



WHAT'S NEW

SOLIDWORKS 2026



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Welcome to SOLIDWORKS Design 2026

This chapter includes the following topics:

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



SOLIDWORKS Design® 2026 contains user-driven enhancements that help streamline and accelerate your product development processes from concept to manufacturing:

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

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

Top Enhancements

The top enhancements for SOLIDWORKS® 2026 provide improvements to existing products and innovative new functionality.

Fundamentals	SOLIDWORKS Appearances on page 18
Parts and Features	Creating Reference Points by XYZ Values on page 27
Structure System and Weldments	Enhanced Corner Treatments on page 36
Detailing and Drawings	Linking PLM Attributes to Bill of Materials Columns on page 14 Automatically Generating Drawings (BETA): Section Views and Hole Callouts on page 41
Configurations	Splitting Out Configurations Into Individual Files on page 46
Import/Export	Selecting Bodies and Features of Multibody Parts on page 29

Performance

SOLIDWORKS® 2026 improves the performance of specific tools and workflows. Some of the highlights for performance and workflow improvements are:

SOLIDWORKS Simulation

The solution time for simulation studies with connectors that support distributed coupling has been improved.

DraftSight

DraftSight performance is improved with switching between sheets, zooming and panning operations, and file opening times.

When you switch to a sheet, performance is improved by an average of 66%. Switching from a sheet back to the model is improved by 78%. These enhancements were measured across several hardware configurations, from low-end setups to high-performance computers, benefiting users no matter what type of system you work on.

Zoom performance is improved by up to 55% in certain cases, and Pan performance is increased by about 38%.

Opening files averages 10% faster. This reduces wait time and maximizes the time you can dedicate to your work.

For More Information



Use the following resources to learn about SOLIDWORKS:

What's New in PDF and HTML

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

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

Interactive What's New

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

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

Online Help

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

SOLIDWORKS User Forum

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

Release Notes

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

Legal Notices

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

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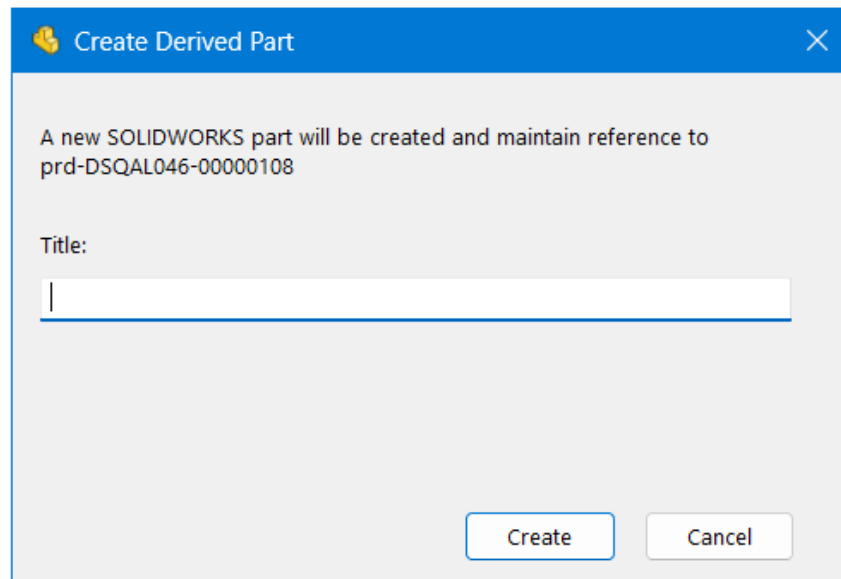
Using SOLIDWORKS on the 3DEXPERIENCE Platform

This chapter includes the following topics:

- **3DEXPERIENCE Components in SOLIDWORKS**
- **3DEXPERIENCE Transition Task in SOLIDWORKS Task Scheduler**
- **Linking PLM Attributes to Bill of Materials Columns**

This chapter covers all enhancements that affect how you use SOLIDWORKS® with the **3DEXPERIENCE**® platform. Unless otherwise noted, the entries in this chapter are available in both SOLIDWORKS Design CC and in SOLIDWORKS Design with the **3DEXPERIENCE** (Design with SOLIDWORKS) add-in.

3DEXPERIENCE Components in SOLIDWORKS



The workflow for handling **3DEXPERIENCE** components in SOLIDWORKS is optimized.

Inserting 3DEXPERIENCE Components into SOLIDWORKS Assemblies

2025	2026
A dialog box prompts you to insert the component as a 3DEXPERIENCE component or create a new SOLIDWORKS derived part.	The software inserts the component as a 3DEXPERIENCE component. No dialog box appears.

Creating Derived Parts in SOLIDWORKS Assemblies

After you insert a **3DEXPERIENCE** component into a SOLIDWORKS assembly in the FeatureManager® design tree, you can right-click the component and select **Create Derived Part**.

2025	2026
The New Document Name dialog box appears.	The Create Derived Part dialog box appears.
During the workflow, the Save to 3DEXPERIENCE dialog box appears.	The Save to 3DEXPERIENCE dialog box no longer appears. You can save to the 3DEXPERIENCE platform whenever you want after you create the derived part.

Editing 3DEXPERIENCE Components in SOLIDWORKS Assemblies

You right-click an inserted **3DEXPERIENCE** component in the Assembly's FeatureManager design tree and click **Edit Part**.

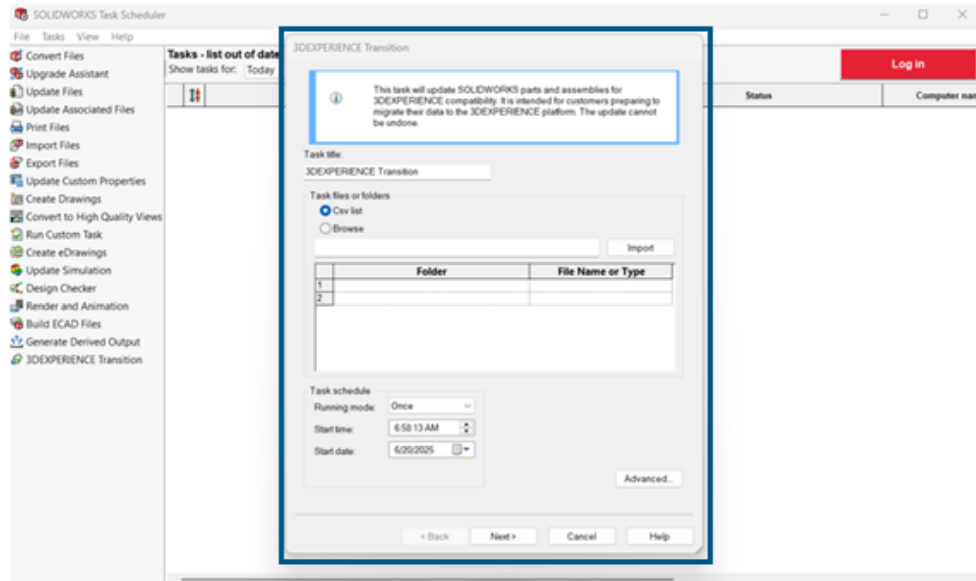
2025	2026
You can edit the component but you cannot save the changes.	The Create Derived Part dialog box appears. You can create the derived part and make changes to it.

Opening a 3DEXPERIENCE Component in SOLIDWORKS

2025	2026
You can open, view, and edit the component but you cannot save the changes. The software warns you that you cannot save changes but only when you try to save the component.	When you open the component, you cannot edit it. All tools that could modify the model are unavailable. A message bar notifies you that the document is view-only. You can still use commands that do not modify the model such as Measure , Rotate , Pan , or Zoom . In the message or the CommandManager, click Create Derived Part to open a

2025	2026
	SOLIDWORKS derived part and insert the 3DEXPERIENCE component into that part.

3DEXPERIENCE Transition Task in SOLIDWORKS Task Scheduler



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

The **3DEXPERIENCE** Transition Task replaces the **3DEXPERIENCE** Compatibility Task.

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

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

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

Creating a 3DEXPERIENCE Transition Task

To create a 3DEXPERIENCE Transition task:

1. In SOLIDWORKS Task Scheduler, click **3DEXPERIENCE Transition**.
2. Under **Task title**, create a name for your task.

- Under **Task files or folders**, select the content you want to update by doing one of the following:

- Browse for a file or folder to add to **Task Files or Folders**.
- Import a .csv file that specifies the content to add to **Task Files or Folders**.

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

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

- Run the task immediately or schedule the task.
- Click **Next**.
- In the Options dialog box, specify options:

Option	Description
Configuration option	<p>Saves only the active configuration or activates all configurations before saving.</p> <div> <p>Activating all configurations before saving can add significant time to the task.</p> </div>
3DEXPERIENCE Compatibility	<p>Updates SOLIDWORKS content for compatibility with the 3DEXPERIENCE platform. See 3DEXPERIENCE Compatibility and 3DEXPERIENCE Integration Options.</p>
File Upgrade Settings	<ul style="list-style-type: none"> Upgrades custom properties. Adds rebuild mark to all configurations. Adds display data mark to all configurations. <div> <p>Add display data mark to all configurations is unavailable if you selected 3DEXPERIENCE Compatibility.</p> </div>
Backup Files	<p>Specifies the location to back up the updated files.</p>

- Optional:** Select a macro to run on the files.
- Click **Finish**.

Running a Macro with the 3DEXPERIENCE Transition Task

To run a macro with the 3DEXPERIENCE Transition task:

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

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

Sample SOLIDWORKS Macro

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

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

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

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

Sub main()

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

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

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

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

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

    Else
```

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

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

End If

End Sub
```

Linking PLM Attributes to Bill of Materials Columns

3DEXPERIENCE users can link PLM attributes to bill of materials (BOM) columns that automatically update when you connect to the platform.

Benefits: When you migrate data to the **3DEXPERIENCE** platform, you can directly link a BOM to platform attributes.

To link PLM attributes to bill of materials columns:

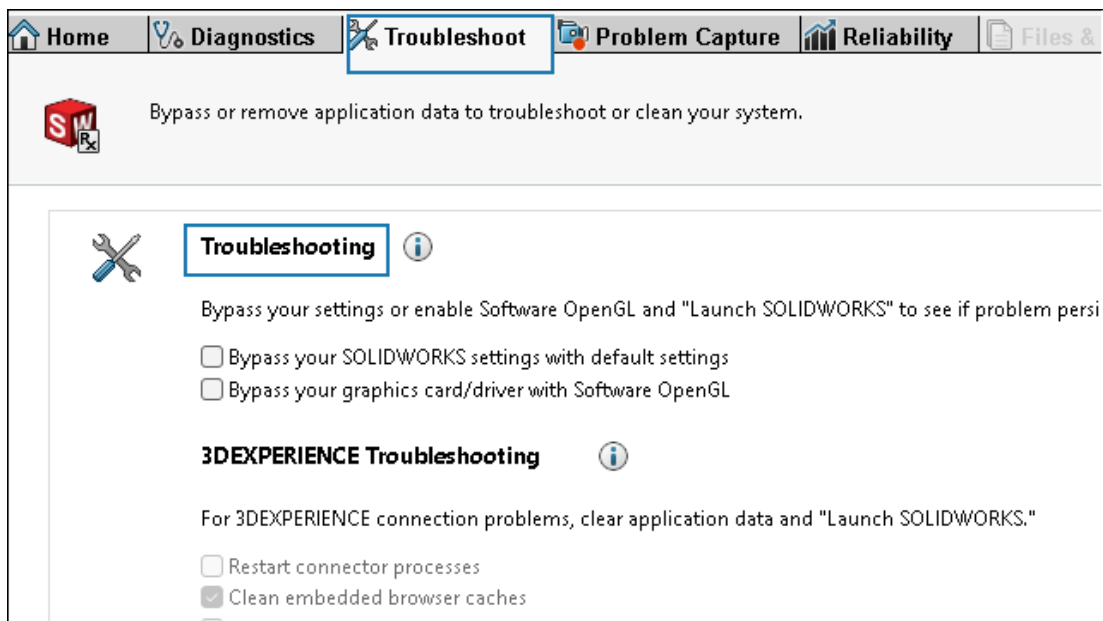
1. Double-click the top of a BOM column, above the column heading.
2. In **Column type**, select **PLM Attributes**.
3. In **Property name**, select an attribute.

When you are in offline mode, two asterisks (**) display in columns that link to PLM attributes, indicating the property is not up to date. When you connect to the **3DEXPERIENCE** platform, SOLIDWORKS updates the property.

3

Administration

Updates to SOLIDWORKS Rx



SOLIDWORKS Rx is easier to use and organizes its functionality more logically.

The **Troubleshoot** tab contains functionality previously located under **System Maintenance** and **Home > Safe Modes**.

The **Problem Capture** tool generates identical .zip content regardless of whether you select **SOLIDWORKS** or **MONITOR** as the source.

In SOLIDWORKS® Design CC, you can open SOLIDWORKS Rx from the **Start** menu. SOLIDWORKS Rx includes a **Benchmark** tab for performance evaluations.

For **3DEXPERIENCE**® users, you have additional troubleshooting options. If you encounter sign-in issues while connecting to **3DEXPERIENCE**, you can reset the connection:

1. **Close Connector processes.** Closes down the connector processes ENOPLMCSAClient.exe and EdmServerV6.exe, and restarts the **3DEXPERIENCE** connection.
2. **Clear embedded browser caches.** Clears the SOLIDWORKS Chromium Embedded Framework (CEF) cache (%temp%\swcefcache) and the WebView2

(%temp%\DSTempWebview2) used by the sign-in dialog box and the **3DEXPERIENCE** Task Pane.

3. **Clear the 3DEXPERIENCE temporary directory.** Clears settings and cached data used by SOLIDWORKS and other **3DEXPERIENCE** apps to customize and accelerate interactions with the **3DEXPERIENCE** platform. The temporary directory is located at %localappdata%\DassaultSystemes\CATTemp.

4

SOLIDWORKS Fundamentals

This chapter includes the following topics:

- **Changes to System Options**
- **SOLIDWORKS Appearances**
- **Render with SOLIDWORKS Visualize from SOLIDWORKS**
- **Deleting Equations for Sketches and Features**
- **Application Programming Interface**

Changes to System Options

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

System Options

Option	Description	Access
Mark assembly as modified when cosmetic changes are made to referenced documents	Specify if a rebuild is required for nonessential changes.	Performance
Enhanced graphics performance (requires SOLIDWORKS restart)	Required for SOLIDWORKS to use the DSPBR model.	Performance


Option	Description	Access
DSPBR (Dassault Systèmes Physically Based Rendering)	<p>Renders models with appearances from DSPBR. These appearances are consistent with SOLIDWORKS Visualize and the 3DEXPERIENCE platform and result in more realistic visuals.</p> <p>When selected, the Appearances PropertyManager includes a DSPBR tab. This tab ensures direct material-to-material translation. All parameters from SOLIDWORKS map directly to Visualize.</p>	Display
Legacy Appearances	Uses older rendering functionality prior to SOLIDWORKS 2026.	Display
Show messages icon in status bar	When selected, lets you quickly access dismissed messages from the status bar.	Messages/ Errors/ Warnings > Dismissed messages

SOLIDWORKS Appearances



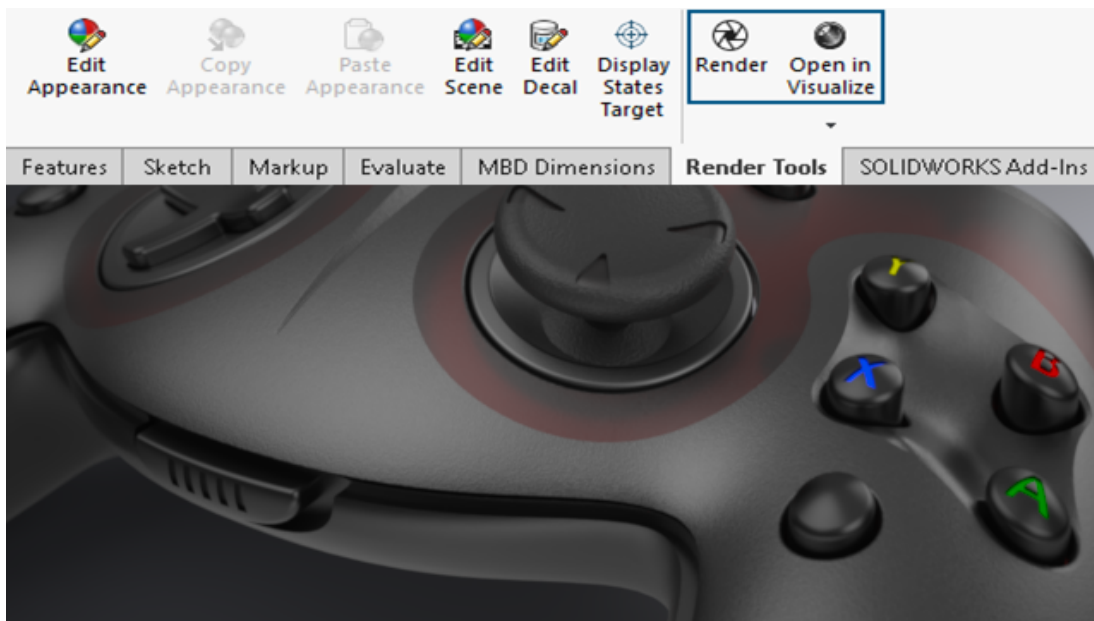
You can use a more extensive library of appearances in SOLIDWORKS® through Dassault Systèmes' Enterprise PBR Shading Model (DSPBR). These appearances are consistent with SOLIDWORKS Visualize and the **3DEXPERIENCE** platform and result in more realistic visuals.

You can render appearances with DSPBR, which is available when you:

- Select **Enhanced graphics performance (requires SOLIDWORKS restart)** in **Tools > Options > System Options > Performance**
- Select **DSPBR (Dassault Systèmes Physically Based Rendering)** under **Appearance Visual Style** in **Tools > Options > System Options > Display**
- Turn on **RealView Graphics** 



The rendering lighting model includes high dynamic range and image based lighting for more physically correct depictions of lighting.

Render with SOLIDWORKS Visualize from SOLIDWORKS





If you have a SOLIDWORKS Visualize installation with a license, you can generate high-quality photorealistic final renders from SOLIDWORKS using SOLIDWORKS Visualize.

The RenderTools CommandManager includes the following tools.

	Render	Generates high-quality renders from SOLIDWORKS using the SOLIDWORKS Visualize Render PropertyManager.
	Open in Visualize	Specifies options that load a SOLIDWORKS model into Visualize.

Rendering with SOLIDWORKS Visualize from SOLIDWORKS

To render with SOLIDWORKS Visualize from SOLIDWORKS:

1. In the CommandManager, click **Render**  (Render Tools tab) or **Render Tools** > **Render**.
2. Specify options in the PropertyManager and click .



SOLIDWORKS Visualize Render PropertyManager

You can use this PropertyManager to specify settings for final renderings.

To open this PropertyManager:

1. In the CommandManager, click **Render**  (Render Tools tab) or **Render** (Render Tools toolbar).


Output Image Settings

	File Name	Specifies the model name.
	Output Folder	Specifies the model location.

Media

Format	Specifies the rendering's output format.
Include Alpha	Specifies whether to add the alpha channel (to preserve transparency) in the final rendering (RGB or RGBA).

Size

Presets	Lists standard aspect ratios for the render camera.
Landscape/Portrait	Specifies a horizontal or vertical orientation for the render camera.
	Width/Height Specifies a custom aspect ratio for the render camera.

Quality





Specifies the number of render passes.

Quality	Number of render passes
Low	50
Medium	100
High	500

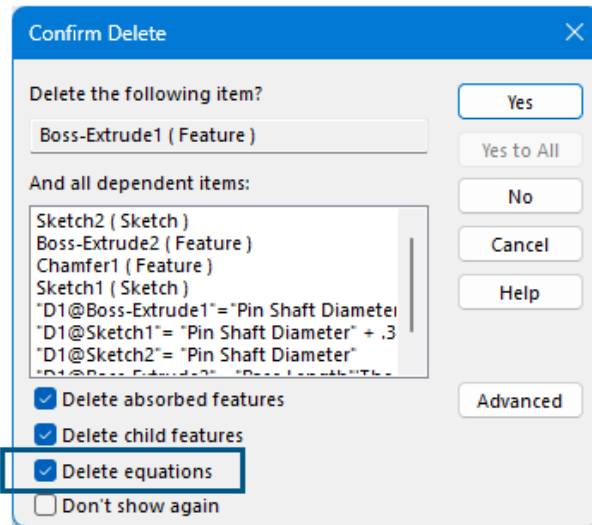
Loading SOLIDWORKS Models into SOLIDWORKS Visualize

You can load SOLIDWORKS models directly into Visualize and adjust them further using Visualize functionality.

To load SOLIDWORKS models into SOLIDWORKS Visualize:

1. In the CommandManager, click **Open in Visualize**  (Render Tools tab) or **Render Tools > Open in Visualize**.
2. Select an option:
 - **Group by Appearance** . Opens the SOLIDWORKS model in Visualize, grouping all parts based on the SOLIDWORKS appearances that you applied to them.
 - **Group by Part** . Opens the SOLIDWORKS model in Visualize, grouping all parts based on SOLIDWORKS components.
 - **Import with Options** . Imports the SOLIDWORKS model into Visualize where you can choose import options.

Deleting Equations for Sketches and Features



For sketches and features that contain equations, if you delete the sketch or feature, you can delete the equation directly from the Confirm Delete dialog box. Previously, you had to manually delete the equation using the Equations, Global Variables, and Dimensions dialog box.

Benefits: You have more control over how the deletion of sketches or features impacts related equations. This helps you maintain or clean up the model as required.

This functionality applies to parts only.

When you delete the sketch or feature, in the Confirm Delete dialog box, select **Delete equations** to delete the associated equations.

Application Programming Interface

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

- SOLIDWORKS API includes the ability to:
 - Notify programs when you reorder cut list and solid body folder features
 - Add and edit family tables in drawings
 - Move drawing views independently of child views
- Simulation API. Support for cable connectors

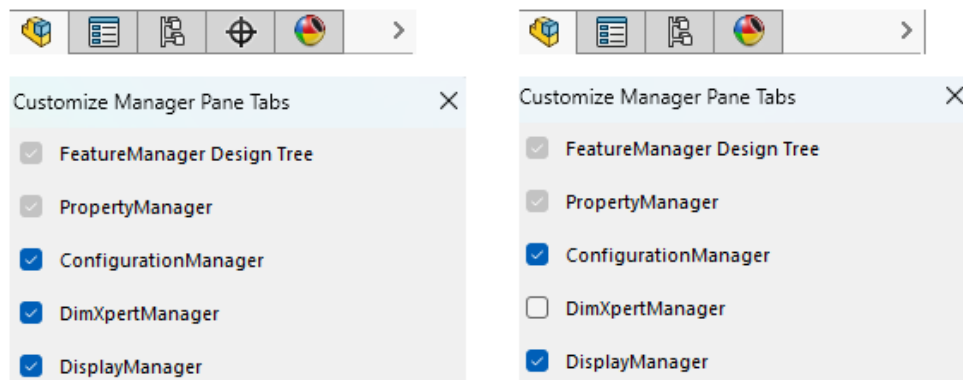
5

User Interface

This chapter includes the following topics:

- **Hiding and Showing Manager Pane Tabs**
- **Viewing Dismissed Messages**
- **Usability**
- **Selection Filters**
- **Other User Interface Enhancements**

Hiding and Showing Manager Pane Tabs

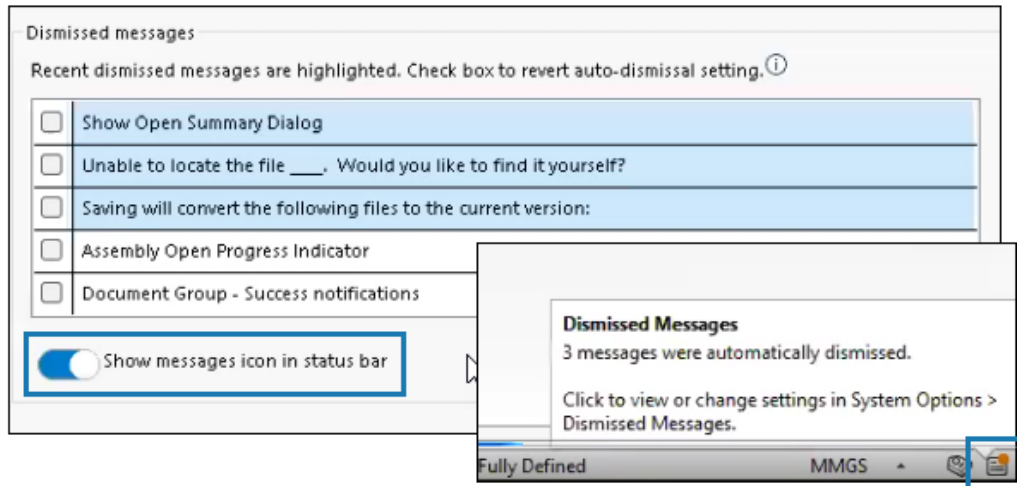


You can hide and show Manager Pane tabs to let you focus on the tabs that are most relevant to your work.

To hide and show Manager Pane tabs:


1. Right-click any Manager Pane tab and click **Customize**.
2. In the dialog box, specify the tabs to hide or show.

Viewing Dismissed Messages

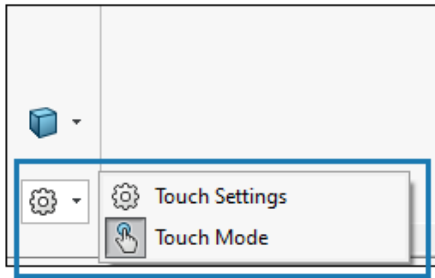
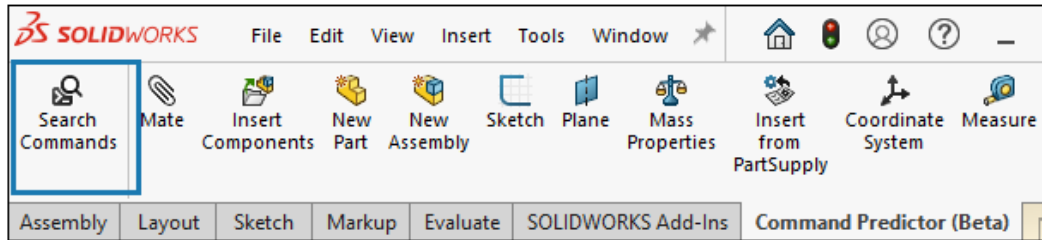


The status bar in the SOLIDWORKS window has a **Dismissed Messages** icon that lets you quickly view the messages that you have dismissed earlier and define the settings to view those messages again.

Improvements:

- In **Tools > Options > System Options**, click **Messages/Errors/Warnings > Dismissed messages**. You can turn on **Show messages icon in status bar** to quickly access messages from the status bar.
- Hover over **Dismissed Messages**  on the status bar. The message box displays the number of recently dismissed messages. Click the icon to access the Dismissed Messages page.

Usability



The user interface is enhanced to improve productivity.

- The warning message for a Toolbox upgrade displays only once. Previously, this message displayed for each missing component of an assembly.

The warning message box has a dismiss timer that automatically closes the message box.

- The smart positioning of the Modify Dimension dialog box avoids overlapping the dimension being edited.
- In touch mode, when the **Lock 3D Rotate** option is off, you can pan a drawing with single-touch drag. This avoids accidental modifications to the drawing, especially while marking up the file.
- On the Touch Mode toolbar, the **Touch Settings** tool has options to directly open the System Options - Touch dialog box and to hide the Touch Mode toolbar.
- On the Command Predictor tab, click **Search Commands** to search for a SOLIDWORKS® tool.

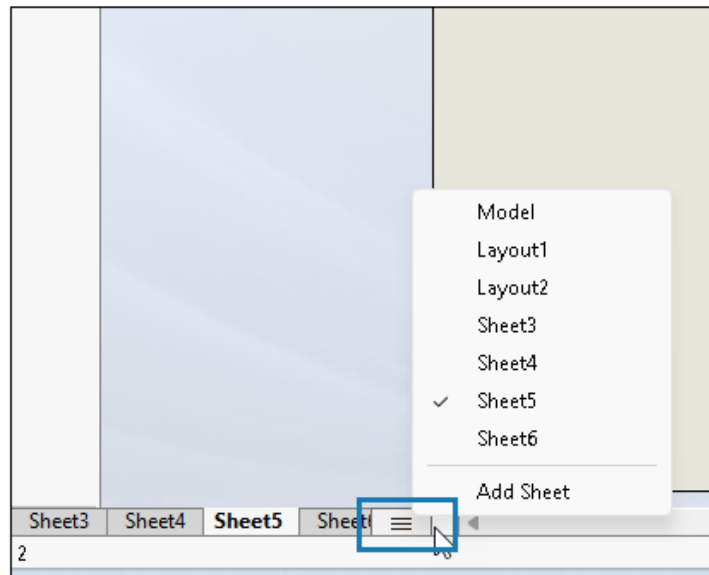
Selection Filters

Improvements to the Selection Filter toolbar let you work more precisely and efficiently when modeling.

- **Feature Filter.** Lets you select and delete features in parts and assemblies. You can select feature like cut extrudes, fillets, and holes directly from the graphics area.
- **Component Filter.** Lets you select top-level parts and subassemblies within an assembly.
- You can assign shortcut keys to the following tools:
 - **Filter Surface Bodies**




- **Filter Solid Bodies**
- **Filter Midpoints**
- **Filter Center Marks**
- **Filter Centerlines**
- The **Tools > Customize > Keyboard** includes a **Selection** category with all **Selection Filter** tools.

