WHAT'S NEW

SOLIDWORKS 2026

Contents

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Welcome to SOLIDWORKS Design 2026

This chapter includes the following topics:

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 Electrical Schematic
- Continue to design anywhere with the latest in browser-based product development on the **3D**EXPERIENCE® platform

This document covers all enhancements that affect how you interact with the **3D**EXPERIENCE platform in SOLIDWORKS Design. It also includes other apps that can connect to the platform such as DraftSight.

Top Enhancements

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

Fundamentals	**SOLIDWORKS Appearances** on page 40
Parts and Features	**Creating Reference Points by XYZ Values** on page 55
Structure System and Weldments	**Enhanced Corner Treatments** on page 64
Detailing and Drawings	**Linking Bill of Materials Columns to PLM Attributes (2026 SP1/FD01)** on page 19 **Automatically Generating Drawings (BETA): Section Views and Hole Callouts** on page 77
Configurations	**Splitting Out Configurations Into Individual Files** on page 84
Import/Export	**Selecting Bodies and Features of Multibody Parts** on page 57

Performance

The current release improves the performance of specific tools and workflows.

Some of the highlights for performance and workflow improvements are:

SOLIDWORKS Design

(2026 SP1/FD01)

Rendering performance is improved for part views with highlighted edges (**Wireframe**, **HLR/HLV**, and **Shaded With Edges**) when you select features and bodies.

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 Design:

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 Design, appears next to new menu items and the titles of new or significantly changed PropertyManagers. Click to display the topic in this guide that describes the enhancement. To enable Interactive What's New, click > **What's New** > **Interactive**.
Online Help	Contains complete coverage of our products, including details about the user interface and examples.
SOLIDWORKS User Forum	Contains posts from the SOLIDWORKS user community on the **3D**EXPERIENCE® platform (login required).
Release Notes	Provides information about late changes to our products, including changes to the *What's New* book, online help, and other documentation.
Legal Notices	SOLIDWORKS Legal Notices are available **online**.

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Using SOLIDWORKS Design on the 3DEXPERIENCE Platform

This chapter includes the following topics:

- **SP1 and FD01**
- **SP0 and GA**

This chapter covers all enhancements that affect how you use SOLIDWORKS® Design when you are connected to the **3D**EXPERIENCE® platform. Unless otherwise noted, the information in this chapter applies to both SOLIDWORKS Design installed from the **3D**EXPERIENCE platform and to SOLIDWORKS Design with the Design with SOLIDWORKS (**3D**EXPERIENCE) add-in.

SP1 and FD01

Applying a Unified Date Format (2026 SP1/FD01)

3DEXPERIENCE users can apply a single uniform date format across SOLIDWORKS Design and **3D**EXPERIENCE apps from Preferences in the platform. When you select **ISO 8601 (YYY-MM-DD)**, all `$PLMPRP` date properties in SOLIDWORKS Design display in the same format.

Benefits: If you do not apply the ISO 8601 format, you can still manage date formats through drawing templates or Windows regional settings. Applying the ISO 8601 format ensures that dates appear consistently across all users and regions. This standardized format removes regional differences, improves communication across global teams, and keeps data consistent.

The platform saves all dates in GMT (UTC) time and displays them in your local time zone. Depending on the region, this date can shift by one day. For example, a model created in France on 2025-04-13, shows the same date when viewed in India.

To apply a unified date format:

1. In the platform, on the top bar of 3D Dashboard 3DS Home, click your profile icon and select **Preferences** > **Language**.
2. Under **Region**, select **ISO 8601 (YYYY-MM-DD)**.

The format applies to the following `$PLMPRP` properties:

- Standard properties such as `$PLMPRP:"created"` and `$PLMPRP:"modified"`.

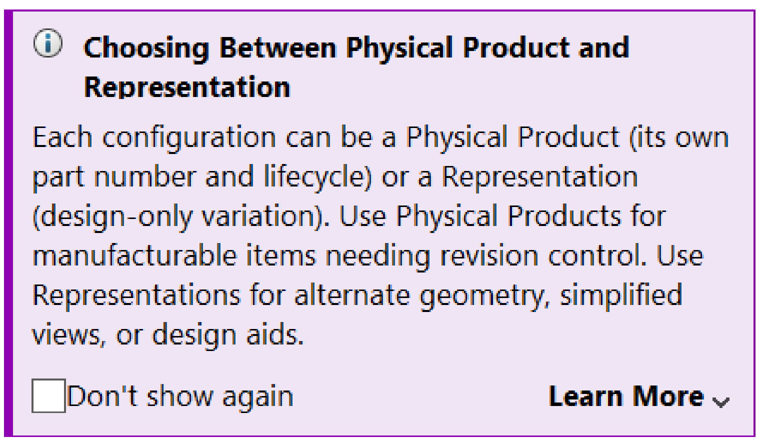
- Additional properties such as `$PLMPRP:"ea_changedstatusdate"` and `$PLAMPRP:"ea_releasestatusdate"` (including 0.1, 0.2, and other indexed values).
- Custom mapped properties such as `$PLMPRP` values, such as `$PLMPRP:"XP_VPMReference_Ext.Start_Day"`.

Starting with SOLIDWORKS Design 2026 SP1, dates in drawing title blocks, tables, and **3D**EXPERIENCE revision tables also follow the ISO 8601 format.

Limitations:

- Applies only to `$PLMPRP date properties`, not native SOLIDWORKS date fields.
- Works only when you select **ISO 8601** in **Preferences**. Other selections revert to the Windows local format.
- Does not apply to `PDF` generation.
- If the local cache includes old date mappings, use **MySession** > **Refresh** to update them to the ISO 8601 format.

Assistant Notifications (2026 SP1/FD01)



For **3D**EXPERIENCE users, Assistant Notifications appear in the graphics area to help guide you when you first use various functionalities. You can quickly learn the functionality using these guided contextual notifications.

