



WHAT'S NEW

SOLIDWORKS 2024



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1

Welcome to SOLIDWORKS 2024

This chapter includes the following topics:

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



At SOLIDWORKS®, we know that you create great designs, and that your great designs get built. To streamline and accelerate your product development process from concept through manufactured products, SOLIDWORKS 2024 contains new, user-driven enhancements focused on:

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

Top Enhancements

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

Parts and Features

on page 24

- Hole Wizard
- Making Multibody Parts from Assemblies

Sheet Metal

on page 32

- Slot Propagation
- Stamp Tool
- Tab and Slot

Structure System and Weldments

on page 39

- Corner Management

Assemblies

on page 45

- Defeature Rule Sets
- Repairing Missing References in a Linear or Circular Pattern

Detailing and Drawings

on page 58

- Overridden Dimensions
- Keeping Chain Dimensions Collinear
- Reattaching Dangling Dimensions

SOLIDWORKS MBD

on page 129

- Hole Tables
- Repairing Dangling Dimensions

Performance

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

Some of the highlights for performance and workflow improvements are:

SOLIDWORKS Fundamentals

- Graphics rebuild after exiting SOLIDWORKS options.

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

- Silhouette edges.

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

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

Sketching

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

Sheet Metal

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

Import/Export

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

SOLIDWORKS PDM

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

The following operations are approximately two times faster:

- Add files
- Change state
- Copy tree

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

SOLIDWORKS Electrical

Archiving a project for remote users (VPN connection) is improved and is much faster.

eDrawings

Performance improvements include:

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

For More Information

Use the following resources to learn about SOLIDWORKS:



What's New in PDF and HTML

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

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

Interactive What's New

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

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](#).

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Installation

This chapter includes the following topics:

- **Installation Access Starting with SP0 for SOLIDWORKS Student and Education Editions**
- **Render Installation Manager with Microsoft Edge WebView 2**

Installation Access Starting with SP0 for SOLIDWORKS Student and Education Editions

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

Render Installation Manager with Microsoft Edge WebView 2

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

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

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SOLIDWORKS Fundamentals

This chapter includes the following topics:

- [Changes to System Options and Document Properties](#)
- [Accelerate the Display of Silhouette Edges](#)
- [Application Programming Interface](#)
- [Saving SOLIDWORKS Documents as Previous Versions](#)

Changes to System Options and Document Properties

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

System Options

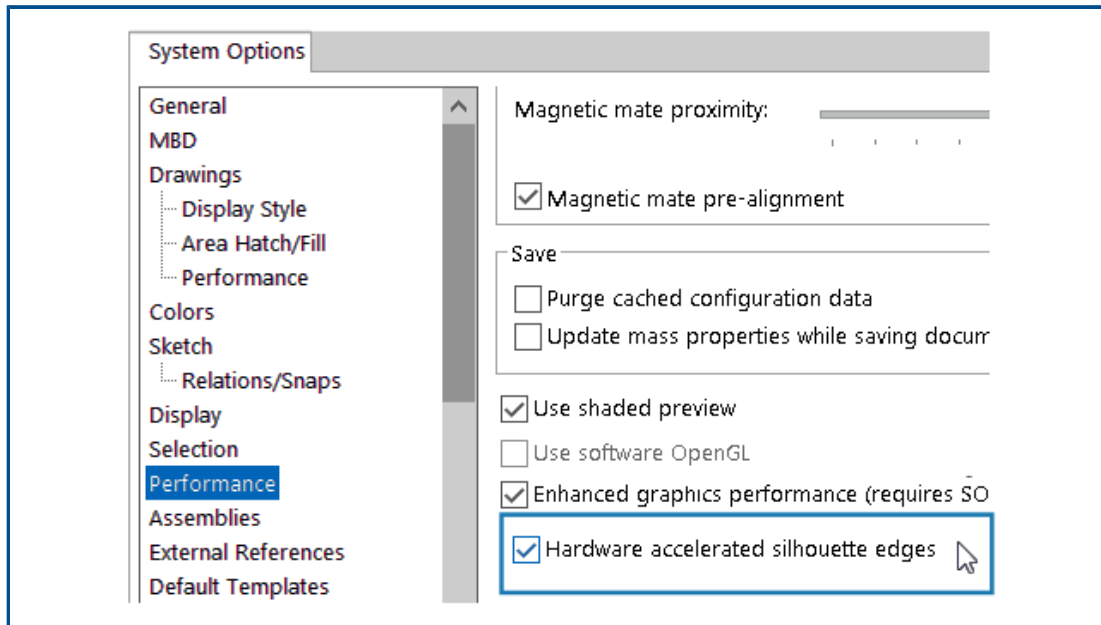
Option	Description	Access
Opposite hand mirror components	Defines default values for Add Prefix and Add Suffix when creating opposite-hand components.	Assemblies
Prefix for virtual components created from external files	Defines a default prefix for virtual components that are created from external files.	Assemblies
Display DimXpert dimensions on top of model	Controls the visibility of dimensions.	Display
Drawings, Overridden dimensions	Specifies a color for overridden dimensions.	Colors
Hardware accelerated silhouette edges	Enables the GPU hardware to improve the display of silhouette edges in HLR, HLV, and wireframe view modes.	Performance
Preview sketch dimension when selected	Turns on sketch dimension previews.	Sketch

Option	Description	Access
Always open a drawing in detailing mode	Opens a drawing by default in Detailing mode.	Drawings > Performance
Defeature Rule Sets	Under Show folders for , specifies a location for defeature rule sets, *.slddrs, and related log files.	File Locations

Document Properties

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

Accelerate the Display of Silhouette Edges



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

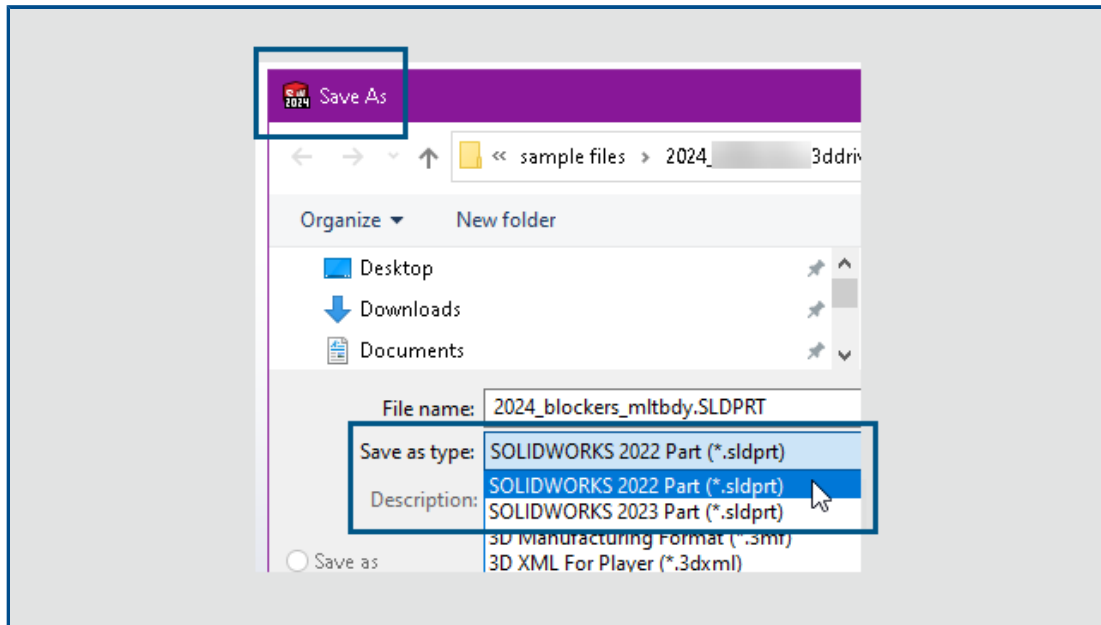
In **Tools > Options > System Options > Performance**, select **Hardware accelerated silhouette edges**.

Application Programming Interface

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

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

Saving SOLIDWORKS Documents as Previous Versions



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

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

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

Workflow

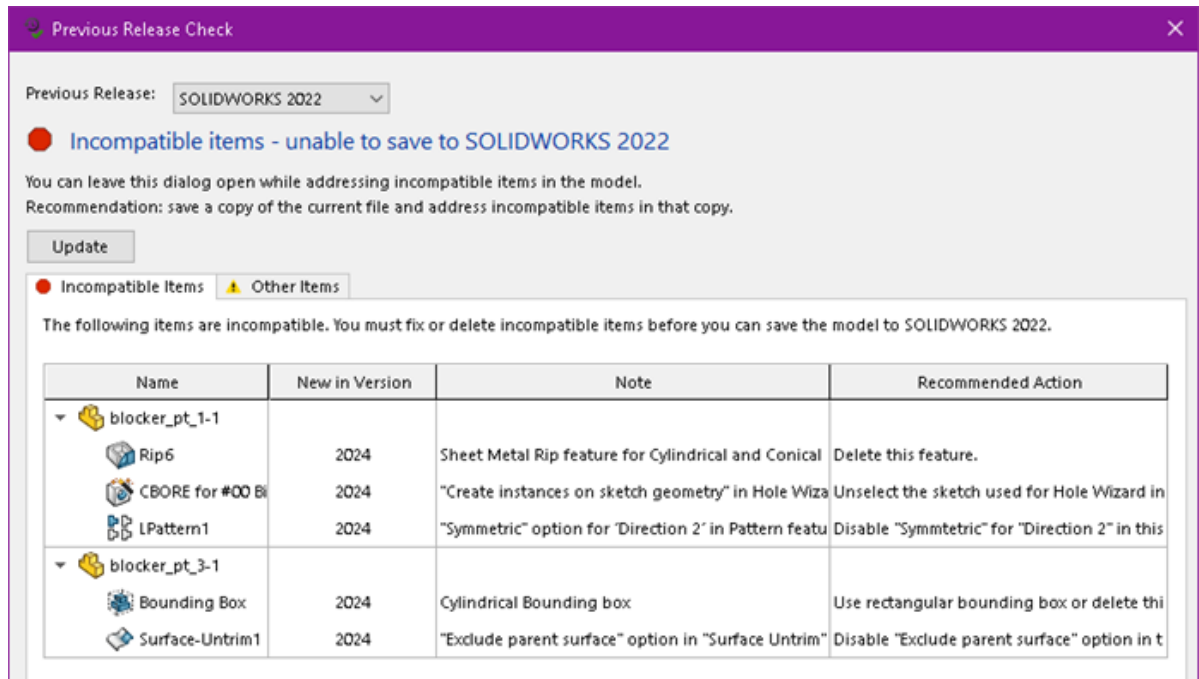
You must manually address incompatible items in this process. Incompatible items, as described in the table below, are items that do not exist or are not supported in the selected previous release.

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

To save a SOLIDWORKS document as a previous version:

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

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



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

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

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

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

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

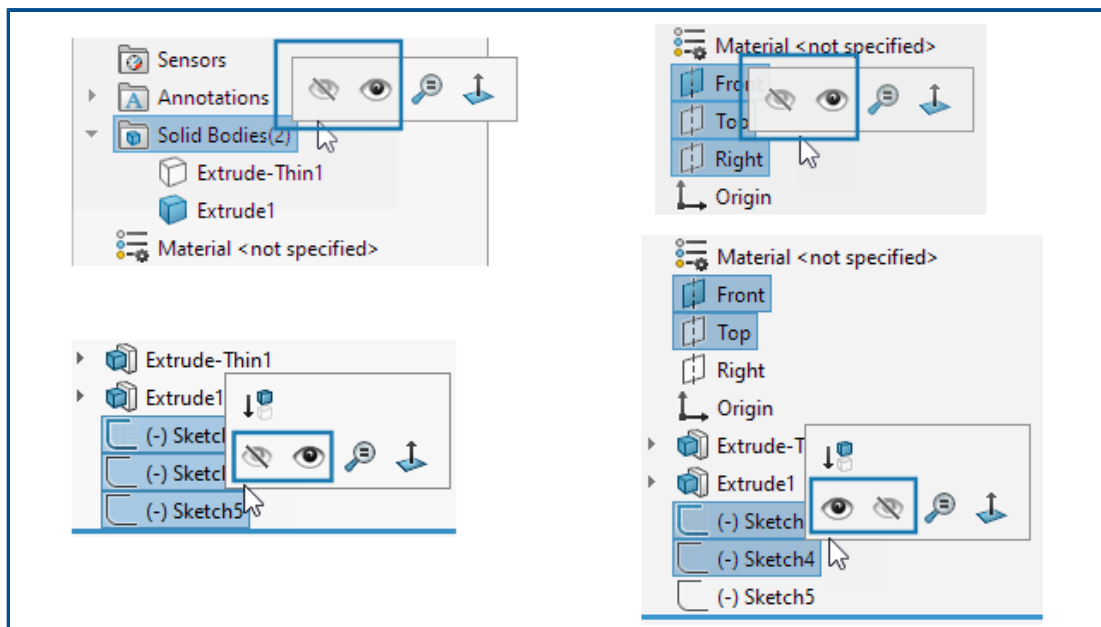
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User Interface





This chapter includes the following topics:

- **Hide and Show**
- **Usability**
- **Icon Updates for Open, Save, and Properties Commands**

Hide and Show



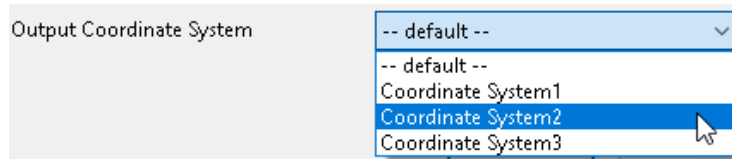
When you multiselect bodies, planes, or sketches that have a combination of shown and hidden states in the FeatureManager® design tree, the context toolbar shows both the **Hide**  and **Show**  tools. You can click **Hide** or **Show** to change the visibility state of all the selected entities.

The **Hide** and **Show** tools also appear when you multiselect a combination of hidden and shown planes and sketches. The **Show Hidden Bodies**  tool is added to the **Tools > Customize > Commands > Features** tab so you can add it to toolbars and the CommandManager. You can use the **Search**  tool or the **S** key to find **Show Hidden Bodies**  and **Show Hidden Components** .

Usability

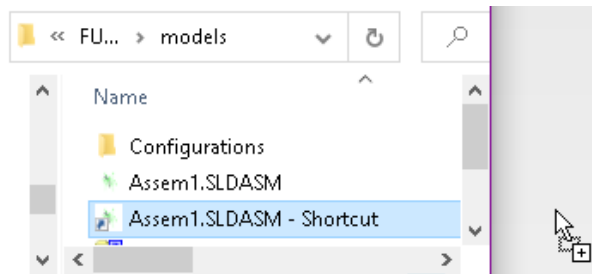
The user interface is enhanced to improve productivity.

Coordinate System to Save



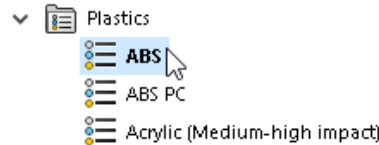
In the Save As dialog box, you can choose which coordinate system to save with a file. In the dialog box, in **Output Coordinate System**, specify the coordinate system to save. When you open the file, the new coordinate system is the origin.

Opening SOLIDWORKS Files from Shortcuts



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

Selecting Materials















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

Icon Updates for Open, Save, and Properties Commands

Tool icons are updated for Open, Save, and Properties commands for SOLIDWORKS and SOLIDWORKS 3DEXPERIENCE apps.

Tool	2023	2024	Change
Open			Arrow color
Open Drawing			Arrow color
Save			Removed label lines and modernized

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

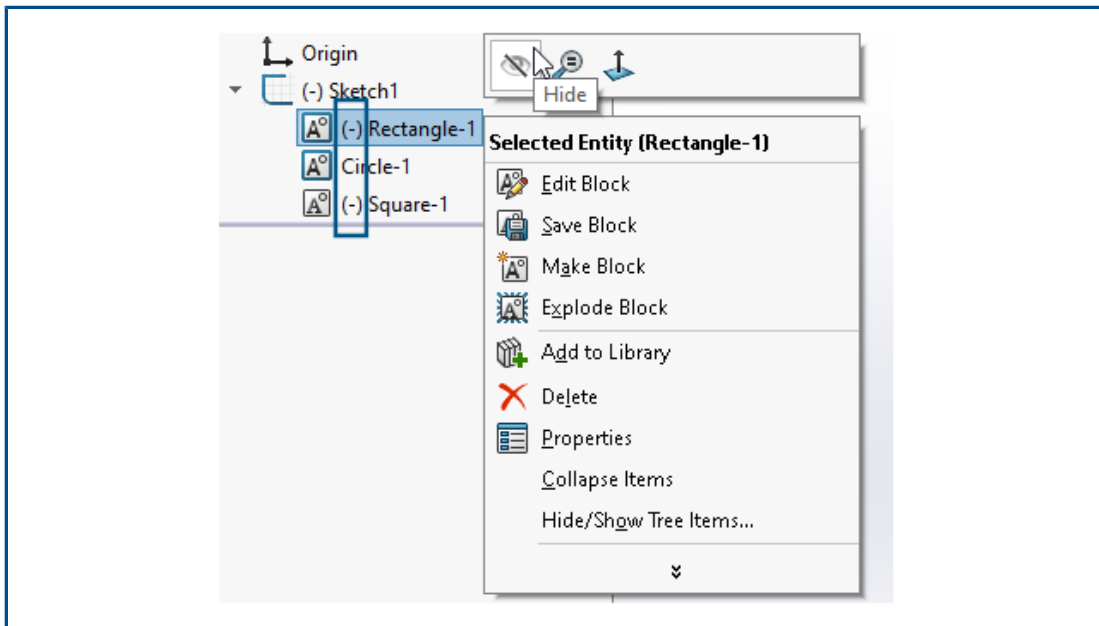
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Sketching

This chapter includes the following topics:

- **Sketch Blocks**
- **Sketch Dimension Previews**

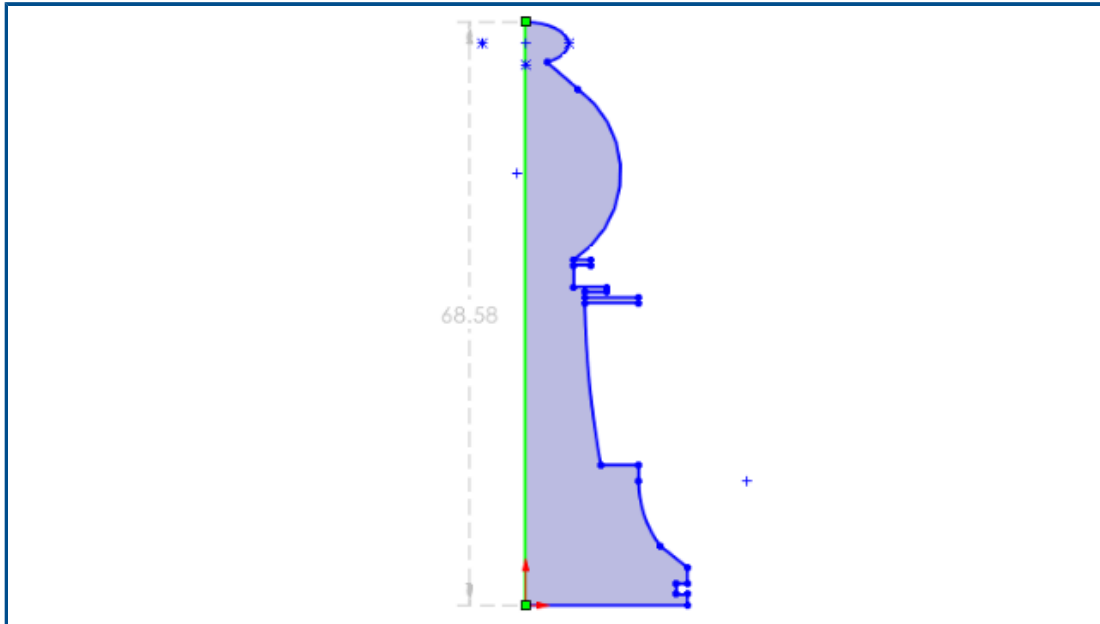
Sketch Blocks



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

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

Sketch Dimension Previews



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

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

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

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

Sketch dimension previews are not supported for path lengths.

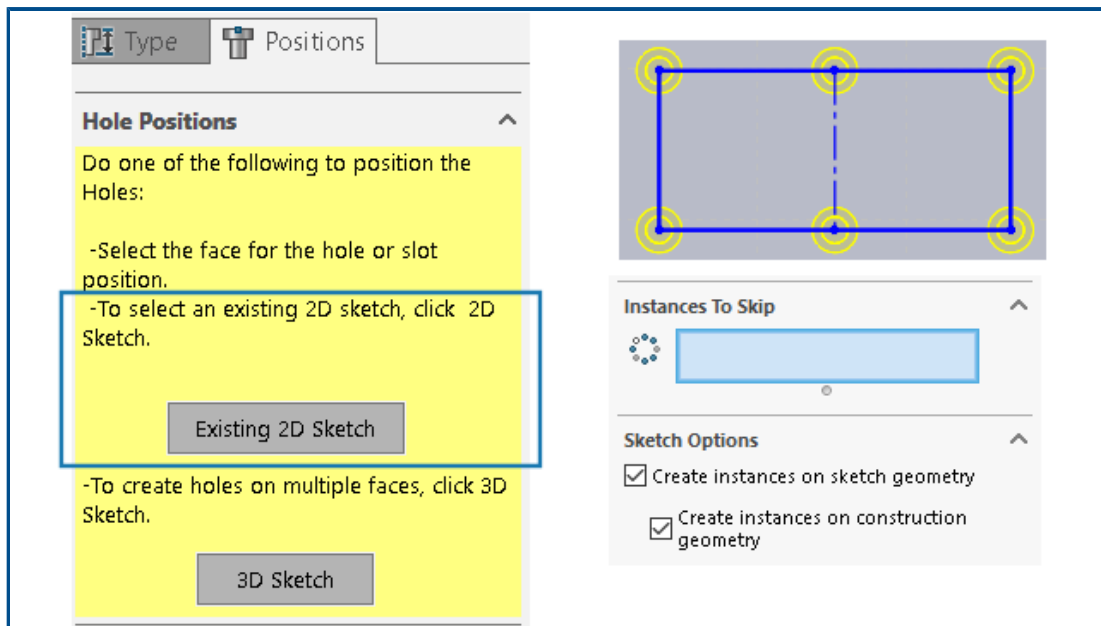
6

Parts and Features

This chapter includes the following topics:

- **Hole Wizard**
- **Making Multibody Parts from Assemblies**
- **Body Transparency for Combine Features**
- **Cylindrical Bounding Boxes**
- **Excluding Parent Surfaces in Untrim Features**
- **Flip Side to Cut for Cut Revolves**
- **SelectionManager for Projected Curves**
- **Stud Wizard**
- **Symmetrical Linear Patterns**

Hole Wizard



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

Under **Hole Positions**, you can click **Existing 2D Sketch** and select an existing 2D sketch to position and automatically create the holes at all endpoints, vertices, and points

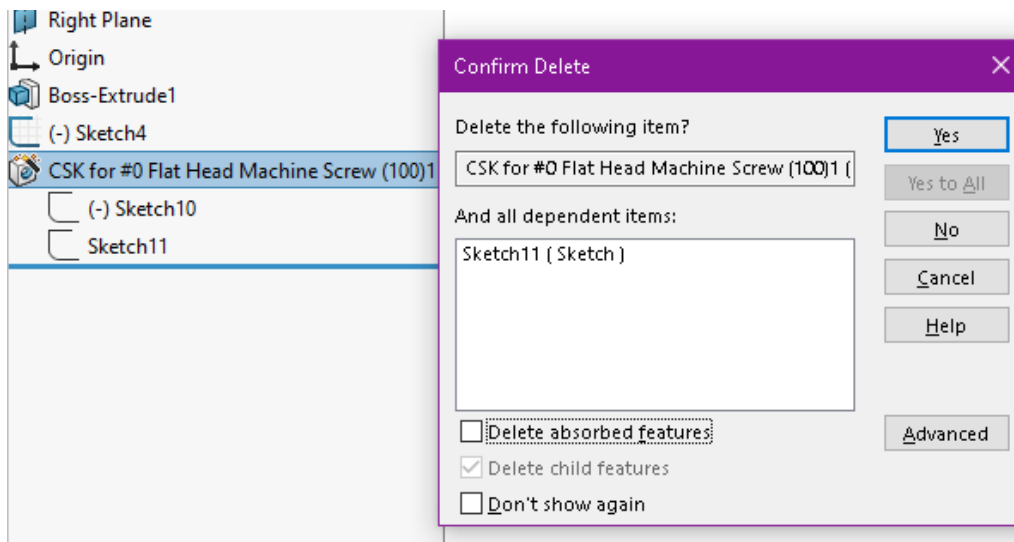
of the sketch geometry. You can select sketch entities like lines, rectangles, slots, and splines. **Sketch Options** specify the geometry used to automatically create the instances.

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

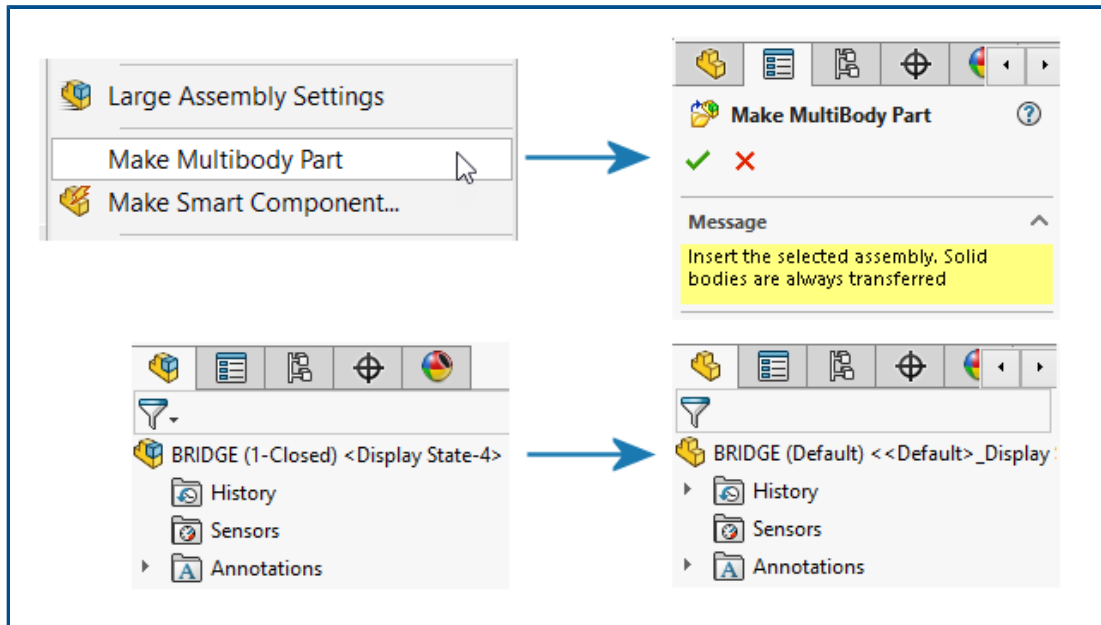
- **Create instances on sketch geometry** (Enabled by default). Positions holes at all endpoints, vertices, and points of the sketch geometry.
- **Create instances on construction geometry**. Positions holes at all endpoints, vertices, and points of the construction geometry.


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

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




Making Multibody Parts from Assemblies

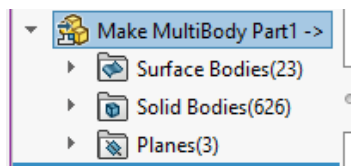


The **Make Multibody Part**  tool converts an entire assembly into a separate, single multibody part that is linked to the parent assembly.

The parent assembly reflects all features that you create in the linked multibody part. Downstream platform applications also support these features so you can perform post-assembly operations on the multibody part, such as material removal.

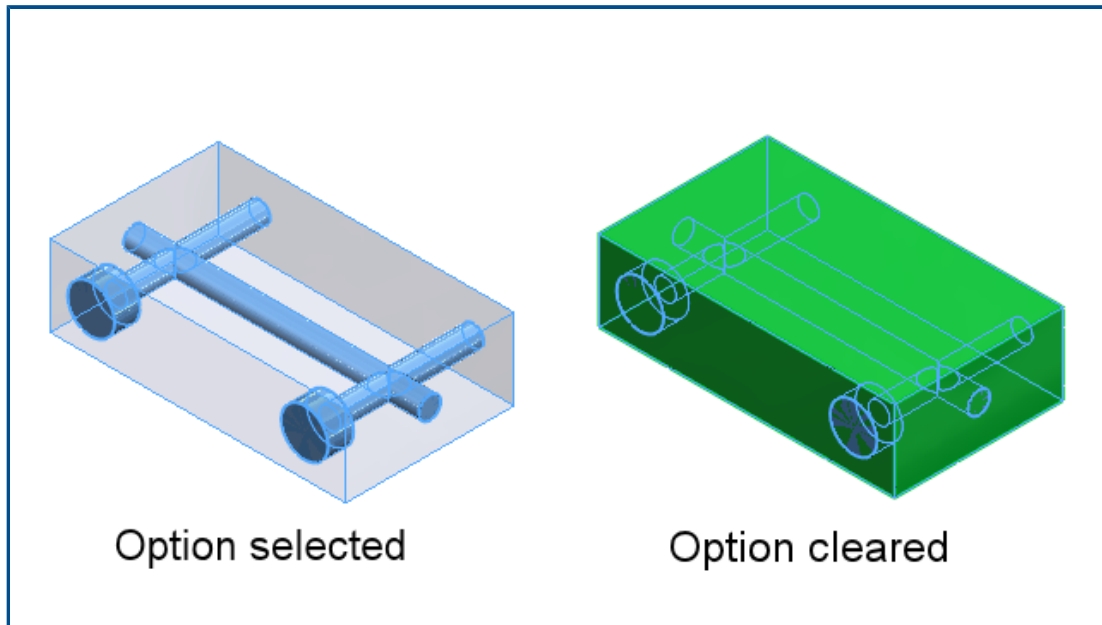
To create a multibody part, in an assembly, click **Tools > Make Multibody Part**.

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



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

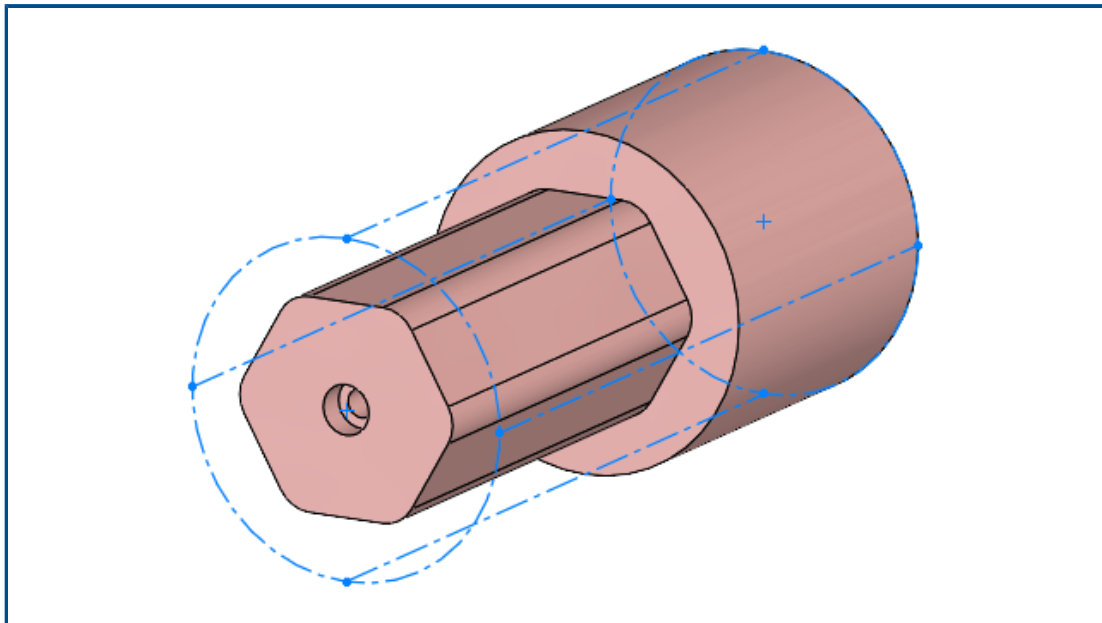
Body Transparency for Combine Features



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

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

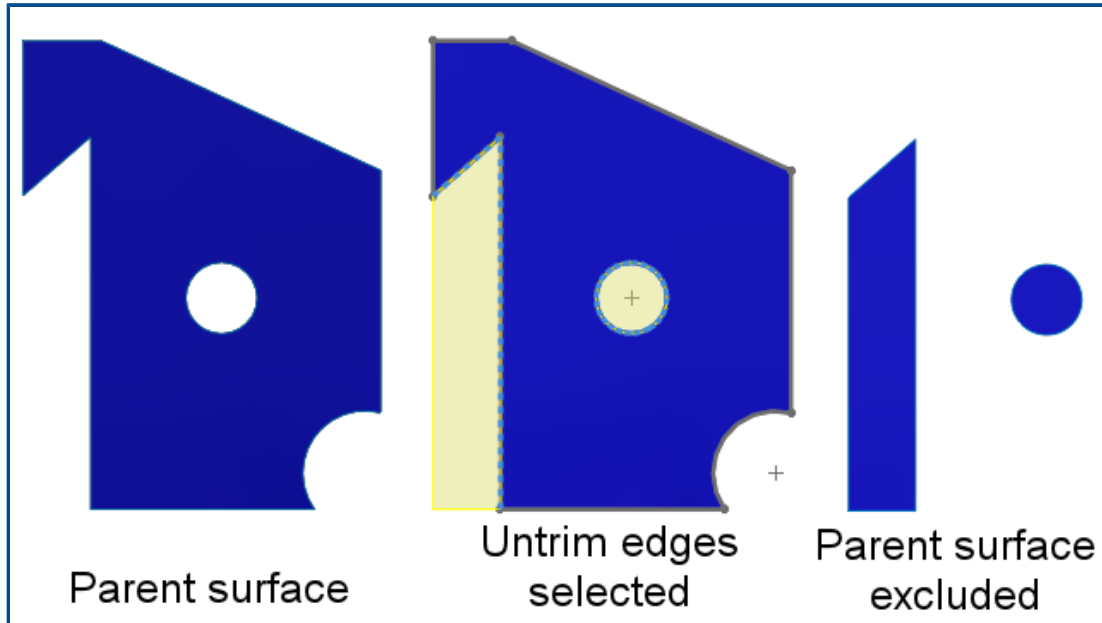
Cylindrical Bounding Boxes



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

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

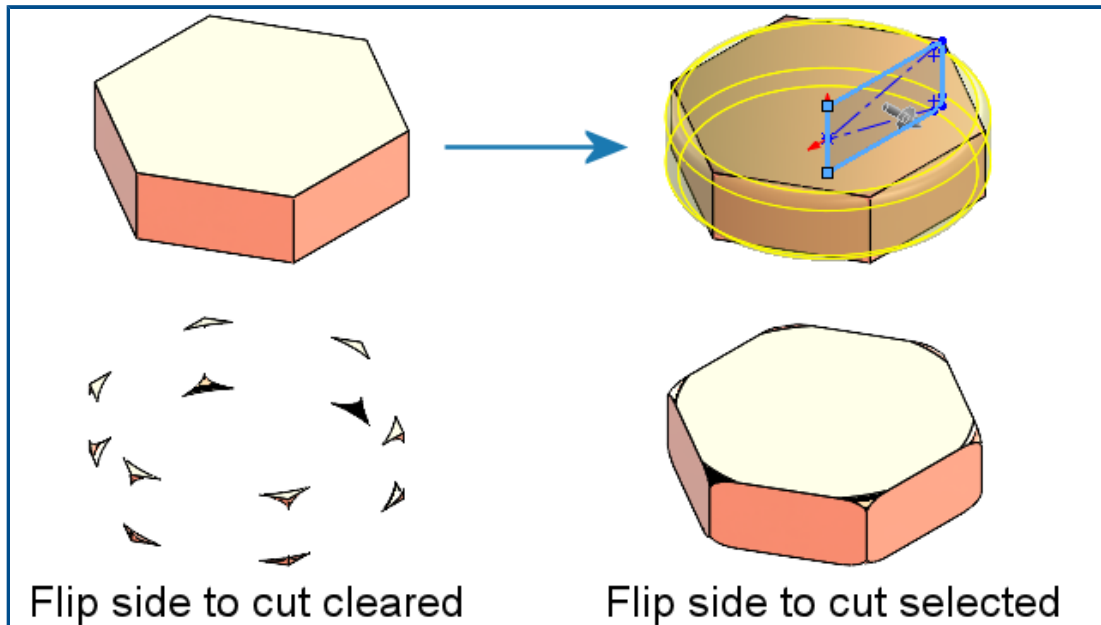
Excluding Parent Surfaces in Untrim Features



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

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

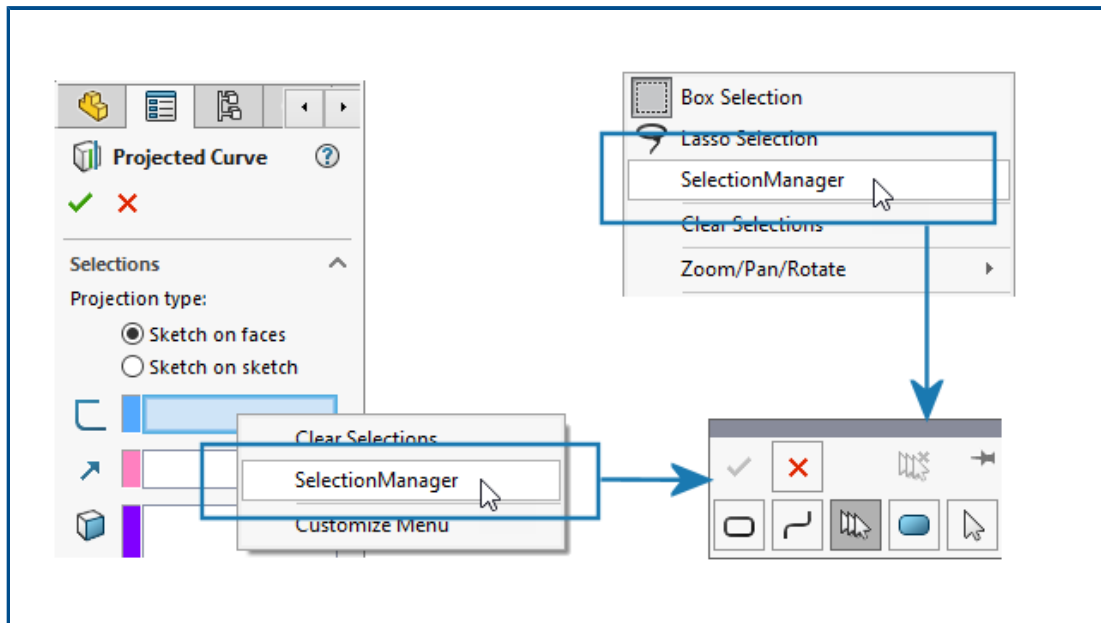
Flip Side to Cut for Cut Revolves



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

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

SelectionManager for Projected Curves



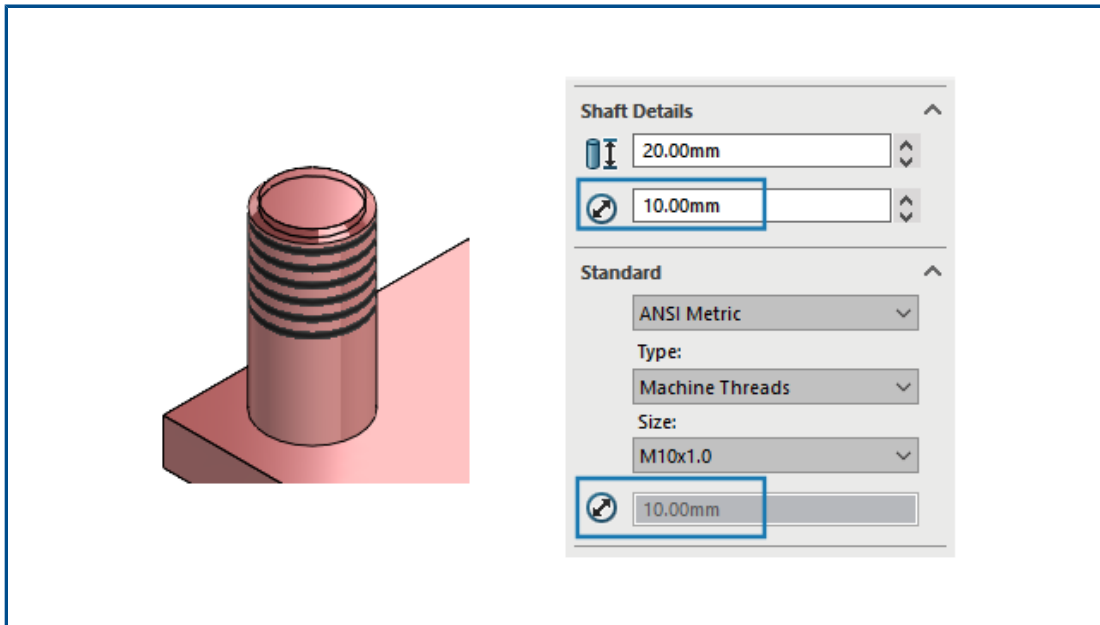
In the Projected Curve PropertyManager or if you right-click in the graphics area, you can use the SelectionManager to select portions of sketches to create projected curves.