Other User Interface Enhancements



The user interface gives you a better experience when working with multiple drawing sheets or tabs, moving folders in the FeatureManager® design tree, and working on multiple monitors.

Improvements:

- At the bottom of a drawing sheet or tab, click **List**  to view a list of sheets or tabs that are open. At the bottom of the list, **Add Sheet** or options to create a new study are available.
- Limitations while moving the folders below the **Origin**  and above **Mates**  feature are resolved. You can move components and folders up or down in the FeatureManager design tree.
- While working with multiple monitors, when you have a number of files open on a single monitor and you click **Window > Tile Horizontally/Tile Vertically**, the software evenly distributes files for display and tiles them to all the available monitors. For example, if there are six files and three monitors, the software tiles and displays two files in each monitor.

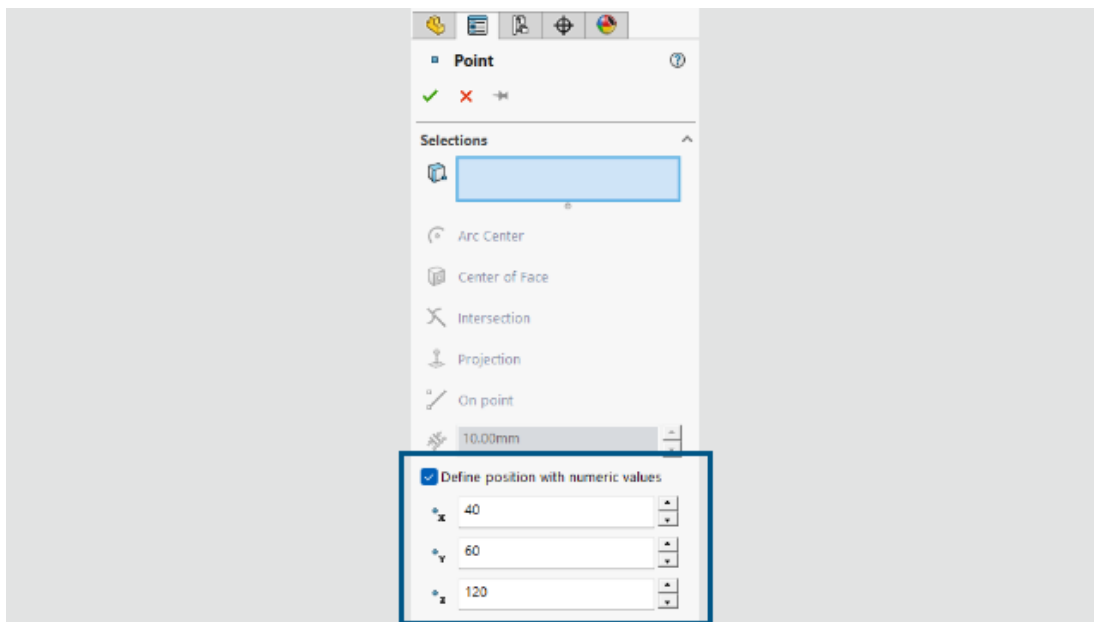
6

Parts and Features

This chapter includes the following topics:

- **Creating Reference Points by XYZ Values**
- **Exiting Part Processes with the Escape Key**
- **Selecting Bodies and Features of Multibody Parts**
- **Using a Coordinate System to Define a Bounding Box**

Creating Reference Points by XYZ Values







You can create reference points by specifying absolute numeric values for the X, Y, and Z coordinates.

Benefits: You have improved control over positioning reference points.

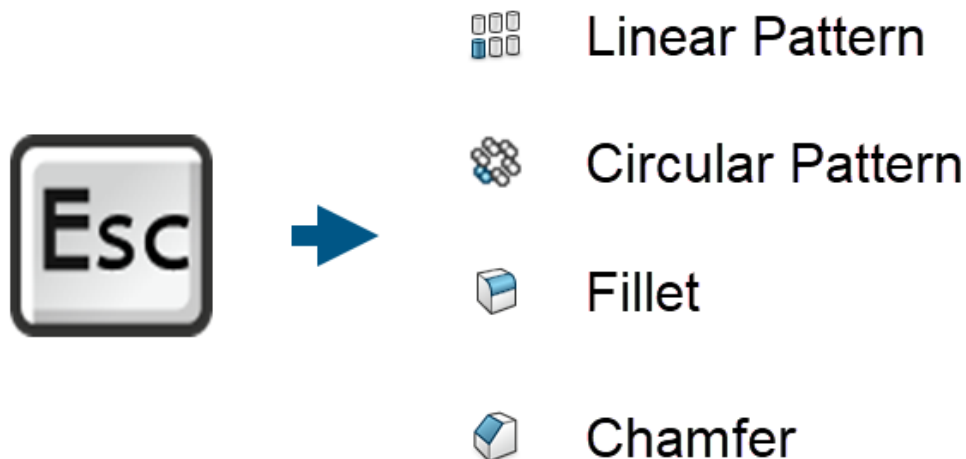
To create reference points by specifying XYZ values:

1. In a part or assembly file, click **Insert** > **Reference Geometry** > **Point**.
2. In the PropertyManager, select **Define position with numeric values**.

The options under **Selections** become unavailable.

3. Specify the **X Coordinate** , **Y Coordinate** , and **Z Coordinate**  values to define the reference point position relative to the origin (0,0,0).
4. Click .

Exiting Part Processes with the Escape Key





To immediately exit out of lengthy part processes, press **Esc** to cancel the ongoing tool and revert the model to its previous state. This applies to the **Linear Pattern**, **Circular Pattern**, **Fillet**, and **Chamfer** tools.

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

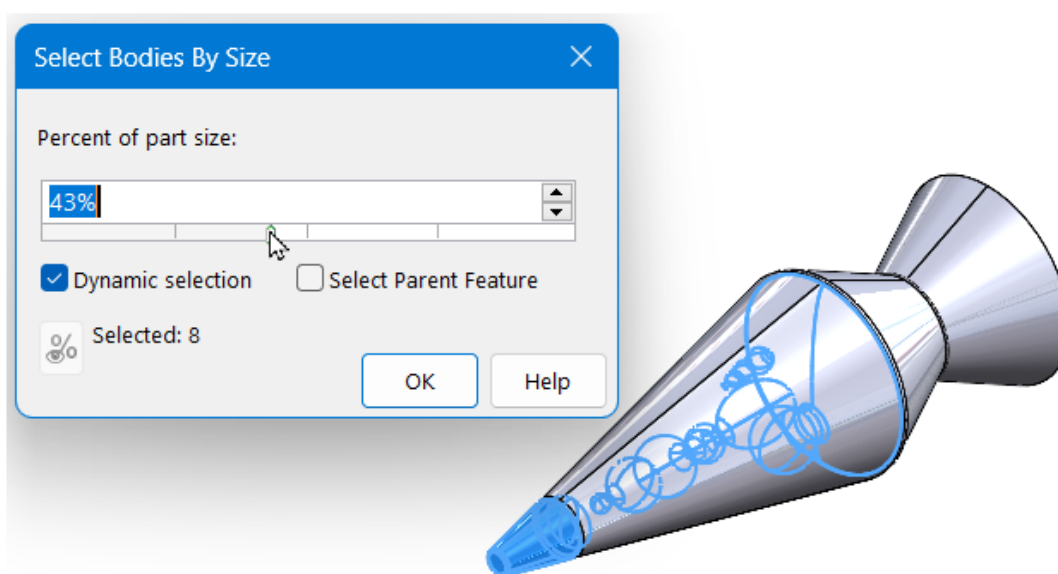
Status bar messages during a preview or main operation alert you that this functionality is available: Press <ESC> to cancel Preview or Press <ESC> to cancel <Linear Pattern/Circular Pattern/Fillet/Chamfer> command.

Press **Esc** during these tools to exit the described processes.

Tool	PropertyManager Actions That You Can Exit
Linear Pattern and Circular Pattern	<ul style="list-style-type: none"> Click  to start running the tool. Select a feature or face. Specify options under Direction 1 or Direction 2. Specify a Preview type. Select Instances to Skip. Select Instances to Vary.

Tool	PropertyManager Actions That You Can Exit
Fillet and Chamfer	<ul style="list-style-type: none"> Click  to start running the tool. Select Items to Fillet or Items to Chamfer. Specify a Preview type.

Selecting Bodies and Features of Multibody Parts




When you open multibody parts in SOLIDWORKS®, you can use several selection tools to view discrete bodies and features of the model. Previously, no selection methods were available.

Benefits: You can hide, add, delete, or suppress nonessential bodies and features to improve model performance and help you complete your task more quickly.

These tools let you select discrete bodies and features of multibody parts:

- **Select Bodies By Size**
- **Select Bodies By Volume**



To access these tools, open a multibody part and click **Tools > Selection** or right-click the **Selection Tools** flyout menu  and select a tool.

Select Bodies By Size

Specify a percentage of the part size that you want to select. Specify these options:

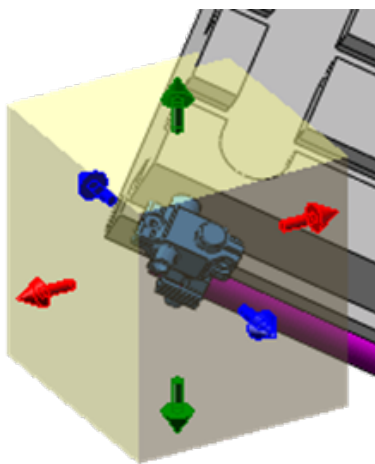
- **Dynamic selection.** Displays a dynamic preview of the selections as you change the value for **Percentage of part size**.

- **Select parent features.** Selects the parent features in the FeatureManager® design tree. This selects the discrete features that make up the bodies. When cleared, the software selects only the bodies themselves. Depending on your selection, you can proceed with your required actions such as deleting, hiding, or suppressing entities.

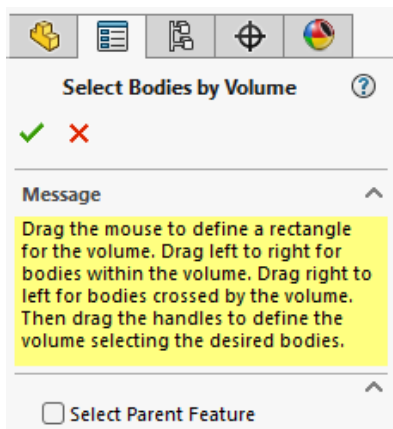
If you import neutral-format multibody parts, to create **Imported**  parent features, in the FeatureManager design tree, right-click the Imported part icon  and click **Break Link**. You cannot undo this action.

Select Bodies By Volume

Select bodies based on a temporary volume that you define.



Follow the directions in the PropertyManager.

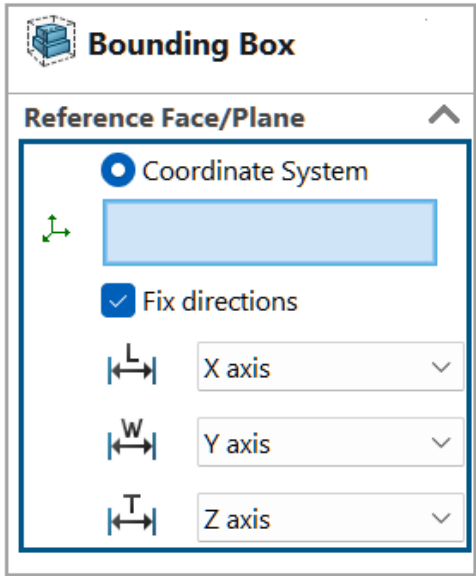


Drag to define a rectangle for the volume.

- Drag left to right to select bodies within the volume.
- Drag right to left to select bodies crossed by the volume.

Then drag the handles to define the volume that selects the required bodies.


Using a Coordinate System to Define a Bounding Box



You can define a bounding box using a coordinate system.




For a rectangular bounding box, you can specify the X, Y, and Z axes for length, width, and thickness. For a cylindrical bounding box, you can specify one of the X, Y, and Z axes for the axis of the cylinder.

To use a coordinate system to define a rectangular bounding box:

1. Open a model, and click **Insert** > **Reference Geometry** > **Bounding Box** .
2. In the PropertyManager, under **Type of Bounding Box**, select **Rectangular**.
3. Under **Reference Face/Plane**, select **Coordinate System** and select a coordinate system.

A rectangular bounding box is created. The edges are parallel to the X, Y, and Z axes of the coordinate system.

4. To change the axis for a direction, select **Fix directions** and select an axis for a direction:

	Length	Specifies the axis for length.
	Width	Specifies the axis for width.
	Thickness	Specifies the axis for thickness.

To use a coordinate system to define a cylindrical bounding box:

1. In the PropertyManager, under **Type of Bounding Box**, select **Cylindrical**.

2. Under **Reference Face/Plane**, select **Coordinate System** and select a coordinate system.

A cylindrical bounding box is created. The axis is parallel to one of the X, Y, or Z axes of the coordinate system with the Z axis as the default axis.

3. To change the axis direction, select **Fix directions** and select an axis:

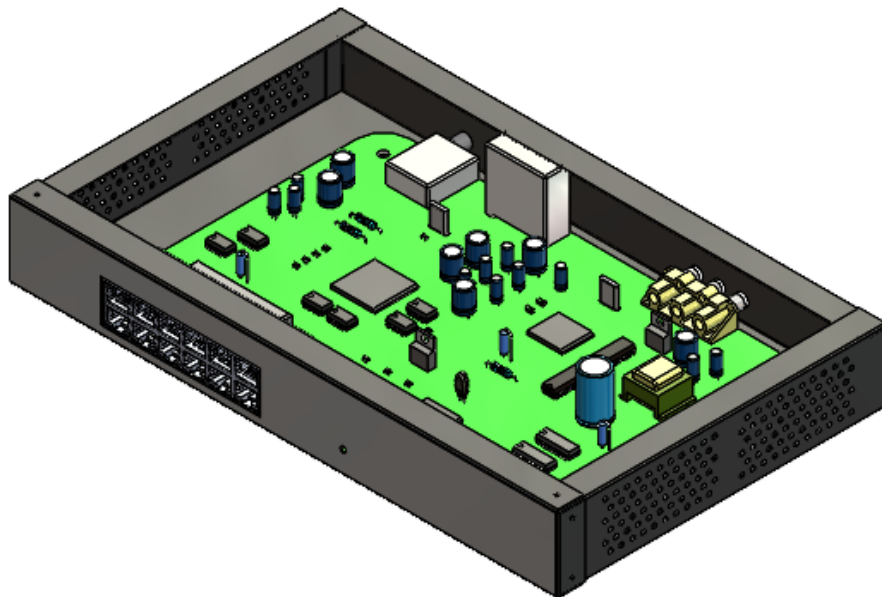


Axis

Specifies the axis for the cylindrical bounding box.

Sheet Metal

Base Flange Starting Conditions



You can specify starting conditions when creating base flanges such as sketch planes, surfaces, and offsets.

In the Base Flange PropertyManager, under **From**, specify a starting condition:

Starting Condition	Description
Sketch Plane	Starts the base flange from the plane on which the sketch is located.
Surface/Face/Plane	Starts the base flange from the specified surface, face, or plane. The entity must be planar.
Vertex	Starts the base flange from the selected vertex. The entity can be a sketch point or model vertex.

Starting Condition	Description
Offset	Starts the base flange on a plane that is offset from the sketch plane at the distance you specify.

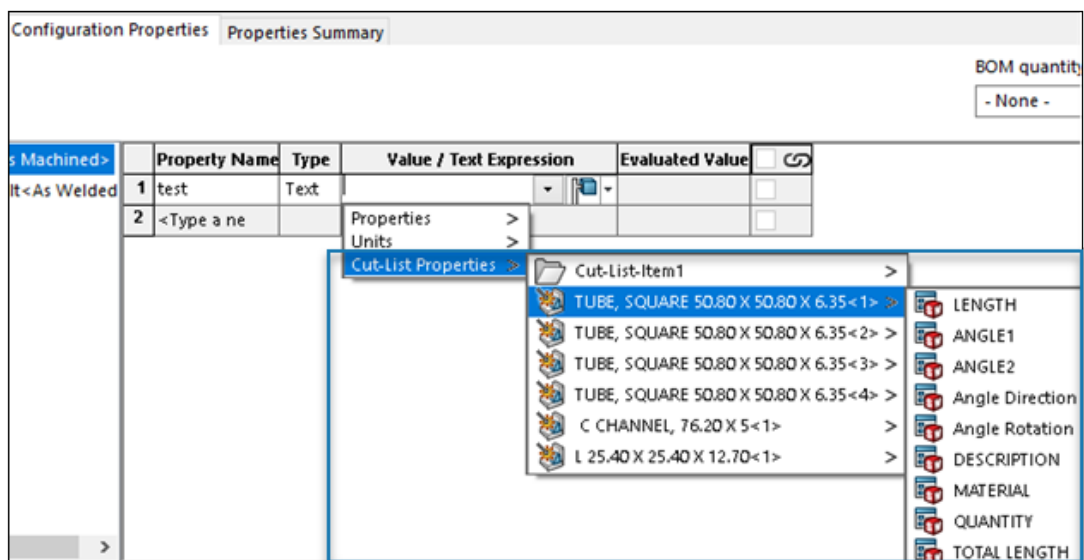
8

Structure System and Weldments

This chapter includes the following topics:

- **Accessing Cut List Properties from File Properties**
- **Enhanced Corner Treatments**

Accessing Cut List Properties from File Properties



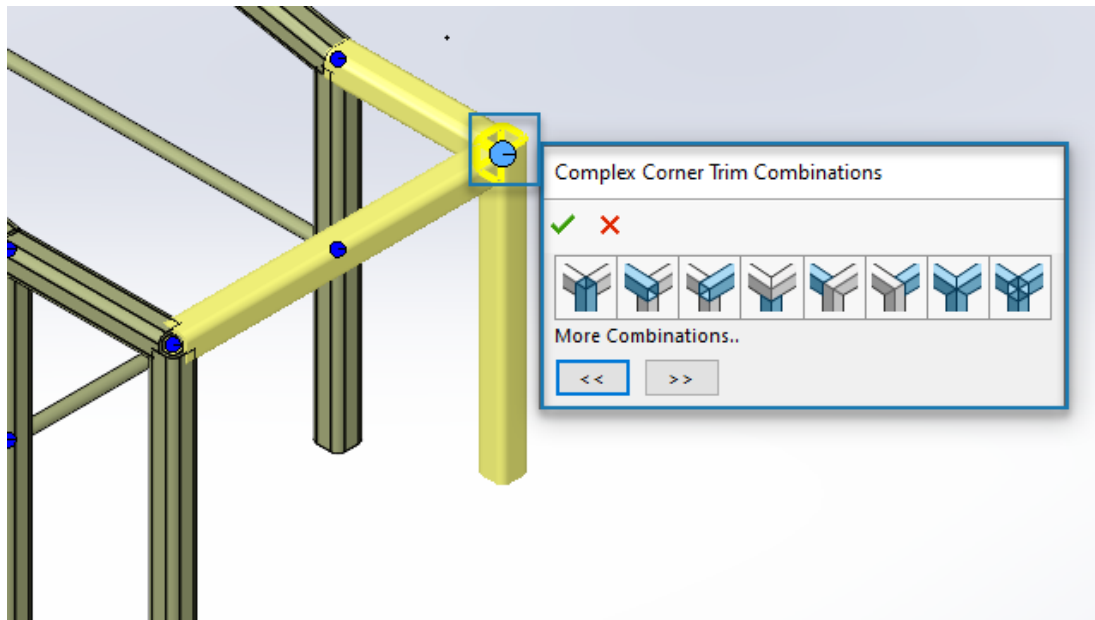
You can access cut list properties from the Configuration Properties tab of the Properties dialog box.

To access cut list properties from file properties:

1. Click **File Properties** (Standard toolbar).
2. In the dialog box, on the Configuration Properties tab, select the configuration name.
3. For **Property Name**, enter the name of the property.
4. For **Value/Text Expression**, select **Cut-List Properties**.
5. From the **Cut-List Properties** flyout menu, select a cut list body and its specific cut list property.





Value/Text Expression displays the formula: `$PRPWLID:"*Cutlist Item name*:*Cutlist Property Name"`. SOLIDWORKS® links the selected cut list property to the formula.

Enhanced Corner Treatments



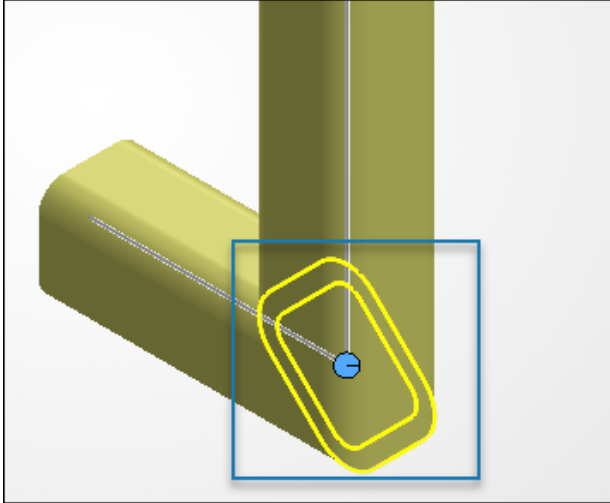
You can select the appropriate combination of trim type and trim order for a complex corner from the graphics area.

To select outputs of corners based on the trim type and trim order:

1. Open a structure system part.
2. In the FeatureManager design tree, right-click **Corner Management** and click **Edit Feature** .
3. In the graphics area, select a complex corner.
4. In the Complex Corner Trim Combinations dialog box, select options to represent the outputs of a corner based on the trim type and trim order.
5. Click  or  to select additional combinations.
6. Click .

The output options appear in the graphics area when three or more members meet at a point.

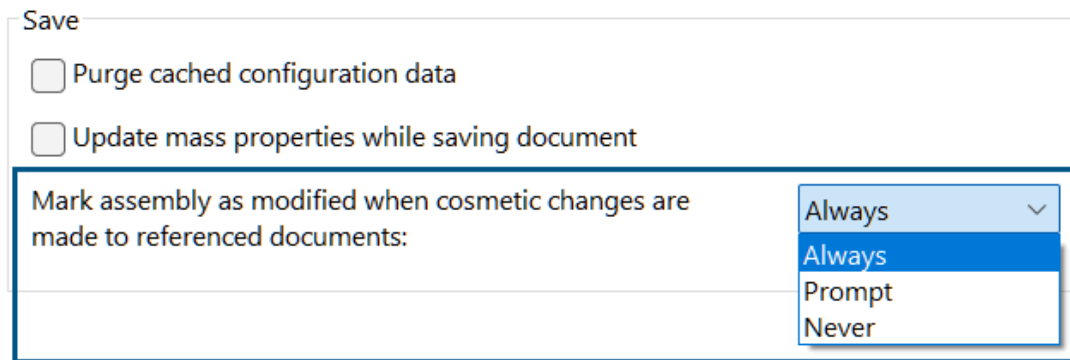
When members meet at a point, yellow borders display the trimmed surface.



9

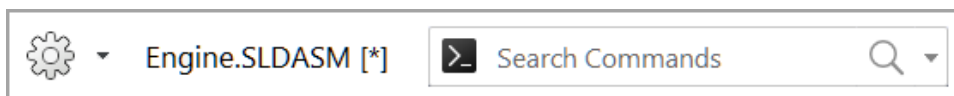
Assemblies

Specifying Rebuild Requirements for Cosmetic Changes



You can use **Mark assembly as modified when cosmetic changes are made to referenced documents** to specify rebuild requirements for nonessential changes.

When you select **Prompt** or **Never** and make a cosmetic change, the model is not marked as modified. In the menu bar, the document name shows with brackets and an asterisk.



Modifications that do not require a rebuild:

- Adding, modifying, or deleting reference geometry in a part.
- Changing the visibility of reference geometry in a part.
- Adding, modifying, deleting, or hiding a sketch in a part where the sketch does not drive any geometry.
- Hiding or showing bodies when the referencing assembly includes hidden bodies or components in the mass property calculations.

- Hiding or showing bodies when the referencing assembly does not include hidden bodies or components in the mass property calculations.
- Adding, modifying, or removing a part appearance.
- Adding, modifying, or deleting a decal from a part.
- Canceling changes to a feature in a part.

To specify rebuild requirements for cosmetic changes:

1. Click **Tools > Options > System Options > Performance**.
2. Under **Save**, select **Mark assembly as modified when cosmetic changes are made to referenced documents** and select an option:

Always	Rebuilds the assembly for all cosmetic changes.
Prompt	Asks if you want to rebuild for each cosmetic change.
Never	Does not rebuild the assembly for cosmetic changes.

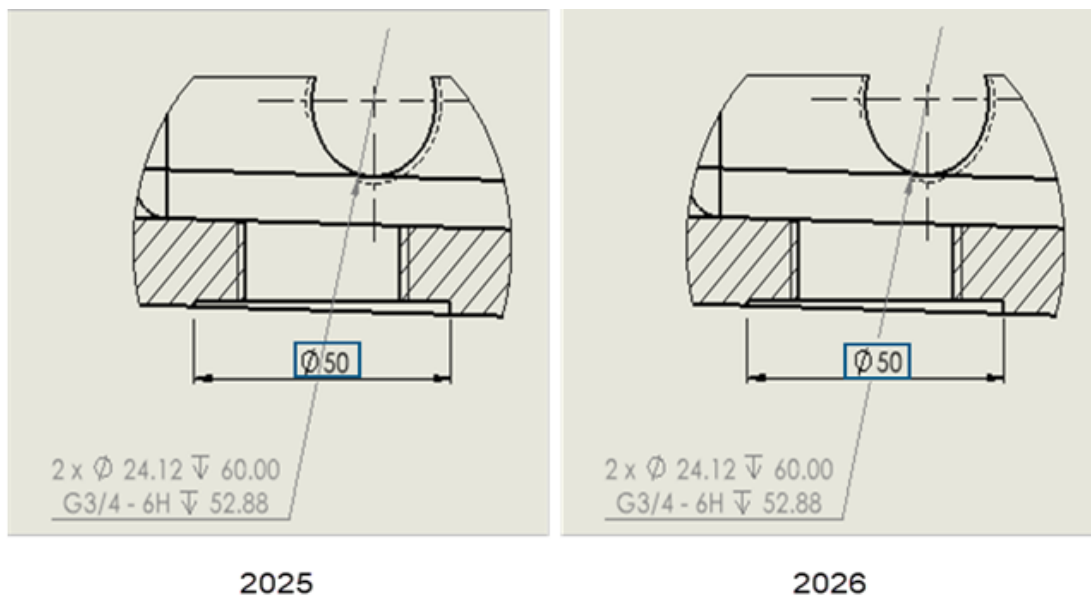
10

Detailing and Drawings

This chapter includes the following topics:

- **Adding Breaks to Dimension Lines around Dimension Text**
- **Automatically Generating Drawings (BETA): Section Views and Hole Callouts**
- **Specifying Text and Symbols in Geometric Tolerance Symbol Ranges**
- **Using Magnetic Lines to Align Annotations**
- **Using Indicators with Surface Finish Symbols**

Adding Breaks to Dimension Lines around Dimension Text



You can add breaks to dimension lines to avoid overlapping dimension text.

To add breaks to dimension lines around dimension text:

1. Right-click a dimension line that overlaps dimension text and click **Add Dimension Break**.
2. Optional: After you add a dimension break, you can right-click and select:
 - **Remove Dimension Breaks**. Removes an existing dimension break.
 - **Update Dimension Break**. Modifies an existing dimension break.

Automatically Generating Drawings (BETA): Section Views and Hole Callouts

3DEXPERIENCE® users can automatically generate drawings (BETA) of parts and assemblies including details such as section views and hole callouts.

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

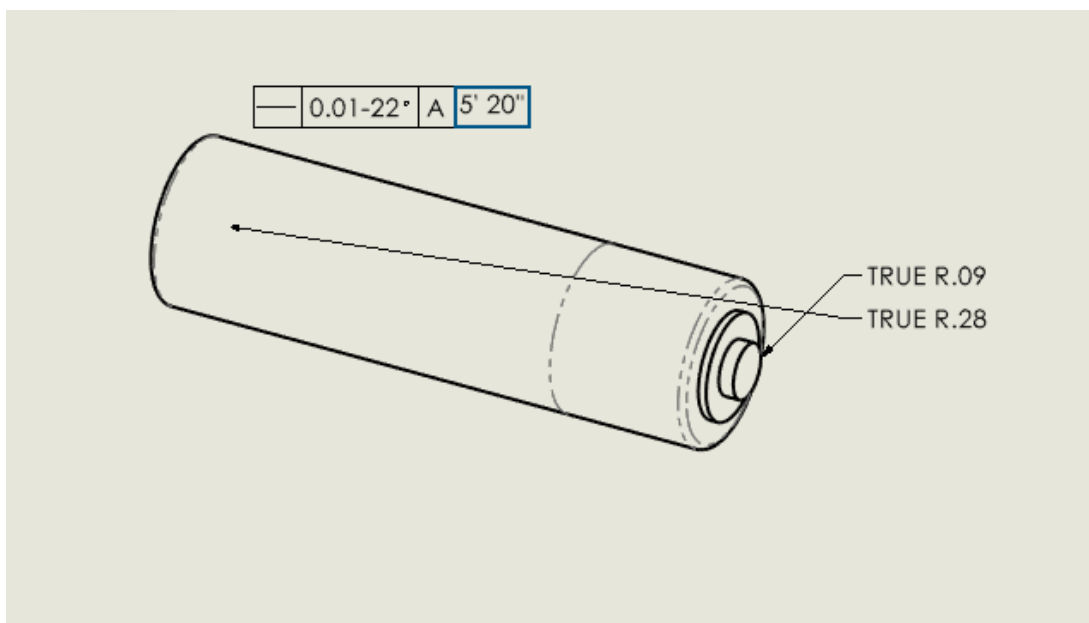
THIS IS A BETA FEATURE UNDER EVALUATION. ANY DECISION TO USE IT IS SUBJECT TO IMPORTANT TERMS AND CONDITIONS THAT CUSTOMER UNDERSTANDS AND ACCEPTS BY USING IT; Please refer to the OST available at <https://www.3ds.com/terms> for these important terms and conditions.

