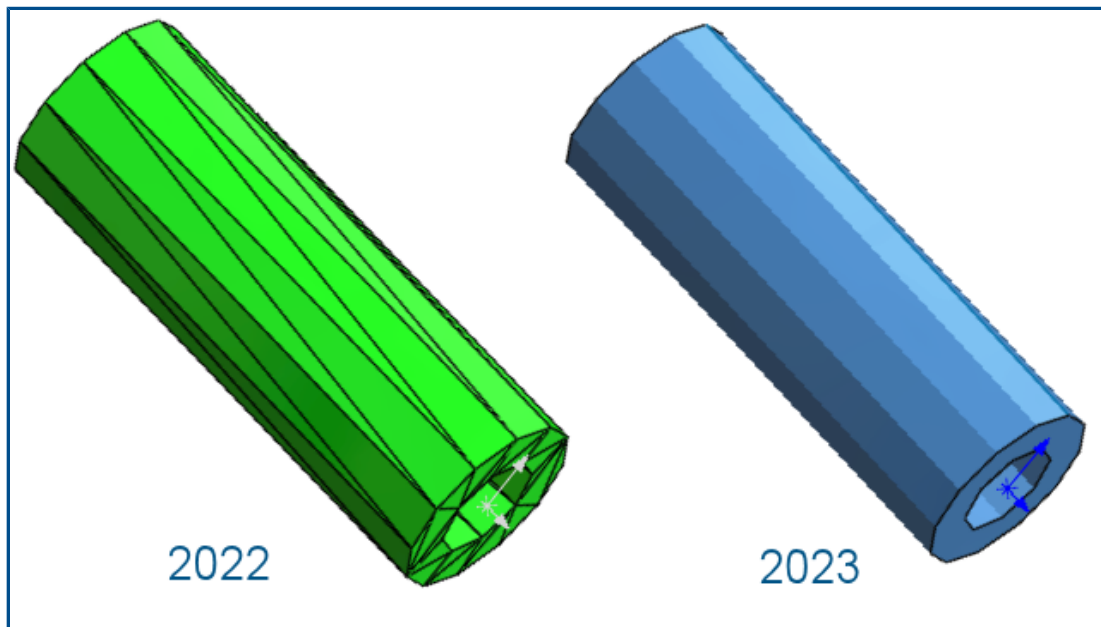
## Exporting Large Assemblies as STEP Files



You can export large SOLIDWORKS® assemblies as STEP files.


In **Tools > Options > System Options > Export > STEP**, select **Export assembly components as separate STEP files (recommended for large assemblies)**. This option exports assemblies as atomic STEP files. Separate STEP files get created for each component in the assembly.

## Improved Display of CGR Files (2023 SP2)



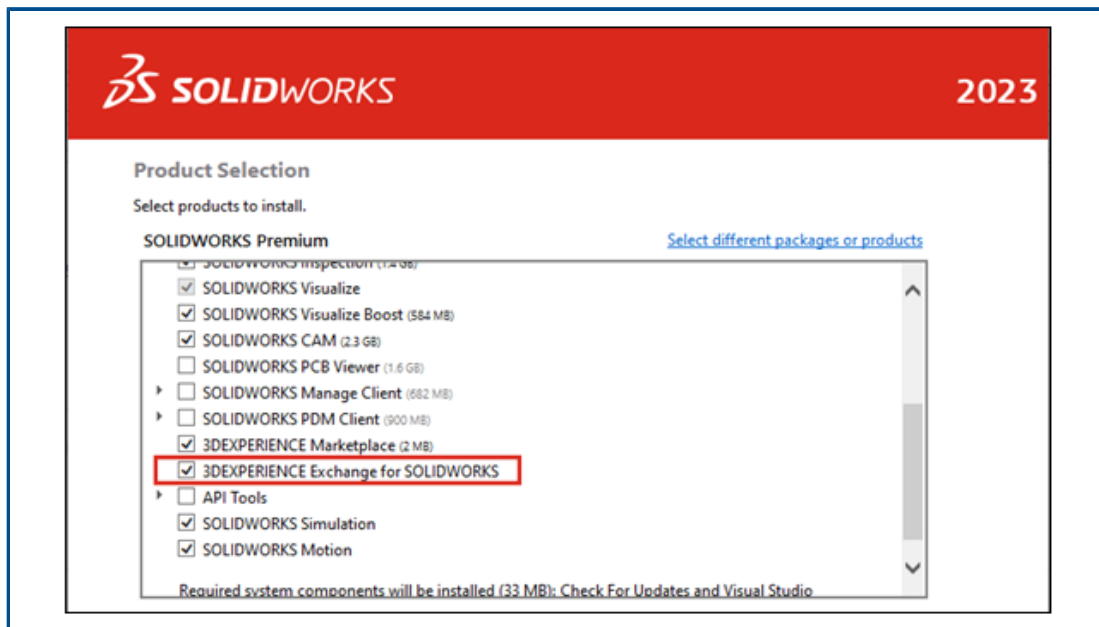
When you import \*.cgr files, facet fins are always hidden, which delivers models that are visually equal to SOLIDWORKS models. The amount of time to open \*.cgr files is reduced.

If you import \*.cgr files as mesh BREP bodies, you have the option to display or hide facet fins. In **Tools > Options > System Options > Display**, select or clear the option **Display facet fins in mesh BREP bodies**.

This option does not affect the display of regular edges on mesh BREP bodies, which includes edges created directly on import or created later with the **Segment Imported Mesh Body**  tool.


Previously, imported \*.cgr files always displayed facet fins and did not visually match SOLIDWORKS models.

## Installing the 3DEXPERIENCE Exchange Add-In



The **3DEXPERIENCE** Exchange add-in installs by default with your installation.

The **3DEXPERIENCE** Exchange add-in lets SOLIDWORKS users safely work with a package of content from a **3DEXPERIENCE** source, such as SOLIDWORKS Connected or the Design with SOLIDWORKS app. SOLIDWORKS users can seamlessly open the package, modify files, and return the **3DEXPERIENCE** content package to the **3DEXPERIENCE** source.

To enable the add-in, in SOLIDWORKS, click **Tools** > **Add-ins** and under **Other Add-ins**, select **3DEXPERIENCE** Exchange. The **3DEXPERIENCE** Exchange  appears in the Task Pane.

See *SOLIDWORKS Help: **Import and Export** > **3DEXPERIENCE** Exchange*.

## OBJ Mesh Import Improvements

Textured .OBJ import by ScanTo3D is improved.

.OBJ import to graphics body supports textures, excluding bump maps.

# 11

## SOLIDWORKS PDM

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This chapter includes the following topics:

- **Managing Deleted Users**
- **Configuring Transition Actions**
- **Adding Tooltips to the Data Card Controls**
- **Automatic login to the Administration Tool**
- **Data Card Controls (2023 SP2)**
- **Performance Improvements for File Version Upgrade Tool**
- **Logging File Retrieval Operations**
- **Support for Microsoft Edge WebView2-Based Controls**
- **Removing Computers from the Task Execution List**
- **Display Groups for Inherited Permissions**
- **Notification Templates**
- **Synchronizing Users with Windows Active Directory (2023 SP1)**
- **Days in State Variable (2023 SP1)**
- **Archive Workflow States (2023 SP2)**
- **Grid View in Web2 (2023 SP2)**
- **Administrative Permissions for Tasks (2023 SP2)**
- **SOLIDWORKS PDM Performance Improvements (2023 SP04)**
- **Other SOLIDWORKS PDM Enhancements**

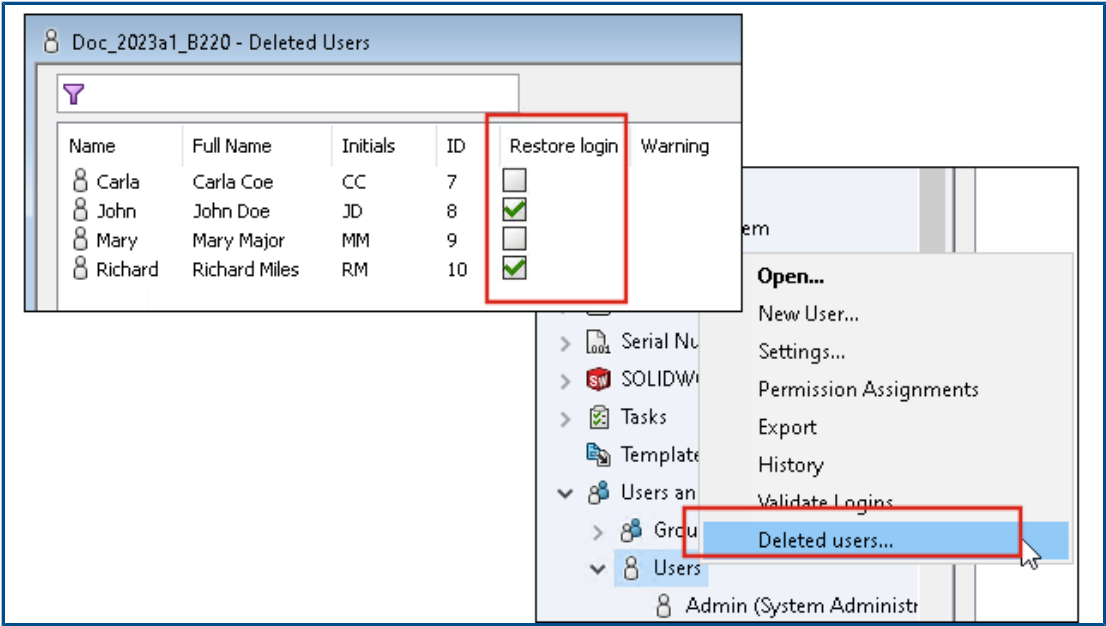
SOLIDWORKS® PDM is offered in two versions. SOLIDWORKS PDM Standard is included with SOLIDWORKS Professional and SOLIDWORKS Premium, and is available as a separately purchased license for non-SOLIDWORKS users. It offers standard data management capabilities for a small number of users.

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



**Video: What's New in SOLIDWORKS 2023 - SOLIDWORKS PDM**

Managing Deleted Users



You can use the Administration tool to restore deleted users and to refuse login to users.

You must have **Can administrate users** administrative permission.

To restore a deleted user, expand **Users and Groups Management**, right-click **Users**, and select **Deleted users**. In the Deleted Users dialog box, select the user in the **Restore login** column.

To refuse login to a user, expand **Users and Groups Management**, right-click **Users**, and select **Open**. In the Users dialog box, select a user in the **Refuse login** column.

Deleted Users Dialog Box

Use the Deleted Users dialog box to restore the login for deleted users.

**To display this dialog box:**

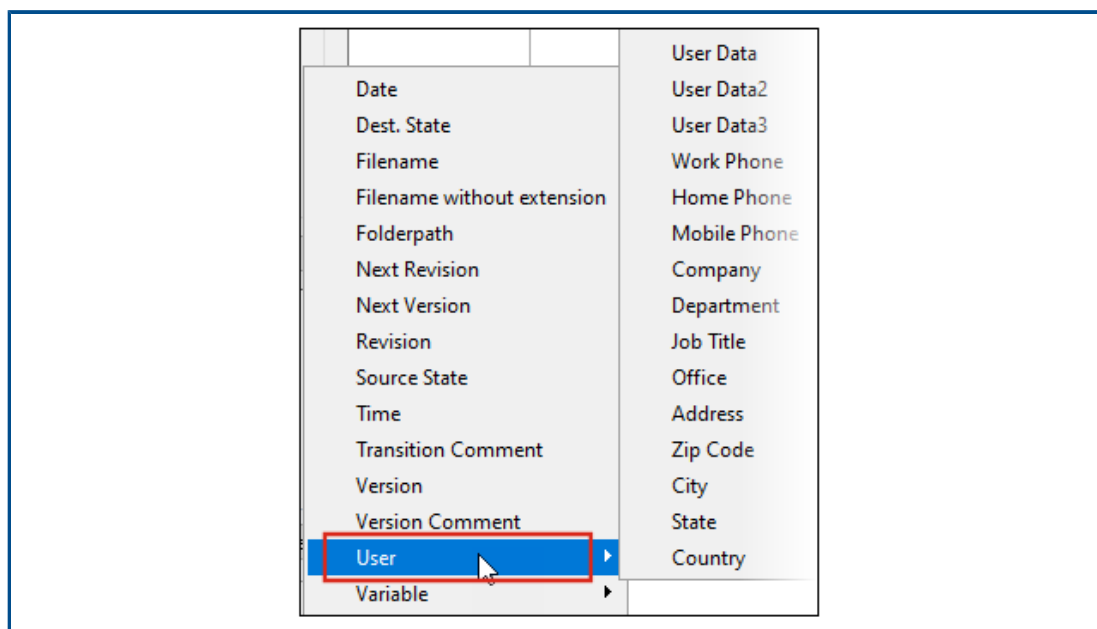
Expand **Users and Groups Management**, right-click **Users**, and select **Deleted users**.

<b>Name</b>	Name the user enters in the login dialog box.
-------------	---

<b>Full Name</b>	The user's full name. The default is the user's login name.
<b>Initials</b>	The user's initials. The default is the first letter of the user's login name.
<b>ID</b>	ID that is unique for each user.
<b>Restore login</b>	When selected, restores the login of the selected user.
<b>Warning</b>	Displays a warning message if the login name you are attempting to restore is already active.

To save changes, click **Save** or close the dialog box and click **Yes** when prompted to save changes.

## Configuring Transition Actions



The variable list in the Transition Action dialog box includes the user properties fields that were introduced in SOLIDWORKS PDM 2022.

These fields are:

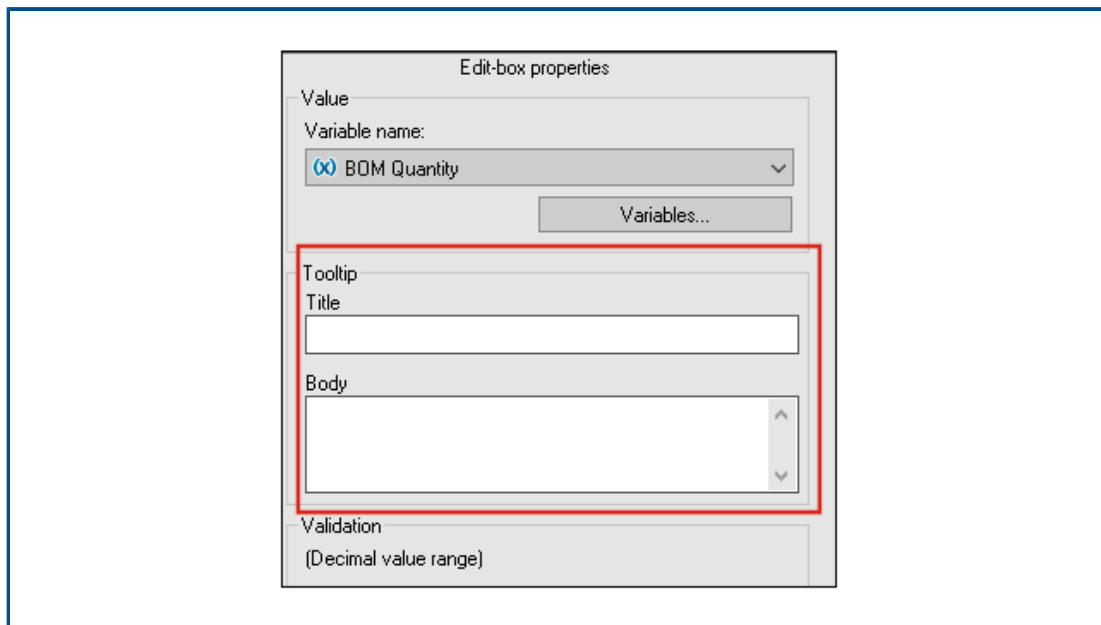
- **First name**
- **Last name**
- **Middle name**
- **Prefix**
- **Suffix**
- **E-mail**
- **E-mail2**

- **E-mail3**
- **User Data**
- **User Data2**
- **User Data3**
- **Work phone**
- **Home phone**
- **Cellphone**
- **Company**
- **Department**
- **Job title**
- **Office**
- **Address**
- **Zipcode**
- **City**
- **State**
- **Country**

These are available under the **User** list and administrators can use these fields in the following transition actions:

- **Import data from XML**
- **Inc. Revision**
- **Send Mail**
- **Set Variable**

## Adding Tooltips to the Data Card Controls



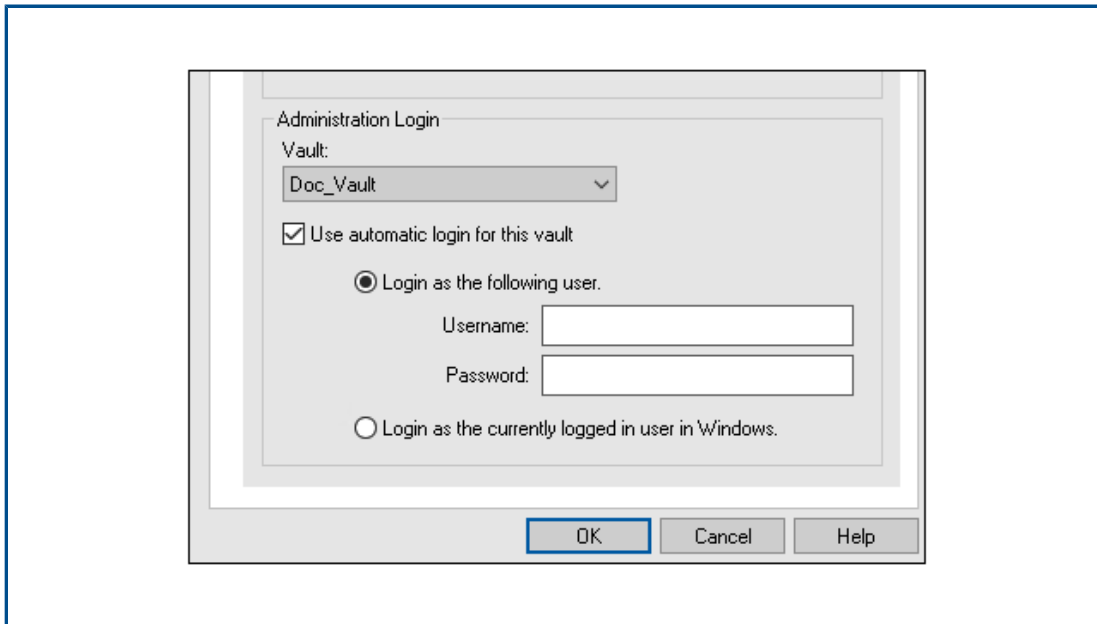
In the Card Editor, you can add tooltips to the data card controls.

In the **Tooltip** pane, enter the **Title** and **Body** of the tooltip.

You cannot specify tooltips for tab and frame controls.

In SOLIDWORKS PDM File Explorer, when you hover over the corresponding data card control, the tooltip displays.

## Automatic login to the Administration Tool



You can configure SOLIDWORKS PDM to log you in to the Administration tool using a specific name or the name of the current Microsoft® Windows® profile.

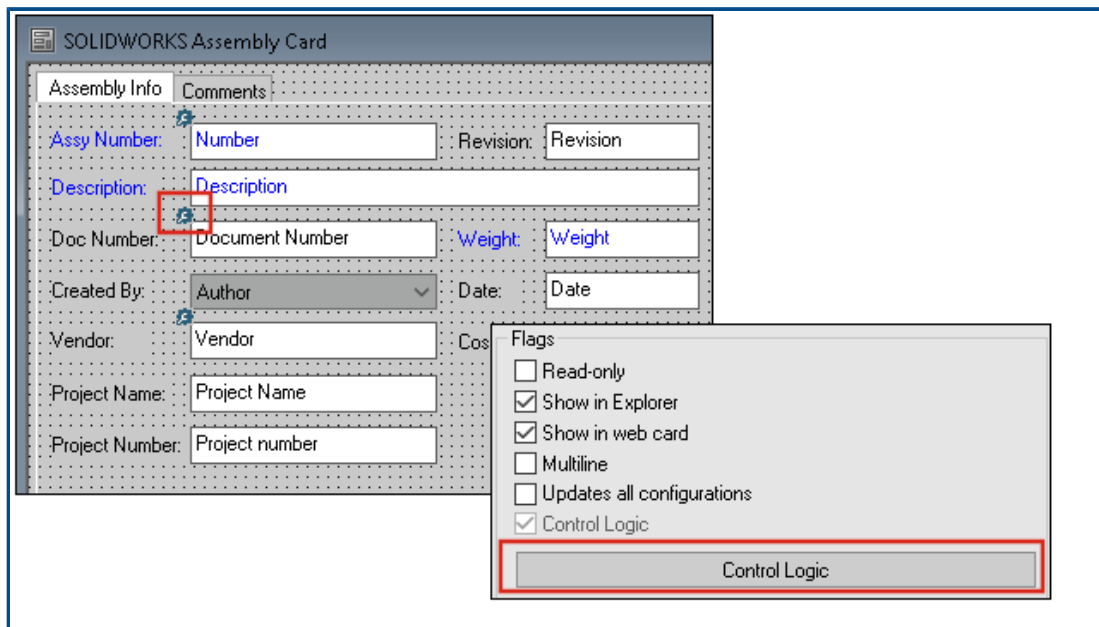
In the Administration tool, under **Local Settings**, double-click **Settings** or right-click **Settings** and click **Open**.

In the dialog box, under **Administration Login**, select a **Vault** and **Use automatic login for this vault**. You can specify:

- **Login as the following user.** Logs you in using the specified user name and password.
- **Log in as the currently logged in user in Windows.** Logs you in using the user name and password of the current Microsoft Windows profile. (Available only for vaults that are configured to use a Microsoft Windows login.)



## Data Card Controls (2023 SP2)



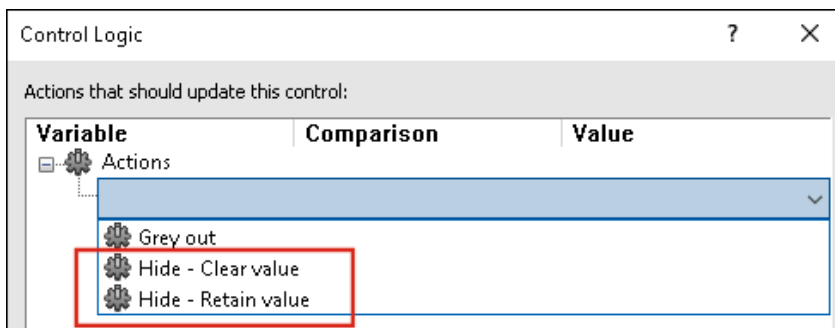
You can apply control logic in data cards more easily.

In addition to the **Control** menu, you can also access the **Control Logic** command when you:

- Click  in the Controls toolbar.
- Click **Control Logic** in the properties pane.

Not available for card search control and variable search control.

When adding control logic, you have more flexibility in selecting the behavior of a control. In the Control Logic dialog box, the **Hide - Clear value** and **Hide - Retain value** options let you specify whether the hide action in the control logic clears or retains the value. Previously, only the **Grey out** and **Hide** options were available.



In the Controls toolbar, if you select **Show Controls with Control Logic**, an indicator appears in the top-left corner of the controls that have control logic applied.

You can also configure the **Find User** command type for a button control to return more user properties. Previously, only **Login name** was available as the user value to populate the **Destination variable** for the **Find User** list.

Command type:  
Find User

Dialog box caption:

Destination variable:  
SenderName

User Value:  
Full name

☐ Permit multiple selection

## Performance Improvements for File Version Upgrade Tool

You can upgrade SOLIDWORKS files in a SOLIDWORKS PDM vault faster.

If you are upgrading assembly files and you select **Overwrite** and **Latest Version** in the Version Settings page, the software skips upgrading and uploading references that are not the latest in the assembly's as-built version. This improves the performance of the upgrade process.

## Logging File Retrieval Operations

Type	Date	Description	Destination Machine	Destination IP Address	User	ID
Information	2022-03-04 10:30:10	Get via PDM Client	ClientMachine	10.10.10.10	Admin	9
Information	2022-03-04 10:37:30	Get via PDM Client	ClientMachine	10.10.10.10	Admin	3
Information	2022-03-04 11:00:17	Get via PDM Client	ClientMachine	10.10.10.10	Admin	4
Information	2022-03-04 11:24:12	Get via PDM Client	ClientMachine	10.10.10.10	Admin	7
Information	2022-03-04 11:25:27	Get via PDM Client	ClientMachine	10.10.10.10	Admin	2
Information	2022-03-04 11:28:55	Get via PDM Client	ClientMachine	10.10.10.10	Admin	3
Information	2022-03-04 11:29:36	Get via COM API	Local	Local	Admin	1

You can maintain a server-side log of files retrieved for a vault. (SOLIDWORKS PDM Professional only.)

To enable logging, in the Administration tool, right-click a vault and select **Properties**. In the File Vault Properties dialog box, under **Logging operations**, select **Get**. This enables logging of the **Get** operation for the vault on all archive servers.

You must have **File vault management** administrative permission.

To view the file retrieval log, in the Administration tool, right-click a vault, and select **Show the Get Log**. The Log File dialog box displays log information for each file in the vault that is downloaded from the archive server to the local view of a client machine, Web2 server, or Web API server.

## Log File - Get Operations Dialog Box

The Get Operations dialog box lets you view informational and error events for the files retrieved for a vault on all archive servers.

### To display the log file:

In the Administration tool, right-click a vault, and select **Show the Get Log**.

<b>Type</b>	Event type: error, informational, or warning.
<b>Date</b>	Date and time of the event.
<b>Description</b>	Method used to invoke the Get operation: <ul style="list-style-type: none"> <li>• Desktop client</li> <li>• COM API</li> <li>• Web API</li> <li>• Web2 server</li> </ul>
<b>Destination Machine</b>	Name of the destination machine that receives data from the archive server.
<b>Client IP Address</b>	IP address of the client machine <div style="border: 1px solid black; padding: 5px; margin-top: 5px;"> For file retrieval operations through Web2 or Web API servers, the IP address of the server machine is logged. </div>
<b>User</b>	Login name of the user who performed the Get operation.
<b>Document ID</b>	ID of the document
<b>Document Name</b>	Name of the document
<b>Version</b>	Version of the document
<b>Document Path</b>	Path of the document in the vault
<b>Archive Server</b>	Name of the archive server on which the event occurred.

## Support for Microsoft Edge WebView2-Based Controls

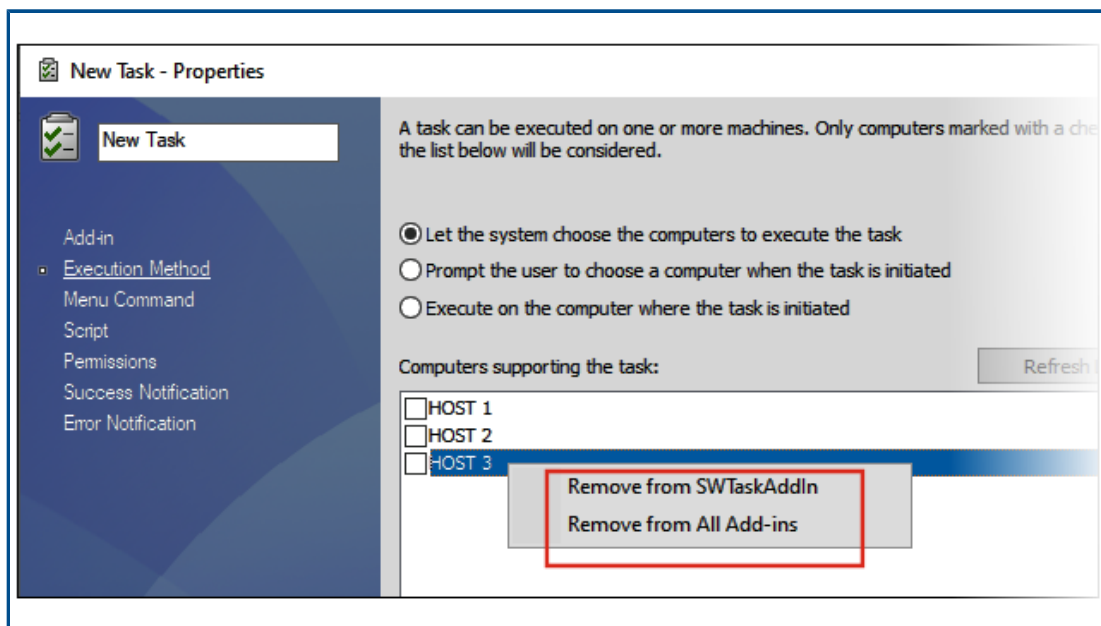
In SOLIDWORKS PDM, the Internet Explorer-based controls are replaced with WebView2-based controls.

This applies for:

- Preview pane in SOLIDWORKS PDM File Explorer. You can preview the HTML documents that are present in the vault.
- EXALEAD OnePart Search view
- Web Card Configuration Editor dialog box

You must have the WebView2 runtime installed on your machine.

## Removing Computers from the Task Execution List



You can remove computers from the list of hosts that support task execution.

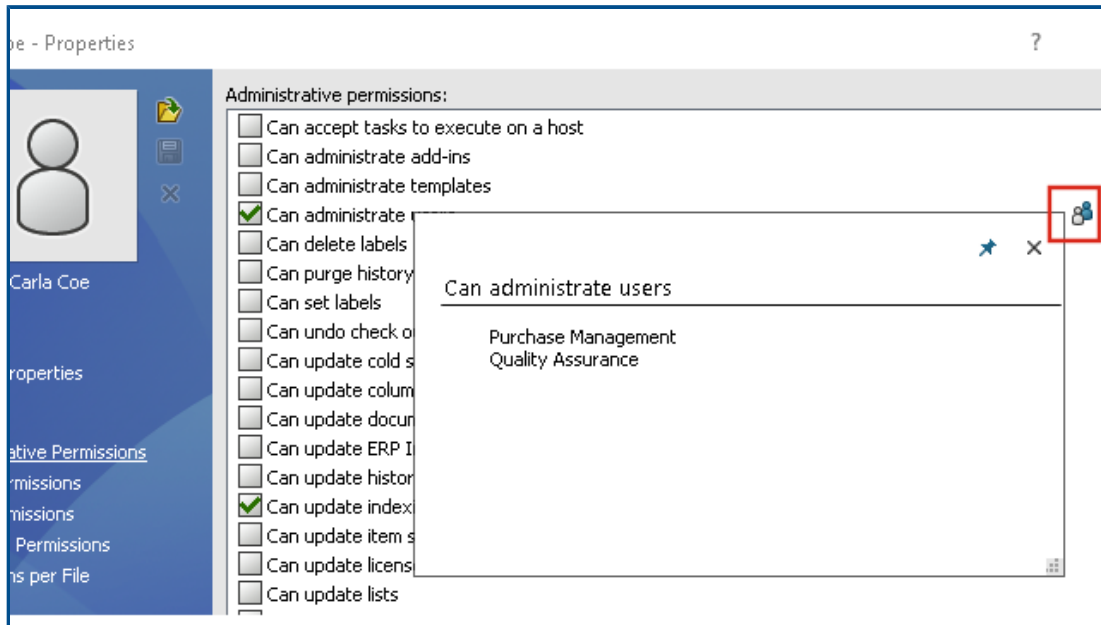
In the Administration tool, open the task's Properties dialog box. In the left pane, click **Execution Method**. In the **Computers supporting the task** list, right-click a computer, and select one of the following:

- **Remove from Add-in Name.** This removes the computer as a supported host for all tasks that use the same add-in.
- **Remove from All Add-ins.** This removes the computer as a supported host for all task add-ins.

A confirmation message lists all the affected tasks.

The Task History dialog box displays a list of computers that are removed for that task.

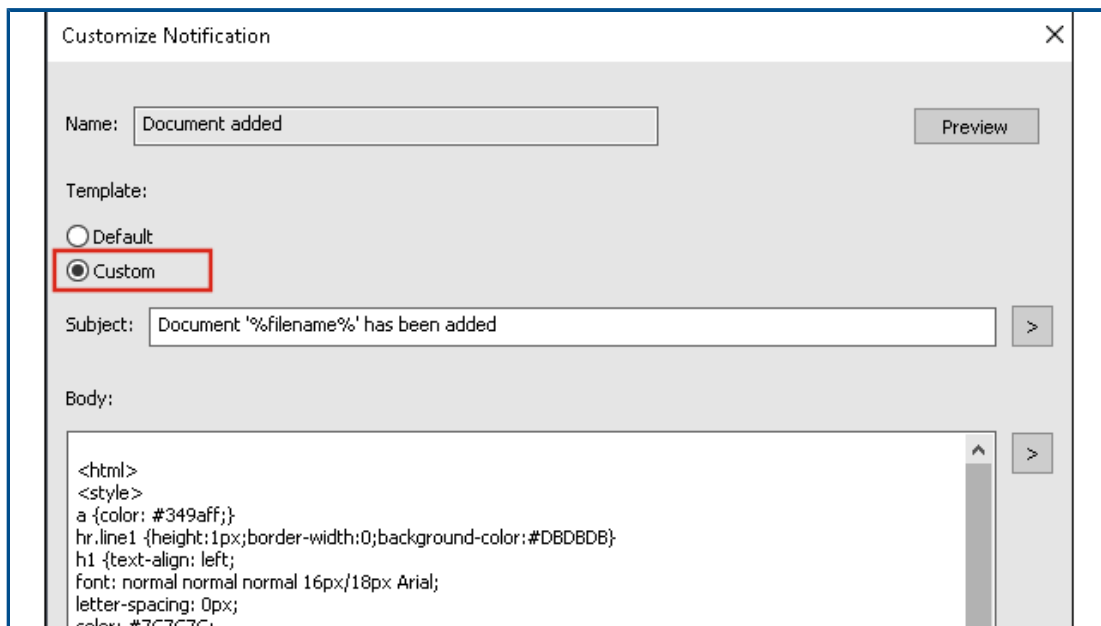
## Display Groups for Inherited Permissions



In the Administration tool, in the user Properties pages, you can hover over the group icon of a permission to view the groups from which the permission is granted to the user.

A pop-up dialog box appears that lists these groups.

## Notification Templates

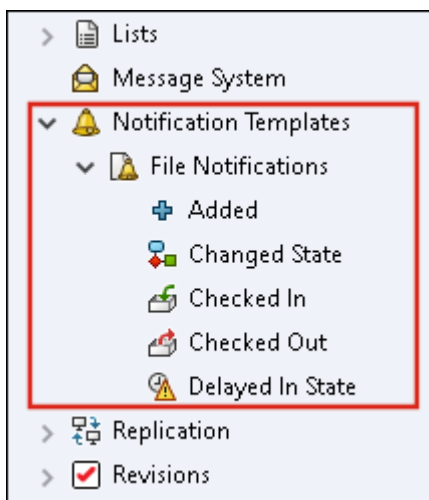


SOLIDWORKS PDM lets administrators customize the templates for automatic notifications for a vault. This helps to display information that is more relevant to the user.

- You must have **Can update mail configuration** administrative permission.
- You can customize notifications only for files and not for items and Bills of Material (BOMs).

You can customize notification templates for the following file operations:

- **Added**
- **Changed State**
- **Checked In**
- **Checked Out**
- **Delayed in State**



You can edit the subject and body of the template in HTML format and also use variables in the notification message.

You can also export notification templates. To export:



- All notification templates. Right-click the **Notification Templates** node and select **Export**.
- File notification templates. Expand **Notification Templates**, right-click **File Notifications**, and select **Export**.
- The notification template for a file operation. Expand **Notification Templates > File Notifications**, right-click a file operation, and select **Export**.