Available Assistant Notifications can help with:

- Configurations
- Opening filters created in the Product Structure Explorer app
- Pattern-driven component patterns

Assistant Notifications provide information about the specific action that you are doing in the app. For example, if you create a new configuration, the Assistant Notification appears with helpful information about that action. If you are creating patterns in assemblies, the Assistant Notification appears with guiding information. Assistant Notifications appear only the first few times that you use a pertinent functionality.

You can:

- Expand and collapse sections. Click **Learn More** to display more or less content.
- Drag the Assistant Notification to reposition it.
- Dismiss the notification. Click **Don't show again** so it does not reappear. To reactivate the dismissed message, see **Tools** > **Options** > **System Options** > **Messages/Errors/Warnings** > **Dismissed Messages**.

Automatically Replicating Windows Folder Structure for Bookmarks (3DEXPERIENCE Users Only) (2026 SP1/FD01)

Users who install DraftSight from the **3D**EXPERIENCE platform can use **Batch Save to 3DEXPERIENCE** to automatically create a bookmark structure that replicates the Windows folder structure.

To automatically replicate the Windows folder structure for bookmarks:

1. On the ribbon, click **DraftSight** > **Batch Save to 3DEXPERIENCE**.
2. In the Batch Save to 3DEXPERIENCE dialog box, click **Add Folder**.
3. Select a folder to upload.
4. In the Batch Save to 3DEXPERIENCE dialog box, click **Bookmark**.
5. In the Select a Bookmark dialog box, select the bookmark to which you want to upload the folder and click **Select**.
6. In the Batch Save to 3DEXPERIENCE dialog box, click **Save**.

DraftSight uploads all subfolders and `DWG` files of the selected folder to the bookmark in the same hierarchy as the Windows folder structure.

If the folder or subfolders include hidden files, the upload process stops and you get an error message.

Deleting Remote and Local Files from the 3DEXPERIENCE Files on This PC Tab (2026 SP1/FD01)

When you select files to delete from the 3DEXPERIENCE Files on This PC tab, SOLIDWORKS Design deletes files in both your local and remote work folders. The option, **Delete files outside the cache folder from the original location (Recommended),** helps you avoid creating duplicate files on the **3D**EXPERIENCE platform.

You can choose to:

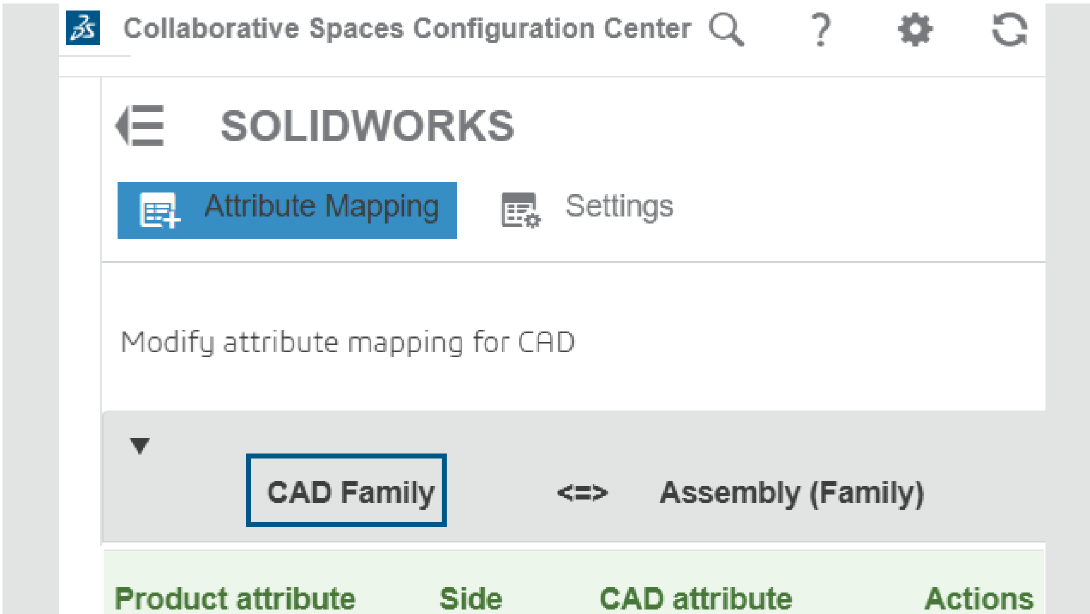
- Delete only the unmodified files and keep the modified ones
- Delete all files, including the modified ones

Benefits: You can delete local and remote files in a single step. It makes file clean-up faster and more reliable.

The same behavior is available in the following:

- **Cleanup** tool in the 3DEXPERIENCE Files on This PC tab
- Choose a Delete Option dialog box

Deprecated CAD Family Attribute Mapping (2026 SP1/FD01)



For **3D**EXPERIENCE users, CAD Family attribute mapping is removed and no longer available in the administration tool. This applies to new tenants installed after SW2026x GA.