Auto-Generate Drawings (BETA) automatically creates:

- Section views, such as views with internal feature dimensions.
- Hole callouts for drawings generated for imported models, such as STEP.

SOLIDWORKS determines the best sheet size from the drafting standard that you select for a part or assembly so the view layout scales to fit the sheet.

Specifying Text and Symbols in Geometric Tolerance Symbol Ranges




When you create geometric tolerance symbol ranges, you can add text and symbols.


You can add:

- Angle degree minute (') and second (") symbols and letters
- Text and symbols
- Text boxes in the second section of a feature control frame

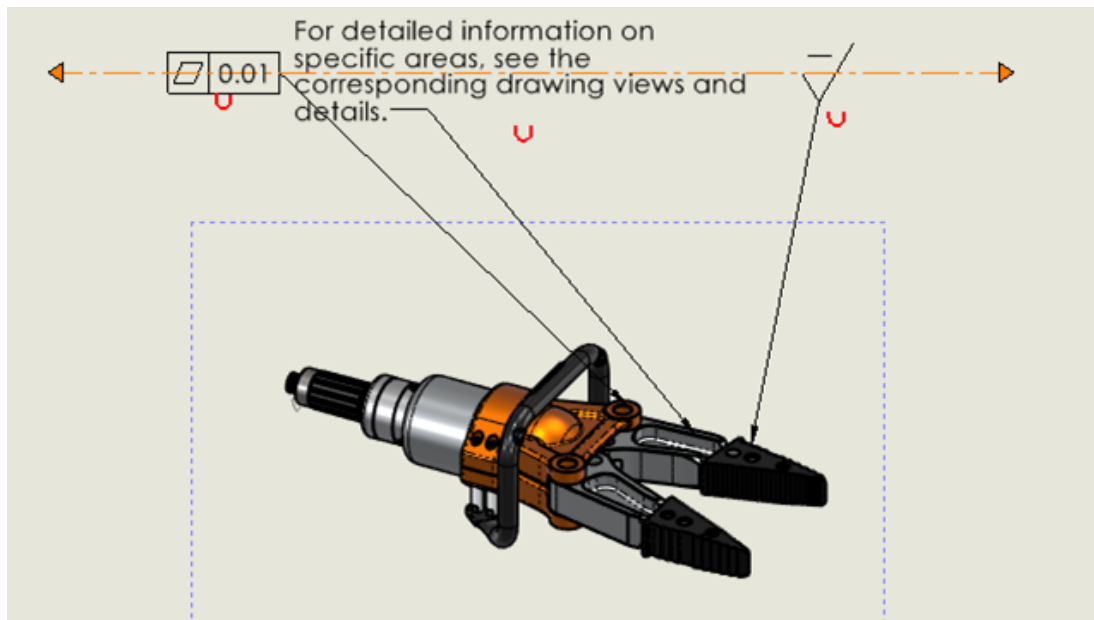
To specify text and symbols in geometric tolerance symbol ranges:

1. In a drawing, click **Geometric Tolerance**  (Annotations toolbar) or **Insert** > **Annotations** > **Geometric Tolerance**.
2. In the graphics area, click to place the symbol.

A feature control frame appears with handles and a Tolerance dialog box surrounding it.


3. In the Tolerance dialog box:
 - a. Specify a symbol.
 - b. Select **Range**.
 - c. Specify text and symbols.
 - d. Click **Done**.
4. In the Geometric Tolerance PropertyManager, click .

Using Magnetic Lines to Align Annotations

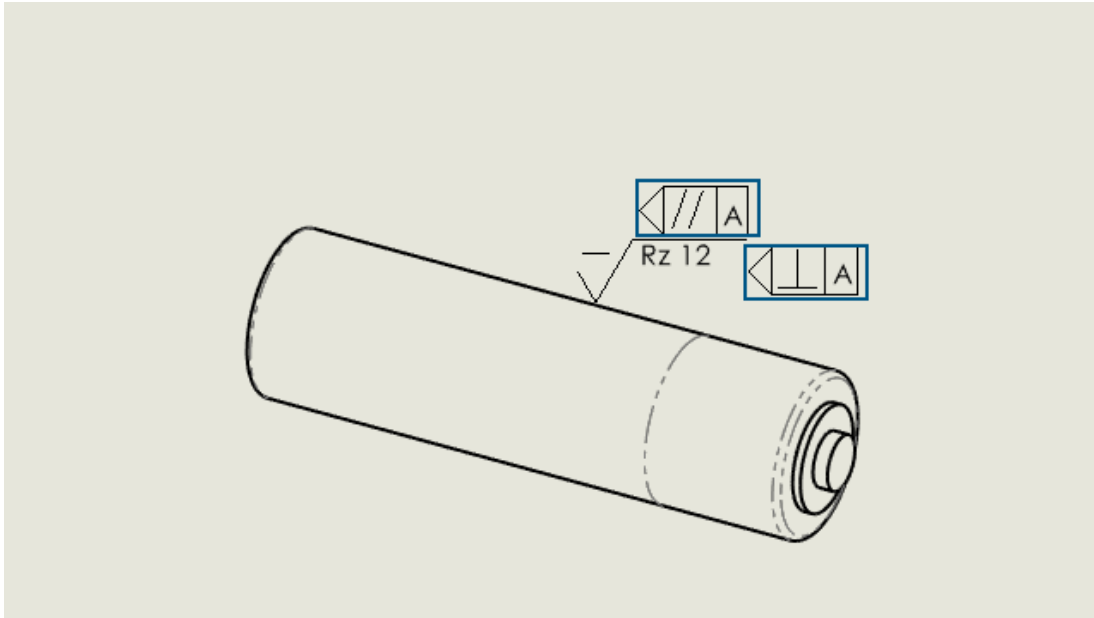



You can use magnetic lines to align annotations, such as notes, weld symbols, geometric tolerance symbols, surface finish symbols, and revision symbols, to improve drawing presentations.

To use magnetic lines to align annotations:




1. Click **Magnetic Line**  (Annotations toolbar) or **Insert** > **Annotations** > **Magnetic Line**.

Using Indicators with Surface Finish Symbols



When you insert a new surface finish symbol, you can use indicators  to place the symbols.

To use indicators with surface finish symbols:

1. In the Surface Finish PropertyManager, under **Symbol Layout**, click an indicator .
2. In the Indicator dialog box:
 - Under **Tolerance Type** specify **Parallel**  or **Perpendicular** .
 - (Optional) Under **Datum Reference**, type a datum.
 - Click **Close**.
3. In the graphics area, click to place the symbol.

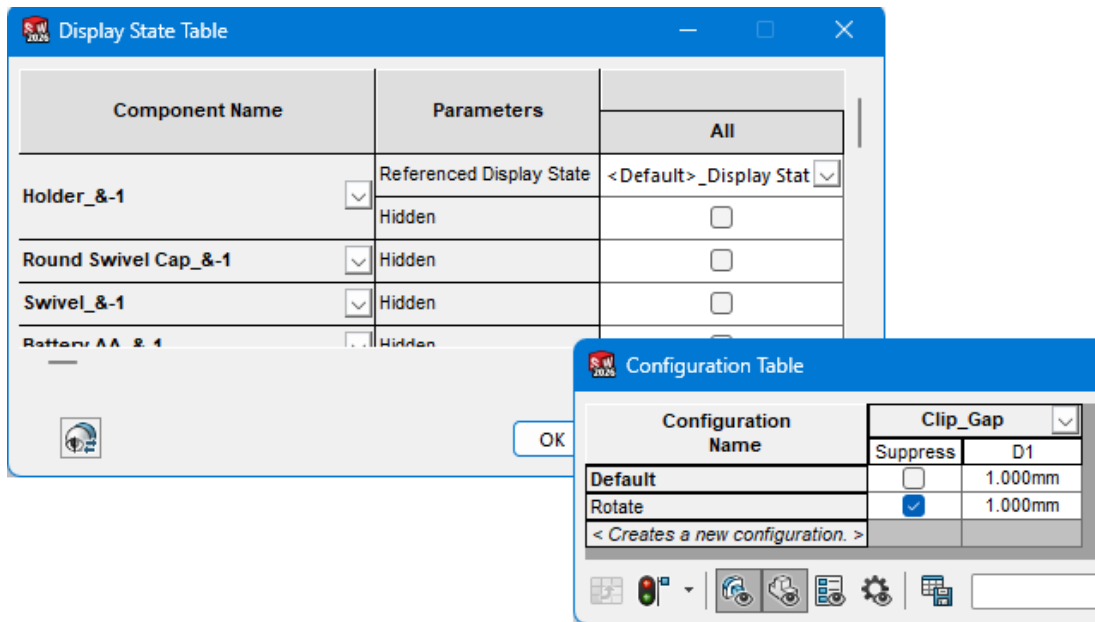
11

Configurations

This chapter includes the following topics:

- **Configuration Tables and Display State Tables Usability**
- **Splitting Out Configurations Into Individual Files**

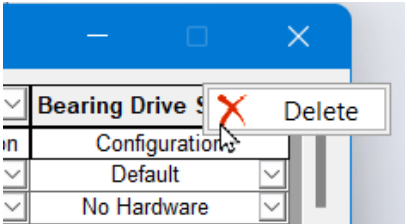
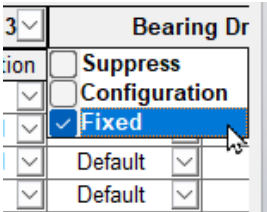
Configuration Tables and Display State Tables Usability



The usability is improved for configuration tables and display state tables.

Configuration Table Improvements

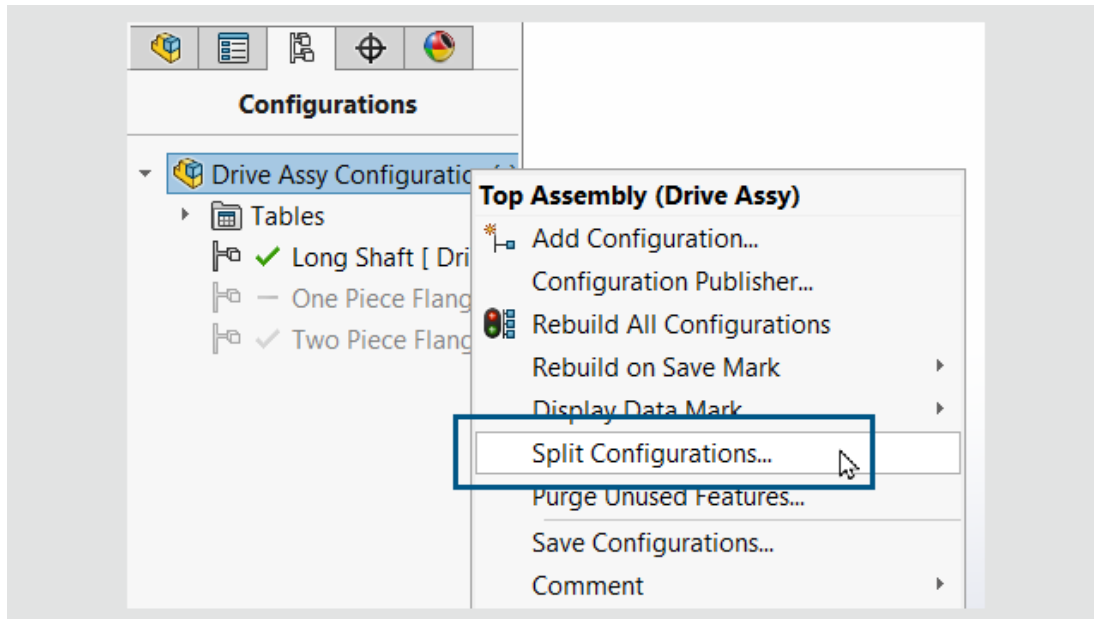
Area	Improvement
Suppress and Fixed columns	When you double-click a component in the graphics area to add it to the configuration

Area	Improvement
	table, the Suppress and Fixed columns get added only when required.
Removing columns	<p>To remove a column, you can right-click the column header and select Delete.</p> 
Hiding and showing columns	<p>You can use a drop-down list to hide or show the Fixed column.</p> 

Display Table Improvements

Area	Improvement
Column width	Column width is optimized and reduced. Long text is wrapped.
Table display	Table flickering is eliminated.
Parameters column	You can resize the Parameters column.
Resizing	When you resize a display state column, the resizing sticks.
Check boxes	Reaction to selecting and clearing of check boxes is improved.
Minimum overall size	The tables and columns have a minimum size that shows all content.
Managing rows	You can add or remove a reference display state row by selecting it from the drop-down list.

Splitting Out Configurations Into Individual Files




In parts or assemblies that have multiple configurations, you can split all the configurations out into individual part or assembly files and save those individual files. You can update the where-used references after the split.

Benefits: This functionality offers you a different way to work with configurations of models.

The **Split Configurations** command is not available for virtual components.

To split out configurations into individual files:

1. Open a part or assembly that has multiple configurations.
2. In the ConfigurationManager , right-click the file name at the top of the tree or any configuration in the tree and click **Split Configurations**.

The Split Configurations as New Files dialog box opens.

3. You can specify these options:
 - **Update where used.** Updates all the references of the original part in the assemblies or drawings to newly created single physical product files in open and out-of-memory files. If you clear this option, the **Where used** table is unavailable.
 - **Where used.** Displays all open in-memory assemblies or drawings of the part or assembly being configured. If you select a check box, the software updates the references of the selected assemblies and drawings with the new split-out parts.
 - **File Locations.** Opens the File Locations dialog box. You can update the **Where used** references of open and out-of-memory assembly and drawing files when the configurations are split.
 - **Update for 3DEXPERIENCE Compatibility.** Creates single physical product files.
4. Click **Save**.

All the configurations are split out into separate files that are in the same location as the original file. The file names are <original file name>.<configuration name>.SLDPRT or SLDASM. For example, for a part named BasePart.SLDPRT that has a split configuration named LongHandle, the split configuration name is BasePart.LongHandle.SLDPRT.

12

Import/Export

Face and Edge Identifiers During Import

When you import certain source CAD files into SOLIDWORKS®, SOLIDWORKS strives to keep the face and edge identifiers stable during the entire current full release cycle.

Benefits: Stable identifiers ensure that downstream SOLIDWORKS features remain valid if you reimport the source CAD file in a newer version.

This information applies to source CAD files from these formats that you import into SOLIDWORKS:

- Autodesk® Inventor®
- CATIA® V5
- PTC Creo®
- SLDXML
- Solid Edge®
- Unigraphics®/NX™

During import, SOLIDWORKS imports the face and edge identifiers (IDs) from the source CAD file. When you then create SOLIDWORKS features in the imported file, SOLIDWORKS uses these IDs as references.

If you update the source CAD file to a newer version that you reimport into SOLIDWORKS, SOLIDWORKS reimports these IDs, keeping their original values. SOLIDWORKS regenerates all downstream SOLIDWORKS features and uses these IDs to match the changed source geometry.

These IDs are not guaranteed to be persistent forever. The source CAD system may change the way it saves these IDs or the algorithm used to convert these IDs during import into SOLIDWORKS may change to improve performance.

SOLIDWORKS strives to keep these IDs stable through all service packs of the current major release, for example, SOLIDWORKS 2026. SOLIDWORKS would only allow IDs to change in a service pack of a major release if the stable IDs might cause regeneration issues for features created using imported geometry when the next major release is available.

13

SOLIDWORKS PDM

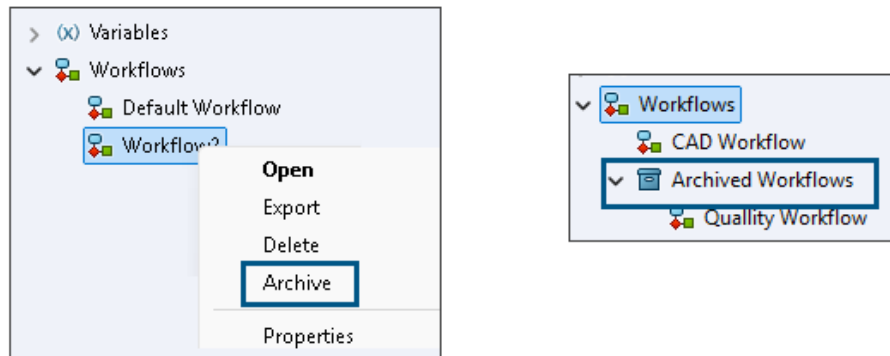
This chapter includes the following topics:

- **Archive Workflows**
- **Lower Level Folder Access**
- **File Version Upgrade Tool**
- **Disabling Custom Triggers before Database Upgrade**
- **Named BOM and File Details in the Web2 Client**
- **Data Encryption Standard**
- **Support for the Kerberos Windows Authentication Protocol**
- **Convert Task Options**
- **Automatic Synchronization of Vault Views**

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

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

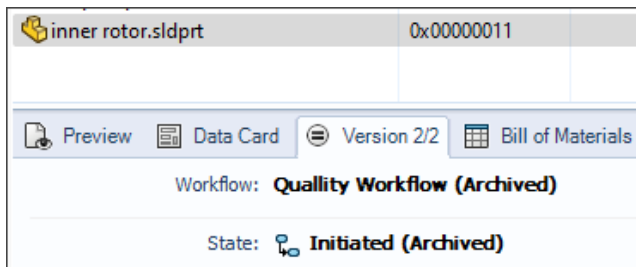
Archive Workflows



In the SOLIDWORKS PDM Administration tool, you can archive a workflow. You cannot transition or rollback the file to another state within the archived workflow.

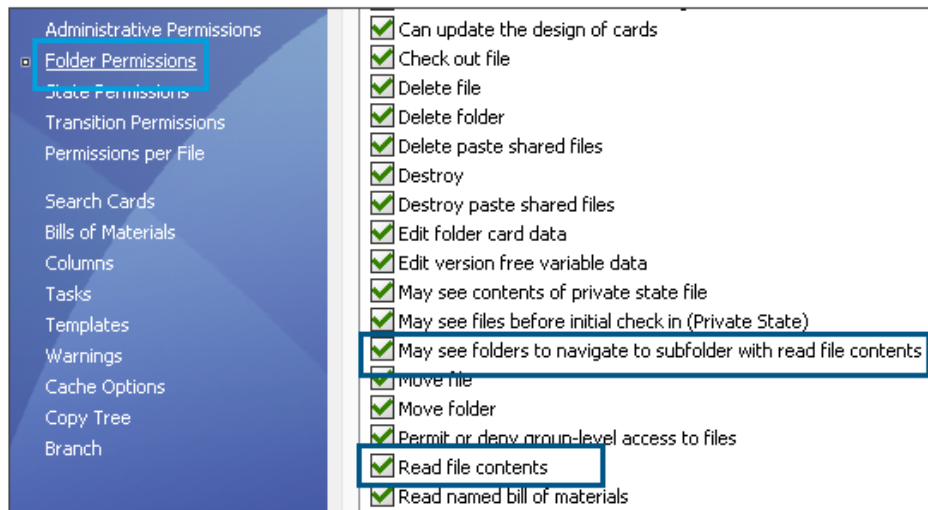
To archive a workflow, right-click a workflow and select **Archive**. The workflow then moves under the **Archived Workflows** sub folder.

You can also see the **Archived** tag in the version tab of the file on the explorer view.



You can unarchive the workflow later whenever required.

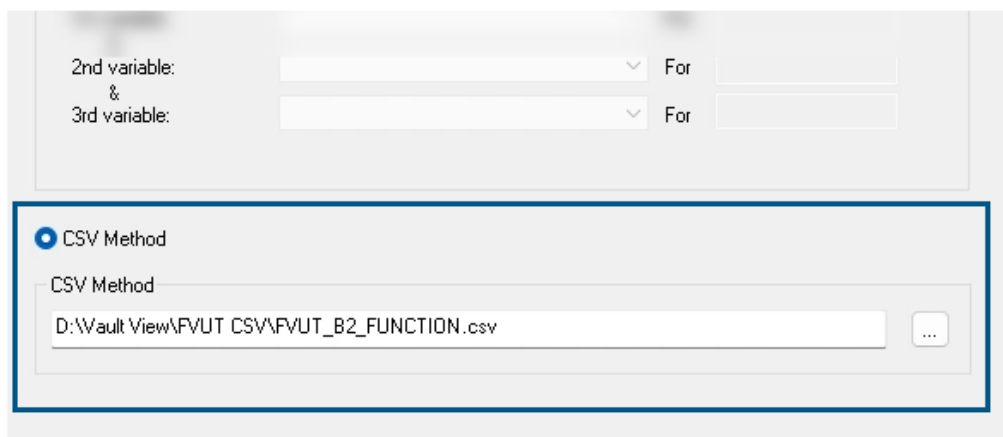
Lower Level Folder Access



If you have **Read file contents** access to a lower-level folder, you can browse the hierarchy of folders from a higher-level folder to a lower-level folder.

You can do this even if you do not have **Read file contents** access on the parent folders of the hierarchy. In the **Admin - Properties** under **Folder Permissions**, you need to select **May see folders to navigate to subfolder** for this functionality.

File Version Upgrade Tool



The SOLIDWORKS PDM File Version Upgrade Tool has the following enhancements to improve your productivity:

- You can specify a **CSV** file under **Search Files To Upgrade** in the SOLIDWORKS PDM File Version Upgrade tool. The **.csv** file has two columns, the **Document ID**, and the **Folder ID**. The **CSV** file is useful to narrow down the search results by letting you mention specific files to upgrade instead of clearing the check boxes in the later **Search Results** page.
- You can select **Latest version with revisions** under **Version Settings > Overwrite existing versions of files > Overwrite**.

You can select either **Latest version with revisions** or **Version with a revision**.

Overwrite existing versions of files

☐ Overwrite all versions

☒ Overwrite

☒ Latest version

☐ Version with a revision

☒ Latest version with revisions

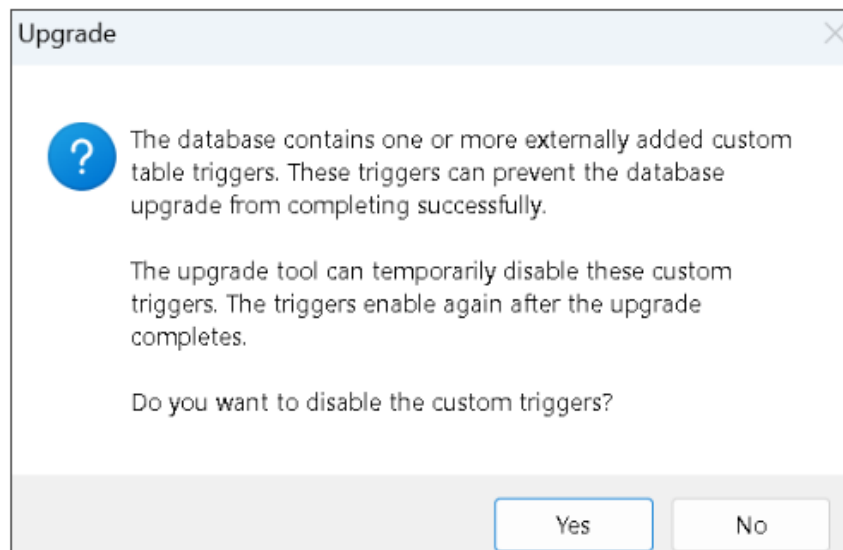
☐ Skip check for broken references

With this option, you get a version attached with the latest revision. For example, if the following are the revisions and versions for a file, this option picks version 5 of the file to overwrite.

Version	Revision
1	
2	
3	A
4	
5	B
6	

- In the logging feature, you can:
 - Add a prefix and suffix to the log file name.
 - Get details of the error in the logs if the upgrade fails.
 - Get details if the tool skips a file during upgrade.

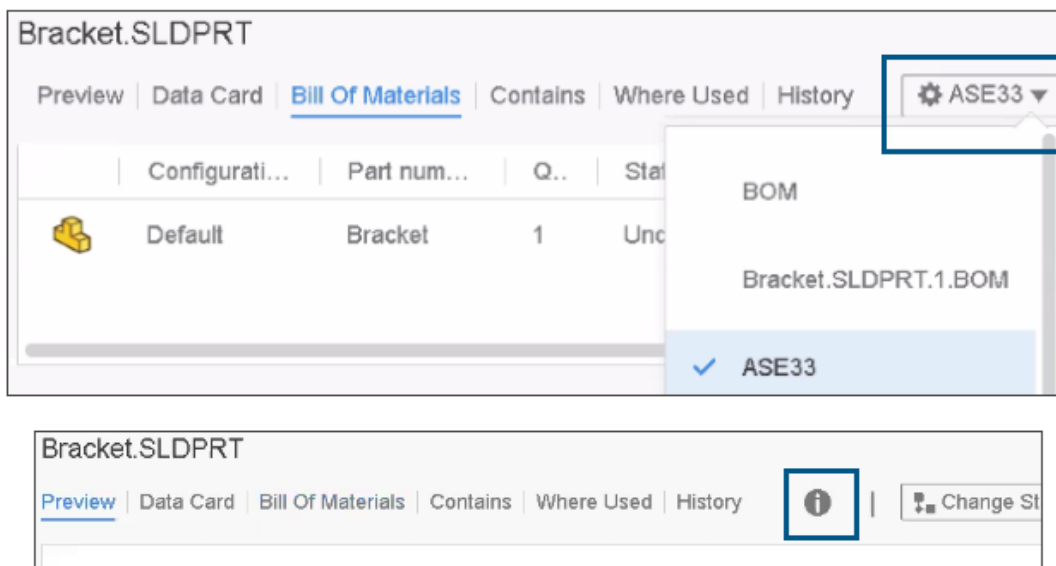
Disabling Custom Triggers before Database Upgrade



You can disable the custom triggers before starting an upgrade to the SOLIDWORKS PDM database.


The custom triggers slow down the upgrade process, so disabling them makes the upgrade faster.

Named BOM and File Details in the Web2 Client



In the SOLIDWORKS PDM Web2 client, some tabs have been updated for you to have better visibility of the information.

You can:

- View the Named BOM under BOM types in the Bill Of Materials tab for the file.
- Click  under either Preview or Data Card to display the file information.

Data Encryption Standard

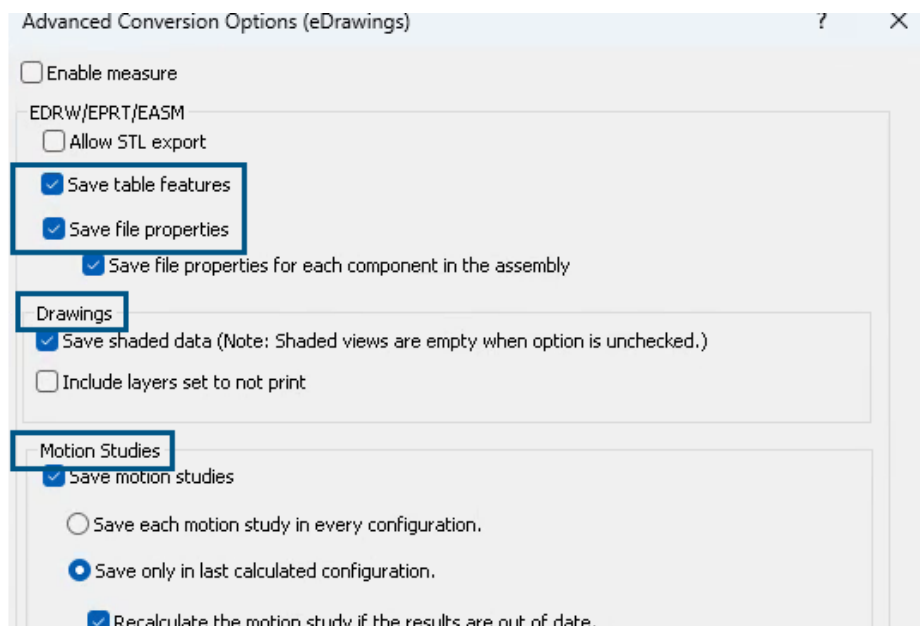
The Advanced Encryption Standard (AES) for data transfer between the archive server and the client has been upgraded from AES-128 to AES-256. This makes the data transfer more secure.

Support for the Kerberos Windows Authentication Protocol

You can use the Kerberos Windows Authentication protocol when logging into the SOLIDWORKS PDM vault using Windows authentication.

You can use Kerberos when NTLM is disabled in the domain.