## Customize Notification Dialog Box

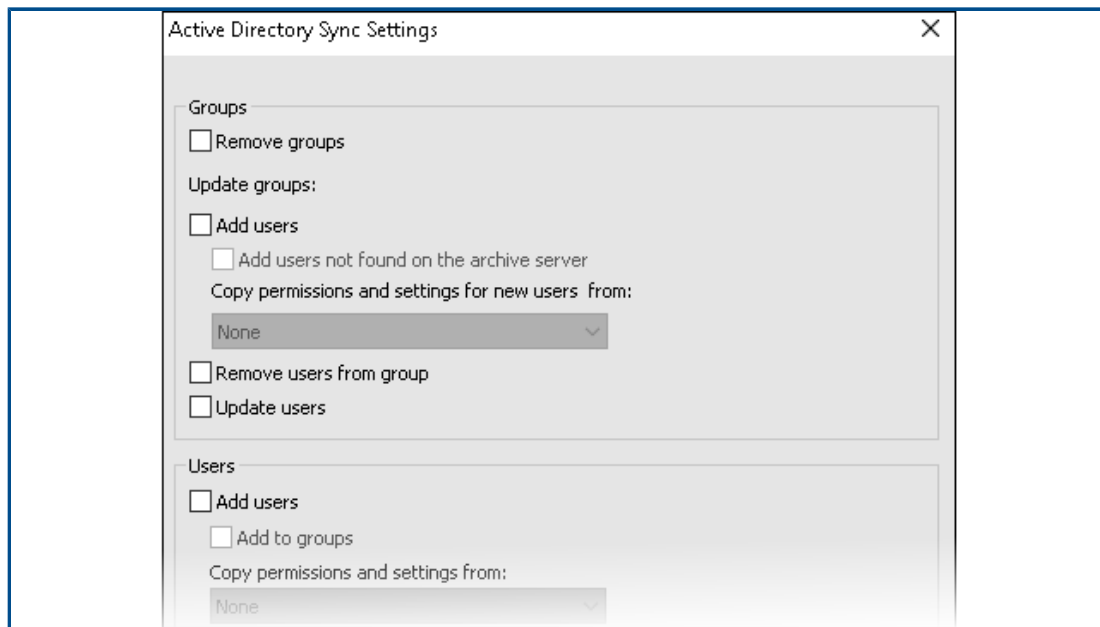
This dialog box lets you customize the templates for automatic notifications for a vault.

**To access this dialog box, do one of the following:**

- Expand **Notification Templates > File Notifications**, right-click a subnode, and select **Open**.
- Expand **Notification Templates > File Notifications**, and double-click a subnode.

<b>Name</b>	Displays the name of the notification template. This field is read-only.
<b>Preview</b>	Displays how the template appears to users.
<b>Template</b>	<p>Lets you select either of the notification templates:</p> <ul style="list-style-type: none"> <li>• <b>Default</b></li> <li>• <b>Custom</b></li> </ul> <p>If you select <b>Custom</b>, the fields listed below in the table are enabled. This lets you customize the notification template.</p>
<b>Subject</b>	<p>Displays the notification subject. You can edit the notification subject for custom notifications. To include variables, click  and select a variable.</p>
<b>Body</b>	<p>Displays the notification body. You can edit the notification body for custom notifications. To include variables, click  and select a variable.</p>
<b>Load from File</b>	Imports HTML text into the body of the notification.
<b>Reset</b>	Resets the notification template to the default template.

## Synchronizing Users with Windows Active Directory (2023 SP1)



You can synchronize users and groups in a vault with Windows® Active Directory as an on-demand operation.

You must have **Can administrate users** administrative permission.

You can define preferences for sync in the Active Directory Sync Settings dialog box. To access this dialog box, right-click the **Users and Groups Management** node and select **Active Directory Sync > Settings**.

To perform Active Directory sync, right-click the **Users and Groups Management** node and select **Active Directory Sync > Run Sync**.

The **Update from Active Directory** command is now also available for Windows users in a vault. Previously, it was available only for Windows groups in a vault.

## Active Directory Sync Settings Dialog Box

Use the Active Directory Sync Settings dialog box to define preferences for on-demand sync of users and groups in a vault with Windows Active Directory.

### To access this dialog box:

Right-click the **Users and Groups Management** node and select **Active Directory Sync > Settings**.

## Groups

<b>Remove Groups</b>	Removes Windows groups from SOLIDWORKS PDM vault that no longer exist in Active Directory.
----------------------	--

## Update Groups

<b>Add users</b>	Adds existing users or creates new users in the vault based on the group membership in the Active Directory group.
<b>Add users not found on the archive server</b>	Creates new users in a vault even if they are not found on the archive server.
<b>Copy permissions and settings from</b>	For new users, copies permissions and settings from the selected user.
<b>Remove users from group</b>	Removes Windows users from the SOLIDWORKS PDM group who are no longer members of the Active Directory group.
<b>Update users</b>	Updates Windows user properties for existing members of a Windows users group with the attributes of the user account in Active Directory.



## Users

<b>Add users</b>	Adds new Windows users that exist in users and groups defined in the Windows login settings on the archive server but not in the vault.	
	<b>Copy permissions and settings from</b>	For new users, copies permissions and settings from the selected user.
<b>Remove users</b>	Removes Windows users from the vault if they no longer exist in the Windows login settings on the archive server.	
	<b>Delete</b>	Deletes the user from the SOLIDWORKS PDM vault.
	<b>Refuse login</b>	Specifies the <b>Refuse Login</b> option in the Administrative permissions page of user properties.
<b>Update users</b>	Updates user properties for existing Windows users with the attributes of the user account in the Active Directory.	

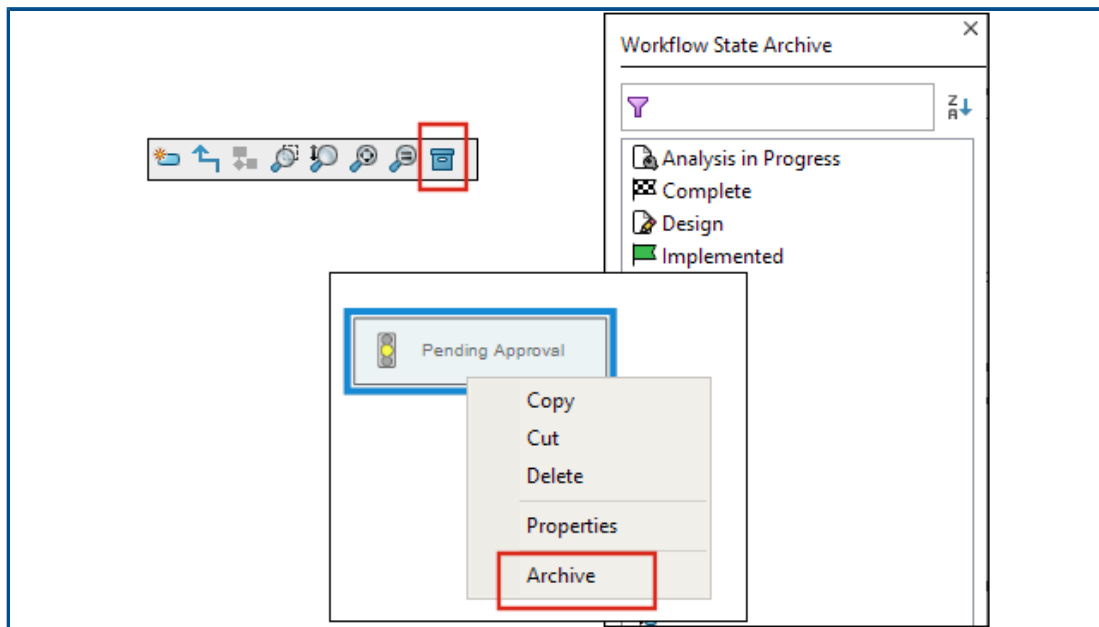
## Days in State Variable (2023 SP1)

In the Customizable Columns dialog box, for a column, you can select **Days in State as Variable**.

This system variable is available for the following column set types:


- **File List**
- **Quick Search Result**
- **Search Result**

## Archive Workflow States (2023 SP2)



You can archive a workflow state that is not connected to any transition. This lets you see a workflow without any obsolete states in the view. Administrators can view archived states and unarchive them.

You cannot archive states that are connected to other states.

On the Workflow toolbar, click  to display the Workflow State Archive pane. In this pane, you can view all the archived states. You can also sort the list alphabetically.

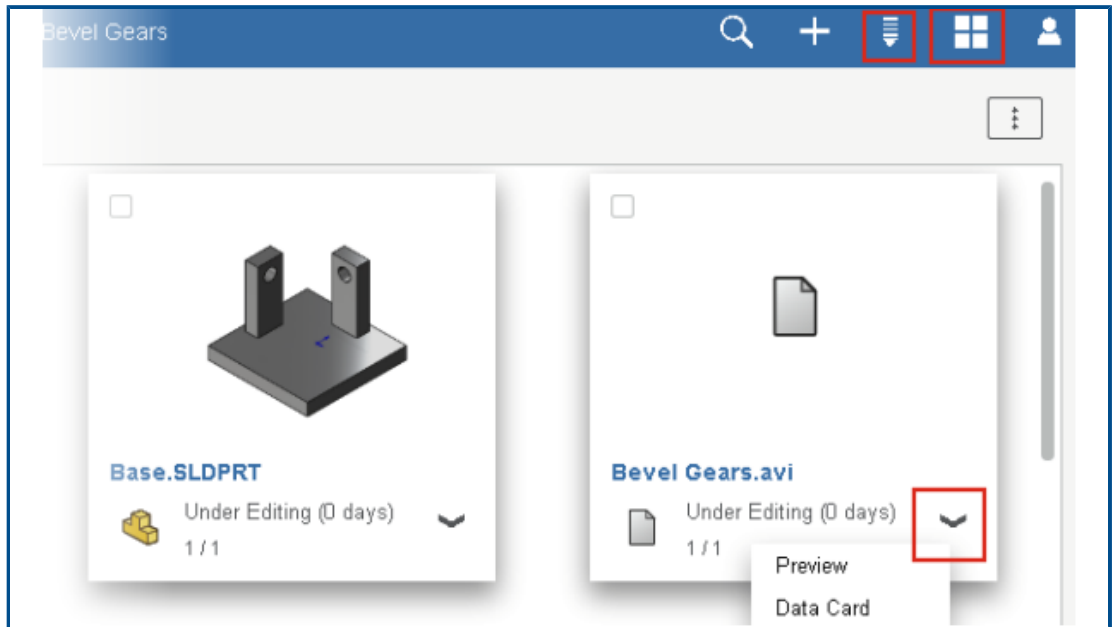
To archive a workflow state:

- Right-click the state and select **Archive**.
- Drag the state to the Workflow State Archive pane.

To unarchive a workflow state:

- In the Workflow State Archive pane, right-click the state and select **Unarchive**.
- Drag the state from the Workflow State Archive pane to the workflow.

## Grid View in Web2 (2023 SP2)





In the Web2 client, you can see a grid view of thumbnail previews in file lists and search results.

The grid view layout is available in the following pages of large screen and small screen layouts:



- File or folder list
- Search results

To switch the layout from the list view to the grid view:

- In the large screen layout, click  and select **Grid View**. In this view, you can click  and select a direction and parameter to sort the list.
- In the small screen layout, touch the ellipsis, expand **List View**, and then select **Grid View**. To sort the list, expand **Sort By** and select a direction and parameter.

The **Select All** and **Deselect All** options are available in the grid view layout.

To view more information for a file, you can access the File View tabs through the quick links menu. To access this menu in:

List view	Hover over a row and click  or right-click the file.
Grid view	Click  or right-click the file.

The quick links menu is available only in:

- File or folder lists
- File details view
- Search results

## Administrative Permissions for Tasks (2023 SP2)

Administrative permissions for tasks are redefined.

Permission	Description
<b>Can view task lists</b>	View the task list and perform all the actions that are available in the Task List dialog box.
<b>Can update task settings</b>	Create and modify tasks.

## SOLIDWORKS PDM Performance Improvements (2023 SP04)

SOLIDWORKS PDM 2023 makes significant performance improvements in the presence of high latency to SOLIDWORKS PDM servers.

The improvements might vary depending on the amount of latency, size of datasets, and operations.

- The following operations perform better when latency to the database server is high:
  - Check-in
  - Check-in with automatic transition
  - Undo Check-out
  - Interactive operations such as menus, shortcut menus, reference dialog boxes, SOLIDWORKS PDM File Explorer tabs
- The following operations perform better when latency to the archive server is high:
  - Add files
  - Change state on file modifications
  - Get files
  - Get latest version of files

In SOLIDWORKS PDM 2023 SP04, the mechanism for loading of SOLIDWORKS PDM and custom add-ins is reworked. As a result, operations such as logging in to a vault and starting the SOLIDWORKS PDM Search tool perform significantly better.

## Other SOLIDWORKS PDM Enhancements

SOLIDWORKS PDM 2023 offers enhanced data security, support for the latest Windows® operating system, and other improvements.

- SOLIDWORKS PDM encrypts all the data traffic to and from the archive server.
- SOLIDWORKS PDM 2023 supports the Windows® 11 operating system.
- In SOLIDWORKS PDM File Explorer, you can copy static text from data cards.
- The language for the user interface of eDrawings® Web Viewer follows the language specified in Web2.

# 12

## SOLIDWORKS Manage

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This chapter includes the following topics:

- **Automatic Update of Mapped Linked Record Properties**
- **Using the New From Option**
- **Task Administration Access**
- **Linked Record BOM Compare Field Selection**
- **Support for Variable Driven BOM Quantities**
- **Timesheets to Approve Node**
- **Cancelled Task Line in Capacity Planning**
- **Capacity Planning User Efficiency**
- **Hide Labels**
- **Marking Tasks as Complete**
- **Task Snooze Reminder Time**
- **Tasks Board Time Display**

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.

## Automatic Update of Mapped Linked Record Properties

 **Linked Record Options**

**Fields Mapping**  
Document fields on the LEFT - Record fields on the RIGHT

Document Object Column	Record Object Column	Update method	Update Automatically	Info
Constructed by				
ContextCategory				
Copyright				
Cost				
CreationTime				
Customer Name				
Date				
Derived From				
Description	Description	Always	<input checked="" type="checkbox"/>	
Description				
Description 2				
Dest. doc. ID				

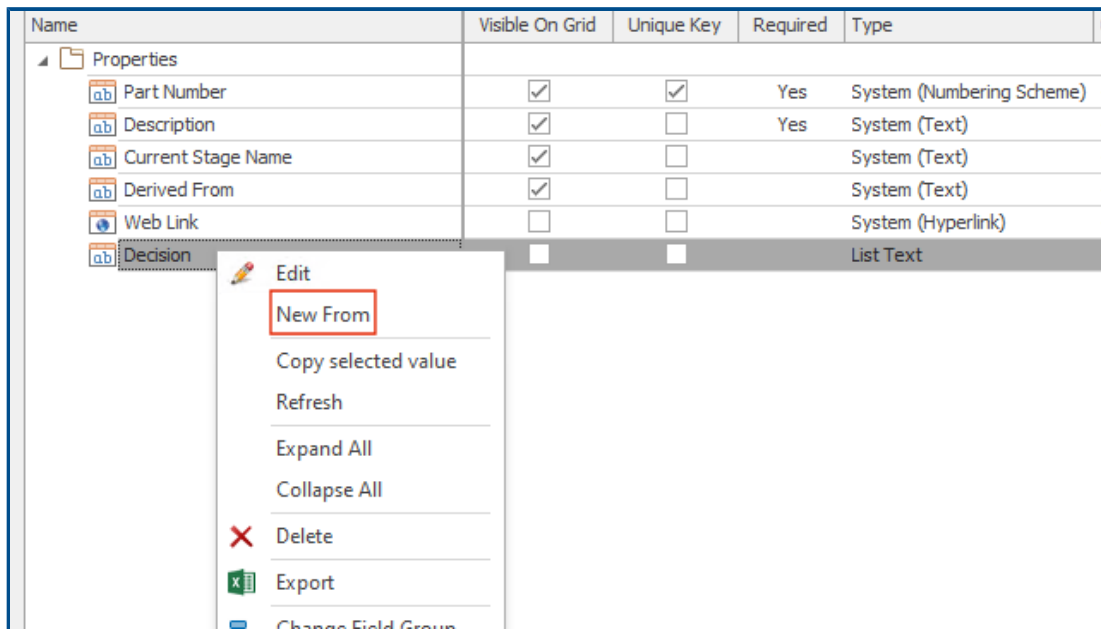
You can update mapped properties automatically to connect primary references and the linked record they are associated with.

The **Linked Record** updates when the **Primary Reference** changes and the record is checked in. Previously, you must check out the linked record and click **Get values from Primary Reference**. You can define each mapped field to update automatically.

### Updating Mapped Fields Automatically

1. In the SOLIDWORKS Manage Administration Settings tool, edit the **Primary Reference** object.
2. On the Linked Record Object wizard page (for PDM objects) or on the Linked Record tab for document objects, click **Configure**.
3. On the Fields Mapping wizard page:
  - a) Select the check box in the **Update Automatically** column for the field to update automatically.
  - b) In the right pane, for **Linked Record** with the status, select a status to update when the primary reference is updated.

## Using the New From Option



**New From** lets you copy a field within an object to create new fields with the same settings.

It saves the time required for administrators to configure objects.

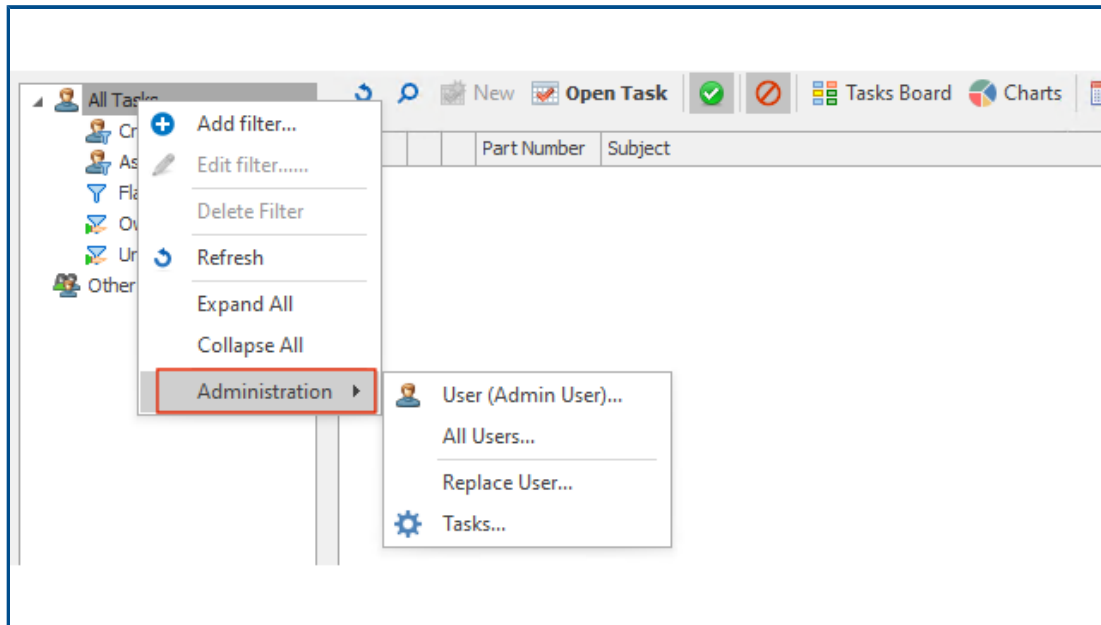
**New From** is not available when you configure BOM fields or process item fields.

To use the **New From** option:

1. In the Administration tool, edit an object or right-click an object in the main user interface.
2. Navigate to the Fields tab or wizard page.
3. Right-click a field and select **New From**.
4. In the New From dialog box, enter a display name for the new field and do one of the following:
  - Click **Save** to create the new field and close the dialog box.
  - Save and create a new field based on the originally selected field.



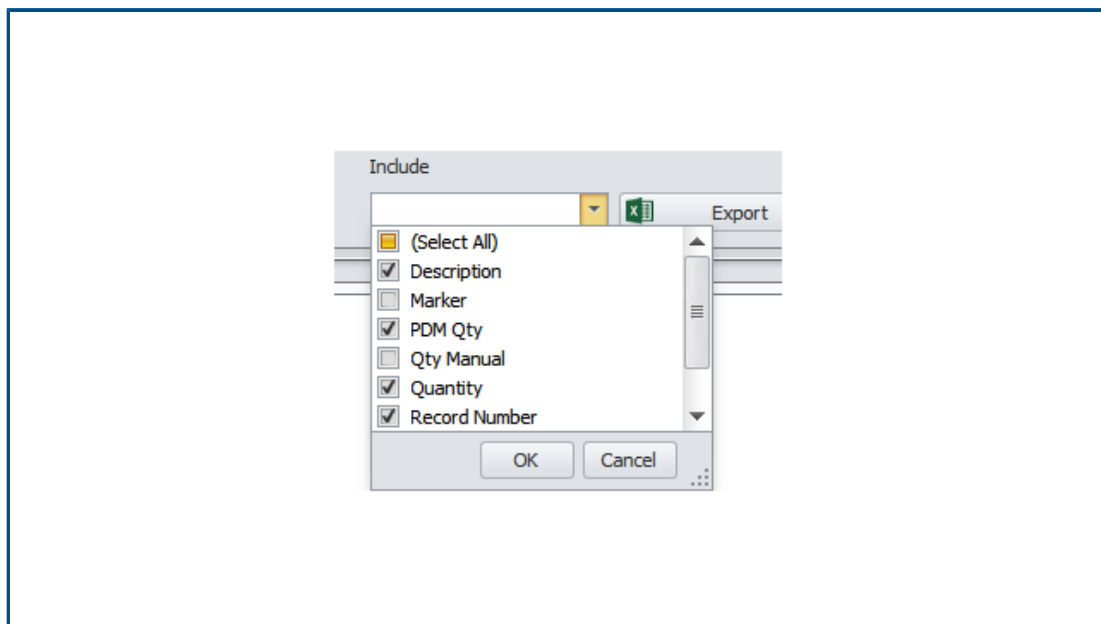
## Task Administration Access



You can open the Task Administration dialog box using the **Tasks** option.

To open the dialog box, in the Administration tool, click **All Tasks > Administration > Tasks**.

## Linked Record BOM Compare Field Selection



When you compare a linked record BOM to the BOM of its primary references, you can select fields to compare.

For example, when you assign a new part number to the linked records, the field is always different from the primary reference part number. So you can exclude this field from comparison because it always shows a difference.

## Support for Variable Driven BOM Quantities

**Edit Variable**

Variable name:

Variable type:

☐ Version free  
☐ Mandatory values  
☐ Only look for mandatory values in @ tab for SLDD files  
☐ Unique values

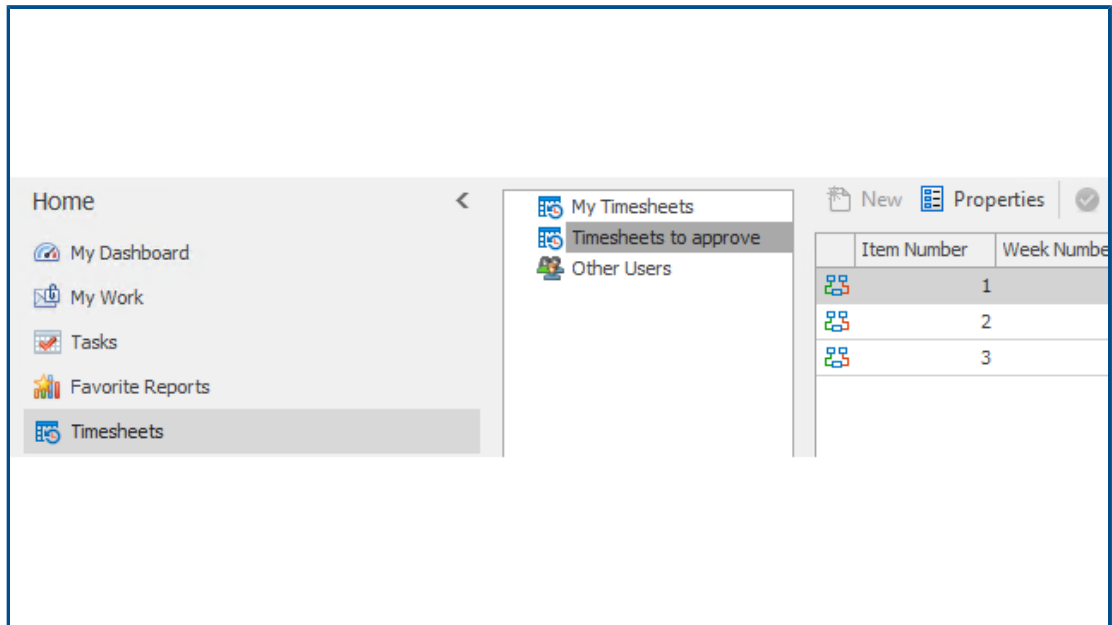
Attributes:

Block	Attribute	Extensions
CustomProperty	UNIT_OF_MEASURE	sldprt

SOLIDWORKS Manage reads the BOM quantity from a SOLIDWORKS PDM variable set and uses it for the *UNIT\_OF\_MEASURE* attribute.

Previously, SOLIDWORKS Manage used the instance count for the *UNIT\_OF\_MEASURE* attribute.

## Timesheets to Approve Node



**Timesheets to approve** lets you see the timesheets that a user needs to work on, to complete the timesheet approval process.

## Cancelled Task Line in Capacity Planning

User Details									
User:	Admin User			Year:	2022				
Activity	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug	
Σ Total Time	31	28	31	30	31	30	31	31	
⏏ Non-Working Time	10	8	8	9	9	8	10	8	
📁 Tasks	7.2	0	0	0	0	0	0	0	
📁 Assigned Tasks	22.2	0.2	0	0	0	0	0	0	
📁 Completed	0	0.2	0	0	0	0	0	0	
⏏ Cancelled	15	0	0	0	0	0	0	0	
✅ Available capacity	13.8	20	23	21	22	22	21	23	

Task Charts

You can see the number of hours of cancelled tasks in the **Cancelled** row of the User Details dialog box.

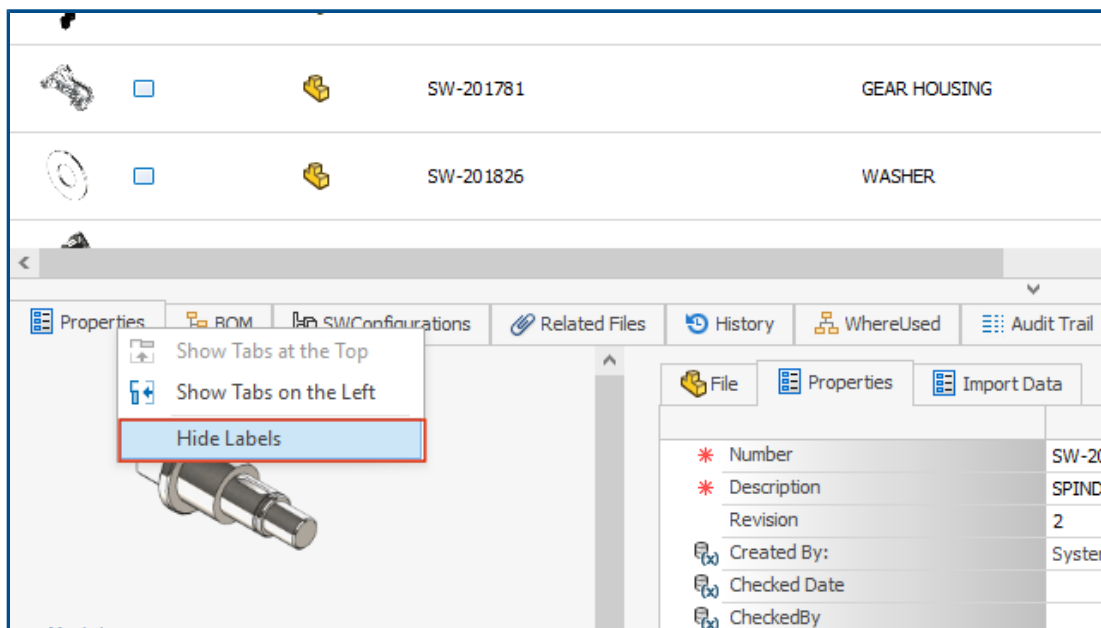
The **Assigned Tasks** row displays total assigned task hours including the cancelled task hours. The **Available capacity** row displays assigned task hours minus cancelled task hours.

## Capacity Planning User Efficiency

When you view user details, the **Efficiency** (%) value that appears in the Capacity Planning dialog box is applied to the user's time.

Previously, the **Efficiency** in user details was 100% regardless of the setting in the Capacity Planning dialog box.

## Hide Labels



**Hide Labels** lets you hide captions. This creates more space in the software so you can see the displayed tabs on the user interface.

To hide captions for the following items, right-click tabs, and select **Hide Labels**:

- Bottom tabs in the main user interface for all objects
- Property card tabs for processes
- Administration for document and record objects (not SOLIDWORKS PDM)
- Administration for relationships and resource objects

To hide labels for property cards for document and record objects, click .

## Marking Tasks as Complete

Relationships  
Resources  
Web Options  
Dashboards  
Timesheets  
Tasks

Other options

[Order tasks by...](#)

☐ Always disable "Complete (%)"  
If disabled, the "Progress" section will be used. This field will be calculated automatically.  
Note: If a task is assigned to multiple users, this field will be disabled for the specific task.

☒ Allow more time than allocated  
☐ Do not allow more time than allocated

Create new tasks as "private" or "public"  
Allow users to decide (My Options) This setting does not overwrite task template

Default time for new tasks (hours) 8

☐ Email task owner when task is completed This setting can be changed in individual tasks

☒ Enable "Mark as completed" for any assignee when multiple users are assigned

In the System Administration dialog box, you can select a check box that enables **Mark as Completed** for a task that is assigned to multiple users.

Previously, the assigned user must enter the required time in the progress area to mark the task as complete.

## Task Snooze Reminder Time

SOLIDWORKS Manage Reminder

TSK000792 Create CAD Models in SOLIDWORKS

Due 8/9/2021 15:00

Subject	Due In	Type
TSK000792 Create CAD Models in SOLIDWORKS	Overdue 179 Days	Normal

Dismiss All Open Dismiss

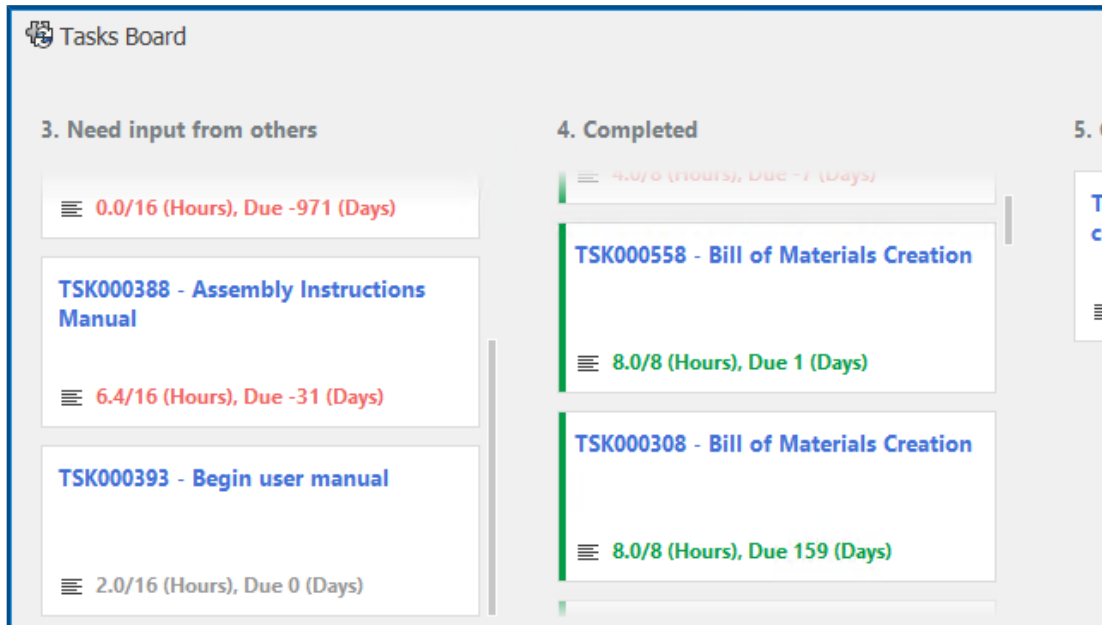
Click Snooze to be reminded again in:

30 Minutes Snooze

The SOLIDWORKS Manage Reminder saves the last task snooze time.

If you want to keep the same reminder time, only click **Snooze**. Previously, the task snooze time was 5 minutes by default so every time user must change the time.

## Tasks Board Time Display



The Tasks Board displays the time required for completion, total assigned time, and number of due days until the next task.

The text is color coded depending on the state of the task. Green indicates complete tasks, red indicates overdue tasks, and gray indicates tasks that are not overdue.

# 13

## SOLIDWORKS Simulation

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This chapter includes the following topics:

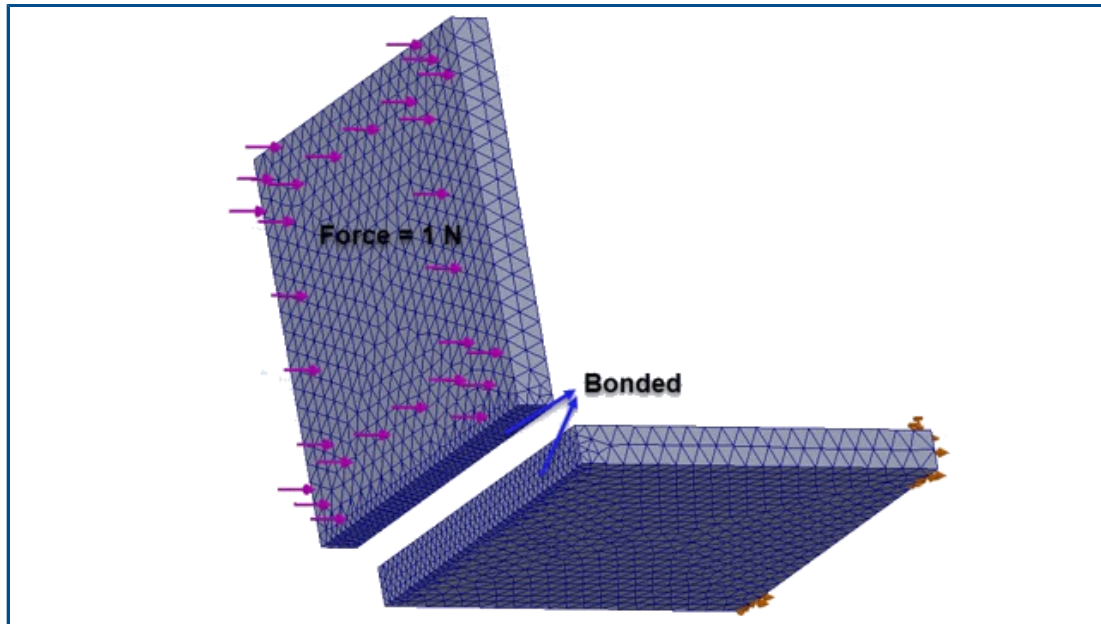
- **Bonding Interactions**
- **Diagnostic Tools**
- **Linkage Rod Connector**
- **Penalty Stiffness Control for Contact**
- **Simulation Solvers**
- **SOLIDWORKS Simulation Knowledge Base (2023 SP1)**
- **Underconstrained Bodies**

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



## Video: What's New in SOLIDWORKS 2023 - SOLIDWORKS Simulation

### Bonding Interactions



An improved surface-to-surface bonding algorithm enforces bonding interactions for faces that are not facing each other and have no projection area between them.