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

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

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

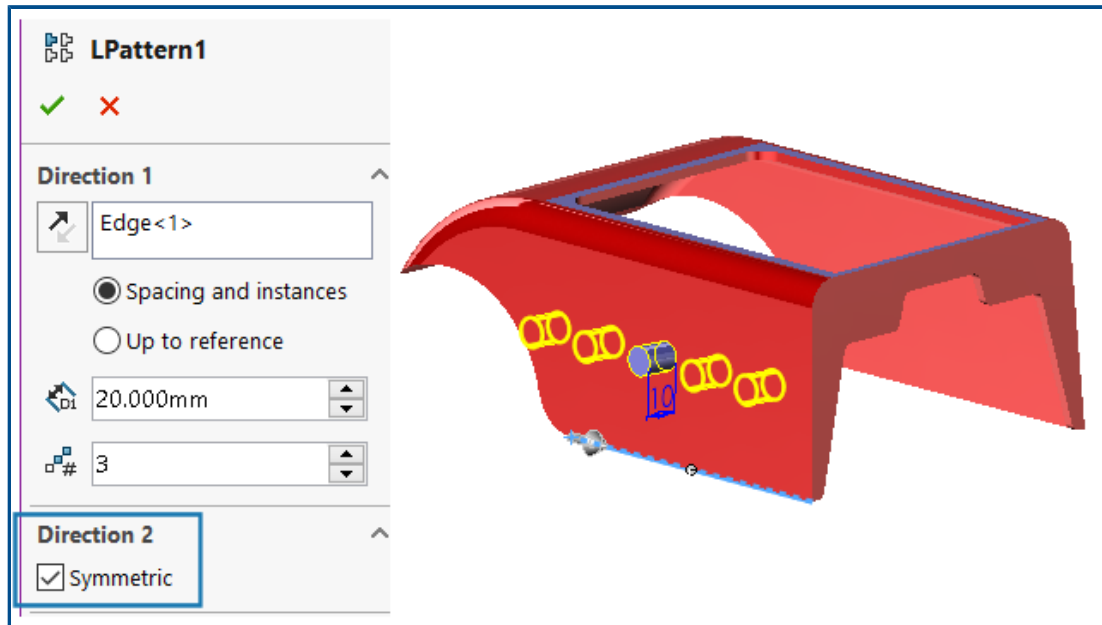
Stud Wizard



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

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

Symmetrical Linear Patterns



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

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

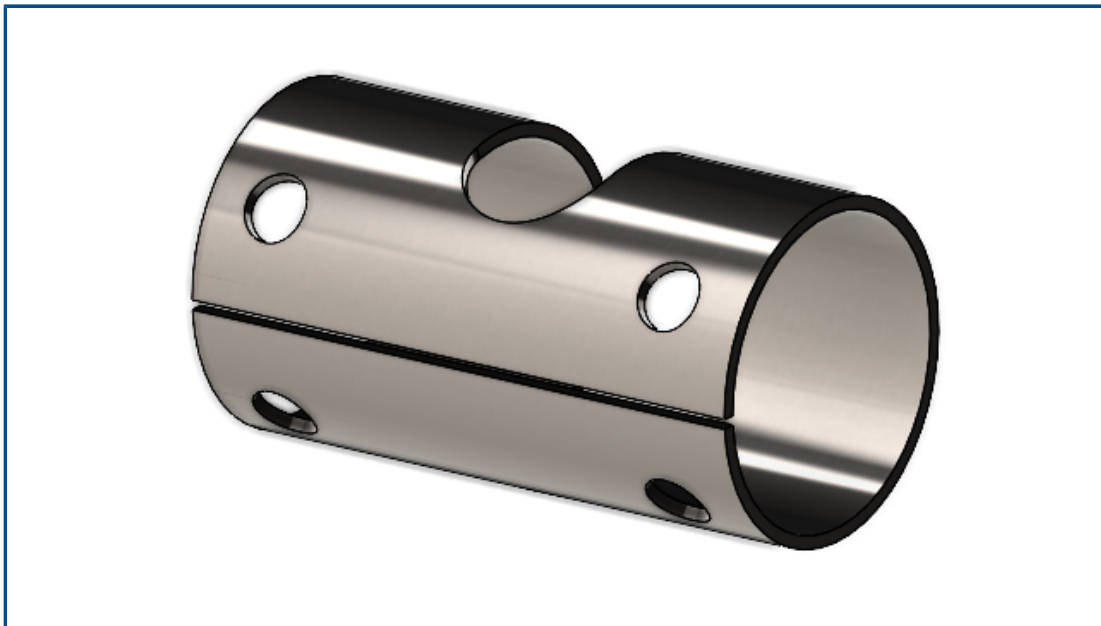
7

Sheet Metal

This chapter includes the following topics:

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

Rip Tool



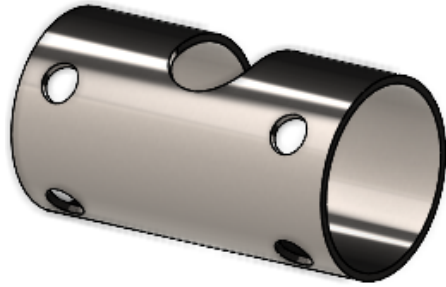
You can use the **Rip** tool to create rips in hollow or thin-walled cylindrical and conical bodies. By selecting an edge on a cylindrical or conical face, you can flatten the part as sheet metal.

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

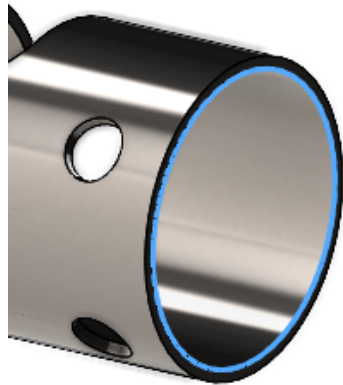
SOLIDWORKS supports straight cuts only, not slanted cuts.

To use the rip tool in a cylindrical part:

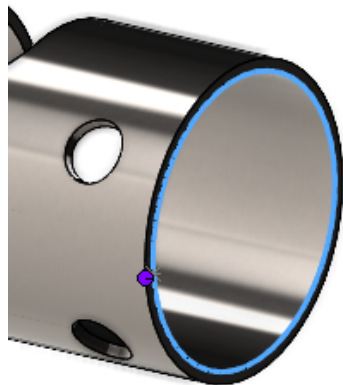
1. In a hollow or thin-walled cylindrical or conical part, click **Rip**  (Sheet Metal toolbar).



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

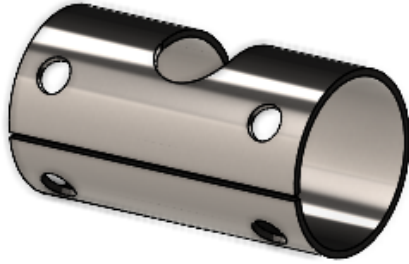



- b. A reference point on the model.



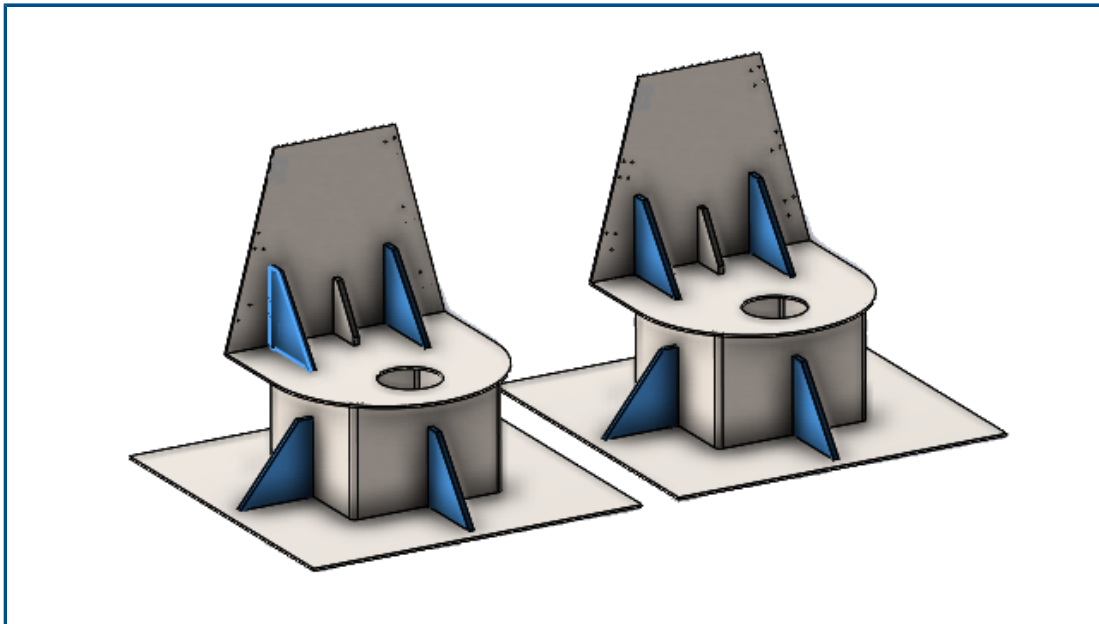
The reference point can be on the model or anywhere in the graphics area. If you select a reference point that is not on the model, the software projects the point onto the model.

3. Specify options in the PropertyManager and click ✓.



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

Slot Propagation



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

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

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

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

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

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

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


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

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

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

4. Click .

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

1. In an assembly with a component that has a tab and slot, right-click the component and click **Propagate Slots**.
2. In the Slot Propagation PropertyManager, under **Instances for slot propagation**, specify an option:
 - **Only selected.** Propagates slots to the selected component only.
 - **All instances in same parent assembly.** Propagates slots to all instances of the selected component that are in the same parent assembly.
 - **All instances.** Propagates slots to all instances of the selected component.
3. Click .

Slot Propagation PropertyManager

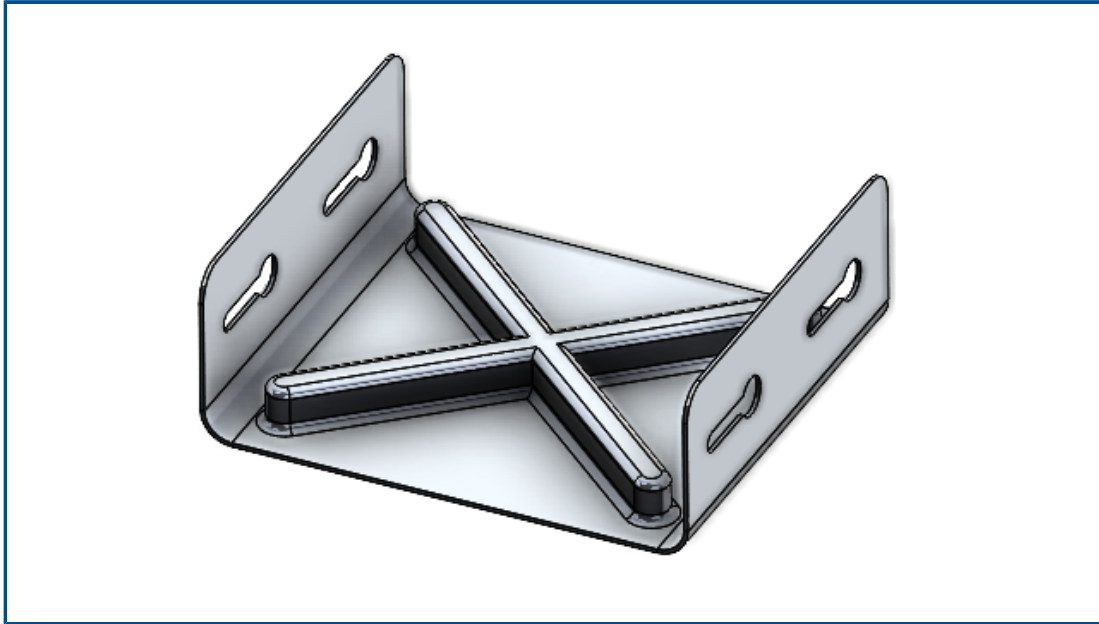
To open this PropertyManager:

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

Selection

Propagate slots for these component(s)	Lists the components to apply the slots to.
Instances for slot propagation	<p>Specifies which components to propagate slots to:</p> <ul style="list-style-type: none"> • Only selected. Propagates slots to the selected components. With this option, you can delete specific components from the list. • All instances in the same parent assembly. Propagates slots to all instances of the selected components that are in the same parent assembly. • All instances. Propagates slots to all instances of the selected components. With this option, if some components already have a slot, they are ignored.

Stamp Tool




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

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

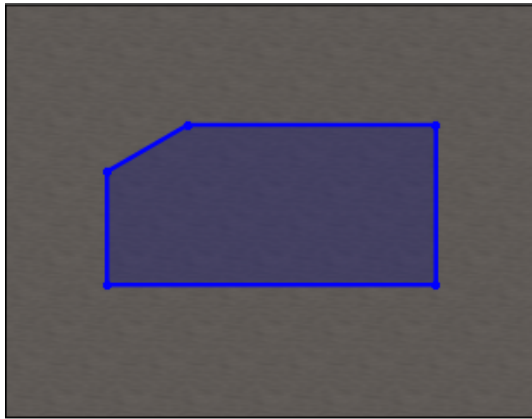
Using sketches to create forming tools is a faster way to apply forming tools to sheet metal parts. The **Stamp** tool allows more flexibility to experiment with different designs and parameters.


Using the Stamp Tool

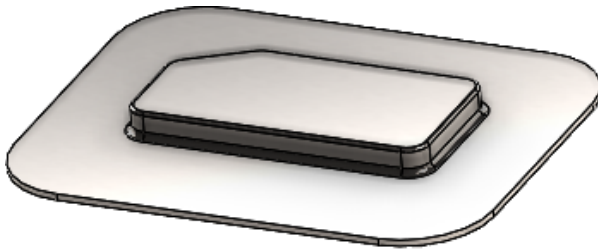
To use the stamp tool:

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

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



3. In the PropertyManager, specify options and click .





Stamp PropertyManager

To open this PropertyManager:

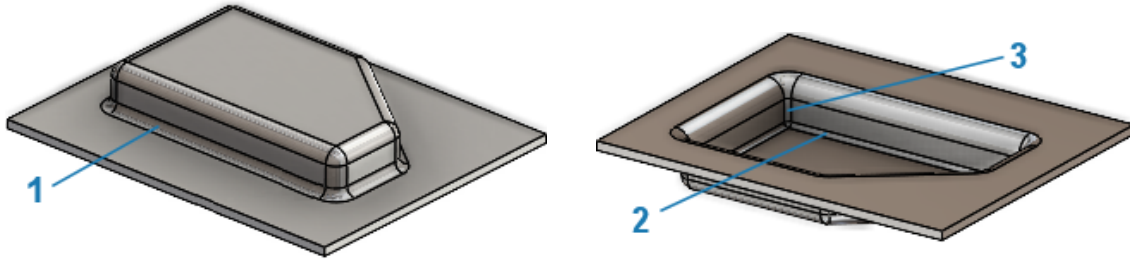
1. In a sheet metal part, click **Stamp** (Sheet metal toolbar) or **Insert > Sheet Metal > Stamp**.




Stamp Parameters

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

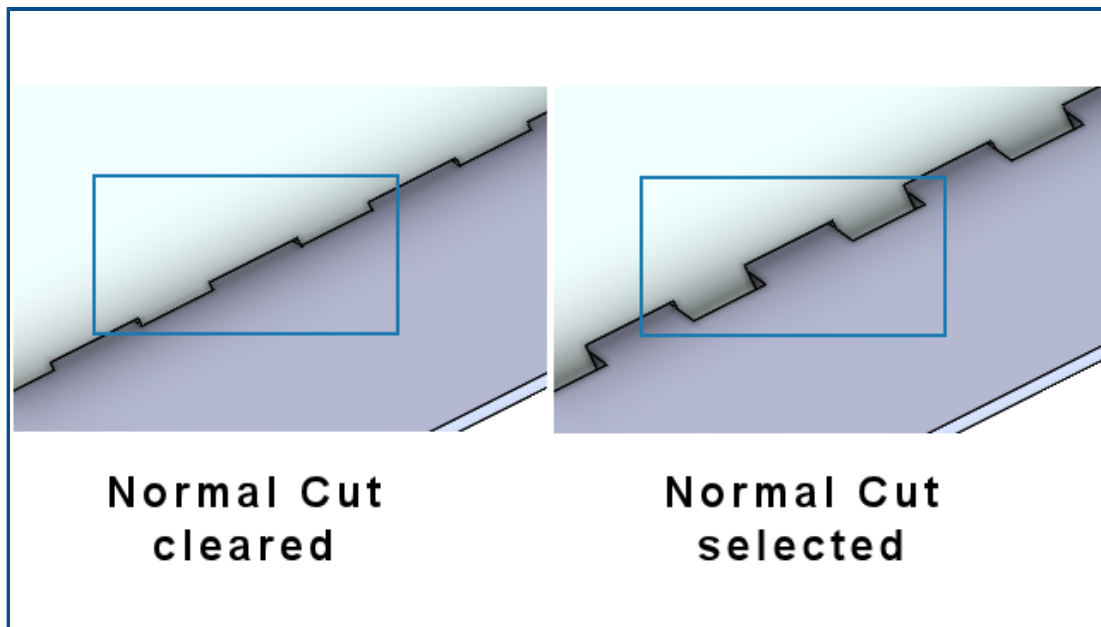
Fillet

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



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

Normal Cut in Tab and Slot



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

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

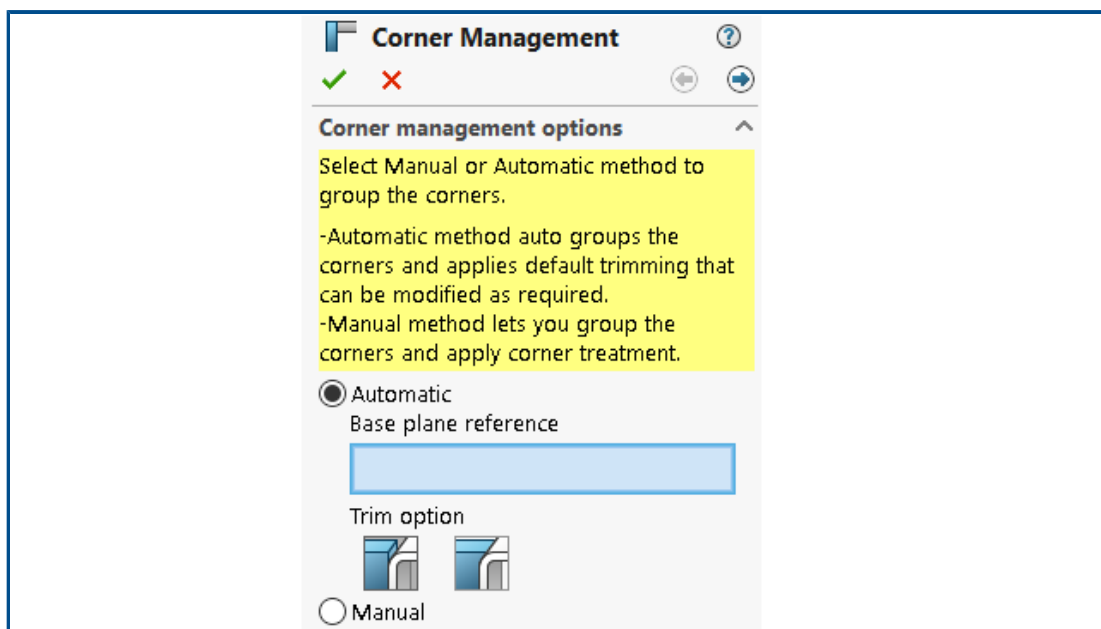
8

Structure System and Weldments

This chapter includes the following topics:

- **Corner Management**
- **Displaying Units in File Properties**
- **Structure System**

Corner Management




You can apply corner treatments manually or automatically.

To open the Corner Management PropertyManager:

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

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





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


Two Member PropertyManager

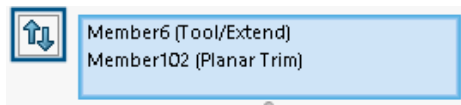
The user interface of the Two Member PropertyManager is enhanced.

Enhancements include:




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

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

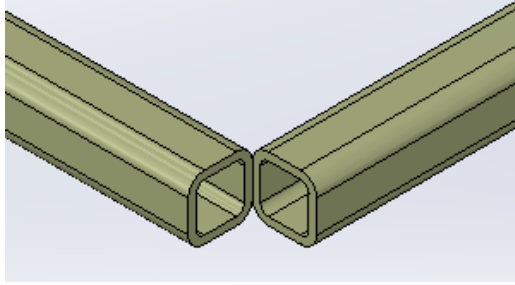
- You can use the **End Butt1** and **End Butt2** trim options for swapping. Previously, you could swap the tool and body to trim using the arrows .



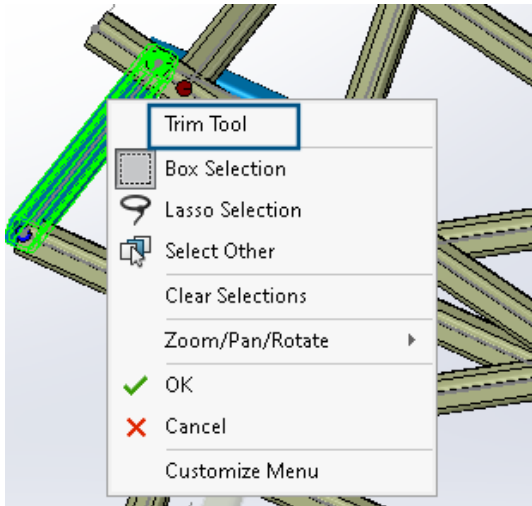
- Updated icons:

Icon	Trim Option
	Planar Trim
	Body Trim
	Miter Trim

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



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

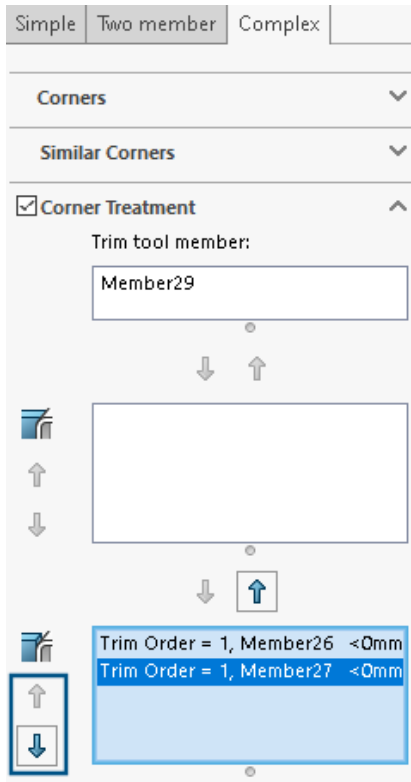


- In the PropertyManager, for **Trim Tool**, you can select **Automatic** or **User Defined**. The **User Defined** option lets you select a face or a plane to trim.

Complex Corner PropertyManager

The user interface of the Complex Corner PropertyManager is enhanced.


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



Editing the Corner Management Options

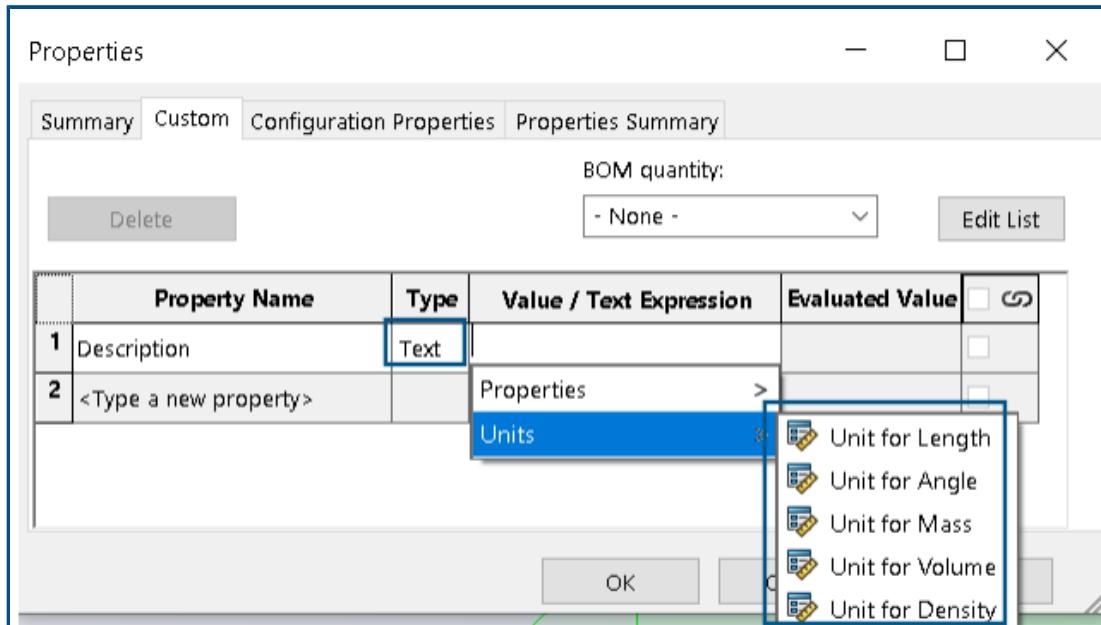
You can modify the corner treatment.

To edit the corner management options:

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


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

Displaying Units in File Properties



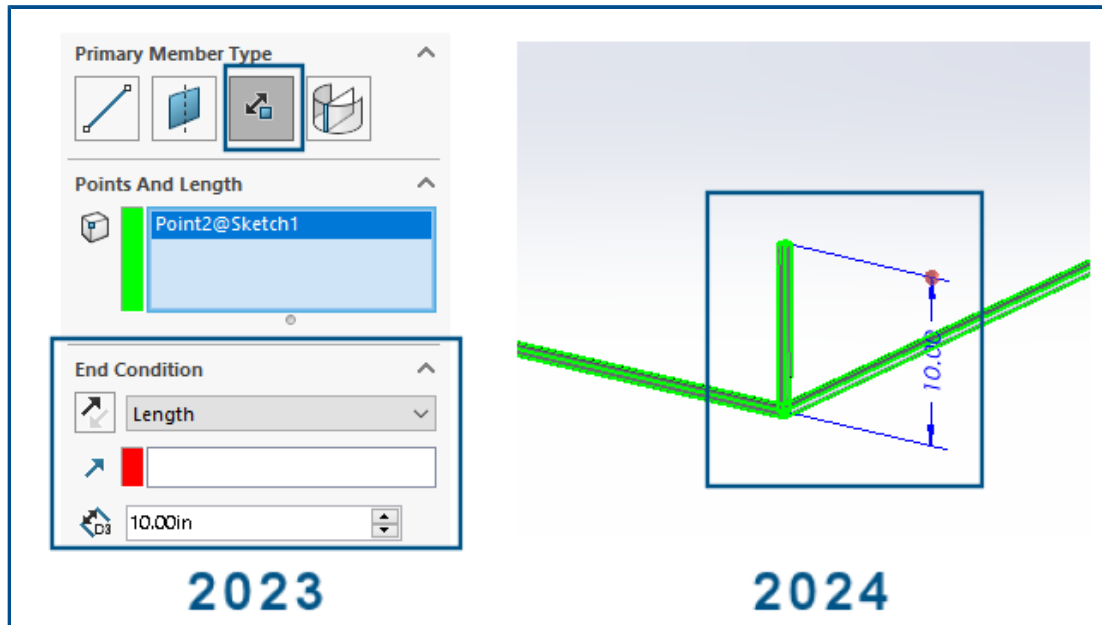
You can capture and display the units for the **Text** type of file properties.

To display units in file properties:

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

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

Structure System



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

- When editing the structure system in the graphics area, you can change the length of the point length member.
To change the length, double-click the member and click the dimensions. Previously, you had to edit the length of the point length member from the Primary Member PropertyManager.
- You can use corner management for profiles of less than 2mm.

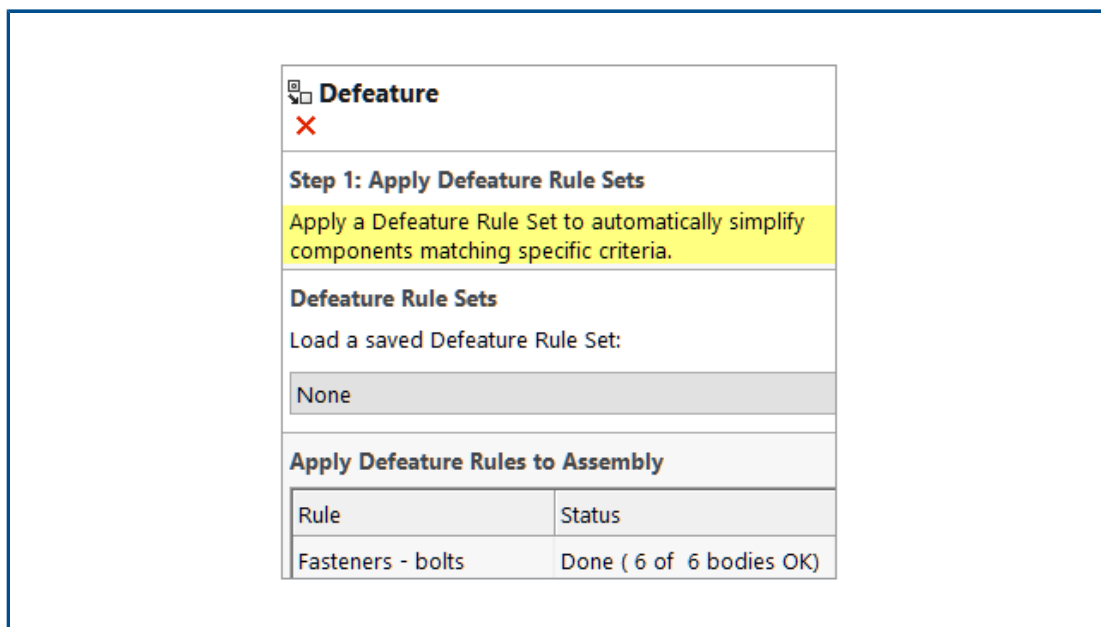
9

Assemblies

This chapter includes the following topics:

- **Defeature Rule Sets**
- **Propagating Visual Properties in Defeature Groups**
- **Repairing Missing References in Linear or Circular Component Patterns**
- **Mate References**
- **Auto-Repair for Missing Mate References**
- **Assigning Component References to Top-Level Components**
- **Specifying a Prefix and Suffix for Components**

Defeature Rule Sets



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

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

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

Specifying a File Location for Defeature Rule Sets

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

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




To specify a file location for defeature rule sets:

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

Creating Defeature Rule Sets

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

To create a defeature rule set:

1. Open a model, and click **Defeature**  (Tools toolbar) or **Tools > Defeature**.
2. In the PropertyManager, select **Silhouette** .
3. Click **Next** .
4. Under **Apply Defeature Rules to Assembly**, click **Edit Rules**.
5. In the Defeature Rules Editor dialog box, under **Name**, enter a name.

Defeature Rules Editor						New rule set
List of Rules:						
	Name	Selection Criteria	Defeature Type	Enclose in one ...	Defeature Ori...	Visual prop...
1	Click to a...	Define selection cr...	Choose type	Off	Choose orientat...	Propagate

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


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

Advanced Component Selection					
Define Search Criteria					
	And/Or	Category1	Category2	Condition	Value
1		File Type		=	Fastener
2	And	Document name -- SW Sp		contains	bolt

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

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

Defeature Rules Editor						New rule set
List of Rules:						
	Name	Selection Criteria	Defeature Type	Enclose in one ...	Defeature Ori...	Visual prop...
1	Fastener	File Type = "Faste...	Cylinder	Off	Automatic	Propagate

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

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

Apply Defeature Rules to Assembly	
Rule	Status
Fasteners - bolt	Pending

- Click **Apply**.

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

Apply Defeature Rules to Assembly	
Rule	Status
Fasteners - bolts	Done (6 of 6 bodies OK)

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

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

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

```

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

```

Defeature - Apply Defeature Rule Sets PropertyManager

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

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

To open the Defeature - Apply Defeature Rule Sets PropertyManager:

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

2. In the PropertyManager, select **Silhouette** .
3. Click **Next**  until the **Apply Defeature Rule Sets** page appears.

Defeature Rule Sets

Load a saved Defeature Rule Set

Specifies the rule set to load.
None displays when there are no loaded rule sets. Saved rule sets display in the list.

To specify the file location for the saved rule set, click **Tools > Options > System Options > File Locations**. Under **Show folders for**, select **Defeature Rule Sets**. Click **Add** to specify a location.



Apply Defeature Rules to Assembly




Rule	Lists the rules.
Status	<p>Displays the results of applying the rule:</p> <ul style="list-style-type: none"> • Pending. Displays when the rule is not applied or when an existing rule is modified but not reapplied. • Done (x of y bodies OK). After applying the rule, displays the number of components processed, x, and the number of components, y that meet the criteria.
Apply	<p>Applies all rules to the model in the order that the rules are listed. The defeatured geometry generates and a preview displays in the graphics area.</p> <p>After a rule is applied to a component, no other rules are applied to that component.</p> <p>After saving the model as a part, the defeature components display in the FeatureManager design tree.</p> <p>The log file includes a list of components with a status of OK where components are defeatured or Failed where components are not defeatured.</p> <div style="border: 1px solid gray; padding: 5px; margin-top: 10px;"> <p>Rules apply to part level components. Rules do not apply to subassemblies.</p> </div>
Clear	Removes all the rules and deletes the simplified geometry applied to the model.
Edit Rules	Opens the Defeature Rules Editor dialog box.
Save log	Saves the log file.

Defeature Rules Editor Dialog Box

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

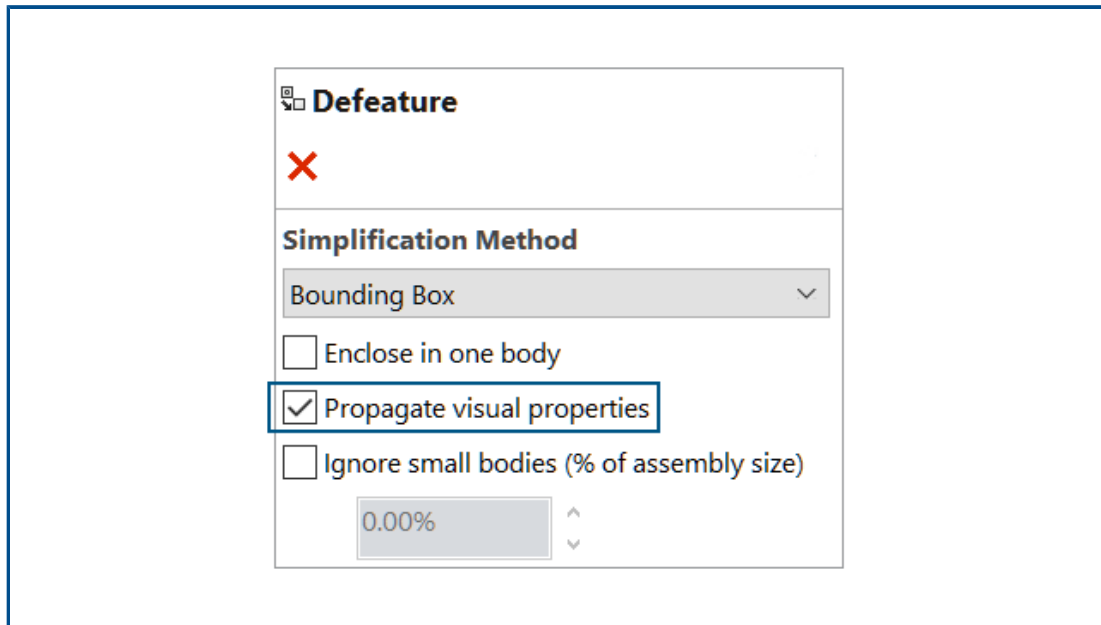
To open the Defeature Rules Editor dialog box:

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

	New	Creates a new rule set.
	Open	Opens an existing rule set.
	Save	Saves the rule set in a Defeature Rule Set file, .slddrs.
	Name	Specifies a name for the rule set.
	Selection Criteria	<p>Displays the selection criteria. For a new rule, click Define selection criteria to open the Advanced Component Selection dialog box where you define the selection rules.</p> <p>To modify a rule, click the selection criteria for the rule. Under Rule Definition, click Selection Criteria.</p> <div style="border: 1px solid gray; padding: 5px; margin-top: 10px;"> <p>In the Advanced Component Selection dialog box, the following functionality is not available when you open the dialog box from the Defeature PropertyManager:</p> <ul style="list-style-type: none"> • Manage Searches tab • Name of Search • Apply </div>


Defeature Type	<p>Specifies a simplification method:</p> <p>Bounding Box Creates a cuboid bounding box.</p> <p>Cylinder Creates a cylinder derived from the dimensions of a cuboid bounding box.</p> <p>Polygon Outline Creates an extruded polygon that fits around the outline of the selected bodies and components.</p> <p>Tight Fit Outline Creates an extruded body by using the outlines of the selected bodies and components.</p> <p>None (Copy Geometry) Creates an exact copy of the selected bodies and components.</p>
Enclose in one body	<p>Creates a single body that includes the specified components.</p> <ul style="list-style-type: none"> • Off • Per part • Entire group <div style="border: 1px solid gray; padding: 5px; margin-top: 10px;"> <p>When you select Cylinder or None (Copy Geometry) for Defeature Type, Enclose in one body is set to Off.</p> </div>
Defeature Orientation	<p>Specifies a defeature orientation:</p> <ul style="list-style-type: none"> • Automatic • Component XY • Component YZ • Component XZ • Global XY • Global YZ • Global XZ
Visual properties	<p>Propagate Includes appearances and textures in the defeatured model.</p> <p>Don't propagate Omits appearances and textures from the defeatured model.</p>
Rule Definition	<p>Displays the selected rule. Click Selection Criteria to modify the rule.</p>

Propagating Visual Properties in Defeature Groups

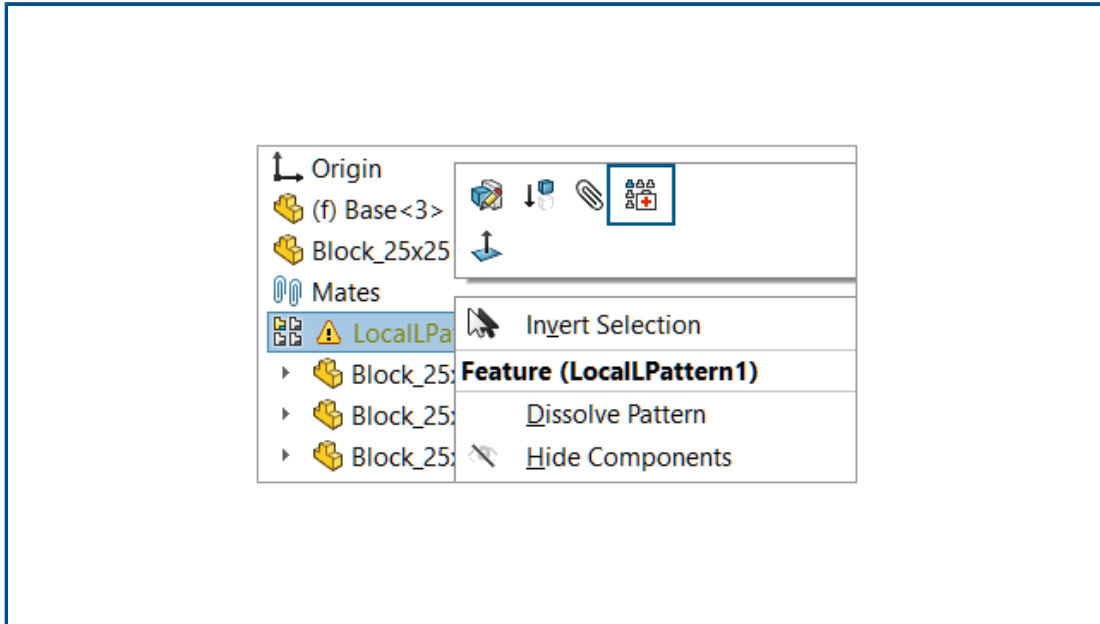


You can include appearances and textures in a defeatured group.

To propagate visual properties in defeature groups:

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


Repairing Missing References in Linear or Circular Component Patterns




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

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

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

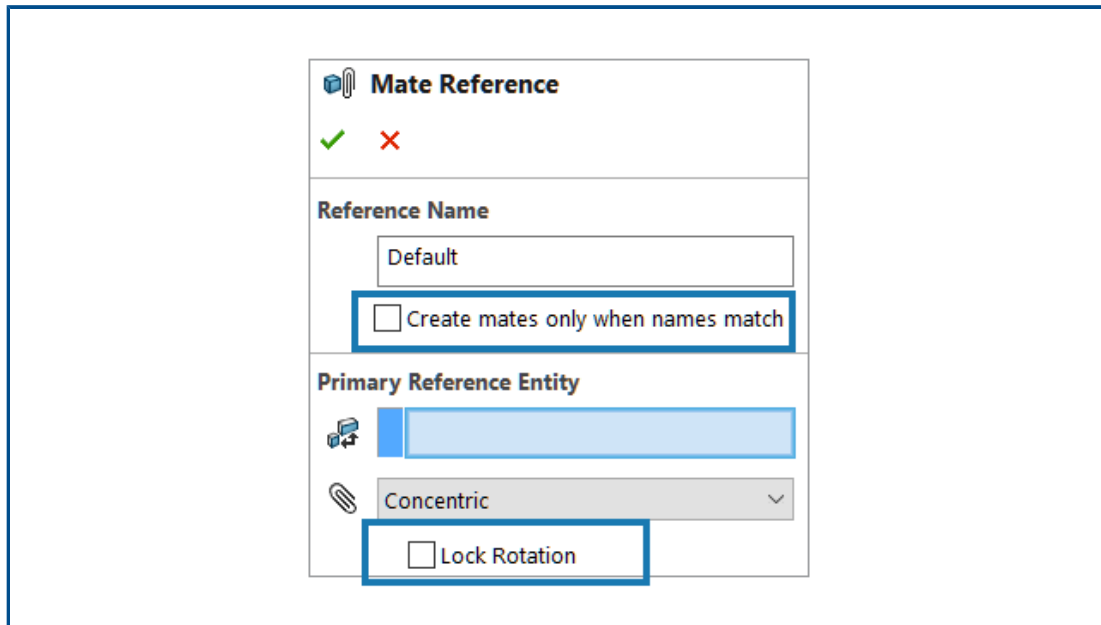
You cannot use **Auto Repair**  in Large Design Review mode.

To repair missing references in linear or circular component patterns:

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


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

Mate References



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

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



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

The dialog box can appear when using these workflows:

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

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

To create mates only when the names match:

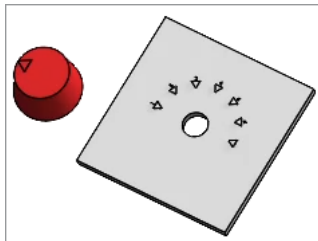
1. Open a model with a mate reference where the name of the mate reference is different for each component.
2. Open one of the components from the mate reference.
3. In the FeatureManager design tree for the component, under the **Mate References**  folder, right-click a mate reference  and click **Edit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

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

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

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

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

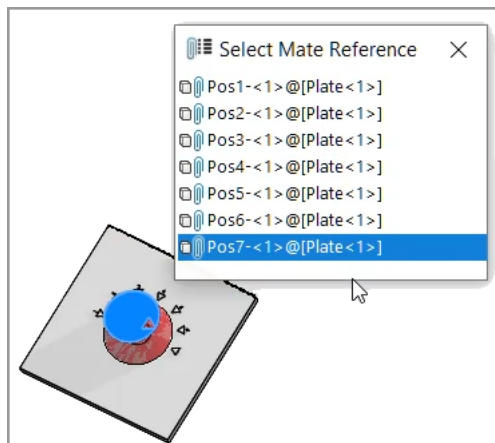


2. Drop the knob over the plate.

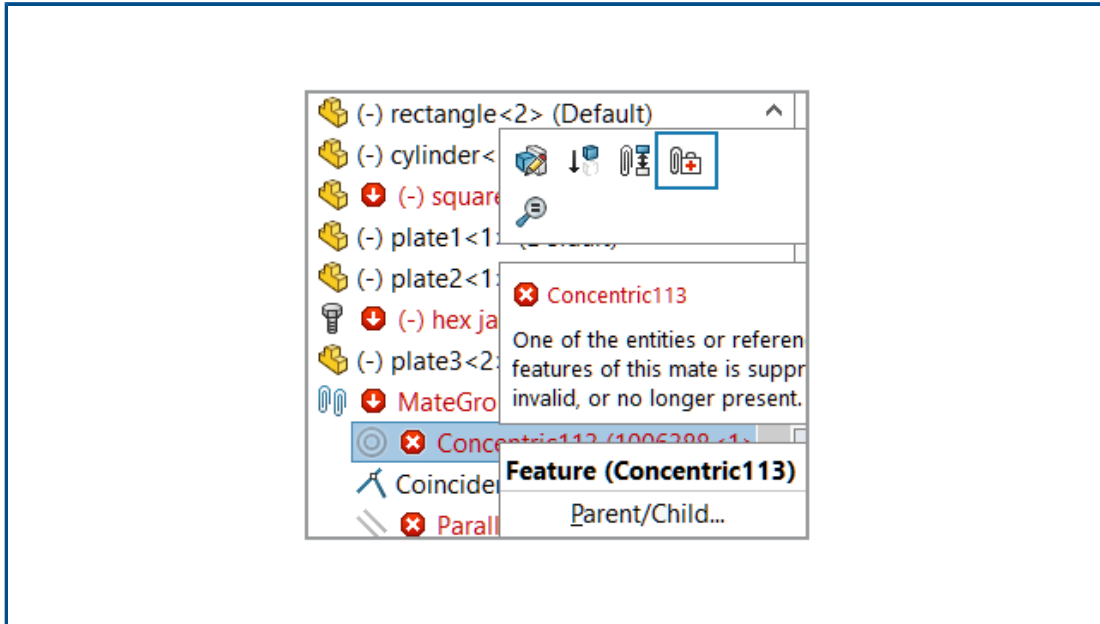
Select Mate Reference  appears when the knob is over the plate.