Convert Task Options

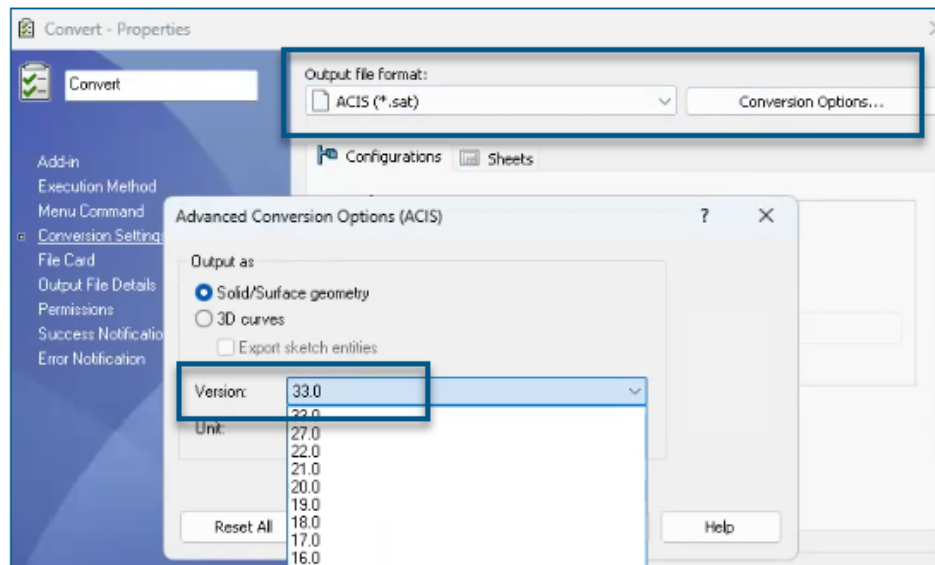


The SOLIDWORKS PDM Administration tool includes enhancements for the convert task options for Parasolid™, ACIS®, and eDrawings® file format.

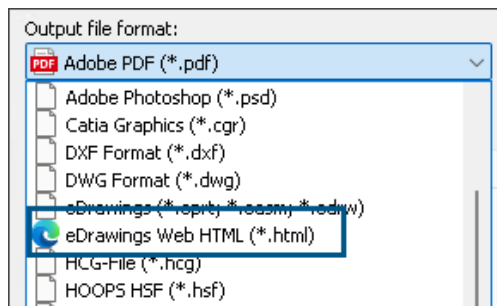
The enhancements are:

- A modified user interface for the eDrawings file format similar to the SOLIDWORKS export **System options** for better clarity and usability. For example, the existing options are grouped under sections and the following options are added:

- Save table features
- Save file properties

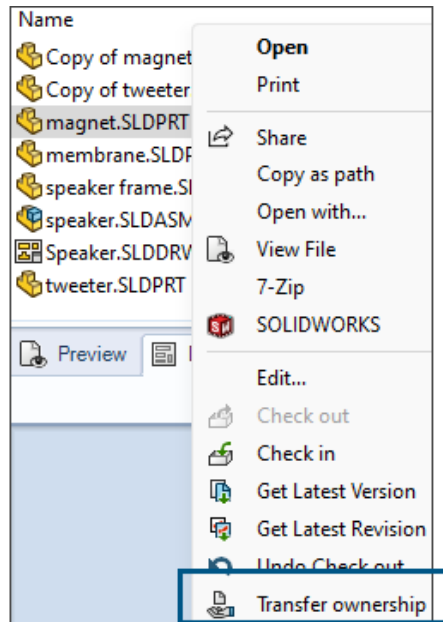


- Support of higher versions for Parasolid (up to 35.1) and ACIS (up to 33.0) file formats.



- A new **eDrawings Web HTML (*.html)** option under **Output file format**. The user interface of the **Advanced Conversion Options** dialog box for this new option is similar to eDrawings options (All options are unmodifiable except **Enable measure**).
- Ability to change the output path or file name in the **Draftsight to PDF** and **Office to PDF** tasks using the **Advanced Scripting Options**.

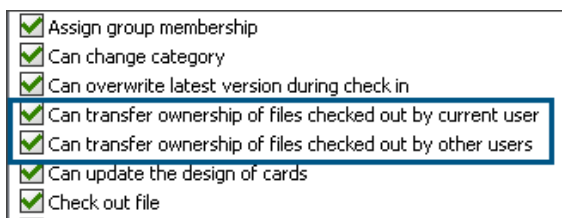
Automatic Synchronization of Vault Views



SOLIDWORKS PDM automatically synchronizes modified checked out files to the archive server where the local view is connected. This lets different users work on these files on different computers without checking in the files to the SOLIDWORKS PDM vault by taking up the ownership using **Transfer Ownership**.

For example, if you have checked out a file and modified and saved it on one computer, you can sign in to a second computer, transfer ownership of the file to the second computer, and start working with the file.

In the same way, another user can take ownership of the file on the same or a different computer and can further view, modify, or check in the file.



To use this functionality, you need the following folder and state permissions:

- **Can transfer ownership of files checked out by current user**
- **Can transfer ownership of files checked out by other users**

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SOLIDWORKS Manage

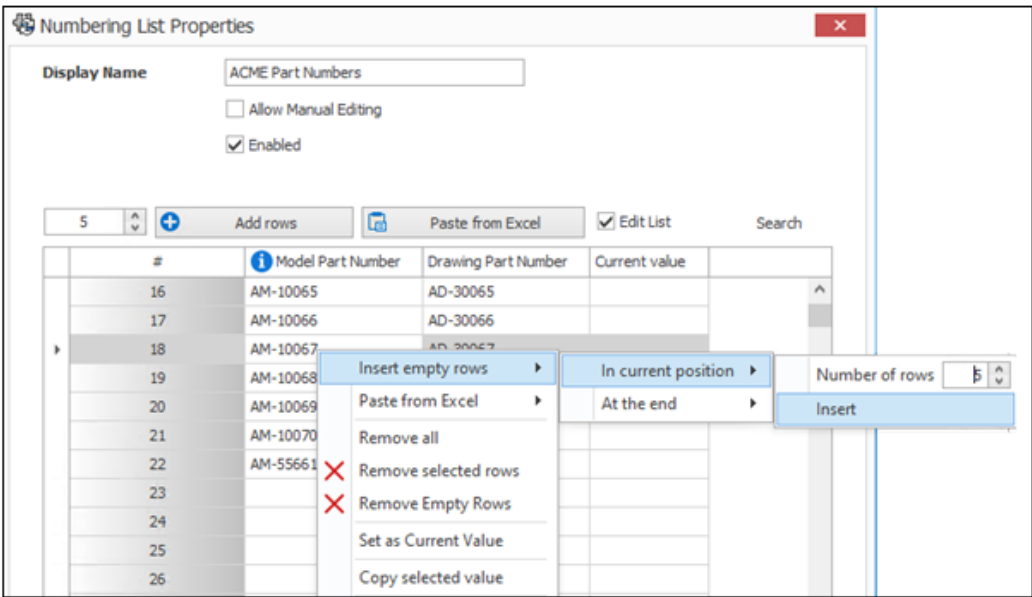
This chapter includes the following topics:

- **Numbering Lists**
- **Previewing Related Files**
- **Accessing Timesheets by Targeted Web Client**
- **Providing access of Root Objects to Users or Groups**
- **Excluding New Users from Groups**
- **Securing Database Updates with an SQL Password**
- **Set the End Date for a Task**
- **Including On Hold Tasks**
- **Viewing Task Details from the Capacity Planning Tool**
- **Reports Module in the Plenary Web Client**
- **Creating Links to the Desktop Client**
- **Children Only Flat BOM**
- **Defining a User Access Condition**
- **Processing Output Conditions**
- **Messaging API Event Triggers**

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.

Numbering Lists



Numbering Lists lets you assign part numbers to SOLIDWORKS files.

Administrators can add a series of numbers to a list. When the users save a new file into a document object, instead of using the **Numbering Scheme**, they can use the new number in the list.

It is easier to integrate parts numbers:

- That you receive from a third party.
- For models and drawings that do not follow sequential numbering.

Numbering List Properties Dialog Box

To open the Numbering List Properties dialog box:

1. In the System Administration tool, under **Advanced**, select **Numbering List**.
2. Click **New**.
3. Specify options below.

Options inside the Dialog Box

Option	Description
Display Name	Displays the name of the numbering list.
Allow Manual Editing	Lets you edit the part number manually before a new record is saved.
Enabled	Assigns the numbering list to a document object.

Option	Description
Add Rows	Adds the specified number of rows at the bottom.
Paste from Excel	Pastes the copied text from the Microsoft® Excel spreadsheet to the list.
Edit List	Edits an existing value in the list.
Model Part Number	Numbers to assign to parts and assemblies. <div>If a value in the Model Part Number column is empty, SOLIDWORKS Manage removes the entire row when you save.</div>
Drawing Part Number	Numbers to assign to drawing files.
Current Value	Yes indicates the row from which, the software assigns the next numbers.
Enter empty rows	Interleaves empty rows in between the existing rows.
Set as Current	Defines the selected row as the current row.

Shortcut Menus

Option	Description
Remove all	Clears the entire numbering list.
Remove selected rows	Removes selected rows from the list.
Remove Empty rows	Removes rows that do not have a model or a drawing value.
Set as Current Value	Specifies the selected row as the current value.
Copy selected value	Copies the selected value to the clipboard.
Export to Excel	Creates a new Microsoft® Excel file with the selected data.
Reload All	Removes all the changes since the last save.

Defining a Numbering List

To define a numbering list:

1. In the System Administration tool, under **Advanced**, select **Numbering List**.
2. Click **New**.
3. In the Numbering List Properties dialog box, enter a **Display Name** for the new list.
4. Select **Allow Manual Editing** to overwrite the automatic value on the property card.
5. Click and edit the part number.
6. Optional: Clear **Enabled** when you do not want to use the list for a document object. You can add the data manually or from Microsoft® Excel spreadsheet.

If the **Model Part Number** column is empty, SOLIDWORKS Manage removes the entire row when you save. If the **Drawing Part Number** column is empty, the software maintains the row.

Adding Data to the List

You can enter data manually, for example by copying it from a Microsoft Excel spreadsheet or another source.

You can add the data only from the first two columns of the Excel spreadsheet.

To add data to the list:

1. Enter the number of rows.
2. Click **Add rows**.
3. Enter values for **Model Part Number** and **Drawing Part Number**.
4. Optional: To enter blank rows:
 - a) Right-click the cell and select **Insert empty rows > In current position**.
 - b) In **Number of rows**, enter a number.
 - c) Select **Insert**.
5. Click **Save**.
6. Optional: Copy the data and click **Paste from Excel** to add the data at the bottom of the list from an Excel spreadsheet.
7. Optional: Click a row, right-click, and select **Paste from Excel > In current position** to add the copied values in the middle of existing rows.

Using a Numbering List in a Document Object

To use a numbering list in a document object:

1. In the System Administration tool, select Options.
2. In **CAD Options**, click **SOLIDWORKS Options**.
3. In the Main Options (SOLIDWORKS) dialog box, click **Part Number options**.
4. In the Part Number Options dialog box, in **Field Group**, select the field group that uses a numbering list.
5. Select **Numbering List**.

6. In **Numbering Scheme Versions**, for each file type, select **Numbering List**.

Parts and assemblies use the numbers from the **Model Number** column.

7. Optional: Include a prefix or suffix to add to the number.
8. Click **Save** and close the dialog box.

Linked Models and Drawings

The way numbers are assigned from the list depends on the **Part Number Options** specified for linking models and drawings.

The assignment of numbers from the list also depends on whether:

- A model and a drawing can have the same part number.
- The list has a model and a drawing number in the current value row.

The following tables describe the different scenarios. If the list does not have numbers, SOLIDWORKS Manage uses the default numbering scheme assigned to the object.

Table 1: Linked Model and Drawings That Cannot Have the Same Part Number: List Contains Both Numbers

Model first, then drawing	Drawing first, then model
The model gets the current value Model Part Number , then the drawing gets the Drawing Part Number from the same row.	The drawing gets the current value Model Part Number , then the model gets the next Model Part Number in the list.

Table 2: Linked Model and Drawings That Cannot Have the Same Part Number: List Contains Only The Model

Model first, then drawing	Drawing first, then model
The model gets the number from the row which has the current value, then the drawing gets the next number in the list.	The drawing gets the current value Model Part Number , then the model gets the next Model Part Number in the list.

Table 3: Linked Model and Drawings That Can Have the Same Part Number: The List Contains Both Numbers

Model first, then drawing	Drawing first, then model
The model gets the current value Model Part Number , then the drawing gets the Drawing Part Number from the same row.	The drawing gets the current value Model Part Number , then the model gets the next Model Part Number in the list.

Table 4: Linked Model and Drawings That Can Have the Same Part Number: The List Contains Only The Model

Model first, then drawing	Drawing first, then model
The model gets the number from the row which has the current value, then the drawing gets the next number in the list.	The drawing gets the current value Model Part Number , then the model gets the next Model Part Number in the list.

Table 5: Model and the Drawing Are Not Linked: The List Contains Both Numbers

Model first, then drawing	Drawing first, then model
The model gets the current value Model Part Number , then the drawing gets the next Model Part Number in the list.	The drawing gets the current value Model Part Number , then the model gets the next Model Part Number in the list.

Table 6: Model and Drawing Are Not Linked: The List Contains Only The Model

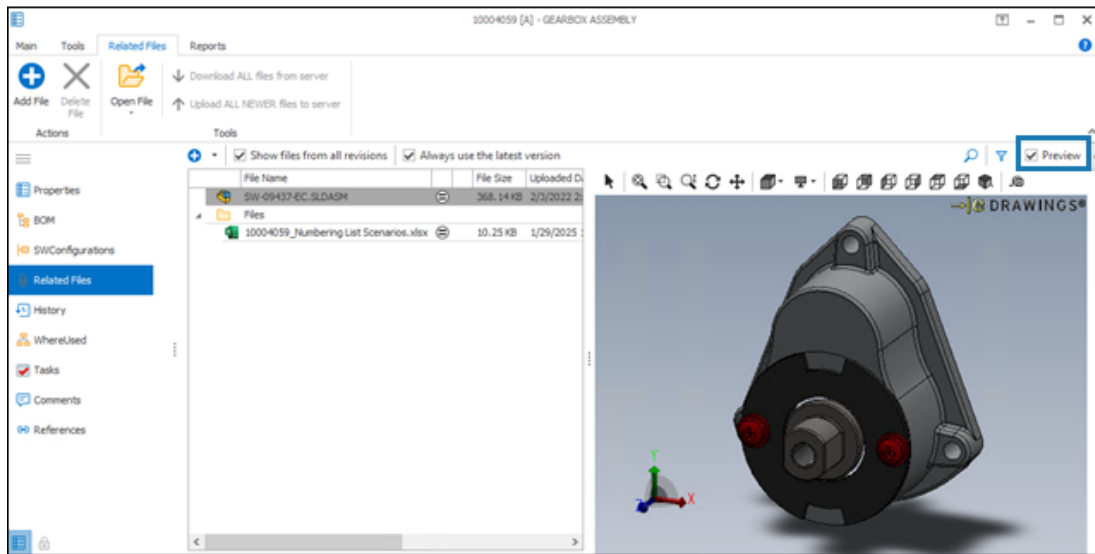
Model first, then drawing	Drawing first, then model
The model gets the current value Model Part Number , then the drawing gets the next Model Part Number in the list.	The drawing gets the current value Model Part Number , then drawings get the next Model Part Number in the list.

Applying a Number to a SOLIDWORKS File

To apply a number to a SOLIDWORKS file:

1. In SOLIDWORKS, open the SOLIDWORKS Manage add-in.
2. Create a new model or drawing.
3. In the SOLIDWORKS Manage add-in, click **Save As**.
4. In the dialog box, select a group that has Numbering List selected in **Type**.
5. Enter the required fields and click **Save**.

Previewing Related Files



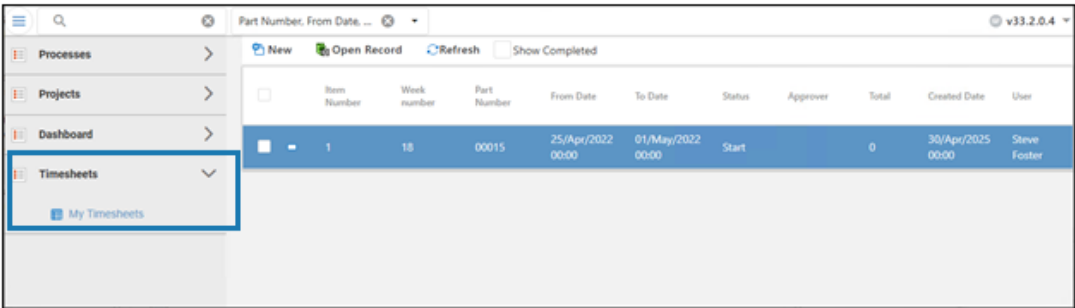
You can preview related files that include primary files for document and PDM objects in the Related Files tab.

Previously, you could preview documents and PDM objects only in the right-side fly-out pane on the main grid.

To preview related files:

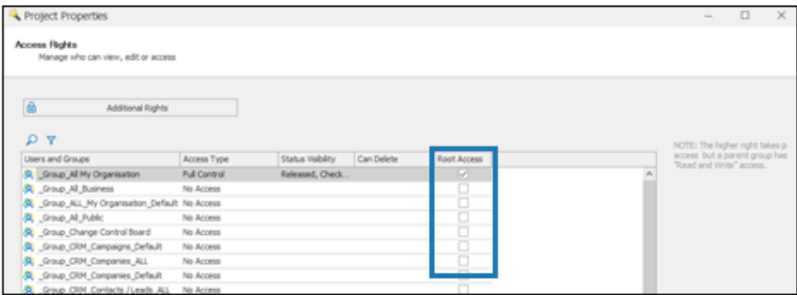
1. Click Related Files in the ribbon.
2. Select **Preview**.

Accessing Timesheets by Targeted Web Client



You can access timesheets by the targeted web client. This lets external users submit their work without having full user access.

Providing access of Root Objects to Users or Groups

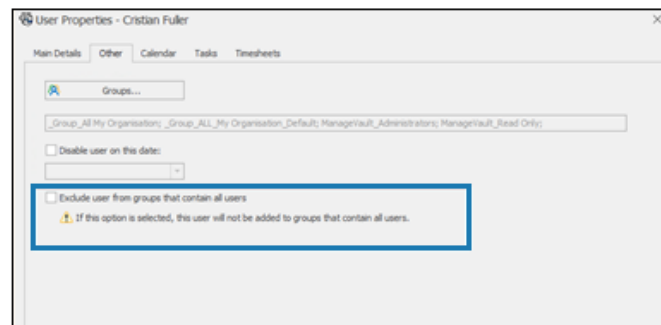


You can grant access to an object's root location for specific users and groups. This level of access lets users and groups view only the records within the subfolders.

To provide access of root objects to users or groups:

1. In the Project Properties dialog box, select **Access Rights**.
2. Under **Root Access**, select users or groups.

Excluding New Users from Groups



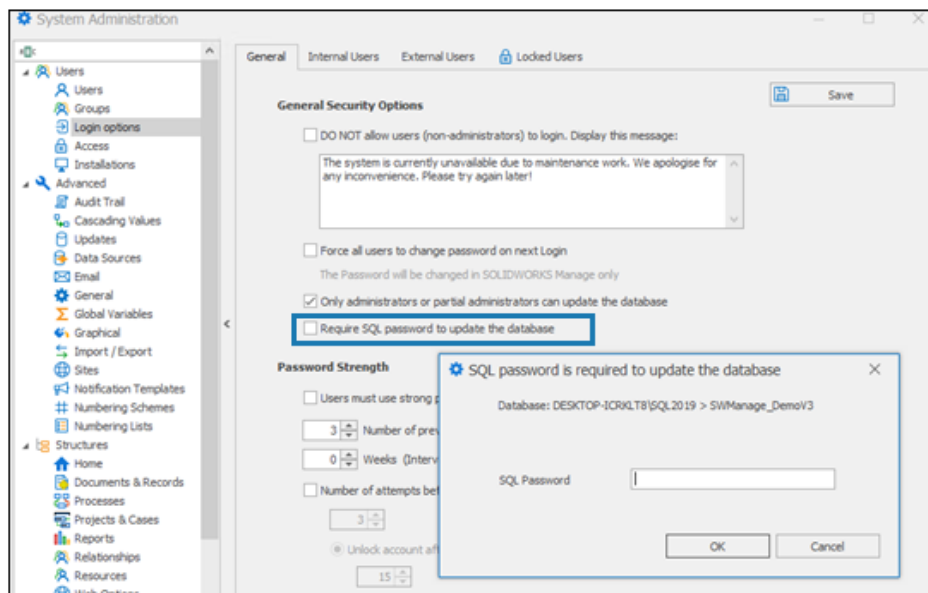
You can exclude new Manage-only users from groups that automatically include all users.

Previously, you had to manually remove new users from such groups. For example, the system-defined group **_Group_All My Organisation** includes all users automatically.

To exclude new users from groups:

1. In the User Properties dialog box, on the Other tab, select **Exclude user from groups that contain all users**.
2. Click **Save**.

Securing Database Updates with an SQL Password

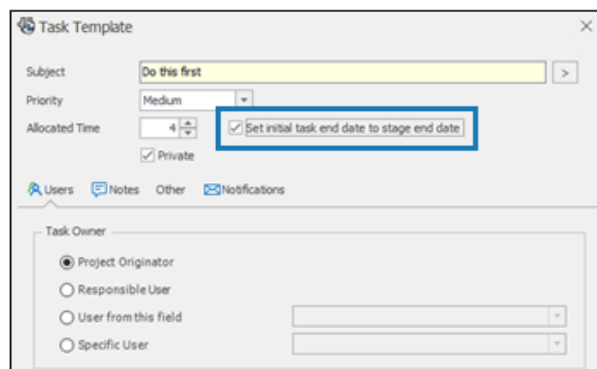


You can secure database updates with an SQL password.

To secure database updates with an SQL password:

1. In the System Administration tool, click **Users > Login Options**.
2. On the General tab, select **Require SQL password to update the database**.
3. Enter the SQL password, then click **OK**.

Set the End Date for a Task



You can specify that the end date of a task is the same as end date of the stage.
 Previously, the end date of the task was based on the duration allocated to each task.
 In the Task Template dialog box, select **Set initial task end date to stage end date**.

Including On Hold Tasks

Capacity Planning

Year: 2022 | Include: Holidays, Absences, Public holidays | Units: Hours

Capacity: Users | 100 | Efficiency (%) | Group By: Months | ☒ Auto Width | ☒ Include "On Hold" tasks in Assigned Tasks

Resource	1	2	3	4	5	6	7	8	9	10	11	12	Total
Larry Jones	168.0	168.0	168.0	168.0	168.0	176.0	168.0	168.0	176.0	168.0	176.0	176.0	2,072.0
Marcel Hand	168.0	168.0	168.0	168.0	176.0	176.0	168.0	168.0	176.0	168.0	176.0	176.0	2,080.0
Mark Murphy	168.0	168.0	168.0	168.0	176.0	176.0	168.0	168.0	176.0	168.0	176.0	176.0	2,080.0
Michael Steel	160.0	160.0	164.0	128.0	176.0	176.0	168.0	168.0	176.0	168.0	176.0	176.0	2,032.0
Michael Coulter	168.0	168.0	168.0	168.0	176.0	176.0	168.0	168.0	176.0	168.0	176.0	176.0	2,080.0
Mike Spender	168.0	168.0	168.0	168.0	176.0	176.0	168.0	168.0	176.0	168.0	176.0	176.0	2,080.0
Paul Anderson	168.0	168.0	168.0	168.0	176.0	176.0	168.0	168.0	176.0	168.0	176.0	176.0	2,080.0
Sparky DeNila	168.0	168.0	168.0	168.0	176.0	176.0	168.0	168.0	176.0	168.0	176.0	176.0	2,080.0
Steve Foster	168.0	168.0	168.0	140.0	152.0	176.0	168.0	168.0	176.0	168.0	176.0	176.0	2,036.0
Sylvain Trudeau	168.0	168.0	168.0	168.0	176.0	176.0	168.0	168.0	176.0	168.0	176.0	176.0	2,080.0
System Administrator	168.0	136.0	168.0	96.0	176.0	152.0	144.0	168.0	-43.0	62.0	-128.0	168.0	1,256.0

Demand

Projects | ☐ Based on tasks | ☒ Based on stages | ☒ Show zero values | ☒ Ignore completed work | ☒ Auto Width | ☒ Highlight values

Project	1	2	3	4	5	6	7	8	9	10	11	12	Total
Total	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
PRJ-00503	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
PRJ-00479	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
PRJ-00477	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
PRJ-00003	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
PROD-00005	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
PROD-00006	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

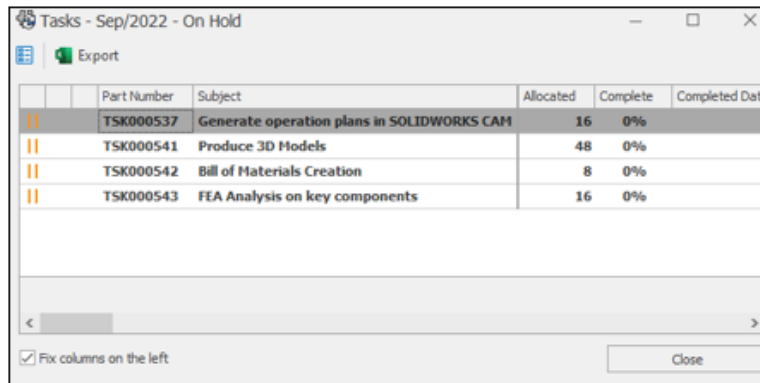
You can identify tasks with the status **On Hold**. This status indicates that you cannot work on the task yet.

To include On Hold tasks:

1. In the **Capacity Planning** tool, select **Include "On Hold" tasks in Assigned Tasks**.

This helps a team evaluate if they can take on the project without exceeding capacity. If too many tasks are **On Hold**, adding the new project may overload the team.

Viewing Task Details from the Capacity Planning Tool

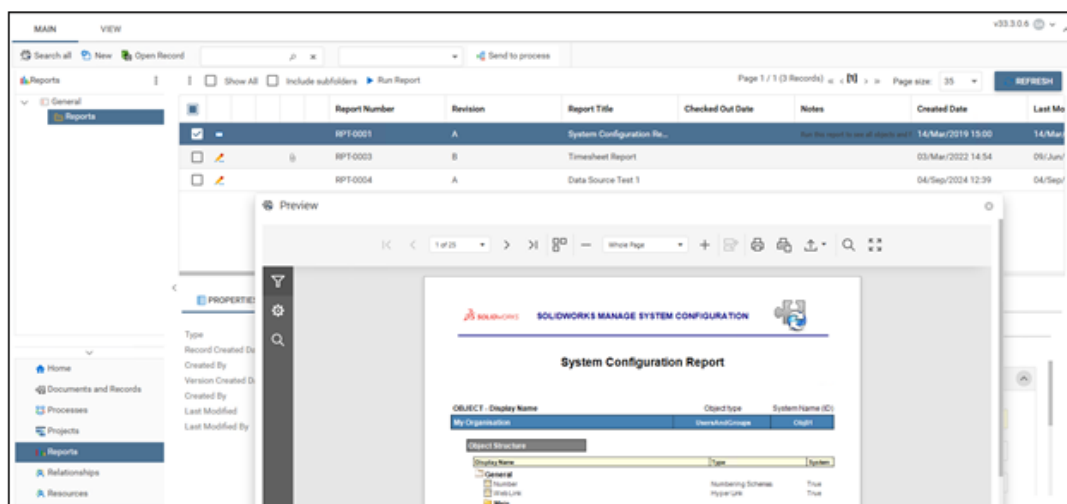


	Part Number	Subject	Allocated	Complete	Completed Date
	TSK000537	Generate operation plans in SOLIDWORKS CAM	16	0%	
	TSK000541	Produce 3D Models	48	0%	
	TSK000542	Bill of Materials Creation	8	0%	
	TSK000543	FEA Analysis on key components	16	0%	

☒ Fix columns on the left Close

You can access the individual tasks from the **Capacity Planning** tool.
Project managers can see the details of the tasks assigned to each user.

Reports Module in the Plenary Web Client



The screenshot displays the 'Reports' module in the Plenary Web Client. The main area shows a list of reports with columns: Report Number, Revision, Report Title, Checked Out Date, Notes, Created Date, and Last Modified. A preview window is open, showing the 'System Configuration Report' for 'My Organization'. The report includes a table with columns: Object Name, Object Type, System Name (ID), and System. The table lists 'General' (Numbering Scheme: True), 'Assembly' (Hyper Link: True), and 'Main' (Hyper Link: True).

You can access **Reports** in the plenary (full) web client.
This lets you access and run consolidated reports from a web browser.

Use a desktop client to edit the reports.

Creating Links to the Desktop Client

You can create links for records on the desktop (thick client). You can include them in notification emails and display them on dashboards.

Previously, you could create links to the web client only.

Children Only Flat BOM

The **Flat BOM (Children Only)** view displays only the rolled-up child items. Previously, the **Flat BOM** view displayed rolled-up parents (for example, subassemblies) and children (for example, parts).

Flat BOM (Children Only) is similar to **Parts Only BOM** in SOLIDWORKS PDM.

Defining a User Access Condition

You can define the users or groups that can work on a particular process stage using conditions. This simplifies configuration by eliminating the need to have conditional workflow paths to provide different access rights.

Processing Output Conditions

Output conditions can use **Affected Items** fields in addition to process fields. This allows you to run outputs on specific affected items.

Messaging API Event Triggers

The API includes event triggers that lets you send messages to a queuing application. It provides a more robust method to send and receive changes from an external system like an ERP system.