A pair of faces qualifies for bonding if these conditions apply:

- The angle between the faces is less than a threshold value.

If the angle between the faces exceeds the threshold value, the pair of faces does not qualify for bonding.

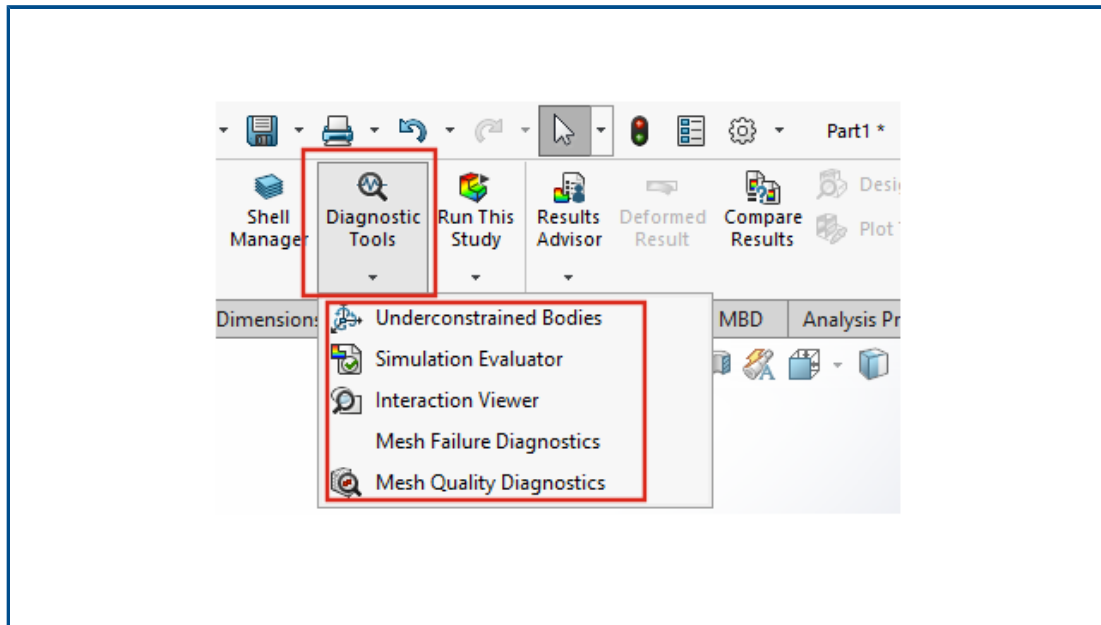
- At least one face of the pair is a flat face (selected as **Set 1** or **Set 2** in the Local Interactions PropertyManager).

The solver considers the flat face as the target face to enforce bonding. The source face can be either flat or curved.

The enhanced bonding algorithm applies to the surface-to-surface formulation and is available for bonded interaction sets that you define using the Local Interactions PropertyManager. It applies to these studies: linear static, frequency, buckling, linear dynamic studies, and also fatigue and design studies associated with linear static studies.



## Diagnostic Tools

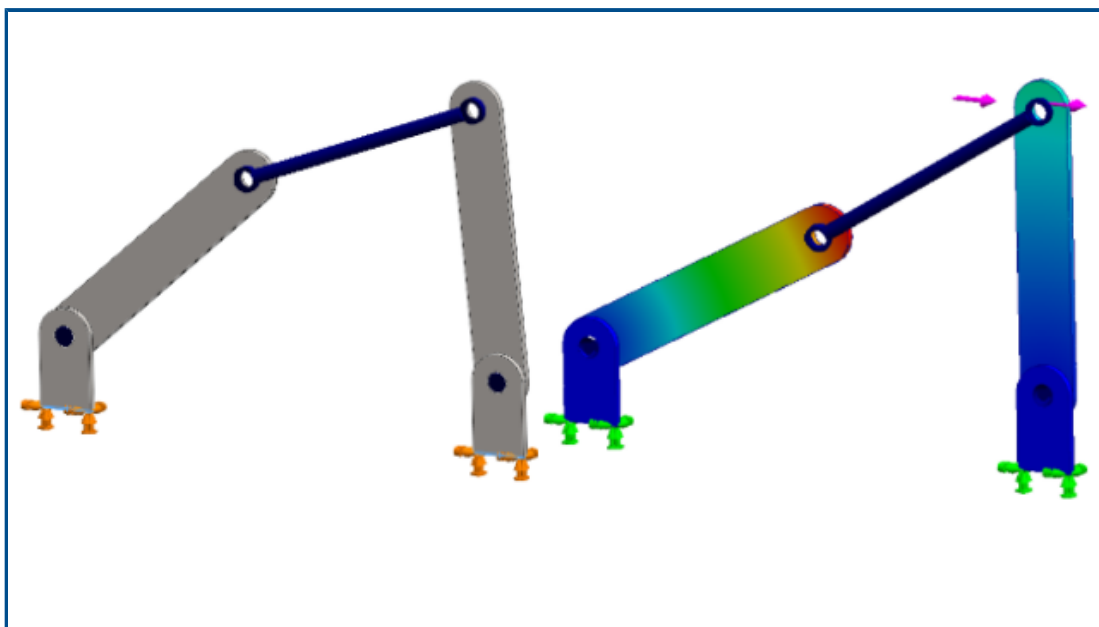


You can access the diagnostic tools available for simulations from the CommandManager.

**To access the diagnostic tools:**

From the **Simulation** toolbar, click **Diagnostic Tools**.

## Linkage Rod Connector




The application of **Linkage Rod** connectors is extended to nonlinear static and nonlinear dynamic studies.

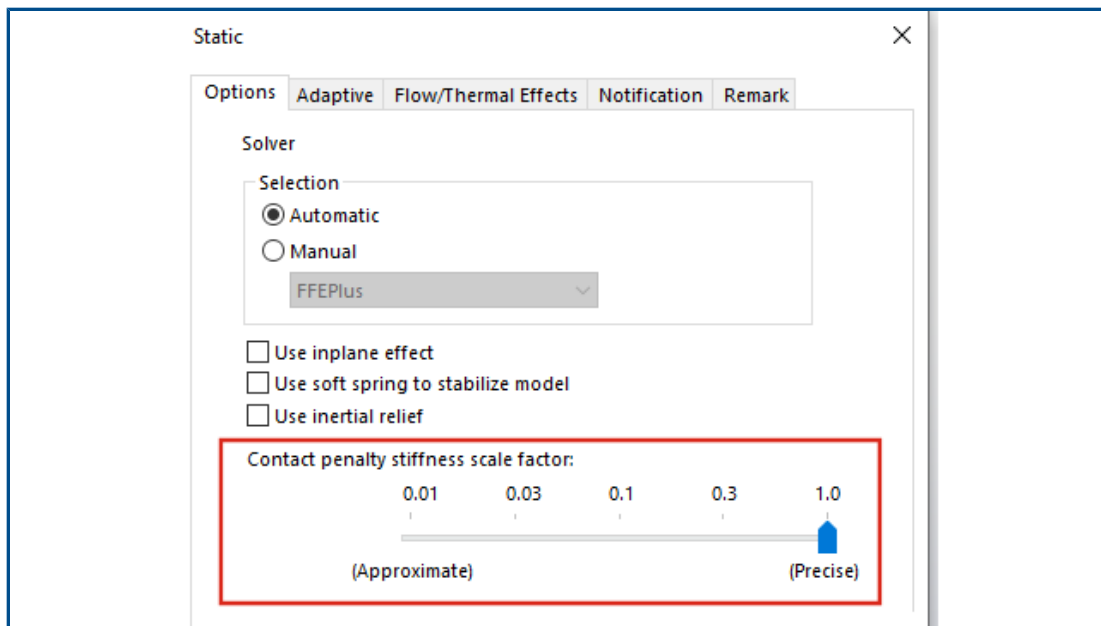
You can specify a **Linkage Rod** connector between cylindrical faces, circular edges (for shells), or vertices to model the behavior of connecting rods.

#### To open the Linkage Rod PropertyManager:

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

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

## Penalty Stiffness Control for Contact

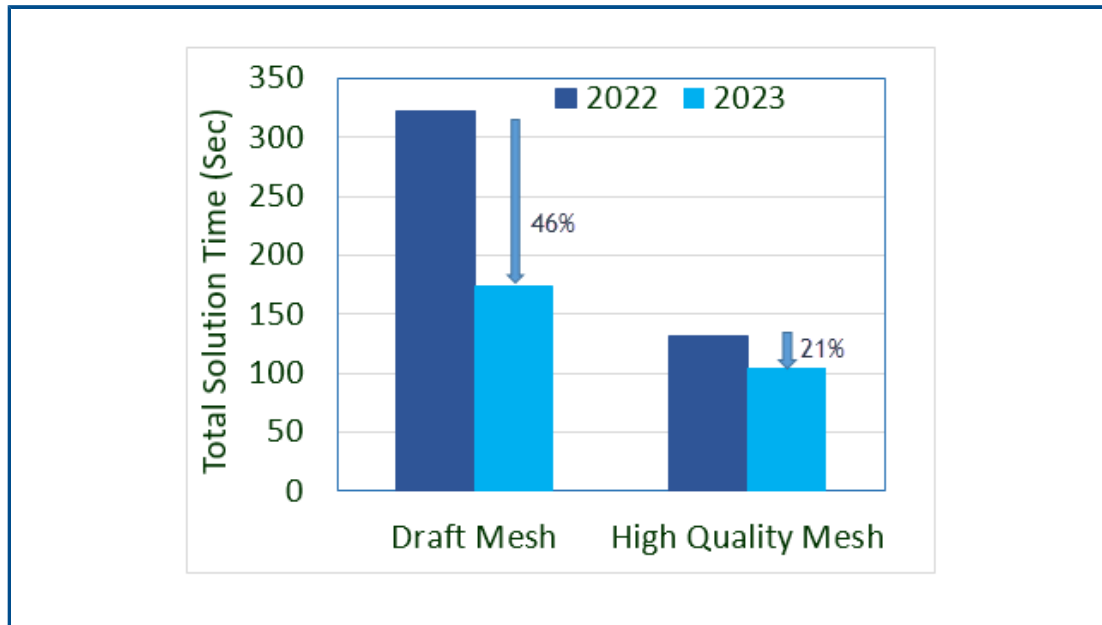


You can specify a scale factor for the penalty stiffness for contact that is used in linear static studies.

SOLIDWORKS Simulation uses a default scale factor of 1.0 for the penalty stiffness to reach an accurate solution for linear static studies with contact interactions. However, you can select a lower scale factor to obtain an approximate solution faster to assess design iterations and the overall behavior of a model.

You can set the study-level scale factor for the penalty stiffness in the Study Properties dialog box. For global-level settings, select **Simulation** > **Options** > **Default Options** > **Interactions**.

## Simulation Solvers



Function-based processing for the FFEPlus iterative solver is extended to frequency and buckling studies, and to the linear static studies that include node-to-surface interactions and virtual wall definitions.

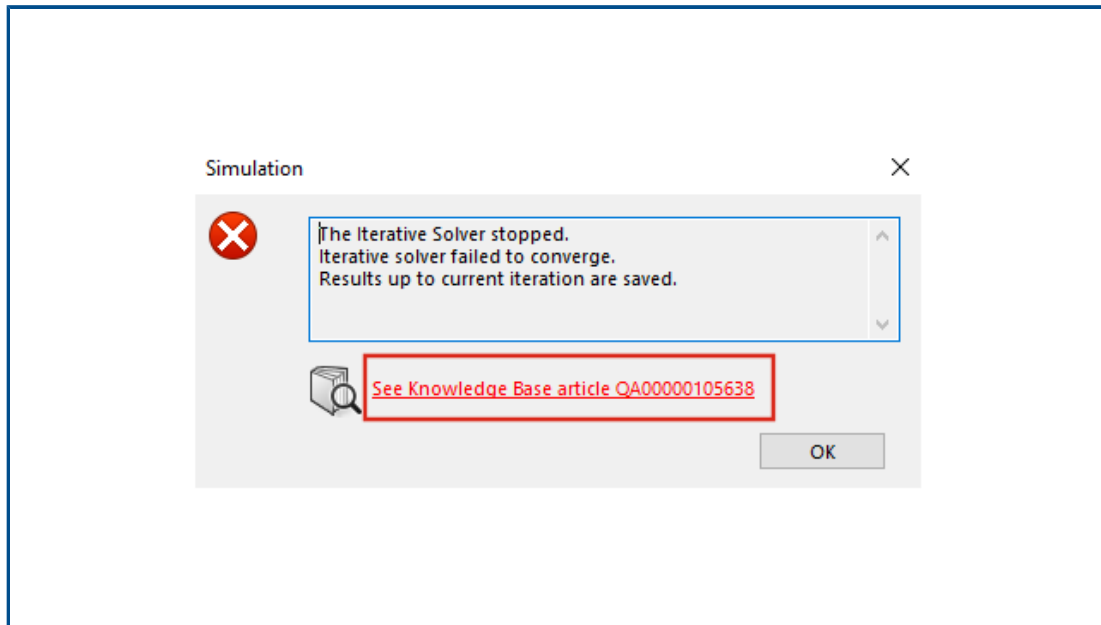
- **FFEPlus Iterative Solver**

The transfer of stiffness data to solve the systems of equations is optimized for linear static, frequency, and buckling studies because file-based processing is replaced with function-based processing. Solution performance is improved for frequency and buckling studies. The image illustrates the solution performance for a typical frequency study.

In addition, solution performance is improved for linear static studies that contain:

- Node-to-surface bonding and contact interactions
  - Virtual wall definitions
- The **FEEPlus** and **Large Problem Direct Sparse** solvers can now handle multiple translational and rotational degrees of freedom that are applied remotely with the distributed connection type.

## SOLIDWORKS Simulation Knowledge Base (2023 SP1)

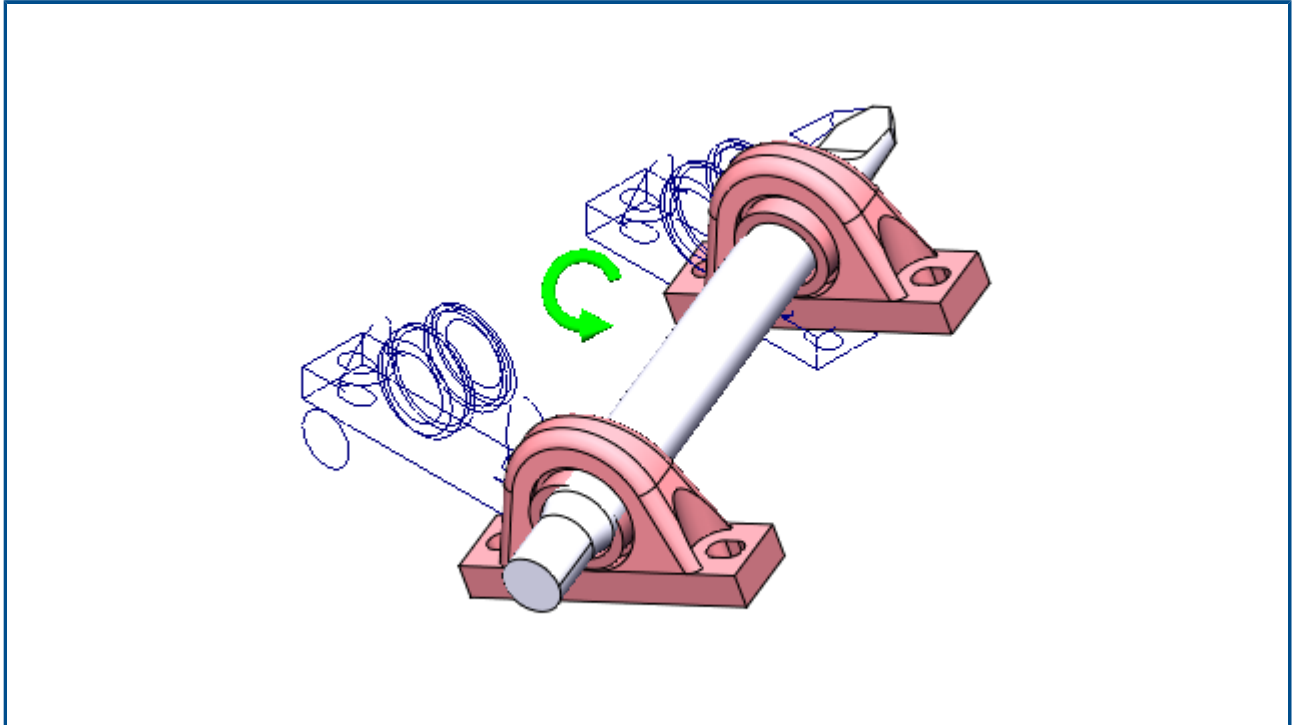


The SOLIDWORKS Simulation Knowledge Base articles have migrated from the SOLIDWORKS Knowledge Base portal to the Dassault Systèmes Knowledge Base portal.

The hyperlinks to the Knowledge Base articles that appear in the simulation solver error messages direct you to the Q&A articles in the Dassault Systèmes Knowledge portal.

You can use the Solution ID of an article published in the SOLIDWORKS Knowledge Base to search for the corresponding Q&A article in the Dassault Systèmes Knowledge Base.

## Underconstrained Bodies



In SOLIDWORKS Simulation Professional and SOLIDWORKS Simulation Premium, for static studies only, the **Underconstrained Bodies** algorithm includes contact interactions and bolt connectors when detecting rigid (or free) body modes of components.

In SOLIDWORKS Simulation Standard and SOLIDWORKS Premium, the **Underconstrained Bodies** algorithm does not include these improvements and remains unchanged from previous releases.

### To open the Underconstrained Bodies PropertyManager:

In the Simulation study tree, right-click the **Connections** node, and click **Find Underconstrained Bodies**.

The improved algorithm offers the following advantages compared to the legacy **Underconstrained Bodies** functionality:

- Faster detection of free body modes because the solver performs Singular Value Decomposition (SVD) over the reduced stiffness matrix.
- Inclusion of contact interactions and bolt connectors when analyzing free body modes.
- Realistic visualization of free body modes in planes that are inclined to the orthogonal planes that are defined by the global X-, Y-, and Z-axes.
- Improved visualization of unconstrained displacements for the whole assembly.

# 14

## SOLIDWORKS Visualize

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This chapter includes the following topics:

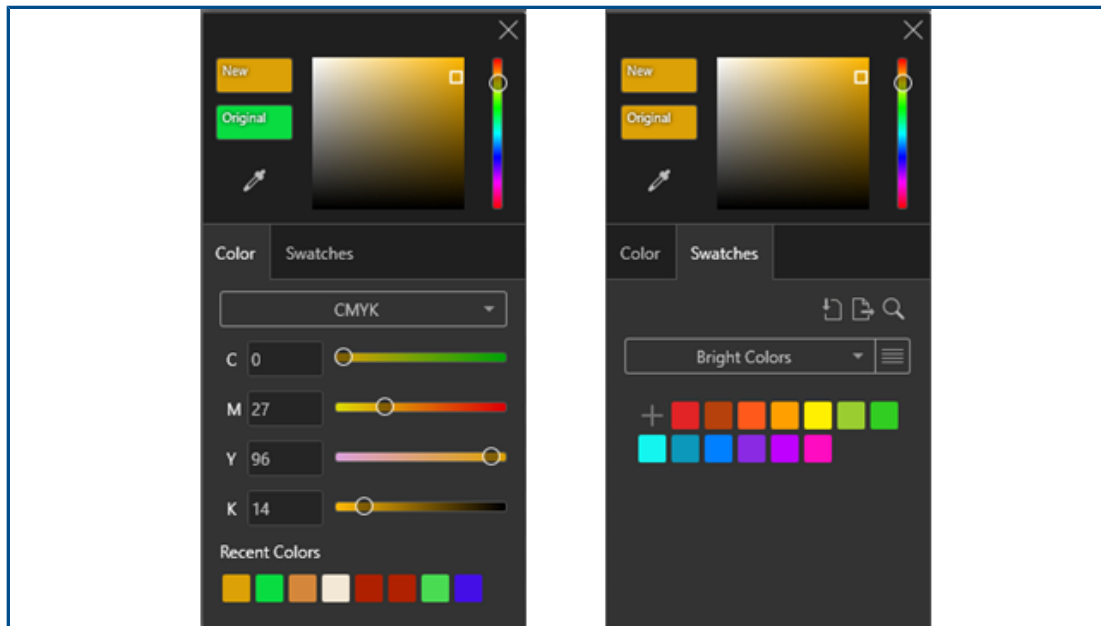
- **Color Picker**
- **DSPBR Material Support**
- **Importing Options**
- **Interactive Images (SW2023 SP2)**
- **PhotoView 360**
- **Shadow Catcher Support with Stellar Physically Correct (2023 SP2)**
- **Stellar Physically Correct**
- **System Info**
- **Sunlight Environment**
- **Transform Manipulator**
- **Render Wizard**

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



## Video: What's New in SOLIDWORKS 2023 - SOLIDWORKS Visualize

## Color Picker




In the Color Picker, you have more ways to define colors, you can display recently used colors, and you can import and export color swatch palettes.


The Color Picker includes two tabs: Color and Swatches.

On the Color tab you can:

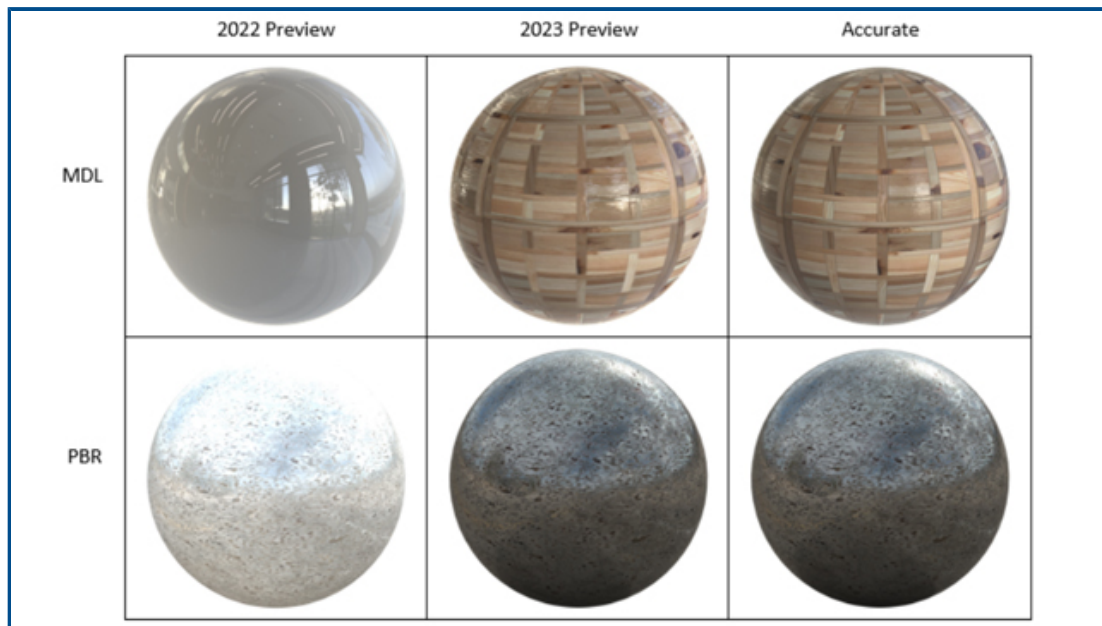
- Define color codes including:
  - **CMYK**, which lets you modify color through CMYK components.
  - **HEX**, which lets you view and edit the hexadecimal code.
- Access the most recently used colors in **Recent Colors**.

On the Swatches tab you can:

- Click **Import Swatch Palette**  to import existing color swatch palettes from:
  - SOLIDWORKS Palettes (.sldclr)
  - ACO files (.aco)
  - SVG files (.svg)
  - HTML files

- Click **Export Swatch Palette**  to export color swatch palettes from SOLIDWORKS Visualize and use them in SOLIDWORKS.
- Select preset swatch palette libraries such as **Bright**, **Pastel**, and **Grayscale**.
- Manage existing or new swatch palettes by adding, naming, duplicating, deleting, and locking them.

## DSPBR Material Support



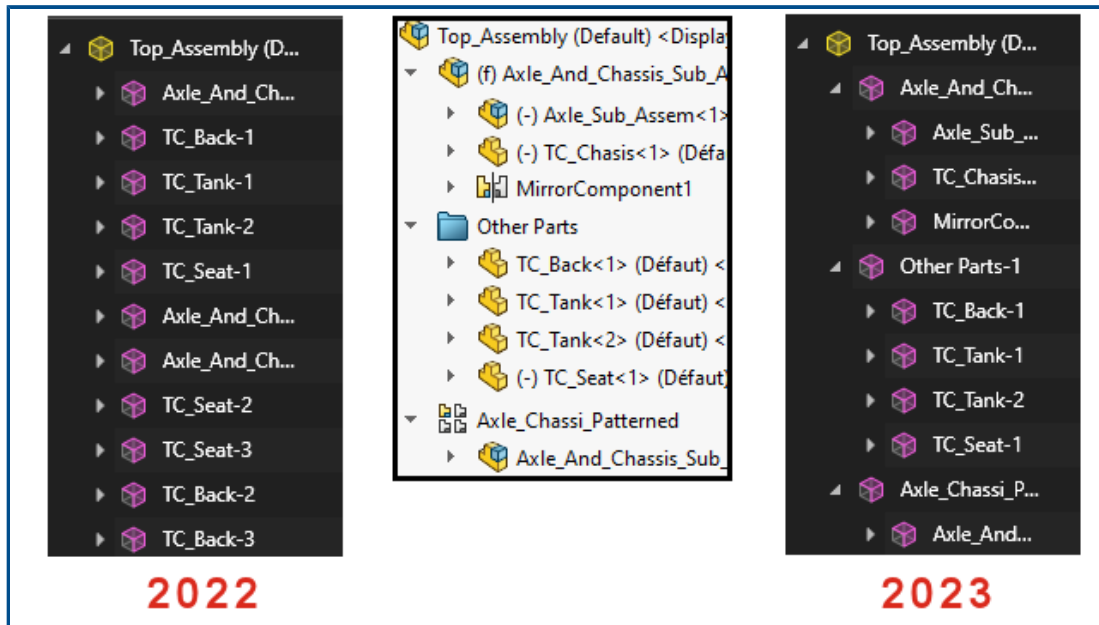
In the **Preview** render mode, display has improved for Physically Based Rendering (PBR) materials, as well as NVIDIA® Material Definition Language (MDL) appearances, and AxF™ appearances.

Improvements include the display of:

- Physically based rendering appearances for better render consistency.
- Materials that use texture distilling in the background, such as MDL and AxF appearances.



## Importing Options



When you import SOLIDWORKS parts and assemblies in SOLIDWORKS Visualize, the **Part Grouping** options include only two choices to streamline the import process.

### To access the importing options:

1. Open a project.
2. Click **File > Import**.
3. Browse to a .SLDPRT or .SLDASM and click **Open**.
4. In the dialog box, on the **Geometry** tab, specify **Part Grouping** options.

**Part Grouping** includes:

- The **Part Grouping** menu, which provides two choices when importing SOLIDWORKS .SLDPRT or .SLDASM files.
  - **Component/Part/Body**
    - Creates nested SOLIDWORKS Visualize groups for SOLIDWORKS assembly components down to the SOLIDWORKS parts and bodies.
    - Creates nested groups for subassembly components, mirror components, pattern components, and SOLIDWORKS folders.
    - Refines SOLIDWORKS Visualize parts based on appearance assignments in each SOLIDWORKS body.
    - Imports saved SOLIDWORKS configurations, animations, and supports **Monitor File** functionality.
  - **Appearance**
    - Creates a SOLIDWORKS Visualize part based on every unique SOLIDWORKS appearance.
    - Combines SOLIDWORKS geometry having the same appearance into a single SOLIDWORKS Visualize part.

- Imports saved SOLIDWORKS configurations.
- Does not import saved SOLIDWORKS animations and does not support **Monitor File** functionality.



Imported part names are based on their ancestry. Previously, the name was based on the imported appearance.

## Interactive Images (SW2023 SP2)



You have more ways to display interactive images.

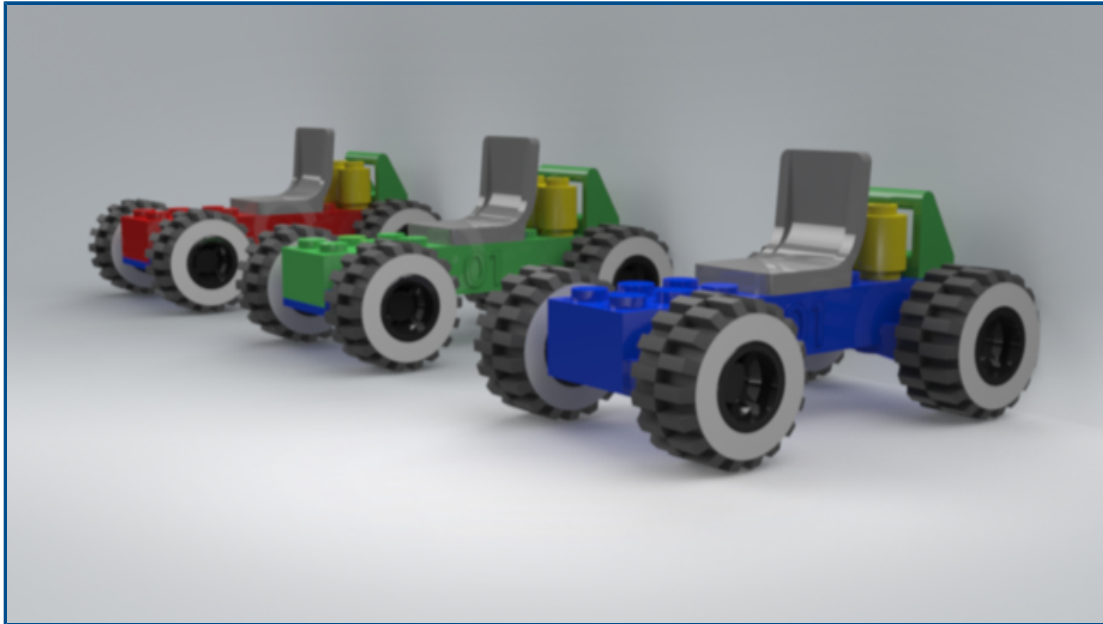
You can:

- Scale the output file in web browsers.
  - Switch (toggle) between **Actual**  and **Fit** . To resize the image to its native resolution, click **Actual**. To fit the image to the screen, click **Fit**.
- Lock the actual size of the image.
- Generate .html files when rendering images.
  - To generate an .html page, double-click the .html thumbnail (Open.html).
- Scrub through a web-compiled image of animations.
- Export files in output formats including .png and .jpg. Previously, the output format was in .jpg only.

## PhotoView 360

SOLIDWORKS 2023 SP5 is the last supported release of PhotoView 360.

## Shadow Catcher Support with Stellar Physically Correct (2023 SP2)



Stellar Physically Correct supports the shadow catcher feature and the ambient occlusion output layer.

You can:

- Simulate any geometry as a shadow catcher to receive shadows from the scene lighting.
- Render an ambient occlusion layer to boost contrast in areas including small crevices and corners.
- Use ambient occlusion for **Accurate** mode when rendering with NVIDIA® Iray®.



## Stellar Physically Correct

Stellar Physically Correct yields photo-realistic rendering results.

SOLIDWORKS Visualize supports Stellar Physically Correct, Dassault Systèmes Global Illumination renderer, using the Deep Learning AI Denoiser for optimal rendering performance. It is designed to run on high-performance devices like GPUs or multicore CPUs.

### To use Stellar Physically Correct:

1. Click **Tools > Options > 3D Viewport**.
2. Under **Render Engine**, select **Stellar Physically Correct**.

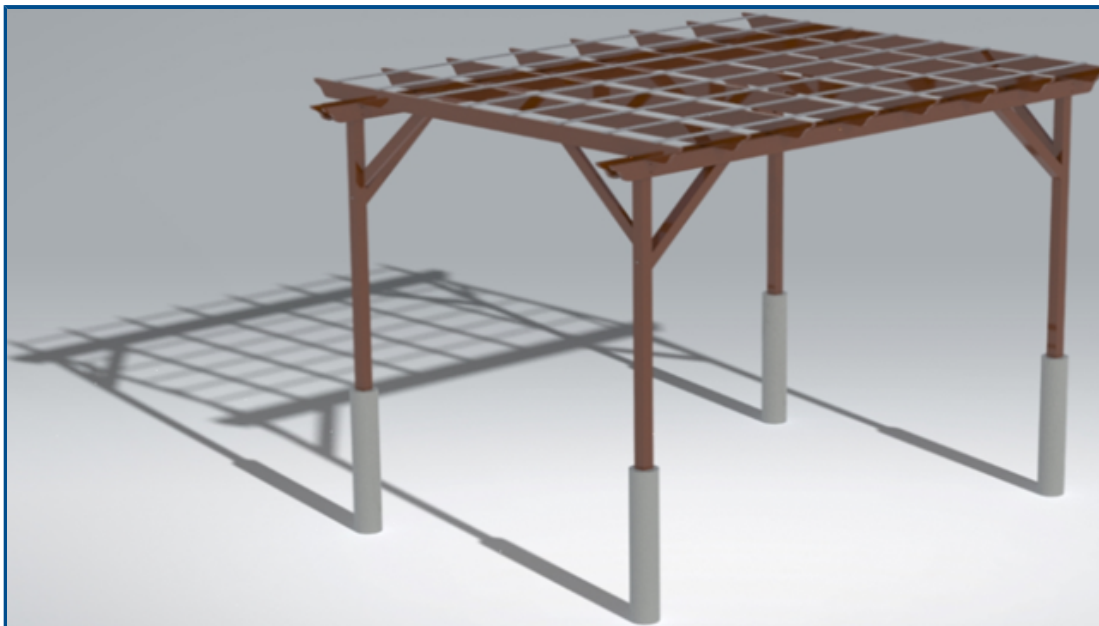
## System Info

In the System Info dialog box, you can view the requirements and hardware status of a given renderer and every GPU on your system.

To access the System Info dialog box, click **Help > System Info**.

Because SOLIDWORKS Visualize supports multiple renderers, the System Info dialog box only reports on the renderer actively running on your system. To select a renderer, click **Tools > Options > 3D Viewport**. Under **Render Engine**, select a renderer.

## Sunlight Environment

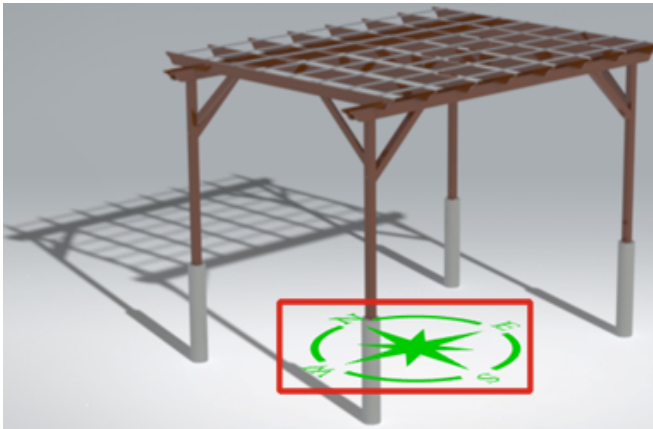



With a sunlight environment, you can stage a model under a realistic sky and light from the sun at a specific time of year and location on earth.

To create a sunlight environment, click **New Project > Scenes > New Sunlight Environment** or import sunlight data from SOLIDWORKS. In SOLIDWORKS, click **SOLIDWORKS Visualize > Export Advanced**.

You can:

- Specify the north direction using the compass manipulator.