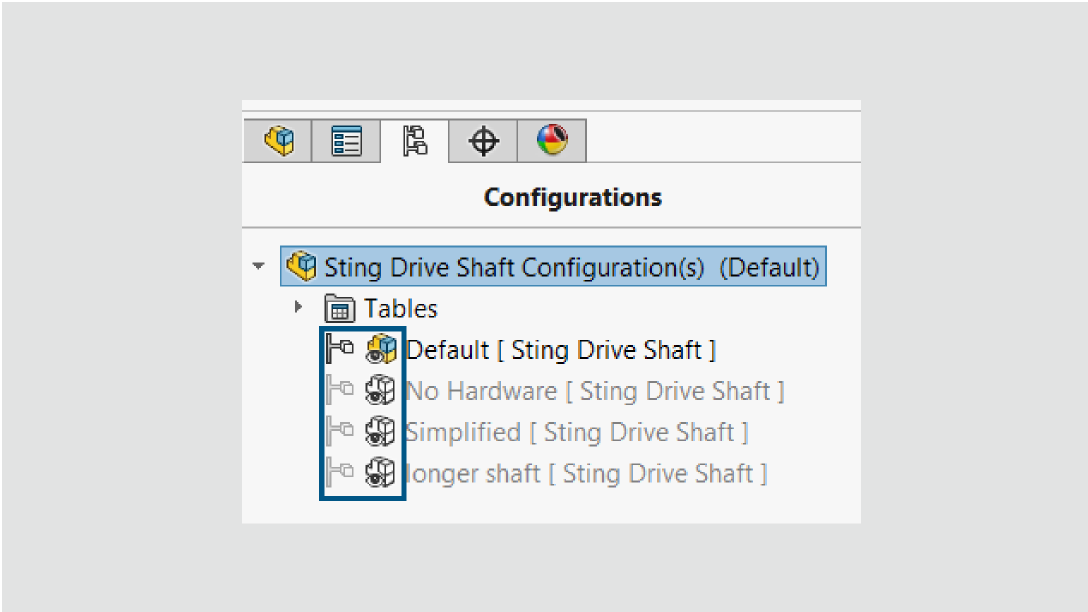
This attribute mapping has limited value and could cause confusion. Removing this mapping increases performance.

CAD Family attribute mapping in legacy on the cloud tenants is still supported but removed for legacy on-premises tenants.

Recommendation: Delete the attribute mapping definition on CAD Family.

To access attribute mapping, go to **Collaborative Spaces Configuration Center** > **CAD Collaboration** > **SOLIDWORKS** > **Attribute Mapping**.

Display Data Marks (2026 SP1/FD01)



In the ConfigurationManager for physical products or representations, **3D**EXPERIENCE users can use the **Display Data Mark** menu commands to add or remove Display Data marks on configurations. The Display Data marks that you specify persist after you update models for **3D**EXPERIENCE compatibility.

3DEXPERIENCE users can see the Display Data Mark state and change it. This is especially important for working with Large Design Review, SOLIDWORKS Visualize, and eDrawings previews in PDM.

This functionality applies to newly updated files or legacy files that were already updated for **3D**EXPERIENCE compatibility. Previously, Display Data marks were unavailable after you updated a file for **3D**EXPERIENCE compatibility.

To add or remove Display Data Marks:

1. In the ConfigurationManager, right-click a configuration and click **Display Data Mark**.
2. Specify an option:
   - **Add Mark for This Configuration**
   - **Add Mark for All Physical Product Configurations**
   - **Add Mark for All Representation Configurations**
   - **Add Mark for Specified Configurations**
   - **Remove Mark and Purge Data for All Configurations**

   If you modify the Display Data mark for a configuration of an assembly component and save the model, the software tracks this as a cosmetic change for the parent assembly.

Displaying Simulation and Configuration Tabs in the Action Bar (2026 SP1/FD01)

By default, the Simulation and Configuration tabs in the MySession action bar are hidden. You can show either or both tabs when using these tools.

Benefits: This enhancement helps keep the MySession interface clean by displaying only the tabs that are relevant to your work.

To display these tabs:

1. In the action bar, click **Tools** > **Options** > **MySession**.
2. Under the Action Bar section of the dialog box, select one or both of the following options:
   - **Show Simulation tab.**
   - **Show Configuration tab.**
3. Click **OK**.
4. Restart SOLIDWORKS Design for the change to take effect.

Email Reminder for Platform Activation (2026 SP1/FD01)

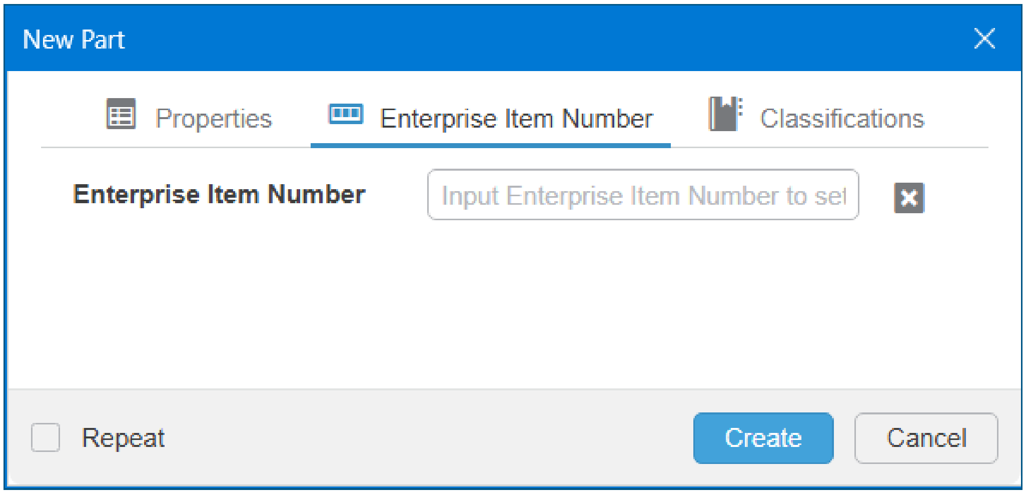
Administrators who manage multiple User Entitlement Service (UES) licenses receive an Email Reminder in the SOLIDWORKS MarketPlace add-in. The reminder prompts them to activate their licenses if they missed the original activation email.

Benefits: This helps administrators activate multiple UES licenses for the **3D**EXPERIENCE platform directly from SOLIDWORKS Design.

When administrators sign in to SOLIDWORKS Design, the platform checks whether their site ID owns UES licenses and whether they have administrative privileges in the DSx Client.

If they meet both conditions, the Email Reminder appears in the SOLIDWORKS Marketplace add-in, accessible from the SOLIDWORKS Task Pane. The reminder also includes a link to a Help topic that explains how to activate the platform.

Enterprise Item Number Tab (2026 SP1/FD01)



When **3D**EXPERIENCE users create new parts or assemblies on the platform, the platform's New dialog box displays the Enterprise Item Number (EIN) tab. You can enter an EIN value.