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



Auto-Repair for Missing Mate References




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

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

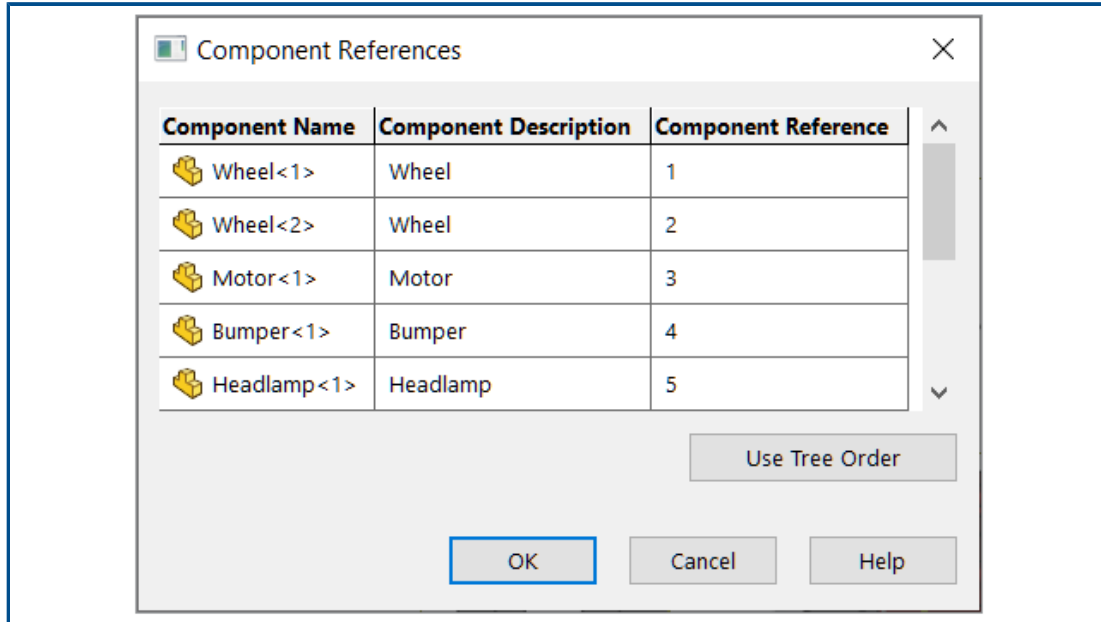
For parallel mates, SOLIDWORKS repairs the missing reference by selecting a reference on the same component that has a different position. For planar faces, the missing reference is repaired with a different planar face that has the same orientation. For plane references, the missing reference is repaired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

To auto-repair missing mate references:

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

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

Assigning Component References to Top-Level Components



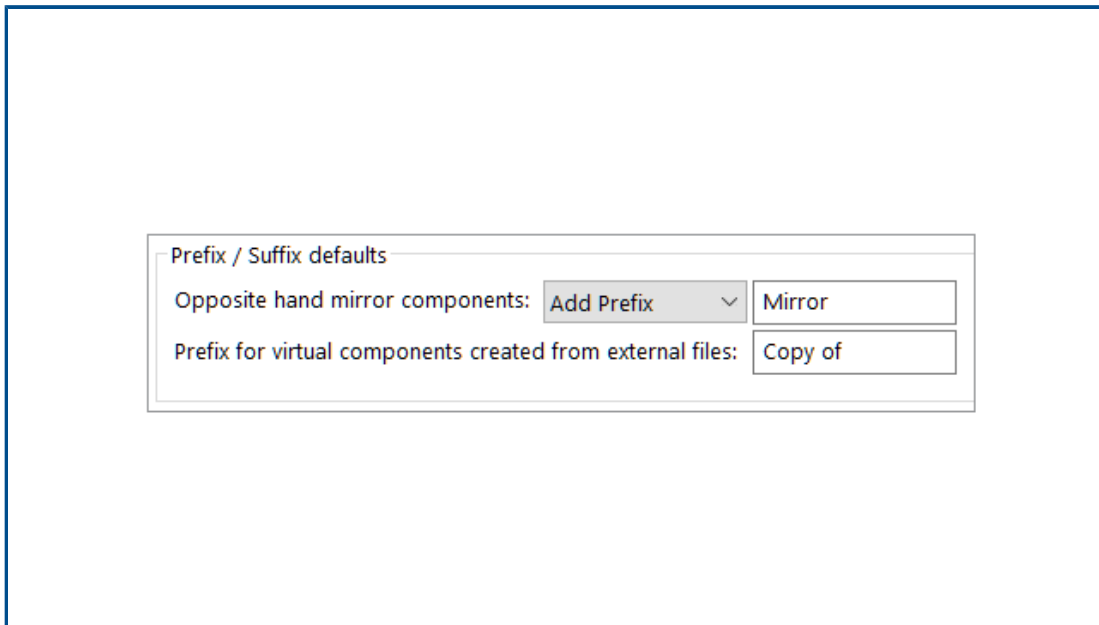
In the Component References dialog box, you can enter component references for all top-level components. You can use the tree order from the FeatureManager design tree as the component reference.

To assign component references to top-level components:

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

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

Specifying a Prefix and Suffix for Components



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

To specify a prefix and suffix for components:

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

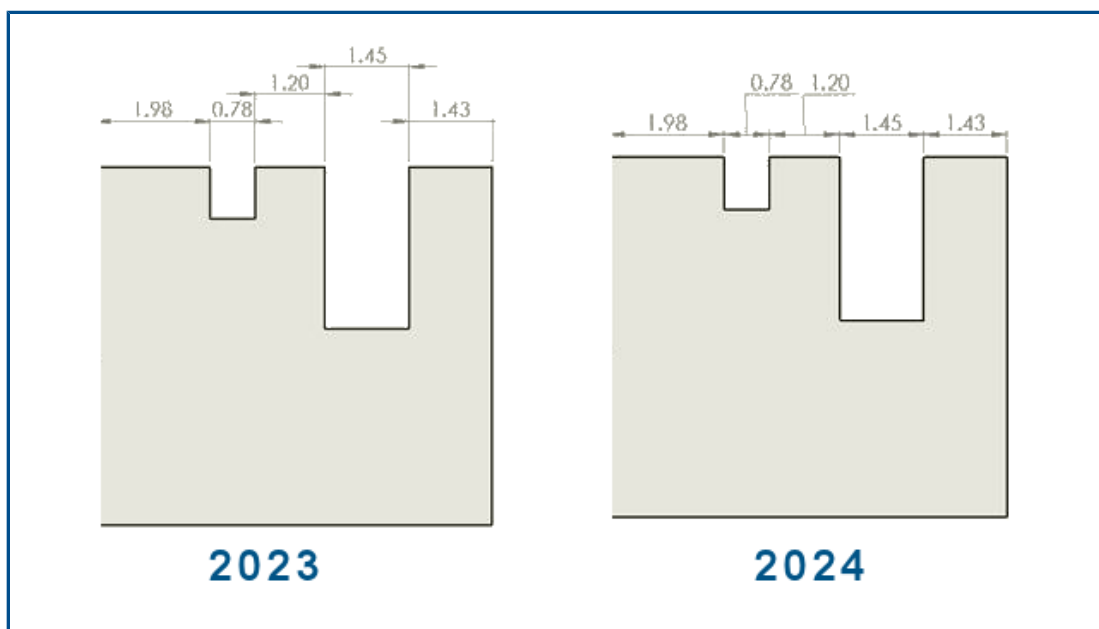
10

Detailing and Drawings

This chapter includes the following topics:

- **Keeping Chain Dimensions Collinear**
- **Overridden Dimensions**
- **Reattaching Dangling Dimensions**
- **Excluding Hidden Sketches from Flat Pattern DXF Files**
- **Highlighting Referenced Elements**
- **Keep Link to Property Dialog Box Open**
- **Opening a Drawing in Detailing Mode by Default**
- **Select Multiple Layers**

Keeping Chain Dimensions Collinear



You can ensure that chain dimensions remain collinear even with limited space. When dimension text and arrowheads overlap, you can select options for the best fit.

To keep chain dimensions collinear when dimension text overlaps:

1. Click **Tools > Options > Document Properties > Dimensions > Linear > Chain Dimension**.

2. Under **Collinearity Options**, select **Offset text automatically when space is limited**.

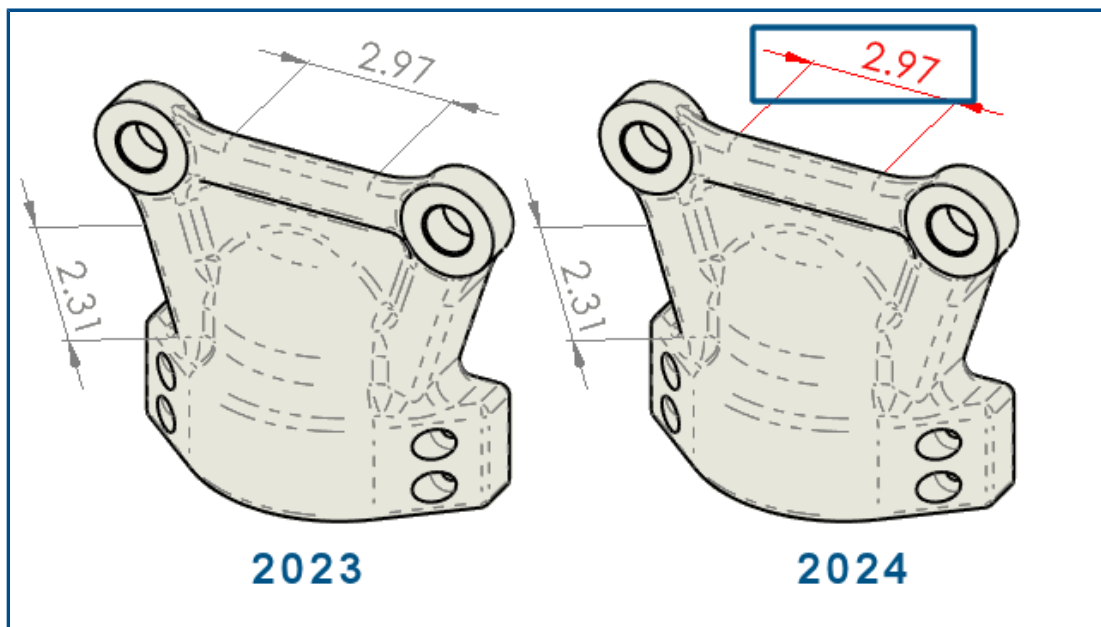
For ISO and ANSI, this option is selected by default.

To keep chain dimensions collinear when arrowheads overlap:

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

For ISO, this option is selected by default.

Overridden Dimensions



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

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

You can:

- Change the color of overridden dimensions automatically.

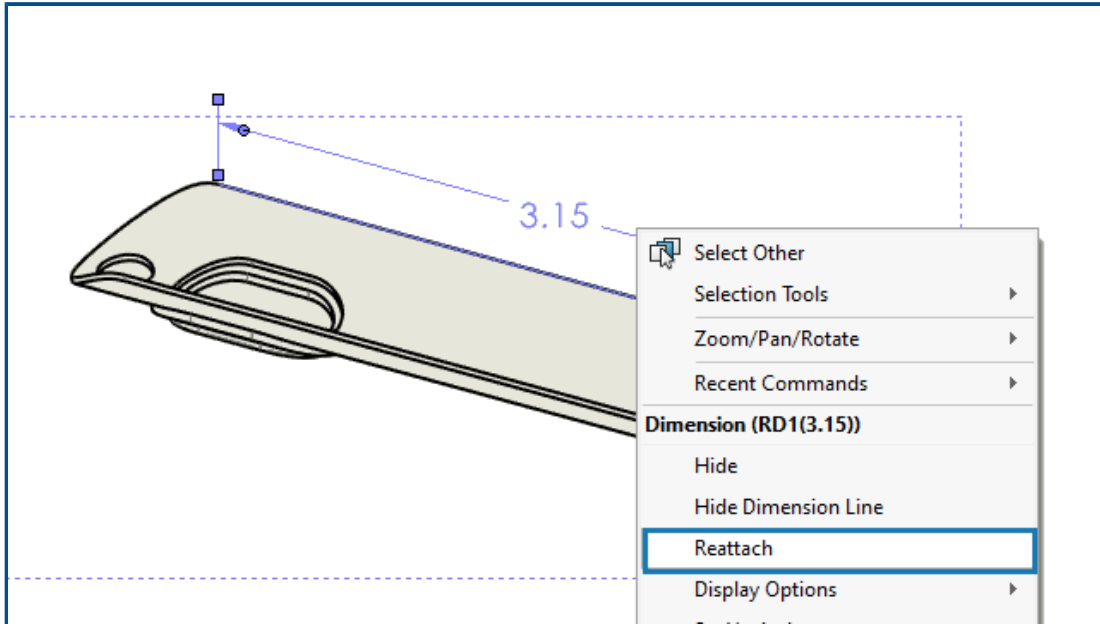
To specify the color, click **Tools > Options > System Options > Colors**. Under **Color scheme settings**, edit the color for **Drawings, Overridden dimensions**.

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

- Restore the overridden dimension values to their original values.

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

Reattaching Dangling Dimensions



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

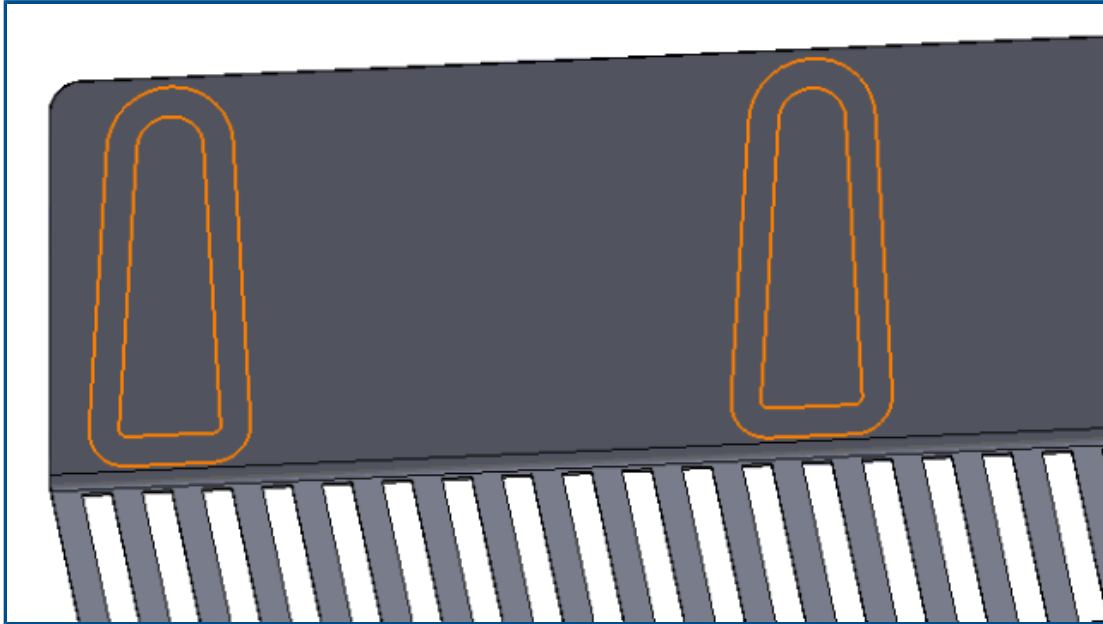
The feature does not support:

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

To reattach dangling dimensions:

1. Right-click the dangling dimension and click **Reattach**.
SOLIDWORKS® highlights the dangling point with an X on the first extension line.
2. Select a point on the model to reattach the dangling point to the model.
The dangling point reattaches to the new selection.
SOLIDWORKS highlights the dangling point with an X on the next extension line.
3. Select a point on the model to reattach the dangling point to the model.
The dangling point reattaches to the new selection.

Excluding Hidden Sketches from Flat Pattern DXF Files

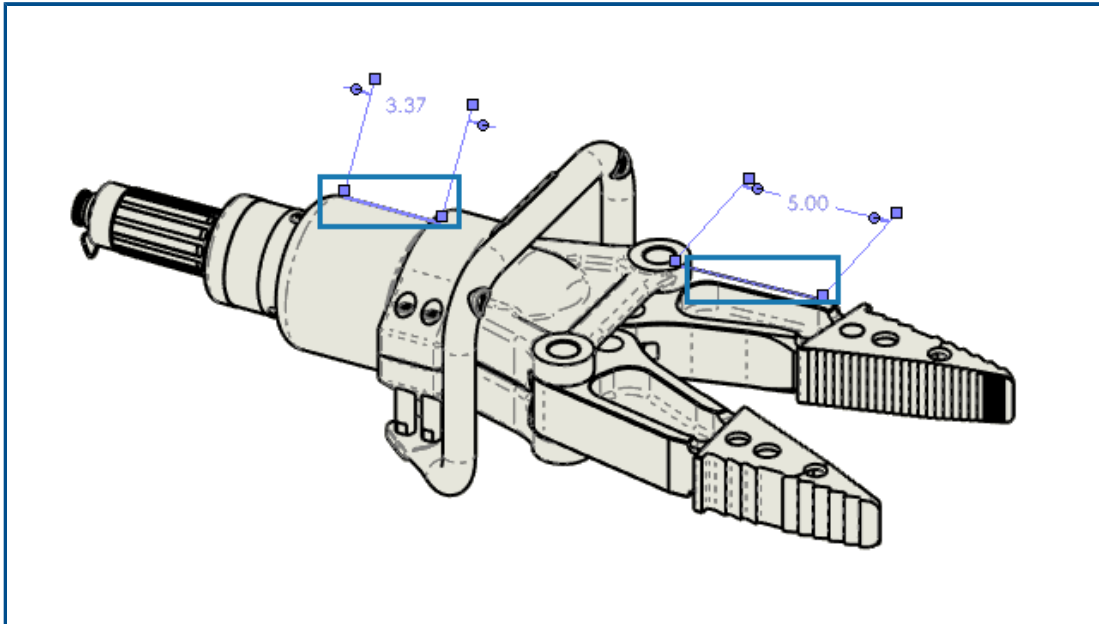


In the DXF / DWG Output PropertyManager, when you export a sheet metal flat pattern as a `.dxf` file, you can exclude hidden sketches.

To exclude hidden sketches from flat pattern DXF files:

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

Highlighting Referenced Elements



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

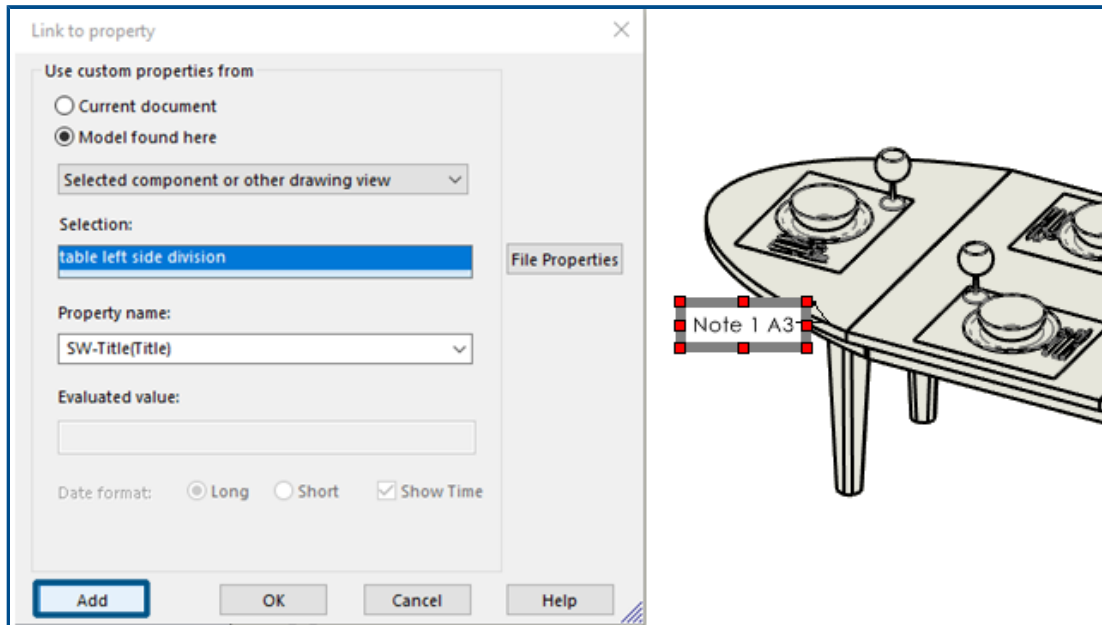
The feature does not support the following dimensions:

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

To highlight referenced elements:

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

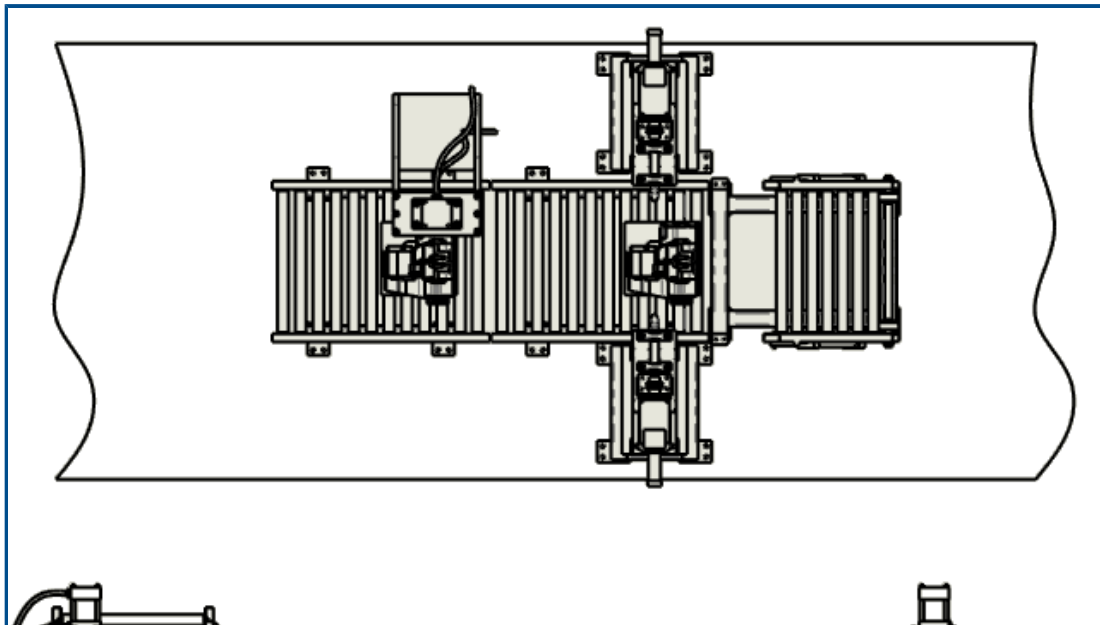
Keep Link to Property Dialog Box Open



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

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

Opening a Drawing in Detailing Mode by Default



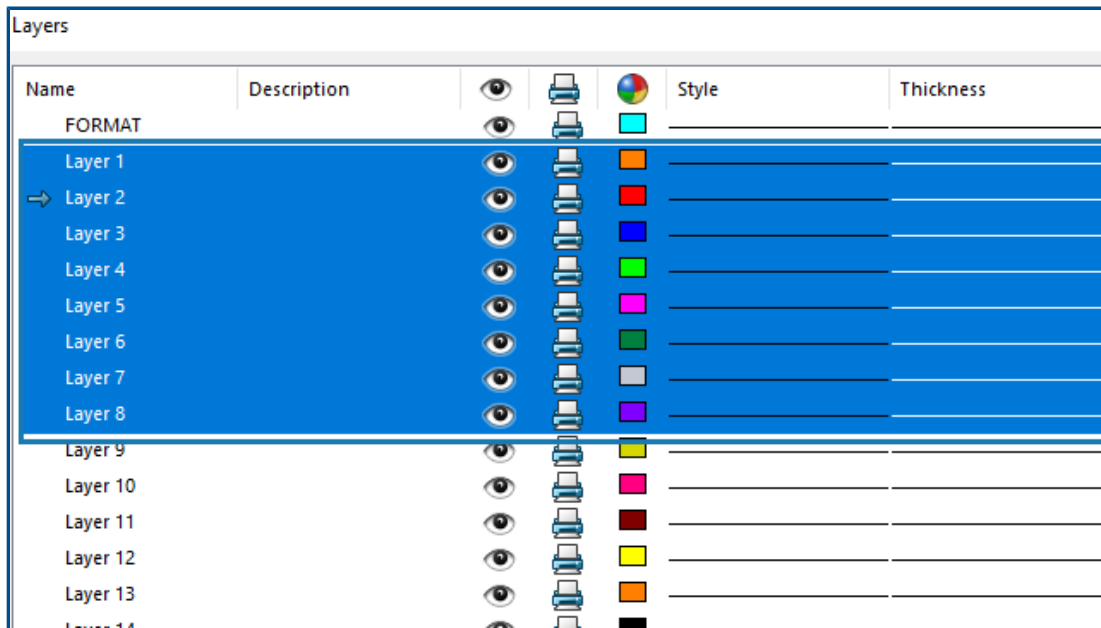
You can open a drawing in Detailing mode by default.

You can use this to automatically open large drawings quickly.

To open a drawing in Detailing mode by default:

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

Select Multiple Layers



You can select multiple layers at once to modify.

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

You can:

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

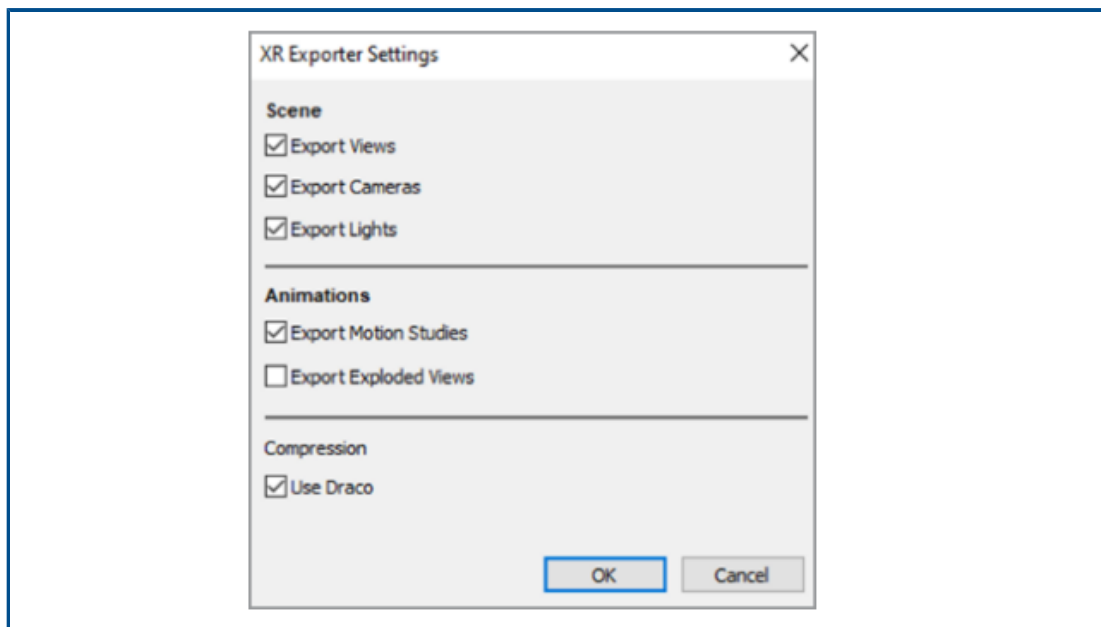
11

Import/Export

This chapter includes the following topics:

- **Exporting to Extended Reality**
- **Canceling the Import of Third-Party CAD Files**
- **Importing STEP Assemblies as Multibody Parts**

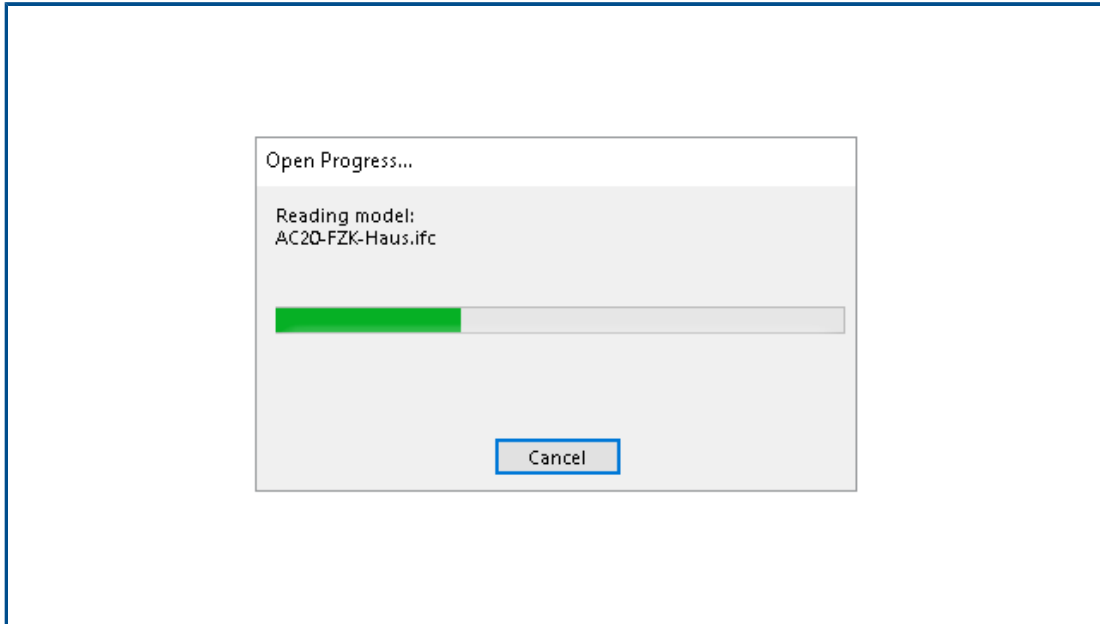
Exporting to Extended Reality



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

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

Canceling the Import of Third-Party CAD Files



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

To cancel the import of third-party CAD files:

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

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

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

Importing STEP Assemblies as Multibody Parts

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

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

12

SOLIDWORKS PDM

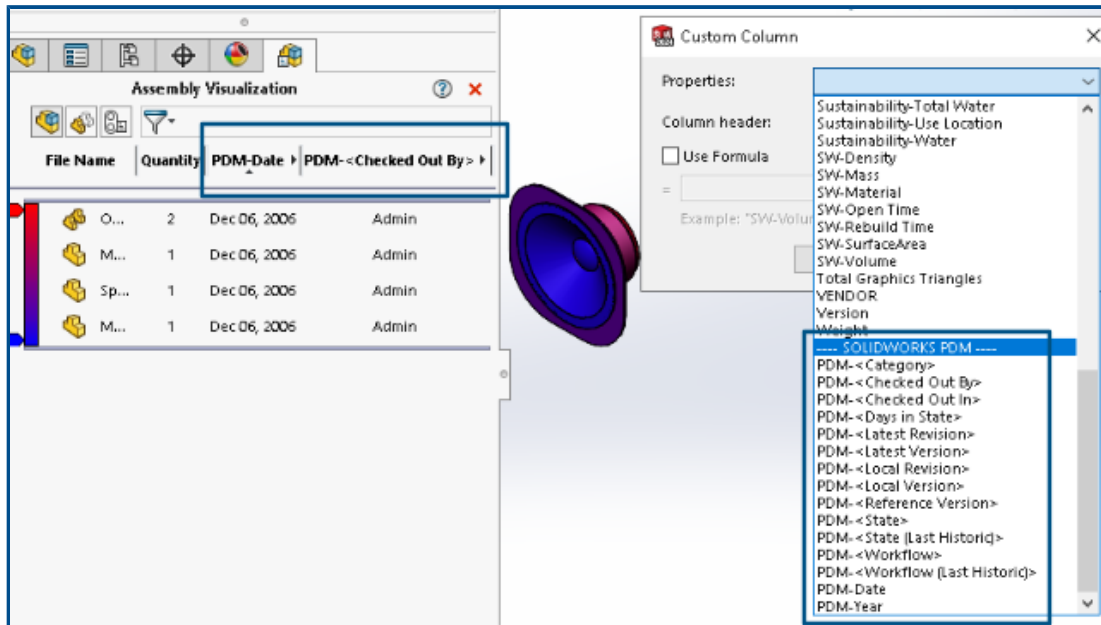
This chapter includes the following topics:

- **Assembly Visualization**
- **Downloading Specific Versions of a File in Web2**
- **File Type Icons**
- **Check Out Option in Change State Command**
- **Copying Tree Dialog Box**
- **Viewing Check-Out Event Details**
- **System Variables**
- **Viewing License Usage**
- **Data Security Enhancements**
- **SOLIDWORKS PDM Performance Improvements**

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.

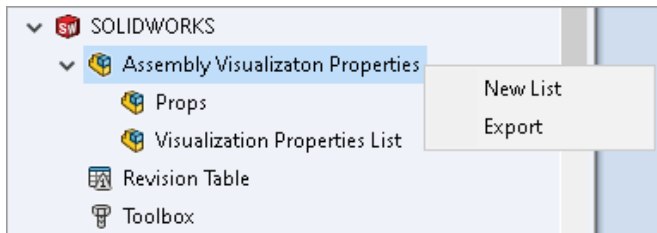
SOLIDWORKS PDM Professional is a full-featured data management solution for a small and large number of users, and is available as a separately purchased license.

Assembly Visualization



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

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



To view SOLIDWORKS PDM custom variables in Assembly Visualization:

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

Customize Assembly Visualization Properties Dialog Box

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

To open this dialog box:

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

Name

Specifies the name of the new properties list.

Variables

Variable	Displays the selected variable.
Name	Displays the name of the selected variable.
Add	Adds the selected variable.
Delete	Deletes the selected variable.
Up and down arrows	Moves the selected variables up or down.

Selected Variable

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

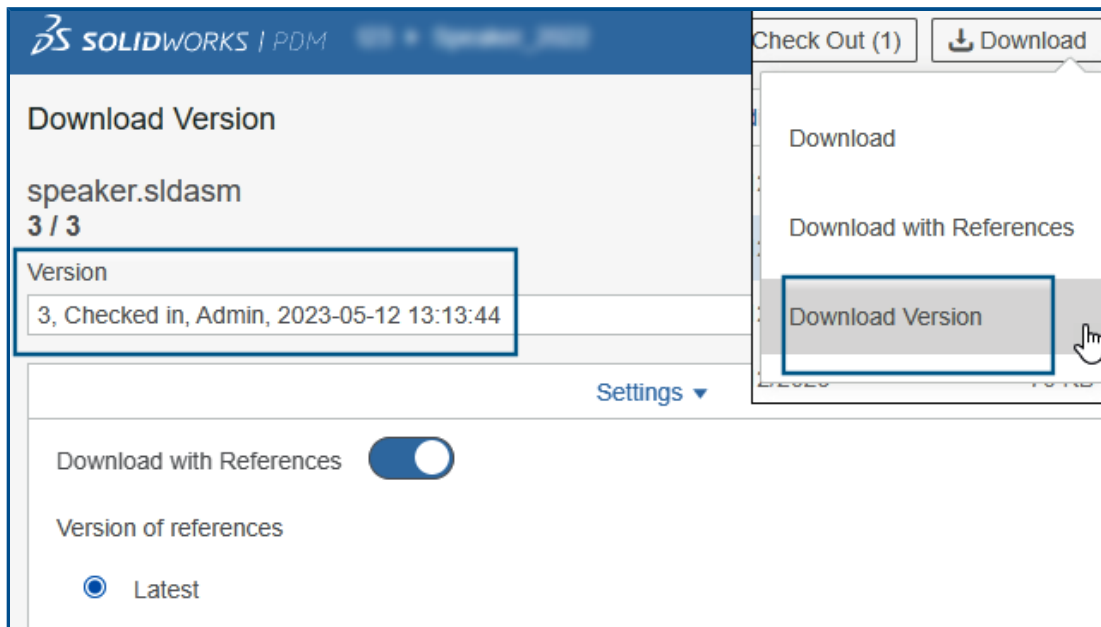
Users

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

Groups

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

Downloading Specific Versions of a File in Web2



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

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

The Download Version dialog box lets you select the version and settings for download.

To access this dialog box:

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

Download Version Dialog Box

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

To open this dialog box:

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

Version


Select the version of the file to download.

Settings

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

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

Files

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

Total Files to Download

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

Download

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

Download Version Dialog Box - Small Screen Layout

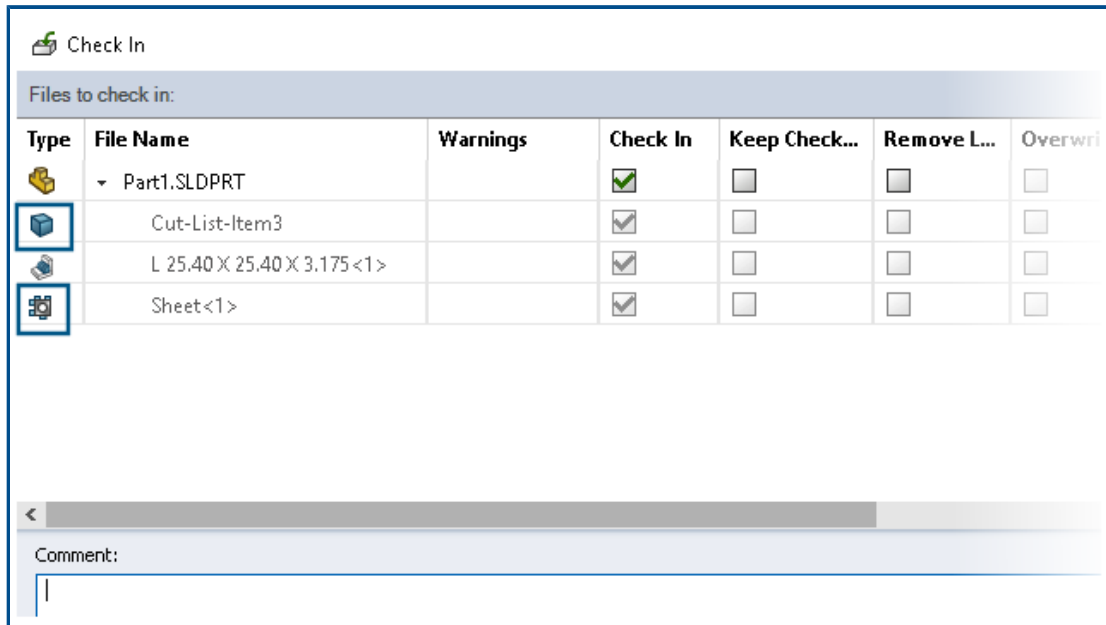
You can use the Download Version dialog box to download a specific version of a file and its references.

To open this dialog box:

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

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

File Type Icons



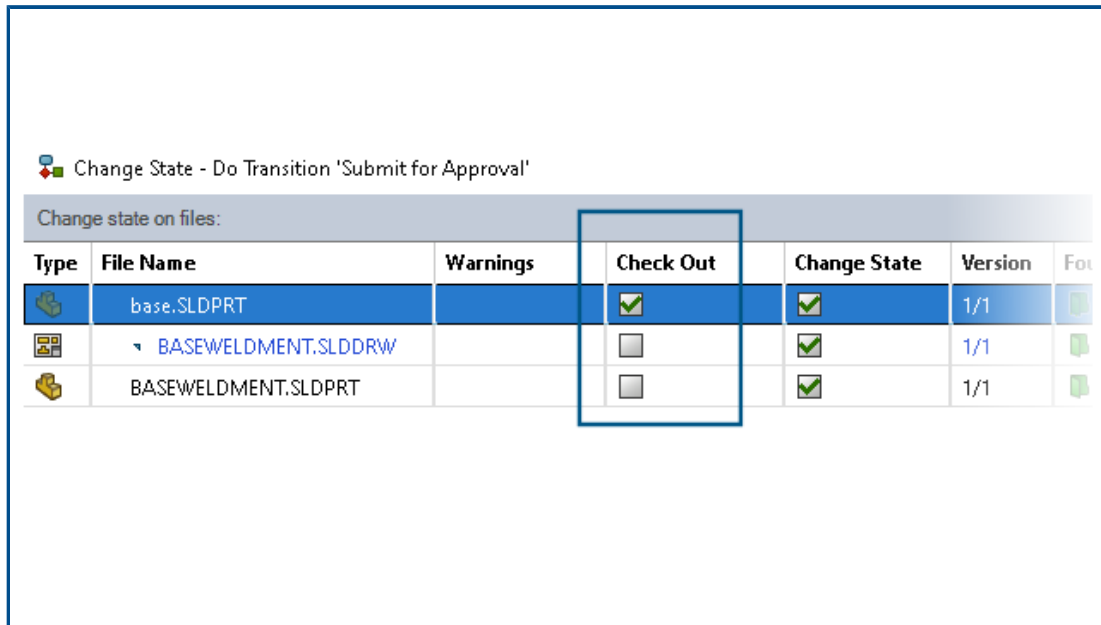
You can view the file type icons for weldment cut list items and the files that were shared using pasted shared overlays.

These icons are available in the dialog boxes for:

- File Details
- File Operations
- Web2

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

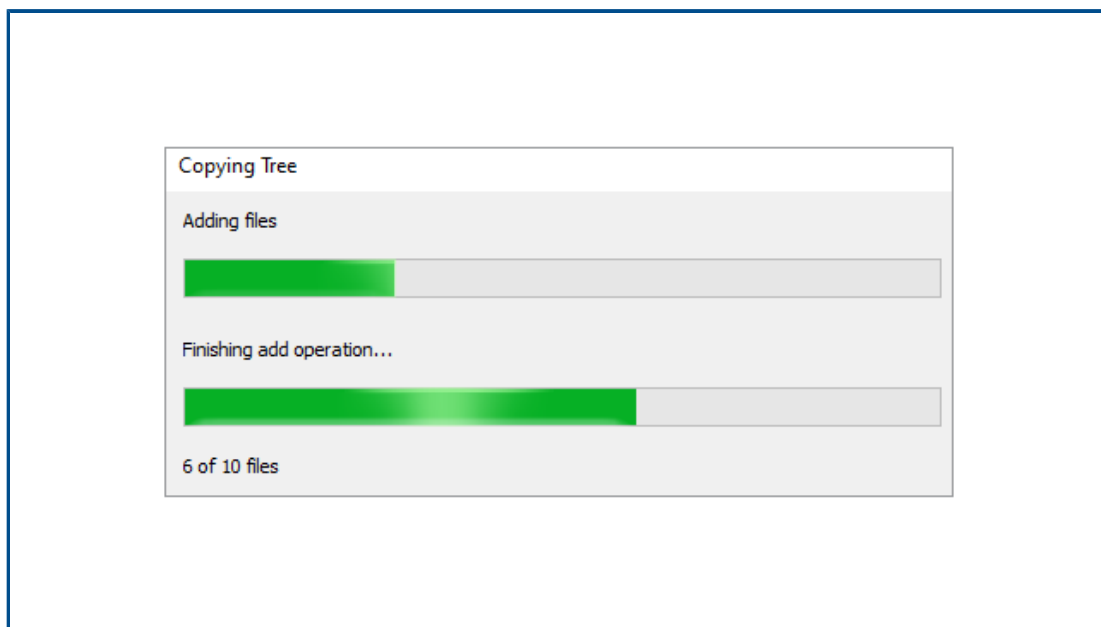
Check Out Option in Change State Command



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

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

Copying Tree Dialog Box

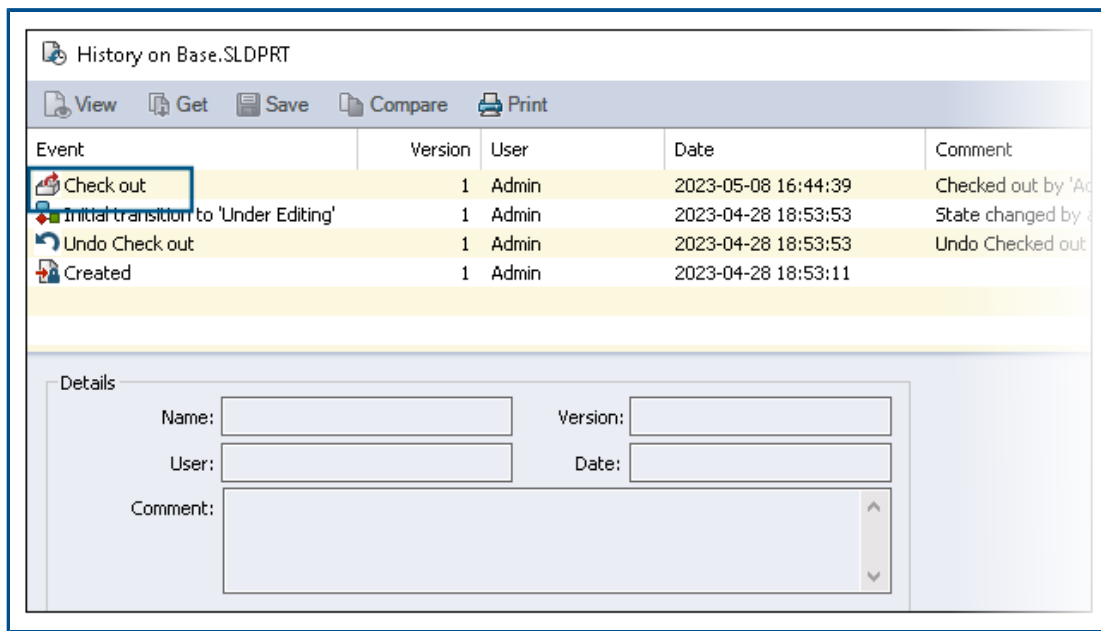


In the SOLIDWORKS PDM File Explorer, the Copying Tree progress dialog box that appears while copying a file tree (from **Tools > Copy Tree**) includes more detailed information.