- Specify an exact date, time, location, and time zone.
- Use the **Animation Wizard**  to animate the sunlight environment and study the design over the course of time.

## Transform Manipulator



You can use the transform manipulator to move, scale, and rotate objects in a scene. Previously, you had two separate tools to move and rotate and another to scale. The transform manipulator includes move, rotate, and scale in one tool.

**To use the transform manipulator:**

1. On the toolbar, click **Selection Tool** > **Model** .
  2. In the 3D Viewport, select the target model.
  3. On the toolbar, click **Object Manipulation Tools** > **Transform** .
- The transform manipulator appears on the selected model.
4. Use the table below to use the transform manipulator.

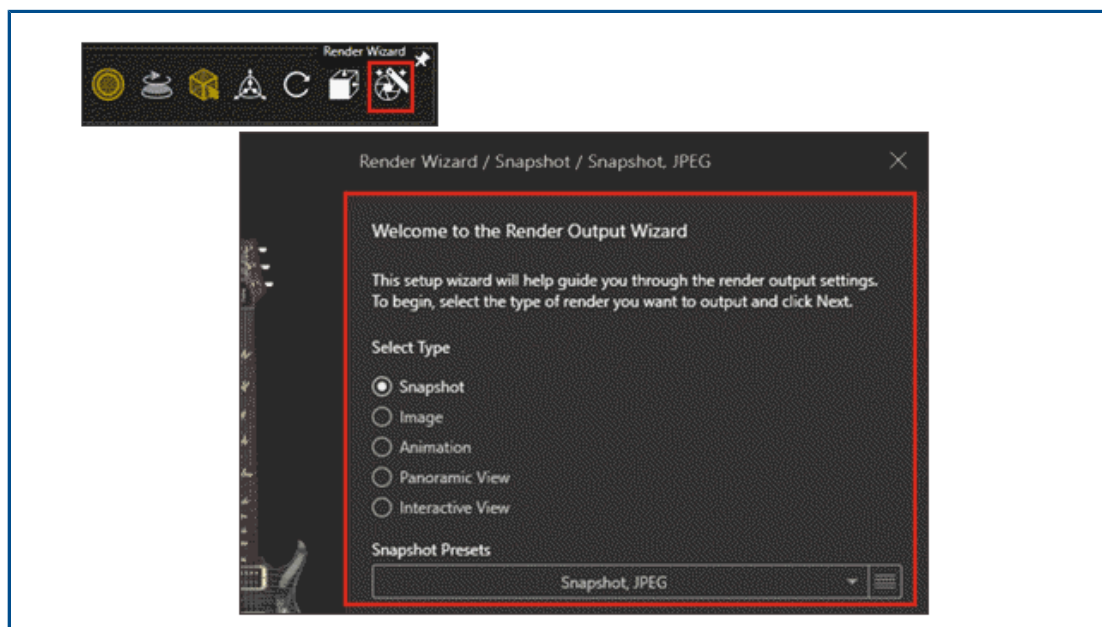
Action	Description
Move the model	Select an axis and drag to move the object along that axis. Use the green, red, or blue planes to move the objects in those planes.
Rotate the model	Select a ring around an axis and drag to rotate the object around the axis.
Scale the model	Select a red, green, or blue cube to scale the object along that axis. Use the yellow cube to scale uniformly.

When you select a handle and drag, the main gray circle and the handle you drag appears. The rest of the transform manipulator is hidden while you drag.

Keyboard shortcuts allow for quick access to functionality without using the transform manipulator.

Keyboard Shortcut	Description
<b>g</b>	Move selection in the camera plane.
<b>r</b>	Rotate selection in camera plane
<b>e</b>	Scale selection uniformly
<b>g/r/e</b> followed by <b>x</b>	Move, rotate, or scale locked to the X-axis
<b>g/r/e</b> followed by <b>y</b>	Move, rotate, or scale locked to the Y-axis
<b>g/r/e</b> followed by <b>z</b>	Move, rotate, or scale locked to the Z-axis
<b>g/r/e</b> followed by <b>Shift + x</b>	Move, rotate, or scale locked to the YZ plane
<b>g/r/e</b> followed by <b>Shift + y</b>	Move, rotate, or scale locked to the XZ plane
<b>g/r/e</b> followed by <b>Shift + z</b>	Move, rotate, or scale locked to the XY plane

## Render Wizard



The Render Output Wizard simplifies the process of specifying parameters and creating presets for rendering output. It makes it easier for you to create high quality, realistic images, videos, and interactive experiences the way you want them.

The wizard provides an easy to use interface for setting up five render types:

- Snapshot
- Images



- Animations
- 360 panoramic views for virtual reality
- Interactive views for websites

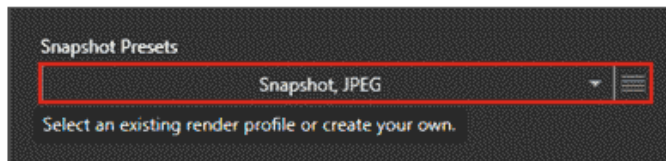
When you select a render type, the wizard steps you through all the parameters available for that type. Depending on the render type, these parameters can include settings for media (image formats), size, image quality, animation, and interactive experiences.

The wizard also makes it easy to create render presets and reuse them every time you want to generate output with the same settings.

#### To use the Render Output Wizard:



1. In the main toolbar, click **Render Wizard**.
2. In the Welcome page, select the render **Type** and select a render **Preset** from the list or create a preset using the **Smart Control Bar**.



3. Navigate through the steps presented by the wizard to adjust the parameters available for the render type.
4. Do one of the following:
  - Click **Render** to begin the rendering process.
  - Select **Queue** in the rendering options to add the current rendering to the SOLIDWORKS Visualize Queue so you can render it later.

When finished, the rendering is saved in the Images library at `SOLIDWORKS Visualize Content\Images`.



# 15

## SOLIDWORKS CAM

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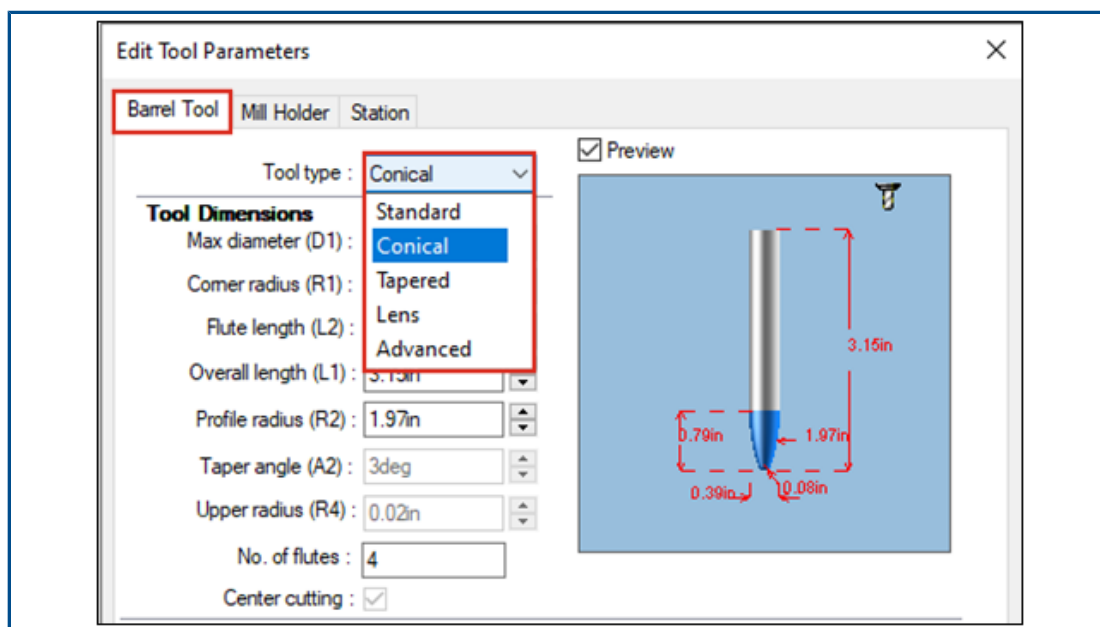
This chapter includes the following topics:

- **Barrel Tool Support**
- **Geometry Tab in Operations Parameters Dialog Box**
- **Improved Leadin and Leadout Calculations for Contour Mill Toolpaths**
- **Lines Only Option in VoluMill Toolpath**
- **Update to Machine to Mean Calculation**

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

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

### Barrel Tool Support



You can define barrel tools to generate toolpaths, mainly for multiaxis finishing operations in SOLIDWORKS CAM and the Technology Database. An advantage to using barrel tools is that they provide a larger area of contact, which results in a better surface finish.

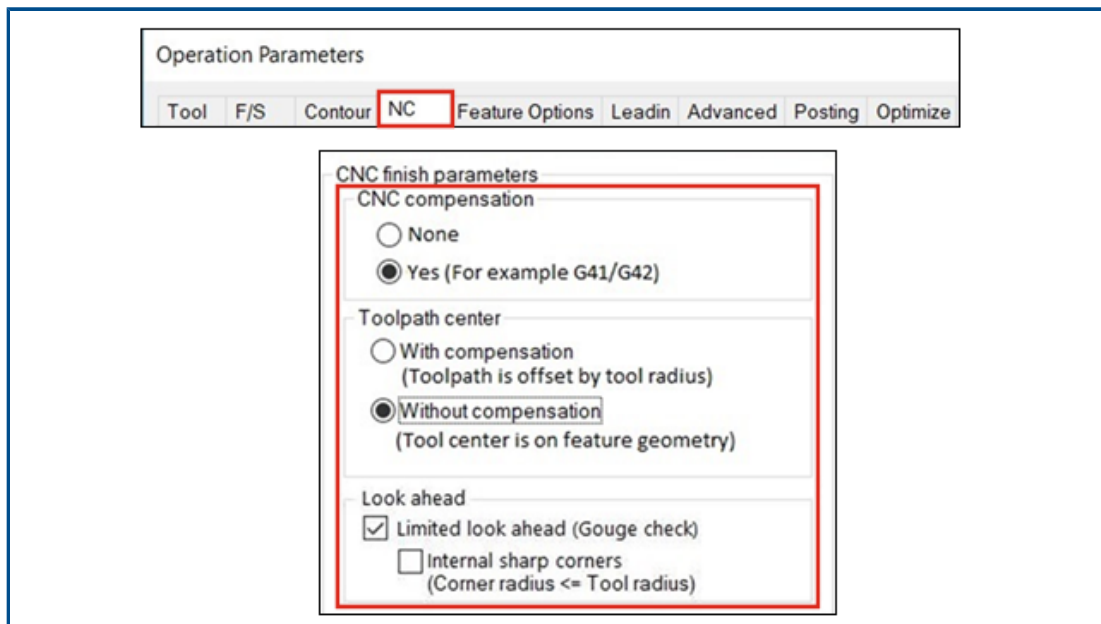
In the Operations Parameters dialog box, under the **Tool** tab, you can select **Standard**, **Conical Tangent**, **Tapered**, **Lens**, or **Advanced** as the barrel tool type. You can also define profiles to create custom barrel tools.

## Geometry Tab in Operations Parameters Dialog Box

The **Geometry** tab in the Operations Parameters dialog box lets you define and modify **Multisurface features**, **Avoid features**, and **Contain** and **Avoid areas** for 3 axis milling.

The tab also includes group boxes for **Automatic contain area** and **Avoid small profiles** formerly in the **Advanced** tab.

## Improved Leadin and Leadout Calculations for Contour Mill Toolpaths



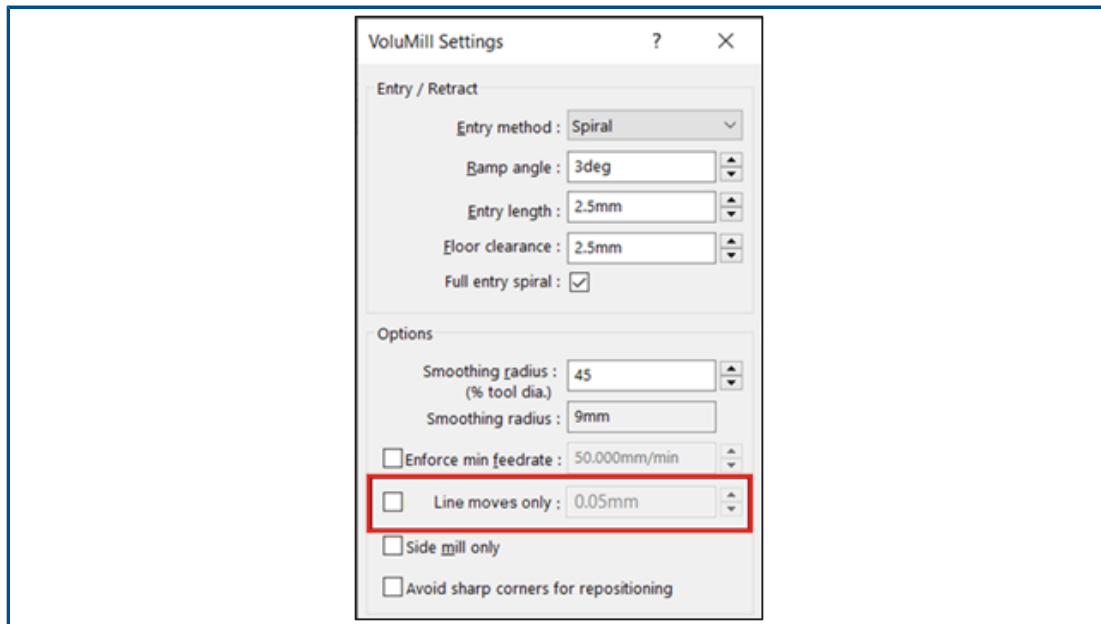
The calculation of **Leadin** and **Leadout** moves is improved so that the **XY start** and **end** points are accurately represented in a compensated toolpath.

In addition to maintaining the same **XY start** and **end** points, SOLIDWORKS CAM automatically adds the tool radius to any arc moves you specify for **Leadins** and **Leadouts**.

For example, if you use a 10mm diameter tool with an arc **Leadin** of 3mm, the calculated **Leadin** arc value is 8mm. When the toolpath is compensated on the machine, the resulting motion produces an arc move of 3mm. For other types of **Leadin** and **Leadout** moves, the length and angle of the **Leadin** and **Leadout** line segments automatically adjust.

For legacy parts, you need to regenerate contour mill toolpaths for these changes to take effect.

## Lines Only Option in VoluMill Toolpath



You can split arcs that are larger than 90 degrees into smaller lines using **Lines only**. This option is available in the VoluMill Settings dialog box in SOLIDWORKS CAM and Technology Database, and applies to 2.5 axis and 3 axis VoluMill toolpaths.

You can specify a deviation for this option. This value splits the arcs into linear moves.

Also when you use a pattern with the VoluMill module, you can select an option to **Convert rapid moves to high feed moves** in the **F/S** tab of the Operations Parameters dialog box.

## Update to Machine to Mean Calculation

SOLIDWORKS CAM tolerance-based machining only uses dimensional tolerances for the Machine to Mean value of a feature. It no longer uses positional tolerances to calculate the Machine to Mean value.

When a feature includes asymmetrical tolerances, the tolerance allowance automatically calculates to the Maximum Material Condition. Positional tolerances are not used because they can overcut or undercut a feature.

# 16

## SOLIDWORKS Composer

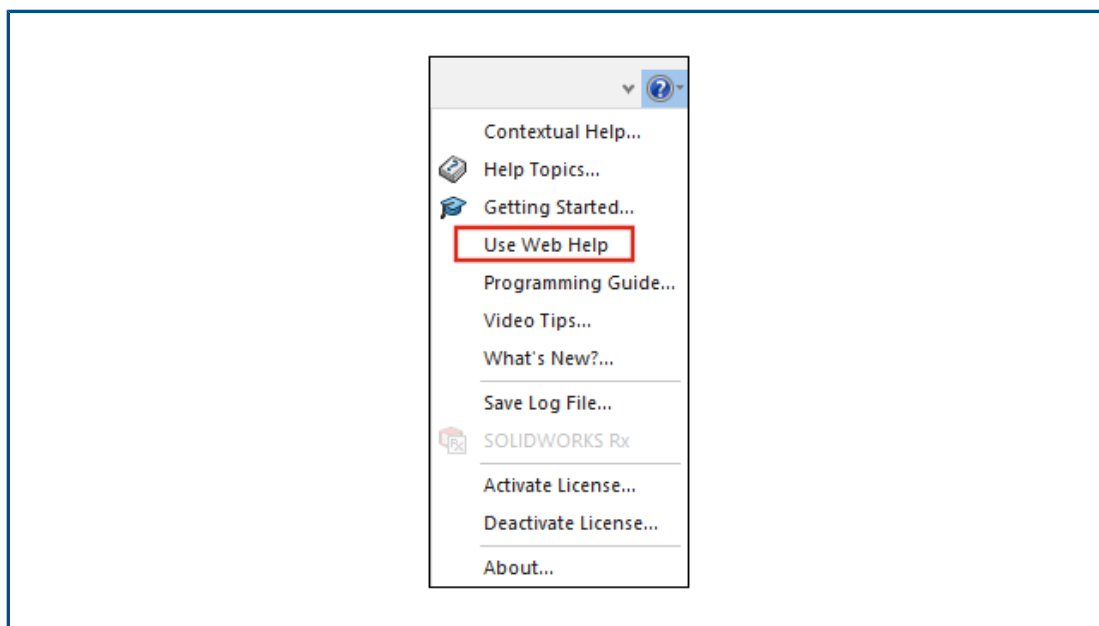
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This chapter includes the following topics:

- **Accessing Online Help for SOLIDWORKS Composer Products**
- **Creating View Collections**
- **Player Documentation No Longer Bundled in Composer Packages**
- **Data Management with ENOVIA 3DLive**
- **Composer Player 64-Bit Version**
- **Supported Import Formats**
- **Support of Meta-Properties from Pro/E Creo and STEP Files**
- **Other SOLIDWORKS Composer Enhancements**

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

### Accessing Online Help for SOLIDWORKS Composer Products

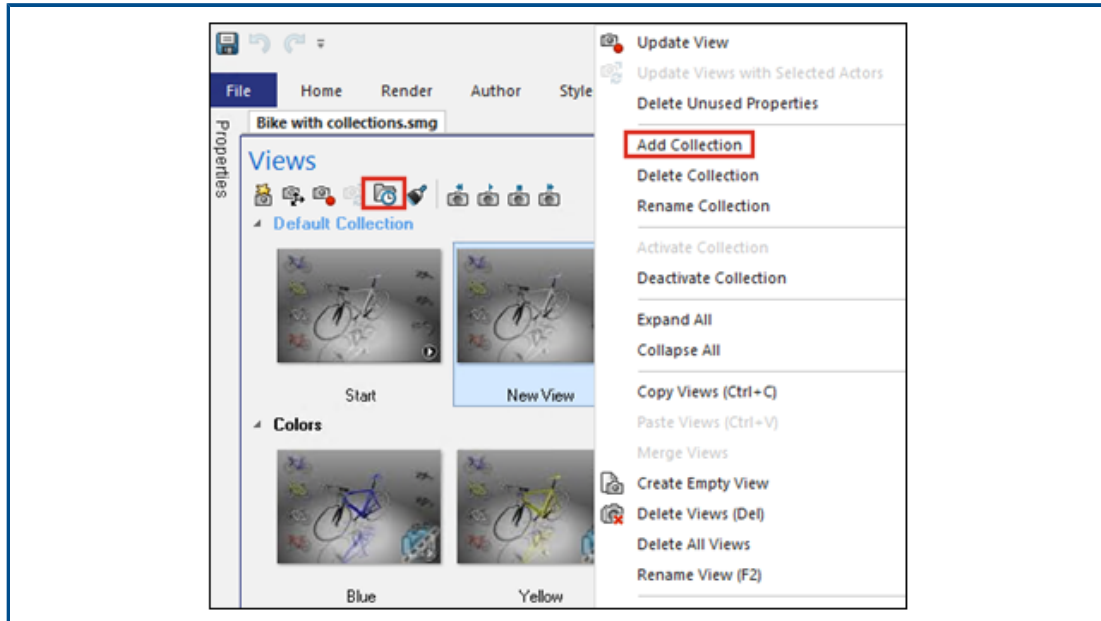


You can access the online help for all SOLIDWORKS Composer products.

Click **Help > Use Web Help** to access the online help, clear it to access the local help. Previously, only local help was available. See *Composer Help: Working with the Ribbon*

> *Help Menu, Sync Help: Menus and Commands, and Player Help: Interface Description*  
 > *Menu Bar > Help Menu.*

## Creating View Collections




You can manage and organize views into collections in the Views pane.

You can also:

- Drag views into collections
- Expand or collapse collections
- Rename collections
- Activate or deactivate collections
- Delete collection views

You can create a collection of views in the Views pane by doing the following:

- In the Views pane, click **Create View Collection** .
- Right-click a view and select **Add Collection**.

See *Composer Help: Working with Views > View Collections* and *Working with the Left Panes > Views Pane*.

## Player Documentation No Longer Bundled in Composer Packages

When you save the active Composer document as an auto-executable package (.exe file), the Player documentation is no longer bundled in the package.

You can access the Player documentation online from the Player. See *Composer Help: Importing and Opening Files > About Composer File Types, Interface Description > Dialog Boxes > Save Package As*, and *Interface Description > Dialog Boxes > Send by E-mail*.

## Data Management with ENOVIA 3DLive

You can no longer import and update models from an ENOVIA® 3DLive session.

The following commands are removed from the user interface:

- **Import from 3DLive** and **Update from 3DLive** from the **File** menu.
- **ENOVIA 3DLive > Import** and **ENOVIA 3DLive > Open in ENOVIA 3DLive** right-click commands from the Assembly pane.

## Composer Player 64-Bit Version

The **Use Composer Player 64-bit version** option is removed from the **Package** options in **(Default) Document Properties > Output**.

See *Composer Help: Managing (Default) Document Properties > Output*.

## Supported Import Formats

SOLIDWORKS Composer and SOLIDWORKS Composer Sync support higher versions of several import formats.

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

## Support of Meta-Properties from Pro/E Creo and STEP Files

You can import meta-properties from Pro/E Creo (\*.prt, \*.asm), and STEP (\*.stp, \*.step) files in SOLIDWORKS Composer and SOLIDWORKS Composer Sync.

Meta-properties are user-defined attributes such as part number (P/N), designation, and revision number. They appear as user properties of the string type in the Properties pane. See *Composer Help: Managing (Default) Document Properties > Input* and *Sync Help: Managing Default Document Properties > Input*.

## Other SOLIDWORKS Composer Enhancements

SOLIDWORKS Composer products have additional enhancements.

- In the Technical Illustration workshop, you can no longer select **SVGZ** as the format for saving vector images. The **SVG** format is still available. If the **SVG** file is too large, you can convert it into the **SVGZ** compressed format using a standard file compression tool. See *Composer Help: Working in Workshops > Technical Illustration Workshop*.
- On the **Publish** page, you can no longer select **SVGZ** as the output format for vector images. The **SVG** format is still available. If the **SVG** file is too large, you can convert it into the **SVGZ** compressed format using a standard file compression tool. See *Sync Help: Configuring Conversion Properties > Publish*.
- In SOLIDWORKS Composer Player, you can see views grouped into collections in the Views pane.

## SOLIDWORKS Electrical

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This chapter includes the following topics:

- **Defining Open-Ended Wires for Routes**
- **Propagating Electrical Properties from 2D to 3D**
- **Associate Feature Improvements**
- **Electrical Manager Tree Improvements**
- **Dynamic Connection Labels**
- **Origin-Destination Symbol**
- **Displaying Column Header Text in All Electrical Project Languages**
- **Manufacturer Part Weight**
- **Automatic Orientation in PDF Export**
- **Deleting Components**
- **Additional Variables in Origin-Destination Formula**
- **Resetting Dismissed Messages**
- **Multilingual Attributes**
- **Microsoft Excel Automation Improvements**
- **Embedding Report Tables**
- **Additional File Formats for Images**
- **Removing Assembly Link While Duplicating Projects (2023 SP3)**
- **Display Warning While Creating a Graph (2023 SP3)**
- **Route Active Location Only (2023 SP3)**

SOLIDWORKS® Electrical is a separately purchased product.



## Video: What's New in SOLIDWORKS 2023 - SOLIDWORKS Electrical

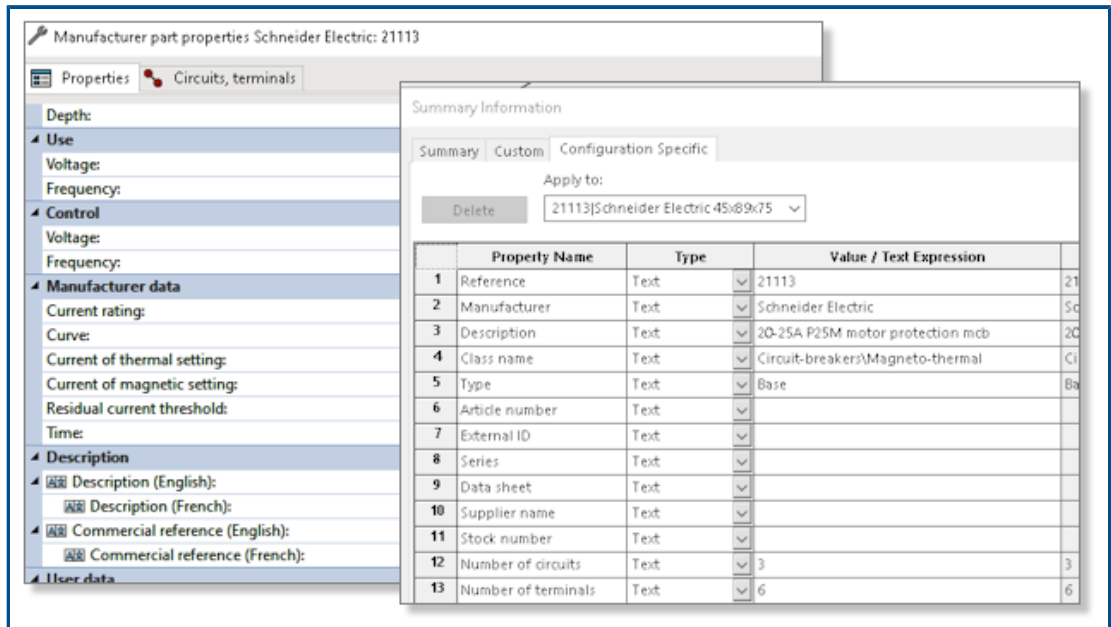
### Defining Open-Ended Wires for Routes



Open-ended wires have one extremity that is disconnected from a component or terminal. The open-ended wire property automatically propagates to the From-To list for routes.



## Propagating Electrical Properties from 2D to 3D

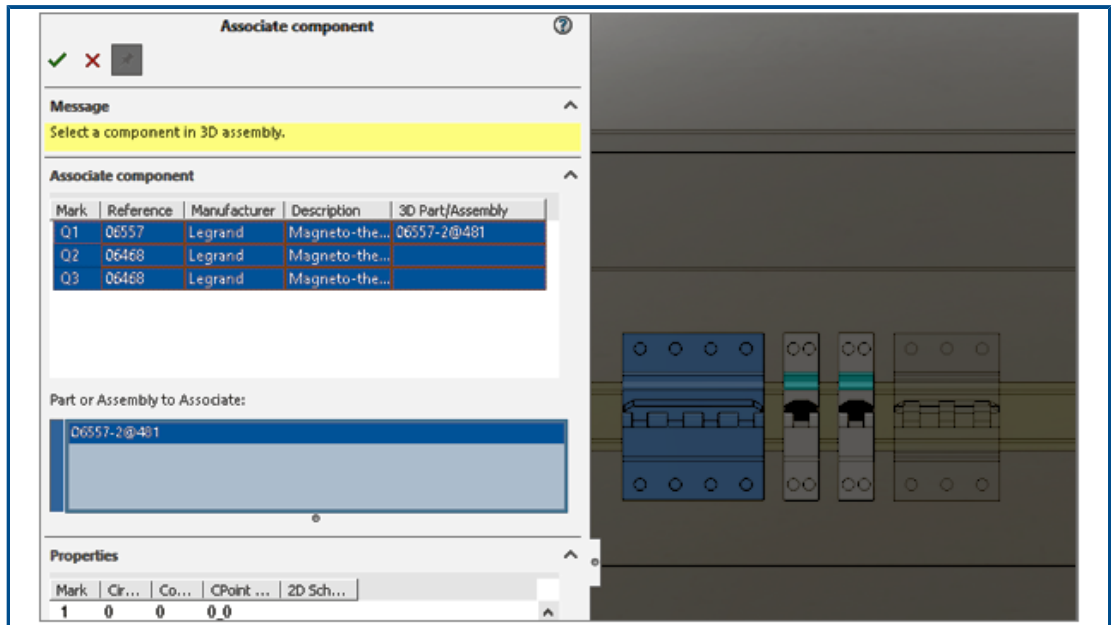


For each component, you can propagate the electrical properties to the 3D part or assembly.

If you modify the electrical properties in the 2D application, you must click **Update BOM Properties** in the SOLIDWORKS Electrical menu, to update the 3D part properties.

You can use these properties to complete tables like the Bill of Materials.

## Associate Feature Improvements

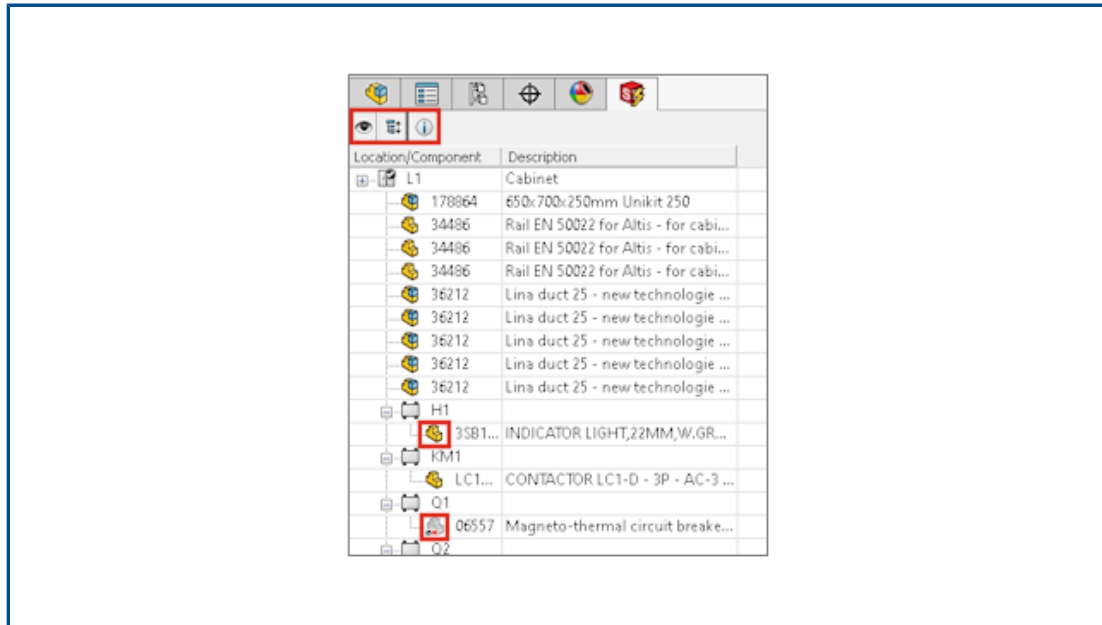


In SOLIDWORKS Electrical 3D, the **Associate** command lets you associate electrical components with 3D parts or assemblies.

The **Associate Component** PropertyManager lets you:

- Select several components to associate them with several 3D parts or assemblies.
- Display the properties of the selected component to check the consistency with the 3D part or the assembly.

## Electrical Manager Tree Improvements



In SOLIDWORKS Electrical 3D, the **Electrical Manager** tree displays the list of components in the electrical project.

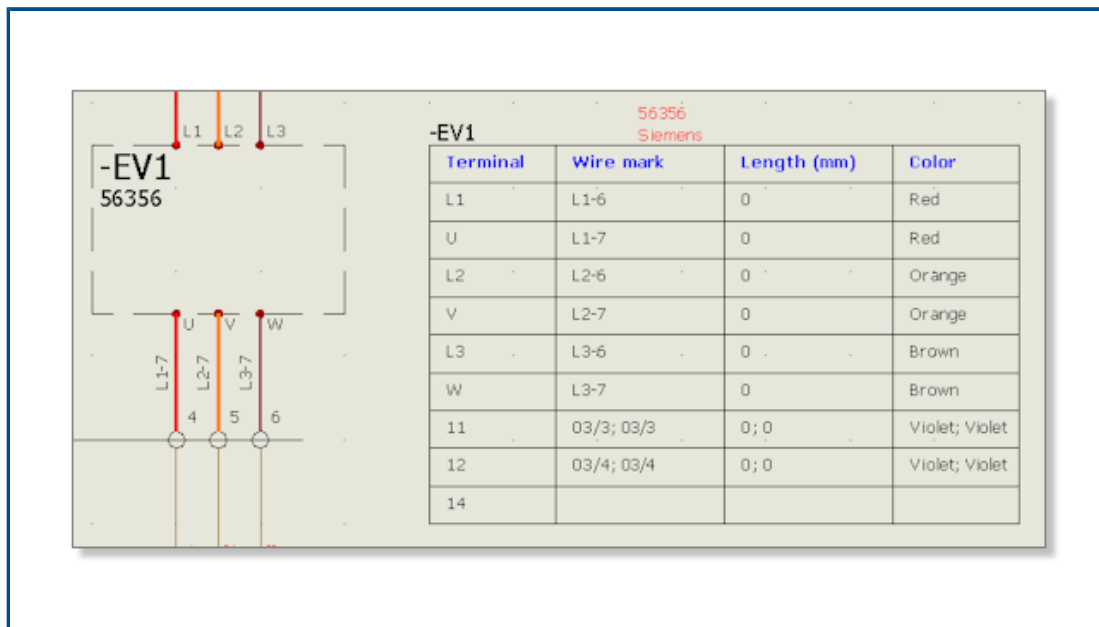
In the **Electrical Manager** tree, an icon before each component indicates whether the component is associated or not associated with an assembly or a 3D part. Selecting a component in the tree highlights the associated assembly or 3D part in the graphical area.

Selecting the location from the **Electrical Manager** highlights the assembly in the **Feature Manager** tree if the corresponding assembly is inserted in the main assembly.

The **Electrical Manager** lets you:

- Show or hide inserted components
- Expand or collapse all tree nodes

## Dynamic Connection Labels

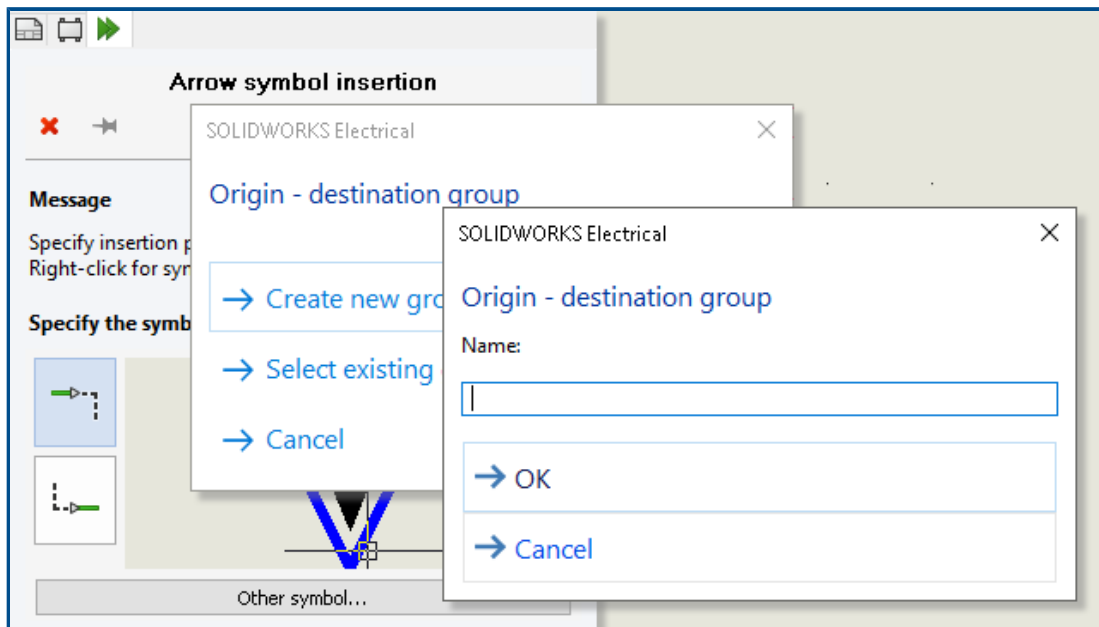


Dynamic connection labels are specific symbols that display connection information about a component. You can dynamically insert the symbols into schemes.