Benefits: You have an additional location where you can enter the EIN value.

To access the Enterprise Item Number tab in SOLIDWORKS Design:

1. Click **File** > **New**.
2. In the New SOLIDWORKS Document dialog box, on the 3DEXPERIENCE tab, select **Part** or **Assembly** and click **OK**.

   The platform's New dialog box appears with the Enterprise Item Number tab.
3. Enter a value for **Enterprise Item Number** and click **Create**.

Improved Handling of Multi-Valuated Attribute Mappings (2026 SP1/FD01)

Administrators can identify and manage legacy CAD-to-platform mappings linked to Multi-Valuated Attributes (MVAs). When such mappings exist, SOLIDWORKS Design warns users during the save process to indicate incompatible mappings.

Benefits: This helps administrators maintain clean data and ensures that property mappings remain consistent between SOLIDWORKS Design and the **3D**EXPERIENCE platform.

An is a **3D**EXPERIENCE attribute that can store multiple values for one property, such as several countries or codes. Because SOLIDWORKS Design properties hold only a single value, mapping them to MVA attributes can cause data conflicts or information loss.

This behavior applies to legacy platform configurations (26xGA or earlier) that still allow **CAD-to-platform** mappings. Administrators can manage these mappings in the **Platform Manager** > **Collaborative Spaces Configuration Center**. From there, they can remove or re-create legacy **CAD-to-platform** mappings in the supported direction (**platform-to-CAD**) to avoid compatibility issues.

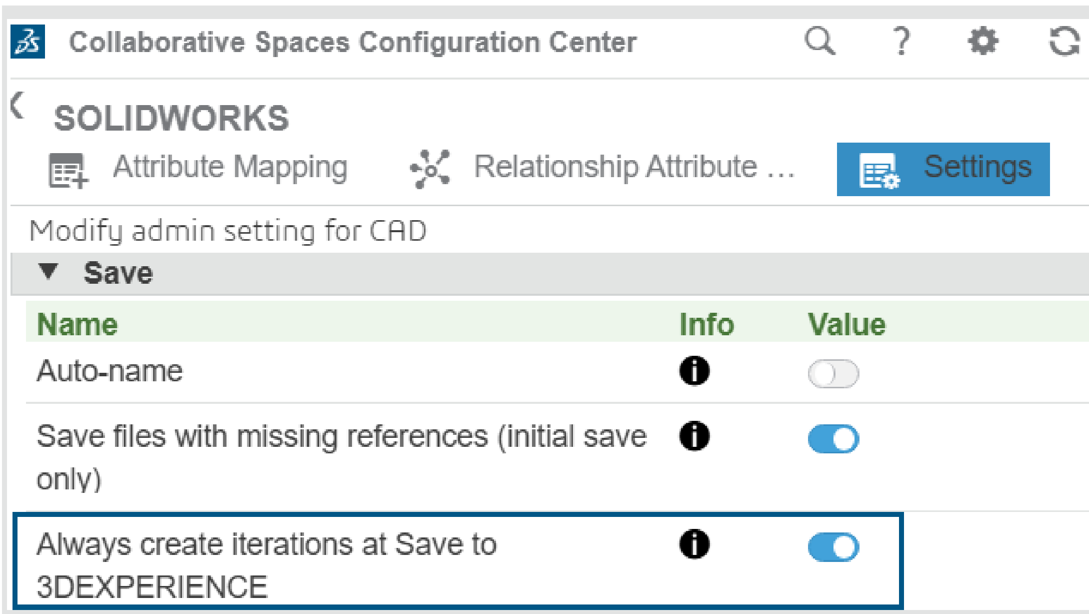
This adds the following behaviors for legacy mappings that use MVA attributes:

- When a CAD-to-platform MVA mapping exists, SOLIDWORKS Design displays a warning during **Save with Options**.
- In **MySession** > **Properties**, MVA-mapped attributes appear in read-only mode and show concatenated multivalues separated by commas.
- Administrators cannot create new CAD-to-platform MVA mappings.
- If an MVA value contains commas, the concatenation can appear ambiguous (for example, the string `3,14,0,693,-1` may represent `"3,14,"` `"0,693,"` and `"-1"`).

This does not affect performance during **Save with Options**.

CAD Family mappings are no longer supported and are ignored if present.

Keep Previous Iteration Option (2026 SP1/FD01)



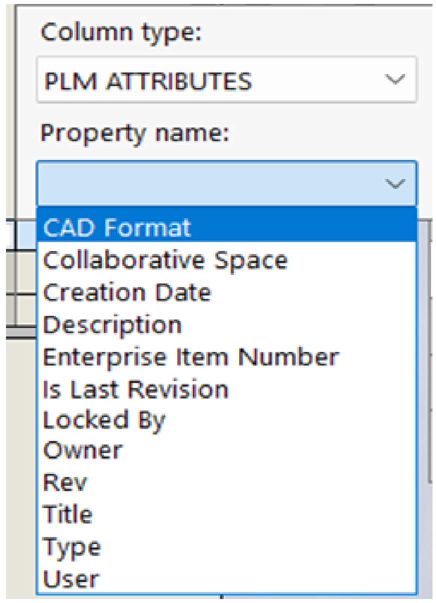
3DEXPERIENCE users can enable an option to always enable the **Keep previous iteration** option in the Save to 3DEXPERIENCE dialog box.

You no longer need to manually select **Keep previous iteration** each time to create a new iteration. This prevents potential data loss.

To enable this option:

1. On the platform, go to **Collaborative Spaces Configuration Center** > **CAD Collaboration** > **SOLIDWORKS** > **Settings**.
2. Under **Save**, select **Always create iterations at Save to 3DEXPERIENCE**.

Linking Bill of Materials Columns to PLM Attributes (2026 SP1/FD01)



3DEXPERIENCE users can link bill of materials columns to PLM attributes from the platform. The PLM attributes can be both default attributes and custom attributes.