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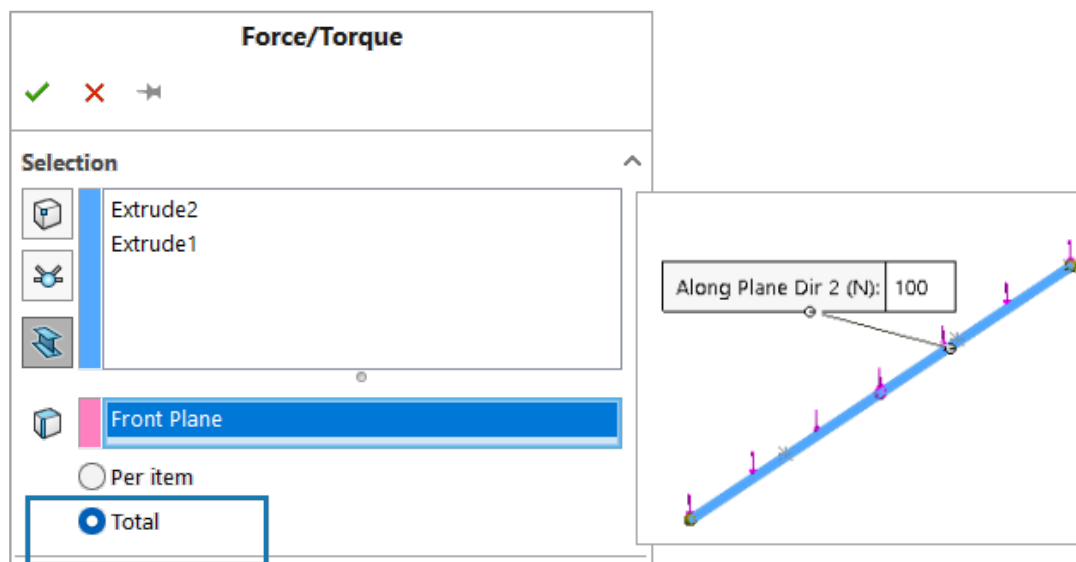
SOLIDWORKS Simulation

This chapter includes the following topics:

- **Applied Forces on Beams**
- **Buckling Studies**
- **Display of Angular Deformations**
- **Distributed Remote Load on Shell Edges**
- **Licensing Updates**
- **Performance Improvement for Studies with Connectors**
- **Pin Connector Forces**
- **Remote Mass Support for Response Spectrum Analysis**
- **Shell Definitions**
- **User Interface**

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

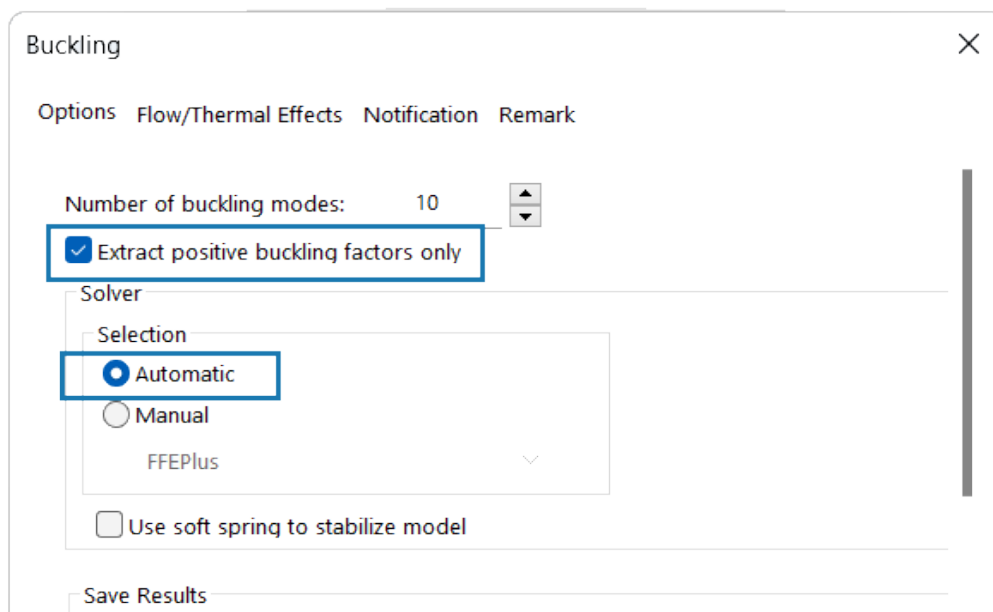
Applied Forces on Beams



You have more flexibility when applying forces on beam bodies.

In earlier releases, you could only apply a force load to beam bodies with the default option **Per item**. In this release, you can select the **Total** option in the Force/Torque PropertyManager to distribute a force load among several beam bodies that is proportional to their lengths.

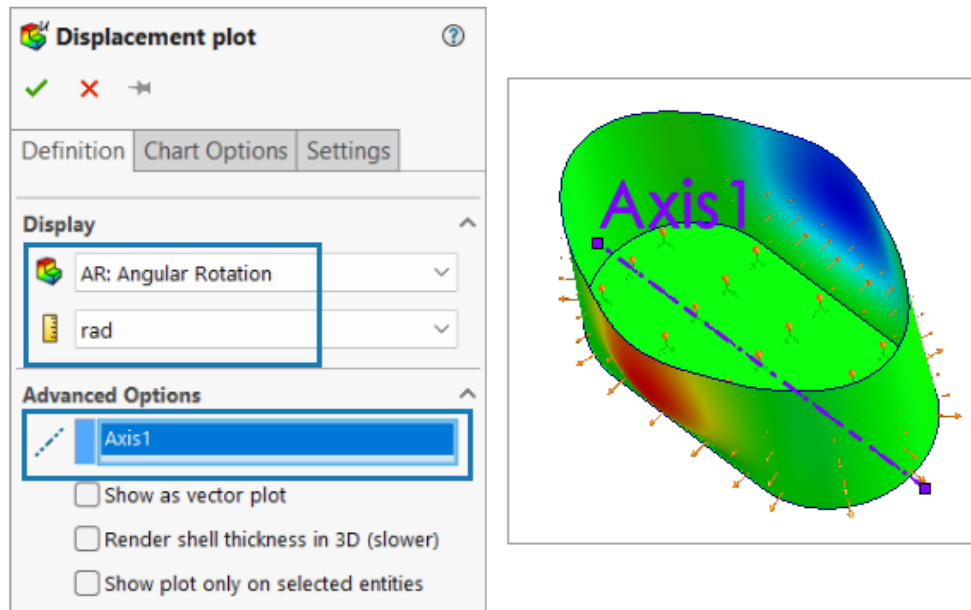
Buckling Studies



You can extract only the positive buckling factors and modes for a model that you use to run a buckling study.

The option **Extract positive buckling factors only** is available from the Buckling Study Properties dialog box. If you choose to extract only the positive buckling factors and modes, the solver switches to the **Automatic** option. SOLIDWORKS Simulation does not report any negative buckling factors and their associated modes that the solver could calculate for the buckling simulation.

Display of Angular Deformations

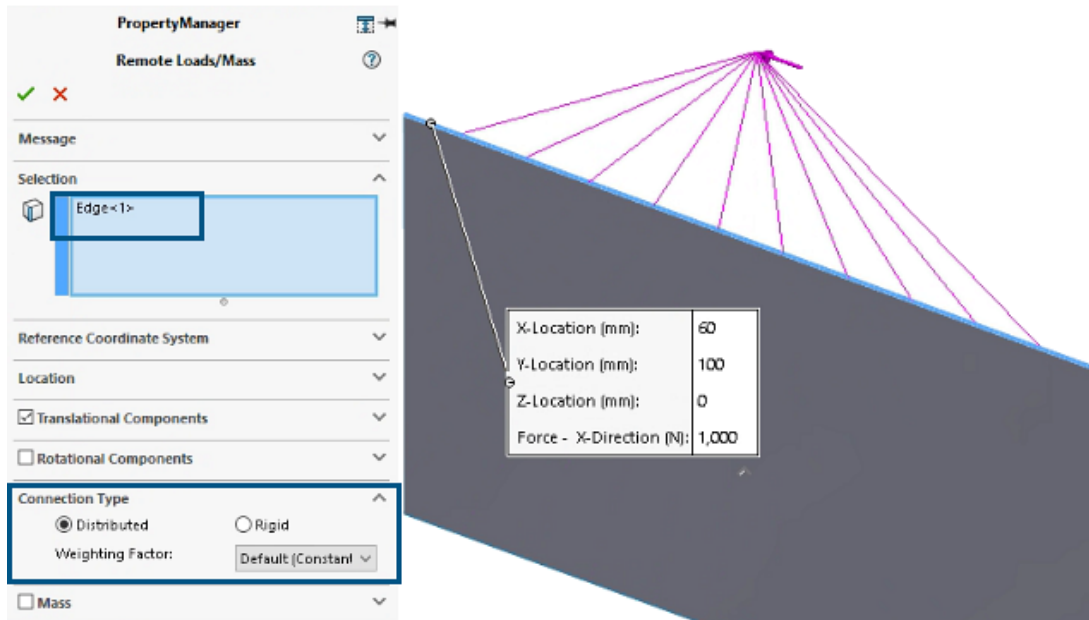


You can plot angular deformation results with respect to a given axis in units of degrees or radians.

In the Displacement plot PropertyManager, select **AR: Angular Rotation** under **Display** and an axis under **Advanced Options**.

This is available for static and nonlinear static studies. Only studies with all solid, shell, or beam meshes support the display of angular rotations. Mixed-mesh studies are not supported. The creation of time-history plots of angular rotations for nonlinear studies is not supported.

Distributed Remote Load on Shell Edges



The distributed coupling formulation for remote load and remote mass now supports shell edges.

When you select a shell edge as support, the remote load or mass is distributed across the edge's coupling nodes. Previously, the distributed coupling formulation was available only for faces.

This is available for linear static studies, along with the associated fatigue, design, and pressure vessel design studies.

Licensing Updates

Functionality that was available only with SOLIDWORKS Simulation Professional and SOLIDWORKS Simulation Premium licenses is now available with SOLIDWORKS Simulation Standard licenses.

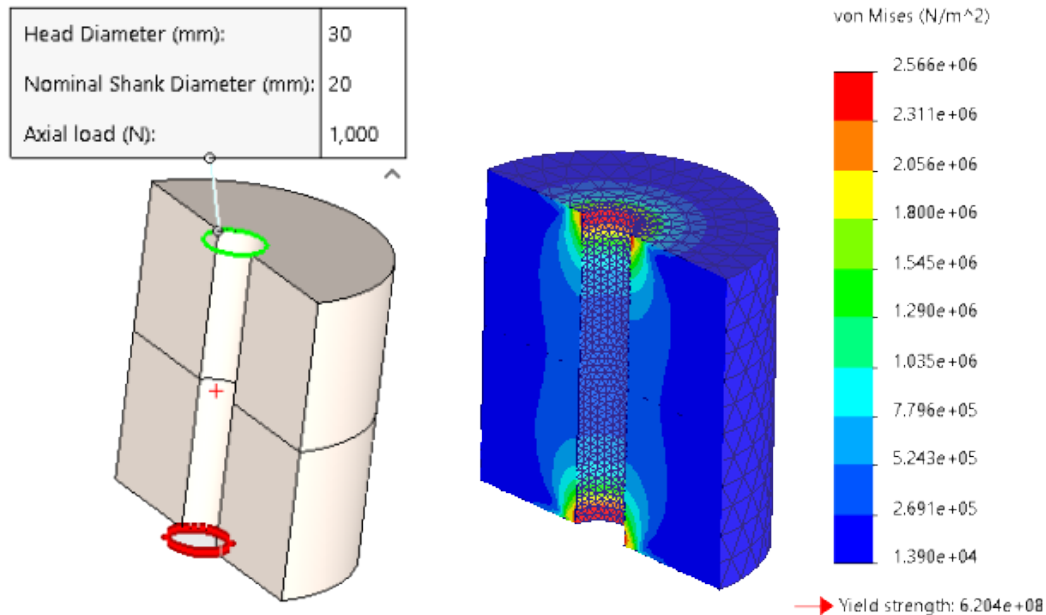
- Automatic detection of underconstrained bodies. The **Automatically detect underconstrained bodies** option that you access from the System Options - General dialog box is available for all SOLIDWORKS Simulation licenses.

When you select **Automatically detect underconstrained bodies**, the solver detects bodies that are not sufficiently constrained during simulation and can exhibit translational or rotational rigid body modes if the model is unstable. The automatic detection of rigid bodies is available for linear static studies.

- Advanced bonding and contact algorithms. Improved memory estimate, allocation, and management by the solver allows the completion of large surface-to-surface bonded and contact interaction sets that previously failed because of insufficient memory.

- Function-based communication. Passing data to the equation solvers FFEPlus and Large Problem Direct Sparse is more efficient for all licenses. Function-based data communication through memory usage replaces file-based communication.

Performance Improvement for Studies with Connectors



The solution time for simulation studies with connectors that support distributed coupling has been improved.

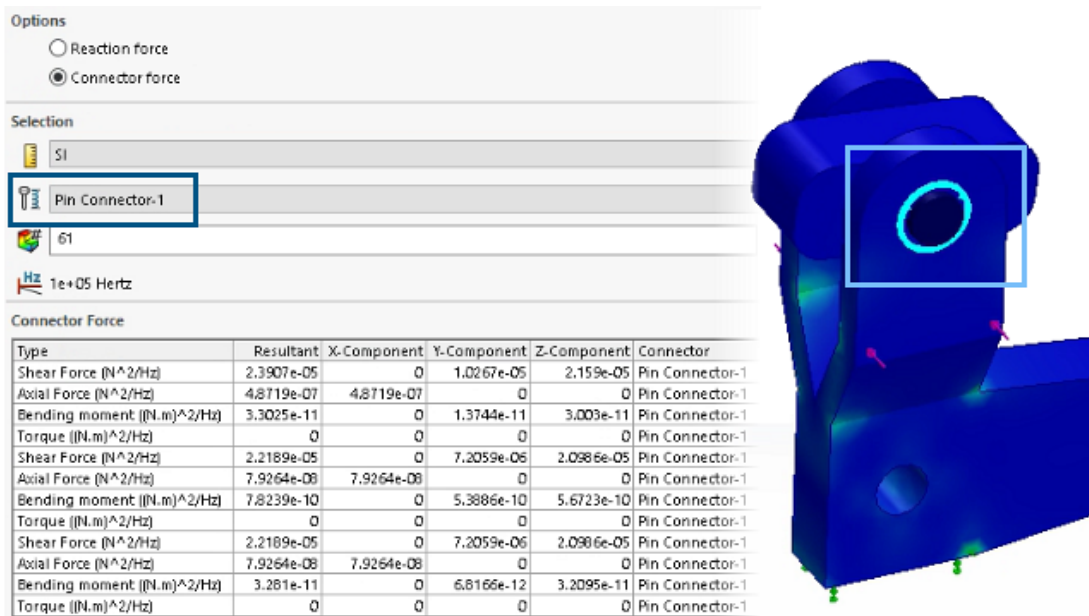
Expected solver enhancements include:

- Intel Direct Sparse solver. Models that previously failed to solve because of limitations on the number of coupling facets (surface elements) exceeding 800 can reach a solution. This restriction is removed. In addition, solution time for models using connectors with distributed coupling and a large number of coupling nodes (for example, bolt, bearing, and linkage rod connectors) is faster.

For example, the image above shows a model of two cylinders that are attached with a bolt with distributed coupling. In earlier releases, the linear static study of this model failed because of the limitation on the number of coupling facets. In this release, the Intel Direct Sparse solver successfully provides a solution for the same study.

- FFEPlus solver. Faster solution time for models using connectors with distributed coupling and a large number of coupling nodes.

Pin Connector Forces



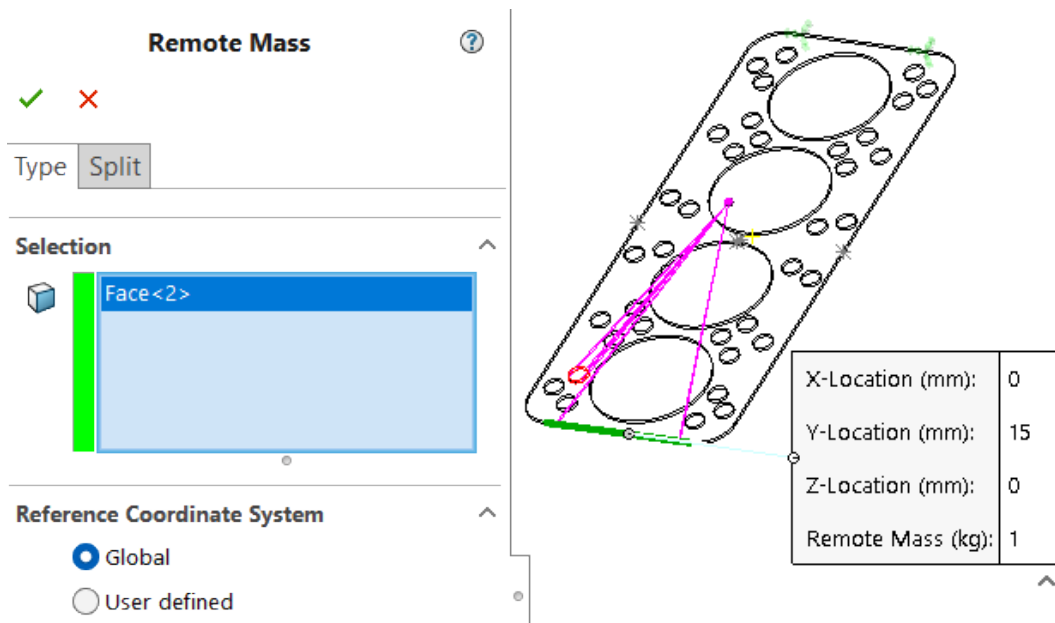
You can extract pin connector forces, including shear force, axial force, bending moment, and torque, in linear dynamic random vibration studies.

In the Result Force PropertyManager, under **Options**, select **Connector force** to view a list of the pin connector forces that the solver calculates. You can list the X-, Y-, and Z- components of the pin connector forces with respect to the global coordinate system or a local coordinate system along with the resultant force. SOLIDWORKS Simulation lists the pin forces and moments based on PSD (Power Spectral Density) values, which represent the force distributions across the frequency domain.

Click **Response Graph** to generate a response graph of the pin forces in the frequency domain.



This lets you evaluate the forces and moments acting on pin connectors during random vibration studies, providing a more accurate representation of load distribution.

Remote Mass Support for Response Spectrum Analysis



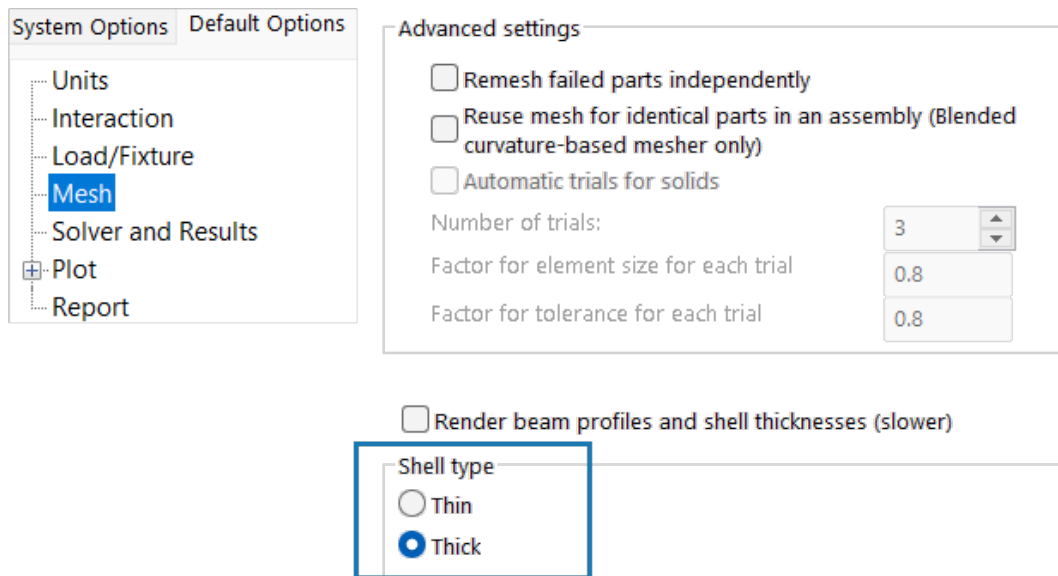
Response spectrum analysis studies support the application of remote masses.

You can include the effect of a component's mass that is not part of the meshed geometry to the rest of the model by treating it as a remote mass. To apply a remote mass to a model for which you want to run a response spectrum analysis:

1. From the response spectrum analysis study tree, right-click **External Loads**, and select **Remote Mass** .
2. Select the faces, edges, or vertices to which you apply the remote mass for **Selection**.
3. Specify the location of the remote mass at the component's center of gravity for **Location**.
4. Enter a value for the **Remote Mass** .

The remote mass is rigidly connected to the rest of the model at the selected faces, edges, or vertices.

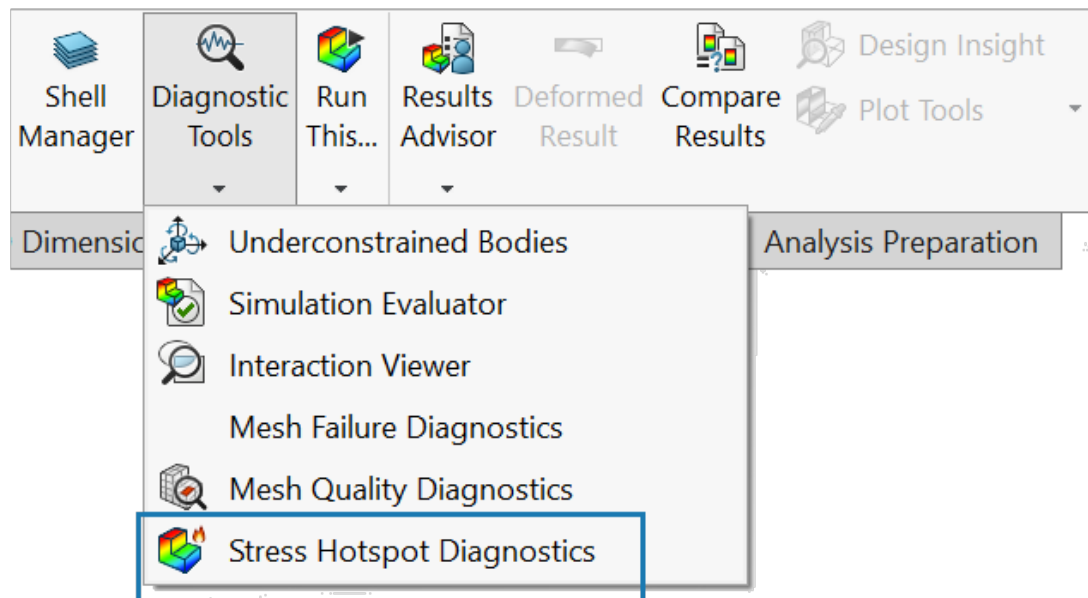
Shell Definitions



You can assign the shell type definition, **Thick** or **Thin** globally, for new studies.

To modify the **Shell type**, from the menu at the top bar, click **Simulation** > **Options...** > **Default Options** > **Mesh**. The specified shell type applies to new shell definitions created from surface bodies, sheet metal bodies, and solid bodies using the **Define shell by selected face** command.

User Interface



Several user interface enhancements improve the user experience.

- You can access the **Stress Hot Spot** tool from the CommandManager under **Diagnostic**

Tools



- In the Report Options dialog box, you can select or clear all report sections with one action before generating a report, thus saving time.
- The wording of error messages is clearer, and it is easier to identify the root cause of errors in simulation studies.

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SOLIDWORKS Visualize

This chapter includes the following topics:

- **Support for AMD Hardware in Stellar Fast Render Mode**
- **DSPBR Support in SOLIDWORKS**

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

Support for AMD Hardware in Stellar Fast Render Mode

System Info

Approved Upgrade for best performance Acceptable configuration Not applicable

GPU: NVIDIA RTX A1000 6GB Laptop GPU

Render Engine: Stellar

Render Mode: Fast

GPU 1: NVIDIA RTX A1000 6GB Laptop GPU

Entry	Current Value	Required Value	Status
Graphics Memory	6 GB	4 GB	Pass
Graphics Driver Vers...	32.0.15.5635	31.0.15.3878	Pass
Compute Capability	8.6	7.5	Pass
Vulkan Version	1.3	1.3	Pass
Vulkan Extensions	Supported	Supported	Pass

SOLIDWORKS Visualize supports native GPU acceleration on AMD hardware (RDNA™ 2 and newer) using the 3DS Stellar **Fast** rendering mode, Visualize's interactive ray tracing engine based on Stellar RealtimeGI.

Previously, if you used SOLIDWORKS Visualize on a computer with AMD GPUs, you had to use the AMD Radeon™ ProRender rendering engine. This change streamlines the user experience and reinforces native AMD hardware support.

You must have an AMD RDNA 2 GPU or newer that meets the minimum Vulkan® ray tracing extension requirements, and have sufficient VRAM (GPU memory).

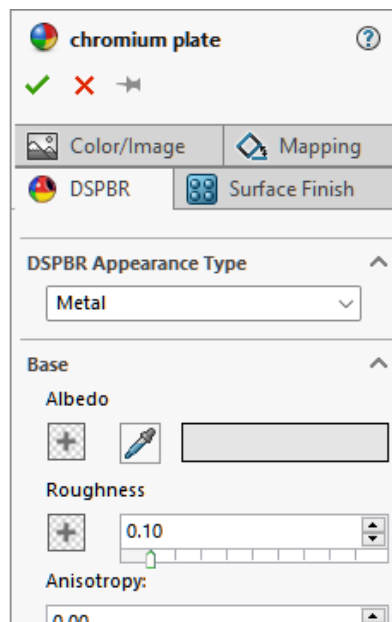
Different render engines and render modes require different minimum requirements and GPU compatibility. Therefore, the System Info dialog box is redesigned to reflect the support for additional hardware.

To access the System Info dialog box:

1. Click **Help > System Info**.
2. In the dialog box, specify the **GPU**, **Render Engine**, and **Render Mode**.

The dialog box displays a system capability assessment. It also displays support for **Vulkan Extensions** if you selected the **Stellar** and **Fast** or the **ProRender** and **Accurate** combinations.

DSPBR Support in SOLIDWORKS



SOLIDWORKS 2026 supports DSPBR appearances, creating a seamless transition when you open designs in SOLIDWORKS Visualize to produce high-quality, photo-realistic images.

DSPBR revolutionizes how SOLIDWORKS handles appearances from the user interface and asset libraries (appearances and scenes) to the quality of real-time rendering. The workflow enhancement aligns SOLIDWORKS more closely with Visualize and the **3DEXPERIENCE** platform.

In the Appearances PropertyManager, a DSPBR tab ensures direct material-to-material translation. All parameters from SOLIDWORKS map directly to Visualize. To use the tab, select **DSPBR (Dassault Systèmes Physically Based Rendering)** under **Appearance Visual Style** in **Tools > Options > System Options > Display**.

Previously, Visualize approximated DSPBR materials when you opened a SOLIDWORKS file in Visualize. Visualize uses the same approximation when you open files created in SOLIDWORKS 2025 or earlier. For SOLIDWORKS 2026 files, the mapping of SOLIDWORKS DSPBR appearances to Visualize uses a 1:1 correlation, eliminating the potential of error-prone approximations and providing a unified experience.

SOLIDWORKS CAM

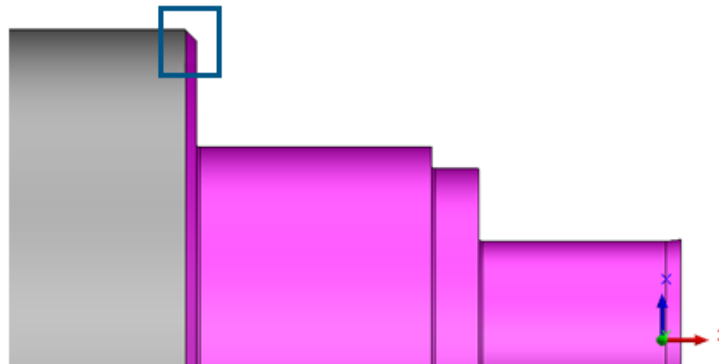
This chapter includes the following topics:

- **Bar Break Chamfers for Stock in Turn Toolpaths**
- **Collet Housing Parameters**

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

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

Bar Break Chamfers for Stock in Turn Toolpaths



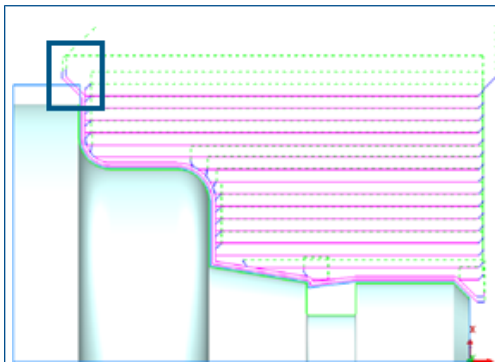
You can add bar break moves to Turn toolpaths generated for ODs to prevent burrs that can damage guide bushings.

During the Turn toolpath machining process, burrs (unwanted sharp edges), can form near the tool inserts when the edges of the cylindrical stock are machined. Burrs can cause damage when the stock material slides through guide bushings. To eliminate burr formation, you can specify an option for bar break moves for these types of Turn toolpaths:

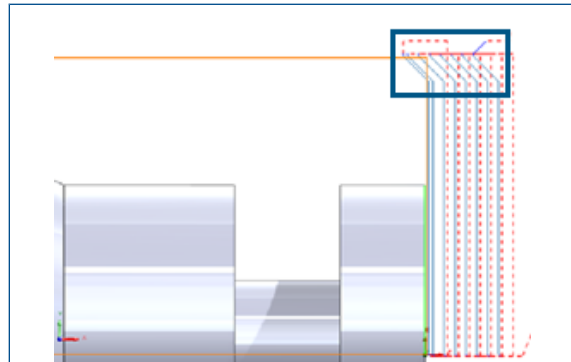
- Turn Rough
- Turn Finish
- Groove Rough (OD features only)
- Groove Finish (OD features only)
- Face Rough
- Face Finish

Adding bar break moves to the cut passes ensures the deburring of stock edges. It also prevents damage to guide bushings when the stock moves in and out of the guide bushing during the machining process.