The dynamic connection labels use configuration files where you enter the parameters to draw them. Click **Electrical project > Configurations > Connection label** to manage the configuration files.

To insert dynamic connection labels, click **Schematic > Connection label > Insert a dynamic connection label for a component**.

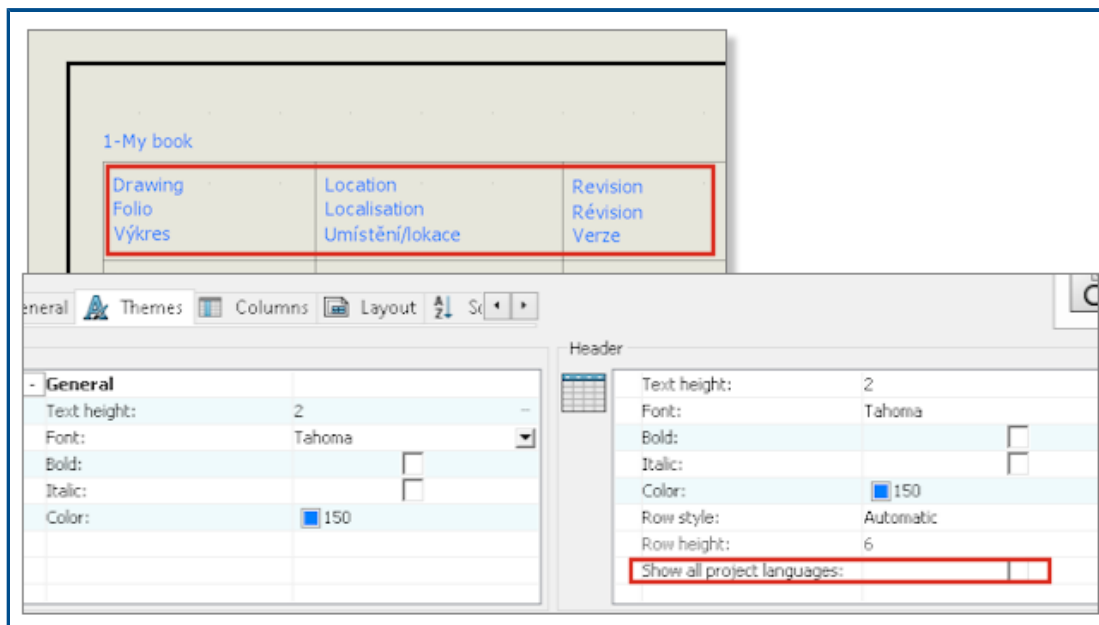
## Origin-Destination Symbol



You can manually insert origin-destination arrow symbols at the end of the wires. You associate a group number with each inserted symbol. All the wires connected on the same group number have the same equipotential number.

To insert an origin-destination symbol, click **Schematic > Origin – destination arrows > Insert a single origin - destination arrow symbol**.

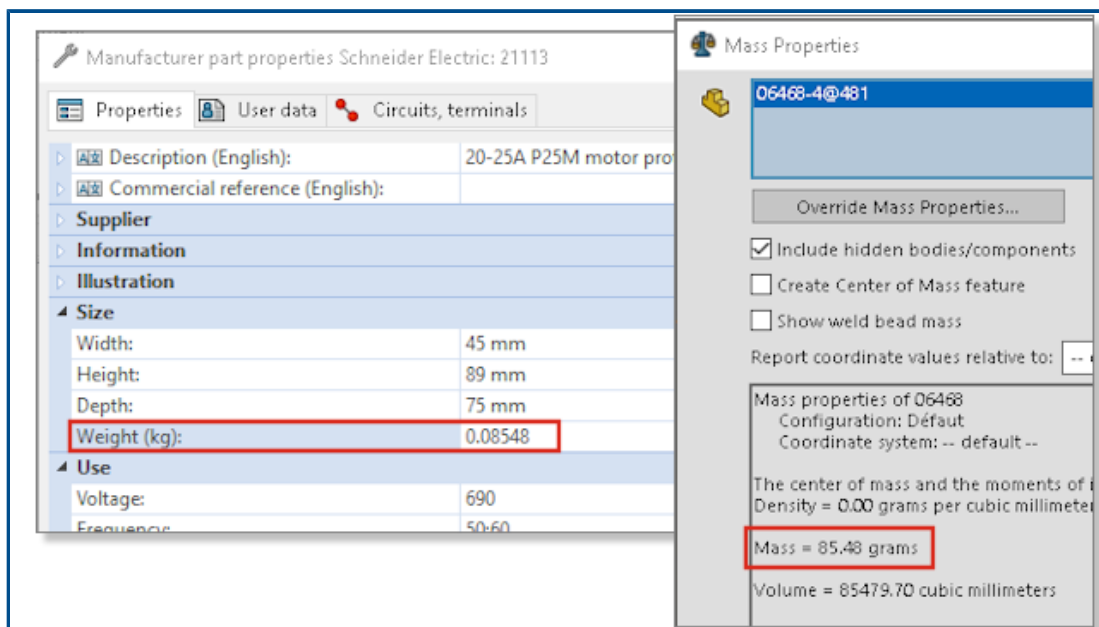
## Displaying Column Header Text in All Electrical Project Languages



In the report, you can display column header text in all three languages specified in the electrical project configuration.

To activate this feature, in the report configuration, select **Show all project languages**.

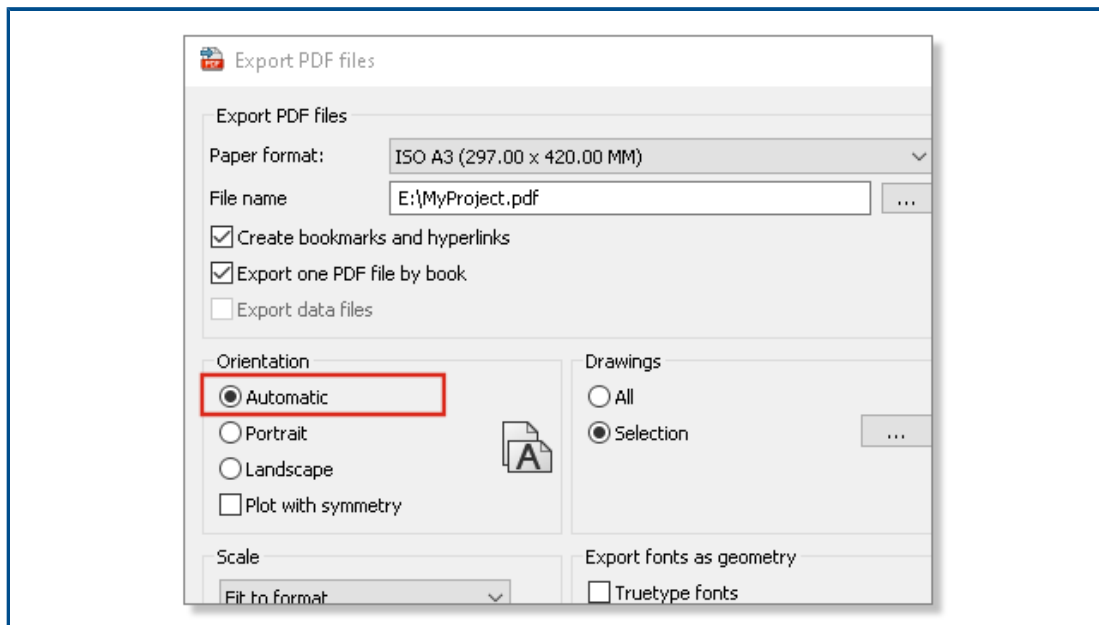
## Manufacturer Part Weight



You can manage the weight of a manufacturer part.

The weight propagates to the **Mass properties** of the 3D assembly associated with the component on which you assign the manufacturer part. For components that are cut, like rails or ducts, the weight updates according to the **Depth** value that you use in the cabinet layout.

## Automatic Orientation in PDF Export

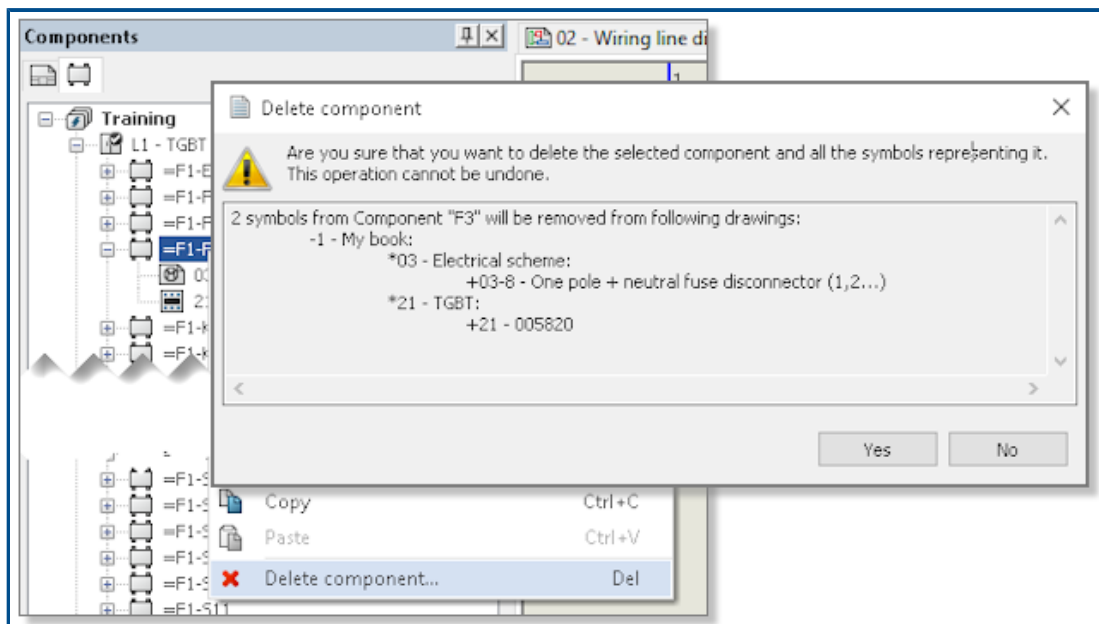


When you export the electrical project drawings to a PDF file, you can keep the same page orientation.

Drawings in **Landscape** orientation convert to PDF pages in **Landscape** orientation, and drawings in **Portrait** convert to PDF pages in **Portrait** orientation.

Select the **Automatic** option, available in **Import/Export > Export PDF files**.

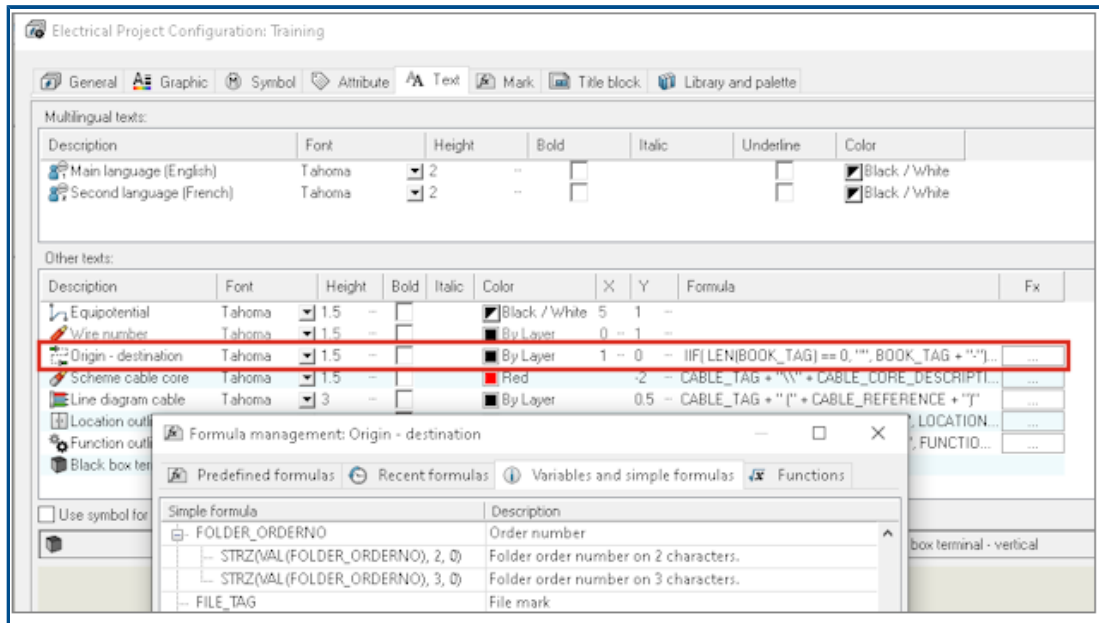
## Deleting Components



In the **Components** dockable panel, you can delete a component and all the symbols representing it in the drawings.

A component without inserted symbols is automatically deleted. If you want to delete a component with inserted symbols in the drawings, a dialog box opens, lets you select the option to delete all the symbols associated with this component. If the component is associated with a 3D part or assembly, it is automatically dissociated.

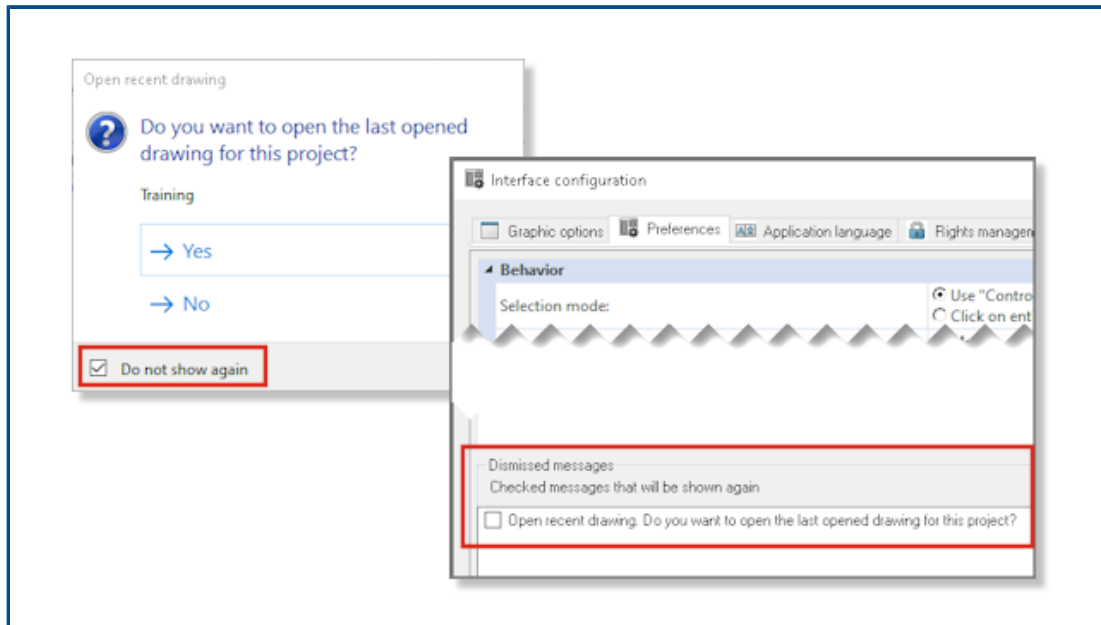
## Additional Variables in Origin-Destination Formula



New variables for components and wires are available in the Formula Management dialog box.

To use these variables, open the electrical project configuration, and in the **Text** tab, click the **FX** cell for **Origin - destination**.

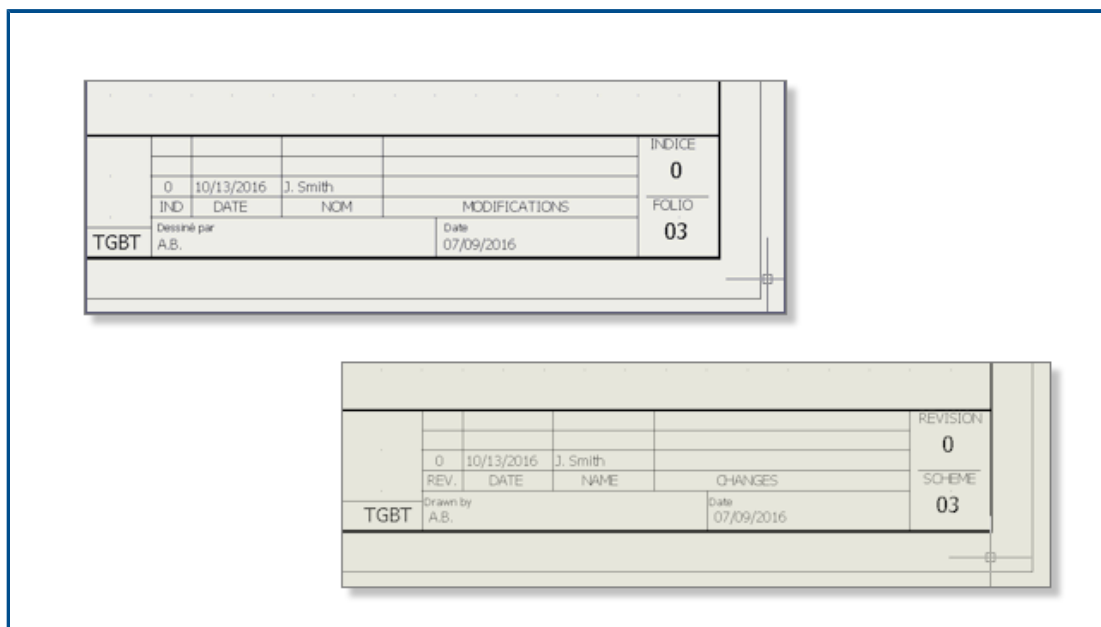
## Resetting Dismissed Messages



A new option lets you manage dismissed messages.

Some dialog boxes have the **Do not show again** option to permanently dismiss them. To reset this status, in the **Interface configuration** dialog box, in the **Preferences** tab, select the dialog boxes to show again.

## Multilingual Attributes

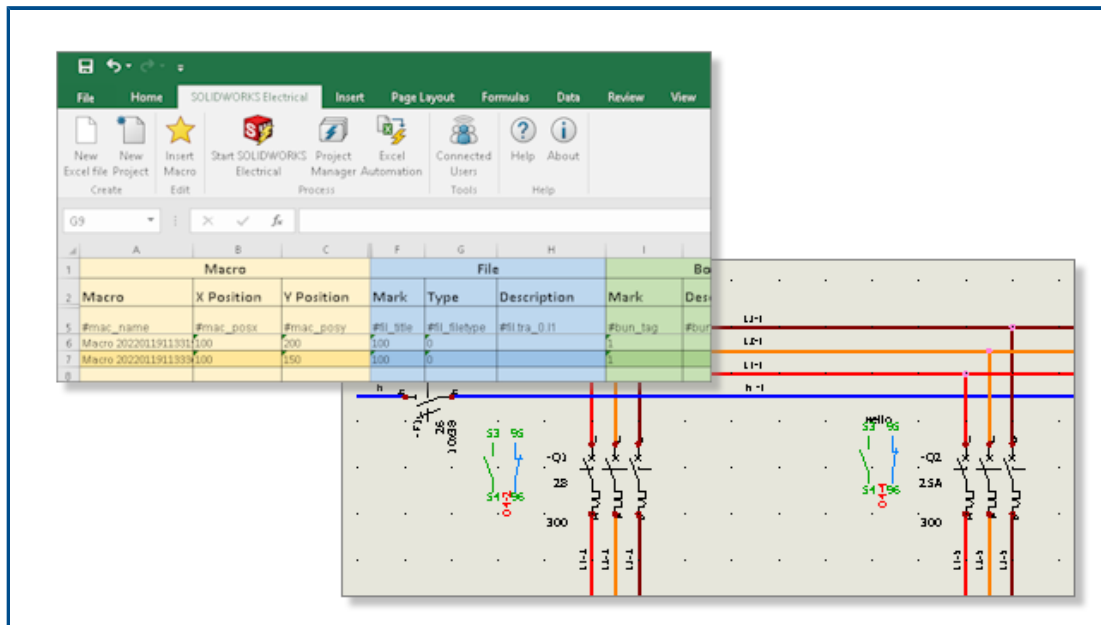




The multilingual attributes are specific attributes that propagate a multilingual text. This multilingual text displays the desired language when you change the main language of the electrical project.

You can use this type of attribute in title blocks or specific symbols like wiring cable labels. You manage multilingual attributes in the title block editor or the symbol.

## Microsoft Excel Automation Improvements

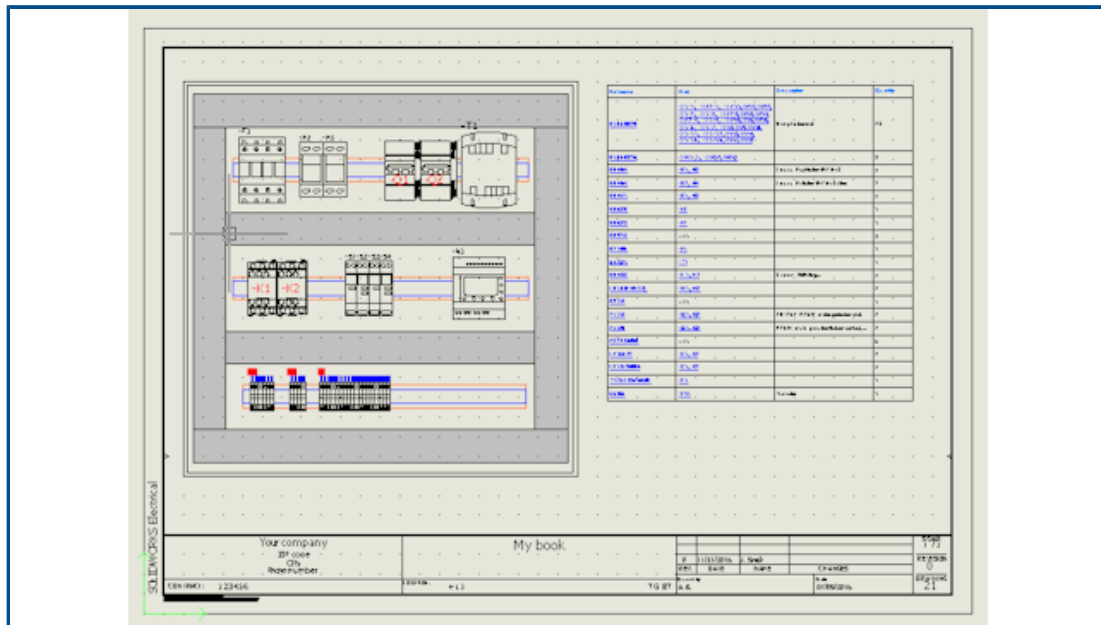


**Excel automation** allows you to automatically generate schematic drawings from data contained in a Microsoft® Excel® file.

With the **Excel automation**, you can:

- Manage the origin and destination arrows
- Replace a symbol while inserting a macro
- Select the folder for the generated drawings

## Embedding Report Tables

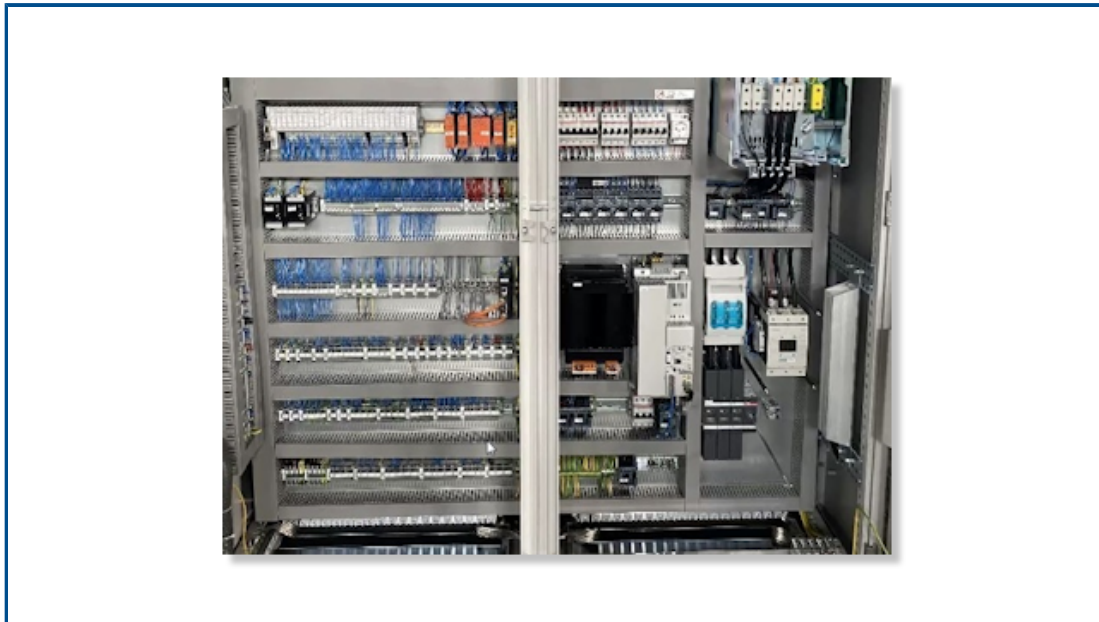


You can insert report tables anywhere into electrical project drawings.

According to configuration parameters, you can automatically insert specific report tables into graphical drawings like schematic, wiring line diagram, or 2D cabinet layout schemes.

After changes of the electrical project data, you can update the report content.

## Additional File Formats for Images

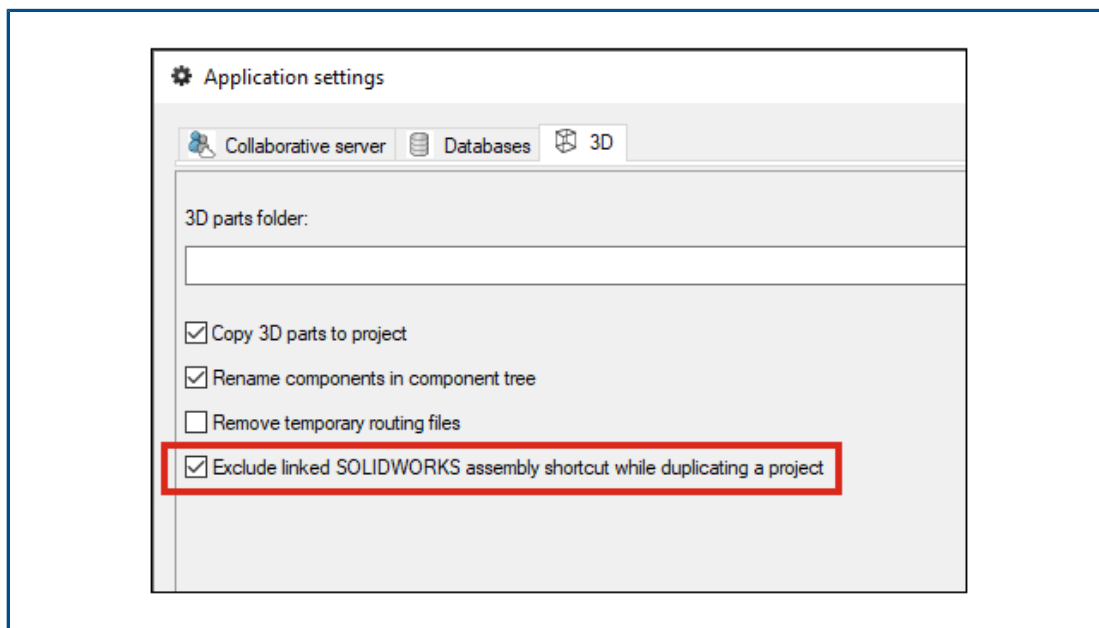


You can use several file formats to insert or attach images to drawings.

The supported file formats are:

- Bitmap images (\*.bmp)
- DIB images (\*.dib)
- JPEG File Interchange Format (\*.jpg)
- Portable Network Graphics (\*.png)
- Tagged Image File Format (\*.tiff)
- Graphics Interchange Format (\*.gif)

## Removing Assembly Link While Duplicating Projects (2023 SP3)



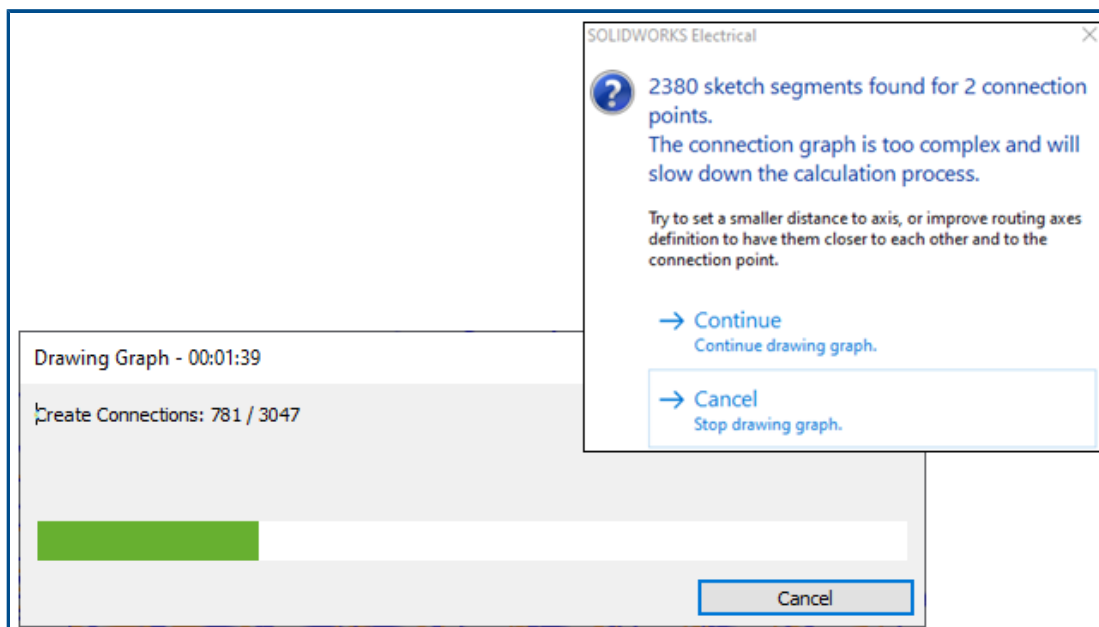
When you duplicate a project, you can remove the link of the SOLIDWORKS assembly to its original path.

To remove the link, from **Tools > Application settings > 3D**, select **Exclude linked SOLIDWORKS assembly shortcut while duplicating a project**.

To relink an existing assembly file inside a project to another already existing SOLIDWORKS assembly, you can browse and select the new SOLIDWORKS assembly from the Creation of assembly files dialog box. SOLIDWORKS Electrical retains the association of the 3D components after you select an existing assembly file from the required location if the same 3D part files are found. Previously, it was impossible to relink the existing SOLIDWORKS assembly file in a duplicated project to a new SOLIDWORKS assembly.

This is available only for the SOLIDWORKS Electrical 3D add-in.

## Display Warning While Creating a Graph (2023 SP3)

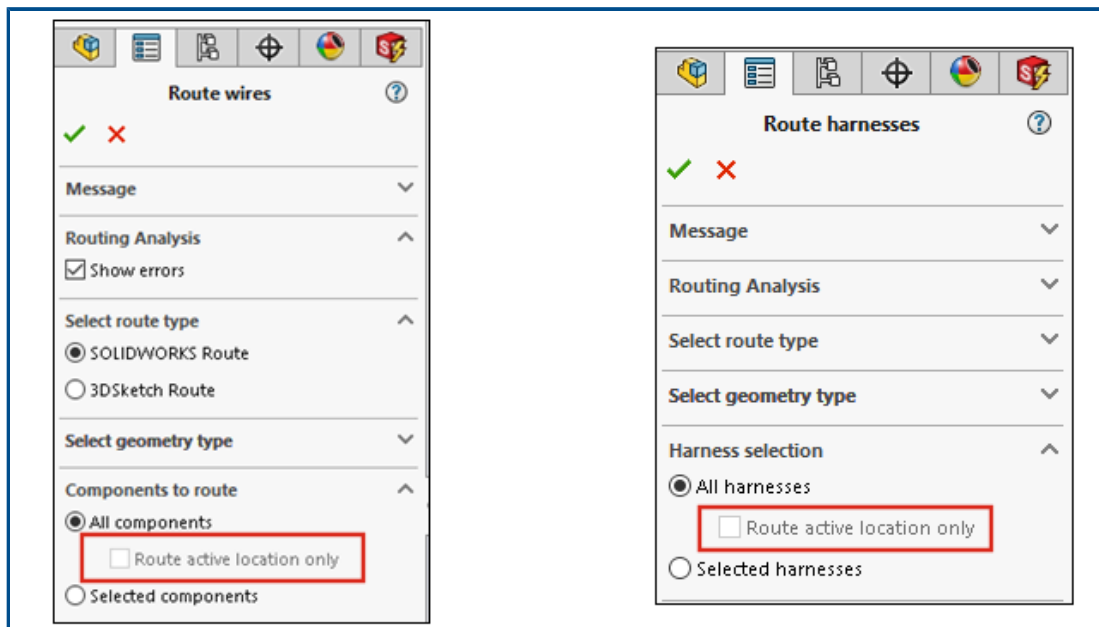


When you draw graphs, dialog boxes and warning messages appear to guide you.

When you draw a graph, a progress dialog box with a **Cancel** button appears. The progress dialog box reports the steps involved in the graph creation process. To end the command without closing the SOLIDWORKS session, click **Cancel**.

When you use **Draw Graph**, if the graph is for a very complex assembly, a warning message appears and prompts you to continue drawing the graph or cancel the operation.

## Route Active Location Only (2023 SP3)



You can define the depth of the routing process by limiting it to the active location instead of repeatedly rebuilding all the sublevels (sublocations) routes.

To limit the routing to the top-level assembly for wires, cables, or harness, do the following:

- In the Route Wires PropertyManager, under **Components to route** > **All components**, select **Route active location only**.
- In the Route Cables PropertyManager, under **Cables to route** > **All cables**, select **Route active location only**.
- In the Route Harnesses PropertyManager, under **Harness selection** > **All harnesses**, select **Route active location only**.

This capability reduces the overall design time spent while working on large electrical projects with nested locations. You can limit the routing process to the active or top location only instead of routing or updating the sublocations each time you execute the route command on the top-level assembly.