This functionality is also supported for family tables.

> All columns linked to PLM attributes are read-only, but you can edit the column names.

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

To link BOM columns to PLM attributes:

1. In a drawing saved to the platform, start to insert a BOM into the drawing.
2. In the BOM, insert a new column, then double-click the top of the column header.

   The **Column type** dialog box appears.
3. In **Column type**, select **PLM Attributes**.
4. In **Property name**, select a PLM attribute to apply to that column.

   > 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 reconnect to the **3D**EXPERIENCE platform and rebuild the file, SOLIDWORKS Design updates the property.

   > When you right-click a PLM attribute column, the options **Merge cells** and **Edit multiple property values** are not available.

Opening Components from the Platform (2026 SP1/FD01)

For **3D**EXPERIENCE users, the performance of opening components from the platform is improved.

The platform components are built dynamically and appear in the FeatureManager design tree. This enhancement requires that the models you open were saved to the platform with the same version of SOLIDWORKS Design that you are using.

The software saves the SOLIDWORKS Design components that you create back to the platform as derived output for all non-SOLIDWORKS Design components, provided they have either ExactGeometry or STEP derived output. When you subsequently open these components, they do not require a rebuild.

> Components with non-SOLIDWORKS Design data that do not have ExactGeometry or STEP derived output are not saved to the platform. These components require conversion to SOLIDWORKS Design data each time you open them.

Opening Filters from the Open Dialog Box (2026 SP1/FD01)

3DEXPERIENCE users can search and open filters that they created in the **Product Structure Explorer** from the Open dialog box.

Benefits: You can open assemblies faster and work on a specific subset of components without opening the entire assembly.

In earlier releases, you had to drag filters from **MySession** to open them.

To open filters from the Open dialog box:

1. Open the Open dialog box on the **3D**EXPERIENCE platform by doing one of the following.
   - Click **Open** (Standard toolbar).
   - Click **File** > **Open**.
2. On the 3DSearch tab in the left pane, search for a filter.

3. Double-click the filter.

A message notifies you to create a filter in the **Product Structure Explorer** for the current assembly. The message appears when a large assembly requires 3 minutes or more to open. Filters from the Product Structure Explorer let you open the same assembly faster next time.

For more information about Product Structure Explorer, see **Product Structure Explorer**.

Opening Latest Revision of References (2026 SP1/FD01)

You can open a document in SOLIDWORKS Design with the latest revision of its first-level references automatically. The app ensures that parts, assemblies, drawings always use the most current **In Work** data from the **3D**EXPERIENCE platform.

Benefits: This saves time by eliminating the need to manually update reference revisions when opening SOLIDWORKS Design documents from the platform.

Administrators must activate the global server setting in **Global Availability** > **Flexibility to update product instance reference at Open** on the platform. After the setting is active, you can select **Open Latest revision of the first-level references** in the MySession action bar under **Tools** > **Options** > **Open**.

When selected, SOLIDWORKS Design automatically retrieves the latest revision of each first-level reference during any open action, including **Open**, **Open from 3DEXPERIENCE**, **Drag and Drop**, and **Reload from Server**.

For drawings, a dialog box appears when you open the document. You can update only the referenced parts and assemblies in the drawing or update the next-level references used in its views. If the first-level references are not **In-Work**, SOLIDWORKS Design notifies you that they cannot be updated.

This functionality only applies to documents **In Work** status. It does not modify **Released** or **Frozen** documents. It applies only to native SOLIDWORKS Design models and does not include **Assembly in Part** relationships or lightweight viewing modes, such as **LDR**, **Quick View**, or **Detailing**.

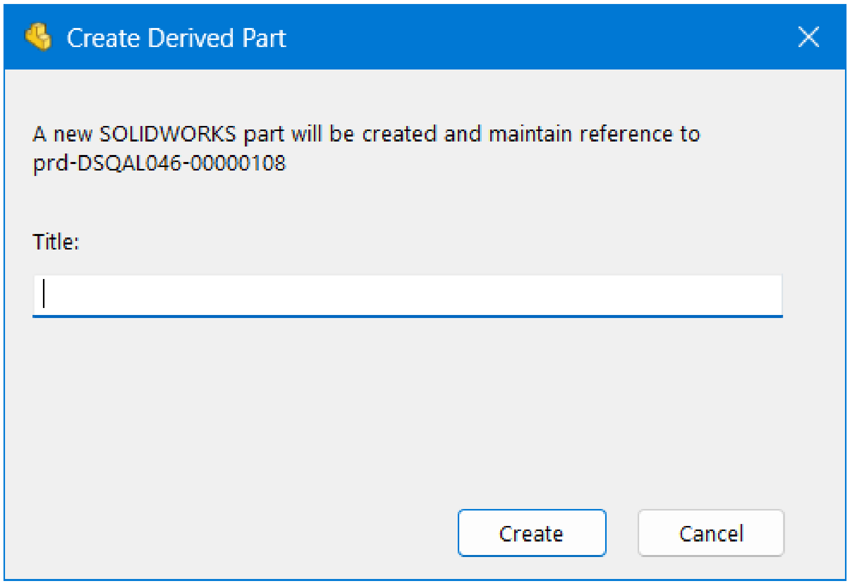
Running Collaboration Workflows from the 3DEXPERIENCE User Interface (2026 SP1/FD01)

3DEXPERIENCE users can run all ECAD-MCAD collaboration workflows (**Push**, **Pull**, **Refresh Connection**, **Update Status**, and **Build Model**) directly from the CircuitWorks collaboration task pane within **3D**EXPERIENCE user interface . This eliminates the need to switch to the CircuitWorks user interface.

Benefits: All collaboration actions are available in one place, saving time and keeping you focused on the design.

SP0 and GA

3DEXPERIENCE Components in SOLIDWORKS Design



The workflow for handling **3D**EXPERIENCE components in SOLIDWORKS Design is optimized.