SOLIDWORKS CAM adds bar break moves to the passes that intersect the maximum stock diameter. It appends these moves to the regular cut moves as required.

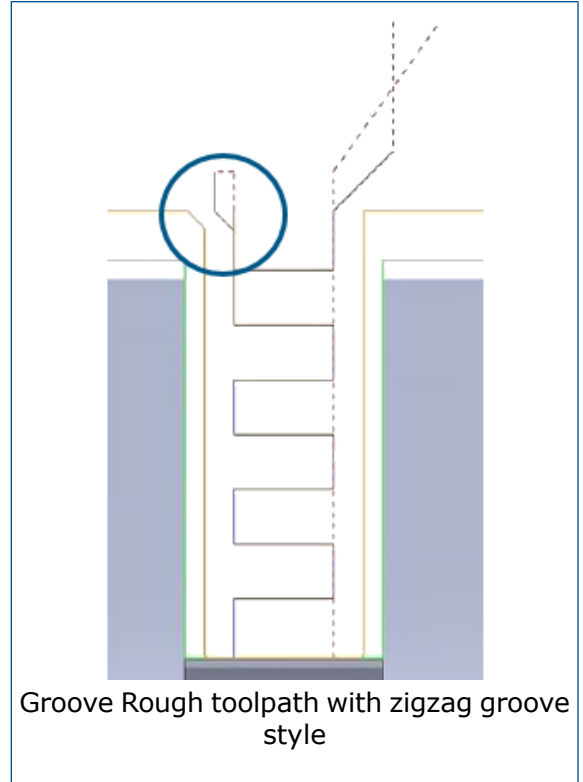
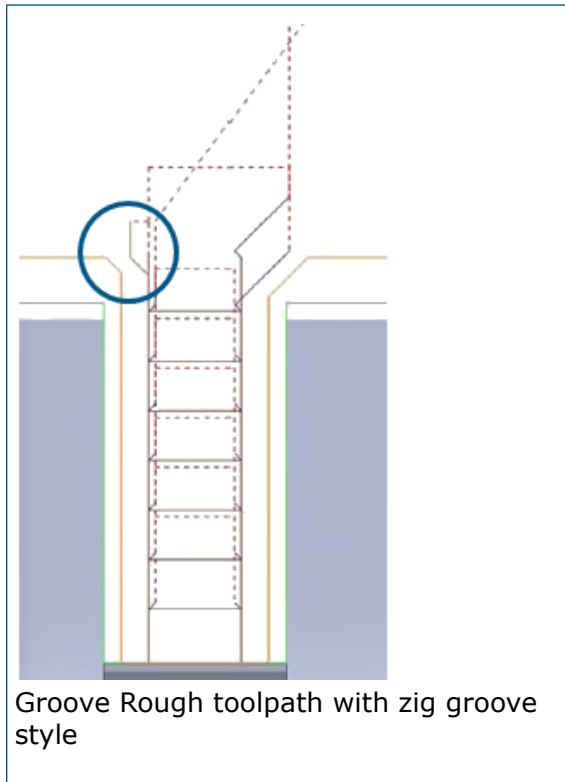


Turn Rough toolpath with bar break chamfers on first and final cut passes

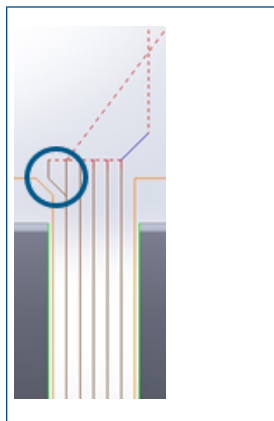


Face Rough toolpath with bar break chamfers for each cut pass

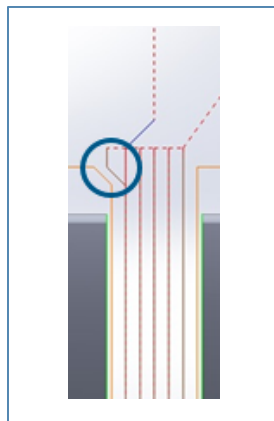
For Groove Rough and Finish toolpaths, SOLIDWORKS CAM adds the bar break moves based on the groove style. The app accounts for the cutting pattern of the groove when adding the bar break moves. There can be either one or multiple bar break moves depending on the groove style.



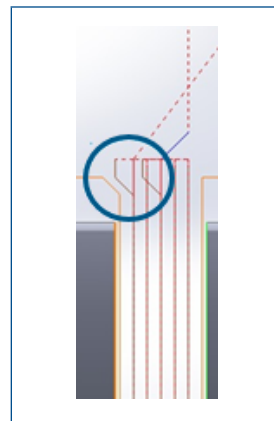
The following Groove Rough toolpaths have normal groove styles and no groove peck types:



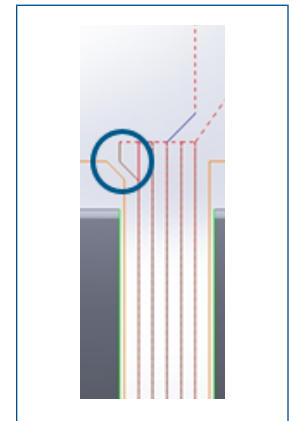
Order: S123
Bar break move
added before final cut
pass



Order: S321
Bar break move
added before first cut
pass



Order: S213
Bar break moves
added before first
and final cut passes



Order: S231
Bar break move
added before final cut
pass

Creating Bar Break Chamfers

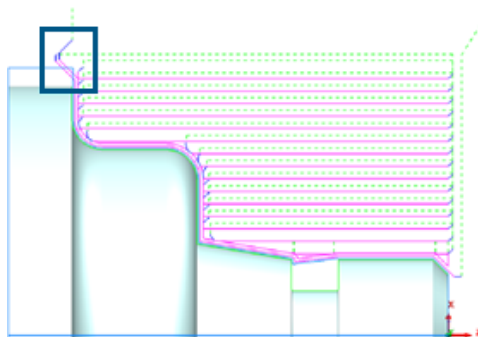
To specify bar break chamfering of stock in turn toolpaths:

1. In the Operation Parameters dialog box, on the NC tab, under **Bar Break**, specify a **Method**:

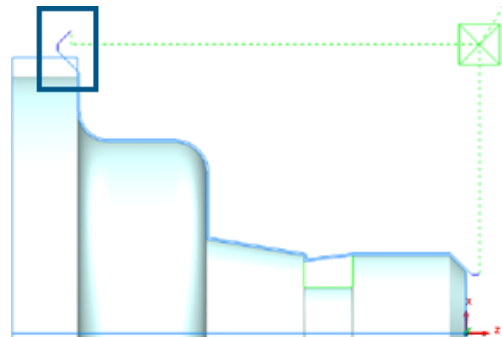
Method	Description
Chamfer	Chamfers the sharp edges of the stock along the OD feature. Specify the Distance and Angle of the chamfer.
Radius	Fillets the sharp edges of the stock along the OD feature. Specify the Distance and Radius of the fillet.
%	Specifies the chamfer distance as a percentage of the nose radius of a turn insert.

2. Optional: Select **Reverse**.

Reverse cuts the bar break move in the opposite direction to the cut passes (the tool approaches the chamfer profile from the maximum stock diameter and machines it). **Reverse** is available only for the Turn Rough and Turn Finish toolpaths but is unavailable if you reverse the **Cut type** on the Turn Rough or Turn Finish tabs of the Operation Parameters dialog box.

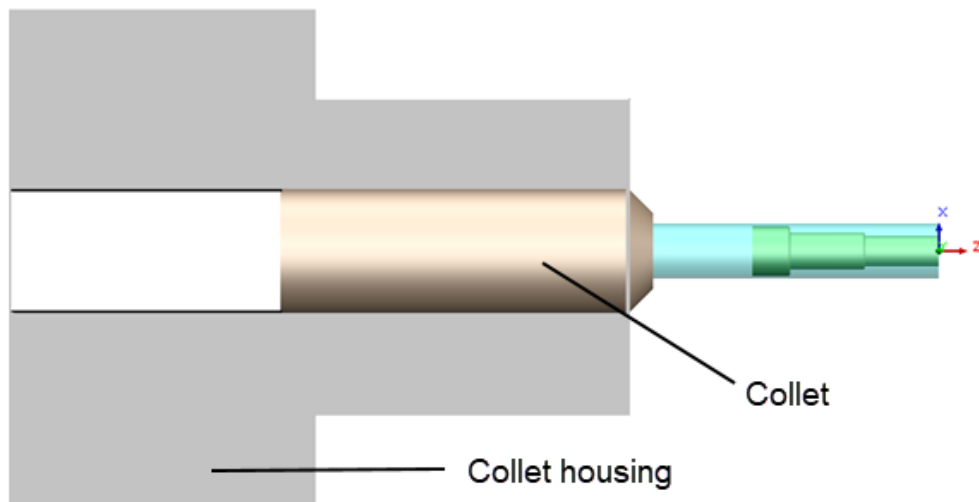


Turn Rough toolpath with **Reverse Cut type**. Bar break chamfers are added to the first and final cut passes.



Turn Finish toolpath with **Reverse Cut type**. Bar break chamfers move all cut passes.

Collet Housing Parameters




You can define parameters of a collet housing so you can visualize it while programming a part. It helps you to define the geometry of the collet housing directly in the graphics area.

In the Collet Parameters dialog box, under **Collet Housing Parameters**, specify options:

Option	Description
Collet Housing Diameter	Specifies the largest diameter for the collet housing.
Collet Housing Length	Specifies the overall length of the collet housing.
Minor Diameter	Specifies the diameter of the front part of the collet housing.
Collar Length	Specifies the length of the front part of the collet housing.

You can save default collet housing parameters in the Technology Database (TechDB™).

In TechDB, on the Turn Tooling  tab, under **Collet Housing Parameters**, specify values.

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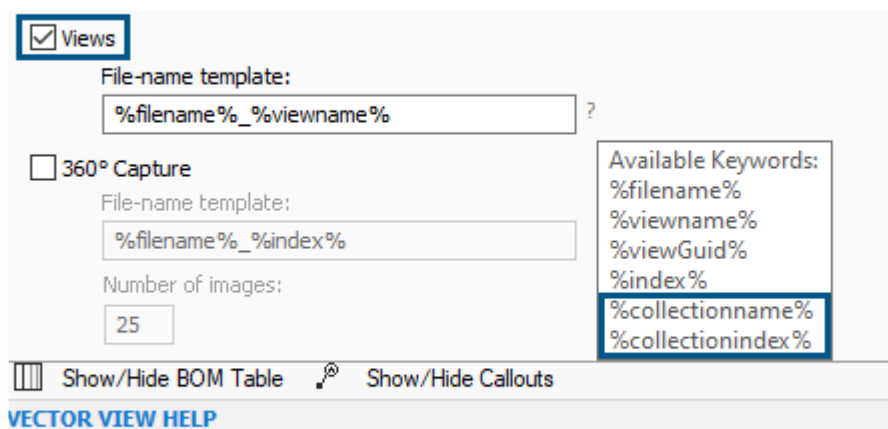
SOLIDWORKS Composer

This chapter includes the following topics:

- **Filename Template Options for Workshops**
- **Multiple Image Formats for Generating Videos**
- **PNG and TIFF Image File Formats**

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

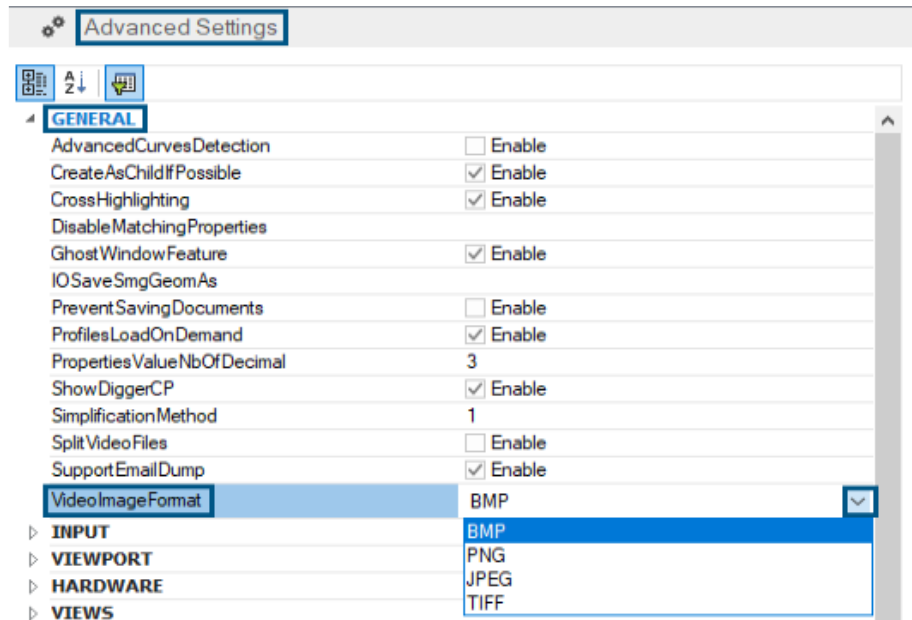
Filename Template Options for Workshops



The `collectionname` and `collectionindex` filename template options are available in the **Views** section of the Technical Illustration and High Resolution workshops.

This makes working with workshops easier. The `%viewindex%` keyword assigns an accurate index for a newly created view.

Multiple Image Formats for Generating Videos

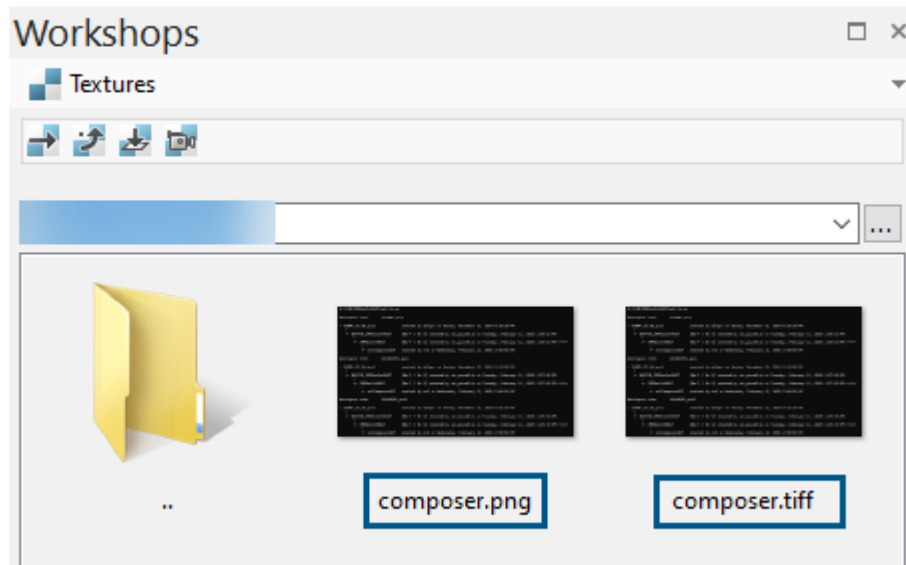


You can use BMP, PNG, JPEG, and TIFF image formats for animation frames while creating videos. You can create videos using the Video Workshop.

You have more image format options for animation frames. Generating videos is easier and faster.

You can use these image formats for MP4, MKV, and FLV output video formats. These video formats use an x264 library. formats.

PNG and TIFF Image File Formats



You can use the PNG and TIFF image file formats in Composer.

You can use the PNG and TIFF formats while:

- Working with the Texture workshop.
- Importing PNG and TIFF image files for textures in Composer.
- Working with the viewport background.

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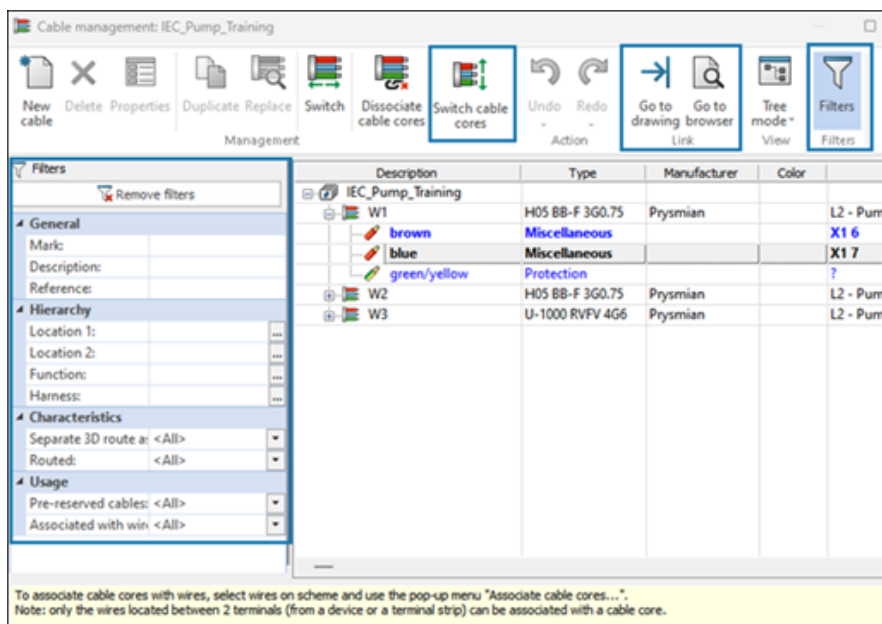
SOLIDWORKS Design Electrical


This chapter includes the following topics:

- **Cable Management**
- **Hiding System Classes**
- **Routing Selected Wires Separately**
- **Connector Dynamic Insertion**
- **Update and Replace Project Data**

SOLIDWORKS® Design Electrical is a separately purchased product.

Cable Management




You can manage cables and cores efficiently with advanced filtering options and the **Switch cable cores**  Command. While managing the cables and cores, you can directly navigate to the schematics and the component browser.

Benefits: You can manage cables and cores more quickly and effectively.

To access commands, click **Electrical Project > Cables**.

Advanced Filtering in the Filters Panel

You can filter cables and cores using the advanced filtering options in the **Filters**  panel.




In the Cable management dialog box, click **Filters**  to display the **Filters** panel.

The **Filters** panel has additional filters:

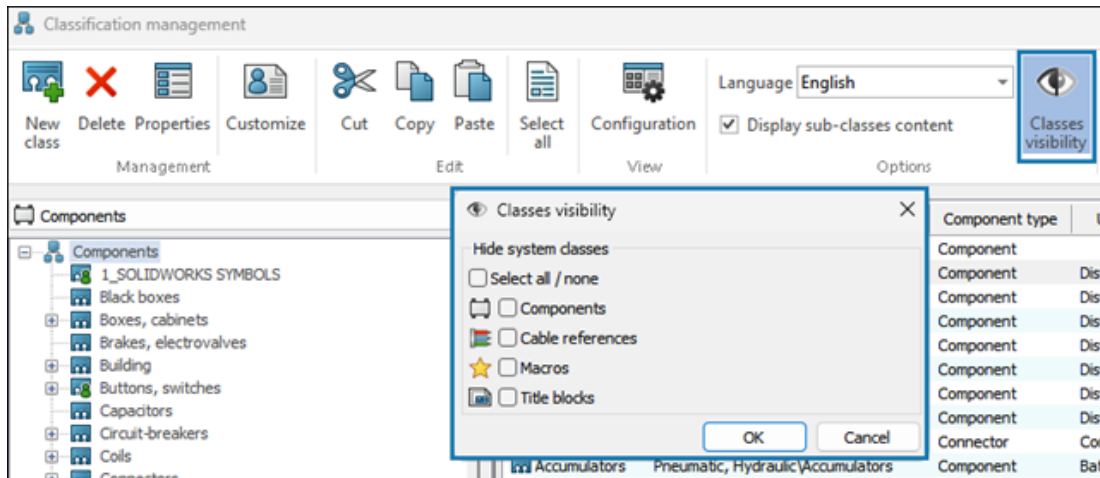
- **Reference**
- **Function**
- **Harness**
- **Separate 3D route assembly**
- **Routed**
- **Pre-reserved cables**
- **Associated with wire**

Additional Capabilities for Productivity of Cable Management


You can improve productivity when managing cables using additional commands in the Cable management dialog box.

	Command	Description
	Switch cable cores	Swaps two selected cable cores. During the switch, the app directly dissociates the wires for the two cores and reassociates them after the switch. You can switch cable cores within the same cable or between different cables.
	Go to drawing	Navigates to the schematic where the cable or core is placed.
	Go to browser	Opens the component browser and highlights the cable's origin component.
	Wire Mark	Displays the wire mark text, which is either the equipotential number or the wire mark. This provides clearer wire information and improves navigation in the schematic.

Hiding System Classes









You can hide default system classes to simplify the Classification design tree.

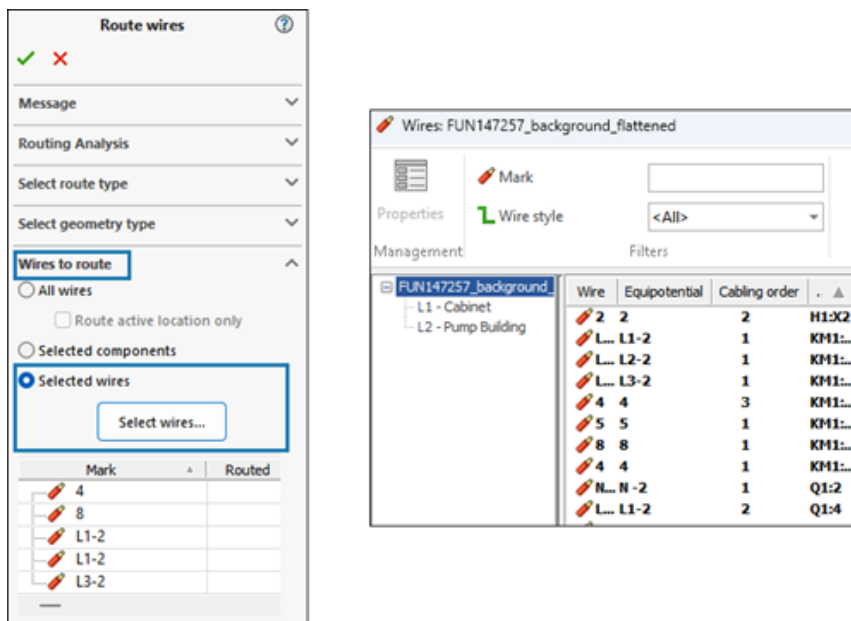
Hidden classes remain visible in the Classification Management  dialog box but are removed from library and selection interfaces. Custom subclasses under system classes stay visible to preserve the hierarchy.

Benefits: You can find relevant components and symbols more easily.

To hide system classes from symbol and manufacturer:

1. In the ribbon, click **Library > Classification management** .
2. In the Classification management dialog box, click **Classes visibility** .
3. In the Classes visibility dialog box, select the classes to hide:
 - **Select all/none**
 - **Components** 
 - **Cable references** 
 - **Macros** 
 - **Title blocks** 


Routing Selected Wires Separately



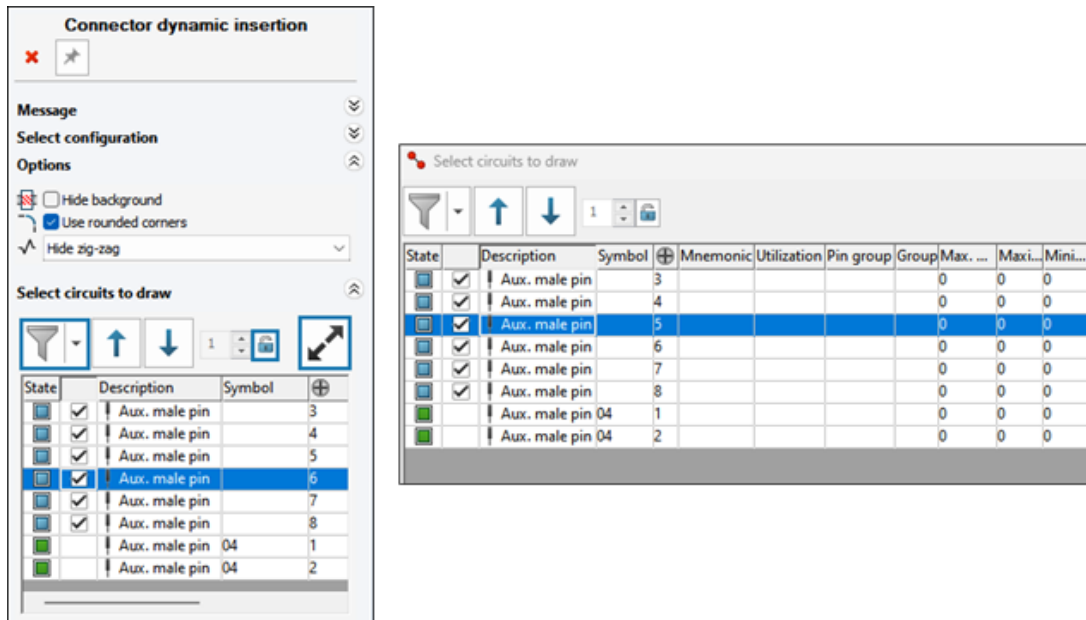
You can route a specific wire or a set of selected wires separately using **Selected wires** in the Route Wires PropertyManager.

In previous releases, you could route wires for all the components or select the components to route their respective wires.

To route selected wires separately:


1. In the ribbon, click **Route Wires**.
2. In the **Route wires** PropertyManager, click:
 - a. Selected wires.
 - b. Select wires.
3. In the Wires dialog box:
 - a. Select the wires to route together.
 - b. Click **Select**.
4. Click .

Connector Dynamic Insertion



The Connector dynamic insertion dialog box improves productivity when working with connectors.




Enhancements include:

- Larger dialog box for showing details of connector circuits and pins
- Better filtering capabilities
- Improved management of connectors with a large number of pins
- Easier access to the **Insert connector** command
-  opens the Select circuits to draw dialog box

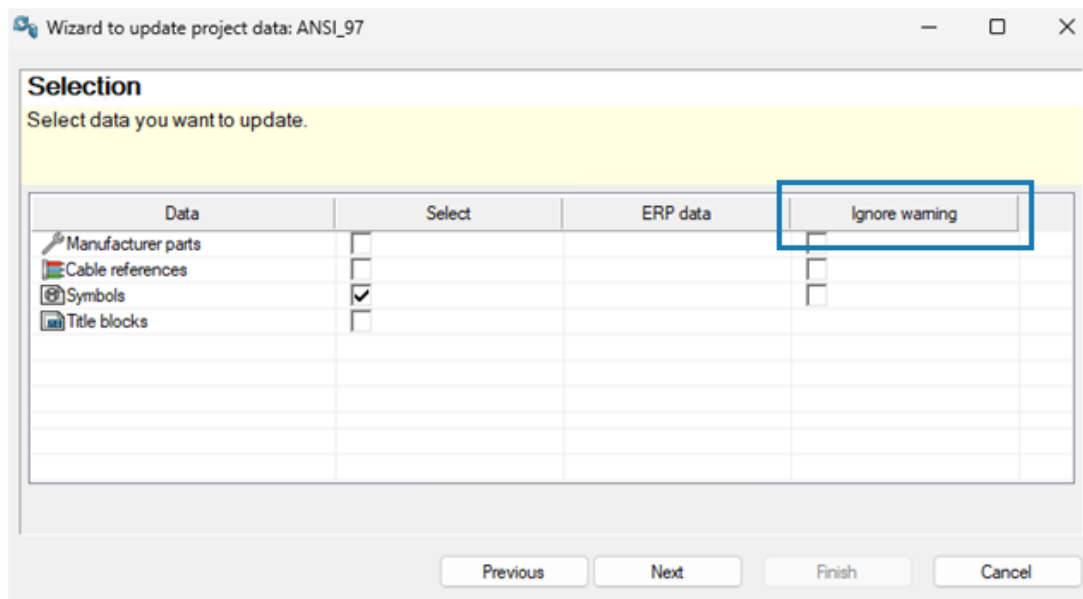
Select Circuits to Draw Dialog Box

The Select circuits to draw dialog box lets you select and manage the number of circuits to include in the scheme drawing when working with dynamic connectors.

To open the Select circuits to draw dialog box:

1. In the scheme drawing, select a symbol.
2. Do one of the following:
 - Click **Schematic > Insert connector** .
 - Right-click a symbol in the scheme, and select **Component > Insert connector** .
3. In the Connector dynamic insertion command panel, under **Select circuits to draw**, click .

Update and Replace Project Data



When you update symbols, manufacturer parts, or cable references that have discrepancies, the system displays a warning message instead of an error message.

Discrepancies include:

- The symbol has more connection points
- The number of circuits/terminals in the manufacturer part is higher
- The number of cable cores in cable reference is higher

This improves the clarity and efficiency of symbol, manufacturer part, and cable references updates, allowing for smoother project management and reducing errors during updates and replacements.