# 18

## SOLIDWORKS Inspection

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This chapter includes the following topics:

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

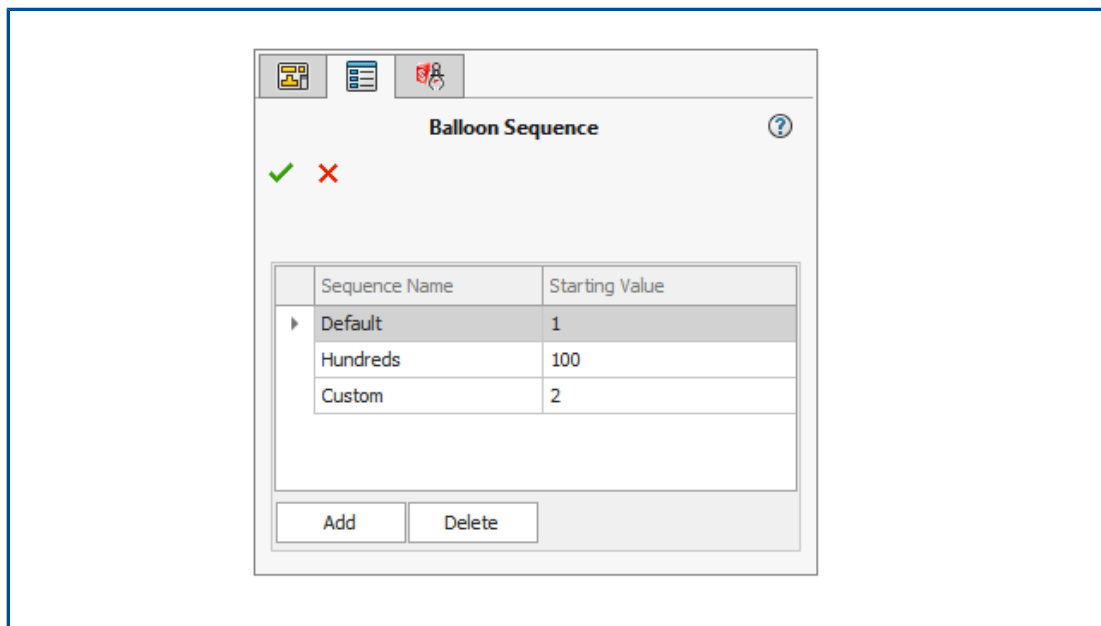
SOLIDWORKS® Inspection is a separately purchased product that you can use with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium, or as a completely separate application (see *SOLIDWORKS Inspection Standalone*).




## Video: What's New in SOLIDWORKS 2023 - SOLIDWORKS Inspection

## SOLIDWORKS Inspection Add-in


### Balloon Sequence



You can use the **Balloon Sequence** tool to define the balloon numbering sequence in multisheet drawings.

In the SOLIDWORKS Inspection add-in, click **Balloon Sequence**  (SOLIDWORKS Inspection toolbar) and specify options in the PropertyManager.

Balloon Sequence PropertyManager

To open this PropertyManager, in the SOLIDWORKS Inspection add-in, click **Balloon Sequence**  (SOLIDWORKS Inspection toolbar).

### Sequence Name

Specifies the different balloon sequences to define. You can change the name of the sequences.

### Starting Value

Specifies the starting numeric value (whole numbers only) of each balloon sequence.

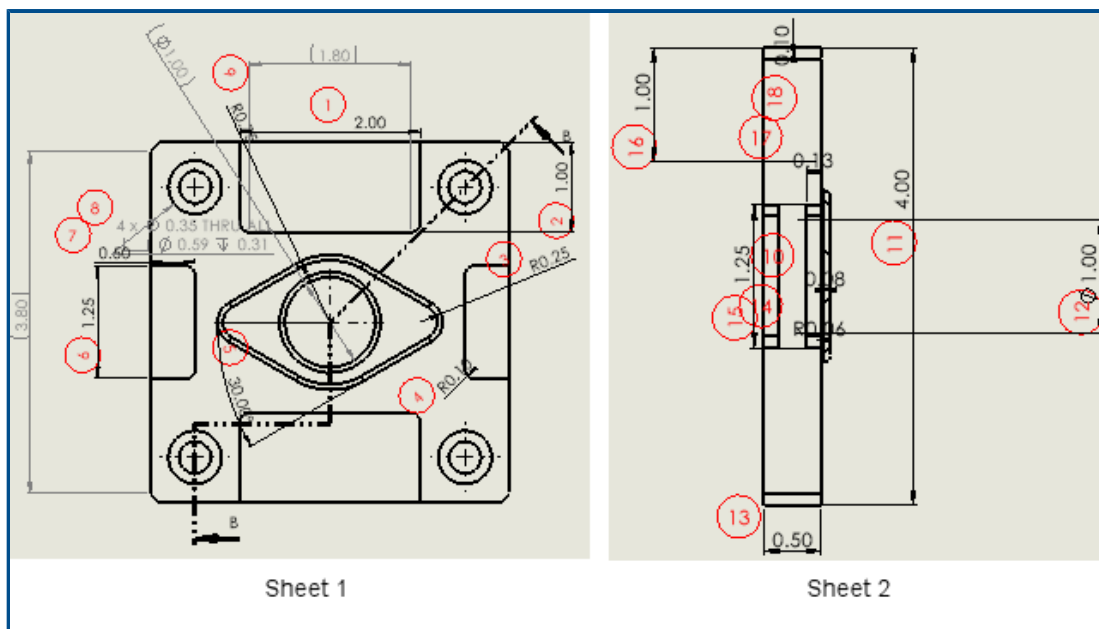
## Add

Adds a row to the balloon sequence table.

## Delete

Removes a selected row from the balloon sequence table.

## Balloon Drawing Sheets Separately



In multisheet drawings, you can specify sheets to balloon instead of ballooning all sheets in a drawing.

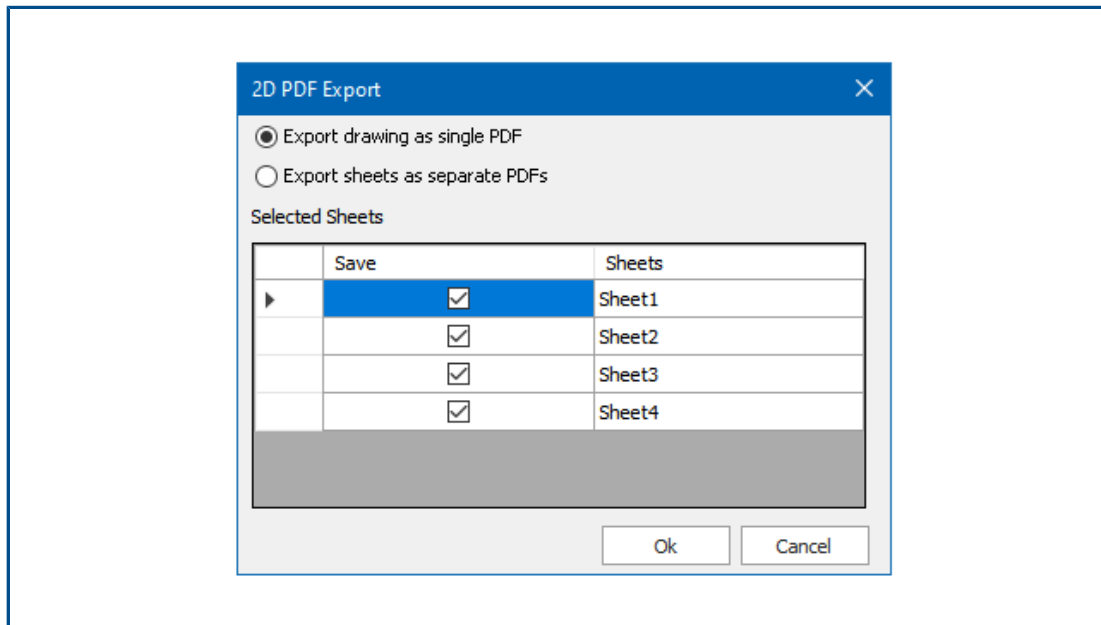
In the Create Inspection Project PropertyManager, under **Sheets**, select **Include All** or you can specify sheets. With **Include All**, the entire drawing is ballooned. When you select sheets, you can specify each sheet in the drawing to balloon in inspection reports. You can also specify the balloon sequence. You define the balloon sequence with the

**Balloon Sequence**  tool or by entering it in **Sequence/Starting Value**.


If you have two or more sheets to balloon continuously, in **Sequence/Starting Value**, select **Continue from previous sheet**.



## Export Sheets as Separate 2D PDF Files

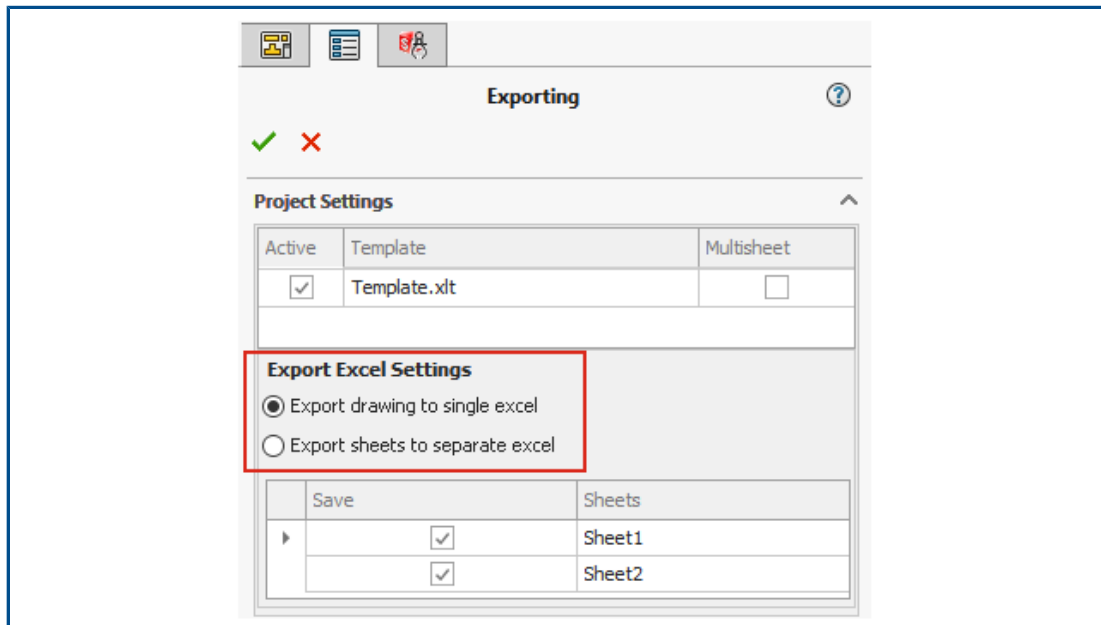


If you balloon drawing sheets separately, you can export the Microsoft® Excel® and .pdf reports separately. Previously, the reports exported as a continuous report with all sheets included.

In the SOLIDWORKS Inspection add-in, click **Export to 2D PDF**  (SOLIDWORKS Inspection toolbar). In the 2D PDF Export dialog box, specify options:

- **Export drawing as single PDF.** Exports the entire drawing as a single .pdf file.
- **Export sheets as separate PDFs.** Exports each drawing sheet as a separate .pdf file.
- **Selected Sheets.** Specifies the drawing sheets to save in the .pdf file.

## Export FAI Reports to Separate Microsoft Excel Documents

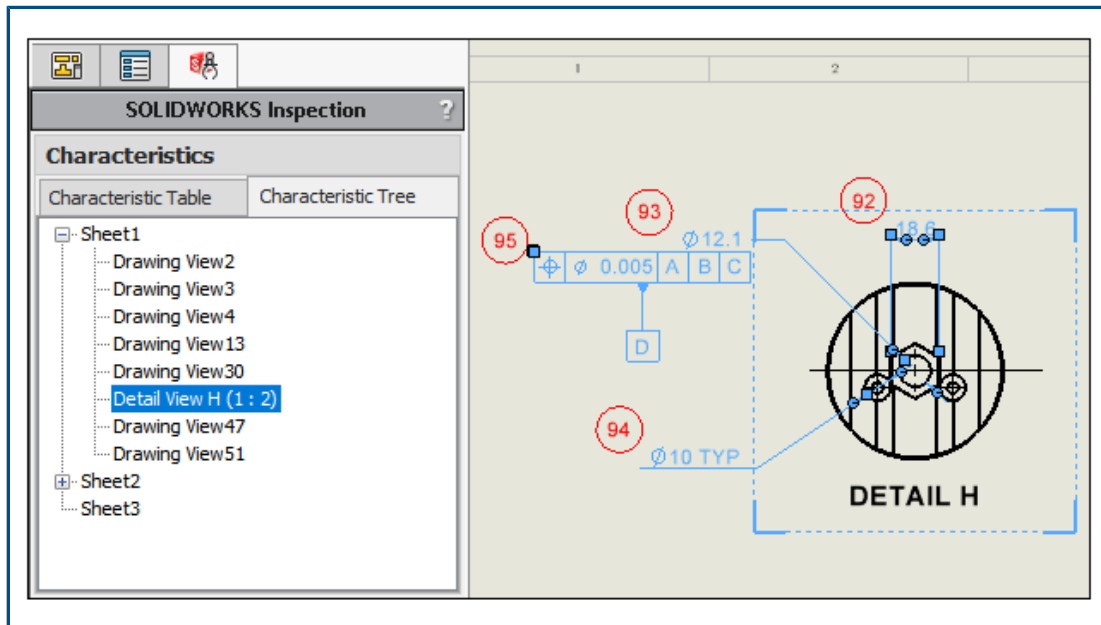


When you export a First Article Inspection (FAI) report to Microsoft® Excel®, you can export characteristics to multiple spreadsheets.

In the SOLIDWORKS Inspection add-in, click **Export to Excel** . In the Exporting PropertyManager, under **Excel Export Settings**, specify options:

- **Export drawing to single excel.** Exports all characteristics from the specified sheets to one Microsoft Excel spreadsheet.
- **Export sheets to separate excel.** Exports all characteristics from the specified sheets to separate Microsoft Excel spreadsheets.

## SOLIDWORKS Inspection Manager



The SOLIDWORKS Inspection Manager contains a Characteristics section with tabs for Characteristic Table and Characteristic Tree. If you select an item in the characteristic table or characteristic tree, the corresponding entity highlights in the graphics area.

The characteristic table is existing functionality that contains the Bill of Characteristics, which lists each ballooned entity. You can select multiple balloons from the same view and reorder them in the characteristic table. When you drag a balloon to reorder it in the table, scrolling speed and usability is improved. The **View** column in the table lists the view to which each characteristic belongs.

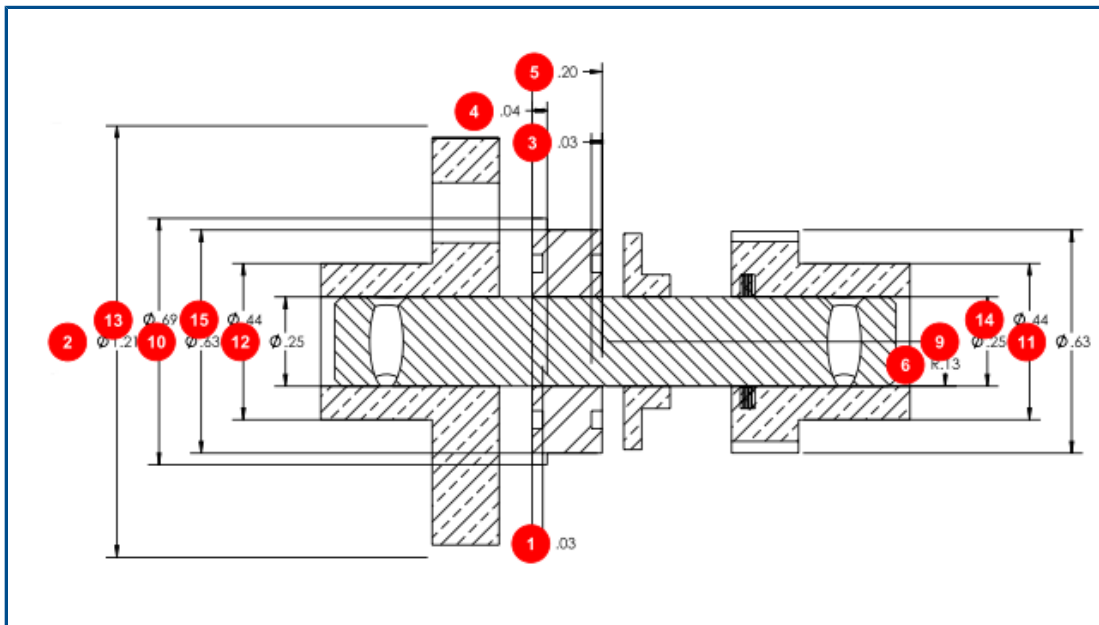
The characteristic tree contains information for each sheet and view in the drawing. It only displays sheets and views that have ballooned items. In the characteristic tree, you can reorder:

- Balloons by dragging views.
- Views that belong to the same sheet.
- Sheets. The sheet and its views are reordered and renumbered according to the new position in the characteristic tree.

## SOLIDWORKS Inspection Standalone

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


## Autoballoons for PDF Files



The **Auto Extract** tool automatically applies balloons to characteristics in PDF files, which reduces the time and effort needed to balloon PDF files.





**Auto Extract** uses a machine learning technique to locate the positions and types of annotations, then attaches balloons to them. You can also use this tool to automatically apply balloons to CAD files. The **Auto Extract** tool replaces the **Smart Extract** tool.

In a PDF file, on the Document ribbon, click **Auto Extract** . **Auto Extract** does not support notes or weld symbols.

You can also manually extract characteristics with the **Manual Extract**  tool (Document ribbon).


### Autoballooning PDF Files

#### To autoballoon PDF files:

1. In SOLIDWORKS Inspection Standalone, click **New** .
2. In the Project Template dialog box, select a template and click .
3. In the Open CAD/PDF File dialog box, select a PDF file and click **Open**.
4. Click **Auto Extract**  (Document ribbon).
5. Specify settings in the Auto Extract PropertyManager and click .

## Auto Extract PropertyManager

You can use the **Auto Extract** PropertyManager to automatically apply balloons to characteristics in CAD models, drawings, and PDF files.

To open the Auto Extract PropertyManager, in a model, drawing, or PDF file, click **Auto Extract** .

## Extraction Settings

(PDF files only.) Specifies the characteristics to balloon. Notes and weld symbols are not supported.

<b>Include All</b>	Balloons all dimensions, geometric tolerance symbols, surface finish symbols, and datums in the file.
<b>Dimensions</b>	Balloons all dimensions.
<b>GD&amp;Ts</b>	Balloons all geometric tolerance symbols.
<b>Surface Finishes</b>	Balloons all surface finish symbols.
<b>Datum</b>	Balloons all datums.

## Dimensions

(CAD models and drawings only.)

<b>Include</b>	Balloons all basic, feature, and reference dimensions. Clear <b>Include</b> to select individual <b>Basic</b> , <b>Feature</b> , or <b>Reference</b> dimensions.
<b>Inspection Only</b>	Balloons all inspection dimensions.
<b>Auto-explode hole callouts</b>	(SOLIDWORKS files only.) Balloons and splits compound hole callouts into subballoons.

## Notes

(CAD models and drawings only.)

**Include**

Balloons all datum feature, datum target, and flag note symbols.  
Clear **Include** to select individual **Datums**, **Datum Targets**, **Flag Notes**, **Correlate Flag Notes**, or **Auto-explode multi-line notes**.

## GD&amp;Ts

(CAD models and drawings only.)

**Include**

Balloons all geometric dimensioning and tolerancing feature control frames.

## Welds

(CAD models and drawings only.)

**Include**

Balloons all weld symbols.

## Surface Finishes

(CAD models and drawings only.)

**Include**

Balloons all surface finish symbols.

## Geometry Options

(CAD models and drawings only.)

**Load Invisible Components**

Loads hidden components into SOLIDWORKS Inspection.

## Page 2 of PropertyManager

Click **Next** ► in the PropertyManager to access these options.

**Entire Drawing**

(Drawings and PDF files only.) Balloons all characteristics in the document.

**Sheets, Include All, sheet<sub>n</sub>**

## Entire Document

(CAD models only.)

<b>Entire Document</b>	Balloons all characteristics in the document.
<b>Include All</b>	Includes all annotation views, otherwise you can select individual annotation views.

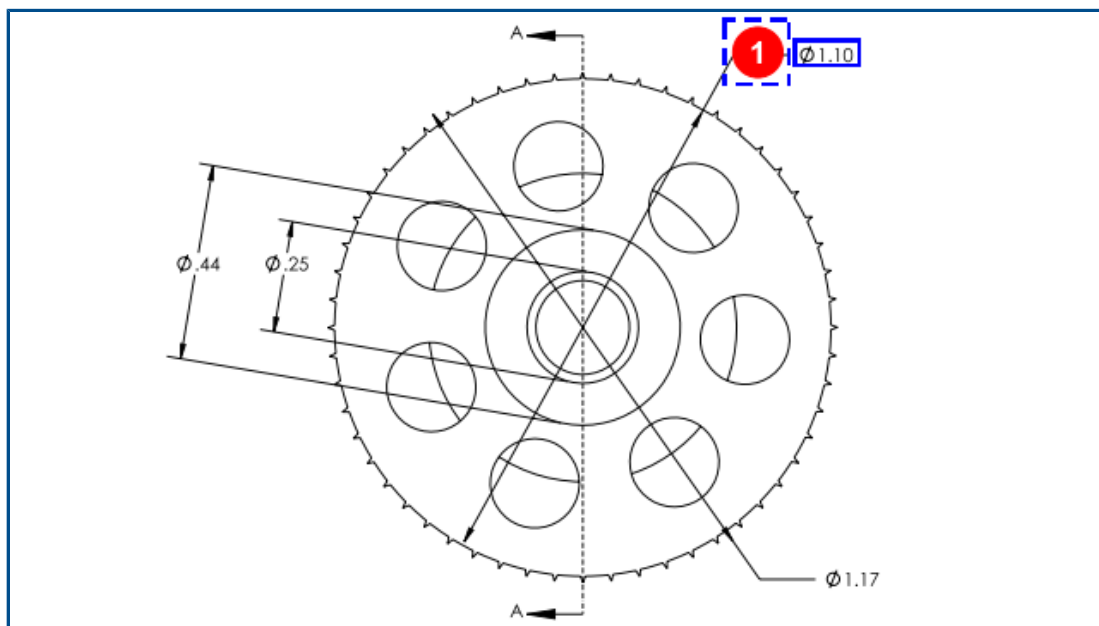
## Captures

<b>Include All</b>	Balloons all model based definition screen captures, or you can specify screen captures to include.
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## Views

<b>Include All</b>	Balloons all annotation views, or you can specify views to include.
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## Manually Ballooning PDF Files



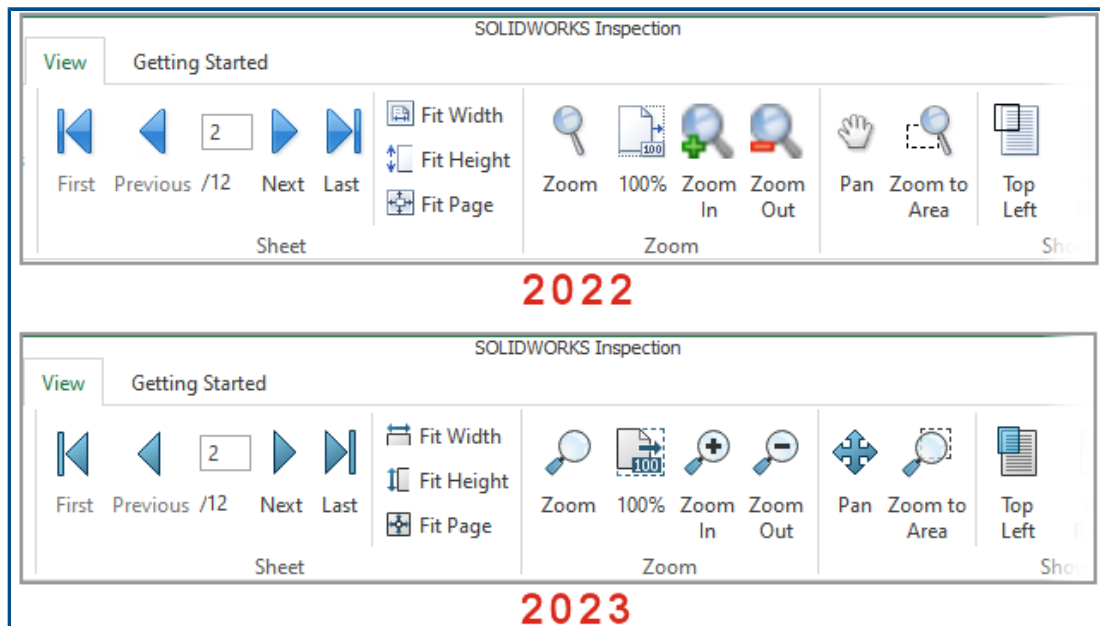
The **Manual Extract** tool replaces the optical character recognition (OCR) editor tool. It is streamlined to make manual extraction of annotations quicker to balloon.

### To manually balloon PDF files:

1. In SOLIDWORKS Inspection Standalone, click **New** .

2. In the Project Template dialog box, select a template and click .
3. In the Open CAD/PDF File dialog box, select a PDF file and click **Open**.
4. Click **Manual Extract**  (Document ribbon) and select the type of annotation to balloon:
  - **Dimension**
  - **Geometric Tolerances**
  - **Note**
  - **Surface Finish**
  - **Weld**
  - **Other**. Includes annotations such as temperature, torque, and electrical resistance.
5. In the PDF file, box-select an annotation of the type you selected in the previous step.
6. In the Properties/Preferences PropertyManager, under **General, Inspection, Ballooning**, and **Custom**, enter values for the remaining data in the annotation.

## User Interface Redesign



The user interface for SOLIDWORKS Inspection Standalone is redesigned to provide better support for high-resolution, high-pixel-density displays. Icons are consistent with those in SOLIDWORKS.

Icons are more crisp and readable, and they scale properly when used with high-resolution displays.



# 19

## SOLIDWORKS MBD

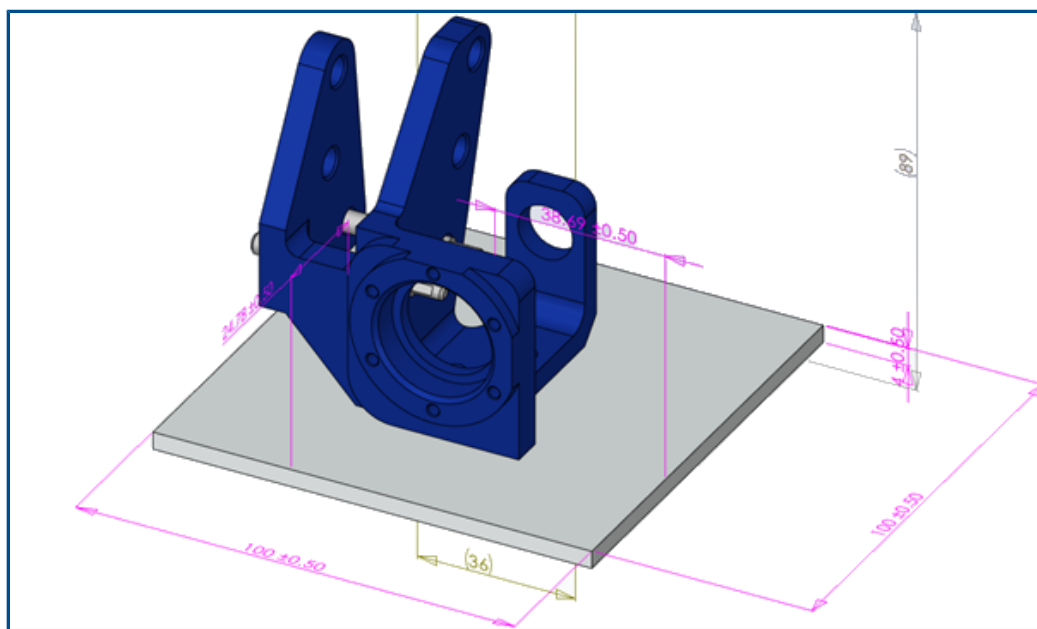
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This chapter includes the following topics:

- **Annotations and Dimensions in 3D PDF Files**
- **Dimension Extension Lines**
- **Editing Compound Features**
- **ISO Tolerance-Modifying Symbols**
- **Wedges**

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

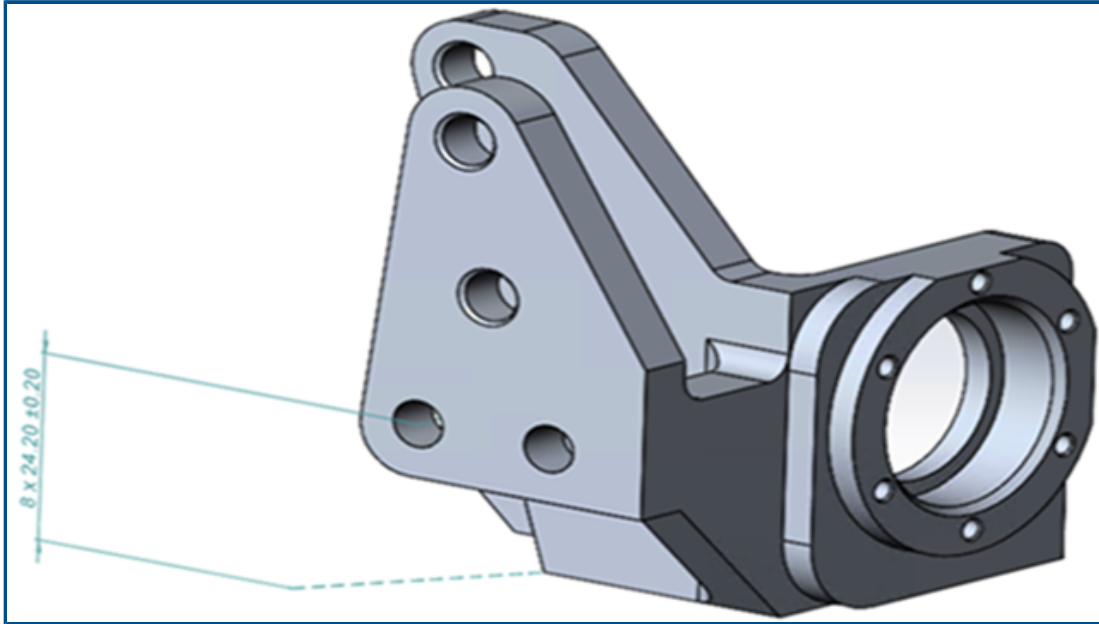
### Annotations and Dimensions in 3D PDF Files



When you create a 3D PDF for an assembly, you can view all dimensions in the assembly and 3D views, even in the viewport, with no data loss.

You can view component dimensions in 3D PDF files, including feature dimensions, reference dimensions, and DimXpert annotations.

## Dimension Extension Lines

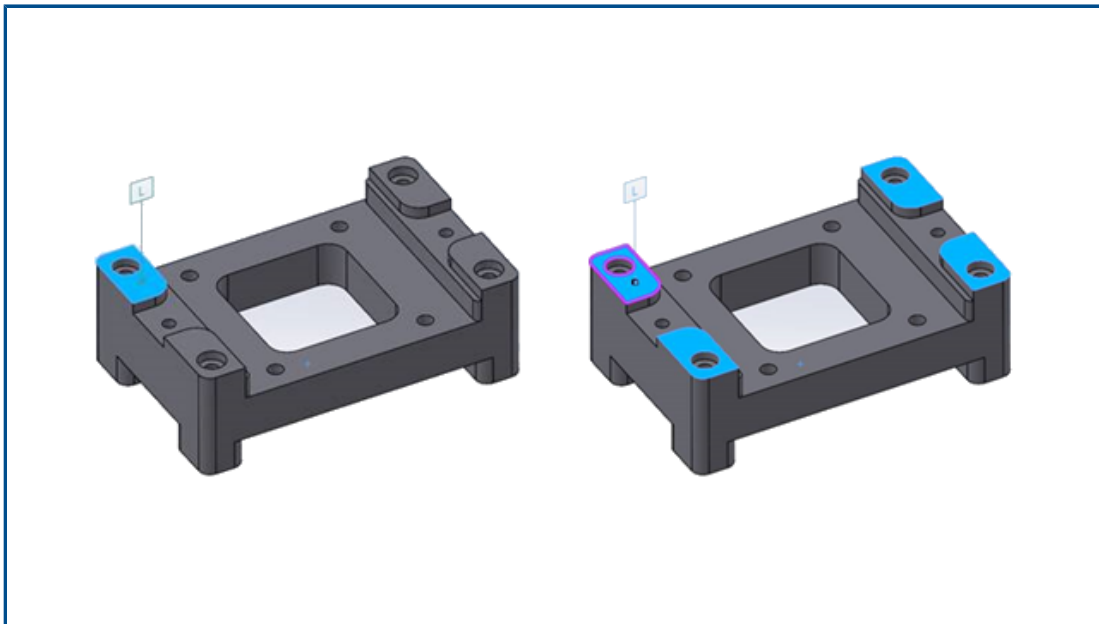


You can use dimension extension lines to connect the dimensions to the model.

**To show dimension extension lines:**

1. Click **Tools** > **Options** > **Document Properties** > **DimXpert** > **Display Options**.
2. Under **Linear Dimension**, select **Connect dimension extension lines to model**.

## Editing Compound Features



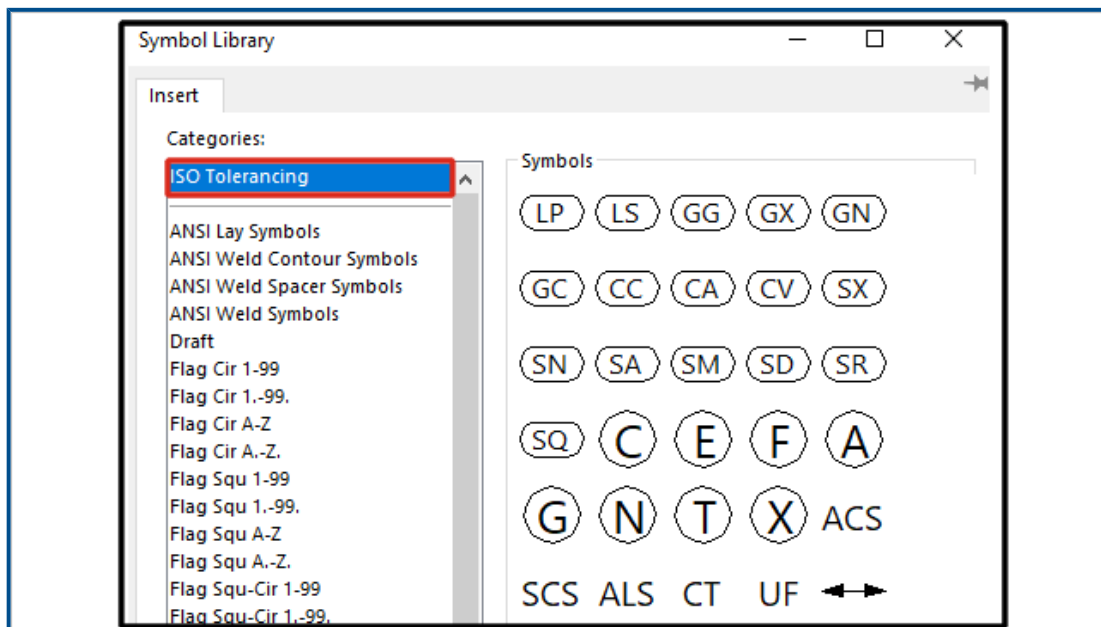
You can add or remove faces on an existing compound feature while maintaining the annotations.

You can edit compound planes, holes, and cylinders.