Inserting 3DEXPERIENCE Components into SOLIDWORKS Assemblies

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

Creating Derived Parts in SOLIDWORKS Assemblies

After you insert a **3D**EXPERIENCE 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

2025	2026
	3DEXPERIENCE platform whenever you want after you create the derived part.

Editing 3DEXPERIENCE Components in SOLIDWORKS Assemblies

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

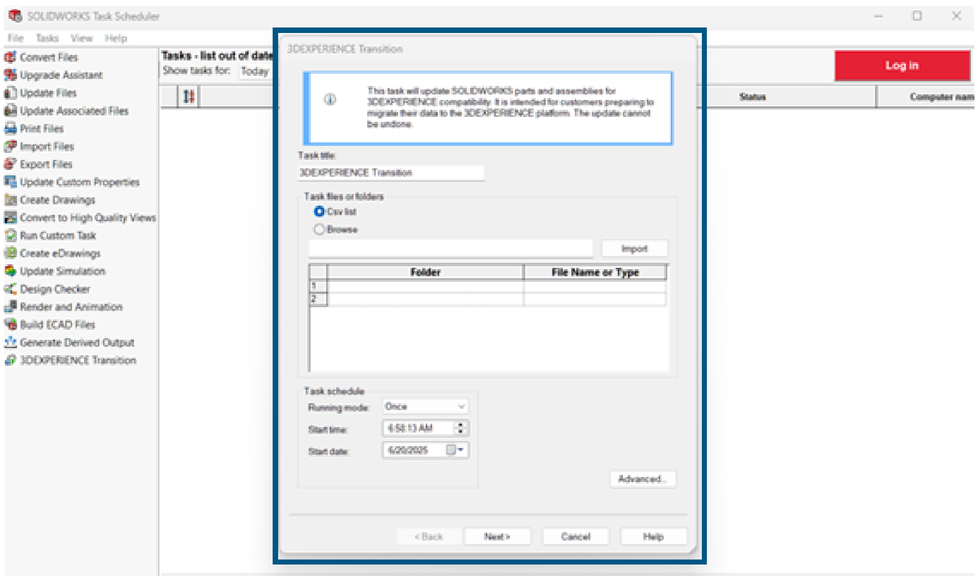
2025	2026
You can edit the component but you cannot save the changes.	The Create Derived Part dialog box appears. You can create the derived part and make changes to it.

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 **3D**EXPERIENCE component into that part.

3DEXPERIENCE Transition Task in SOLIDWORKS Task Scheduler



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

The **3D**EXPERIENCE Transition Task replaces the **3D**EXPERIENCE Compatibility Task.

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

With the **3D**EXPERIENCE Transition task, you can:

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

Creating a 3DEXPERIENCE Transition Task

To create a 3DEXPERIENCE Transition task:

1. In SOLIDWORKS Task Scheduler, click **3DEXPERIENCE Transition**.
2. Under **Task title**, create a name for your task.
3. Under **Task files or folders**, select the content you want to update by doing one of the following:

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

> The format of the `.csv` file is *`path`*, *`filename`*. For example to add `clamp.sldprt` and `bracket.sldrpt`, write:
>
> - `"C:\Users\Public\Documents\SOLIDWORKS\SOLIDWORKS 2025\samples\tutorial\assemblymates","clamp.sldprt"`
> - `"C:\Users\Public\Documents\SOLIDWORKS\SOLIDWORKS 2025\samples\tutorial\assemblymates","bracket.sldprt"`

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

Option	Description
Configuration option	Saves only the active configuration or activates all configurations before saving. Activating all configurations before saving can add significant time to the task.
3DEXPERIENCE Compatibility	Updates SOLIDWORKS content for compatibility with the **3D**EXPERIENCE platform. See **3D**EXPERIENCE Compatibility and **3D**EXPERIENCE Integration Options in *SOLIDWORKS Design*.
File Upgrade Settings	• Upgrades custom properties. • Adds rebuild mark to all configurations. • Adds display data mark to all configurations. **Add display data mark to all configurations** is unavailable if you selected **3D**EXPERIENCE Compatibility.
Backup Files	Specifies the location to back up the updated files.

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

Running a Macro with the 3DEXPERIENCE Transition Task

To run a macro with the 3DEXPERIENCE Transition task:

1. In the **3D**EXPERIENCE 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`).

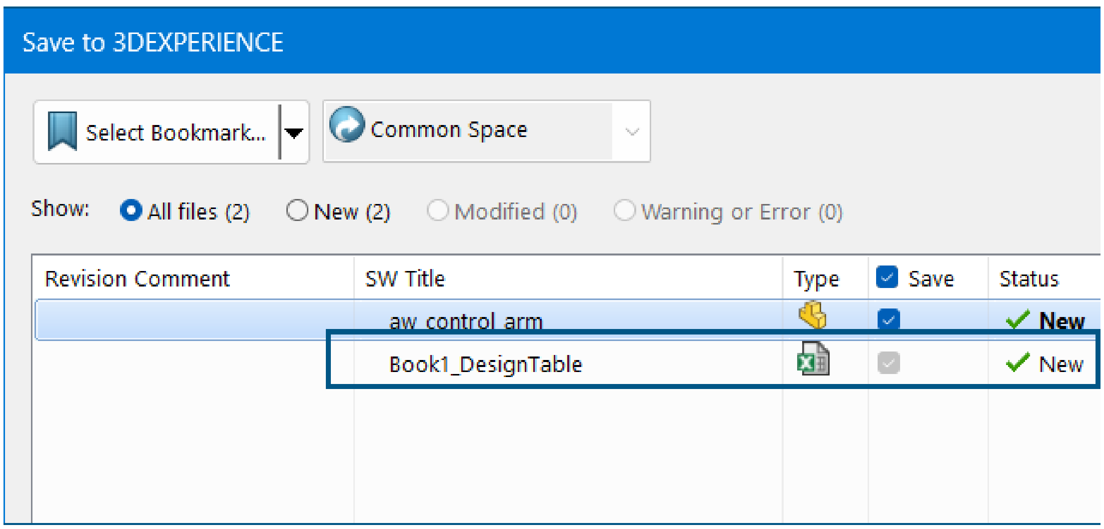
This sample macro adds a property named "Hello" with a value of "Hello World" to any part, assembly, or drawing in the list of task files.