The dialog box includes two progress bars:

- The first progress bar has the primary steps or actions of the overall copy tree operation, such as **Copying Files** and **Copying Variables**.
- The second progress bar has:
 - Secondary steps such as **Preparing Files** and **Finishing add operation**.
 - File names with extensions.
 - The number of files copied and the total number of files.

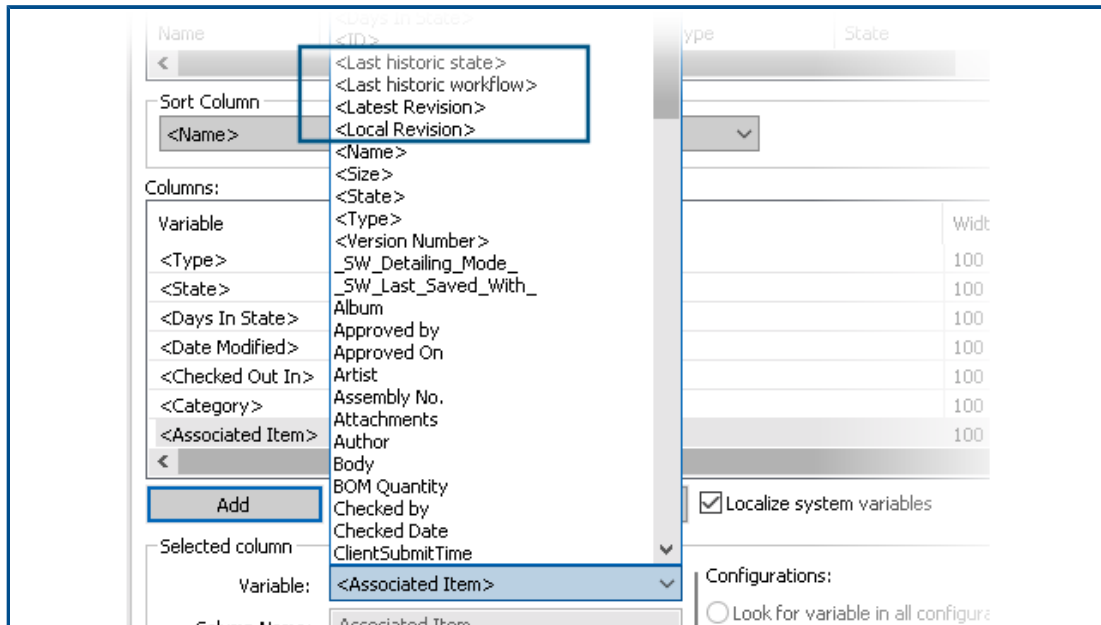
Viewing Check-Out Event Details



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

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

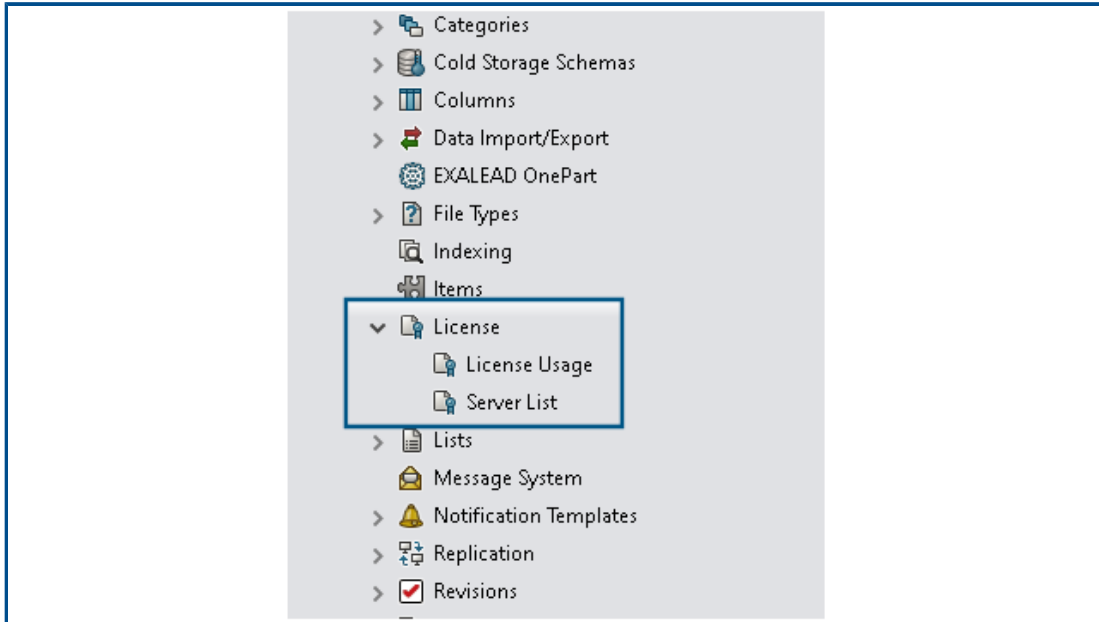
System Variables



System variables are more available and easier to access.

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

Viewing License Usage



You can view license details without any special administrative permissions.

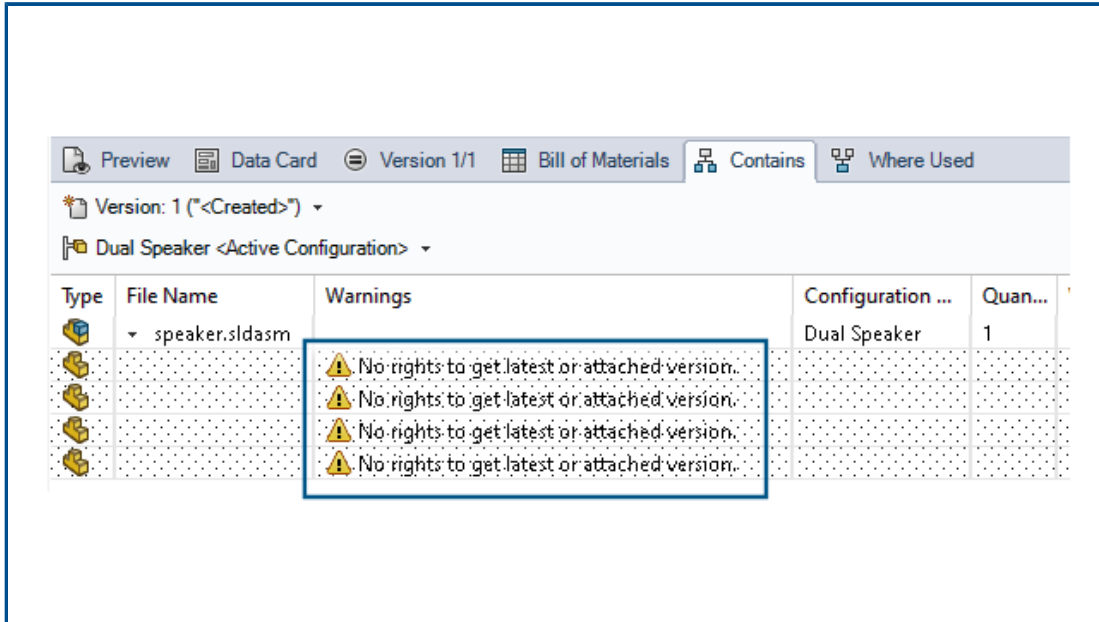
In the Administration tool, the **License** node has the following subnodes:

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

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

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

Data Security Enhancements



In SOLIDWORKS PDM File Explorer and Web2, unauthorized users cannot view file information in the Contains and Where Used tabs.

The warning message **No rights to get latest or attached version** displays.

SOLIDWORKS PDM Performance Improvements

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

The following operations are approximately two times faster:

- Add files
- Change state
- Copy tree

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

13

SOLIDWORKS Manage

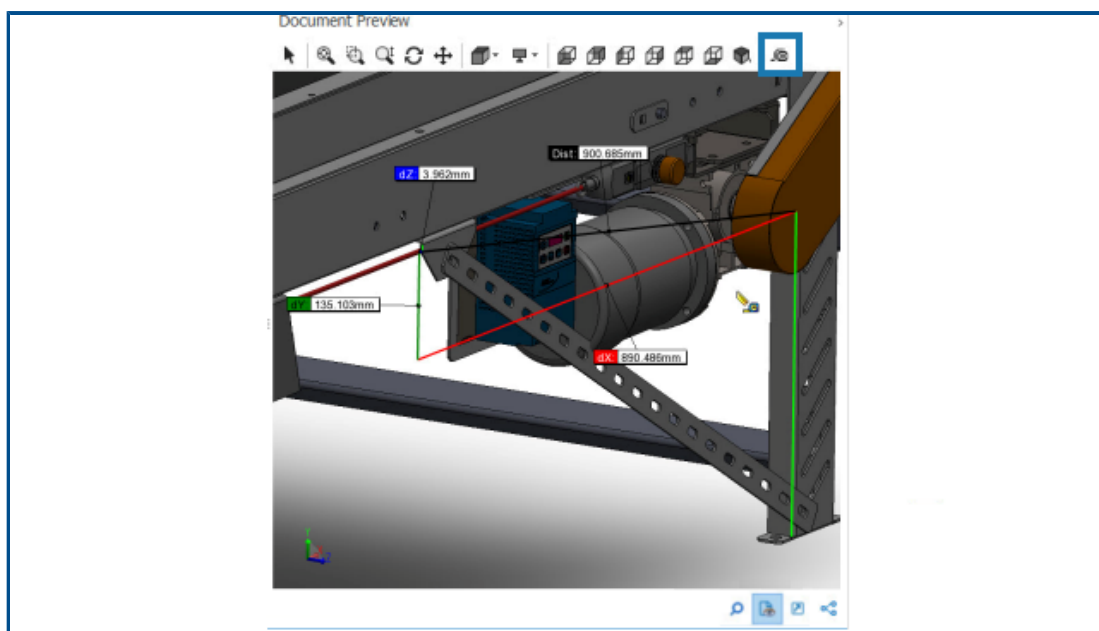
This chapter includes the following topics:

- **Measuring in a Document Preview**
- **Plenary Web Client CAD File Preview**
- **Field Conditions for Affected Items**
- **Task Automation**
- **Task Burn Down Chart**
- **Timesheet Working Hours**
- **Bill of Materials Quantity**
- **Process Output for Replacing BOM Items**
- **Adding Child Conditions to BOMs**

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

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


Measuring in a Document Preview




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

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

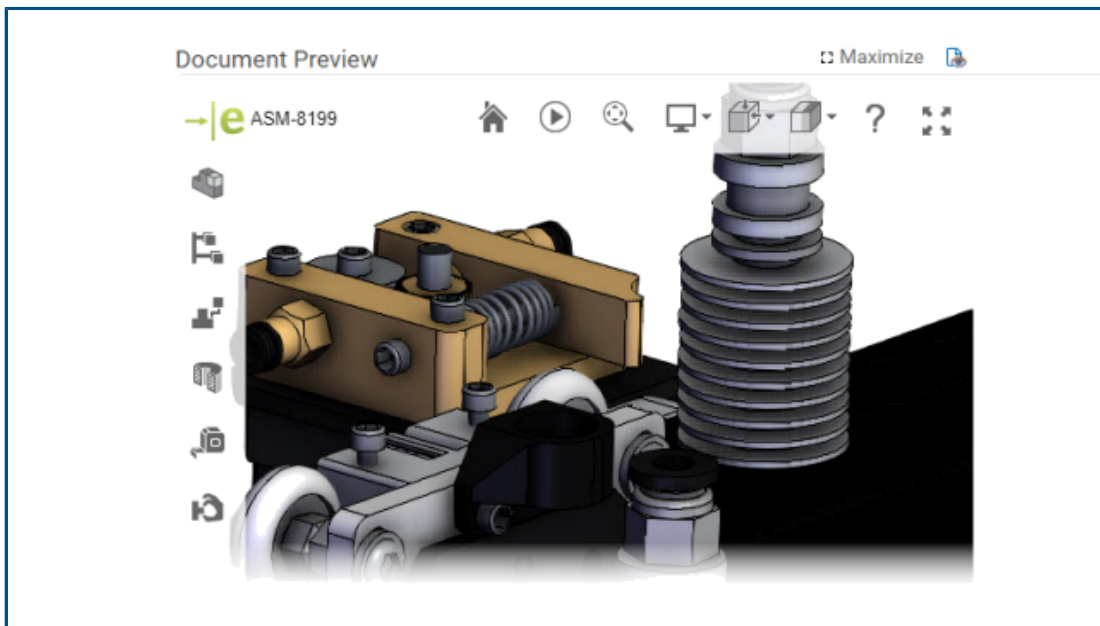
To measure in a Document Preview:

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

The eDrawings® preview displays the selected SOLIDWORKS record.

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

Plenary Web Client CAD File Preview

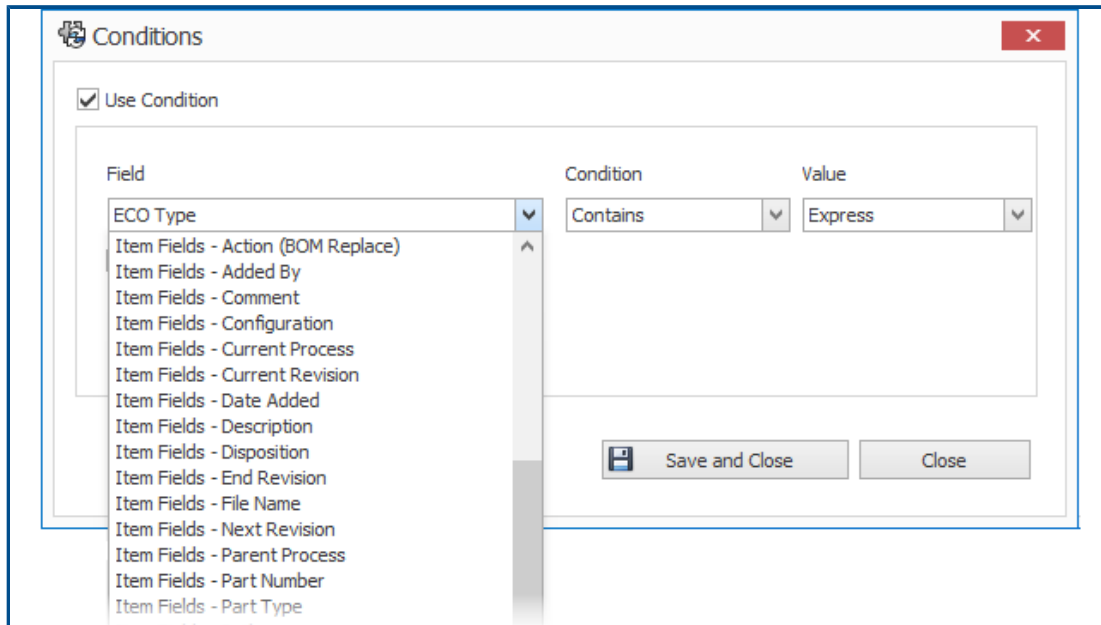


You can dynamically preview CAD files in the Plenary web client windows.

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

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

Field Conditions for Affected Items



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

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

Adding Required Fields to an Affected Item Field

To add required fields to an affected item field:

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

You cannot define mapped fields as required fields.

3. Open the Workflow Properties wizard and select a stage in the workflow diagram.
4. Click **Item Fields**.
5. Select **Required**.

To add a condition, click ellipses in the first **Condition** column to open the Conditions dialog box.

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

6. Click **Save**.

Adding Default Values to an Affected Item Field

To add default values to an affected item field:

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

You cannot define mapped fields as required fields.

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

Mapped fields cannot have a default value.

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

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

Task Automation

Add Edit Delete Refresh

All tasks must be completed before this stage is completed.
 Create these tasks every time this stage is activated

Complete	Subject	Allocated Time	Priority	Created By	Stage
<input checked="" type="checkbox"/>	Feasibility Study	0	Medium	System Administrator	Request Under Review
<input type="checkbox"/>	Cost Benefit Analysis	0	Medium	System Administrator	Request Under Review

Enable conditions for selected Task

Save Conditions

Field: Condition: Value:

Two Conditions

Task automation streamlines the preconfiguration process of handling tasks.

You can add conditions to control the creation of individual tasks. This helps to create tasks that are based on process field values. For example, if multiple departments can

participate in a process, each with their own task, you can add conditions to create the tasks for the required departments.

Adding Task Conditions

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

To add task conditions:

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

Defining Task Completion Requirements

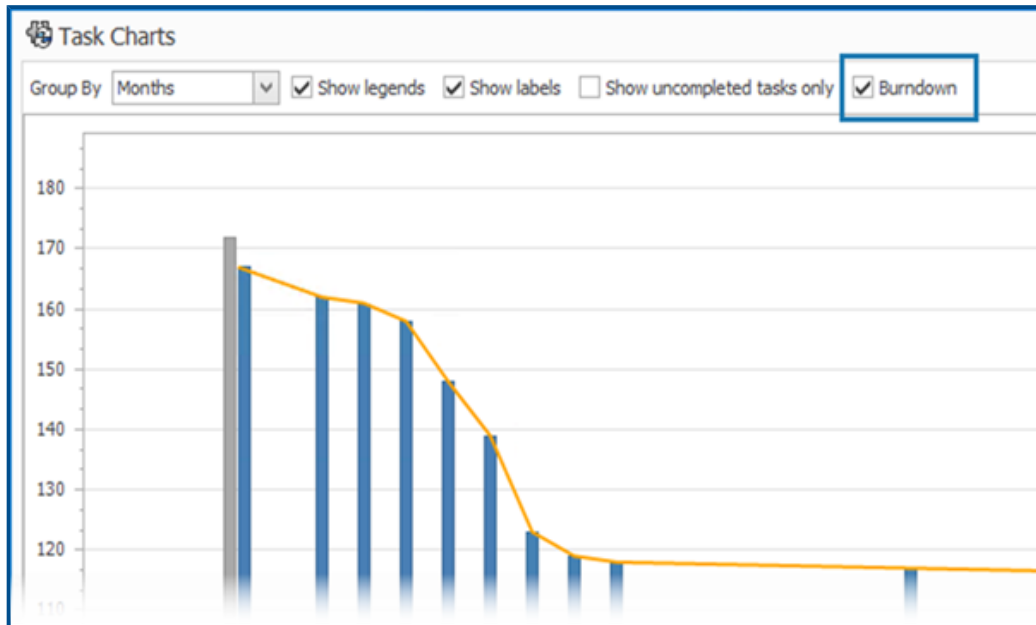
You can define individual tasks to complete before processes can move forward.

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

To define task completion requirements:

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

Task Burn Down Chart



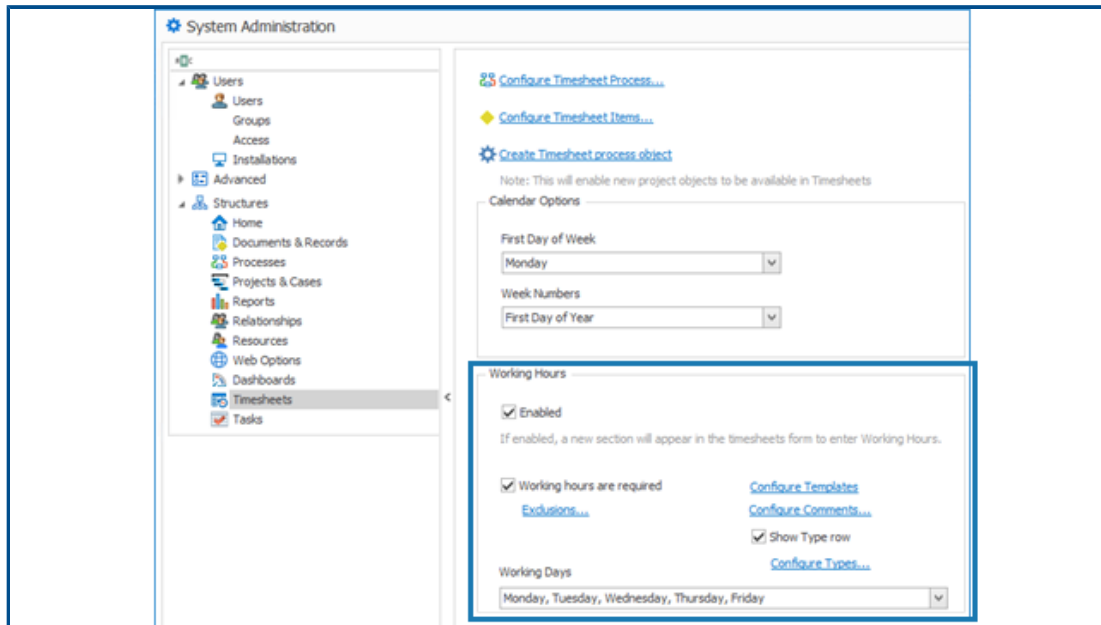
The task burn down chart shows the progression of all project tasks.

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

The burn down chart does not show canceled tasks.

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

Timesheet Working Hours



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

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

Configuring Timesheet Working Hours

To configure timesheet working hours:

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

Working Hours appears in all new and existing timesheets.

3. Specify **Working Hours** options:

Option	Description
Enabled	Lets you specify working hours options.
Working hours are required	Allows total hours for a day other than zero. <div style="border: 1px solid gray; padding: 5px; margin-top: 10px;"> If you select Show Type row and if the value for Exclusions matches the type you enter, you can enter total hours as 0. </div>
Exclusions	Lets you enter values corresponding to the Type .
Configure Templates	Creates workweek templates to reduce the number of entries in a template.
Configure Comments	Lets you add comments for each day and time slot.
Show Type row	Displays a Type row for you to select a type from the list.
Configure Types	Specifies the required Type options.
Working Days	Specifies the days in the workweek.

Configuring Templates

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

To configure templates:

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

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

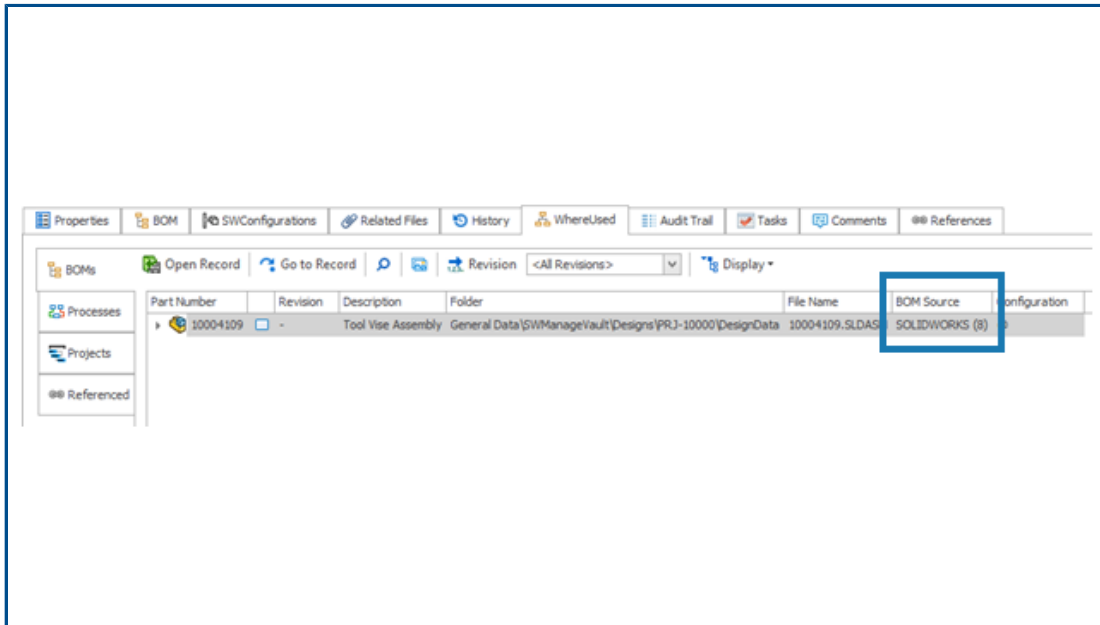
Option	Value	Format
Start	Work start time for a day	24-hour
Pause duration	Break time during the day	hh:mm
End	Work end time for a day	24-hour
Total Time	Calculated based on the other values you specify	

Configuring Comments

You can add comments for each day and time slot.

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

Bill of Materials Quantity




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

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

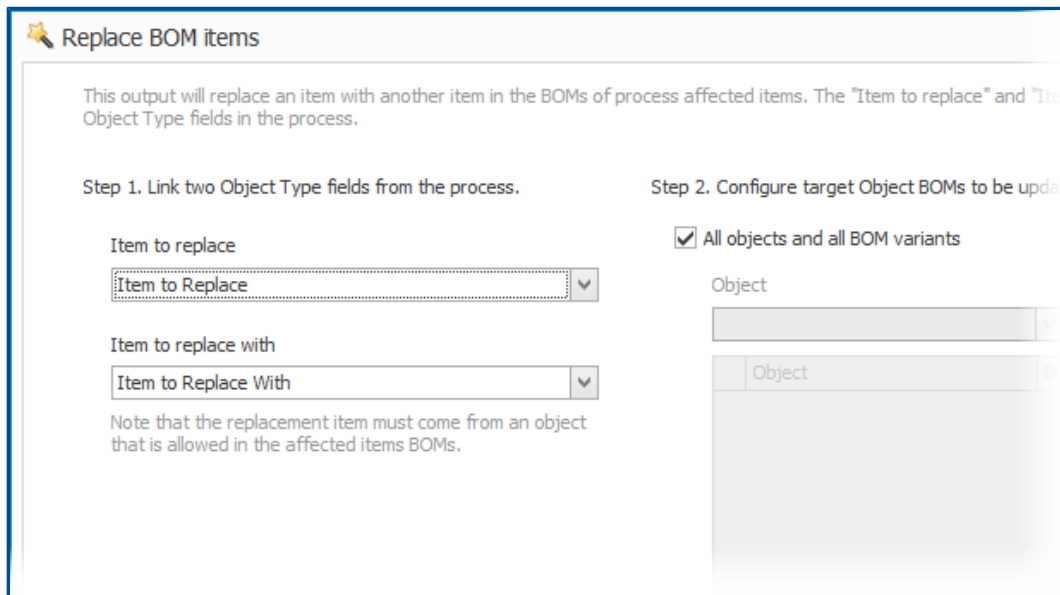
Adding Custom Columns to the Where Used Tab

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

To add custom columns to the Where Used tab:

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

Process Output for Replacing BOM Items



Replace BOM items

This output will replace an item with another item in the BOMs of process affected items. The "Item to replace" and "Item to replace with" fields in the process.

Step 1. Link two Object Type fields from the process.

Item to replace

Item to replace with

Note that the replacement item must come from an object that is allowed in the affected items BOMs.

Step 2. Configure target Object BOMs to be updated.

All objects and all BOM variants

Object	Object Type
Object	Object


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

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


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

Enabling Mass Replace in a Process

To enable mass replace in a process:

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

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


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

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

21. Click **Save and Close**.

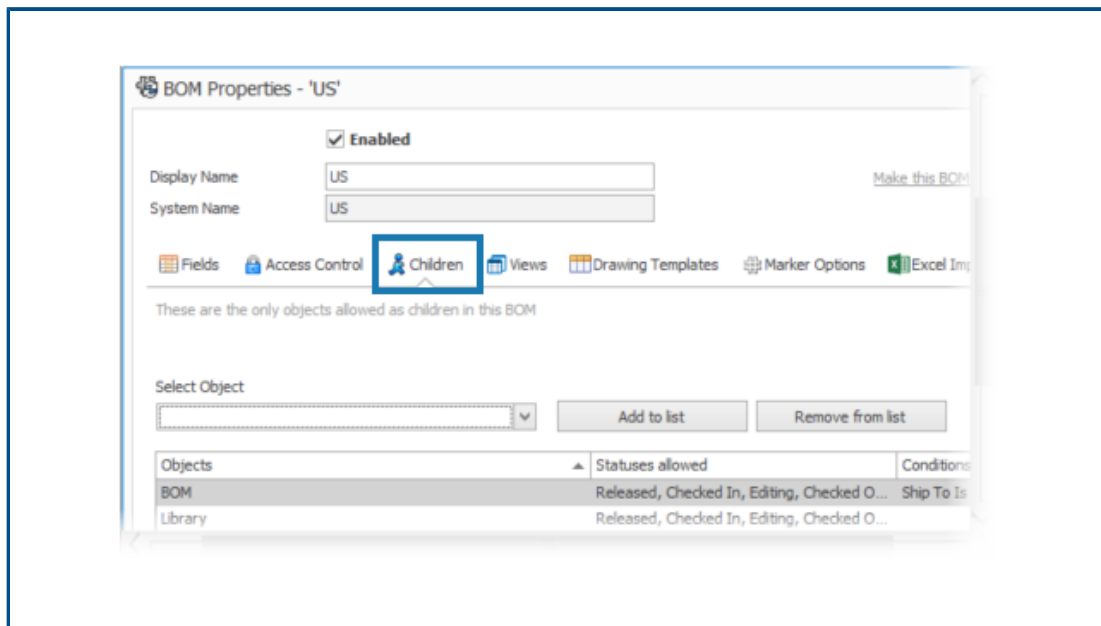
Replacing BOM Items

To replace BOM items:

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



6. Click **Add to list** to close the dialog box and add the selected records to the affected item list.
7. Move the process through its workflow past the stage where you added the **Replace BOM items** output.
To see the updated BOMs, open the record for an affected item.

Adding Child Conditions to BOMs



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

To add child conditions to BOMs:

1. In the System Administration tool, under **Structures**, select an object and click **Edit** .
2. Open the Bill of Materials wizard.
If you edit a record or document object other than a SOLIDWORKS PDM object, click the BOM tab.
3. Select the **Bill of Material** object in the list and click **Edit** .
4. In the BOM Properties dialog box, click the Children tab.
5. Click the cell under **Statuses allowed** for the BOM variant and select the required status.
6. In the **Conditions** column for a BOM object, click ellipses in the cell to add conditions that restrict items to add to the BOM.
7. In the Do not allow adding items to BOM if these conditions are met dialog box, enter the required conditions and warning message.
8. Click **Save and Close**.

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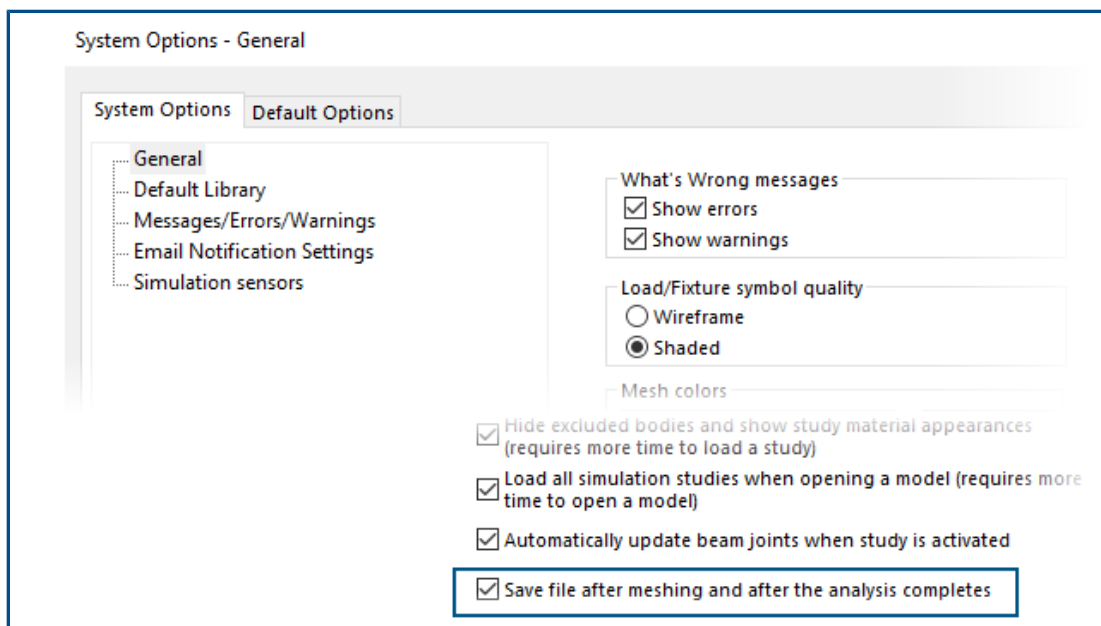
SOLIDWORKS Simulation

This chapter includes the following topics:

- **Automatic Saving of a Model File**
- **Bonding Interactions for Shells**
- **Convergence Check Plot**
- **Decoupling Mixed Free Body Modes**
- **Direct Sparse Solver Retired**
- **Enhanced Bearing Connectors**
- **Excluding Mesh and Results When Copying a Study**
- **Exporting Mode Shape Data**
- **Mesh Performance**
- **Performance Enhancements**
- **Underconstrained Bodies Detection**

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.

Automatic Saving of a Model File



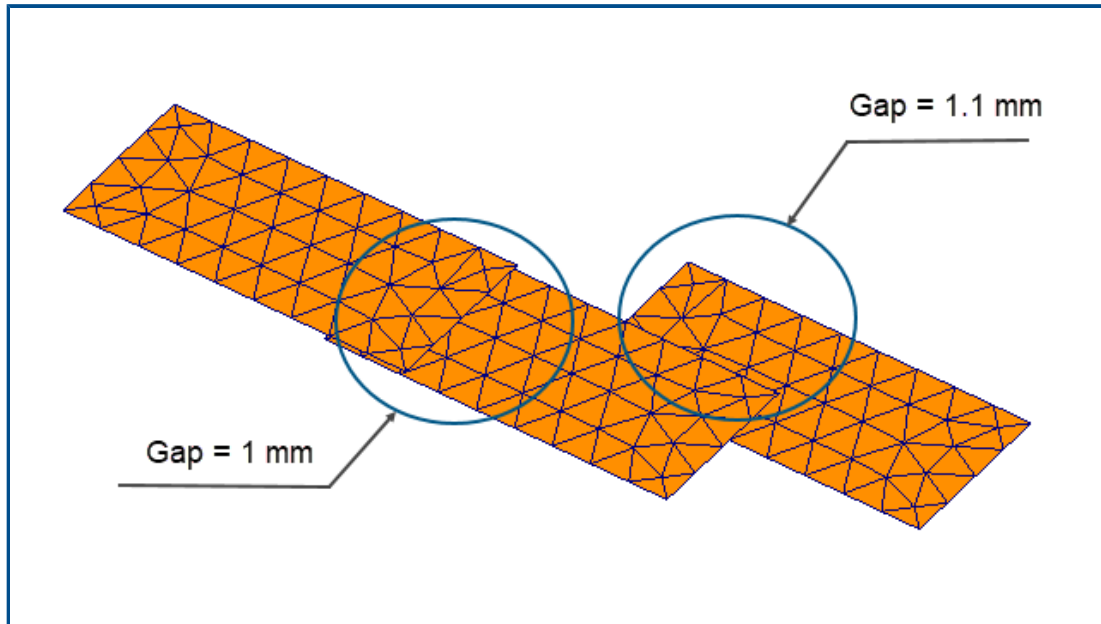
You can save a model file after meshing and after the analysis completes.

To turn on automatic saving of a model file:

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

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

Bonding Interactions for Shells

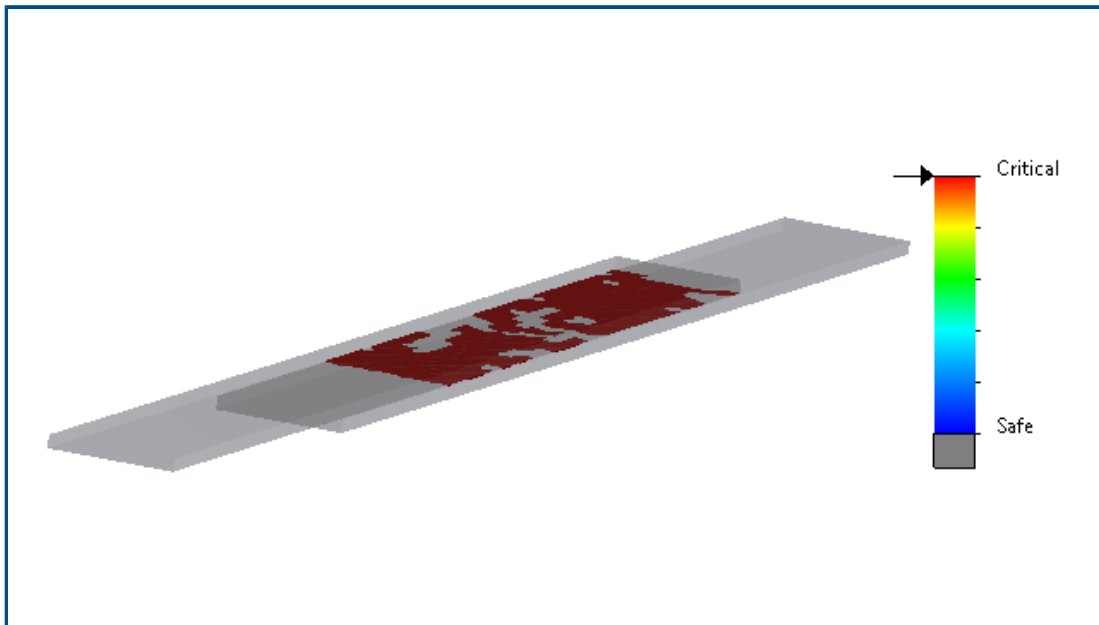


The enforcement of bonding interactions between sets of shell elements that have a physical gap is more robust.

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

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

Convergence Check Plot



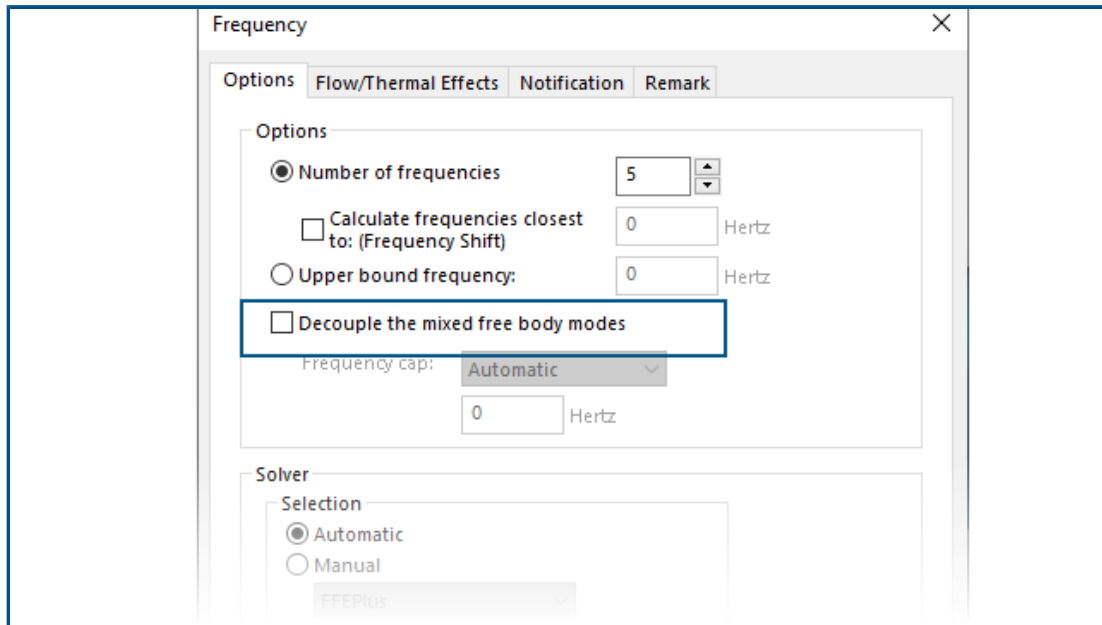
The **Convergence Check Plot** detects regions of the model where the solver has encountered contact convergence issues.

To access the Convergence Check Plot:

Do one of the following:

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

Decoupling Mixed Free Body Modes

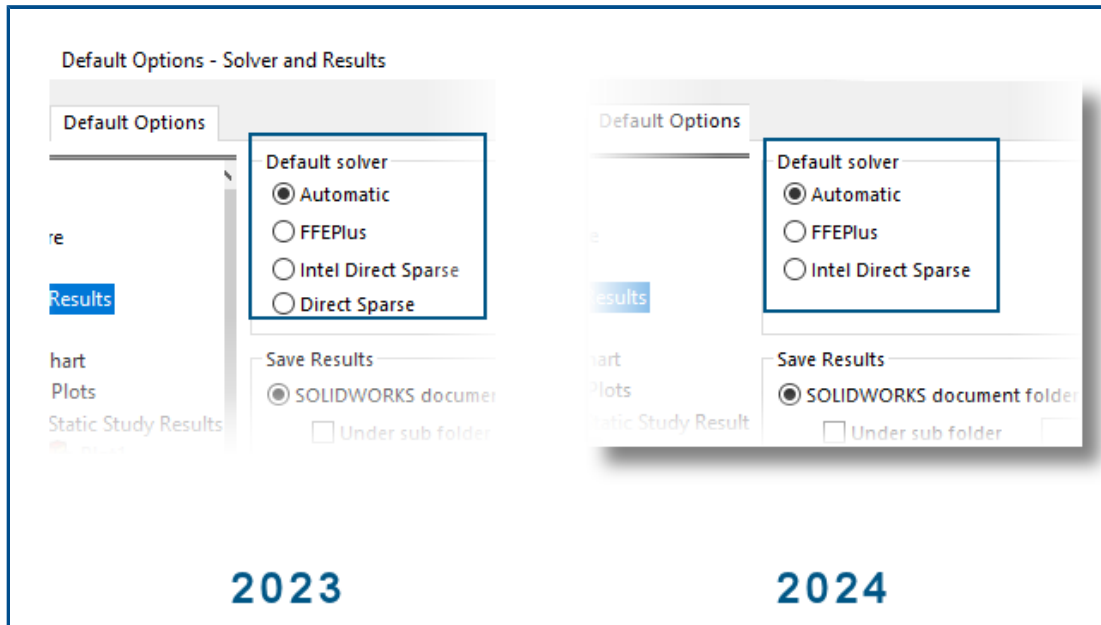


An algorithm can detect and decouple the mixed free body modes while calculating mode shapes.