#### To edit compound features:

1. In the DimXpertManager, right-click the compound feature and click **Edit Feature**.
2. Select features to add or remove from the compound feature.
  - To add a face, select a feature and click **OK**.
  - To remove a face, right-click a feature and select **Delete**.

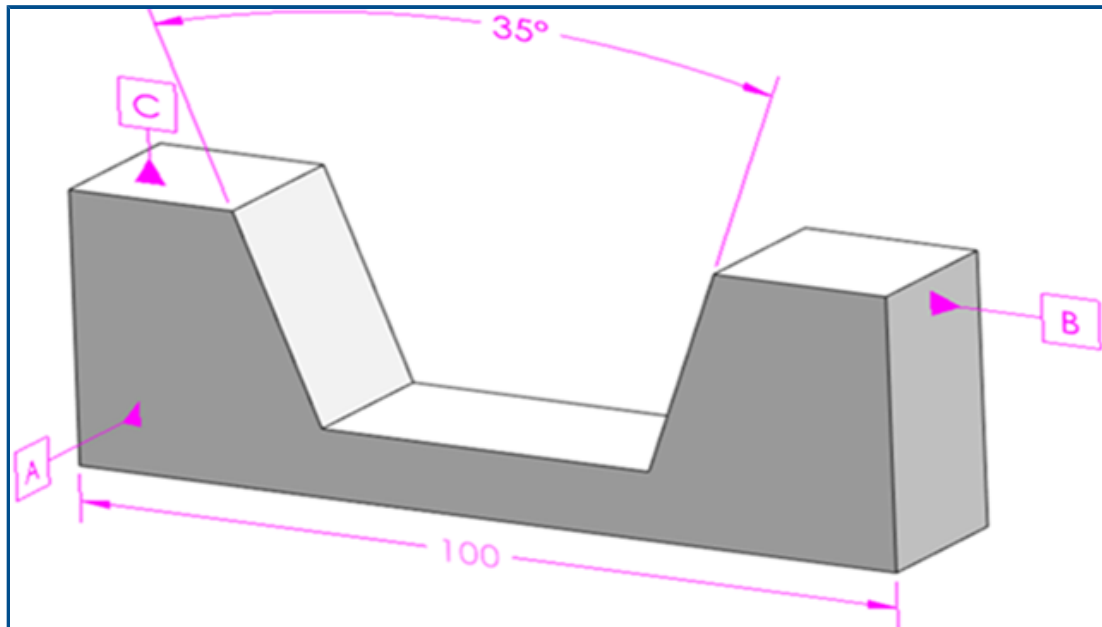
## ISO Tolerance-Modifying Symbols



You can apply ISO 14405-1:2016 standards-based symbols to dimensions and tolerances in DimXpert.

You can add symbols and other text directly to ISO dimensions and their tolerances from the **Tolerance Modifier** section in the DimXpert PropertyManager.

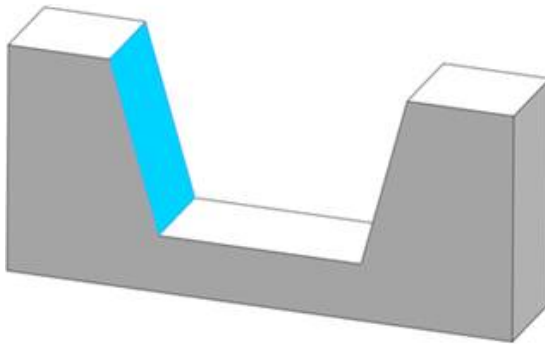
## Wedges



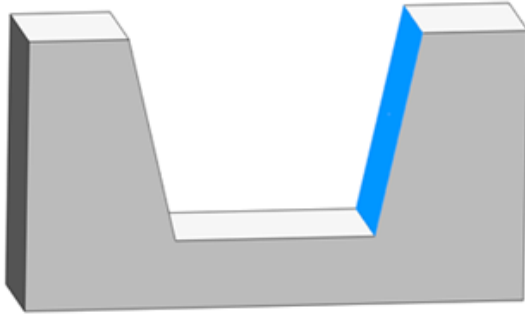
MBD supports wedge features. A wedge has two nonparallel planes with opposing vectors.

### To create a wedge:

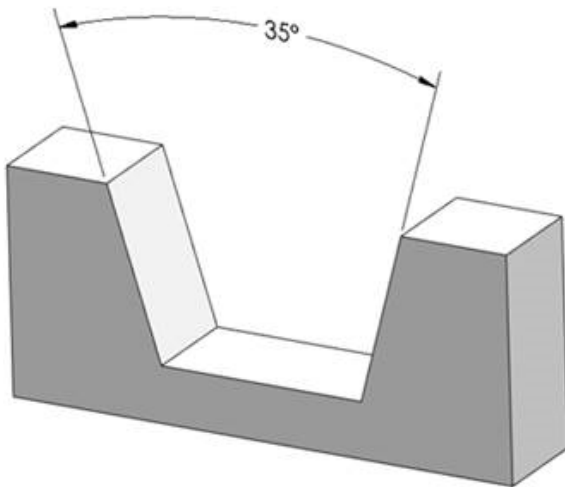
1. Click **Size Dimension**  (MBD Dimension toolbar).
2. Select the first plane.



3. From the **Feature Selector**, click **Create Width/Wedge Feature** .
4. Select the second plane.



5. In the **Feature Selector**, click ✓.
6. Click to place the dimension.



# 20

## eDrawings

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This chapter includes the following topics:

- **Assembly Graphics**
- **Configuration Pane**
- **Components Pane (2023 SP1)**
- **Configuration Support on Mobile Devices**
- **Display States**
- **Display Styles**
- **Measurement Callouts (2023 SP2)**
- **Supported Versions of File Types**
- **Using Localized Versions of eDrawings Web HTML**
- **Using Fractional Measurements**

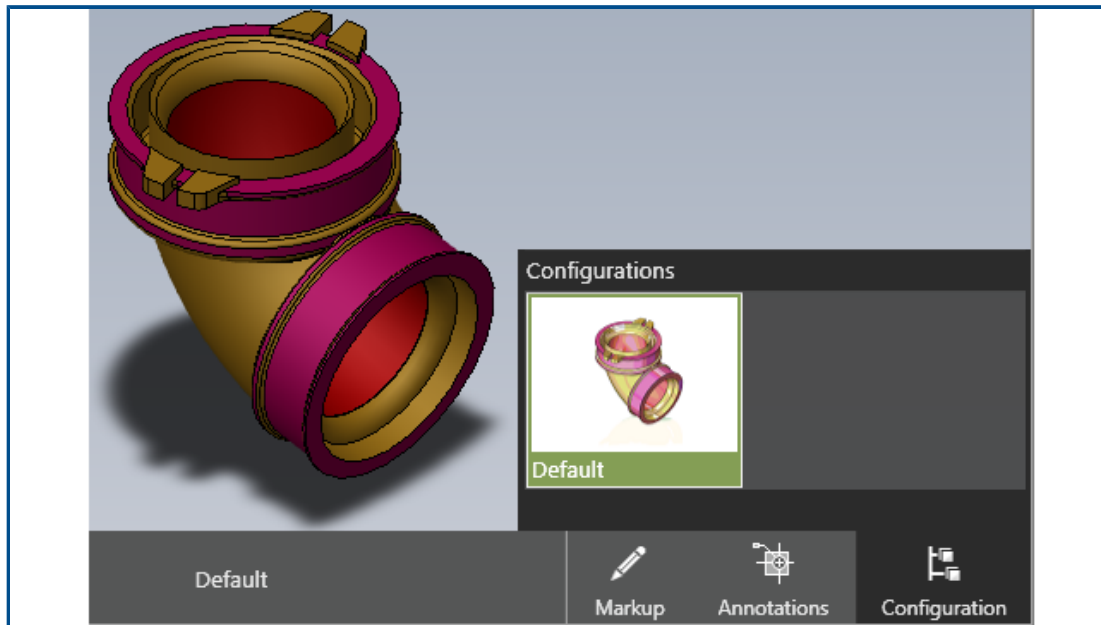
eDrawings® Professional is available in SOLIDWORKS® Professional and SOLIDWORKS Premium.

### Assembly Graphics

Graphics in eDrawings assemblies have improved. When you publish SOLIDWORKS assemblies as eDrawings assemblies, the graphics more closely resemble the original SOLIDWORKS files.

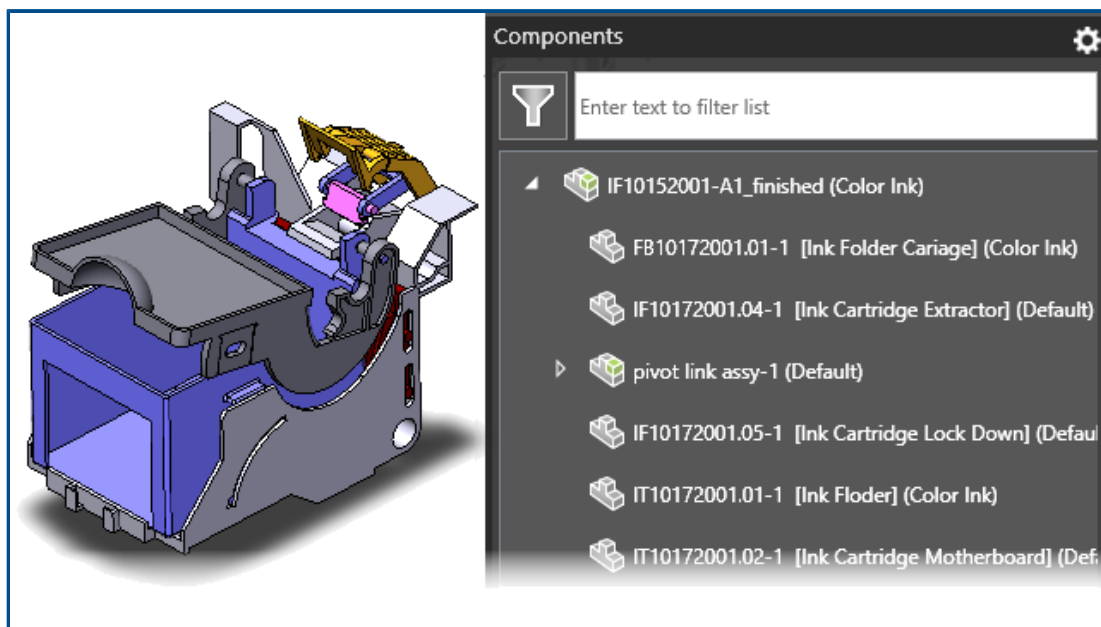
If the eDrawings assembly contains linked display states, the Configuration pane shows **Display States (Linked)**.

## Configuration Pane



When you open files in eDrawings that have only one configuration, the Configuration pane is available. Previously, there was no Configuration pane.

## Components Pane (2023 SP1)

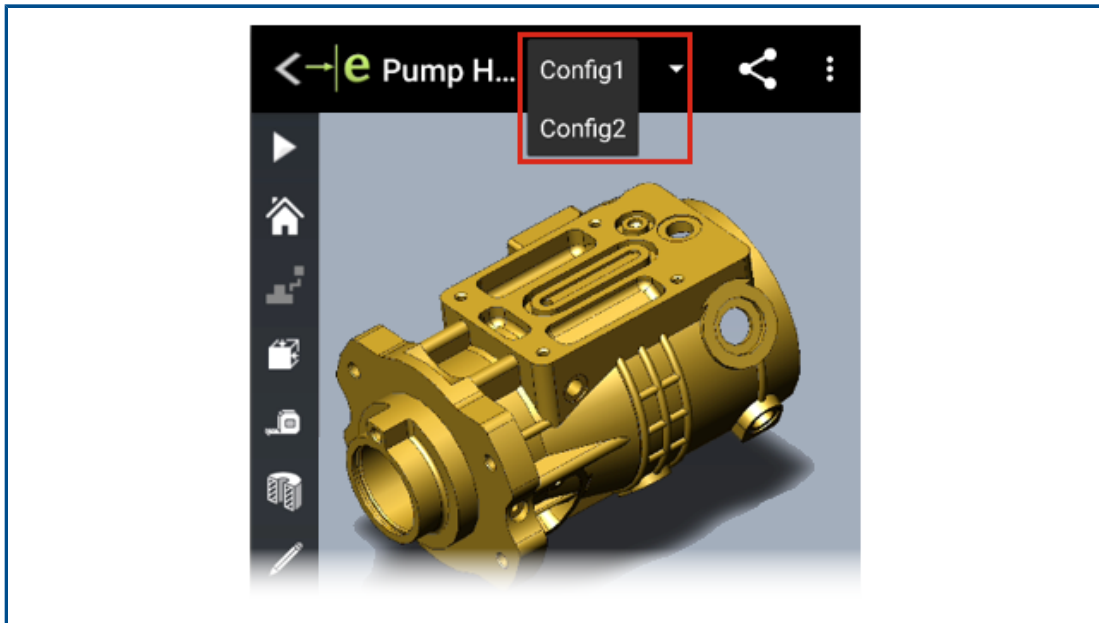


You can display component properties in the assembly tree of the eDrawings Components pane.

Component properties are only available if you published an .EASM file from SOLIDWORKS with **Save file properties for each component in the assembly** selected.

On the Components pane, click **Options**  and select **Show property**. This displays the specified property when you open the Components pane. You can also filter on the text to find components in a large assembly tree.

## Configuration Support on Mobile Devices

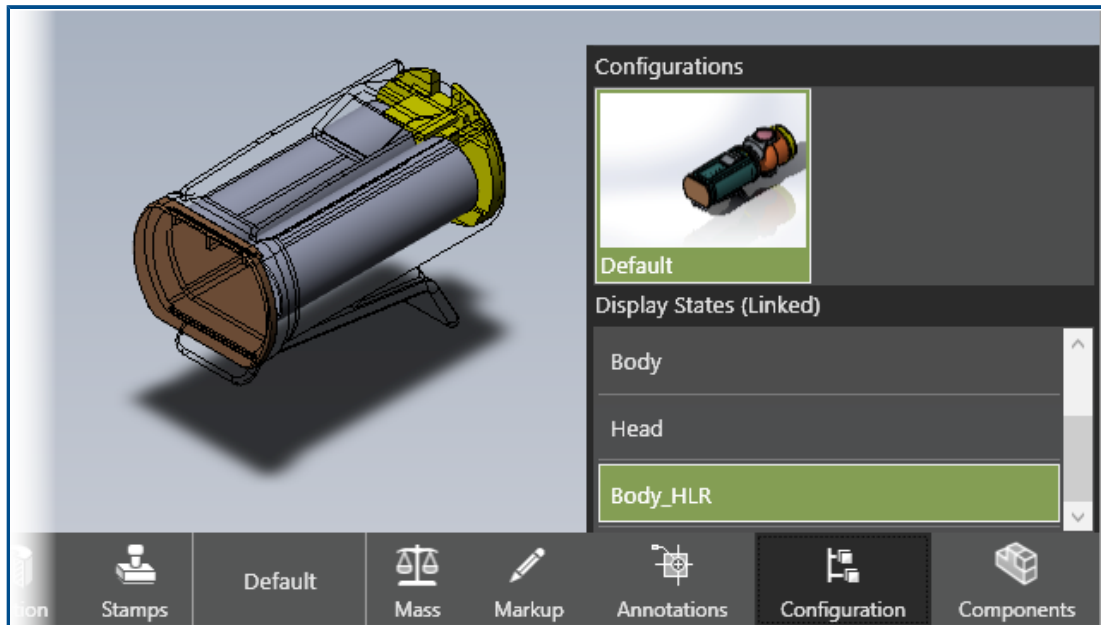


You can specify configurations of SOLIDWORKS parts and assemblies to be available when you open the models in eDrawings for iOS™ and eDrawings for Android™.

In the SOLIDWORKS software, set configurations with **Add Display Data Mark** to view the configurations in eDrawings. For information, see *SOLIDWORKS Help: Generating Display Lists*.



## Display States



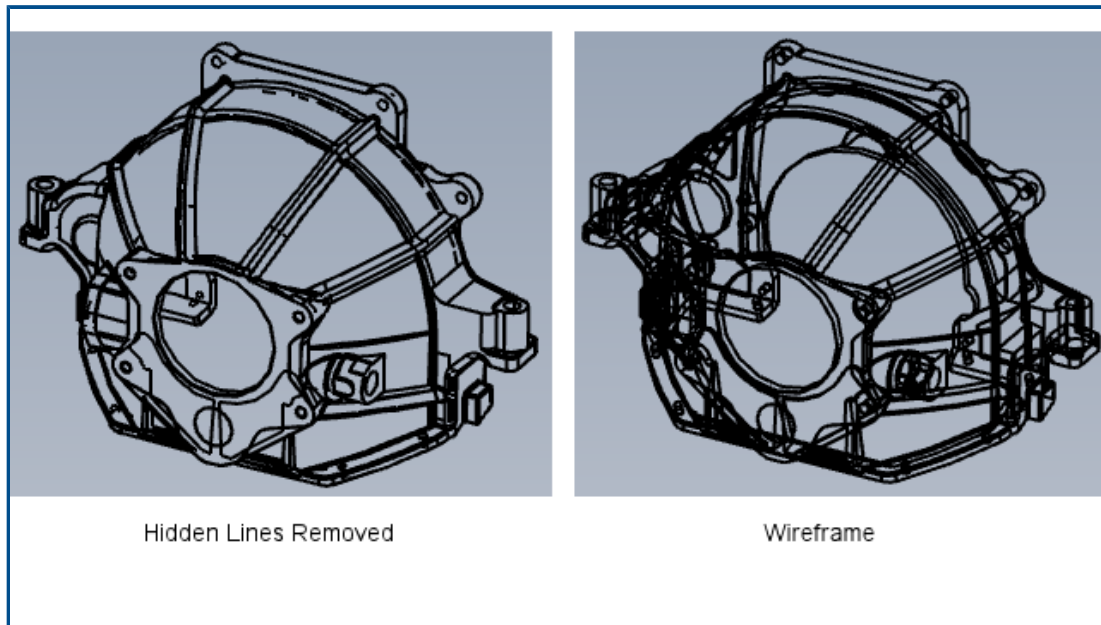
When you publish SOLIDWORKS assemblies as eDrawings assemblies (.EASM), display state data (for the selected configurations in the Save Configurations to eDrawings file dialog box) is stored in the eDrawings assembly files. This gives you important information to communicate and collaborate in eDrawings.

Display state data is also supported when you save an assembly as an eDrawings Web HTML file from eDrawings Professional. You can view the display states in supported browsers and in SOLIDWORKS PDM Web2.

On the Configuration pane, the option, **Link display states to configurations**, is removed. Display states are linked or unlinked based on the setting in the SOLIDWORKS assembly at the time of publishing the eDrawings assembly file from SOLIDWORKS. When the display states are linked to the configuration, only the display states that are associated with the selected configurations are published. If display states are linked to the configurations, the header says **Display States (Linked)**.

Textures are not supported in the display state data.

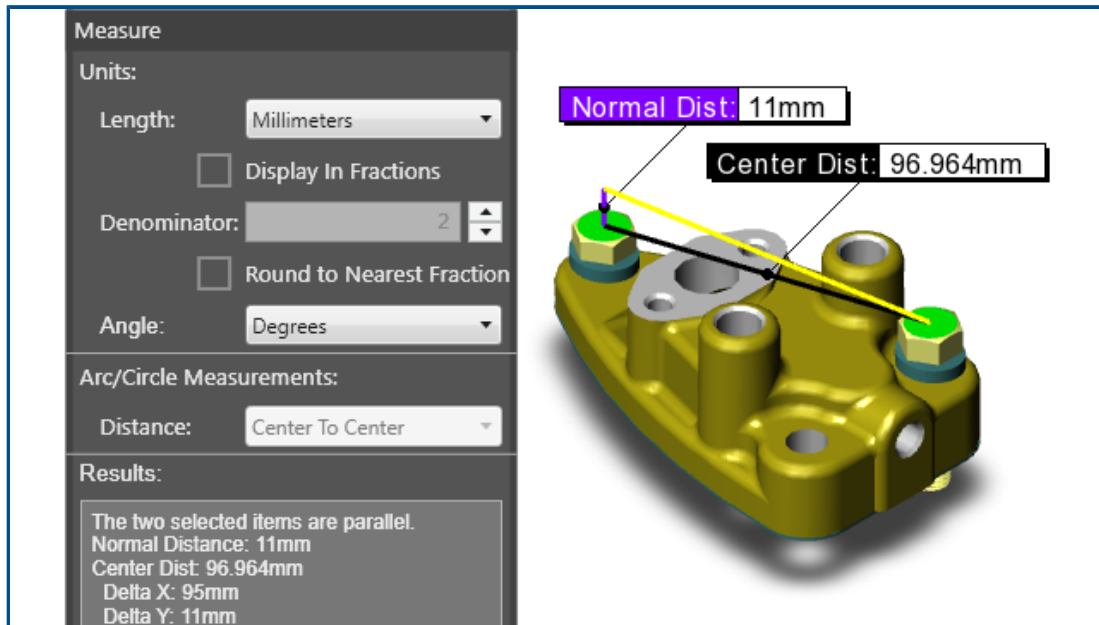
## Display Styles



You can view models in hidden lines visible (HLV) and wireframe modes. The tooltip for **Wireframe** has changed to **Hidden Lines Removed** (HLR).

Click **Display Style**  (Heads-up View toolbar) and select **Hidden Lines Visible** , **Hidden Lines Removed** , or **Wireframe** .

## Measurement Callouts (2023 SP2)



When you measure two parallel entities, the normal distance is displayed in the graphics area. Previously the distance was displayed in the Measure pane only.

Normal distance is the distance of a line perpendicular to the parallel entities. By having the distance displayed in the graphics area, you can see the measurement directly in the model instead of looking in the Measure pane. In the image above, the yellow line is the projection of the entity when the two parallel entities are offset to create the normal distance line, which is purple.

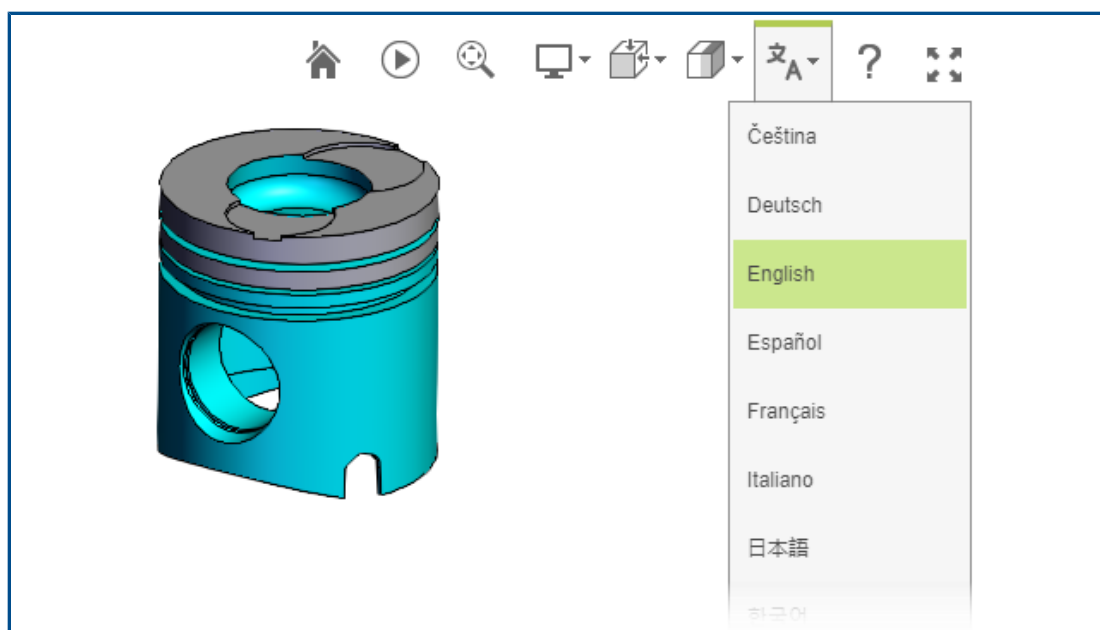
## Supported Versions of File Types

eDrawings has updated the supported versions for several file types.

Format	Version
ACIS (.sat, .sab)	Up to 2020
Autodesk® Inventor® (.ipt., .iam)	Up to 2022
CATIA V4 (.MODEL, .SESSION, .DLV, .EXP)	Up to 4.2.5
CATIA V5 (.CATDrawing, .CATPart, .CATProduct, .CATShape, .CGR)	Up to V5_6R2021
CATIA V6 / 3DEXPERIENCE (.3DXML)	Up to V5-6 R2019 (R29)
Creo - Pro/Engineer (.ASM, .NEU, .PRT, .XAS, .XPR)	Pro/Engineer 19.0 to Creo 8.0

Format	Version
IGES (.igs, .iges)	5.1, 5.2, 5.3
JT (.jt)	Up to v10.5
NX (Unigraphics) (.prt)	V11 to v18, NX to NX12, NX1847 Series to NX1980 Series
Parasolid (.x_b, .x_t, .xmt, .xmt_txt)	Up to 33.1
Solid Edge (.asm, .par, .pwd, .psm)	V19-20, ST-ST10, 2021
STEP (.stp, .step, .stpz)	AP 203 E1/E2, AP 214, AP 242
Wavefront OBJ (.obj)	Any

## Using Localized Versions of eDrawings Web HTML



(Available in eDrawings Professional.) When you save a file as eDrawings Web HTML, then open the .html file, you can select a language to display the user interface in the localized language. The file itself retains its native language.

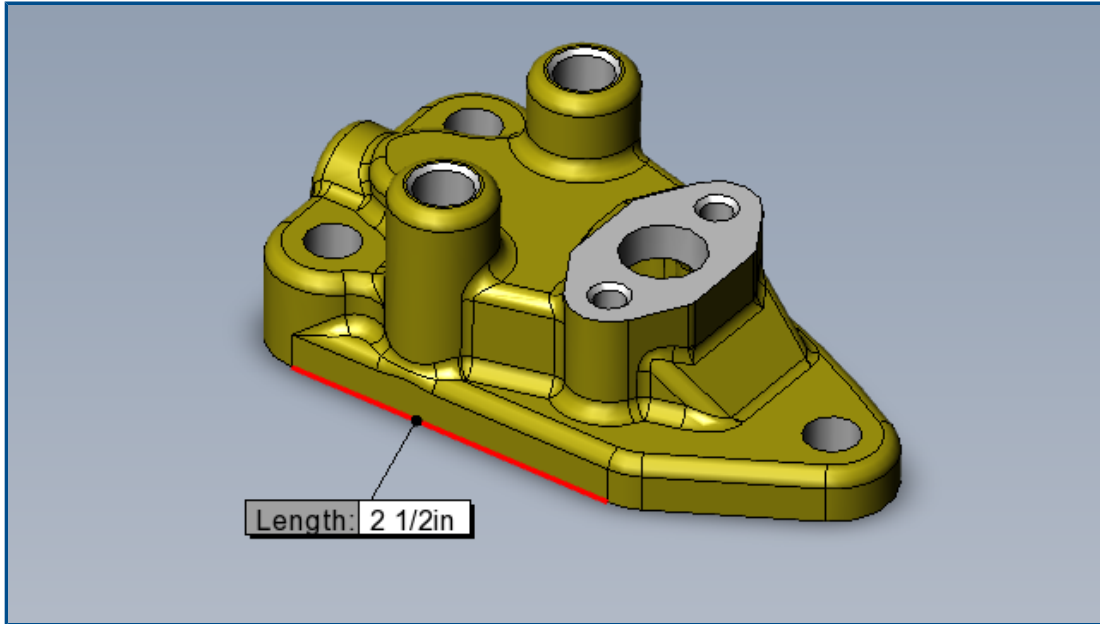
This is helpful if you share the file with a user who speaks a different language.

### To use a localized version of eDrawings Web HTML:

1. In eDrawings, click **File > Save As**.
2. In the dialog box, in **Save as type**, select **eDrawings Web HTML Files (\*.html)**.
3. Double-click the file that you saved.


4. In eDrawings Web HTML, click **Languages**  and select a language.

## Using Fractional Measurements



If you use the English unit system to measure eDrawings models, you can display measurements as fractions with the **Measure** tool.

### To use fractional measurements:

1. Click **Measure** .
2. On the Measure pane, under **Units**:
  - a. In **Length**, select **Inches** or **Feet and Inches**.
  - b. Select **Display In Fractions**.
  - c. For **Denominator**, enter the maximum digits for the fraction denominator.
  - d. Optional: Select **Round to nearest fraction** to round to the next highest fraction if the measurement does not have a fractional value for the denominator.

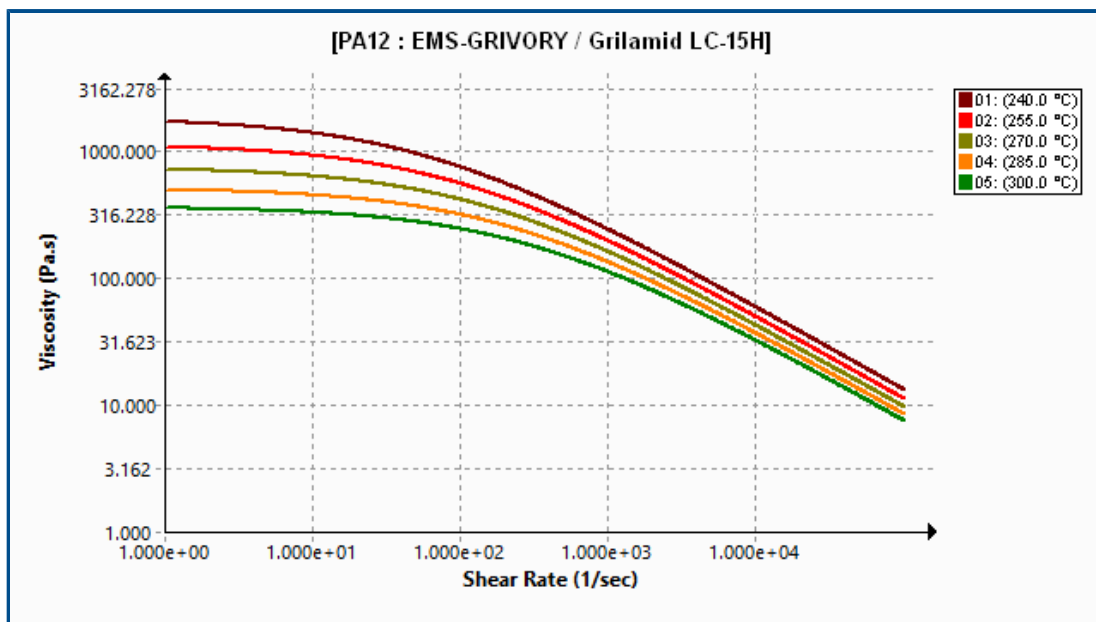
## SOLIDWORKS Plastics

This chapter includes the following topics:

- **Plastics Materials Database - Material Grades**
- **Plastics Materials Database Manager**
- **Summary and Report**

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

### Plastics Materials Database - Material Grades



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

Materials	Description
New Materials	Added 14 new material grades from material manufacturer EMS-GRIVORY.

Materials	Description
Modified Materials	Updated 54 material grades with the latest material property values from material manufacturer EMS-GRIVORY.
Removed Materials	Removed obsolete material grades from the database: <ul style="list-style-type: none"> <li>EMS-GRIVORY: 1</li> <li>CHI MEI Corporation: 37</li> </ul>

## Plastics Materials Database Manager

PC : Covestro / Makrolon 2556		Makrolon 2556	
Melt Temperature	300 °C	Polymer Family	PC
Max. Melt Temperature	320 °C	Manufacturer	Covestro
Min. Melt Temperature	280 °C	Recommended Melt Temperature	300 °C
Mold Temperature	100 °C	Maximum Melt Temperature	320 °C
Max. Mold Temperature	120 °C	Minimum Melt Temperature	280 °C
Min. Mold Temperature	80 °C	Recommended Mold Temperature	100 °C
Ejection Temperature	130 °C	Maximum Mold Temperature	120 °C
Transition Temperature	150 °C	Minimum Mold Temperature	80 °C
Viscosity : 7-Parameters Modified	5.86e+11	Ejection Temperature	130 °C
PVT : Modified Tait Equation	0.0008722	Thermoset Conversions	Not Available
Solid Density	1198 Kg/m3	Transition Temperature	150 °C
Specific Heat : Constant	2100 J/(Kg-K)	Viscosity : 7-Parameters Modified	5.86e+11
Thermal Conductivity : Constant	0.214 W/(m-K)	PVT : Modified Tait Equation	0.0008722
Elastic Modulus : Constant	2400 2400	Density	1198 Kg/m3

2022

2023

Several user interface enhancements to the Plastics Materials Database Manager improve its usability.

Display enhancements:

- Gray color border and gray gridlines replace the yellow color border and gridlines to improve the readability of the material properties.
- The **Elastic Modulus**, **Poisson's Ratio**, and **Thermal Expansion Coefficient** plots are removed.
- The material's polymer family and manufacturer name are removed from the title bar and are listed in separate material property fields.

Material properties enhancements:

- The database stores detailed fiber and filler properties for filled materials. You can view these properties under the **Filler Properties** section.

☐ Filler Properties	
Filler 1	
Description	Glass Fiber
Filler Percentage	55 %
Aspect Ratio	20
Initial Length	Not Available
Average Diameter	Not Available
Density	2500 Kg/m3
Specific Heat	700 J/kg.K
Thermal Conductivity	1 W/m.K
Elastic Modulus (E1)	70000 MPa
Elastic Modulus (E2)	70000 MPa
Tensile Strength ( $\sigma_1$ )	3500 MPa
Tensile Strength ( $\sigma_2$ )	3500 MPa
Poisson's Ratio ( $\nu_{12}$ )	0.29
Poisson's Ratio ( $\nu_{23}$ )	0.29

- In addition to the material's solid density, the database stores its melt density. You can view the material densities under the **Density** section.

Viscosity	PVT	Specific Heat	Thermal Conductivity
Shear Relaxation Modulus	Curing Model	Polymer-Material Parameters	
☐ Density		990	780
Solid Density		990 Kg/m3	
Melt Density		780 Kg/m3	

- The database stores the following Thermoset material properties, if they are available from the manufacturers:
  - PVT data for the cured and uncured states.
  - Heat of Reaction and Induction Time Constants.
  - Ejection Conversion and Initial Conversion.

The solver is also enhanced to account for these additional material properties (when available) to improve the accuracy of Fill, Pack, and Warp simulations.

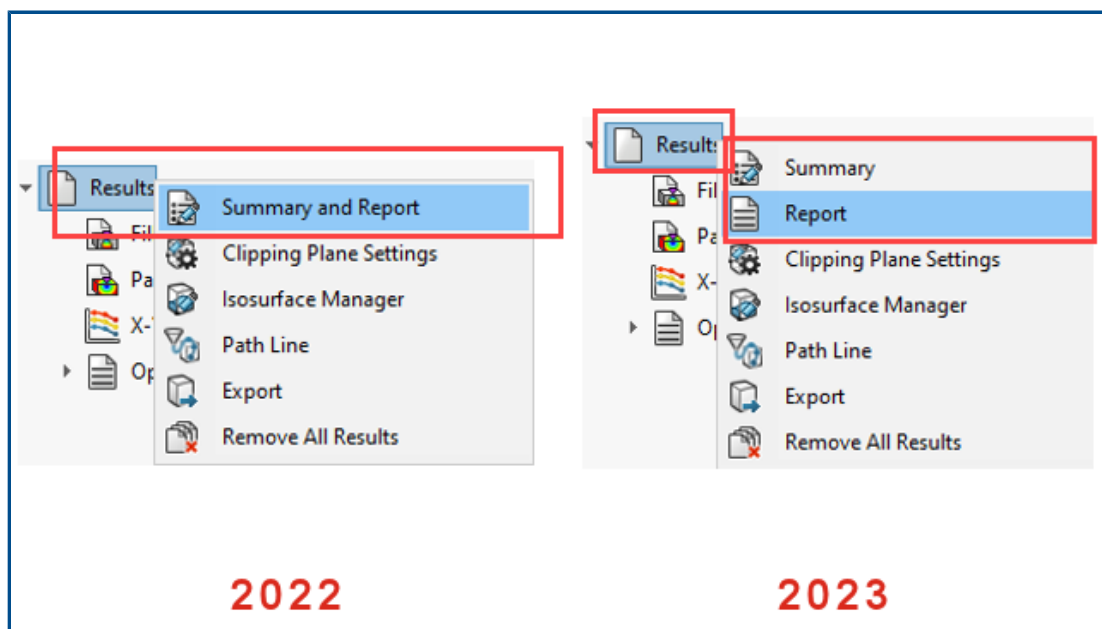
The classifications of the material's **Data Source and Information** field are revised.