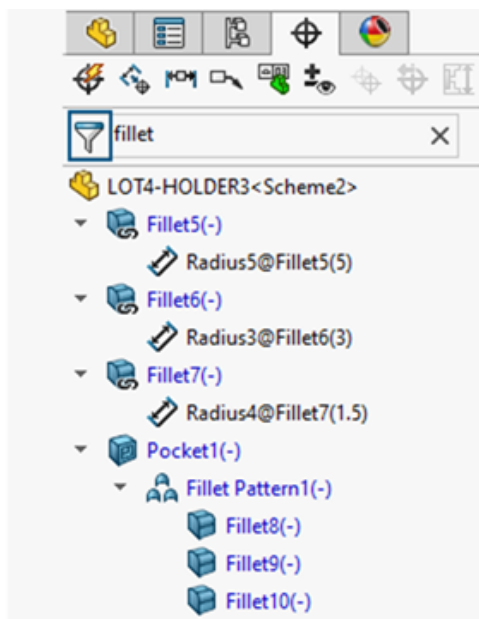
The option **Ignore warning** is added to the Wizard to update project data and Wizard for replacement of project data dialog boxes. This lets you apply the updates and replacements even when the new data is not fully compatible with the existing one. This simplifies the project data updates or replacements and improves the project management during design.

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SOLIDWORKS MBD


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

Filtering the DimXpertManager



You can use a filter in the DimXpertManager to search for DimXpert features, annotations, and annotation views.

To filter the DimXpertManager:

1. At the top of the DimXpertManager, in the filter , enter a keyword to display items to view.

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DraftSight

This chapter includes the following topics:

- **Performance**
- **Start Page Tab (DraftSight Premium Only)**
- **Ribbon Optimization**
- **Powertools Ribbon Tab (DraftSight Premium Only)**
- **Contextual Ribbon for Gradients and Patterns**
- **Manipulating ViewTiles (DraftSight Premium Only)**
- **ViewTiles Controls**
- **Floating Document Windows (DraftSight Premium Only)**
- **ECW Images**
- **CCS Icon Customization**
- **Color Books (DraftSight Premium Only)**
- **PCX Print Configuration Files (DraftSight Premium Only)**
- **Managing Missing External References**
- **Insert Formula Column in Data Extraction**
- **Diesel Expressions**
- **MTEXT Command**
- **RENAME Command**
- **Copying with SCALE Command**
- **Power Dimension Tool (DraftSight Mechanical Only)**

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

Performance

DraftSight performance is improved with switching between sheets, zooming and panning operations, and file opening times.

When you switch to a sheet, performance is improved by an average of 66%. Switching from a sheet back to the model is improved by 78%. These enhancements were measured across several hardware configurations, from low-end setups to high-performance computers, benefiting users no matter what type of system you work on.

Zoom performance is improved by up to 55% in certain cases, and Pan performance is increased by about 38%.

Opening files averages 10% faster. This reduces wait time and maximizes the time you can dedicate to your work.

Start Page Tab (DraftSight Premium Only)



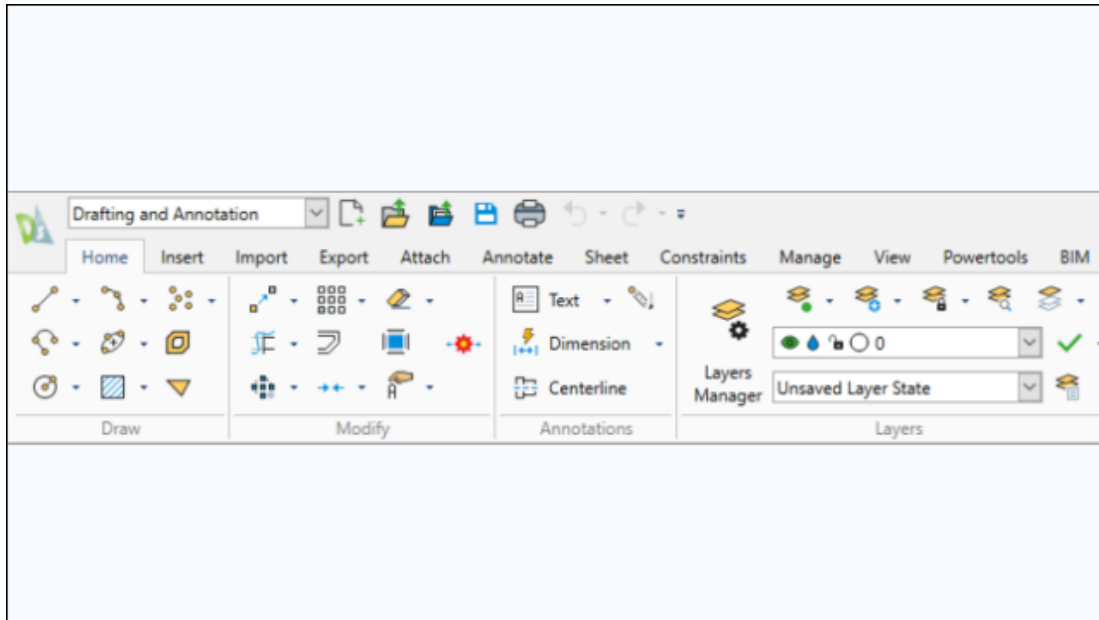
The Start Page tab centralizes key operations and creates a smoother user experience when you open DraftSight.

With the Start Page tab, you can:

- Start new drawings. You can begin new projects instantly, selecting the appropriate units and scale for specific project types. This eliminates project setup time and ensures a productive start for new drawings.
- Continue previous work immediately. The tab provides access to recent files. This lets you continue ongoing projects where you left off, ensuring efficient, uninterrupted progress, whether working alone or on a team.
- Access learning resources. You can access tutorials, documentation, and skill-building resources. This supports new and existing users for enhancing skills, discovering advanced tools, and troubleshooting.
- Customize your workspace. You gain quick access to workspace customization to tailor the user interface according to your preferences and project needs. Customization boosts productivity and creates a more comfortable working environment.
- Review recent projects. You can track and resume recent work, encouraging continuous design improvement and ease of monitoring evolving projects.

To use the Start Page tab, enter `STARTMODE` in the command window.

Ribbon Optimization

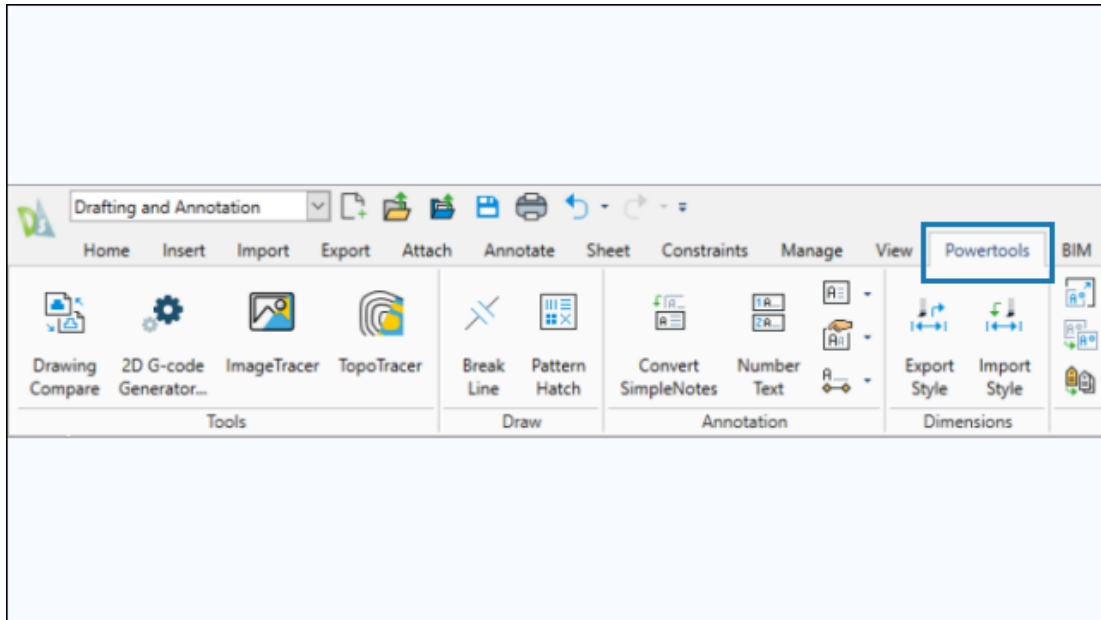


DraftSight has an updated ribbon layout and workspace to improve usability. The ribbon tabs have reduced clutter and distributed commands across additional tabs for better accessibility.

The Home tab includes commonly used commands to enhance efficiency in everyday tasks. The Start tab lets you select a workspace to suit your needs.

The Drafting and Annotation workspace includes panels to group, show, or hide specific entities, and Import and Export tabs.

Powertools Ribbon Tab (DraftSight Premium Only)



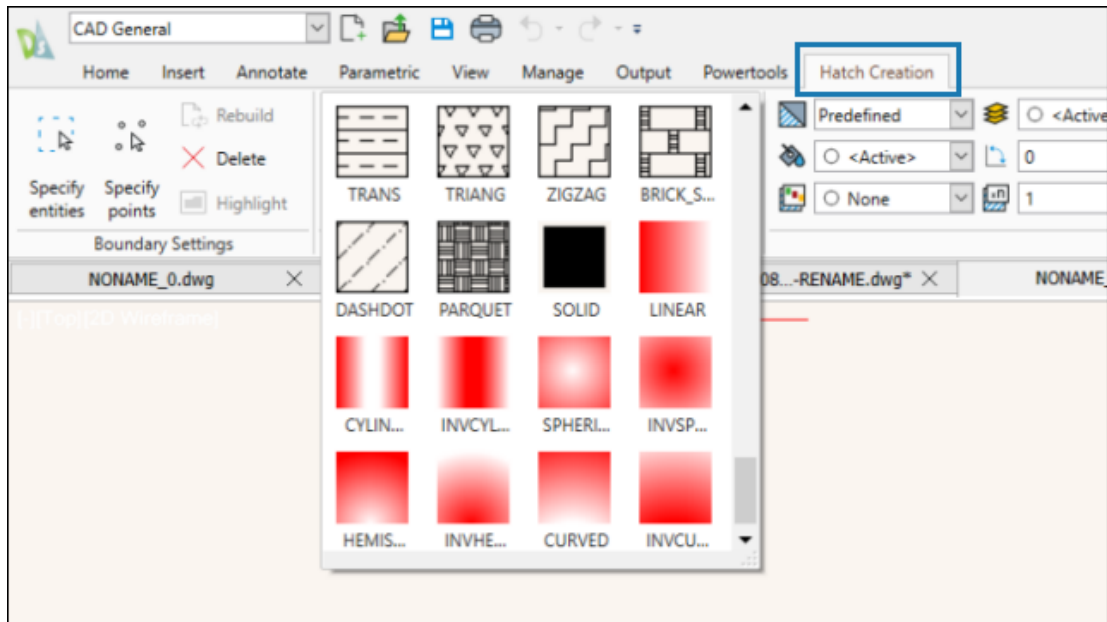
The Powertools ribbon tab has additional tools on it to streamline your workflow.

The tools help you:

- Manage Viewport layouts
- Import and export DimensionStyles
- Create professional text labels
- Scale blocks with precision
- Define draw order by color

These tools provide greater flexibility and automation, increasing your efficiency.

Contextual Ribbon for Gradients and Patterns

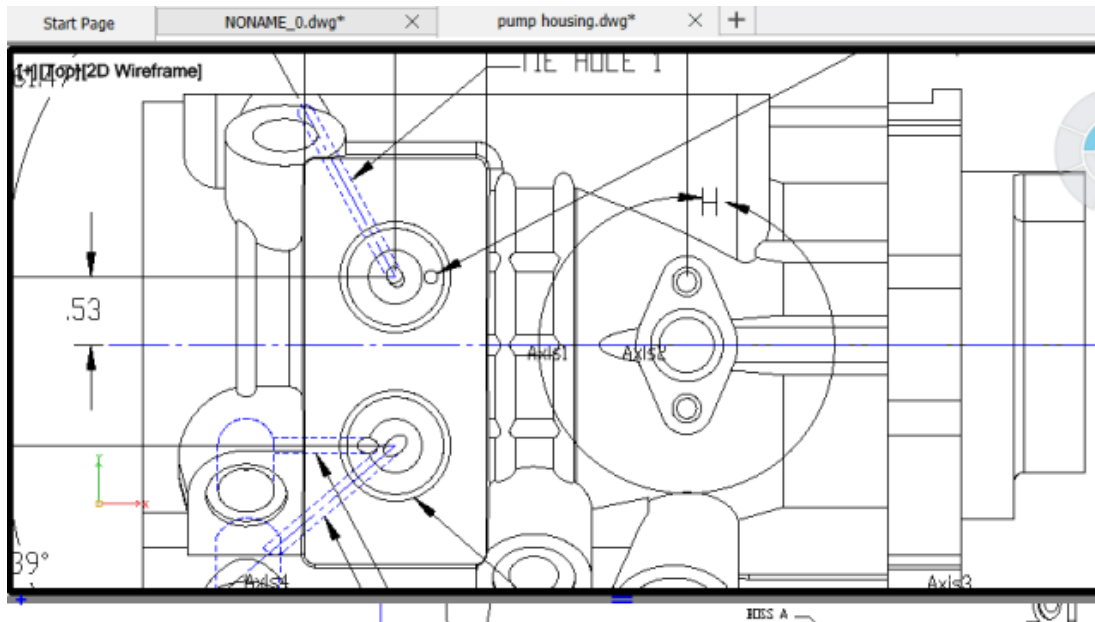


The contextual ribbon has tabs for the `HATCH` and `PATTERN` commands. This enhances productivity by reducing the time spent on finding the commands.

These tabs give you quicker access to adjust gradient fills, format text, and design patterns, making it easier to achieve visually appealing, professional results in less time.

To use the contextual ribbon for gradients and patterns, enter `HATCH` or `PATTERN` in the command window.

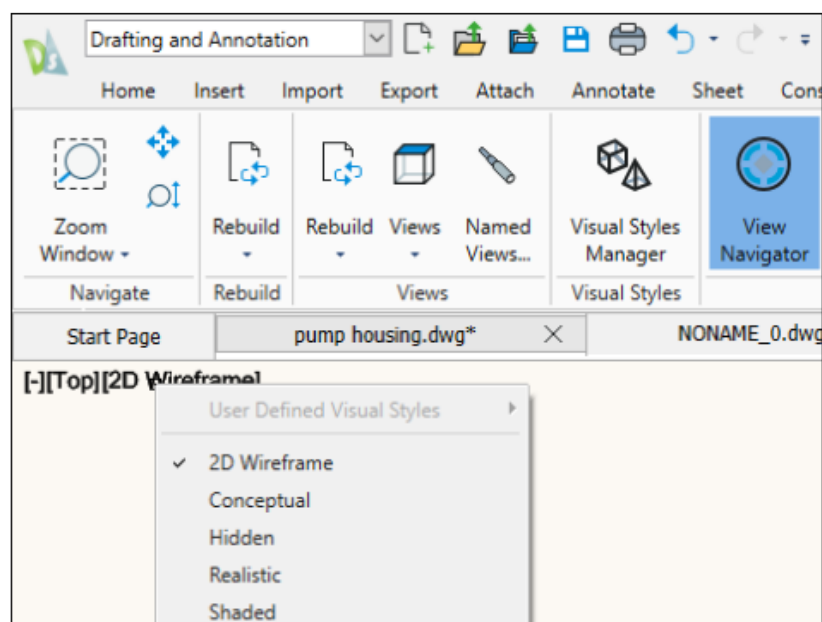
Manipulating ViewTiles (DraftSight Premium Only)



You can resize ViewTiles by adjusting boundaries, merge ViewTiles when boundaries align, and create new ViewTiles with one click or using **CTRL+drag**.

Resizable ViewTiles have adjustable markers and snap-to-boundary alignment, and allow resizing of connected tiles. DraftSight allows flexible arrangements (one-to-one, one-to-multiple, and multiple-to-multiple configurations), and all actions support **Undo/Redo** for corrections.

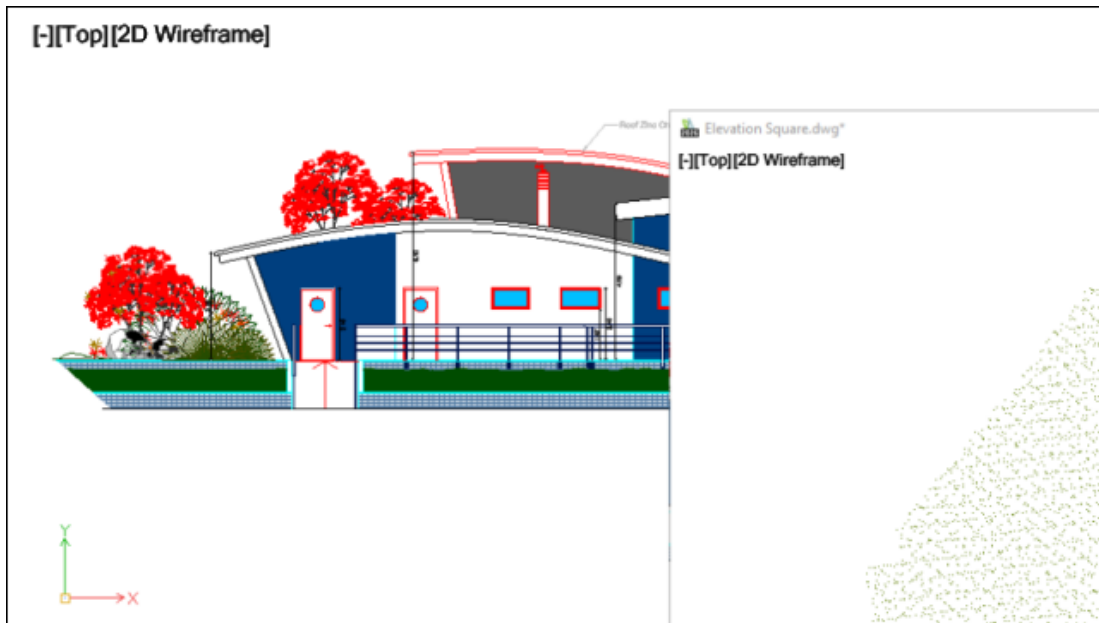
ViewTiles Controls



You can access ViewTile controls in the upper-left corner of each ViewTile (viewport). This gives you convenient access to settings for changing views, adjusting visual styles, and configuring display options. You can modify these settings without navigating through menus or toolbars by clicking within the bracketed areas.

ViewTile controls workflows so you can make adjustments without leaving the drawing or searching for commands. Whether switching between 2D and 3D views or applying different styles to analyze designs, these controls provide an efficient way to work within a drawing.

Floating Document Windows (DraftSight Premium Only)



With floating document windows, you can open drawings in separate windows outside the main application in DraftSight. The windows offer flexible views of multiple drawings side-by-side or across multiple monitors. This functionality provides additional workflow options and enhances productivity by facilitating comparisons, copying and pasting, and multitasking.

To create a floating document window, drag a document tab outside the main window. The floating document window operates independently; you can adjust its size, zoom, and navigate as required.

ECW Images

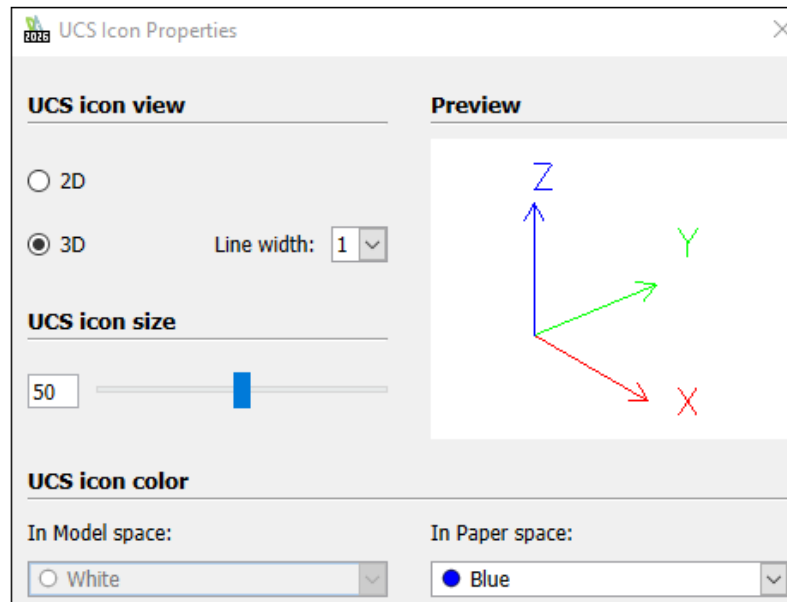
DraftSight supports the Hexagon Enhanced Compression Wavelet (.ECW) image format. .ECW files use efficient compression, making them ideal for large-scale images such as aerial and satellite photography. The files can include embedded map projection data, enhancing their usefulness in geographic information system (GIS) and remote sensing applications.

To use ECW images:

Do one of the following:

- On the ribbon, click **Drafting and Annotation** > **Insert** > **Reference** > **Attach Image**.
- On the menu, click **Insert** > **Attach Image**.
- Enter `ATTACHIMAGE` in the command window.

CCS Icon Customization



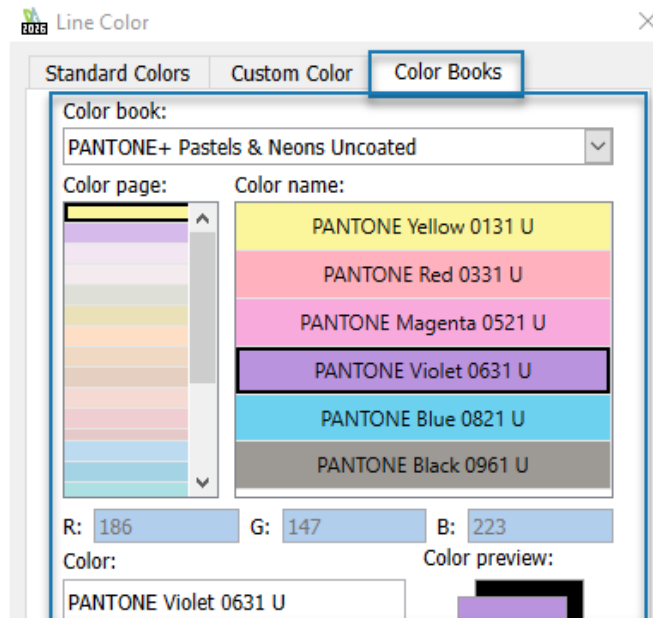
The CCS icon represents the current orientation of the coordinate system. You can adjust this visual indicator to suit your preferences or the needs of a particular project.

The `UCSICON` command includes a **Properties** option in the command prompt. In the CCS Icon Properties dialog box, you can customize the CCS icon's view, size, and color. The dialog box includes a dynamic preview to display the changes.

DraftSight saves the customizations across sessions.

To use CCS icon customization, enter `UCSICON` in the command window.

Color Books (DraftSight Premium Only)



DraftSight supports custom color palettes, also known as color books (.acb files). These give you access to standardized color palettes within projects. Color books are ideal for professionals who rely on precise color standards for branding, industry compliance, or consistent representation of materials.

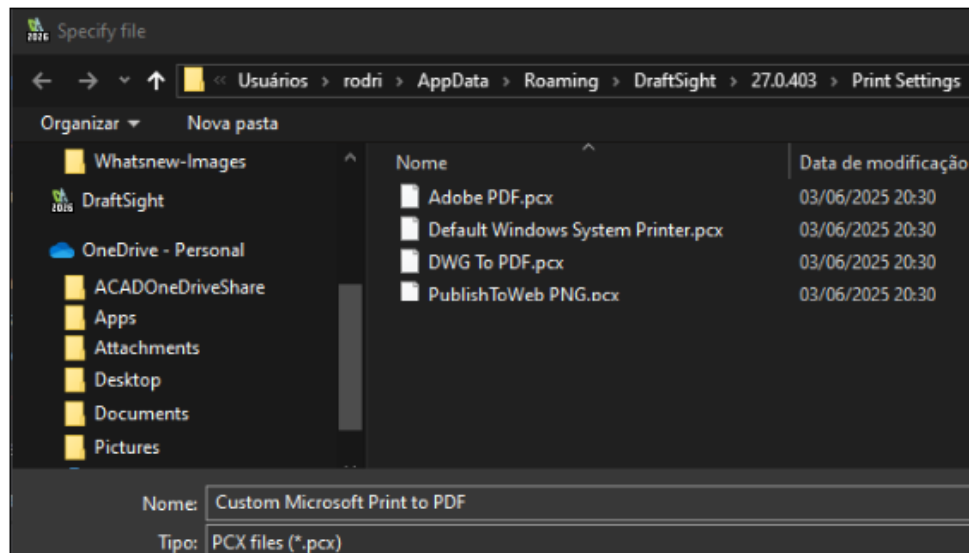
With color books, you can quickly select and apply exact shades from popular color systems such as Pantone® and RAL™, ensuring visual accuracy and consistency across designs.

To use color books:

Do one of the following:

- On the ribbon, click **Home > Properties > Color**.
- On the menu, click **Format > Line Color**.
- Enter `LINECOLOR` in the command window.

PCX Print Configuration Files (DraftSight Premium Only)



DraftSight supports .PCX print configuration files, providing functionality similar to the .PC3 format in other CAD software.

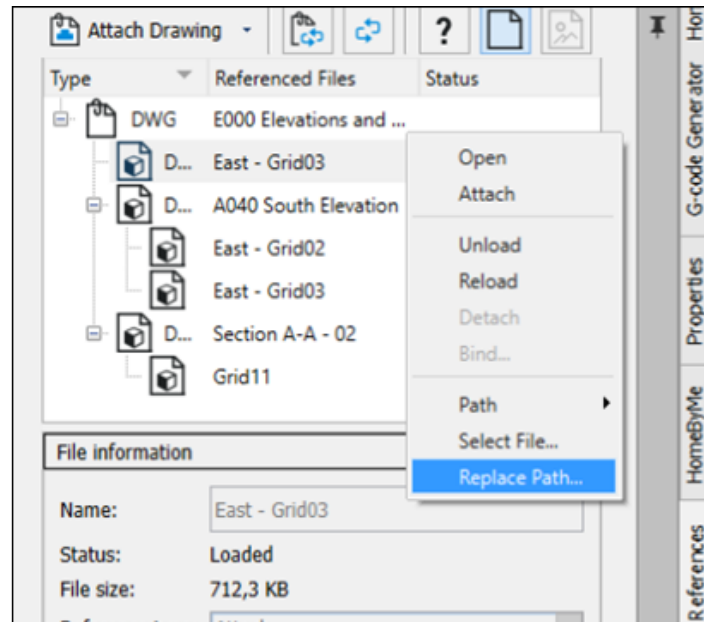
You can import .PC3 files or create and save new print configuration files in the .PCX format. The .PCX format makes it easier to reuse and share print settings for consistent output across multiple print jobs. This improves workflow efficiency by ensuring standardized print configurations.

To use .PCX print configuration files:

Do one of the following:

- On the ribbon (Application menu), click **Print > Print**.
- On the menu, click **File > Print**.
- Enter `PRINT` in the command window.
- Keyboard shortcut: **CTRL+P**.

Managing Missing External References



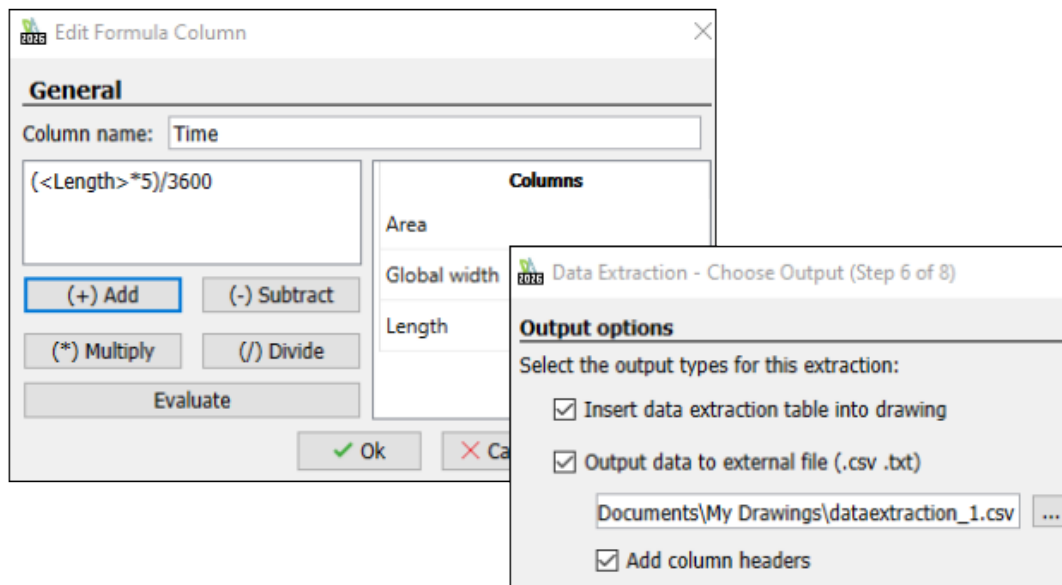
You can manage missing external references more efficiently with tools in the **References** palette. When a referenced file is moved or renamed, you can update the file path once. DraftSight offers to use the same path for other missing references if it finds matching files.

To manage missing external references:

Do one of the following:

- On the ribbon, click **Insert** > **Palettes** > **References Manager**.
- On the menu, click **Tools** > **References Manager**.
- Enter `XREF` in the command window.

Insert Formula Column in Data Extraction



The DATAEXTRACTION command allows for .CSV output and customized fields for mathematical formulas.

With the DATAEXTRACTION command, you can create output data in .CSV format, which includes column headers for readability. This is a convenient way to share information or integrate it with other applications in a text format.