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

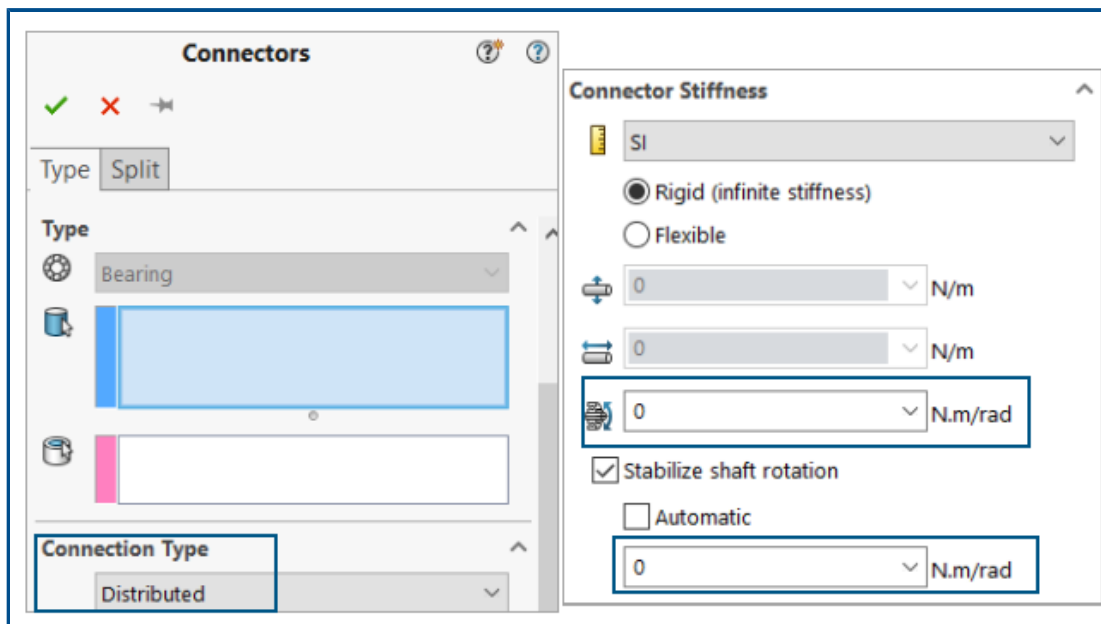
The option to decouple the mixed free body modes is available in Frequency, Linear Dynamic, Harmonic, Random Vibration, and Response Spectrum Analysis studies.

Direct Sparse Solver Retired



The Direct Sparse solver is removed from the list of solvers for simulation studies. For legacy studies that use the Direct Sparse solver, SOLIDWORKS Simulation uses the Intel® Direct Sparse solver.

Enhanced Bearing Connectors



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

The bearing connector is enhanced as follows:

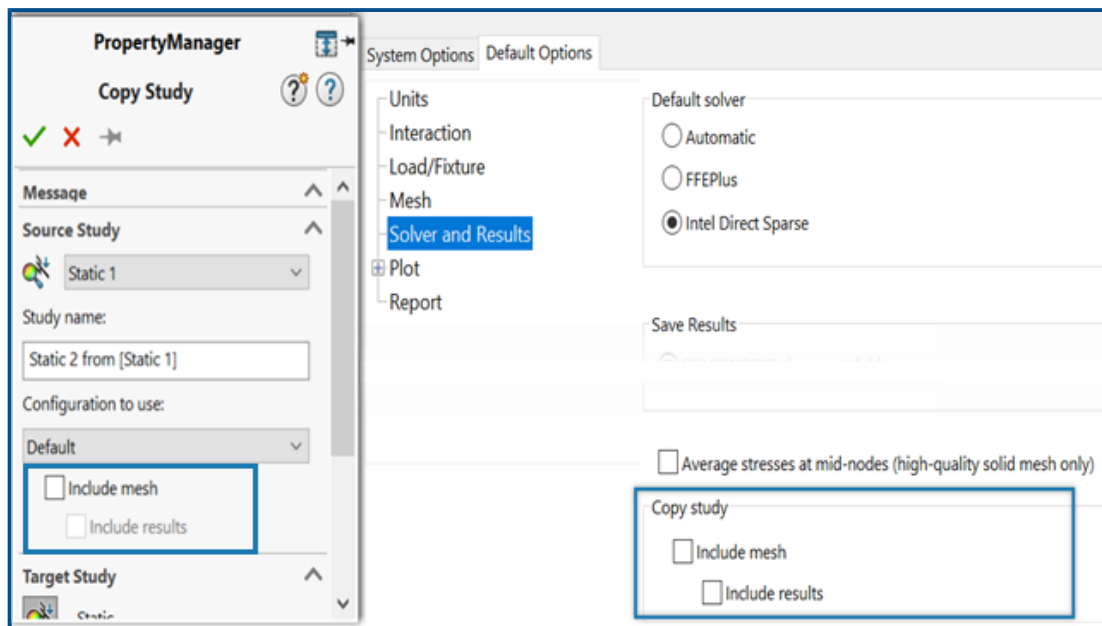
- A **Distributed** type is added to the connector's **Connection Type** options. For a new bearing connector definition, the default **Connection Type** is **Distributed**.
- The addition of **Tilt stiffness** accounts for the bending stiffness of the shaft.

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

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

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

Excluding Mesh and Results When Copying a Study

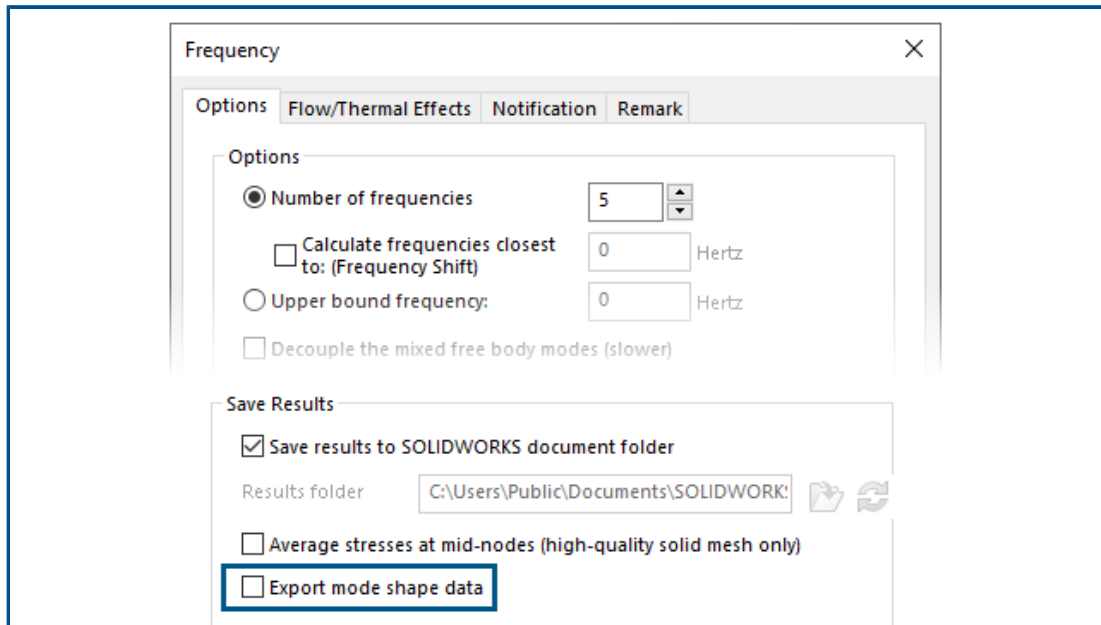


You can save time by excluding mesh and results data when copying a simulation study to a new study.

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

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

Exporting Mode Shape Data

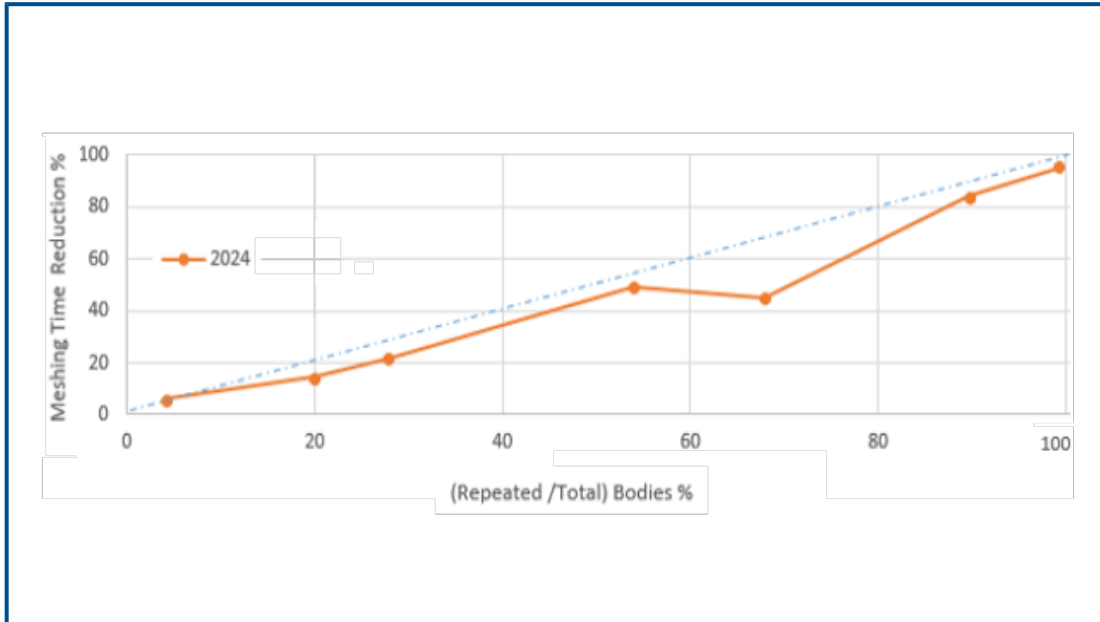


You can export mode shape data to the study's *study_name.out* file.

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

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

Mesh Performance



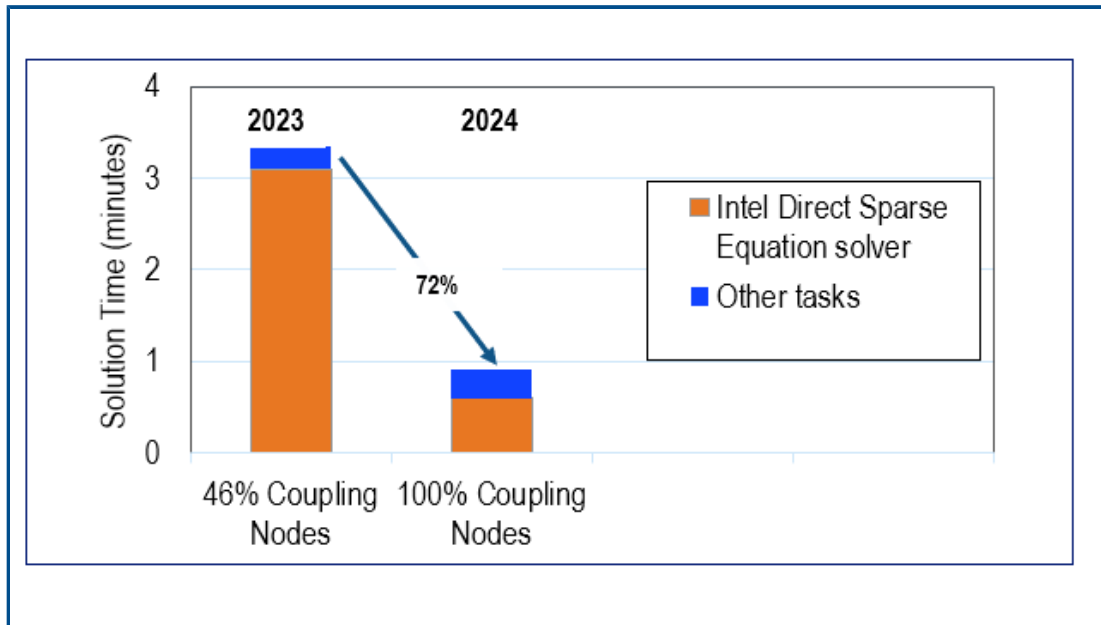
The meshing time with the Blended curvature-based mesher is reduced for assemblies that have multiple identical parts.

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

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

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

Performance Enhancements



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

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

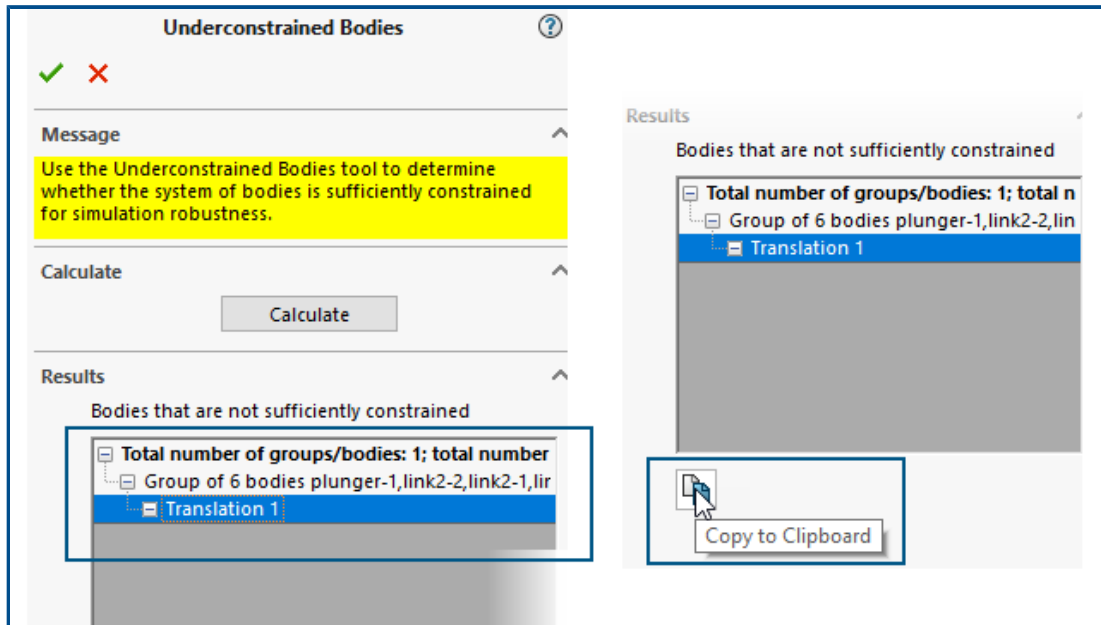
The solution time for these studies is shorter with the Intel Direct Sparse solver. In previous releases, when the number of coupling nodes was very large, only a subset of the coupling nodes participated in the distributed coupling constraints. In SOLIDWORKS Simulation 2024, the distributed coupling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

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

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

- Running larger linear dynamic studies is more efficient. The stress calculation of larger linear dynamic studies is optimized because of improved memory allocation by the solver.
- Improved memory estimate, allocation, and management by the solver allows the completion of large surface-to-surface bonded interaction sets that previously failed because of insufficient memory. This improvement applies to the SOLIDWORKS Simulation Professional and SOLIDWORKS Simulation Premium licenses.

Underconstrained Bodies Detection



There are several usability enhancements for the Underconstrained Bodies PropertyManager.

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

15

SOLIDWORKS Visualize

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.

Enhanced Capabilities for Creating Compelling Appearances



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

DSPBR is a material model for physically based rendering, supported by all renderers in the **3DEXPERIENCE**® platform. The shading model is easy to use and renderer independent. It combines metallic and dielectric scattering distribution functions, including transparency for thin-walled and volumetric objects. It also provides effects, such as emission, clear coat, metallic flakes, and sheen, to cover a range of appearances.

SOLIDWORKS Visualize provides appearances for an expanded range of material types and subtypes. The full Enterprise PBR Shading Model consists of more than 30 parameters, which may be overwhelming. To address this, the software introduces subtypes that expose a subset of the parameters relevant to specific material types. The subtypes simplify the user interface and enhance usability while keeping unnecessary parameters hidden.

Enhancements include:

- A simplified interface for selecting appearance types and optimizing their parameters. You can select appearance types from a list or by clicking thumbnail images.
- The ability to adjust textures and texture maps for almost all parameters, with greater control and fidelity.
- The ability to combine normal and displacement maps and to apply vector displacement.

Aside from the Enterprise PBR Shading Model, appearance subtypes include Basic, Metal, Glass, Plastic, Leather, Textile, Car Paint, Emissive, and Wood.


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

Sample projects and other assets are updated and improved for the DSPBR appearances. Additional appearances and assets are available in the Cloud content library.







Parameters for Basic Subtype

The **Basic** subtype includes the most commonly used parameters for applying textures and effects to surfaces.

Most parameters let you apply a texture and an effect on a surface as follows:

- To apply a texture, click .
- To apply an effect, enter a decimal value, or adjust the slider or spinner.

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

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

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SOLIDWORKS CAM

This chapter includes the following topics:

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

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

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

Additional Probe Cycle Parameters

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

Stop If Tolerance Exceeded

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

Values you can specify for this parameter:

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

The command associated with this parameter in the posted code is

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

Print (Ww) / Measuring Log

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

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

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

Values you can specify for this parameter:

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

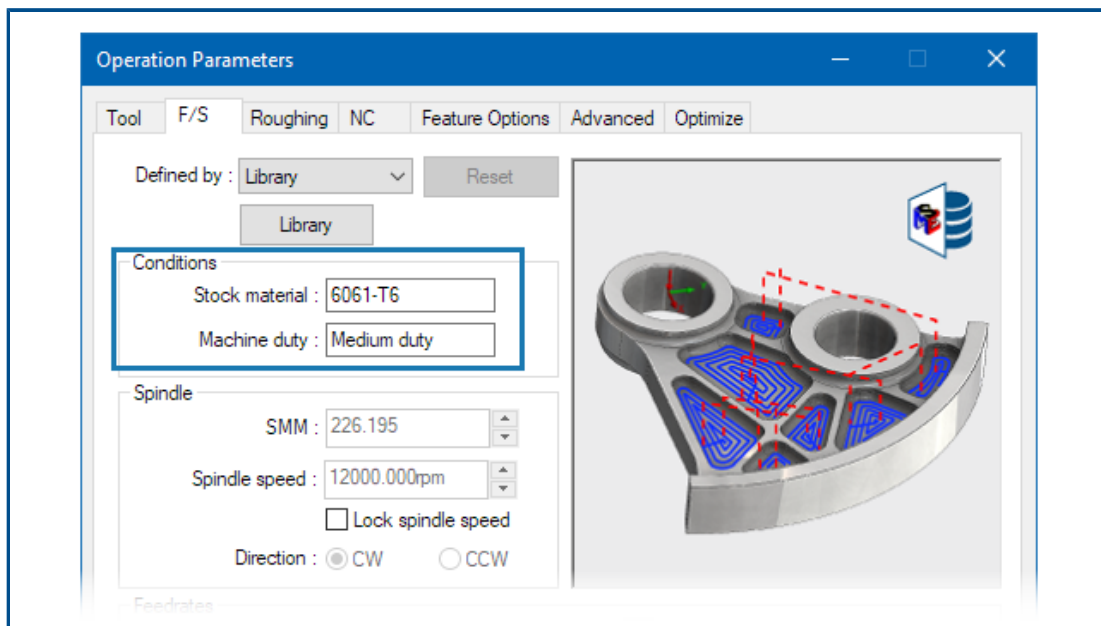
Canned Cycle Threading for Reverse Cuts

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

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

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

Correct Feed/Speed Data for Parts Comprising Assemblies

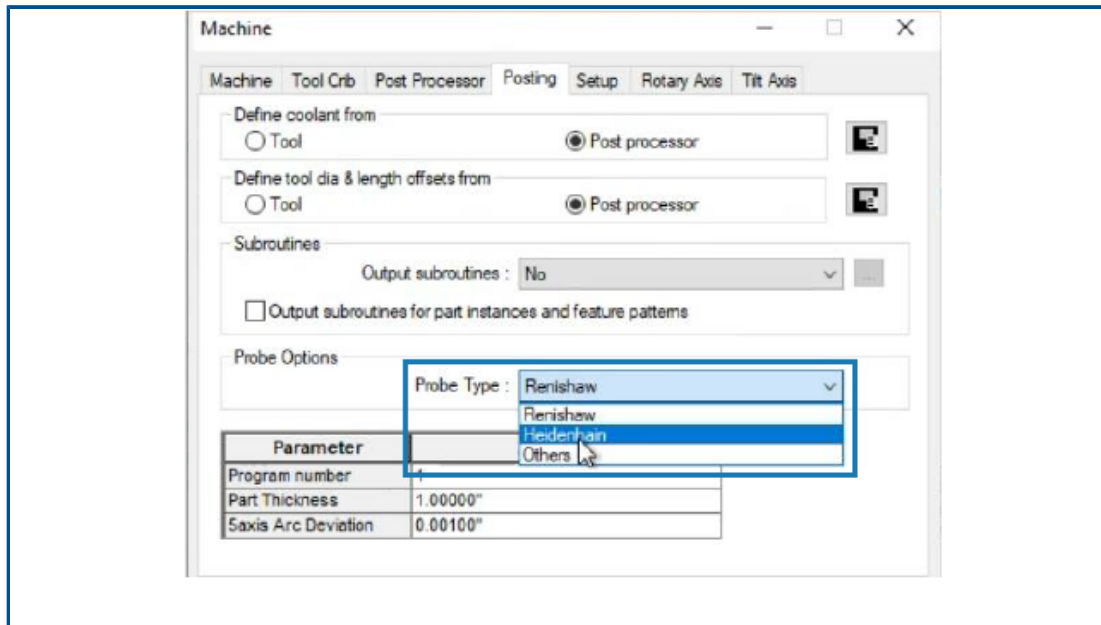


In Assembly mode, if the different parts or the multiple instances of a part comprising an assembly have different stock materials, then for each part or instance, the correct stock material appears.

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

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

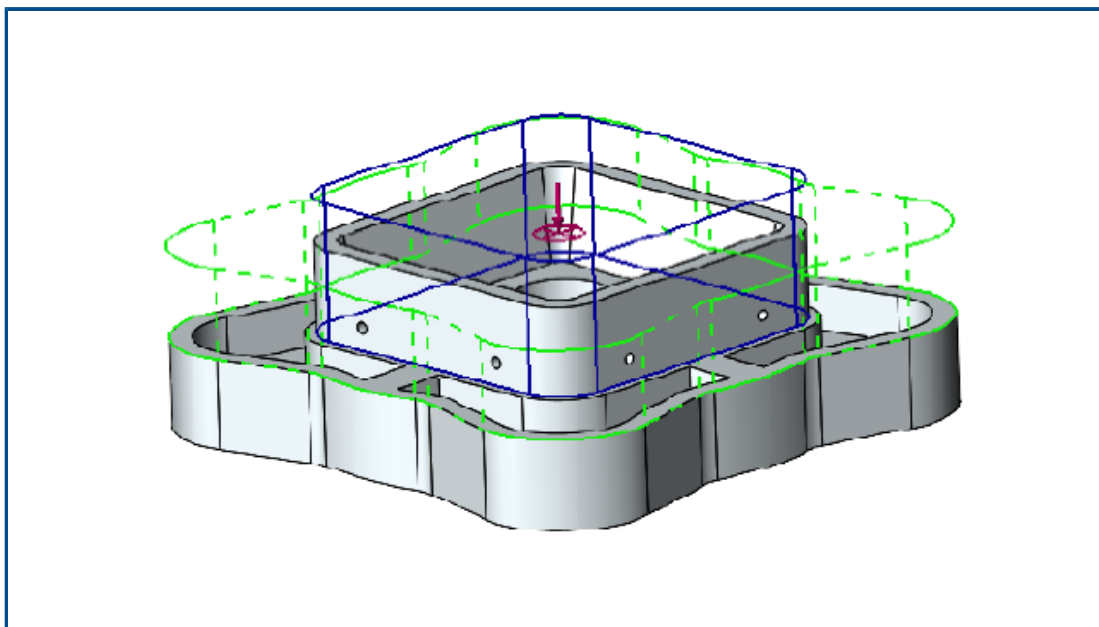
Heidenhain Probe Type



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

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

End Conditions for Islands in the 2.5 Axis Feature Wizard

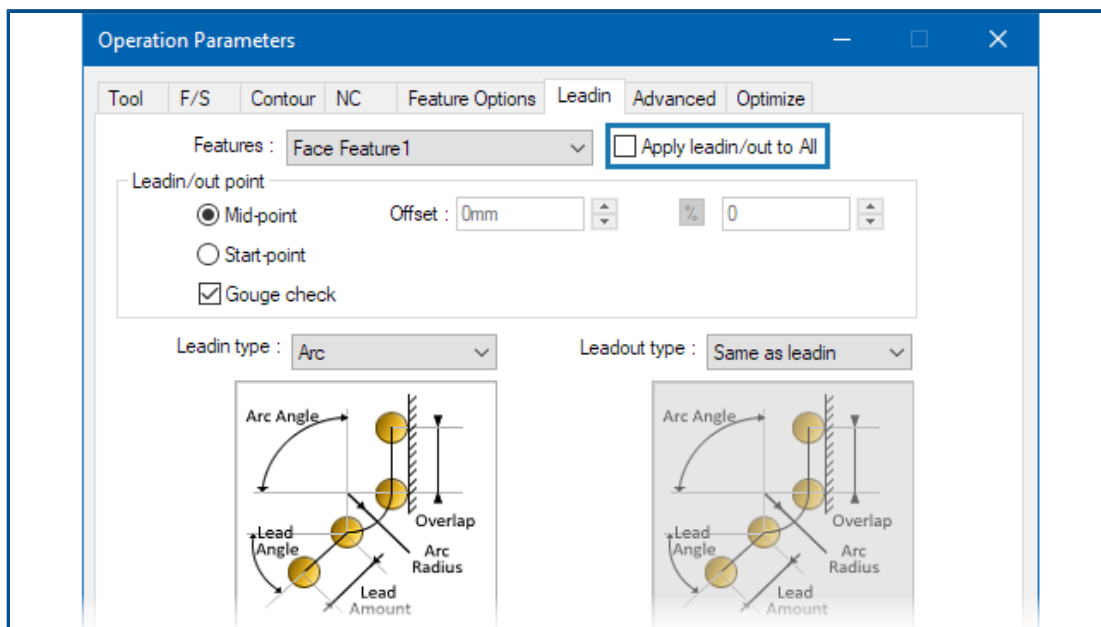


You can define the height of islands for 2.5 Axis features in two directions.

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

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

Leadin and Leadout Parameters for Linked Contour Mill Operations

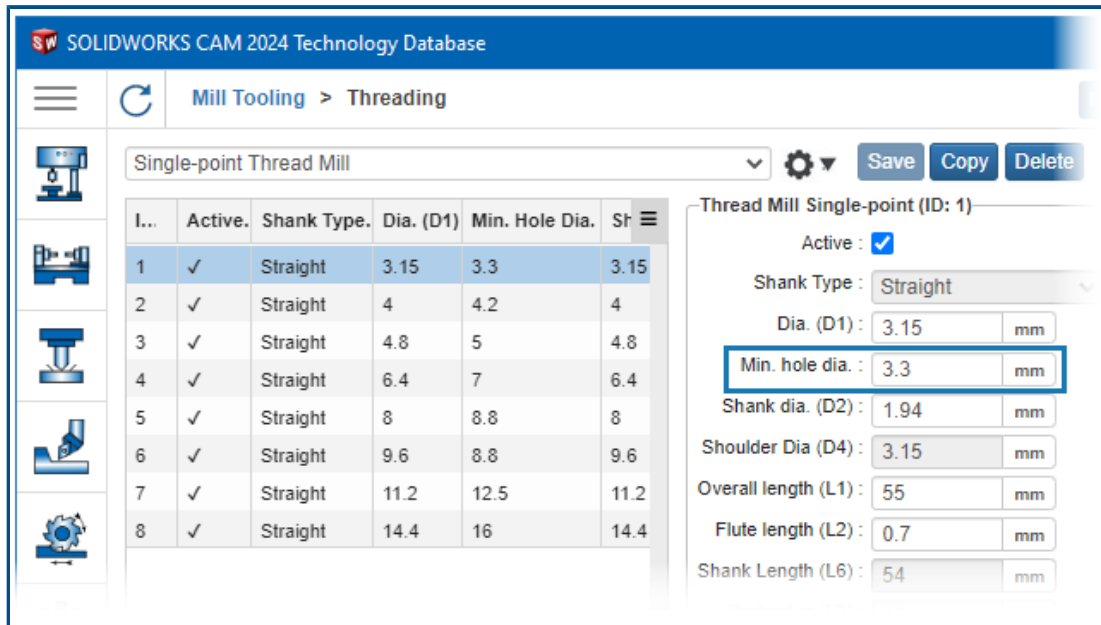


For linked Contour Mill operations, you can specify an option to copy the **Leadin** and **Leadout** parameters of the first Contour Mill operation to the other linked operations.

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

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

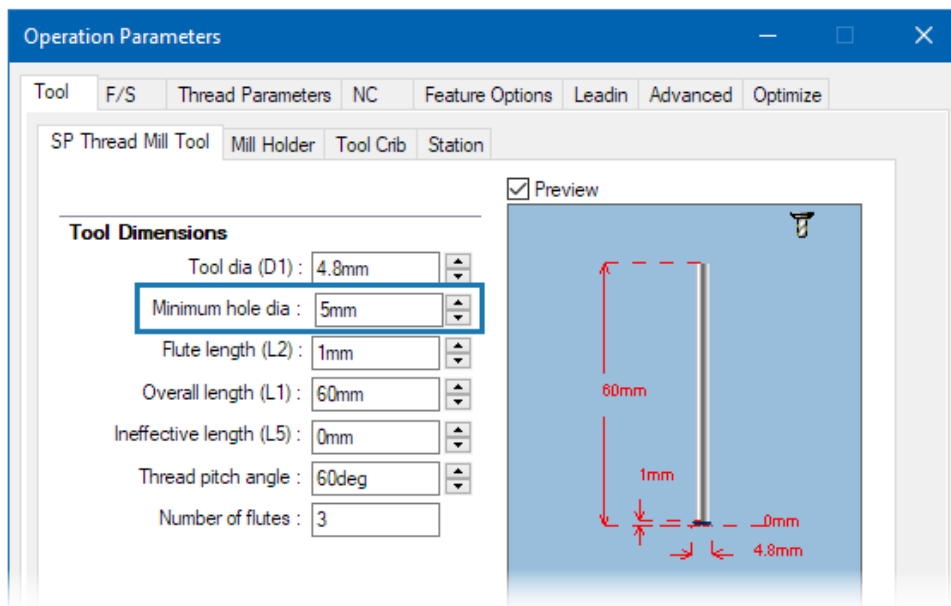
Minimum Hole Diameter for Thread Mill Operations



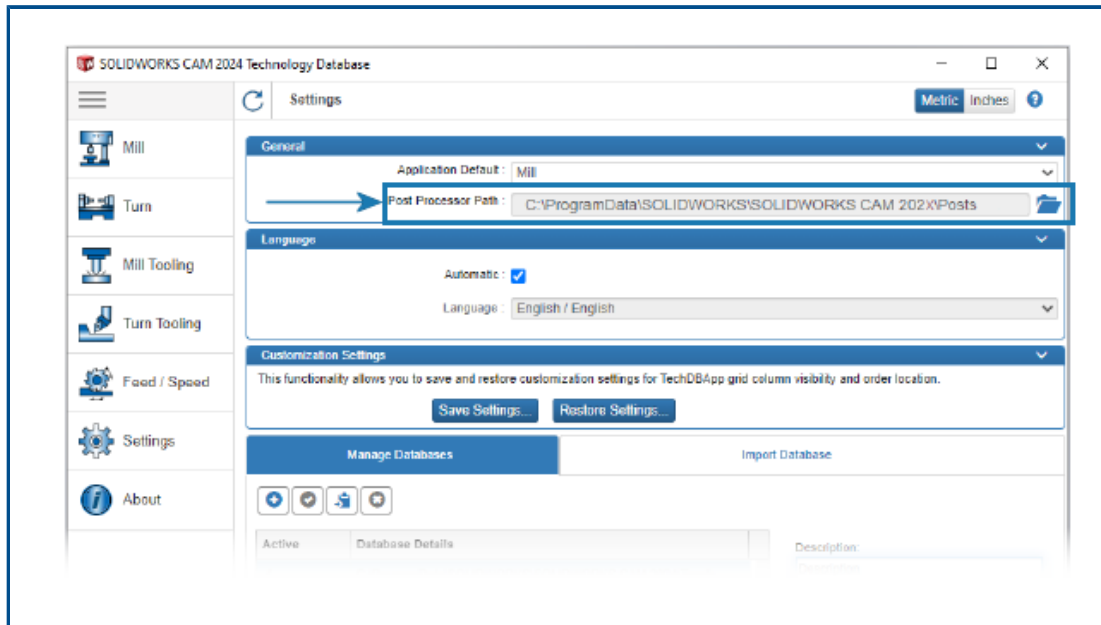
You can specify the minimum hole diameter for thread mill operations. In previous releases, this parameter was read-only.

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

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



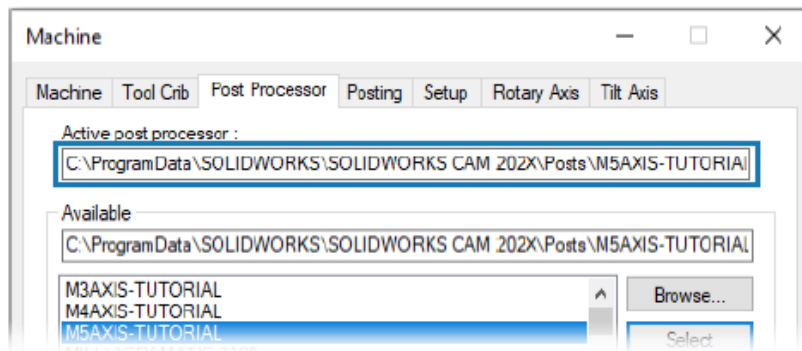
Post Processor Path



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

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

1. SOLIDWORKS CAM determines whether the post processor file is available in the folder for **Active post processor**.
If the folder is unavailable, the software loads the **Post Processor Path**.
2. SOLIDWORKS CAM searches for the post processor file in **Post Processor Path**.
3. When SOLIDWORKS CAM finds the post processor file, it displays the file path of the post processor file in the Machine dialog box on the Post Processor tab for **Active post processor**.



Probe Cycles

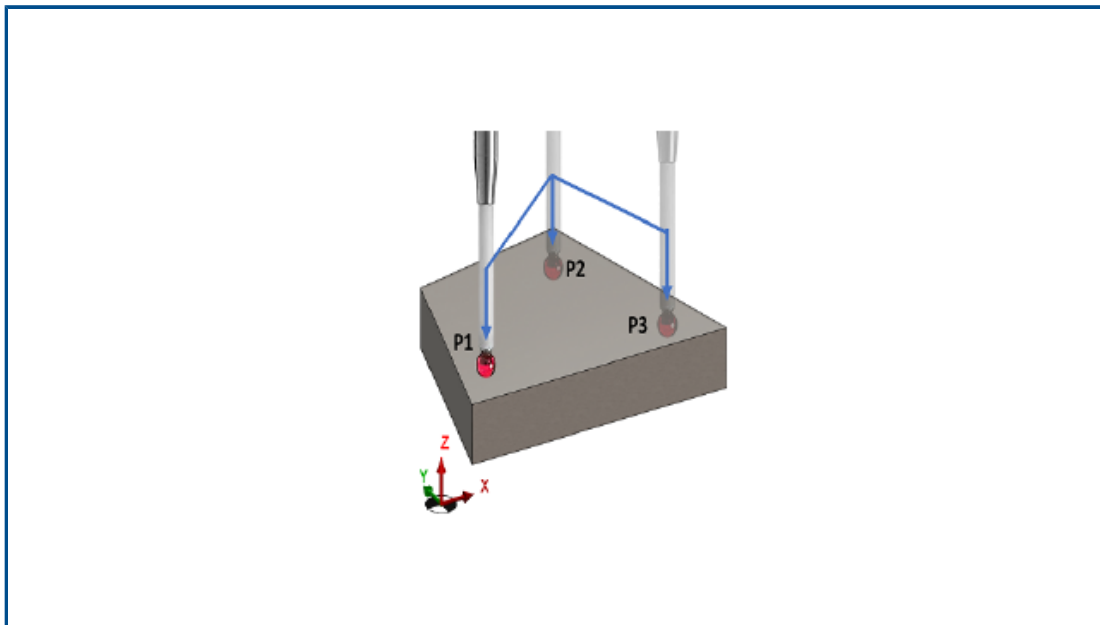
SOLIDWORKS CAM includes additional probing cycles to calibrate and measure planes and axes.

Probing cycles include:

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

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

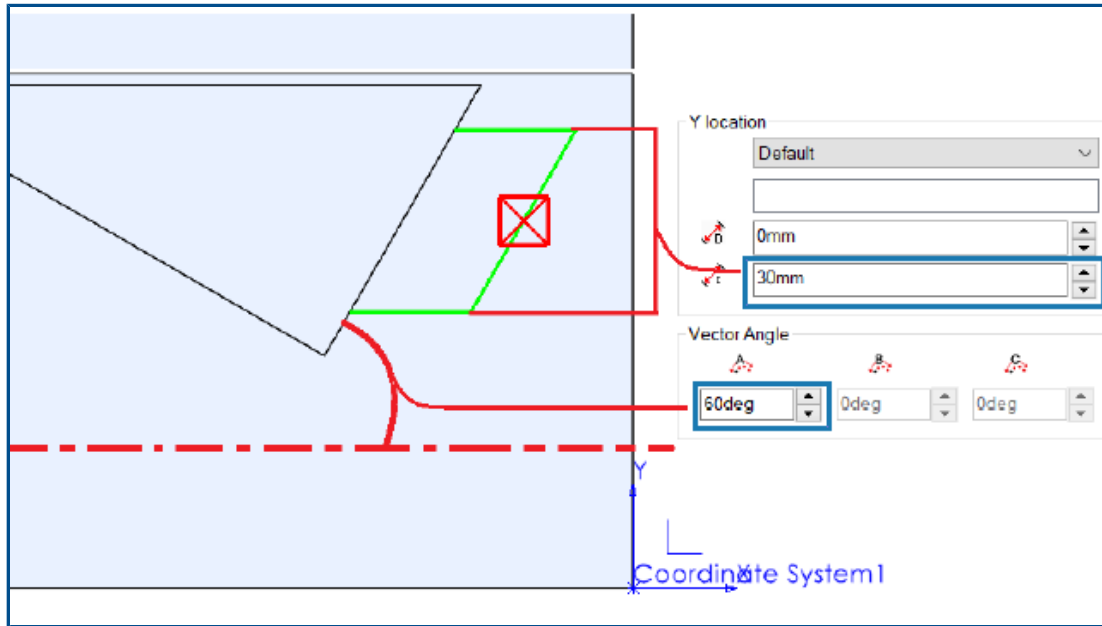
Three Point Plane



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

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

Angle Measurement (X/Y Axis)

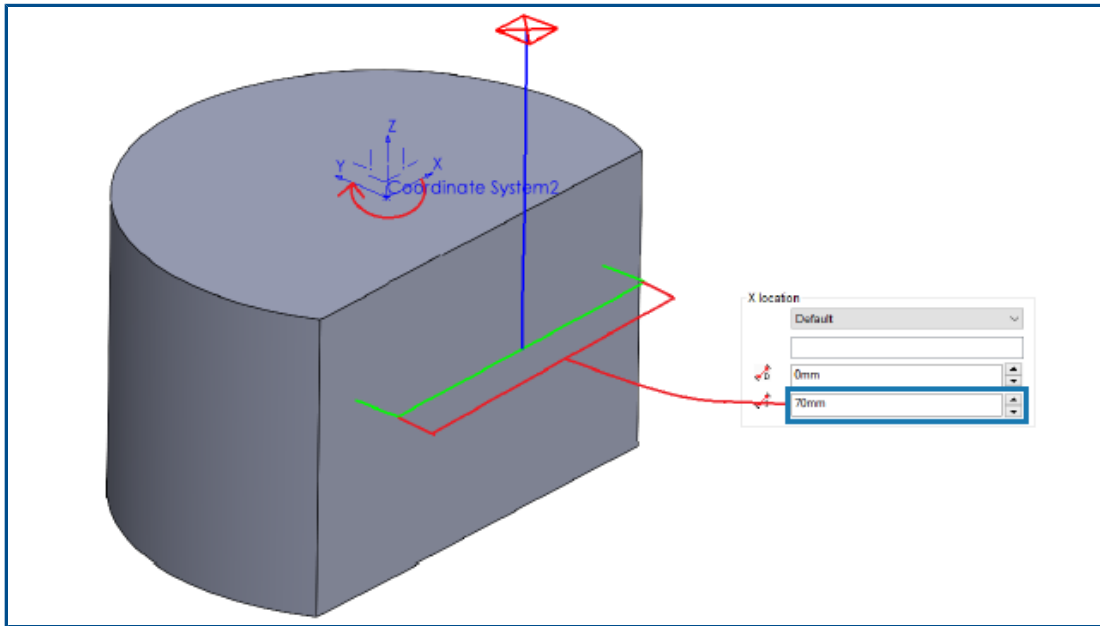


The **Angle Measurement (X Axis)** and **Angle Measurement (Y Axis)** probe cycles probe two points on a selected surface and calculate the angle of the face with respect to the X or Y axis, respectively.

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

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

4th Axis Measurement (X/Y Axis)



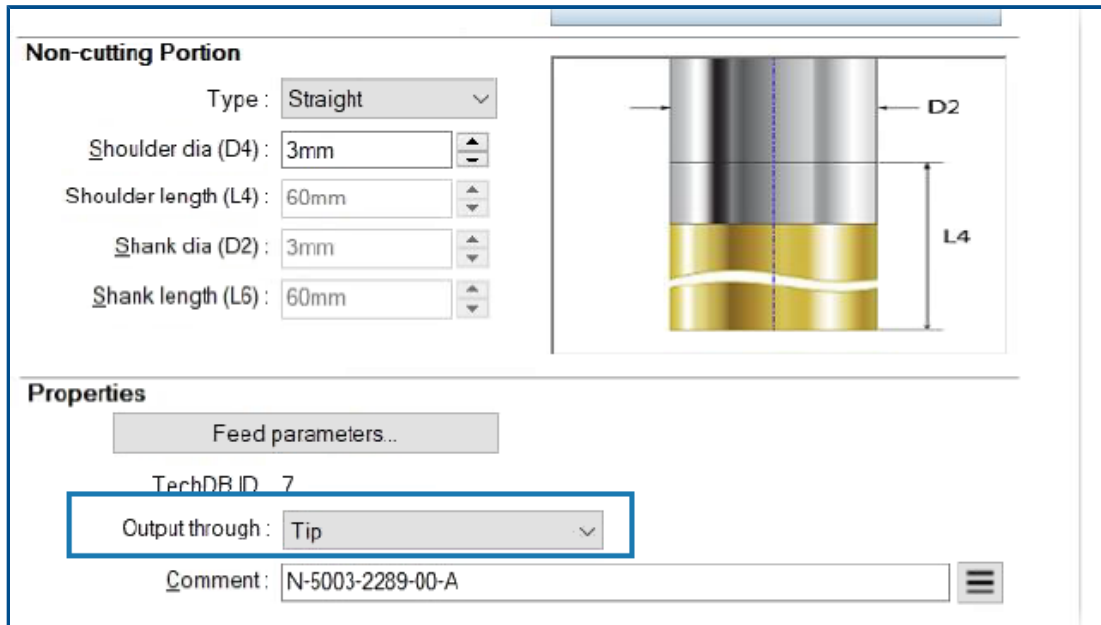
This probe cycle measures the slope of a selected surface between two points with respect to the fourth axis.

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

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

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

Probe Tool Output Options

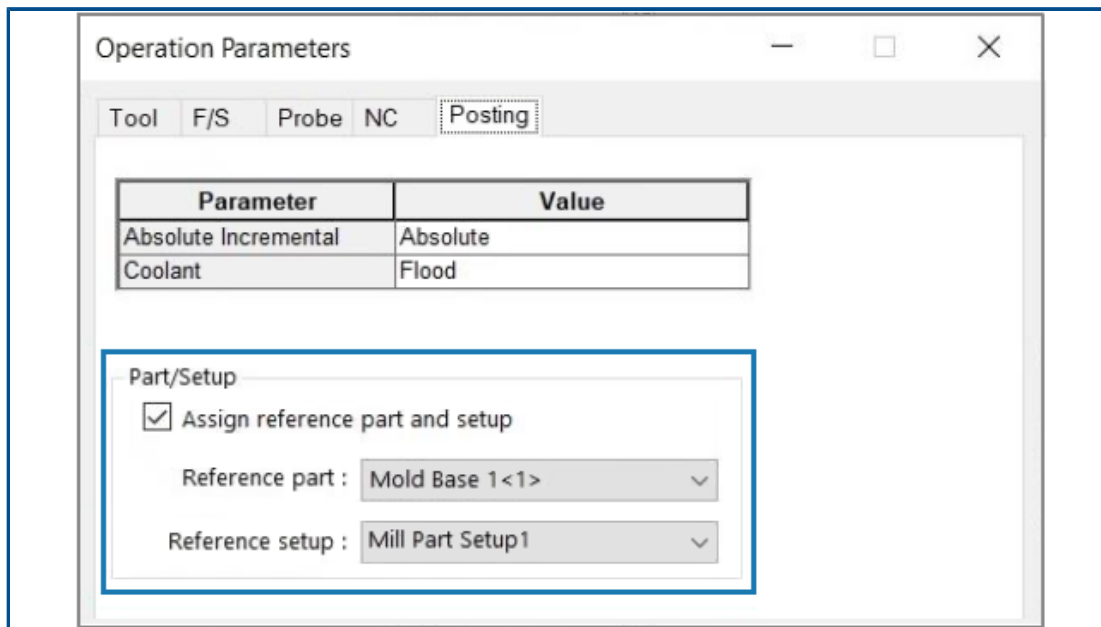


You can specify the **Output through** parameter for probe tools. This parameter generates the toolpath and G-code with the set tool reference point.