Classification	Description
<i>Manufacturer Name</i>	Name of the manufacturer who provided the grade-specific material data.
<i>Manufacturer Name; Generic PVT Data</i>	The Pressure Volume Temperature (PVT) data are generic, while the rest of the data are grade-specific and provided by the material manufacturer.
<i>Manufacturer Name; *to be phased out</i>	The material manufacturer is planning to phase out the specific material grade.



Classification	Description
<b>SIMPOE</b>	The material data are from a legacy source and are not updated.
<b>SIMPOE; Generic PVT Data</b>	The material data are from a legacy source and have been updated from constant density to generic PVT data to improve the accuracy of Fill, Pack, and Warp simulations.

## Summary and Report



You can access the **Summary** and **Report** features from the PlasticsManager independently. In addition, there are several enhancements for the **Summary** and **Report**.

- **To access the Summary:**

In the PlasticsManager, right-click **Results** and click **Summary**.

- **To access the Report:**

In the PlasticsManager, right-click **Results** and click **Report**.

Enhancements to the **Summary** feature include:

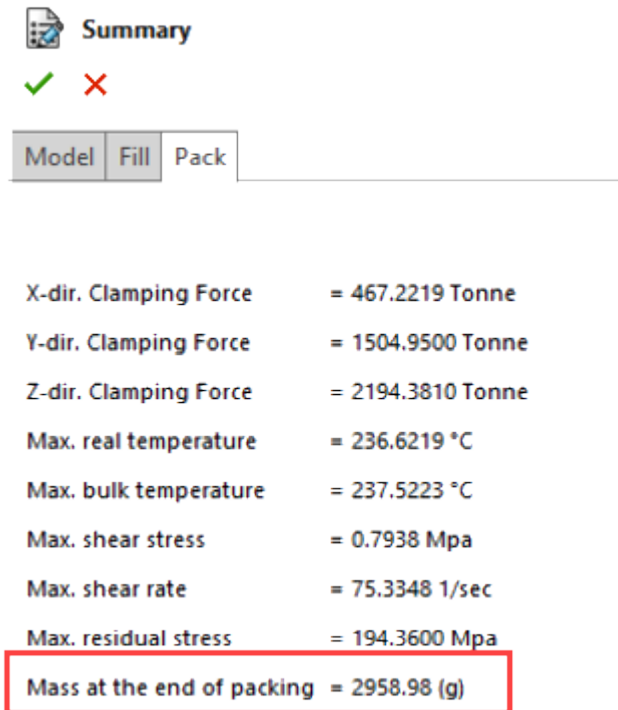
- Quick access to key results.
- Improved content organization into three sections for the Model tab:

**Study Characteristics** Lists the following features: study name, injection process, number of injection units, analysis mesh procedure, and the simulation sequence.

**Injection Unit Settings** Lists the following features for each injection unit: polymer material and the material's melt, mold, transition, and ejection temperatures.

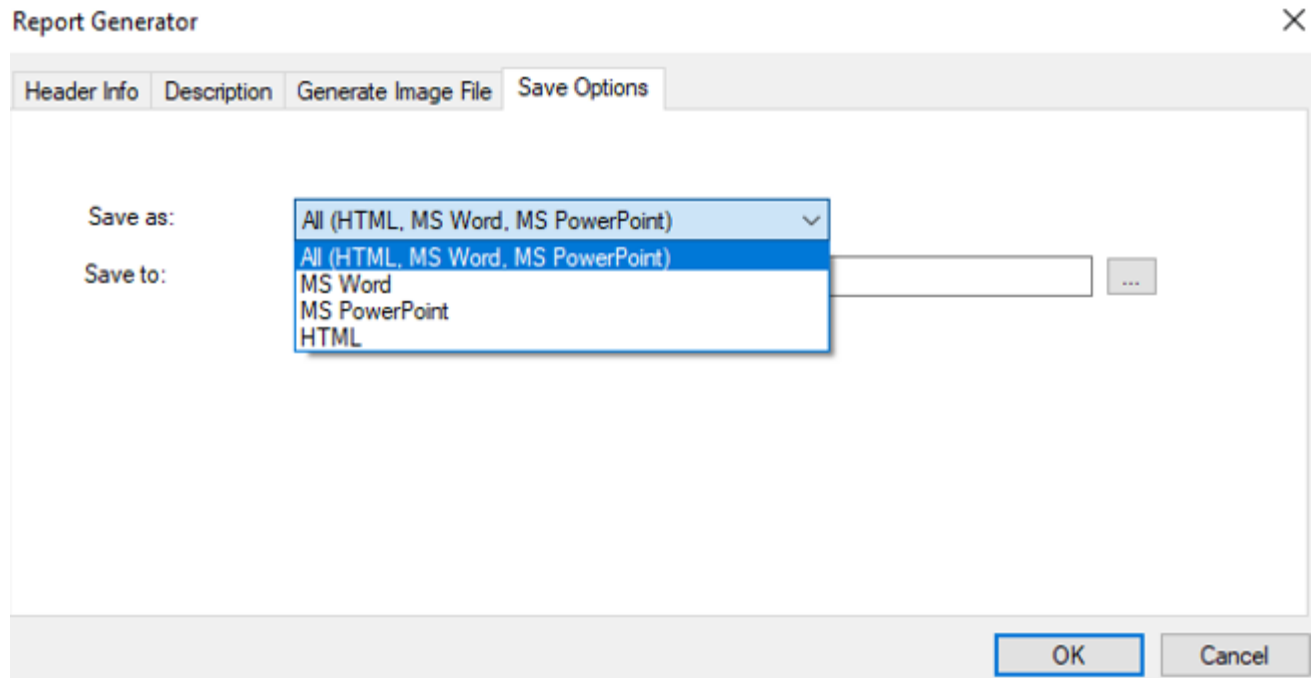
**Physical Attributes** Lists the model volume, mass, and overall dimensions.

- Addition of the plastic part's mass at the end of packing in the Pack tab.



Enhancements to the **Report** feature include:

- Updated Microsoft Word report templates. The **Classic** and **Light** report templates of previous versions are replaced with a single template that has a more modern look and feel.
- Additional formats are available for saving a report. In addition to Microsoft Word, you can save a report as a Microsoft PowerPoint or HTML document. The default option **All (HTML, MS Word, MS PowerPoint)** saves a report to all three formats.



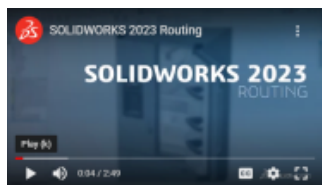
# 22

## Routing

This chapter includes the following topics:

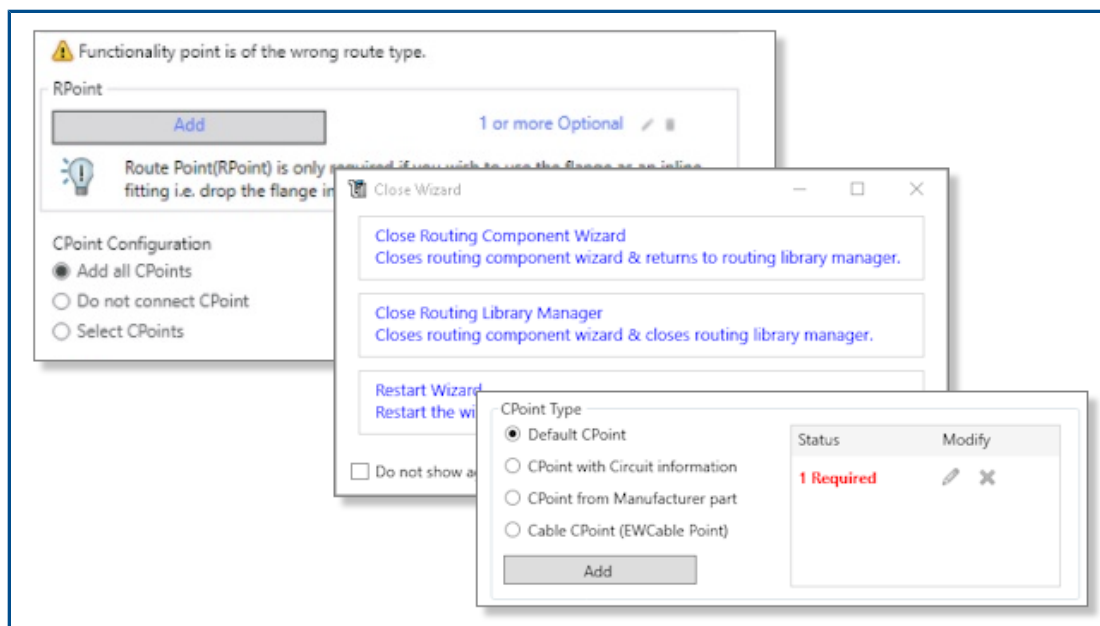
- **Routing Library Manager Enhancements**
- **Complex Splices**
- **Displaying the Cross Section of a Harness Segment**
- **Flatten Route Improvements**
- **Reorienting Connectors in Flattened Routes**

Routing is available in SOLIDWORKS® Premium.



**Video: What's New in SOLIDWORKS 2023 - Routing**

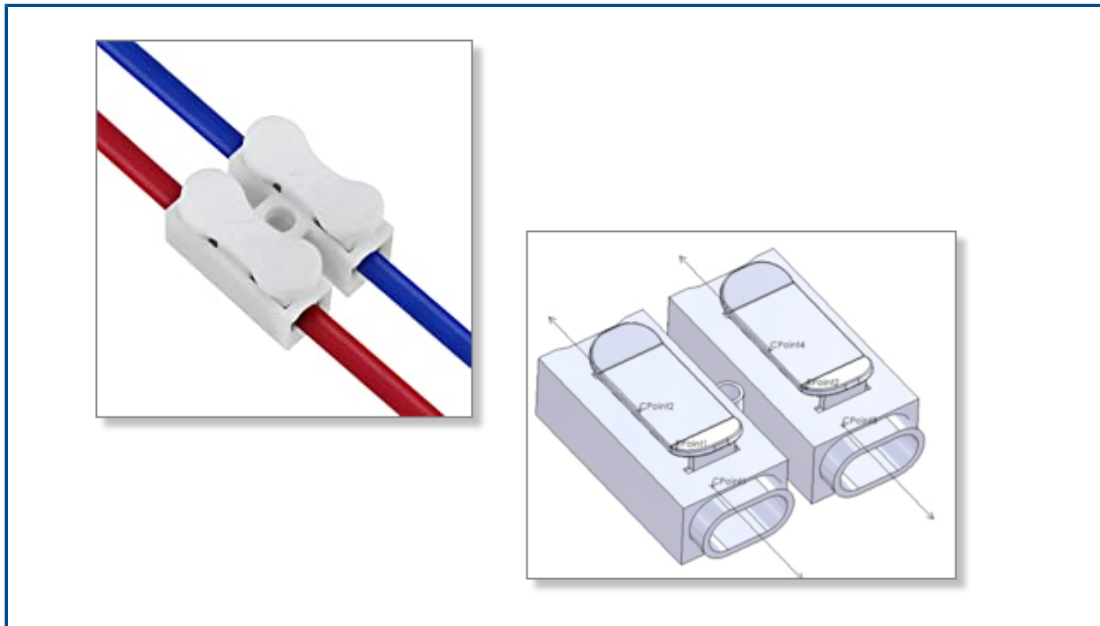
## Routing Library Manager Enhancements



The Routing Library Manager has improved ergonomics and usability, several enhancements have been done:

- Improved discoverability and guidance in the user interface by tooltips.
- Better management of the connection points and Mate References (create, edit, and delete workflows).
- Improved saving and canceling of modifications.

## Complex Splices



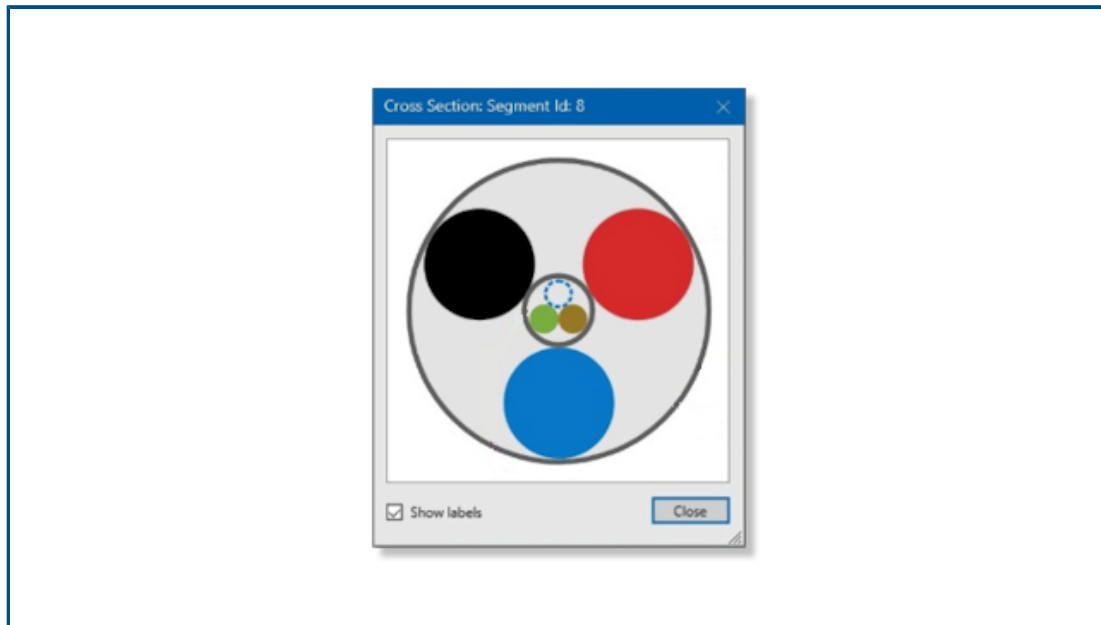
A complex splice has multiple circuits to which you connect wires or cable cores.

To connect wires or cable cores to a complex splice, you must insert the CPoints, and the RPoints if the wire crosses the splice, corresponding to the different circuits.

You can use complex splices in a manual routing or using the From-To list.

The complex splices are not supported in the flattening.

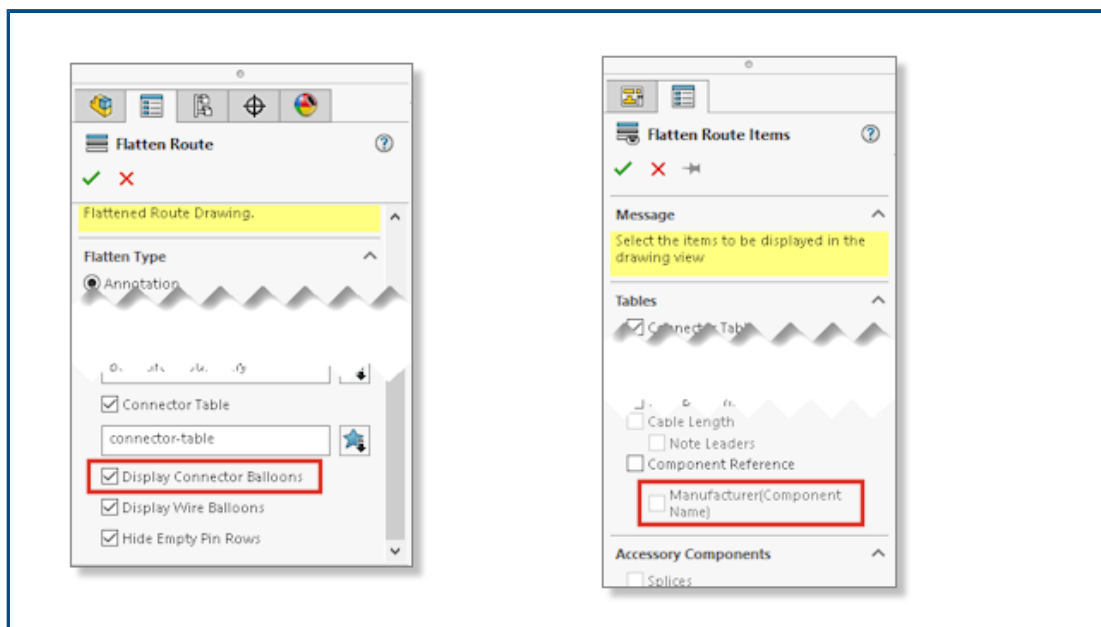
## Displaying the Cross Section of a Harness Segment



You can access the cross section of a harness segment.

In the Electrical Attributes PropertyManager, select **Show Cross Section** to view a graphical cross section of the harness segment. The formula to calculate the diameter of this segment takes into consideration if it contains a cable.

## Flatten Route Improvements

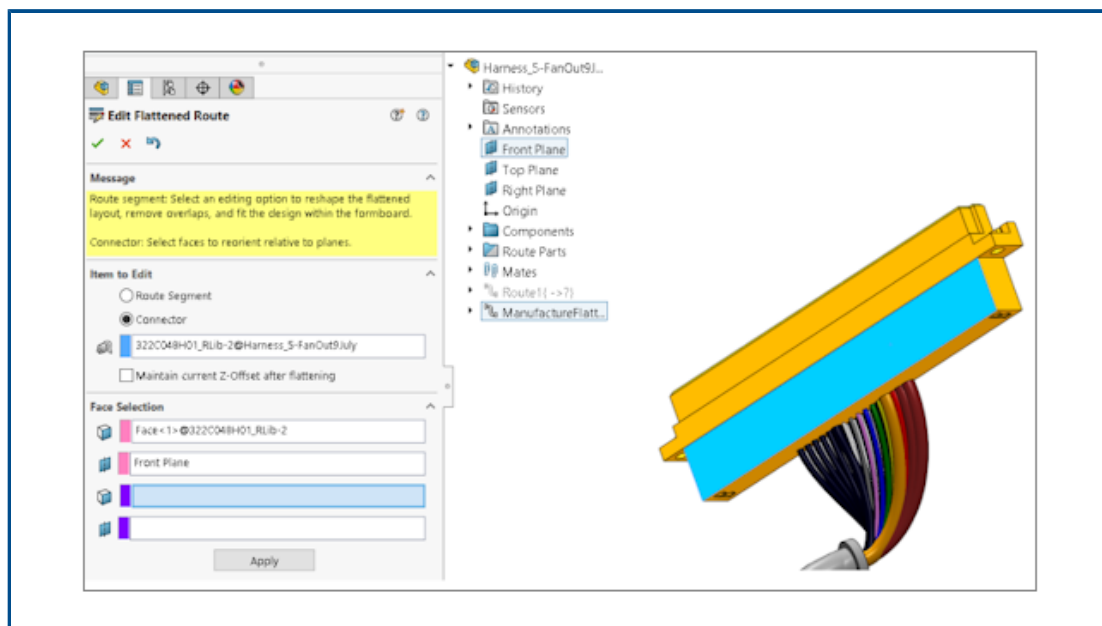


Improvements to the flatten route design deliver higher-quality flatten route drawings.

Improvements relate to the following items:

- The **Auto-Balloon** option applies to balloons for connectors and wires
- When you modify a connector table, you can apply this change to all existing connector tables
- The **Show/Hide Flatten Route Items** command is available in the **Electrical** toolbar
- In the **Flatten Route Items** PropertyManager, you can select **Manufacturer (component name)** to split the text displayed for the component in the flattened route drawing

## Reorienting Connectors in Flattened Routes



You can reorient the connectors to be parallel to a selected plane.

In the **Edit Flattened Route** PropertyManager, select a plane in **Faces and Planes**. This does not affect the **Maintain 3D Orientation** process.

# 23

## DraftSight

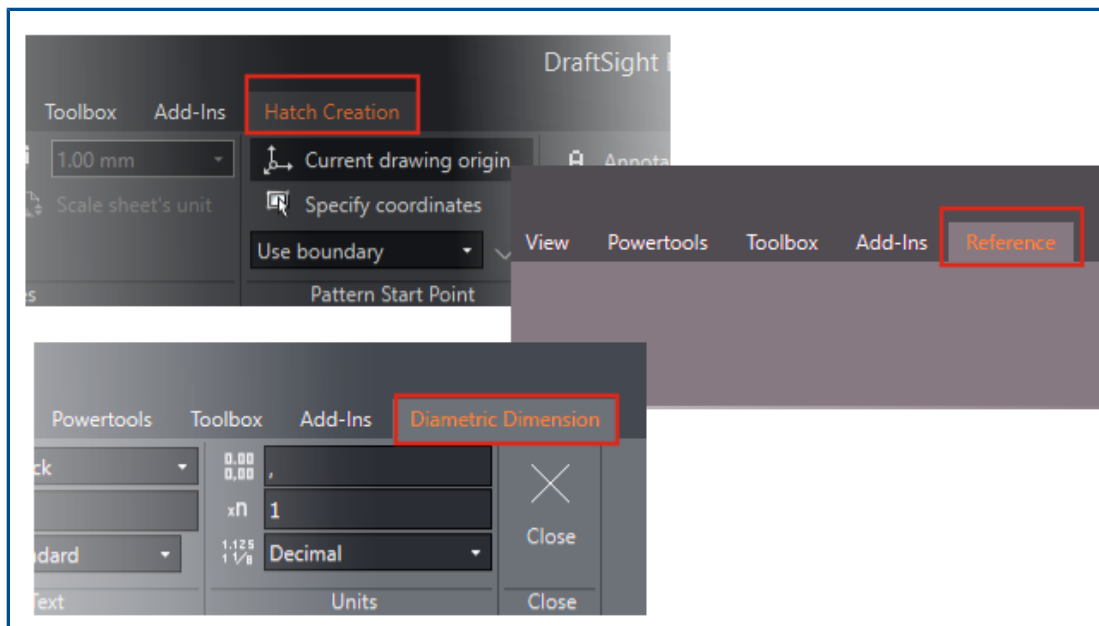
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This chapter includes the following topics:

- **Contextual Ribbon Tabs (2023 SP1)**
- **Cycling Selection (2023 SP1)**
- **Data Extraction (2023 SP1)**
- **Page Layout Manager (2023 SP1)**
- **Annotation Monitor (2023 SP1)**

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

### Contextual Ribbon Tabs (2023 SP1)



You can display specific tools on the ribbon when you select particular entities or run the commands to create new entities.

The contextual ribbon tab provides:

- An uninterrupted view of the graphics area because it is not blocked by the dialog box

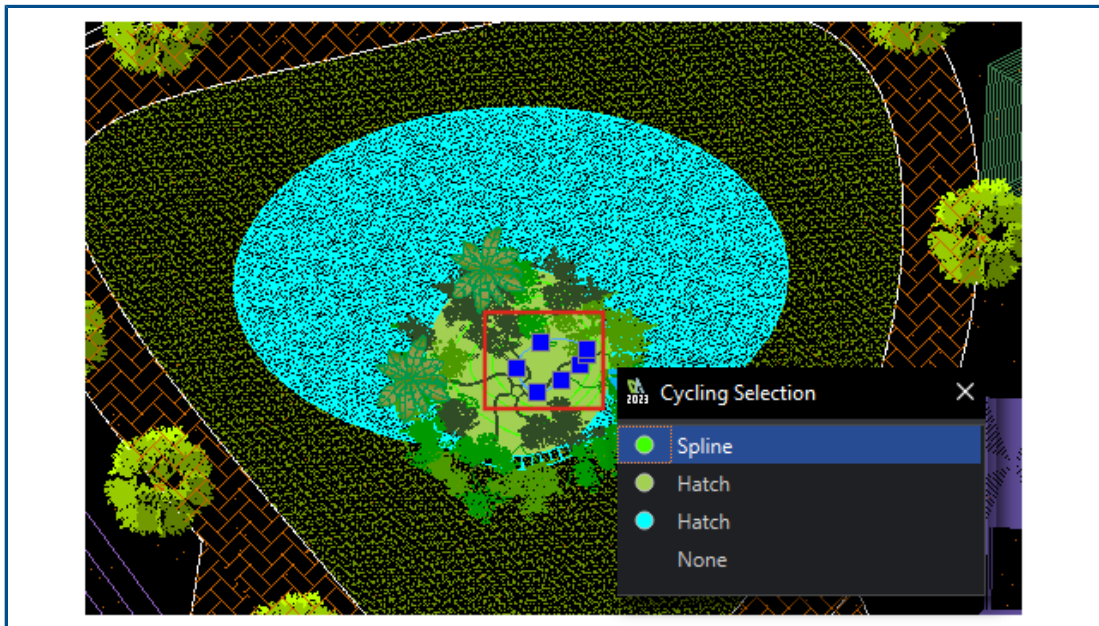


- Immediate visibility of changes that you apply to entities
- Relevant tools for completing a task

DraftSight offers contextual ribbon tabs for the following operations:

- Creating and editing hatches
- Editing table cells
- Editing PDF and DGN underlays
- Editing images and external references
- Editing annotation entities, dimensions, leaders, multileaders, and tolerance symbols.

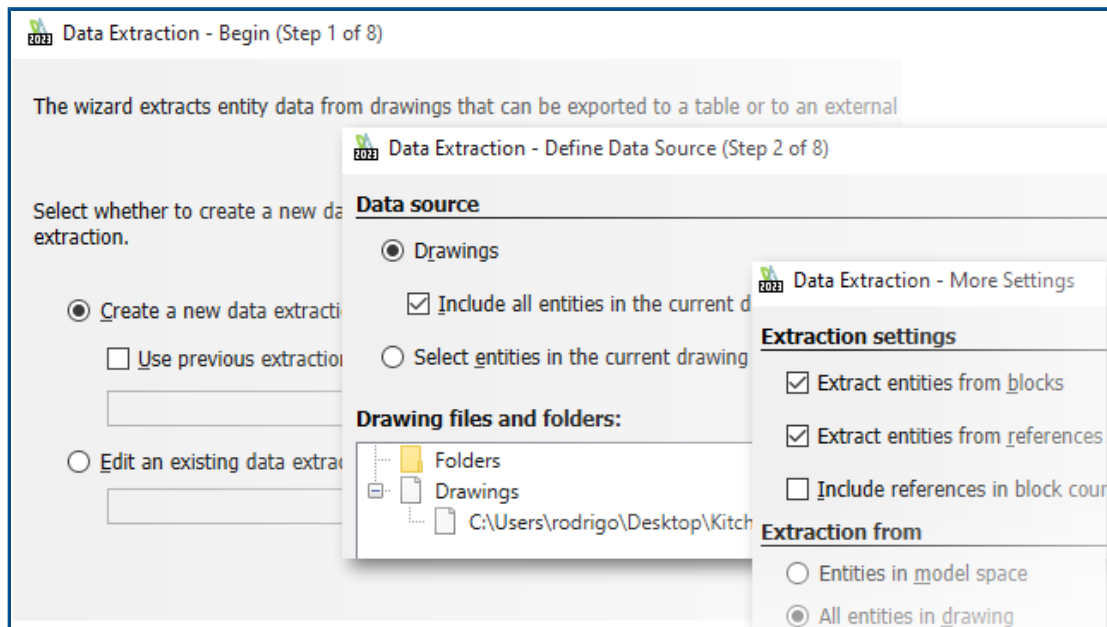
## Cycling Selection (2023 SP1)



You can select specific entities from the list of overlapped entities displayed in the Cycling Selection dialog box.

In previous releases, it was difficult to select an overlapped entity when working on a complex drawing.

## Data Extraction (2023 SP1)



You can use the Data Extraction wizard to specify a set of entities or blocks and extract properties and attributes information.

You can:

- Create a bill of materials from the set of entities.
- Display the result in a table on the drawing.
- Save the result in an external file that you can exchange with others.

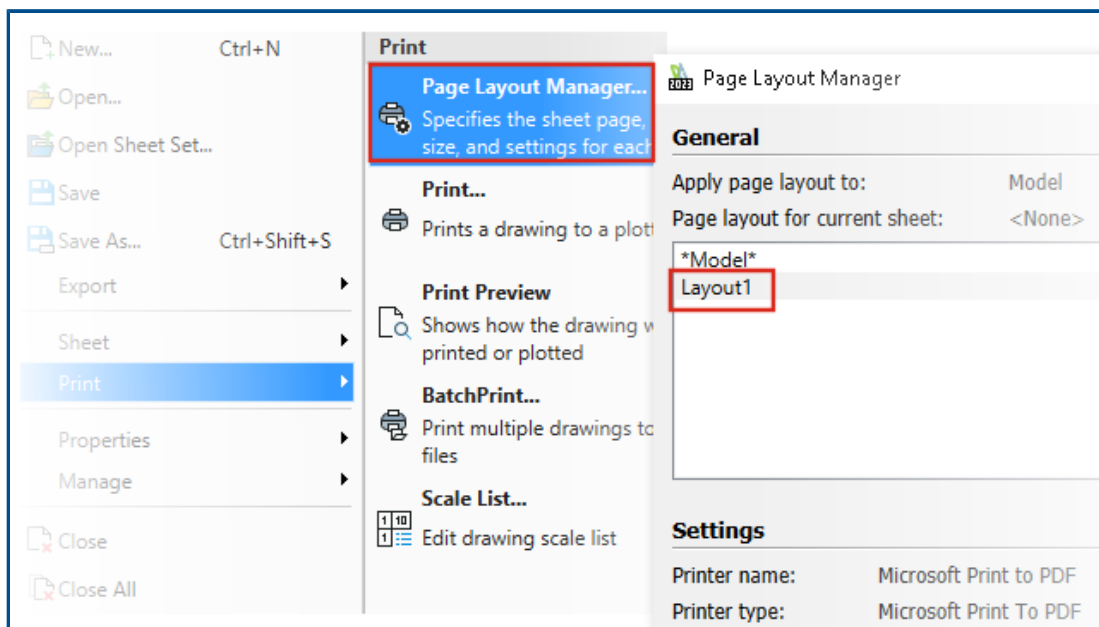
Use the `ExtractData` command to open the Data Extraction wizard.

Options in the Data Extraction wizard let you:

- Specify the set of drawings or folders.
- Filter entities.
- Organize the extracted data.
- Specify an output format.
- Format the output data in a table.

DraftSight saves the data source selections, entities and property selection, and table formatting in a Data Extraction File (`.det`). You can use the already created Data Extraction File as a template to perform the same type of extraction for another drawing.

## Page Layout Manager (2023 SP1)



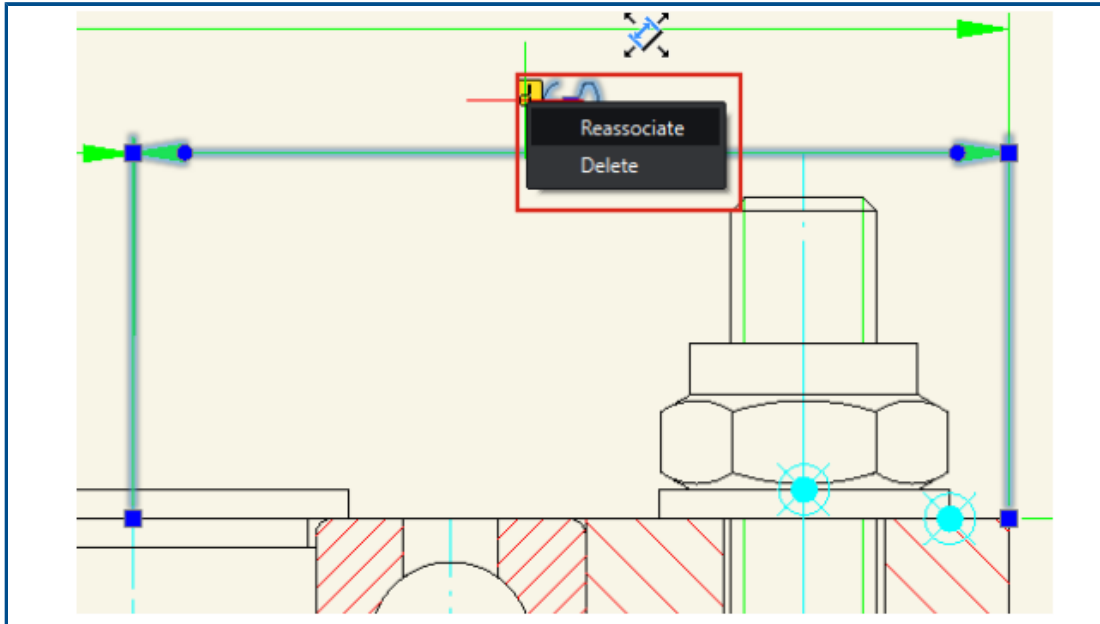
You can use **Page Layout Manager** to save the print settings in a drawing file.

In previous versions, the **Print Configuration Manager** saved the print settings in the .CFG file. When you shared the drawing file with users, you had to share the .CFG file to retain the print settings.

Under **System Options > Printing > General Options**, select **Use Page Layout Manager**.

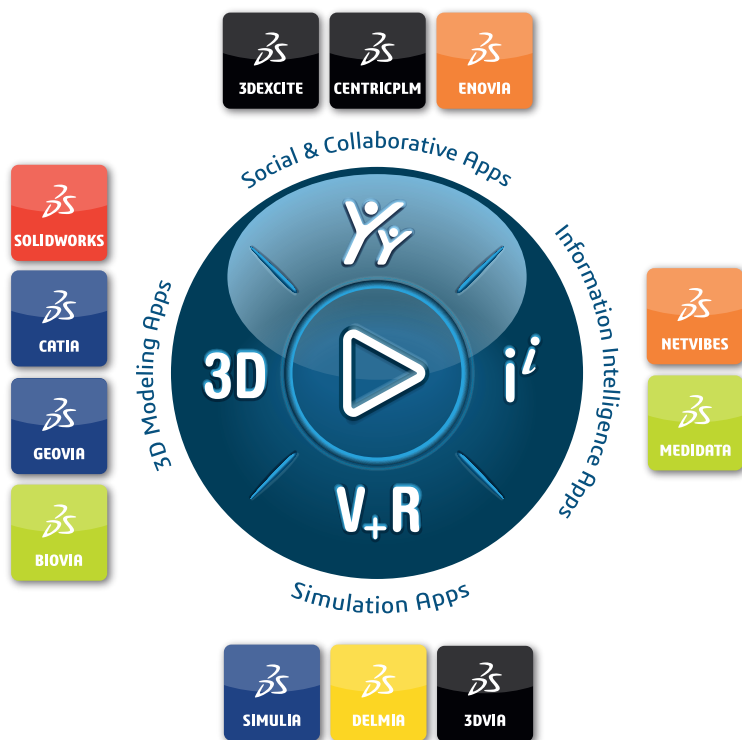
DraftSight reads and lets you print the print settings of a drawing file created in other software, for example AutoCAD®.

## Annotation Monitor (2023 SP1)



Annotation Monitor identifies annotations that are no longer associated with their related geometry.

DraftSight displays the **Reassociate** and **Delete** options. You can either reassociate the annotations or delete them.



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