- 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.

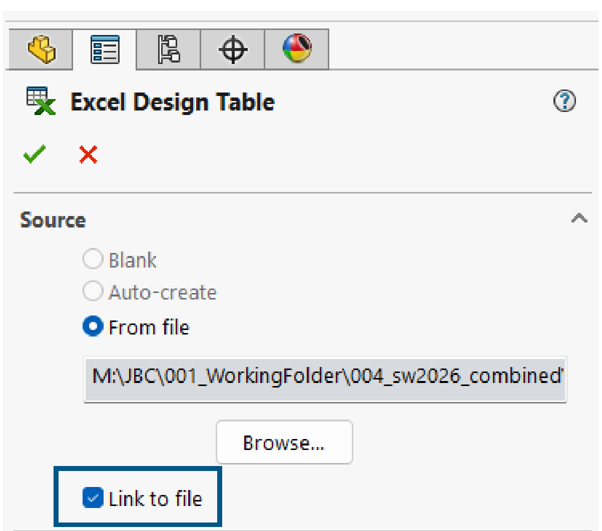
```
Dim swApp As SldWorks.SldWorks
Dim swModel As SldWorks.ModelDoc2
Dim config As SldWorks.Configuration
Dim cusPropMgr As SldWorks.CustomPropertyManager
Dim lRetVal As Long
Dim boolstatus As Boolean
Dim longstatus As Long, longwarnings As Long

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
```

Support for Link to File With Design Tables



For models with design tables that are linked to Excel files with the **Link to file** option, when **3D**EXPERIENCE users save the models to the platform, the linked relationship is saved to the platform. When you open the saved model from the platform, the linked design table is included.



Benefits: Different team members can access the linked design table that is saved with models to the platform.

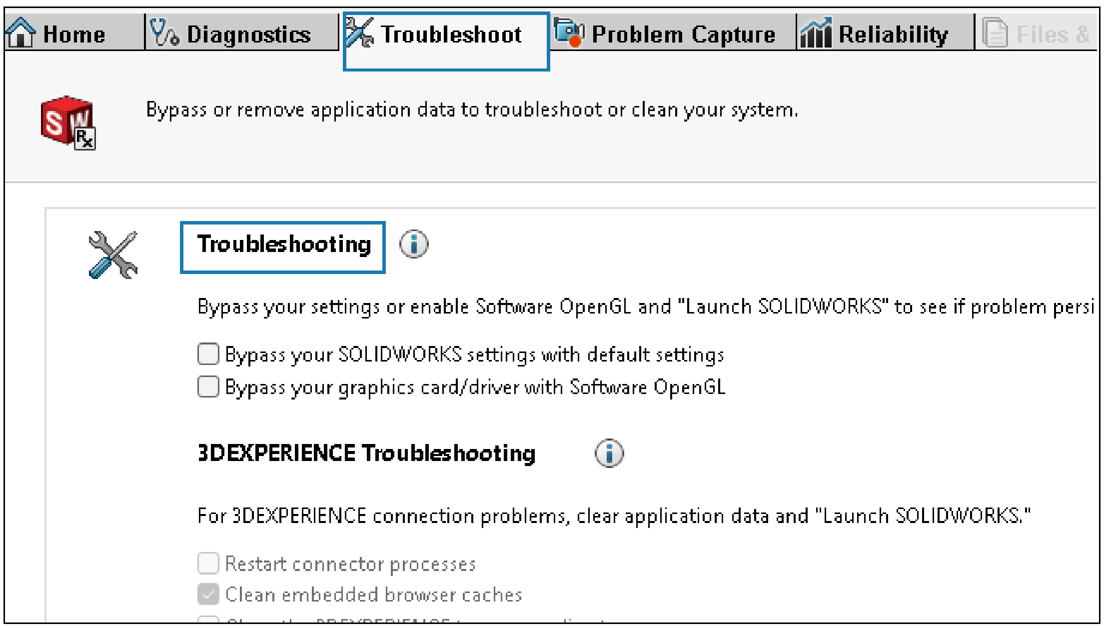
The model on the platform retains all the design table relationships to the model's configurations. When you open the model from the platform, to modify the design table in the ConfigurationManager, right-click the design table and click **Edit Table**. When you save the model to the platform again, the dialog box indicates the design table **Status** as **Modified**.

3

Administration

This chapter includes the following topics:

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.

All users of SOLIDWORKS® Design can open SOLIDWORKS Rx from **SOLIDWORKS Tools**in the Windows **Start** menu.

SOLIDWORKS Rx includes a **Benchmark** tab for performance evaluations.

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

1. **Close Connector processes**. Closes down the connector processes `ENOPLMCSAClient.exe` and `EdmServerV6.exe`, and restarts the **3D**EXPERIENCE 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 **3D**EXPERIENCE Task Pane.
3. **Clear the 3DEXPERIENCE temporary directory**. Clears settings and cached data used by SOLIDWORKS Design and other **3D**EXPERIENCE apps to customize and accelerate interactions with the **3D**EXPERIENCE platform. The temporary directory is located at `%localappdata%\DassaultSystemes\CATTemp`.

Bookmark Support in the Settings Administrator Tool (2026 SP1/FD01)

Administrators can add and manage **3D**EXPERIENCE bookmarks as file locations in the Settings Administrator Tool.

In earlier releases, administrators defined bookmarks in SOLIDWORKS Design and imported them into the tool using an exported `.sldreg` or `.sldsettings` file.