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

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

Probing Cycles in Assembly Mode



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

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

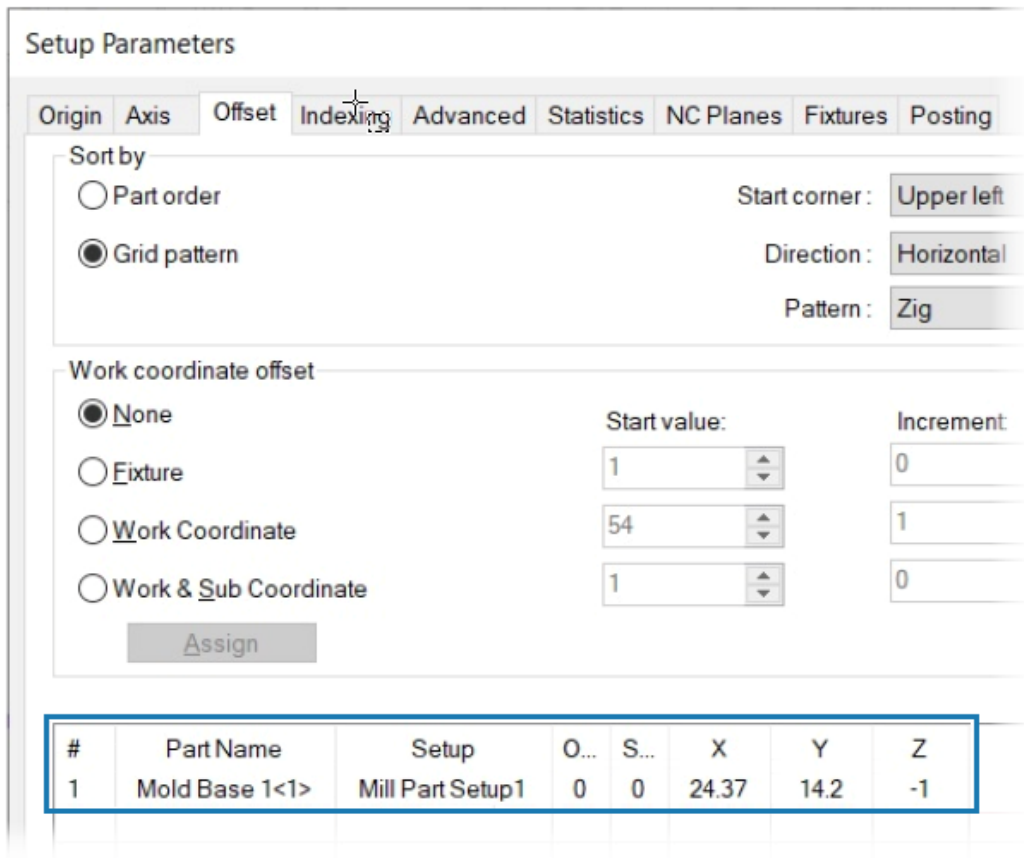
In the Operation Parameters dialog box, on the Posting tab, under **Part/Setup**, you can specify parameters in Assembly mode.

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

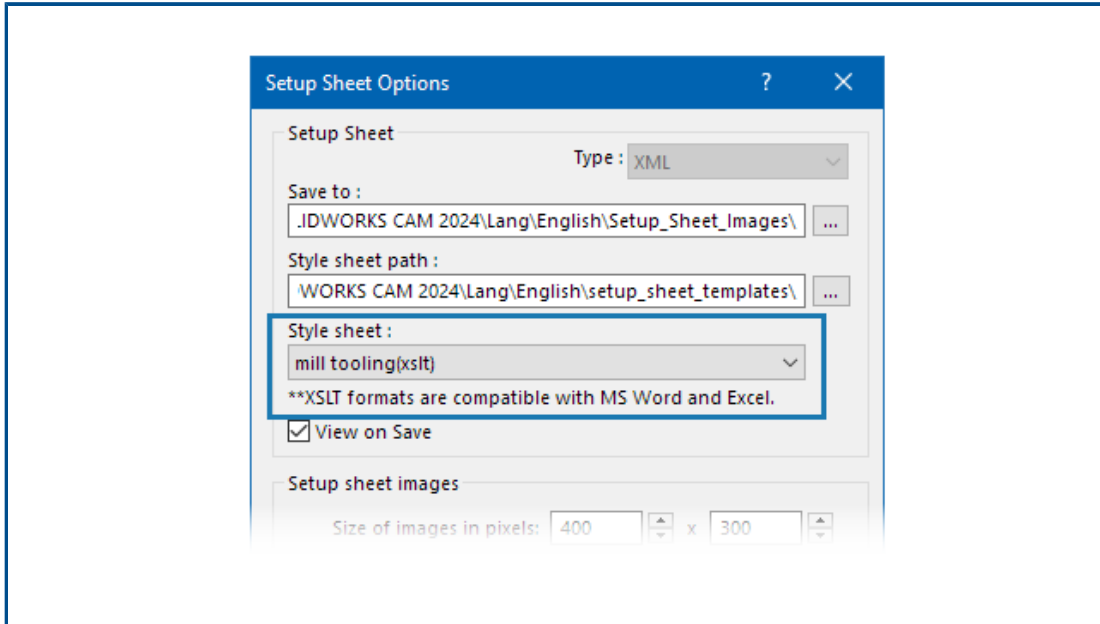
Parameter	Description
-----------	-------------

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

For **Probe** operations, the selections you make for **Reference part** and **Reference setup** are displayed in the part instances and work coordinates on the Offset tab of the Setup Parameters dialog box.



Setup Sheets



The default format for Setup Sheets is `.xslt` for compatibility with the latest browsers.

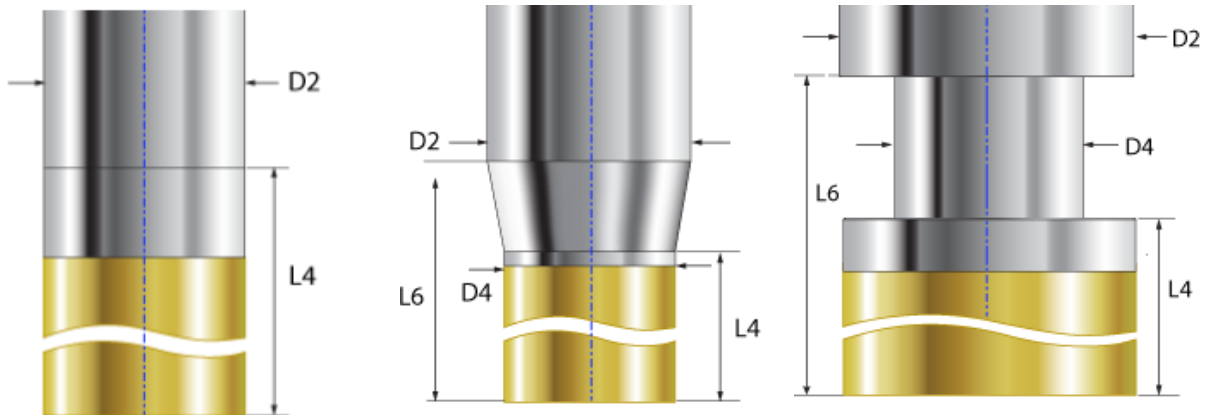
Shank Types for Mill Tools



You can define shank types (**Straight**, **Tapered**, or **Neck**) for any Mill Tool.

In previous releases, only certain Mill Tools could have shank types. You can specify shank types for the noncutting portion of these additional tools:

- Bore Tool
- Center Drill
- Countersink Tool
- Dovetail Tool
- Keyway Tool
- Lollipop Tool



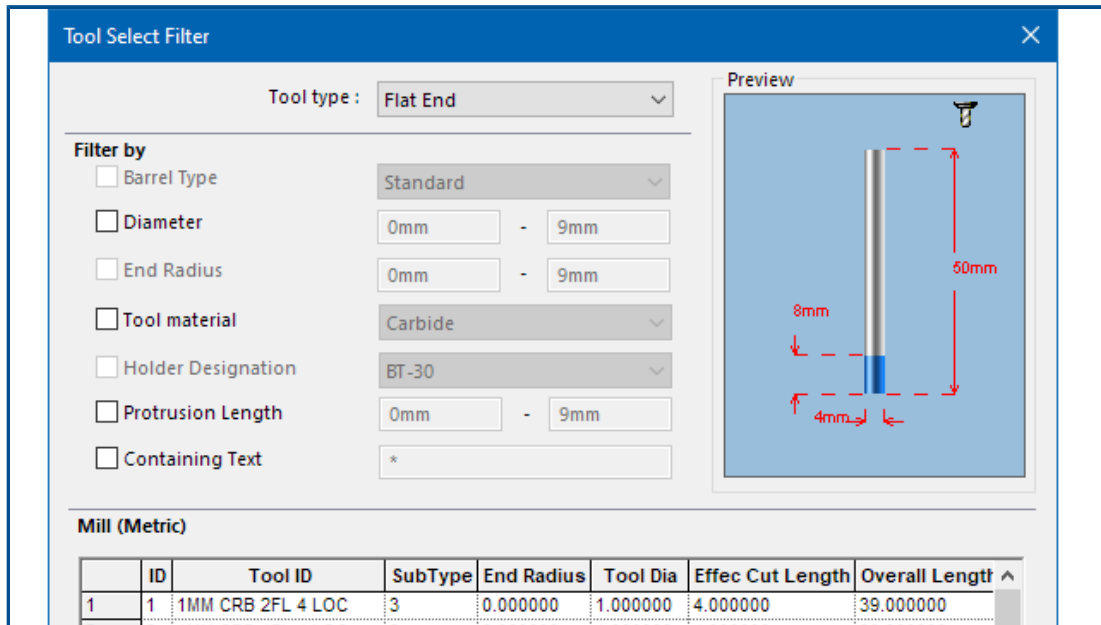
Straight. You can define the shoulder length and shank diameter.

Tapered. You can define the shoulder diameter, shoulder length, shank diameter, and shank length. The tapered portion of the tool is the noncutting portion of the cutting tool.

Neck. You can define the shoulder diameter, shoulder length, shank diameter, and shank length. The neck portion of the tool is the noncutting portion of the cutting tool.

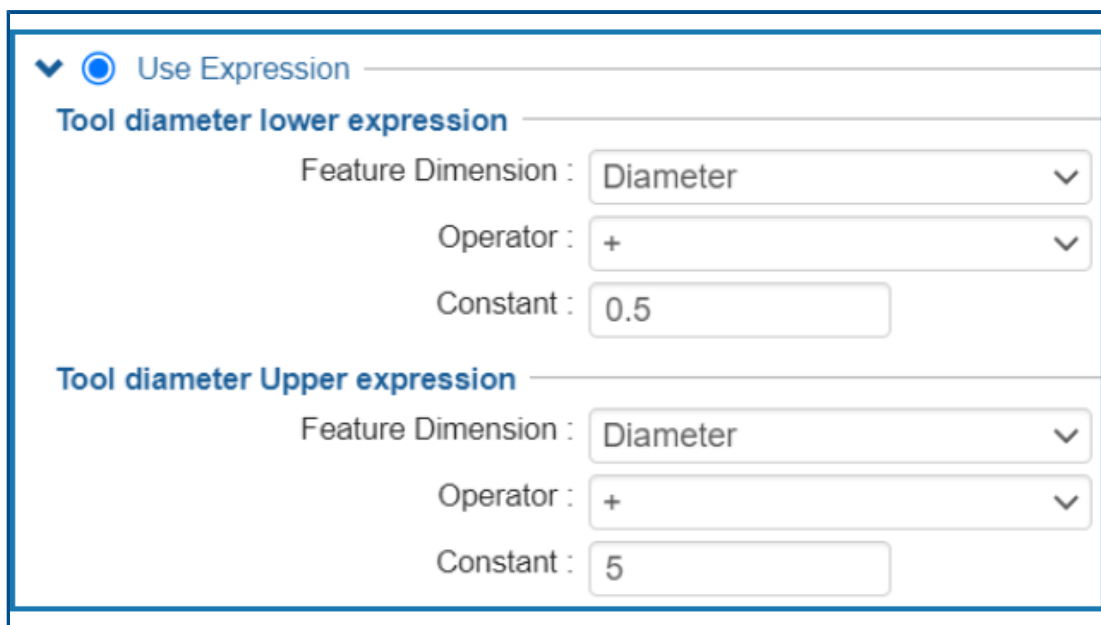
- D2 = Shank diameter
- D4 = Shoulder diameter
- L4 = Shoulder length
- L6 = Shank length

Tool Select Filter Dialog Box



You can resize the Tool Select Filter dialog box to see additional table columns.

Tool Selection - Flute Length



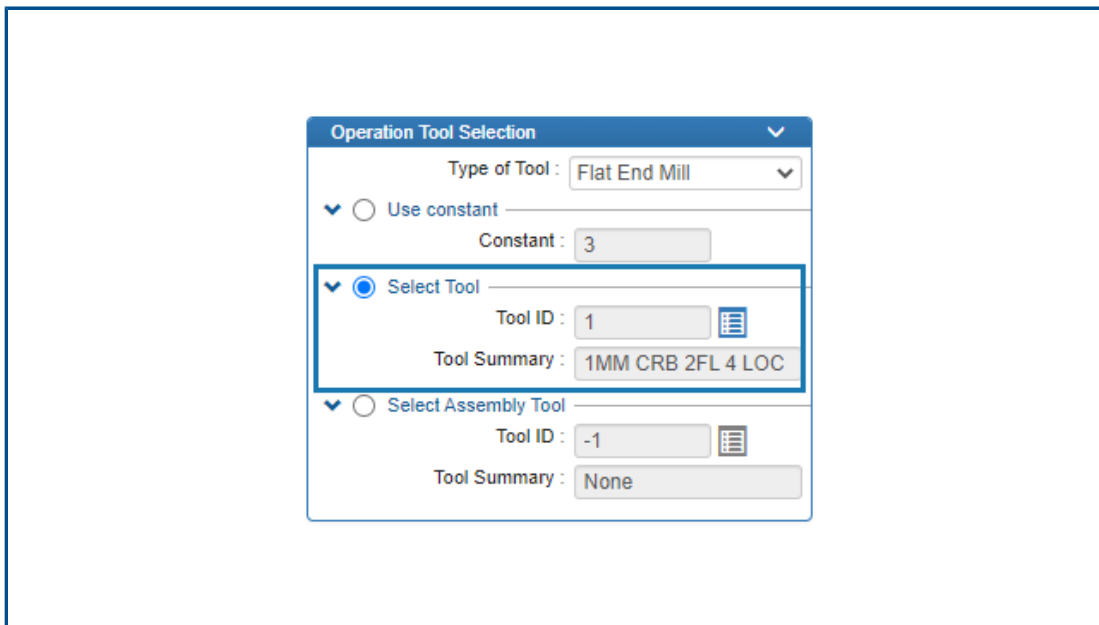
When you specify tool selection criteria based on **Use Expression** and not on a specific tool, SOLIDWORKS CAM accounts for the tool's flute length.

When you run **Generate Operation Plan**, for each operation that you define the tool selection criteria with a Tool diameter lower/upper expression, the following rules apply:

- If the tool crib has two or more tools with identical diameter values matching the expression criteria, SOLIDWORKS CAM accounts for the flute length to assign the tool. It selects the tool with a flute length more than the feature depth. If all tools have a flute length more than the feature depth, SOLIDWORKS CAM selects the tool with a flute length closest to the feature depth.
- If SOLIDWORKS CAM still finds two or more tools, it uses the rules of Stock/Tool Material Mapping to select a tool.

For example, consider a rectangular pocket with a feature depth of 75mm. Based on the feature strategy assigned to this feature, the tool selection criteria selects a 25mm Flat End Mill. The tool crib has two Flat End Mill tools with identical diameters of 25mm. However, one tool has a 50mm flute length and the other has an 80mm flute length. SOLIDWORKS CAM selects the tool with the 80mm flute length because it is closer in value to the feature depth.

Tool Selection - Tool Crib Priority



SOLIDWORKS CAM has better tool selection logic when you select **Tool crib priority** in the Technology Database (TechDB).

SOLIDWORKS CAM has optimized tool selection logic so appropriate tools are available in the active tool crib:

- If the tool assigned in the TechDB for a specific operation is not in the active tool crib, SOLIDWORKS CAM adds it to the tool crib even though smaller tools might be in the active tool crib. (If you selected a tool by referencing it to a specific **Machine ID** in the TechDB.) If another tool with similar parameters is in the active tool crib, SOLIDWORKS CAM uses that tool.
- If you specify that the resultant tool derived from the expressions defined in the TechDB as inactive, SOLIDWORKS CAM does not add it to the active tool crib. It uses the subsequent tool selection rules to add an active tool to the tool crib.

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SOLIDWORKS Composer

This chapter includes the following topics:

- **Offline Help for SOLIDWORKS Composer Products**
- **Support for SpeedPak Configurations in SOLIDWORKS Composer**

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

Offline Help for SOLIDWORKS Composer Products

Offline Help for all SOLIDWORKS Composer products is available as a PDF instead of in HTML format.

In earlier releases, offline Help worked only in Microsoft Internet Explorer. Now it is browser independent.

Support for SpeedPak Configurations in SOLIDWORKS Composer

You can translate SOLIDWORKS assembly files containing components in SpeedPak configurations to SOLIDWORKS Composer.

The SpeedPak components are switched to their parent configurations to enable translation of these components to SOLIDWORKS Composer.

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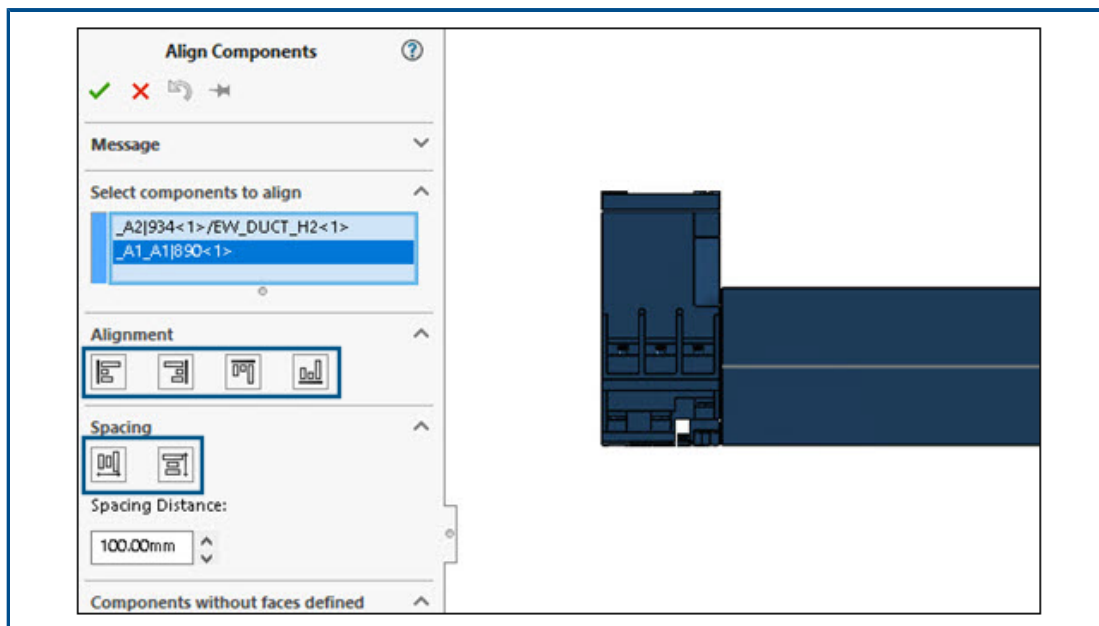
SOLIDWORKS Electrical

This chapter includes the following topics:

- **Aligning Components**
- **Changing the Length of Multiple Rails and Ducts**
- **Filtering Auxiliary and Accessory Parts**
- **Auto Balloons in 2D Cabinets**
- **Removing Manufacturer Part Data**
- **Resetting an Undefined Macro Variable**
- **Shortening Lists Using Ranges**
- **SOLIDWORKS Electrical Schematic Enhancements**
- **SOLIDWORKS Electrical Performance Improvement**

SOLIDWORKS® Electrical is a separately purchased product.

Aligning Components

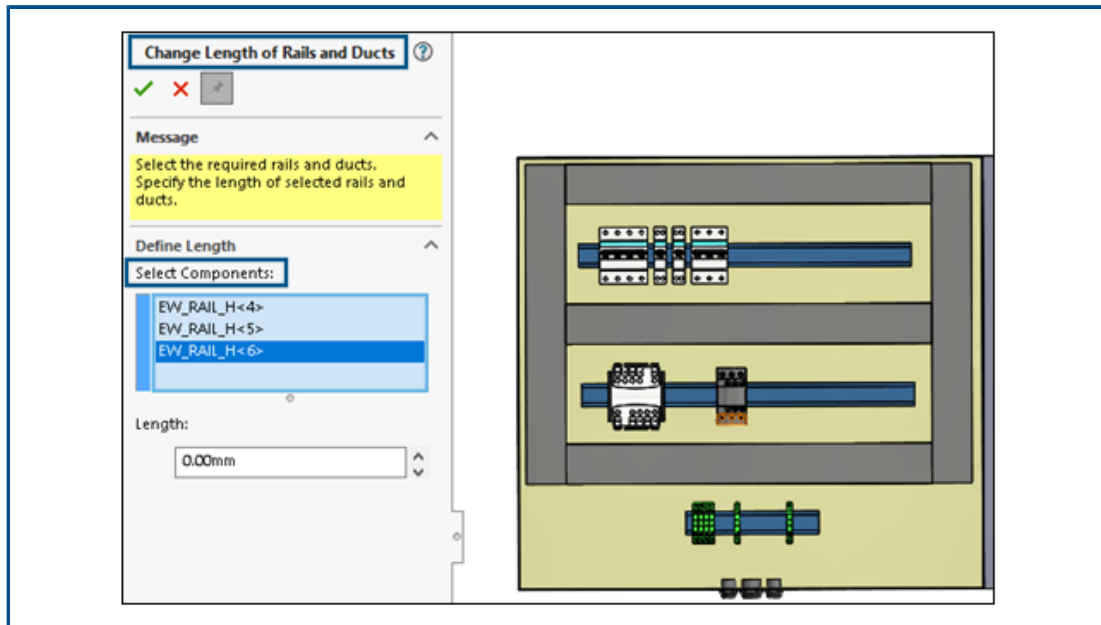


When you use **Align Components** while designing 3D cabinet layouts, you can preview changes in the graphics area.

This significantly reduces the effort required to align SOLIDWORKS components in 3D cabinet layouts.


The Align Component PropertyManager has a simplified and improved workflow.

Changing the Length of Multiple Rails and Ducts

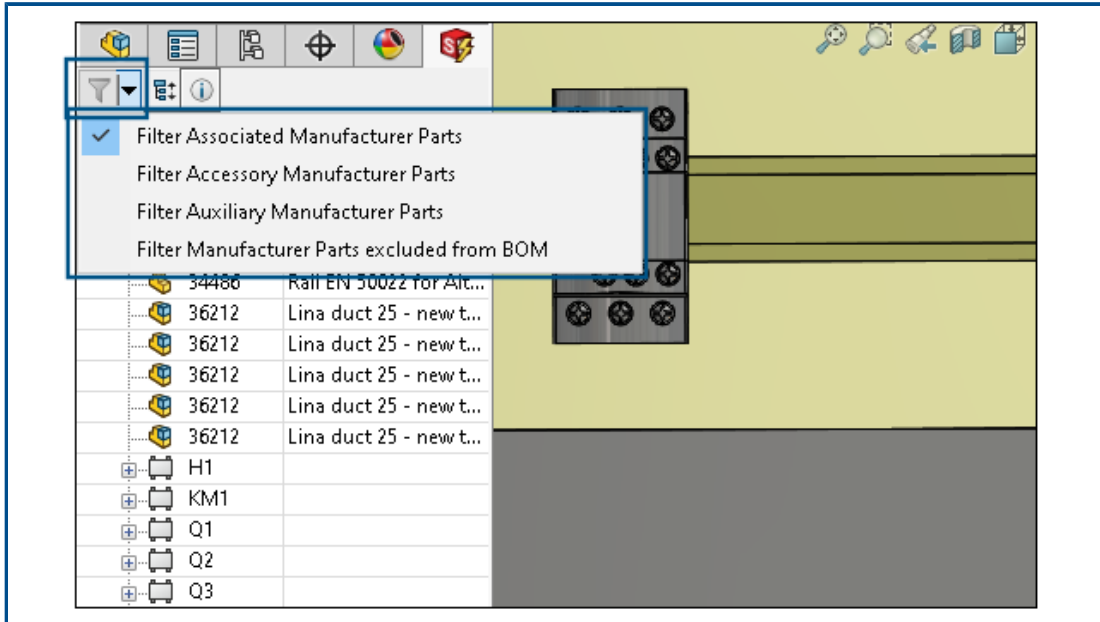


You can change the length of multiple rails and ducts simultaneously. In earlier releases, you could only change the length of a single rail or duct. The multiselection of rails and ducts makes the process of 3D cabinet creation faster.

To change the length of multiple rails and ducts:

1. In the SOLIDWORKS Electrical 3D menu, click **Change Length of Rails and Ducts** .
2. In the PropertyManager, under **Define Length** > **Select Components**, select multiple rails and ducts in the graphics area.

Filtering Auxiliary and Accessory Parts



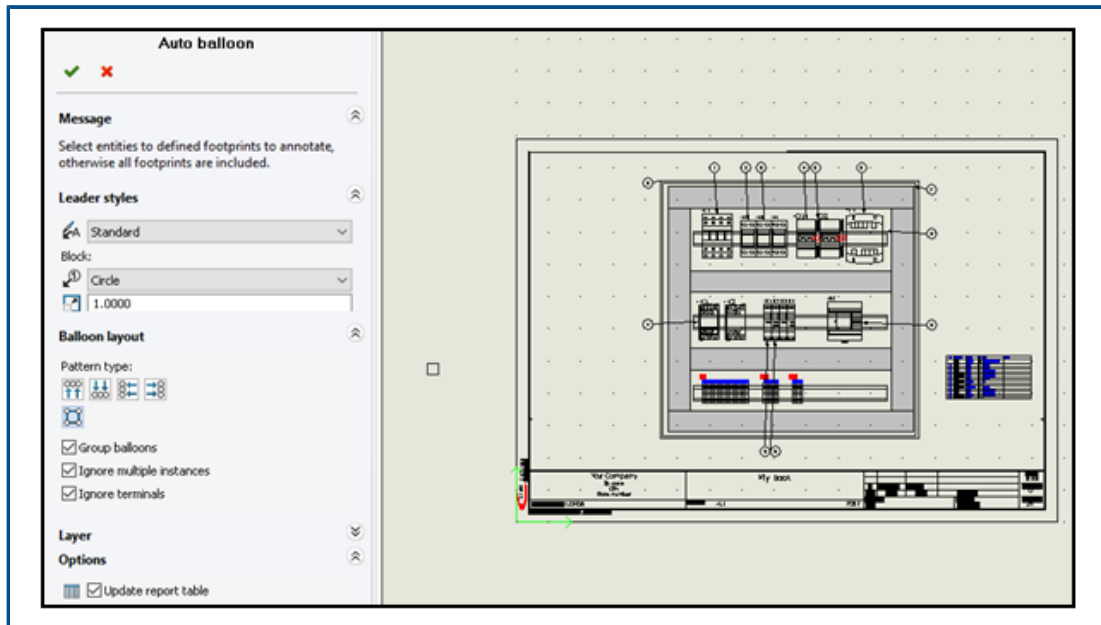
In SOLIDWORKS Electrical, you can filter manufacturer parts based on your selection. You can filter:

- Associated manufacturer parts
- Accessory manufacturer parts
- Auxiliary manufacturer parts
- Manufacturer parts excluded from the BOM

You can use the list in **Filter Manufacturer Parts** in the **Electrical Manager** tree to filter various types of manufacturer parts. **Show/Hide Associated Components** is replaced by this filter option.

This feature is also available in the 2D cabinet layout of SOLIDWORKS Electrical Schematic.



Auto Balloons in 2D Cabinets



You can insert auto balloons in SOLIDWORKS Electrical 2D cabinet layout drawings.

Inserting Auto Balloons in 2D Cabinets

To insert auto balloons in 2D cabinets:



1. Click **Cabinet Layout** > **Auto Balloon** .
2. Select a drawing view in which to insert the balloons.
3. In the PropertyManager, specify options and click .

Auto Balloon PropertyManager

To open this PropertyManager:






1. Click **Cabinet Layout** > **Auto Balloon** .

Leader styles

	Leader style	Specifies the predefined style to apply to leaders.
	Block	Specifies the block to use for the balloons.
	Scale	Specifies a number for the scale to apply to the block used for balloons.

Balloon layout

Specifies the **Pattern type**.

	Top	Displays balloons on the top of the cabinet drawing.
	Bottom	Displays balloons on the bottom of the cabinet drawing.
	Left	Displays balloons on the left of the cabinet drawing.
	Right	Displays balloons on the right of the cabinet drawing.
	Square	Displays balloons in a square surrounding the cabinet drawing.
	Group balloons	Displays the arrows of the grouped balloons with less tilt.
	Ignore multiple instances	Inserts balloons only for the first instance of the same manufacturer part.
	Ignore terminals	Does not insert balloons for the terminal strip.

Layer

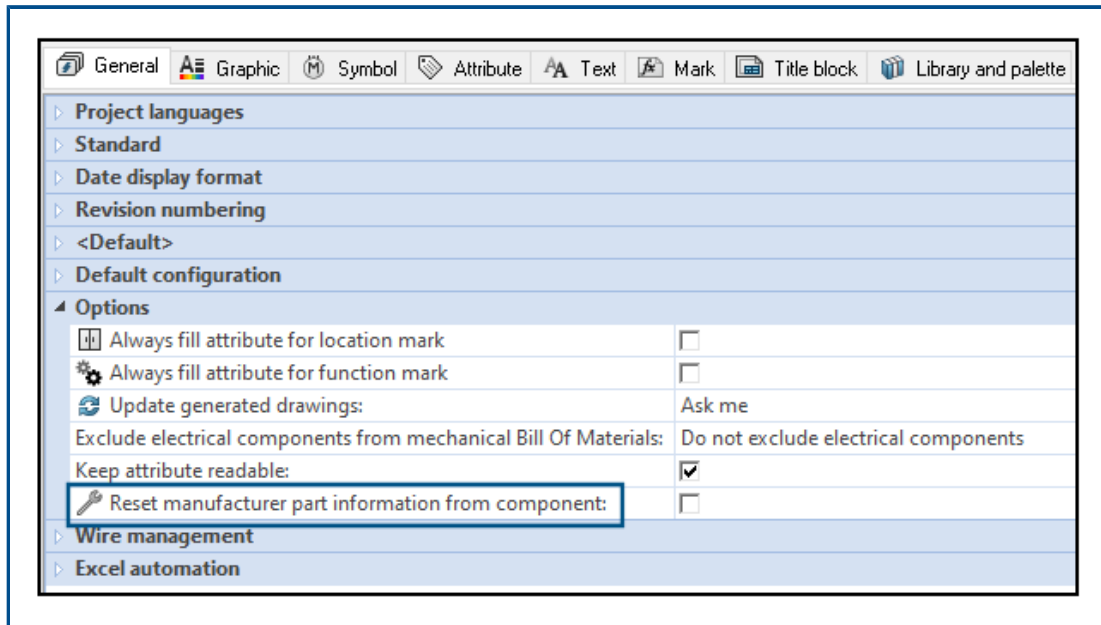
Specifies the layer on which to insert the balloons.

Options

Insert report table. Inserts a report table filtered from the content of the current document.

- If the document contains a report table, select **Update report table** to update the report table.
- The number of balloons associated with parts in the report appears in a separate report template.

Removing Manufacturer Part Data

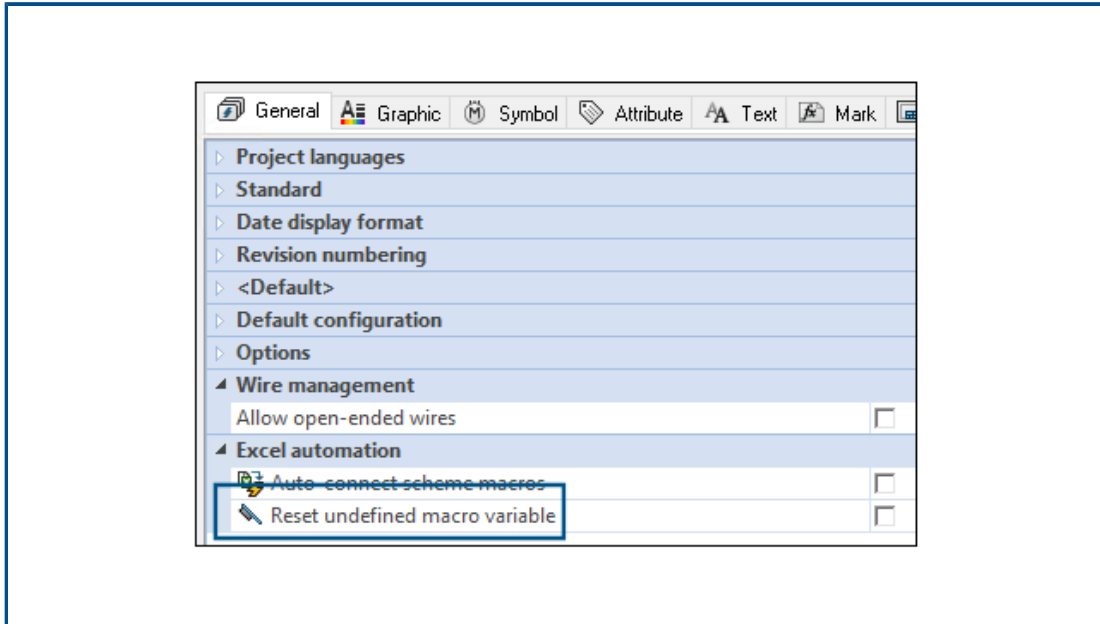


You can clear manufacturer part information when deleting or replacing a part from a component.

To remove manufacturer part data, click **Electrical Project > Configurations > Project**. In the Electrical Project Configuration dialog box, in the **General** tab, under **Options**, select **Reset manufacturer part information from component**. This resets the related information such as manufacturer data, terminal mark when you delete or replace it with a different part.

The option is cleared by default. If you clear this option, the part retains the terminal numbers even after you delete or replace it.

Resetting an Undefined Macro Variable



Excel automation lets you automatically reset undefined macro variables.

To reset undefined macro variables, click **SOLIDWORKS Electrical > Configurations > Project**. In the Electrical Project Configuration dialog box, on the **General** tab, under **Excel automation**, select **Reset undefined macro variable**. When you select this option, the %xxx% variable does not remain in the inserted macro. It is replaced by:

- An empty string
- A removed object
- Associated default object (like function or location)

Shortening Lists Using Ranges

The image shows three tables illustrating the use of ranges in report configuration. The top table shows a list of marks for '1SBL137201R1100' as a long string: '-K1, -K2, -K3, -K4, -K5, -K6, -K7, -K8, -K9, -K10'. The middle table shows the same list shortened to a range: '-K1...-K10'. The bottom table shows a similar shortening for '1B50UM' from '-Q1, -Q2, -Q3, -Q4, -Q5, -Q6, -Q6, -Q6, -Q9, -Q10, -Q11, -Q12, -Q13, -Q14, -Q15' to '-Q1...-Q6, -Q9...-Q15'. To the right, the 'Columns' tab of the Report Configuration dialog box is shown, with 'Merge rows' and 'Value range' options highlighted.

Reference	Mark	Descr
1	-K1, -K2, -K3, -K4, -K5, -K6, -K7, -K8, -K9, -K10	AF09

Reference	Mark	Descr
1	-Q1, -Q2, -Q3, -Q4, -Q5, -Q6, -Q6, -Q6, -Q9, -Q10, -Q11, -Q12, -Q13, -Q14, -Q15	1pole

Reference	Mark	Descr
1	-K1...-K10	AF09

Reference	Mark	Descr
1	-Q1...-Q6, -Q9...-Q15	1pole

In report configuration, when you merge rows, the software lists consecutive values as a range for merged rows instead of listing each individual value in the range.

In the Report configuration edition dialog box, under **Columns**, select **Value range**. To activate this option, select **Merge rows**. You can activate this option for multiple columns at once.

SOLIDWORKS Electrical Schematic Enhancements

SOLIDWORKS Electrical Schematic offers an improved user experience.

- In drawings, you can move entities using the arrow keys.
- The grid point size for the project sheets adapts automatically to the screen resolution.
- In a schematic project, when you set the side panels to **Auto Hide**, the panels retain the auto hide setting. This behavior increases the app's usability.

SOLIDWORKS Electrical Performance Improvement

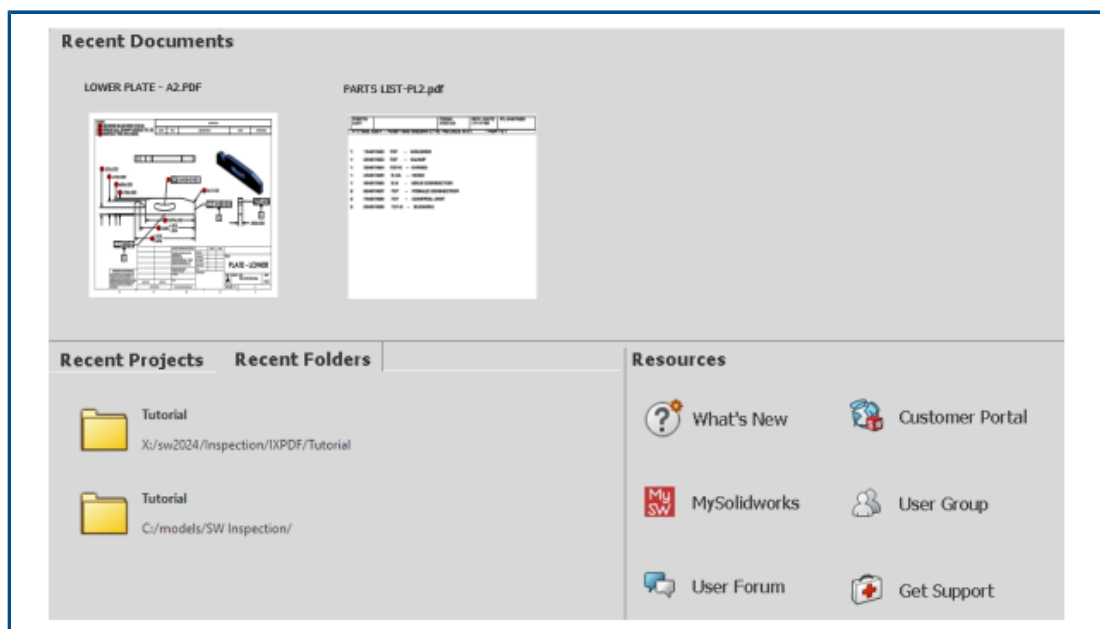
Archiving a project for remote users (VPN connection) is improved and is much faster now.

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SOLIDWORKS Inspection

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

Welcome Page



The redesigned Welcome to SOLIDWORKS Inspection page in SOLIDWORKS Inspection Standalone improves usability.

The welcome page includes:

- **Recent Documents**
- **Recent Folders**
- **Recent Projects**
- **Resources**

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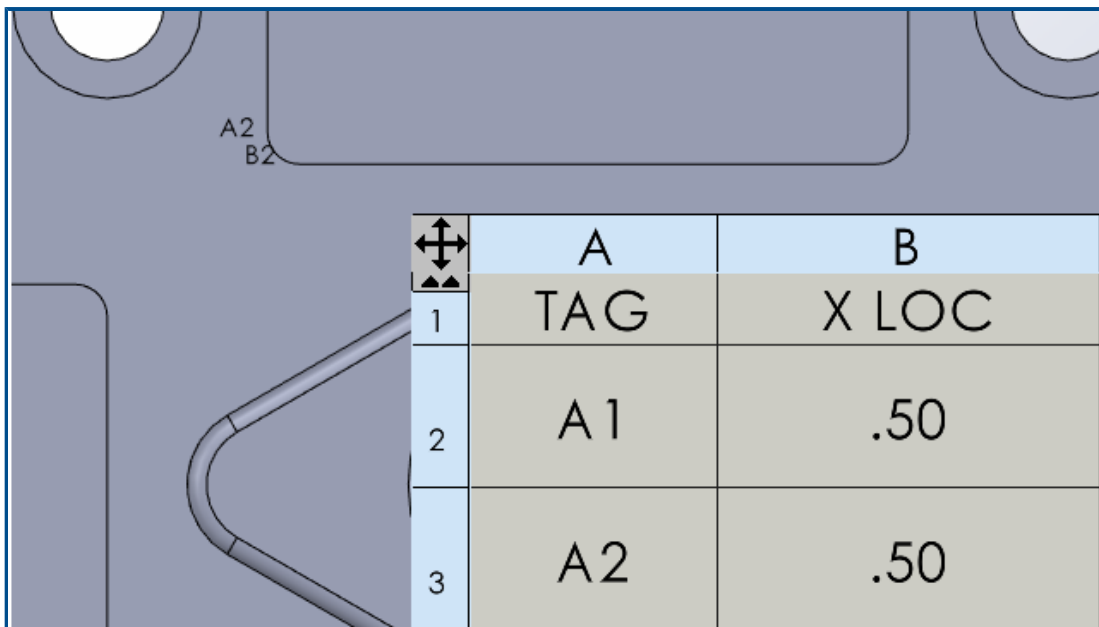
SOLIDWORKS MBD

This chapter includes the following topics:

- **Hole Tables**
- **Repairing Dangling Dimensions**
- **Adding a Decimal Separator in Geometric Tolerance Symbols**
- **Controlling Visibility of Annotations through Solid Geometry**
- **Displaying Dual Dimensions in Geometric Tolerance Symbols**
- **Creating Thickness Dimensions for Curved Surfaces**
- **Displaying Half Angles of Conical Dimensions**
- **Exporting Custom Properties to STEP 242**
- **Viewing Annotations and Dimensions**

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

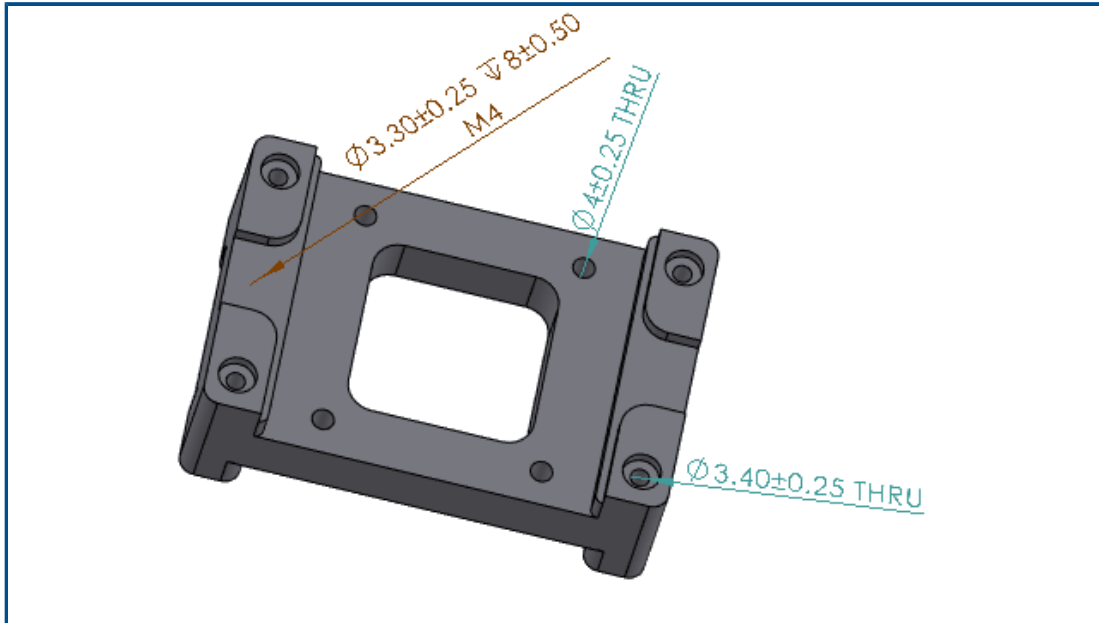
Hole Tables





	A	B
1	TAG	X LOC
2	A1	.50
3	A2	.50

You can include a hole table when you publish a part to 3D PDF.