The DATAEXTRACTION wizard includes an **Insert Formula Column** option where you can define custom fields with mathematical expressions. This provides flexibility when structuring data extraction tables because you can make calculations directly in the wizard. You can create more refined and tailored data exports without external processing.

To use the DATAEXTRACTION command:

Do one of the following:

- On the ribbon, click **Insert > Data Linking > Data Extraction**.
- On the menu, click **Insert > Data Extraction**.
- Enter DATAEXTRACTION in the command window.

Diesel Expressions

The screenshot shows the 'Field' dialog box in DraftSight. The 'General' tab is active. On the left, under 'Category', the 'Others' option is selected. On the right, under 'Name', the 'Diesel Expression' option is selected. Below these, the 'Diesel expression' text box contains the formula: `First 4 char $(substr, $(getvar, "dwgname"), 1, 5)`. The 'Preview' section shows the result: 'First 4 char dies'. At the bottom, the 'Field string' section shows the final output: `%<\AcDiesel First 4 char $(substr, $(getvar, "dwgname"), 1, 5)>%`.

The `FIELD` command supports Diesel Expressions, enabling dynamic text that automatically updates based on drawing properties or user inputs.

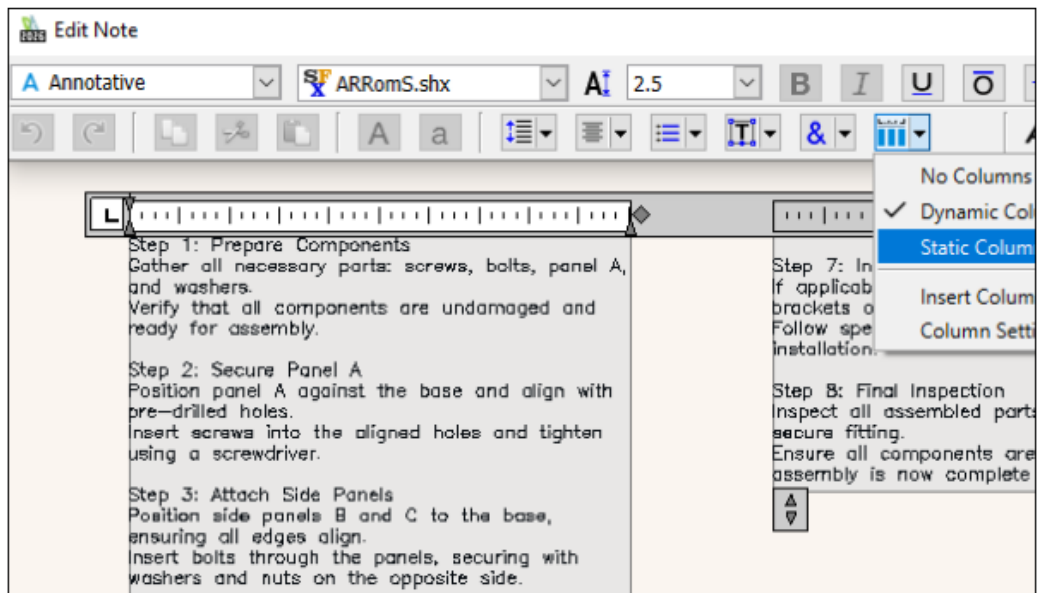
You can create custom text fields that concatenate strings, perform calculations, and apply conditional formatting without needing external scripts or complex programming. This improves automation and flexibility when working with text data in drawings.

To use the `FIELD` command:

Do one of the following:

- On the ribbon, click **Insert > Data > Field**.
- On the menu, click **Insert > Field**.
- Enter `FIELD` in the command window.

MTEXT Command



You can create and edit multiple columns within **MTEXT** entities. With the in-place text editor, you can add columns with equal widths and consistent gutters (spaces between columns), and adjust the height with grips.

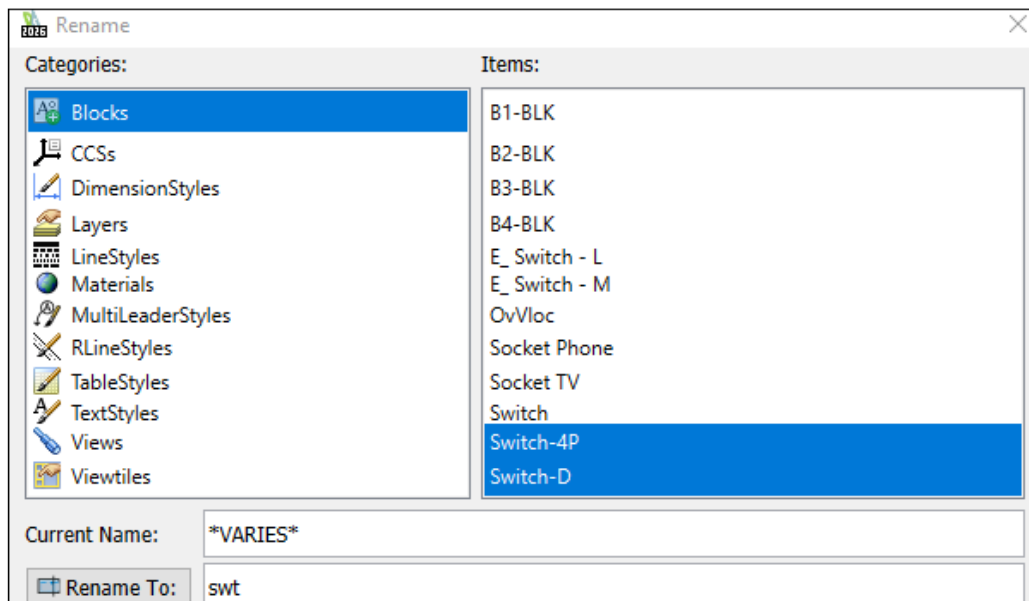
Columns are beneficial for complex documentation or design notes because they help you organize multicolumn text into clearly structured, easy-to-read layouts. They also maximize the available workspace by efficiently managing text within drawings.

To use the **MTEXT** Command:

Do one of the following:

- On the ribbon, click **Annotate** > **Text** > **Note**.
- On the menu, click **Draw** > **Text** > **Note**.
- Enter **NOTE** in the command window.

RENAME Command



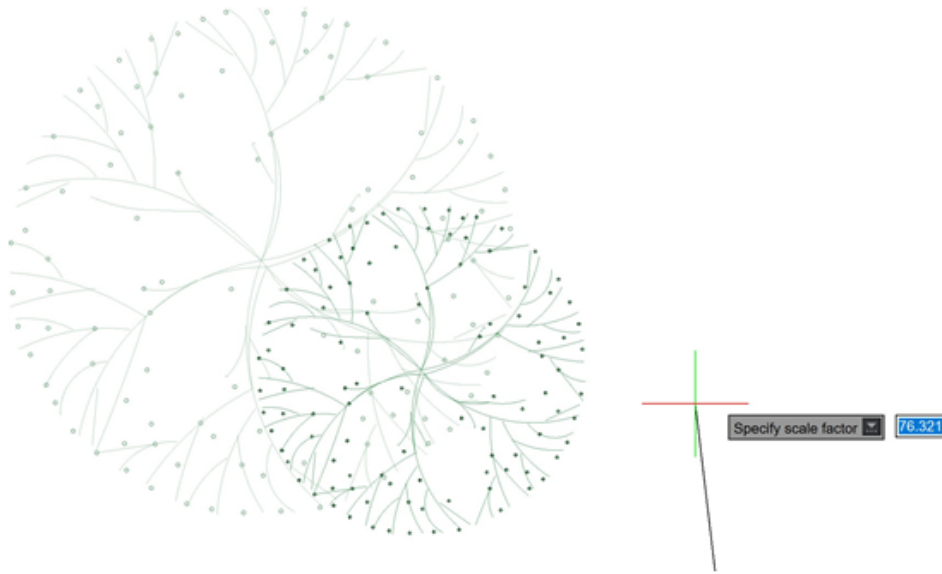
The **RENAME** command supports wildcard characters, making it easier to find and rename multiple named elements at once. You can filter matching items in the **RENAME** dialog box and rename multiple objects more efficiently.

To use the **RENAME** command:

Do one of the following:

- On the menu, click **Format > Rename**.
- Enter **RENAME** in the command window.

Copying with SCALE Command



The **SCALE** command includes a **Copy** option. With this option, you can preserve the original entities while generating scaled duplicates.

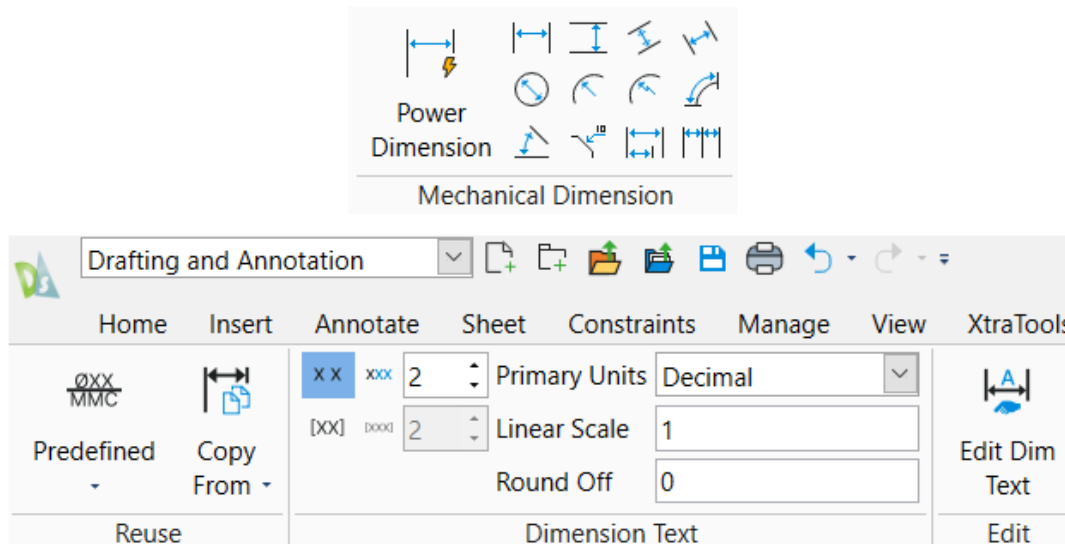
The **Copy** option provides a more streamlined and efficient process. Previously, you had to use the **COPY** command before the **SCALE** command to copy entities.

To use the **SCALE** command:

Do one of the following:

- On the ribbon, click **Home > Modify > Scale**.
- On the menu, click **Modify > Scale**.
- Enter **SCALE** in the command window.

Power Dimension Tool (DraftSight Mechanical Only)



The **Power Dimension** tool provides an advanced and efficient way to create precise dimensions. It selects an appropriate dimension type based on the selected geometry, which ensures consistency and accuracy in technical drawings.

The **Power Dimension** tool:

- Determines the best dimension type for selected objects automatically.
- Supports linear, radial, and angular dimensions.
- Maintains alignment and spacing for a cleaner, more readable layout.
- Enhances productivity by reducing manual adjustments and rework.

To access the Power Dimension tool:

Do one of the following:

- On the ribbon, click **Mechanical Annotate** > **Mechanical Dimension**.
- On the menu, click **Mechanical Annotate** > **Mechanical Dimension**.
- Enter `AM_POWERDIMENSION` or `AMPOWERDIM`.

Power Dimensioning Contextual Ribbon Tab

The Power Dimensioning contextual ribbon tab enhances the dimensioning workflow. It provides quick access to essential tools to modify and refine dimensions.

The following panels are available:

- **Reuse**. Provides tools to apply predefined dimension text, copies properties from existing dimensions, and exports dimension settings to other dimensions.
- **Dimension Text**. Controls dimension text visibility, precision, formatting, scaling, and rounding for primary and alternate units.
- **Edit**. Displays the Note Formatting toolbar that lets you edit the selected dimension.

SOLIDWORKS Plastics

This chapter includes the following topics:

- **Materials Database**
- **Performance**
- **Thermoset Materials**
- **Unfilled Volume Plot**
- **Venting Analysis**

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

Materials Database

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

85 new material grades are added, 12 grades are updated, and 50 obsolete grades are removed from the database.

Manufacturer	Number of New Material Grades
SABIC Specialties®	41
CHIMEI®	22
Roehm GmbH	16
Roehm America LLC	6

Manufacturer	Number of Updated Material Grades
SABIC Specialties®	12

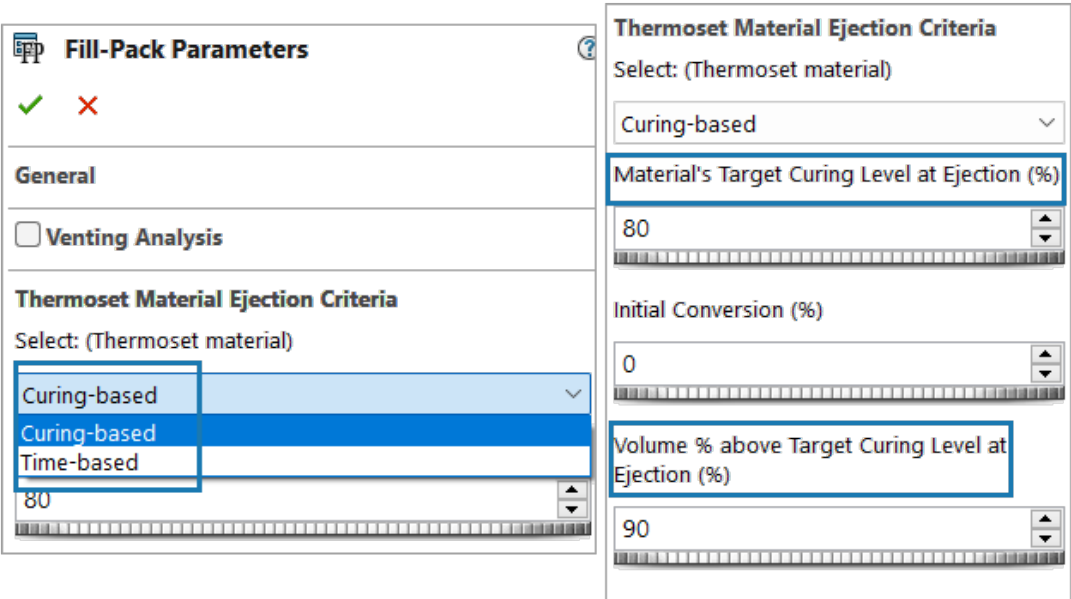
Manufacturer	Number of Removed Material Grades
Rohm GmbH and Company KG	27
Rohm and Haas	19
ICI	3
Mitsubishi Chemical®	1

Performance

Improved efficiency in solving the underlying systems of equations improves the solution times of plastics simulations without affecting robustness and accuracy.

- Up to 15% faster solution for Fill simulations
- Up to 30% faster solution for Pack simulations
- Up to 25% faster solution for Cool simulations

Thermoset Materials



User interface parameter updates for thermoset materials improve usability and solver updates improve the accuracy of Fill, Pack, and Warp simulations.

Simulations account for the orientation effects of fiber-filled thermoset materials which improve the solution accuracy.

The table lists the user interface parameters that are renamed, and one new parameter that is added.

Thermoset Material Parameters - SOLIDWORKS Plastics 2025	Thermoset Material Parameters - SOLIDWORKS Plastics 2026
Reactive Control Type <ul style="list-style-type: none">• Conversion• Time	Thermoset Material Ejection Criteria <ul style="list-style-type: none">• Curing-based directs the solver to continue the curing simulation until the material reaches the specified target curing level, eliminating the guesswork of determining the curing time.• Time-based sets a specific curing time for the solver to complete the thermoset curing simulation. After you run the simulation, you can review the results to determine the percentage of curing achieved across your model and adjust the curing time to either shorten or extend the process.
Ejection Conversion %	Material's Target Curing Level at Ejection sets the percentage of curing at

Thermoset Material Parameters - SOLIDWORKS Plastics 2025	Thermoset Material Parameters - SOLIDWORKS Plastics 2026
	which the thermoset material reaches its gelation point—where its viscosity becomes infinite, and it loses the ability to flow. Curing beyond this level is not beneficial and only adds to the manufacturing cycle time. This characteristic is an intrinsic property of the material determined through characterization, typically provided by the material manufacturer.
	New parameter: Volume % Above Target Curing Level at Ejection sets a threshold volume percentage that determines the ejection point. The default setting is 90%, meaning the part is ejected once 90% of the plastic volume reaches the target curing level.

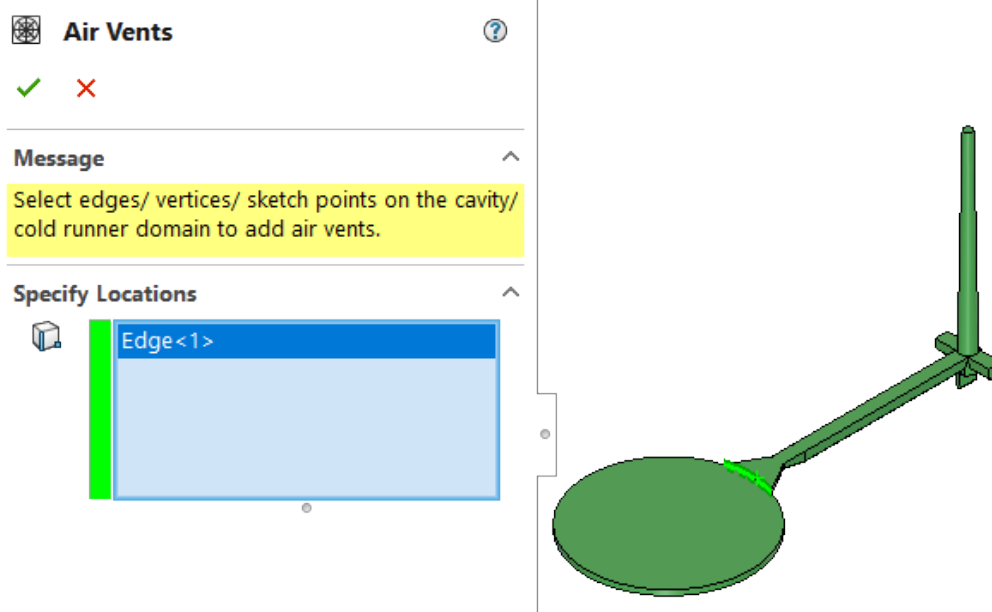
Results Related to Thermosets - SOLIDWORKS Plastics 2025	Results Related to Thermosets - SOLIDWORKS Plastics 2026
<ol style="list-style-type: none"> 1. Curing Time at End of Fill 2. Material Reactive Conversion at End of Fill 3. Curing Time at Post-Filling End 4. Material Reactive Conversion at Post-Filling End 	<ol style="list-style-type: none"> 1. Time to Reach Curing Level 2. Material's Curing Level at End of Fill 3. Time to Reach Curing Level 4. Material's Curing Level at Ejection

Unfilled Volume Plot

A new result plot, **Unfilled Volume**, is available for Fill simulations when a short-shot occurs.

The **Unfilled Volume** plot helps you to visualize regions of the model that remain unfilled because of a short-shot during filling.

Venting Analysis



You can specify **Air Vent** boundary conditions on model edges.

In earlier releases, you could specify air vents for Venting analysis on vertices only. With the addition of edge-based air vents, you can capture more realistically the behavior of actual mold vents. You can assign the **Air Vents** boundary condition to **Cavity** and **Cold Runner System** domains.

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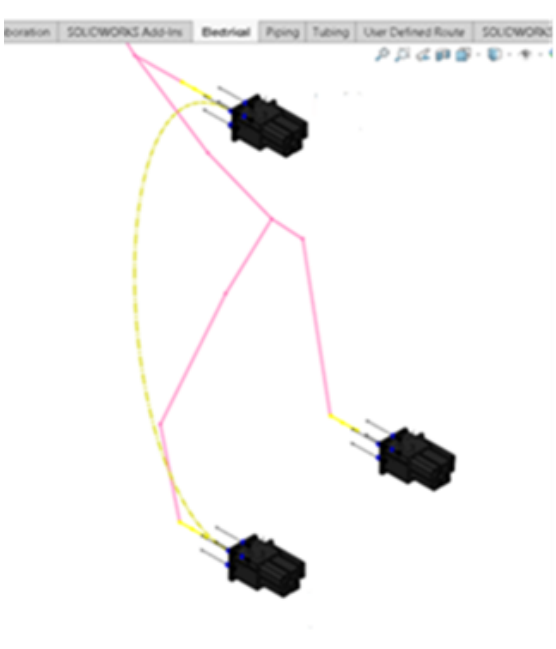

Routing

This chapter includes the following topics:

- **Redirecting Guidelines to Follow a Route Path**
- **Managing a List of Favorites for Coverings**
- **Connector Table Enhancements**
- **Automatically Scaling Drawings to New Sheet Formats**

Routing is available in SOLIDWORKS® Premium and SOLIDWORKS Ultimate.

Redirecting Guidelines to Follow a Route Path

	
<p>Automatically generated guidelines following the shortest possible path between end connectors</p>	<p>Redirected guidelines following a route path</p>

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

Benefits: Redirecting guidelines to follow a route path helps minimize interference with other components and reduces manual adjustments, making routing faster and more efficient.

To redirect guidelines to follow a route path:

1. Create a 2D or 3D sketch that leads to the appropriate end connector.
2. Click **Tools > Routing > Electrical > Routing > Auto Route**.
3. Under **Routing Mode**, select **Guidelines**.
4. Select **Follow Routing Path**.

To ensure accurate results, use **Follow Routing Path** before merging wires.

5. In the graphics area, select sketches to use as route paths.

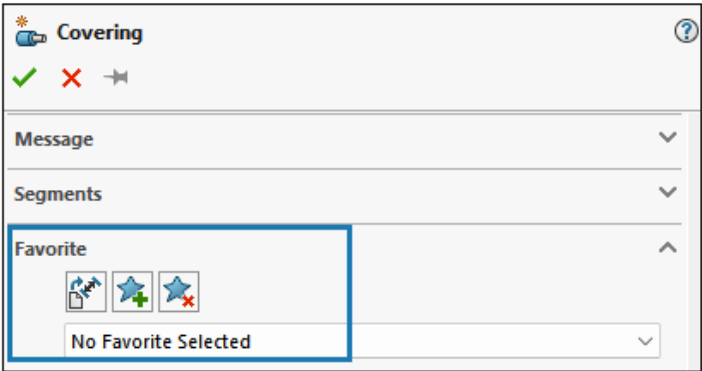
A preview of the guidelines, aligned with the selected sketch, displays in the graphics area.

6. Click **Done**.

The sketches representing the route path appear in the FeatureManager® design tree as **Routing_pathn**

7. Click .

Managing a List of Favorites for Coverings







You can save commonly used or standardized multilayer coverings as Favorites to reuse them in models. This helps you manage a list of preferred coverings for future use.

These favorites save all the covering PropertyManager parameters that you can access while working on other pipe segments.

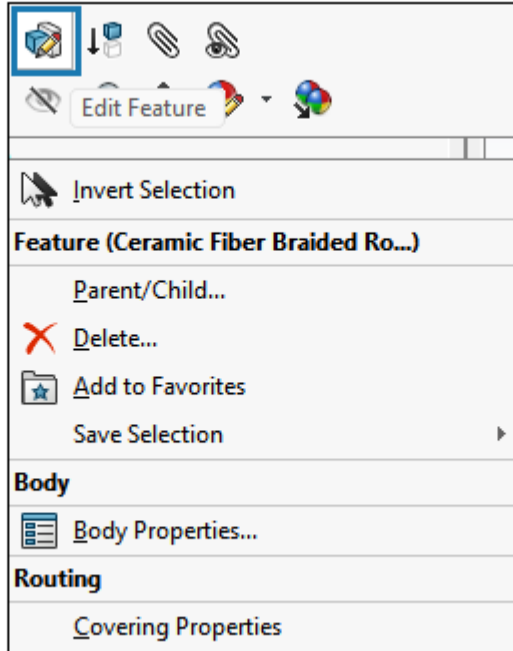
Benefits: This section enhances user efficiency, improves time management by providing quick access to commonly used covering style.

To manage Favorites, in the **Favorite** section of the Covering dialog box, specify options as described in the following table:

Option	Description
 Apply Default/No Favorite	Resets to No Favorite selected and the default settings.
 Add Favorite	Adds the selected covering to the Favorite list. <ul style="list-style-type: none">To add a style, click , enter a name, and then click OK.
 Delete Favorite	Deletes the selected favorite.
Favorites list	Lets you select the saved favorite styles.

This supports for piping and tubing coverings but not electrical covering.


Editing the Covering Element



You can edit the covering elements of a pipe route directly in the FeatureManager design tree.

Benefits: This feature streamlines the designing process by minimizing the number of clicks, saving time and effort.

To edit the covering element:

1. Right-click the covering element in the FeatureManager design tree.
2. Select **Edit Feature** .
3. In the **Covering Layers** section of the Covering PropertyManager, specify the required parameters.
4. Click **Apply**.

Connector Table Enhancements

You can insert and manage connector tables in a more intuitive and efficient way.

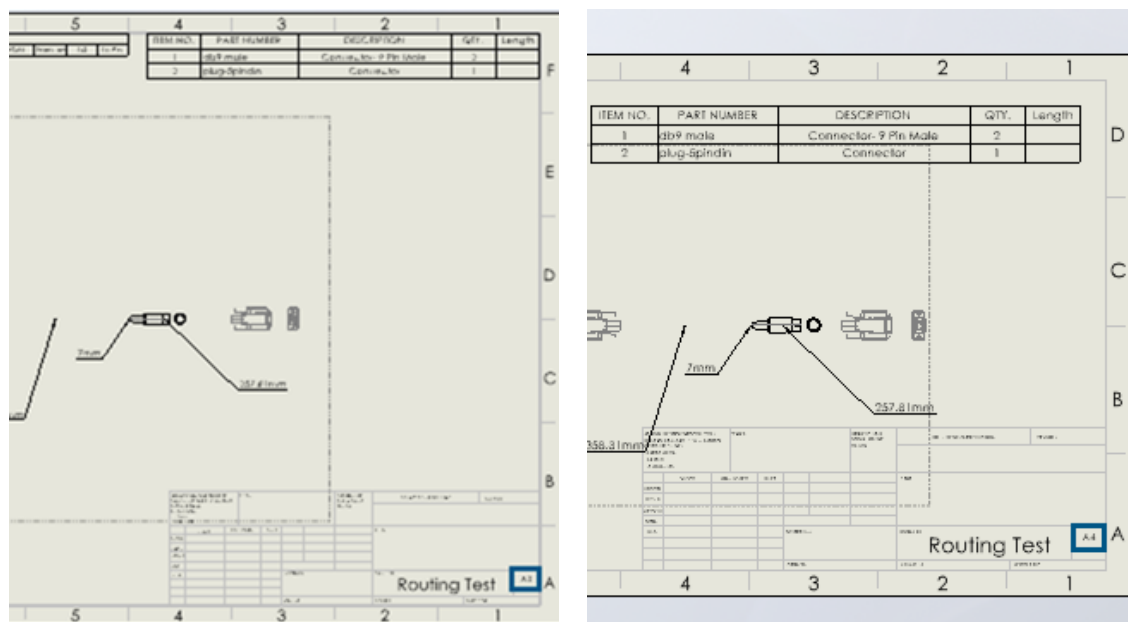
Benefits: These enhancements simplify connector table management. With more informative table names and batch selection tools, you spend less time sorting through connectors.

The Connector Table PropertyManager shows a clearer list of connectors. This list shows the component reference for each connector and helps in reducing errors when inserting the reference.

To help you identify a connector, SOLIDWORKS Routing includes component reference information directly in the name of the connector table. Instead of using generic names like **Connector Table 1**, the updated naming follows a more descriptive format, such as **Connector Table 3Pin<4>**. This format matches what you see in the FeatureManager design tree. It helps you to associate each table with its corresponding connector.

You no longer need to insert tables one by one. You can select and insert multiple connector tables at once, using multi-selection shortcuts in the PropertyManager such as **Ctrl+Click** or **Shift+Click**. You can also remove multiple connectors from the list in a single step.

Automatically Scaling Drawings to New Sheet Formats



A flattened drawing of a routing assembly The same flattened drawing, automatically scaled to a new sheet format

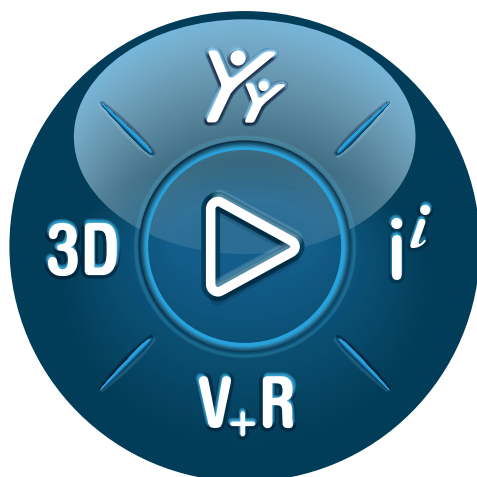
When you change the sheet format template of a flattened drawing, each element automatically adjusts its scale and position to fit within the new sheet size.

For example, the drawing in the table above remains centered after the sheet format changes from **A3 Landscape** to **A4 Landscape**.

Benefits: Automatic scaling ensures that each element of your drawing adapts to a new sheet format correctly, saving time and reducing errors.

Automatic scaling applies to each element of the flattened drawing, including:

- Drawing views
- Electrical tables
- Annotations
- Connector blocks



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France

Asia-Pacific

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

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