Repairing Dangling Dimensions



You can repair dangling dimensions.

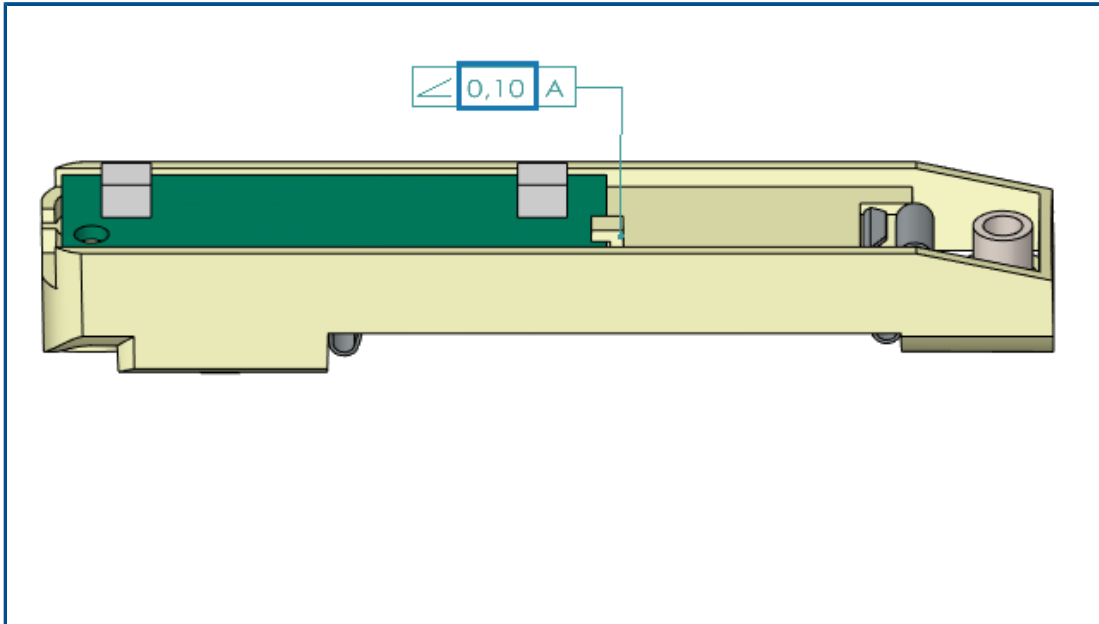
You can edit the dangling dimensions to reattach them to a feature in the model. This applies to dimensions created using the DimXpert tools, such as **Size Dimension** , **Location Dimension** , and the **Angle Dimension** tool.

To repair dangling dimensions:

1. Open a part or assembly that contains dangling dimensions in DimXpert.
2. In the FeatureManager® design tree, right-click a feature and select **Edit Feature**.
3. In the **Reference** dialog box, select the missing reference to repair the dangling dimension.

This tool is available for DimXpert dimensions.

Adding a Decimal Separator in Geometric Tolerance Symbols

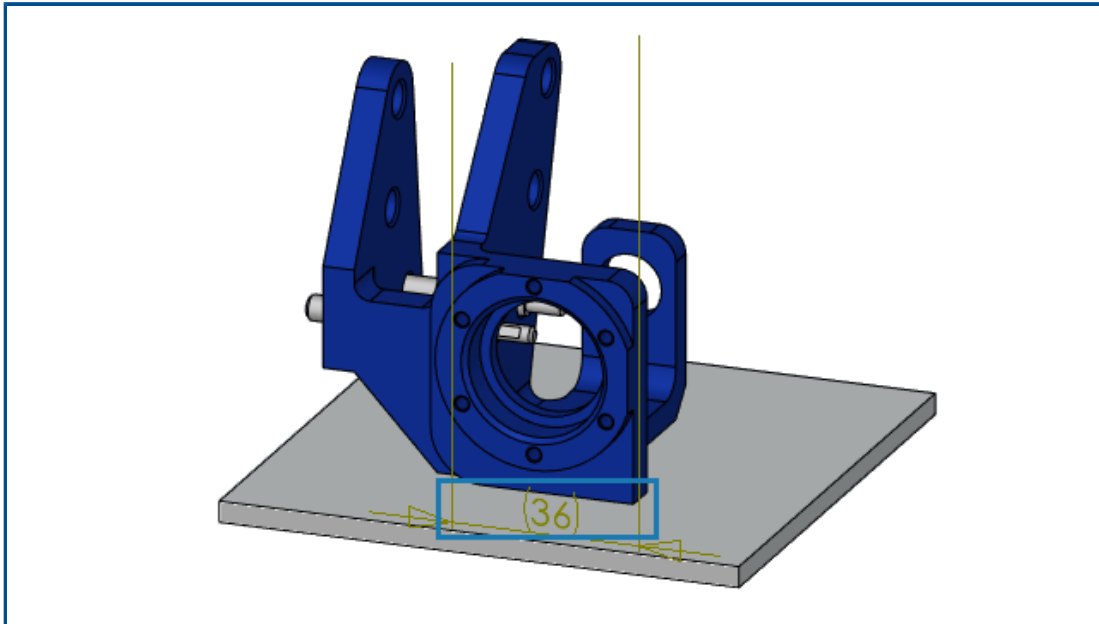


You can add a decimal separator in geometric tolerance symbols.

To add a decimal separator in geometric tolerance symbols:

1. Click **Tools** > **Options** > **Document Properties** > **Annotations** > **Geometric Tolerances**.
2. Under **Decimal Separator**, specify an option:
 - **Comma**. Inserts a comma.
 - **Period**. Inserts a period.

Controlling Visibility of Annotations through Solid Geometry

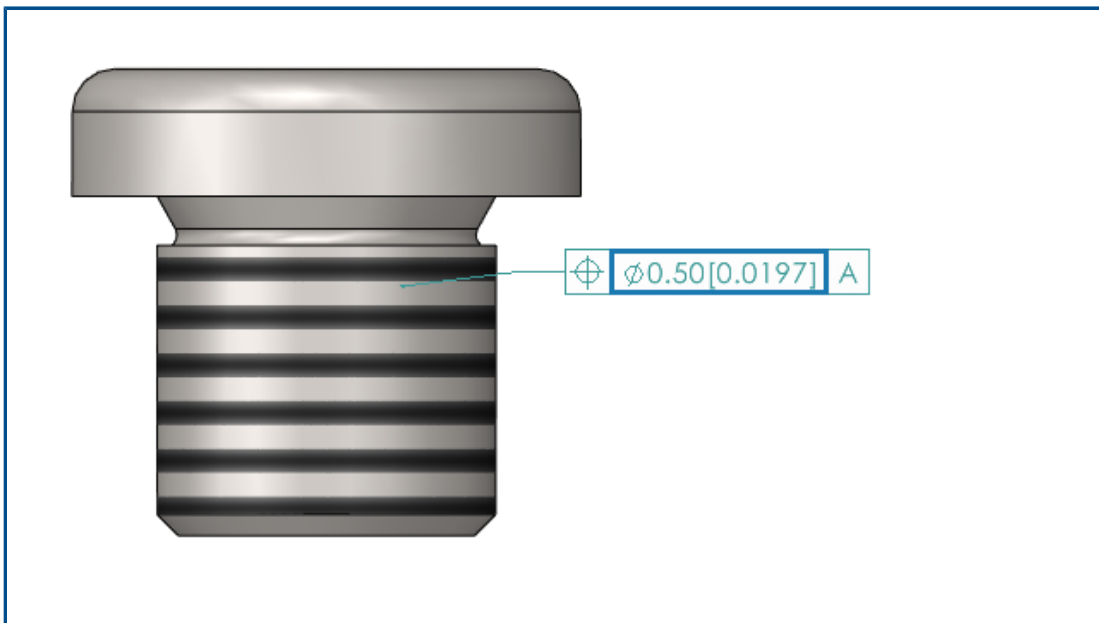


You can make annotations, such as dimensions, stay on top of the model. This lets you see dimensions and extension lines if you rotate the model.

To control visibility of annotations through solid geometry:


1. Click **Tools** > **Options** > **System Options** > **Display**.
2. Select **Display DimXpert dimensions on top of model**.

Displaying Dual Dimensions in Geometric Tolerance Symbols

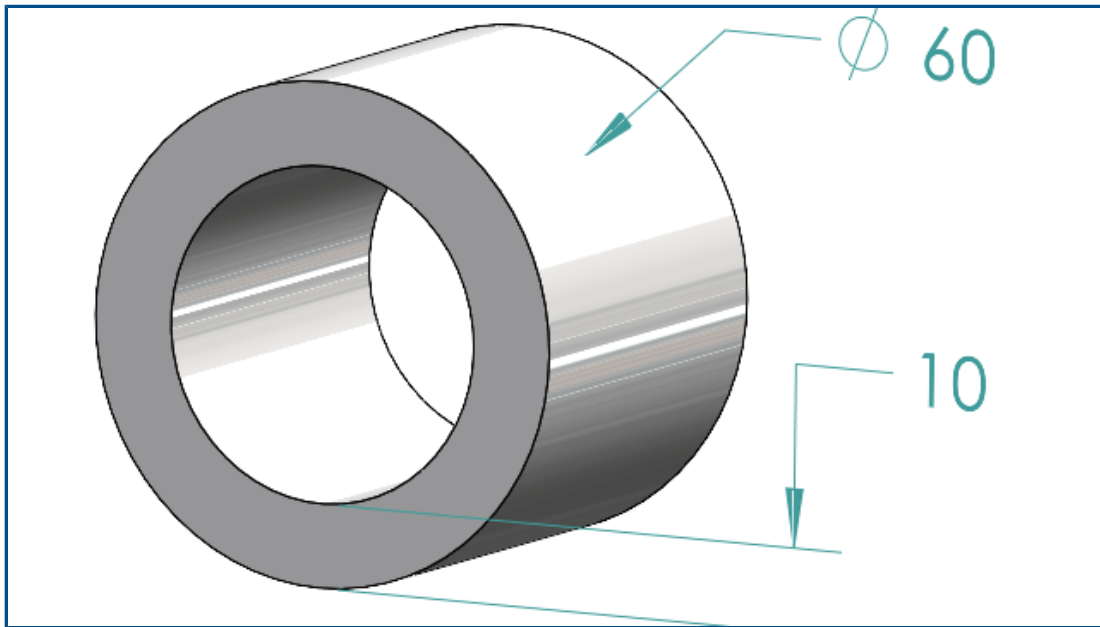


When you create geometric tolerance symbols, you can display dual dimensions, which show two sets of values, such as inches and millimeters, within a single dimension.

To display dual dimensions in geometric tolerance symbols:

1. In a part or drawing, click **Geometric Tolerance**  (MBD Dimension toolbar).
2. In the graphics area, click to place the symbol.
3. Select **Range** in the **Tolerance** dialog box and the **Geometric Tolerance** PropertyManager and select **Display Dual Dimensions**.

Creating Thickness Dimensions for Curved Surfaces



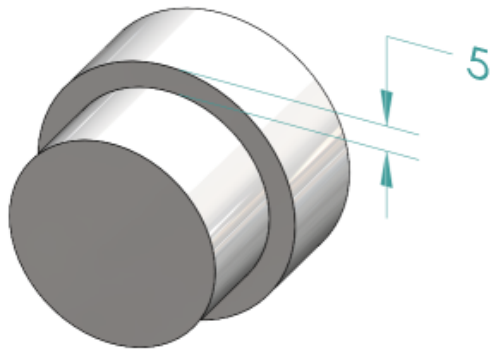
You can create thickness dimensions for curved surfaces.

This helps show the relationships between surfaces. You can apply thickness dimensions to:

- Cylinders
- Bosses
- Simple holes

You can create thickness dimensions between two concentric DimXpert features for:


- An inside and outside diameter, where the inside diameter is a cylinder or a simple hole, and the outside diameter is a cylinder or a boss.
- Two inside diameters of a cylinder or simple hole.
- Two outside diameters of a cylinder or boss. For example:



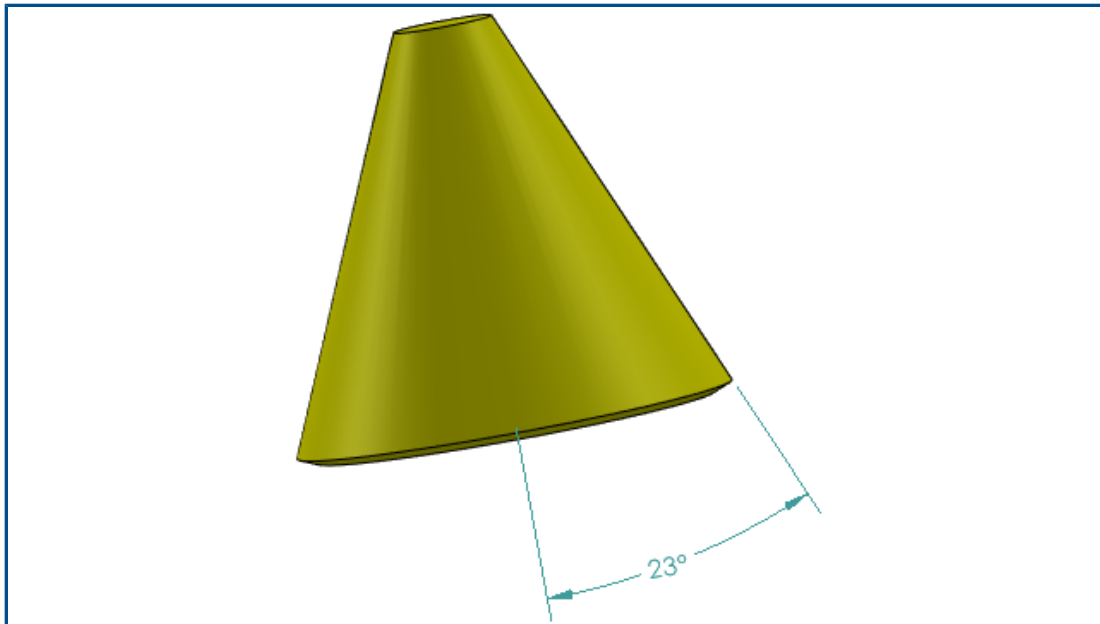
To create thickness dimensions for curved surfaces:

1. Click **Location Dimension**  (MBD Dimension toolbar).

Steps 2 and 3 require that you select two features. For thickness dimensions, the two features must be cylindrical, concentric, and have different diameters.

2. Select the face of the origin feature.
3. Select the face of the tolerance feature.
4. Click to place the dimension.
5. Specify options in the PropertyManager and click .

Displaying Half Angles of Conical Dimensions

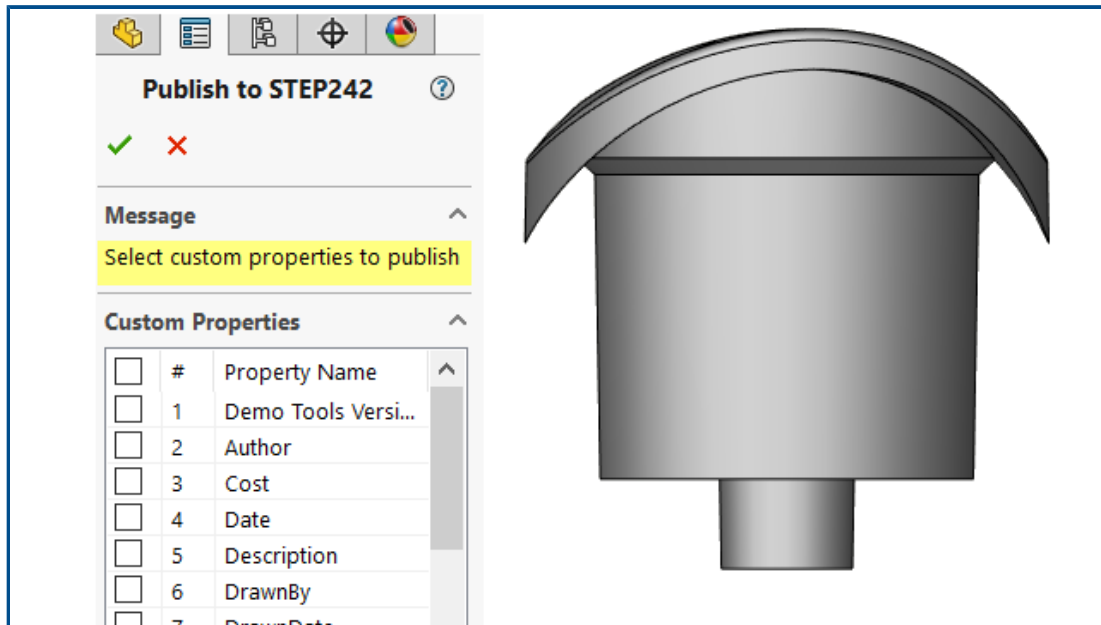


You can display a conical angle dimension as a half angle. This lets you convert a full angle of a cone to a half angle.

To display half angles of conical dimensions:



1. In the DimXpert Value PropertyManager, under **Primary Value**, select **Show as half angle**.

Exporting Custom Properties to STEP 242



You can export custom properties from a part or assembly to the STEP 242 format.

To export custom properties to STEP 242:

1. Click **Publish STEP 242 File**  (MBD toolbar).
2. In the Publish to STEP242 PropertyManager, specify custom properties to export and click .
3. In the Save As dialog box, enter a file name.
4. Click **Save**.

Viewing Annotations and Dimensions

You can view annotations and dimensions in a more organized way.

As of SOLIDWORKS 2024 and later, you do not need a SOLIDWORKS MBD license for this functionality.

You can use the following features:

- List annotations in a tree view. When you select an annotation in the FeatureManager design tree, it highlights the annotation in the graphics area, and you can hide or show annotations.

- Sort by annotation type. You can sort annotations by type, such as smart dimensions, weld symbols, and balloons for better organization.

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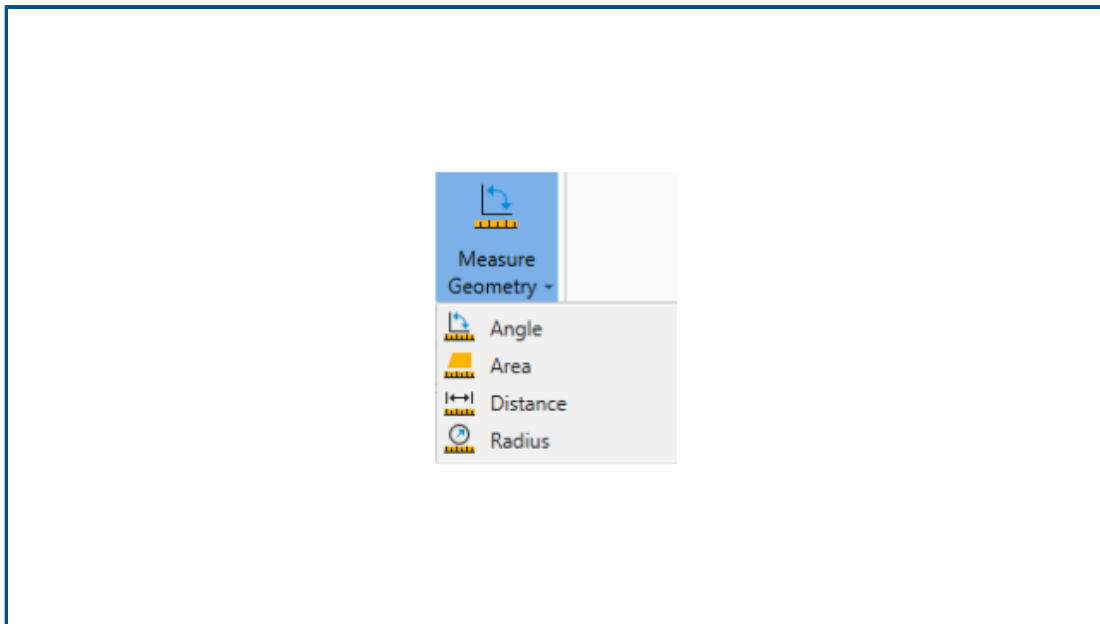
DraftSight

This chapter includes the following topics:

- **Measure Geometry Command**
- **Selecting Multiple Files and Inserting as Reference**
- **Export Sheet Command**
- **Tool Palettes**
- **Make Flat Snapshot Command**
- **View Navigator**
- **Layer Manager Palette**
- **Merge Layer Command**
- **Reshaping Hatches**

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.

Measure Geometry Command



You can use the `MEASUREGEOM` command to measure an area, angle, distance, and radius.

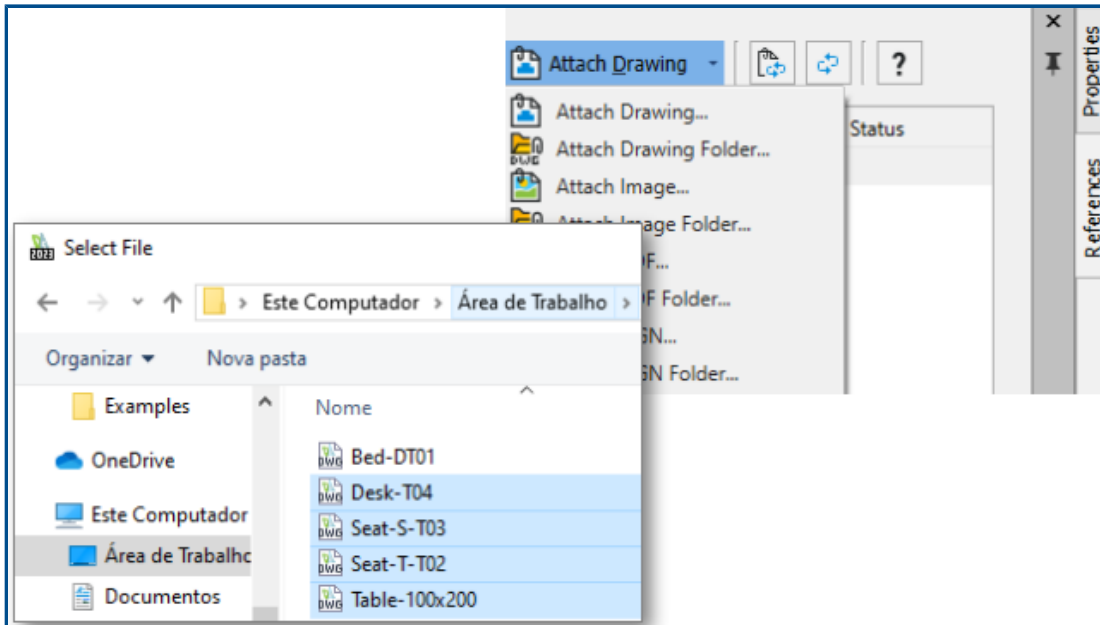
In previous releases, you had to run commands like `AREA`, `DIST`, and `GETANGLE`.

To access the Measure Geometry command:

Do one of the following:

- On the ribbon, click **Home** > **Tools** > **Measure Geometry**.
- Enter `MEASUREGEOM` in the command window.

Selecting Multiple Files and Inserting as Reference



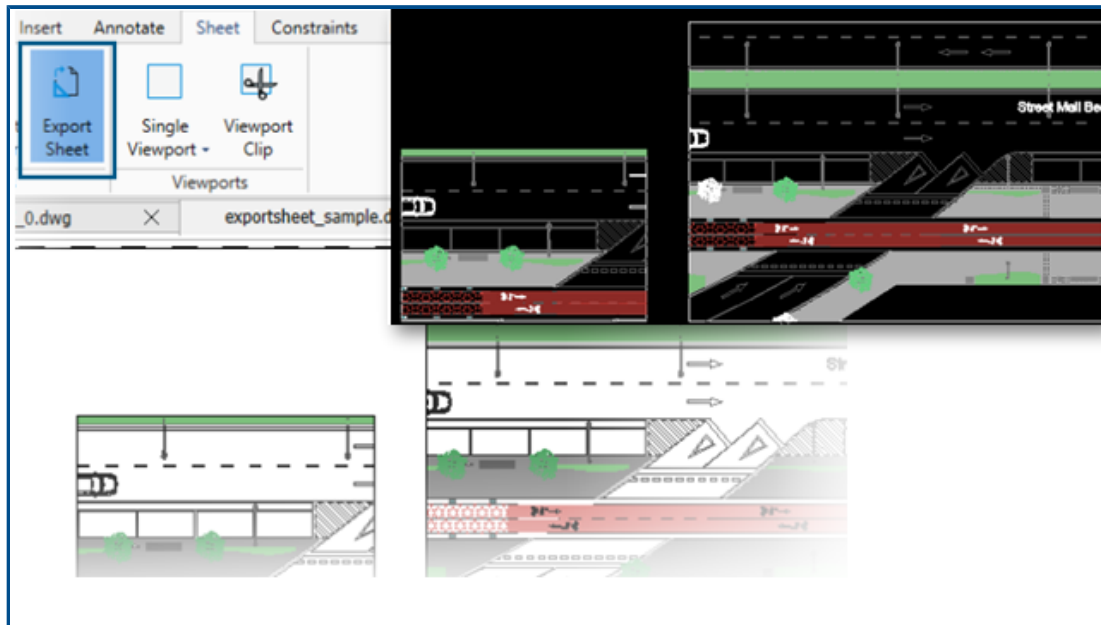
You can select multiple files and folders and insert them as external references to the DWG™ file. This reduces the number of clicks required to insert multiple files and the possibility of failing to insert a file.

To select multiple files and insert them as references:

Do one of the following:

- On the ribbon, click **Insert** > **Block** > **References Manager**.
- On the ribbon, click **Attach**.
- On the menu, click **Tools** > **References Manager**.
- Enter `REFERENCES` in the command window.

Export Sheet Command



You can export all visible entities from an active sheet viewport and entities from the sheets to the new drawing.

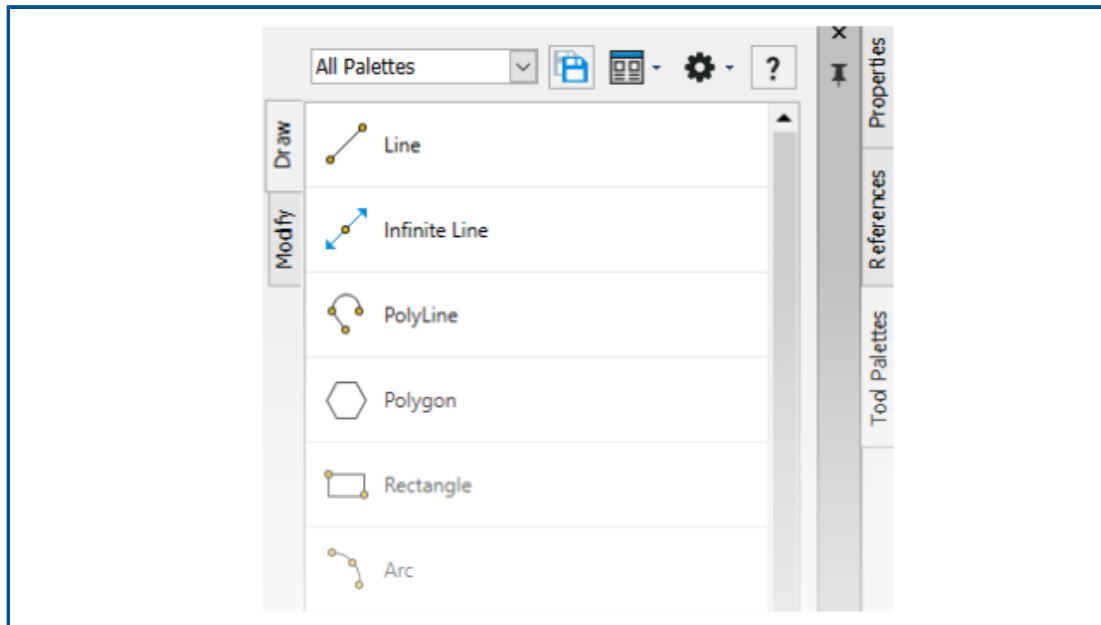
This lets you edit the representation created in the new drawing using commands like TRIM, COPY/PASTE, EXPLODE, STRETCH.

To access the Export Sheet command:

Do one of the following:

- On the ribbon, click **Sheet > Sheets > Export Sheet**.
- On the menu, click **File > Export > Export Sheet**.
- Enter EXPORTSHEET in the command window.

Tool Palettes



You can find frequently used tools and data in the Tool Palettes.

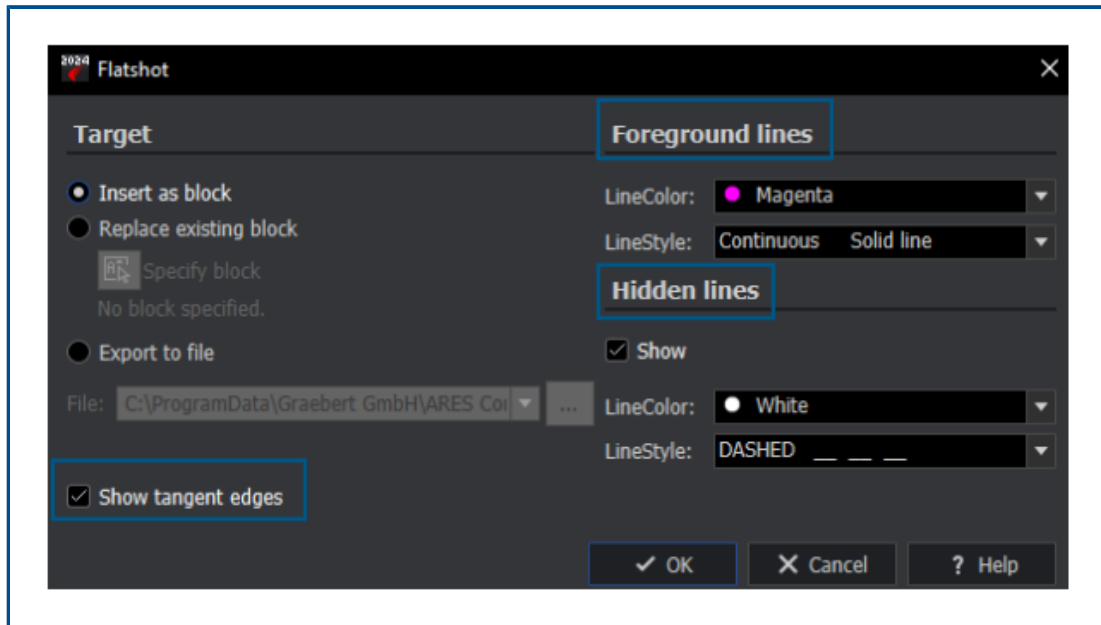
The palettes include all generic properties like docking and auto-hide. You can also create your own palette to store tools and data.

To access the Tool Palettes:

Do one of the following:

- On the ribbon, click **Insert > Palettes > Tool Palettes**.
- On the menu, click **Tools > Tool Palettes**.
- Enter `TOOLPALETTES` in the command window.

Make Flat Snapshot Command



You can use the enhanced features of the `MAKEFLATSNAPSHOT` command for formatting the foreground and hidden lines, and displaying tangent edges.

To access the Make Flat Snapshot command:

Do one of the following:

- On the ribbon, click **Home** > **Snapshot** > **Make Flat Snapshot**.
- On the menu, click **Solids** > **Solid Editing** > **Make Flat Snapshot**.
- Enter `MAKEFLATSNAPSHOT` in the command window.

The enhanced features include:

- **Foreground lines.** **LineColor** and **LineStyle** specify the line color and style of foreground lines.
- **Hidden lines.** **Show** displays the hidden lines. **LineColor** and **LineStyle** specify the line color and style of hidden lines.
- **Show tangent edges.** Displays tangent edges in the flat representation.

View Navigator



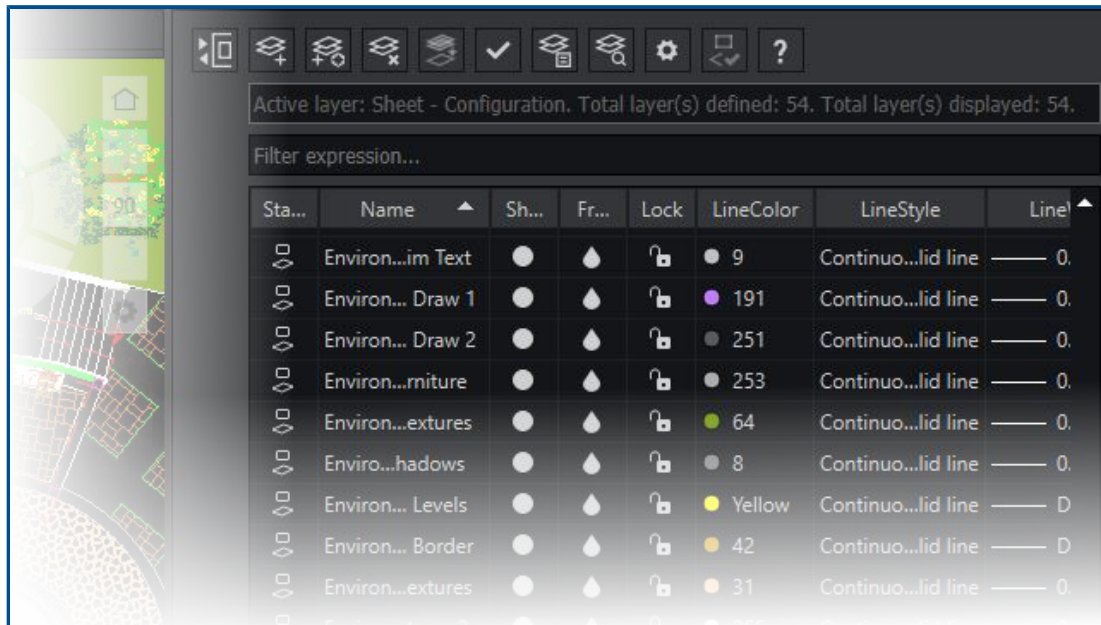
View Navigator lets you switch between standard and isometric views or parallel and perspective views of a model.

Its interface acts as a 3D orientation indicator that lets you see the current view direction.

To access the View Navigator command:

- On the ribbon, click **View > Views > View Navigator**.
- On the menu, click **View > View Navigator**.
- Enter `VIEWNAVIGATOR` in the command window.

Layer Manager Palette



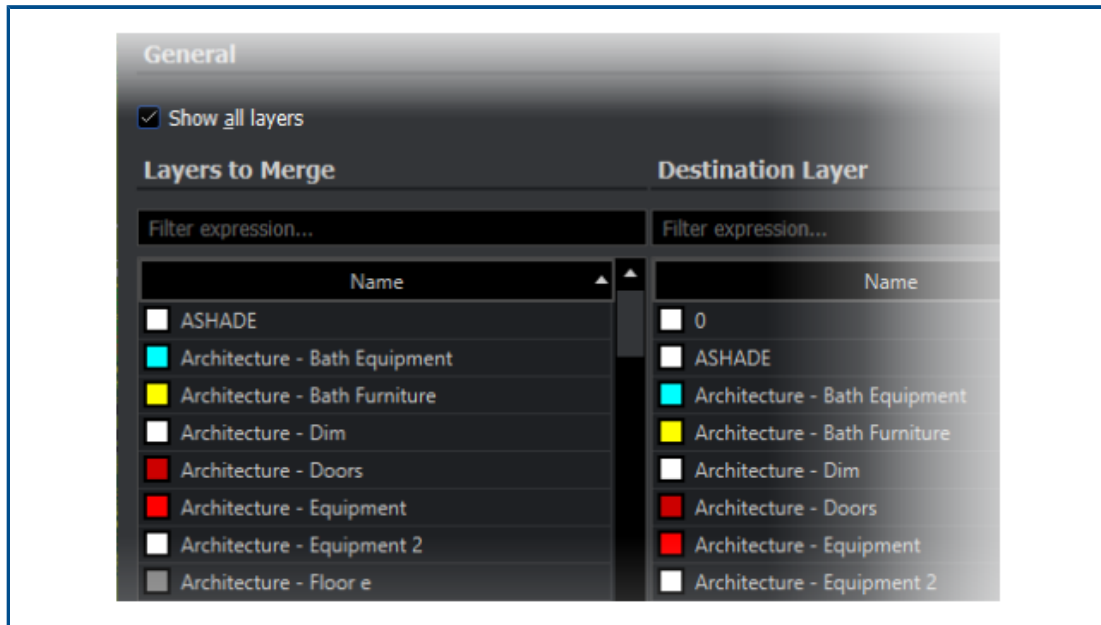
You can use the Layer Manager dialog box as a palette that you can float or dock on the side.

In the Layer Manager palette, you have quick access to layers, layer states, layer previews, or isolating layers.

To open the Layer Manager palette:

- On the ribbon, click **Home** > **Layer** > **Layers Manager**.
- On the menu, click **Format** > **Layer**.
- Enter `LAYER` in the command window.

Merge Layer Command



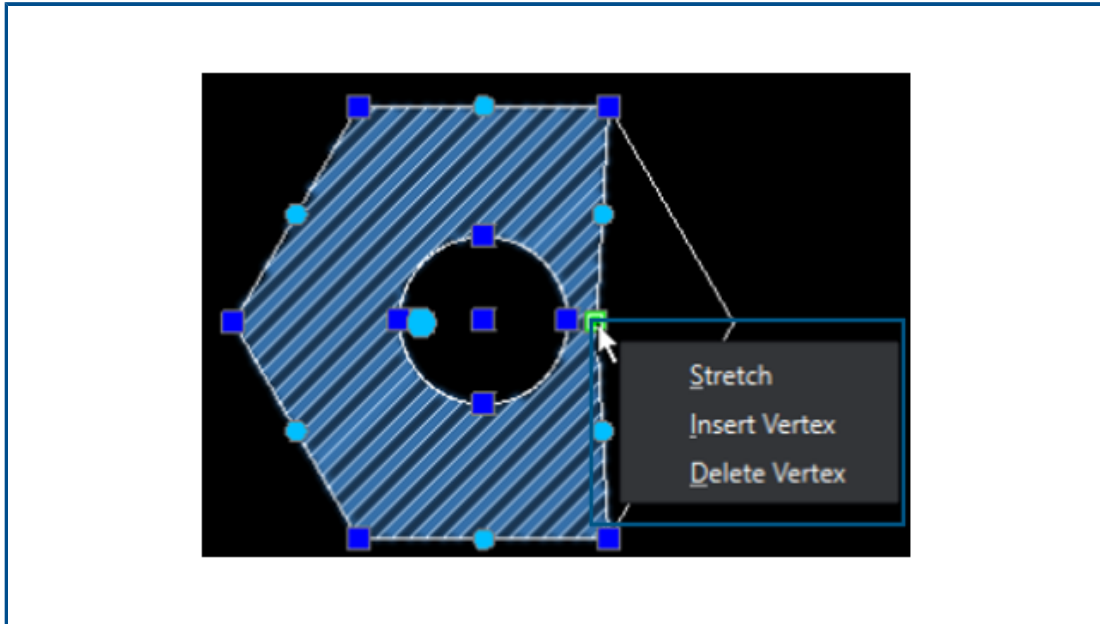
You can use the `MERGE LAYER` command to reorganize layers.

This command is available from the Layer Manager palette that helps you merge the content of selected layers into other layers.

To access the Merge Layer command:

- On the ribbon, click **Home** > **Layers** > **Merge Layers**.
- On the menu, click **Format** > **Layer Tools** > **Merge Layers**.
- Enter `MERGE LAYER` in the command window.

Reshaping Hatches



You can adjust the contour of hatches or gradient hatches.

When you select a hatch entity, the grips appear that help you adjust the shape. When you hover over a grip, the shortcut menu appears with editing options.

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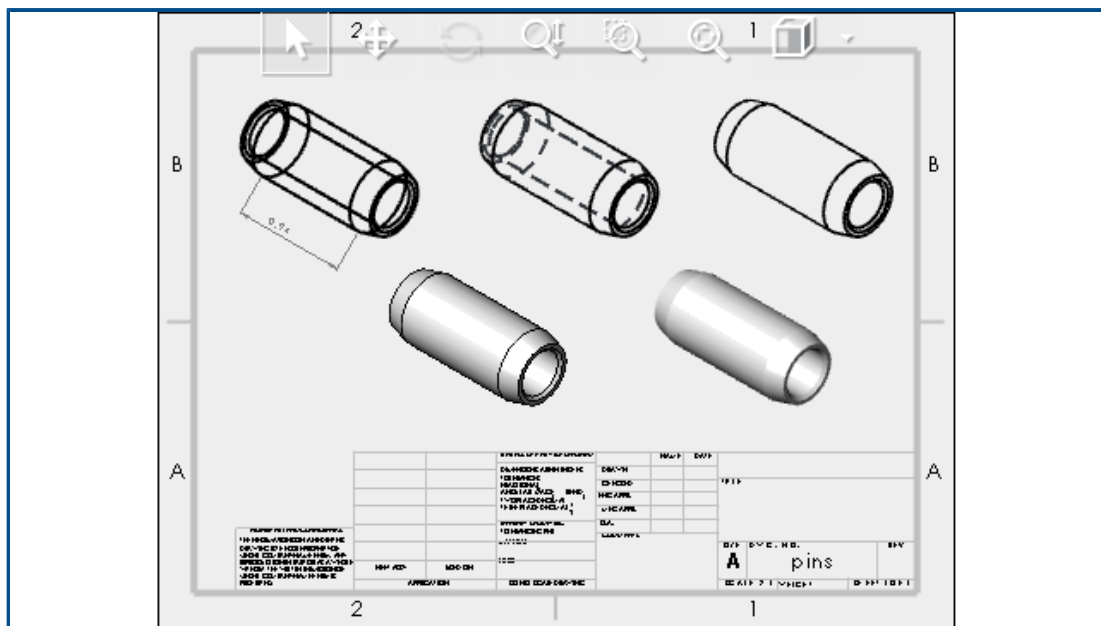
eDrawings

This chapter includes the following topics:

- **Display Styles in Drawings**
- **Supported File Types**
- **eDrawings Performance Improvements**

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

Display Styles in Drawings



If you saved a SOLIDWORKS drawing with specific display styles in drawing views, eDrawings supports each display style for any .EDRW file that you save in eDrawings 2024 and later.

In the Heads-up View toolbar, eDrawings shows all display states if the drawing views have shaded data: **Shaded with Edges**, **Shaded**, **Hidden Lines Removed**, **Hidden Lines Visible**, and **Wireframe**. The **Display Style** tool is only available for drawings with shaded data.

If you change the display style of a drawing view in eDrawings, only the selected view updates with the new display style. All other views remain the same. However, if you

change the display style when you have not selected a drawing view, all views change to the selected display style.

If you rotate a drawing view, the display style is unaffected.

Supported File Types

eDrawings has updated the supported versions for several file types.

Format	Version
ACIS (.sat, .sab)	Up to 2021
Autodesk® Inventor® (.ipt, .iam)	Up to 2023
CATIA® V5 (.CATPart, .CATProduct)	Up to V5_V62023
Creo® - Pro/Engineer® (.ASM, .NEU, .PRT, .XAS, .XPR)	Pro/Engineer 19.0 to Creo 9.0
JT (.jt)	Up to v10.6
NX™ (Unigraphics®) (.prt)	NX1847 Series to NX2212
Parasolid™ (.x_b, .x_t, .xmt, .xmt_txt)	Up to 35.1
Solid Edge® (.asm, .par, .pwd, .psm)	V19 - 20, ST - ST10, 2023

eDrawings Performance Improvements

eDrawings performance is improved with various tools, rendering, printing, and file closure times.

Performance improvements include:

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

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SOLIDWORKS Flow Simulation

This chapter includes the following topics:

- **Importing and Exporting Component Lists**
- **Mesh Generation**
- **Mesh Boolean Operations**

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

For installation of SOLIDWORKS Flow Simulation, see [Load SOLIDWORKS Flow Simulation Modules](#).

Importing and Exporting Component Lists

Components	Materials	Volume Sources	Two-Resistor Components
_Beaglebone	.. / Aluminum [Default]	1 W (Total)	1.4 W (Total)
RJ45_17PINS_2LED_Con-1	Epoxy Resin		
DC_PWR_JCK-1	Epoxy Resin		
usb connector-1	Aluminum [Default]		
Case-1	Aluminum [Default]		
HEADER_23x2-1	Epoxy Resin		
HEADER_23x2-2	Epoxy Resin		
CONN19_HDMI-1	Aluminum [Default]		
beaglebonev8-1	..	1 W (Total)	1.4 W (Total)
Board_beaglebonev8(0)	Insulator [Default]		
U2_U_48_RSL_TYPE_0029(1)	Insulator [Default]		0.8 W

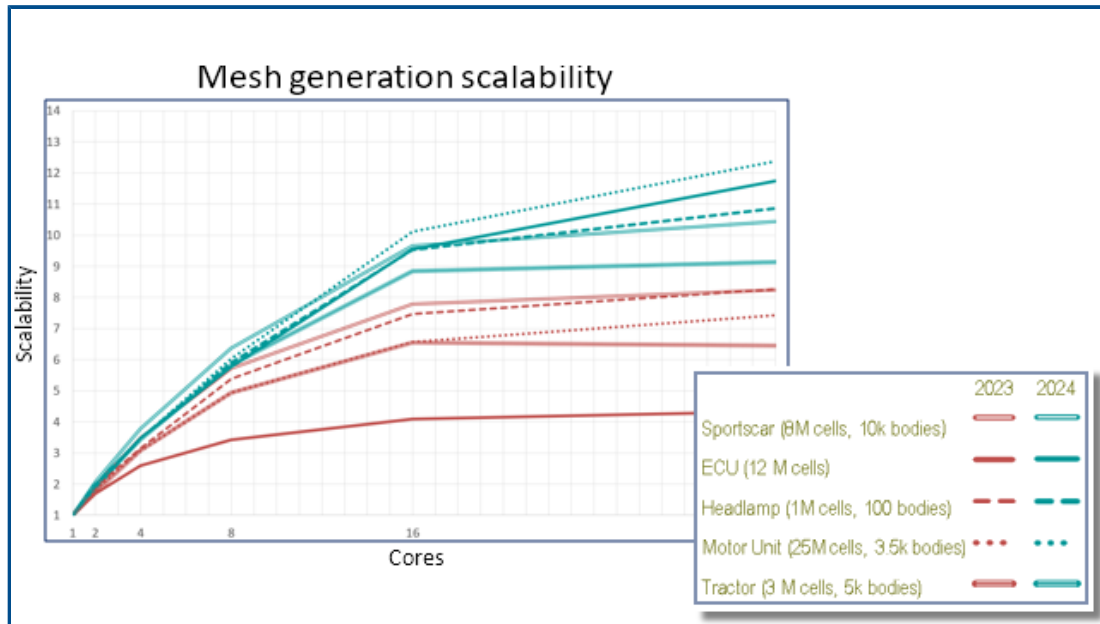
In the Component Explorer dialog box, you can export component lists to a Microsoft® Excel® spreadsheet, edit the properties, and import the component lists back.

By using a spreadsheet, you can manage component properties. You can edit:

- **Materials**
- **Volume Sources**
- **Two-Resistor Components** (library and power)

- **LEDs** (library and current)

Mesh Generation



With the Smart Cell Cartesian mesh generator, you can generate meshes faster and with smaller file sizes.

Speeds are 9-12 times faster on 32 cores for 10-20M cell models in Flow Simulation 2024 compared to 3-7 times faster in 2023. Meshing speed is about 2-3 times faster on 32 cores in 2024 because of scalability.

Mesh Boolean Operations

The Mesh Boolean Operation (MBO) handles complex and extremely bad geometry faster and easier. When SOLIDWORKS cannot conduct Boolean operations successfully because of bad geometry (such as bad topology with missing entities or self-intersecting faces), you can use MBO.

MBO meshes bodies separately and then conducts Boolean operations on the meshed bodies without using CAD Boolean operations.

This technology prepares and meshes even very bad models 5-15 times faster without prior user adjustments or automatic model healing. You can use MBO with the CAD Boolean diagnostic, combining the power of Mesh Boolean and the convenience of getting additional information, such as a diagnostic of the fluid domain.

If the CAD Boolean diagnostic fails to detect the fluid domain, you can still mesh the model with Mesh Boolean. In these cases, the Solver Monitor dialog box shows additional subdomain diagnostics. You can specify how to handle the geometry (CAD Boolean, Preprocessor Boolean (formerly called Improved Geometry Handling), or Mesh Boolean), and you can turn off the CAD Boolean diagnostics.

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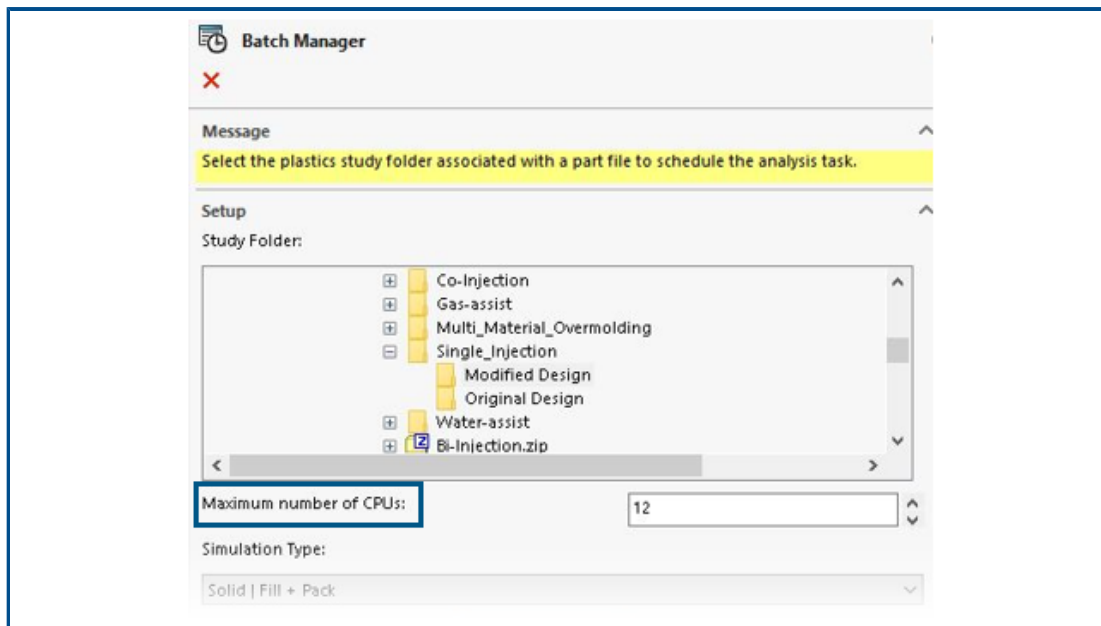
SOLIDWORKS Plastics

This chapter includes the following topics:

- **Batch Manager**
- **Compare Results**
- **Cool Solver**
- **Hot and Cold Runners**
- **Inactivity Timeout for SOLIDWORKS Simulation, SOLIDWORKS Flow Simulation, and SOLIDWORKS Plastics**
- **Injection Location Advisor**
- **Materials with Pressure-Dependent Viscosity**
- **Material Database**
- **Mesh Enhancements**

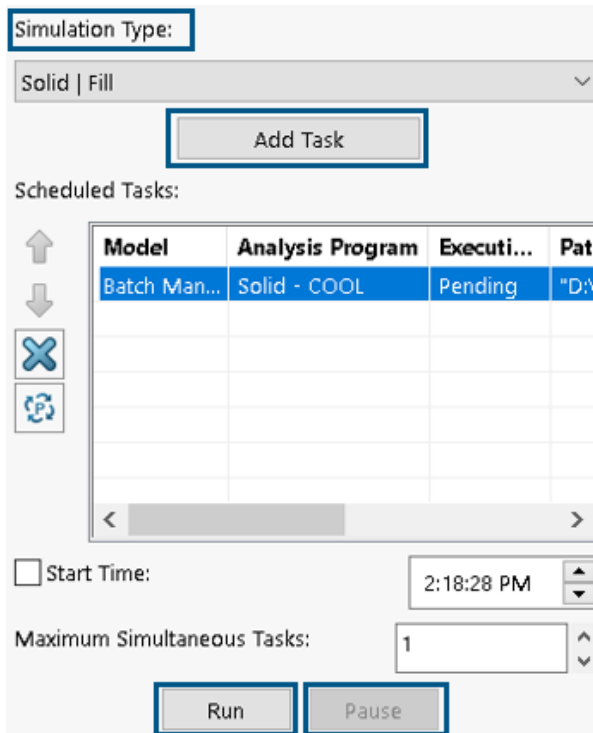
SOLIDWORKS® Plastics Standard, SOLIDWORKS Plastics Professional, and SOLIDWORKS Plastics Premium are separately purchased products that you can use with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium.

Batch Manager

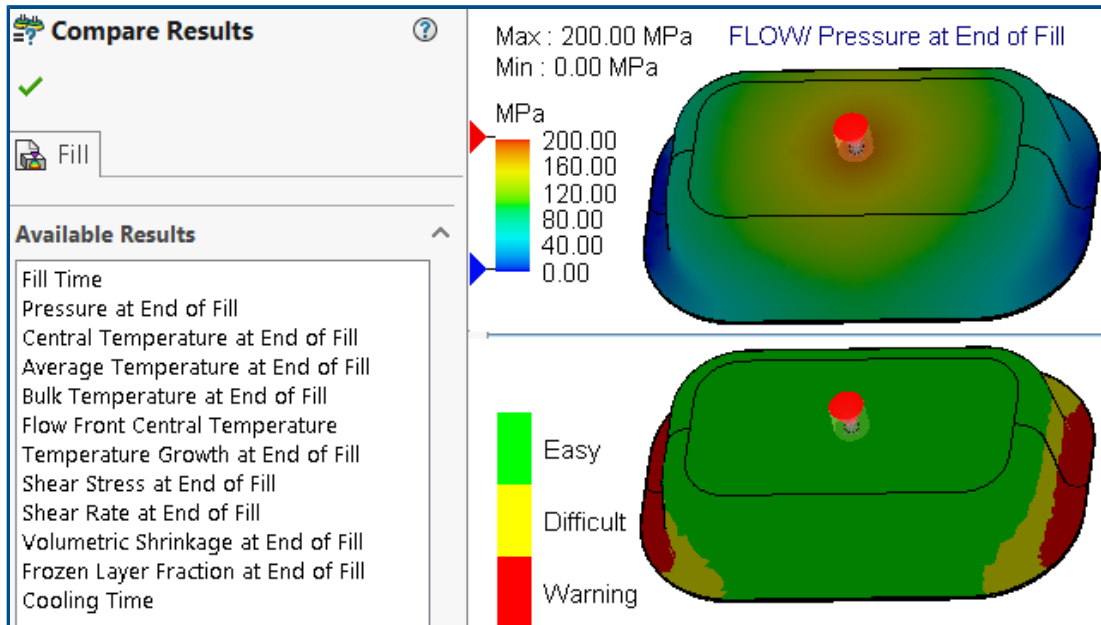


The Batch Manager PropertyManager is redesigned to improve usability.

- Rearrangement of the user interface elements in sections provides a streamlined workflow for the Batch Manager.
- Ability to specify the maximum number of CPUs for an analysis task.
- Improved visibility for the simulation type assigned to an analysis task and for controls to add, run, and pause an analysis task.



Compare Results



You can display four different results plots from one study using split view panes.

To display multiple result plots after running a study:

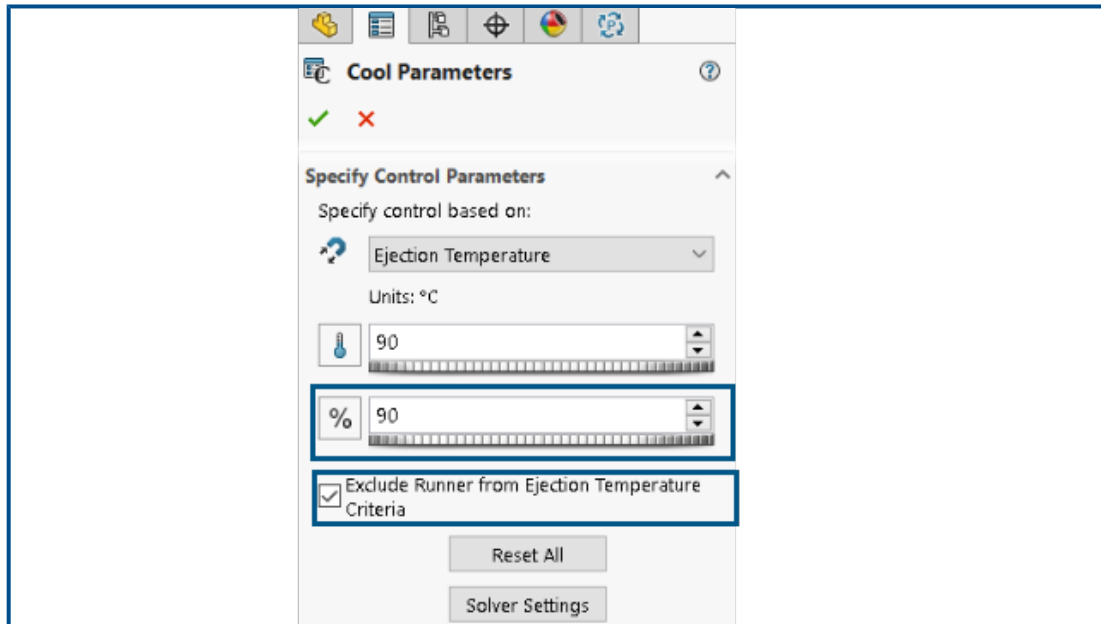
Do one of the following:

- Click **Compare Results** (Plastics CommandManager).
- In a study's PlasticsManager tree, right-click **Results** and click **Compare Results**.

In the Compare Results PropertyManager, you have these options:

Option	Description
Synchronize Views	Applies the same view orientation to all view panes.
Save Image	Saves the split view of the multiple result plots to a .png image format.

Cool Solver



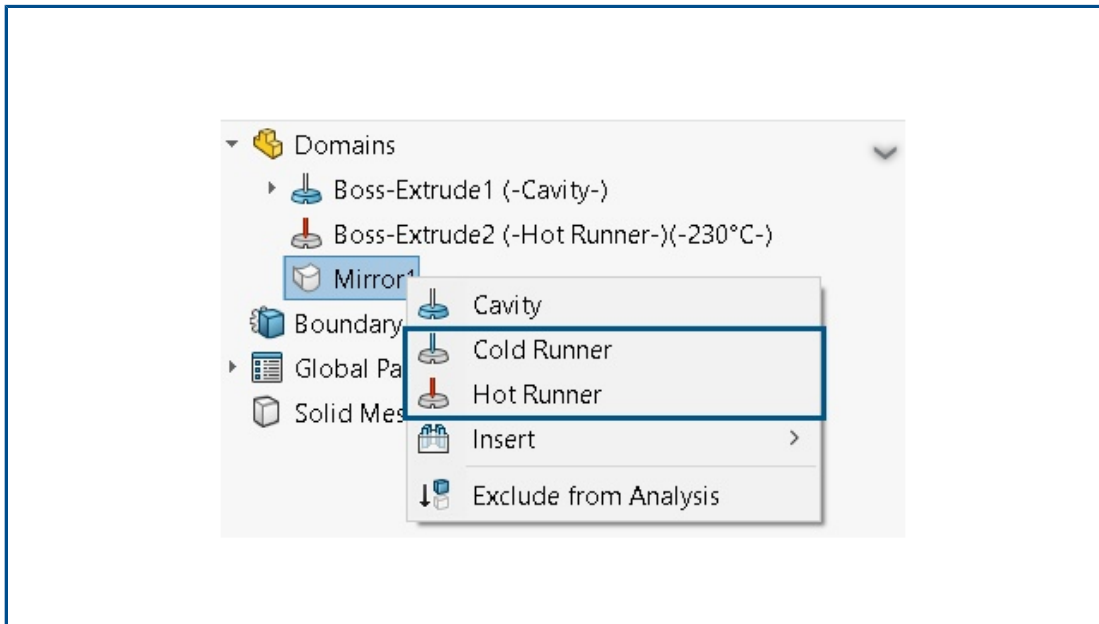
Solver options for ejection criteria enhance the performance of plastic injection simulations for thermoplastic materials.

You can either specify the cooling time or let the cool solver estimate a cooling time based on the following temperature ejection criteria for thermoplastic materials.

Option	Description
Volume % frozen at ejection	Specifies the percentage of the mold's volume that needs to cool down below the ejection temperature. The default is 90%.
Exclude Runner from Ejection Temperature Criteria	Excludes the cooling state of the sprue and runner segments from the ejection criteria. It is common to reduce the overall manufacturing time by ejecting the part

Option	Description
	before the sprue and runner segments have cooled completely.

Hot and Cold Runners



You can more easily assign hot or cold runner domains to components of a plastic injection simulation.

To assign a runner domain type to a body listed under the **Domains** node, right-click the body and click **Hot Runner** or **Cold Runner**.

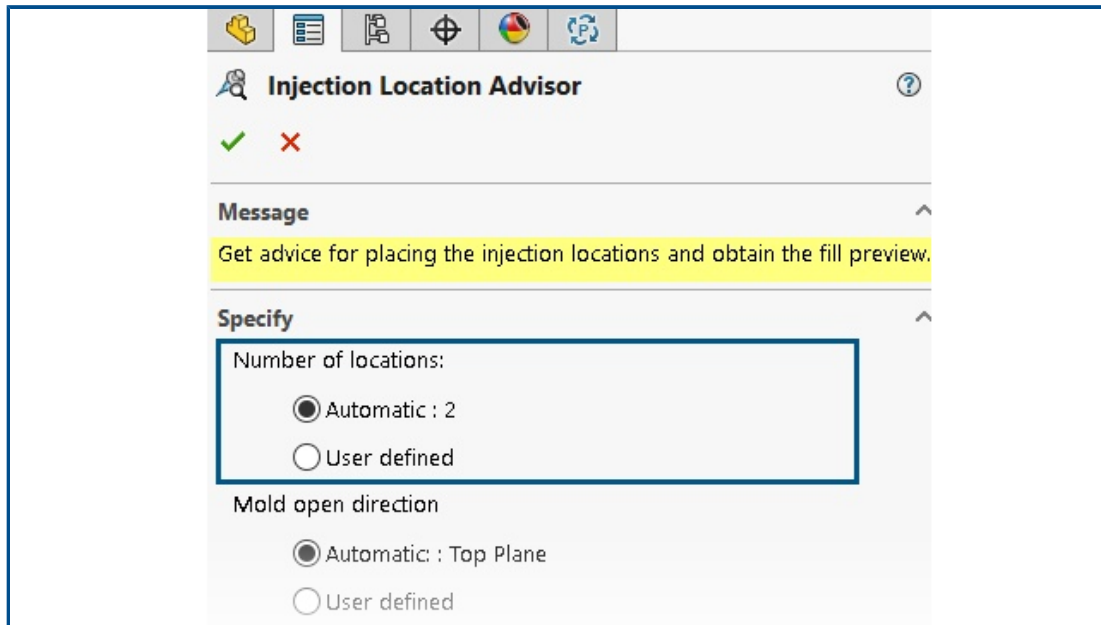
Inactivity Timeout for SOLIDWORKS Simulation, SOLIDWORKS Flow Simulation, and SOLIDWORKS Plastics

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

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

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

Injection Location Advisor




The Injection Location Advisor can iteratively determine an optimal number of injection locations (maximum of 10) to fill a cavity.

The default for **Number of locations** is **Automatic**, which activates the iterative approach for finding an optimal number of injection locations. To specify a custom number of injection locations, select **User defined**.

Materials with Pressure-Dependent Viscosity

/ Viscosity / PVT / Specific Heat / Thermal Conductivity / Shear Relaxation Modulus / Curing Model / Pol	
LUMID HI2252BF	
Polymer Family	PA
Manufacturer	LG Chem
Recommended Melt Temperature	285 °C
Maximum Melt Temperature	300 °C
Minimum Melt Temperature	270 °C
Recommended Mold Temperature	70 °C
Maximum Mold Temperature	80 °C
Minimum Mold Temperature	60 °C
Ejection Temperature	190 °C
Thermoset Conversions	Not Available
Transition Temperature	208 °C
ⓘ Viscosity : 7-Parameters Modified Cross mod	8.10013e+16 373.15 1e-07 41.484 6
⊕ PVT : Modified Iait Equation	0.000831 6.012e-07 1.51761e+08 0.0
Density	1365.5 Kg/m3
⊕ Specific Heat : Variable	32 1261 100 2053 130 2402 140
⊕ Thermal Conductivity : Variable	38.4 0.275 48.9 0.274 69.7 0.275

Fill and Pack simulations support materials with pressure-dependent viscosity.

Materials that have pressure-dependent viscosity are listed in the Plastics Materials Database with an information icon .

Accounting for pressure-dependent viscosity is important for parts that contain long flow lengths or very thin walls, or for cases where you need high injection pressures.

For more information, see *Material Properties (Polymer, Mold, and Coolant Domains)*.

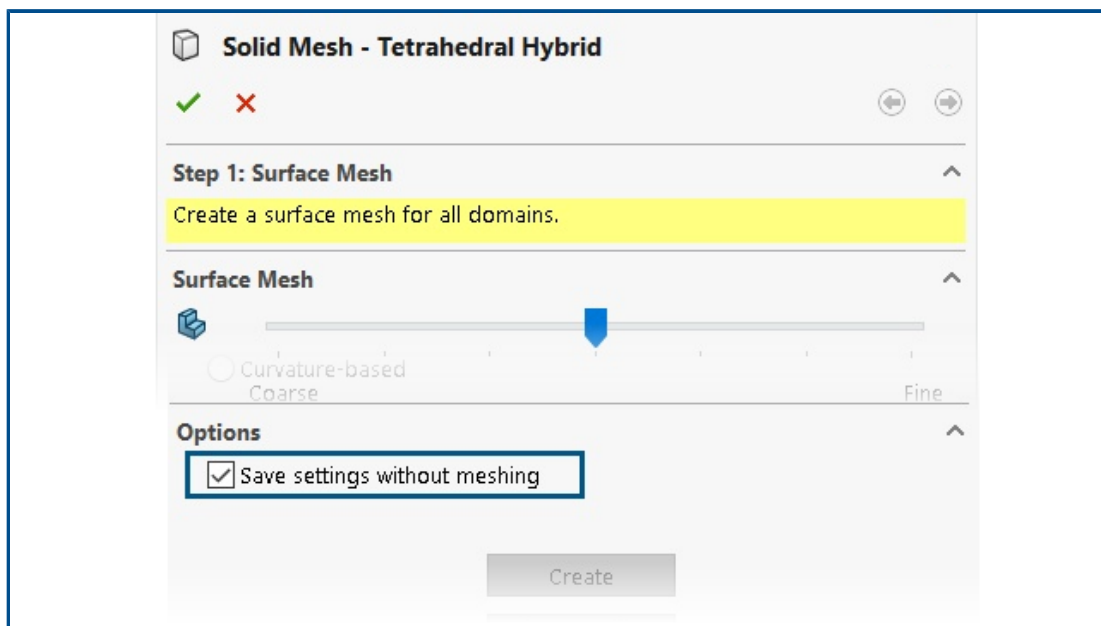
Material Database

The Plastics material database includes the latest data from the material manufacturers.

Materials	Description
New Materials	Added 417 new material grades from the following material suppliers: <ul style="list-style-type: none"> • CHIMEI: 42 • DuPont: 2 • EMS-GRIVORY: 4 • KRAIBURG TPE: 4 • LG Chem: 85 • MOCOM: 128 • ORLEN Unipetrol RPA: 20 • RadiciGroup High Performance Polymer: 2 • SABIC Specialties: 126 • Solvay Specialty Polymers: 1 • Trinseo: 3
Modified Materials	Updated 40 material grades with the latest material property values provided by the following material suppliers: <ul style="list-style-type: none"> • Borealis: 1 • CHIMEI: 2 • EMS-GRIVORY: 10 • ORLEN Unipetrol RPA: 20 • SABIC Specialties: 7

Materials	Description
Removed Materials	<p>Removed 292 obsolete material grades from the following material suppliers:</p> <ul style="list-style-type: none"> • 3M: 1 • ALBIS: 4 • Borealis: 1 • DuPont: 2 • DuPont Engineering Polymers: 2 • KRAIBURG TPE: 1 • LANXESS GmbH: 3 • LG Chemical: 56 • SABIC Specialties: 211 • Solvay Specialty Polymers: 11


Mesh Enhancements



You can save the mesh settings of a study without creating a mesh. You can also preview a surface mesh before creating a solid mesh.

The meshing options are available from the Solid Mesh - Tetrahedral, Solid Mesh - Hexahedral, and Shell Mesh PropertyManagers.

Option	Description
Save settings without meshing	You can save the mesh settings of a model (mesh size, refinement method, and advanced mesh control) without creating the mesh. When you run a study, the mesh settings

Option	Description
	are applied automatically to generate the mesh. In a study's PlasticsManager tree, the icon  next to Solid Mesh or Shell Mesh indicates that you saved the mesh settings for the model.
Show preview	You can preview a surface mesh before creating a solid mesh to check the mesh validity for a model.

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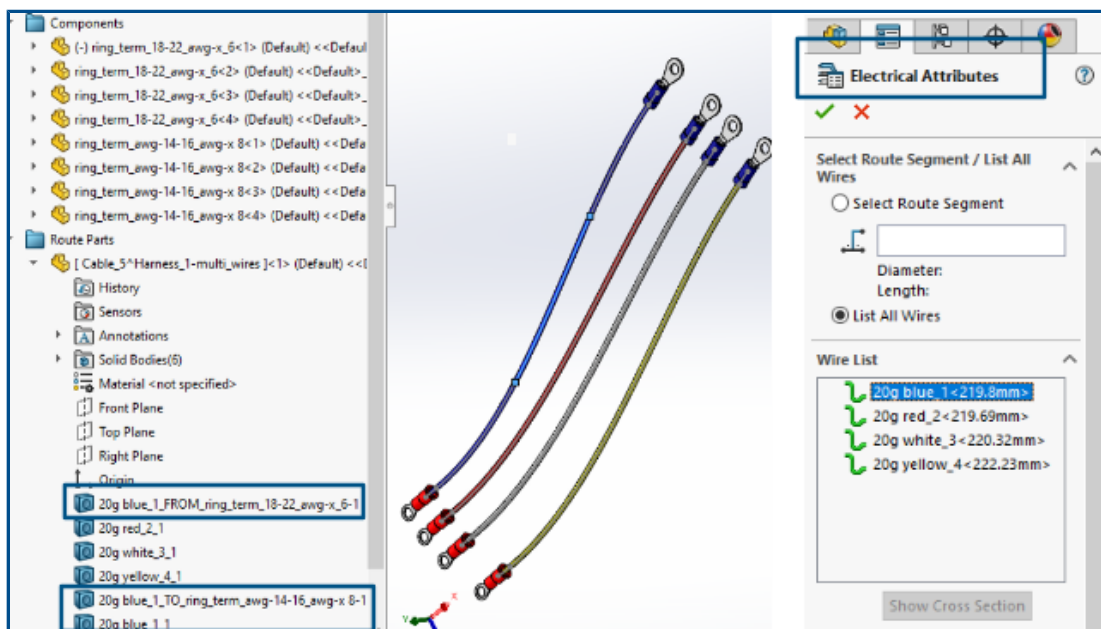
Routing

This chapter includes the following topics:

- **Naming Wires and Cables in the FeatureManager Design Tree**
- **Discrete Wires with Auto Route**

Routing is available in SOLIDWORKS® Premium.

Naming Wires and Cables in the FeatureManager Design Tree



You can view the marks or names of 3D wires, cables, and their cores under **Route Parts** in the FeatureManager® design tree for a routing assembly. The Electrical Attributes PropertyManager automatically preassigns the marks or names.

This helps you correlate the 3D routes in the FeatureManager design tree with the marks or names of the wires, cables, and their cores displayed on the schematic drawing.

The naming convention uses the following to uniquely identify different routes:

- Wire, cable, and cable core marks from the Electrical Attributes PropertyManager.
- Sequential numbers as suffixes (n). Where n is proportional to the number of splits (with split route) and 1 (without split route).
- The directions (FROM/TO) that they connect to the components.

For example, the above image shows the naming for a routing assembly with four wires as follows:

- The three wires red, white, and yellow do not have **Split Route** applied and the naming convention is:

Wire mark_1

For example, 20g_red_2_1

- The blue wire has a **Split Route** applied at two points with three split bodies created and the naming convention is:

- For the two extreme ends connected to the components:

Wire Mark_FROM/TO_Component Mark

For example,

20g blue_1_FROM_Component1

20g blue_1_TO_Component2

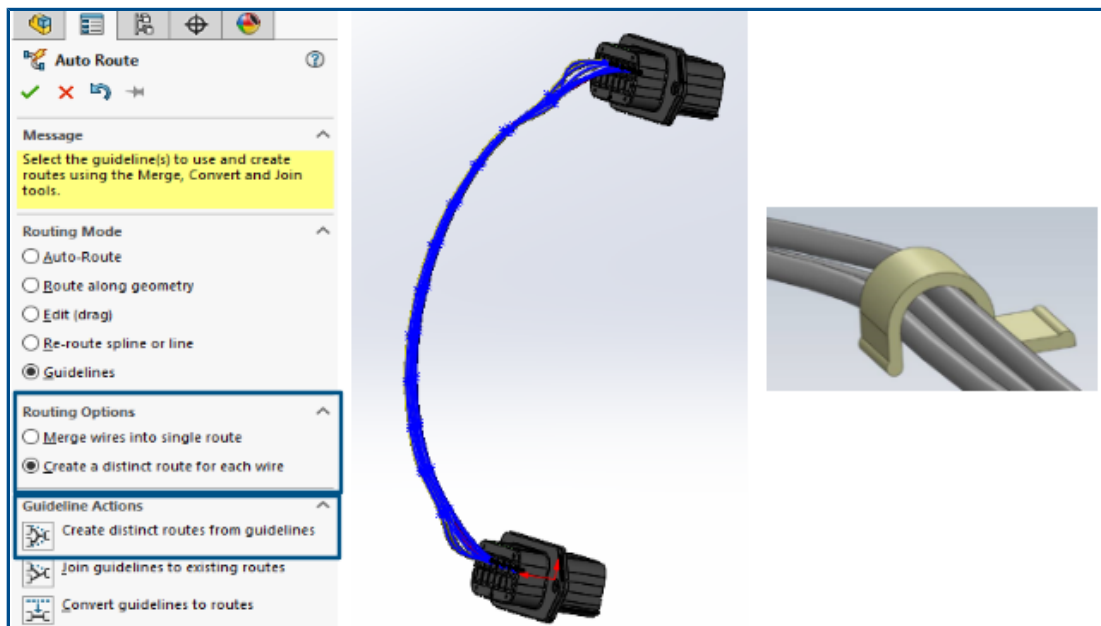
- In-between cable bodies not connected to the components:

Wire Mark_n

For example,

20g blue_1_1

Discrete Wires with Auto Route



You can visualize each wire in a bundle distinctly in 3D and flatten them.

The Auto Route PropertyManager, **Routing Options** include:

- **Merge wires into single route.** Routes the selected wires along a single route.

- **Create a distinct route for each wire.** Routes the selected wires as distinct routes.

You can edit discrete wires by:

- Adding a route to the bundle with **Add Route to Discrete Bundle.**
- Removing a route from the bundle with **Remove Route from Discrete Bundle.**
- Moving the bundle by dragging a spline point on the discrete wire.
- Merging two bundles with **Merge Discrete Bundle.**
- Splitting a single route segment from the bundle.
- Creating a single junction point for multiple discrete bundles coming out from the connector or separate junction point for each discrete bundle.
- Routing the bundle through a clip by selecting one of its splines.
- Handling splices.

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SOLIDWORKS Toolbox

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

Additional Toolbox Hardware



More hardware is available in the ANSI Inch and Metric Toolbox libraries.

Standard	Additional Folders	Additional Hardware
ANSI Inch	<ul style="list-style-type: none"> • The Washers folder includes: <ul style="list-style-type: none"> • Circular Washers • Square Beveled Washers • The Nuts folder includes subfolders for: <ul style="list-style-type: none"> • Hex Nuts - Prevailing Torque Nuts • Wing Nuts • The Pins folder includes subfolders for: <ul style="list-style-type: none"> • Clevis Pins • Cotter Pins • Grooved Pins • Spring Pins • Straight Pins • Tapered Pins 	<ul style="list-style-type: none"> • The Bolts and Screws > Self Tapping Screws folder includes a large hex head tapping screw. • The Bolts and Screws > Machine Screws folder includes a large hex screw.

ANSI Metric **Pins**. Includes coiled spring pins.

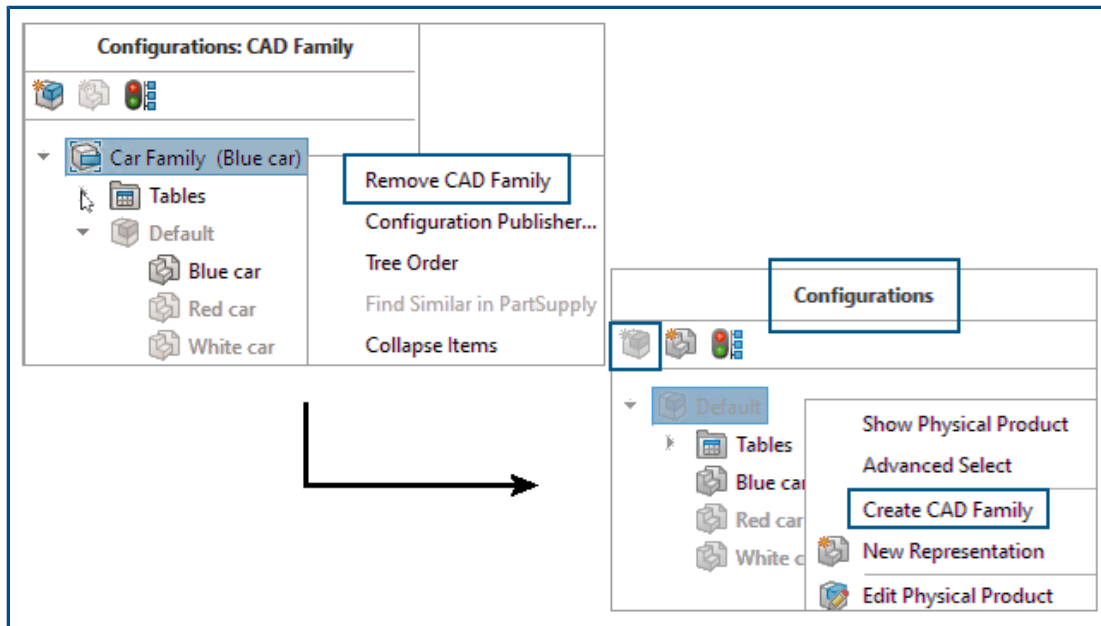
In the ANSI Inch standard, the `hex head tapping screw_ai.SLDPRT` in **Bolts and Screws > Self Tapping Screws > Hex Head Tapping Screw** has been updated. If you copy the updated file, you will lose any customization to the existing file.

Design with SOLIDWORKS

This chapter includes the following topics:


- **Creating a Single Physical Product**
- **Defining Rules for Updating Models to the 3DEXPERIENCE Platform**

Creating a Single Physical Product




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

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

- The part or assembly becomes a physical product.
- If the physical product is the active configuration, SOLIDWORKS uses the physical product as the single physical product. If the representation is the active configuration, SOLIDWORKS uses the parent physical product of the representation as the single physical product.
- Other configurations change to representations of the single physical product.
- **Inserts new physical product**  is disabled.

- The ConfigurationManager title changes from Configurations: <CAD Family> to Configurations.

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

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

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

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

To create a single physical product:

1. Open a model that has multiple physical products.
2. Right-click the CAD Family and click **Remove CAD Family**.

Defining Rules for Updating Models to the 3DEXPERIENCE Platform

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

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

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

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

New configurations are not created when you update a model.

3DEXPERIENCE Integration Rules Editor




You can create rules to specify which parts and assemblies to map to physical products and representations when you update a model to the **3DEXPERIENCE** platform.

You can save the integration rules to a designated folder.

To open the 3DEXPERIENCE Integration Rules Editor:

1. **Tools > Options > System Options > 3DEXPERIENCE Integration.**
2. Optional: Specify a folder for **3DEXPERIENCE Integration Rules Folder.**
3. Click **3DEXPERIENCE Integration Rules Editor.**




Sub-Typing Rules

ID	Incremental number for each sub-type.
Action	Options for working with sub-type rules:
 Add	Adds a new row and opens the Sub-Type Rule Definition dialog box.
 Edit	Opens the Sub-Type Rule Definition dialog box where you can edit the rule.
 Delete	Deletes the row.
Sub-Type Name	Displays the sub-type name.
Sub-Type Rule Description	Displays the rule description.

Configuration Mapping Rules

These options display when you select a sub-type rule.

Single physical product with representations	<p>Specifies the configuration to use as the single physical products.</p> <ul style="list-style-type: none"> • configuration named <div style="border: 1px solid gray; padding: 5px; margin: 5px 0;"> <p>You must enter a configuration name for this option. And you must select either active configuration or first configuration as a secondary configuration to use.</p> </div> <ul style="list-style-type: none"> • active configuration • first configuration
Multiple physical product	Displays additional fields for determining which configurations are mapped to physical products for the selected sub-type.
ID	Incremental number for each sub-type.

Action	Options for working with configuration mapping rules:
	 Add Adds a new row and opens the Configuration Mapping Rule Definition dialog box.
	 Edit Opens the Configuration Mapping Rule Definition dialog box where you can edit the rule.
	 Delete Deletes the row.

Configuration Mapping Rule Description	Description of the configuration mapping rule.
---	--

3DEXPERIENCE Object Type	Object type that configurations are mapped to.
---------------------------------	--

Configurations not included in the mapping rules.	Specifies the action to take for configurations that are not covered by the mapping rules. Options for mapping:
---	--

- **configuration named**

You must enter a configuration name for this option. And you must select either **active configuration** or **first configuration** as a secondary configuration to use.


- **active configuration**
- **first configuration**

Sub-Type Rule Definition Dialog Box

You can create rules to group parts and assemblies before mapping the groups to physical products and representations.

After creating these rules, you use the Configuration Mapping Rule Definition dialog box to specify if the parts and assemblies are physical products or representations when you update the model.

To open the Sub-Type Rule Definition dialog box:

1. **Tools > Options > System Options > 3DEXPERIENCE Integration.**
2. Click **3DEXPERIENCE Integration Rules Editor.**
3. In **3DEXPERIENCE Integration Rules Editor**, click **Add** .

Sub-Type rule description	Describes the sub-type rule.
----------------------------------	------------------------------

If the following condition is met

Specifies conditions for the rule.
Click **Add** to create a condition.

When you have multiple conditions, you can select **All** or **Any**.

Select an input:

Filename	Specifies an operator and a file name.
Custom Property	Specifies a property name, an Evaluated Value , an operator, and a value.
Is Sheet Metal Part	Options are Yes and No . Available for parts.
Is Weldment Part	Options are Yes and No . Available for parts.

Operators for **Filename** and **Custom Property**:

- **contains**
- **does not contain**
- **is**
- **is not**
- **starts with**
- **ends with**

Use **Add** and **Delete**  to create and remove conditions.


Assign the sub-type

Specifies the sub-type name.

Configuration-Mapping Rule Definition Dialog Box

You can define rules for mapping parts and assemblies to a physical product or a representation.

To open the Configuration Mapping Rule Definition dialog box:


1. **Tools > Options > System Options > 3DEXPERIENCE Integration.**
2. Click **3DEXPERIENCE Integration Rules Editor**.
3. In **3DEXPERIENCE Integration Rules Editor**, if there are no rules defined, click **Add**  to add a sub-type rule. Close the **Sub-Type Rule Description** after creating the rule.
4. Select a rule.
5. In **3DEXPERIENCE Integration Rules Editor**, under **Configuration Mapping Rules**, click **Multiple physical product**.

Configuration Mapping Rule Definition

Configuration Mapping Rule Description Describes the configuration mapping rule.

If the following condition is met

Specifies conditions for the rule.

Click **Add**  to add a condition.

When you have multiple conditions, you can select **All** or **Any**.

Select an input:

Configuration name Specifies an operator and a configuration name.

Configuration Property Specifies a property name, an **Evaluated Value**, an operator, and a value.

Configuration BOM option Specifies a BOM part numbering option:

- **Is Document Name**
- **Is Configuration Name**
- **Is User Specified**

Is active configuration Options are **Yes** and **No**.

Is first configuration Options are **Yes** and **No**.

The second list defines options for **Configuration name**, **Configuration Property**, and **Configuration BOM options**.

Operators for **Configuration name** and **Configuration Property**:

- **contains**
- **does not contain**
- **is**
- **is not**
- **starts with**
- **ends with**

Use **Add** and **Delete**  to add and remove rows.

Map the configuration to the 3DEXPERIENCE object type

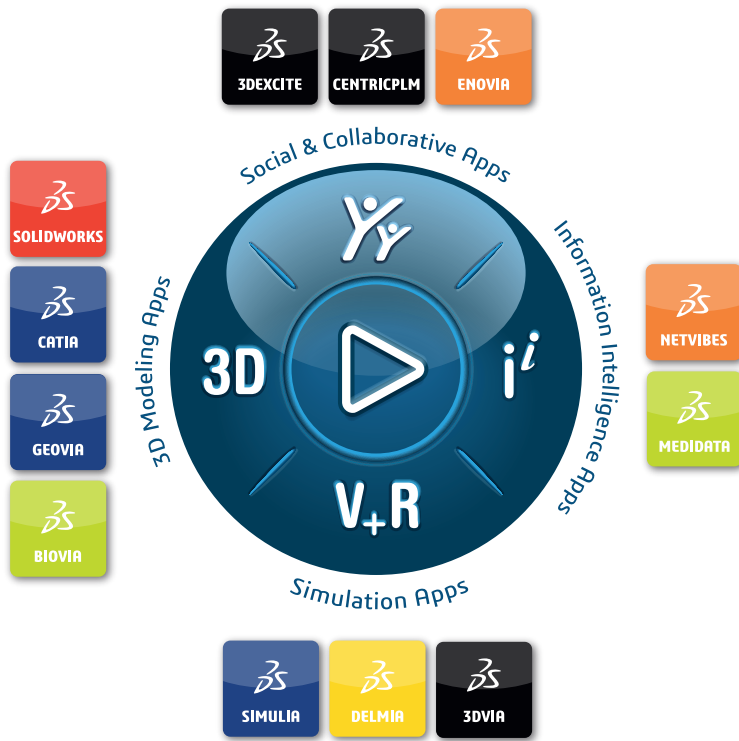
Maps the configurations to a **Physical Product** or a **Representation**.

When you select **Representation**, the following options display:

- **configuration named**

You must enter a configuration name for this option. And you must select either **active configuration** or **first configuration** as a secondary configuration to use.

- **active configuration**
 - **first configuration**
-



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