Benefits: This enhancement gives administrators more flexibility to manage file locations linked to the **3D**EXPERIENCE platform. They can add, lock, and synchronize bookmarks directly in the tool without manually exporting or importing these settings.

The capabilities include:

- Lock bookmarks to prevent user changes.
- Choose between automatic and manual synchronization of bookmarks.
- If multiple tenants are available, select bookmarks only from the same tenant. You cannot combine bookmarks from different tenants..

  This functionality is available only to **3D**EXPERIENCE users.

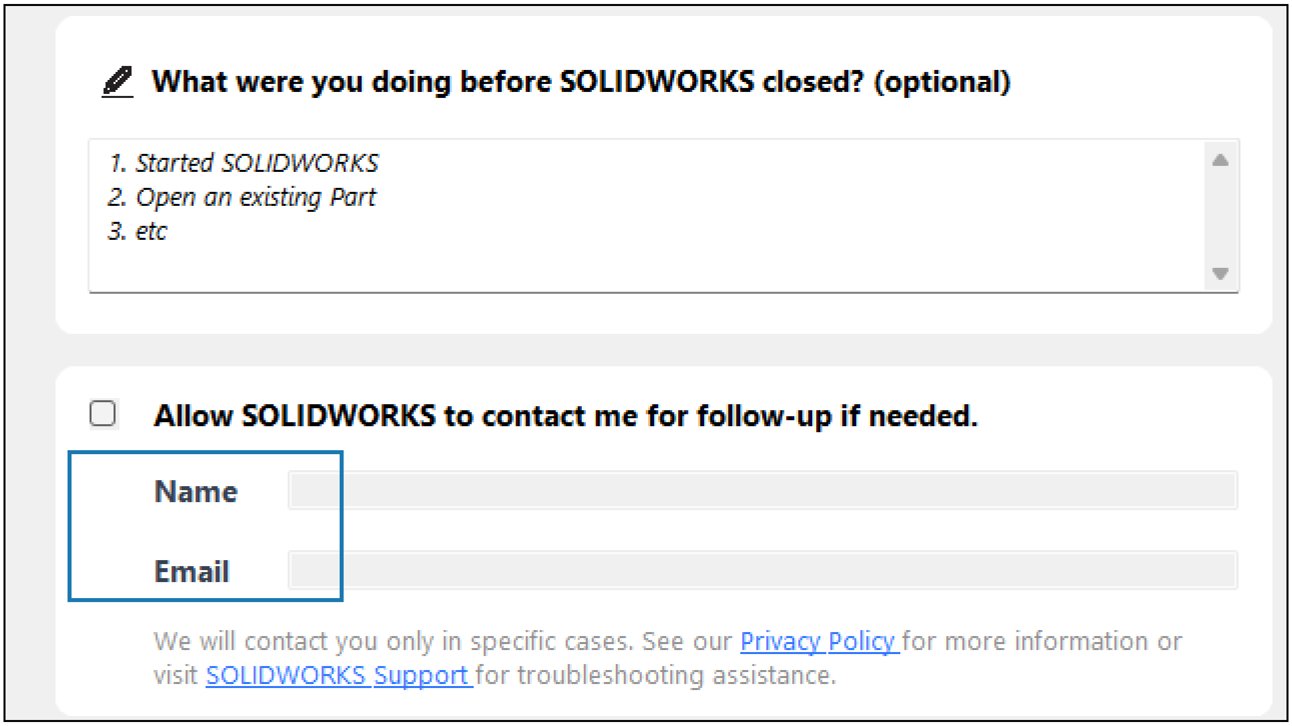
To add and manage bookmark file locations:

1. Launch the Settings Administrator Tool in one of the following ways:
   - From the Administration Image Option Editor after creating an administrative image.
   - By opening an existing `.sldreg` or `.sldsettings` file to review or edit its settings.
2. In the SOLIDWORKS Settings Administration dialog box, under **System Options** > **File Locations**, click **Add**.
3. In the Choose Folder dialog box, click **Select from 3DEXPERIENCE** to access a list of bookmarks.

   If prompted, you may need to sign in with your **3D**EXPERIENCE credentials.
4. In the Select a Bookmark dialog box, choose a bookmark and click **Select**.
5. Next to **Bookmark File Locations**, click **Lock** to apply or restrict changes for users when the settings file is deployed.

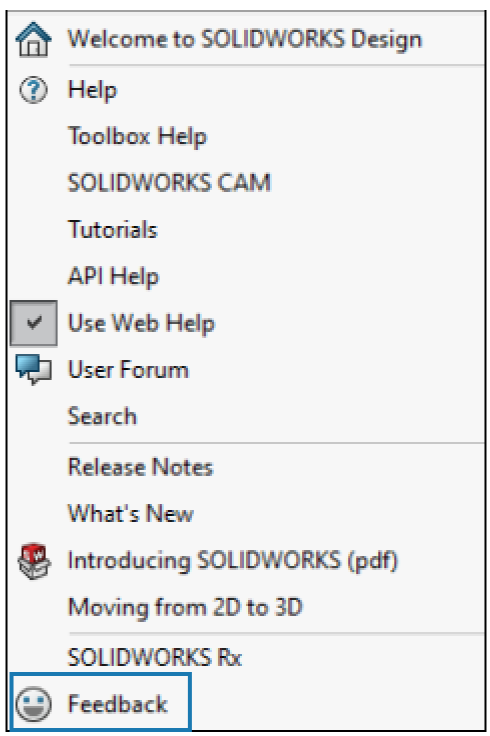
When finished, save the settings file. The new bookmark file location appears in the list and becomes available to users when the settings file is applied.

Crash and Feedback Interface for SOLIDWORKS Design (2026 SP1/FD01)



When SOLIDWORKS Design crashes, the Error Report dialog box now includes a consent section where you can enter your name and email.

You can also submit feedback to help improve SOLIDWORKS Design.



To submit feedback:

1. Click **Help** > **Feedback**.
2. Enter your comments.
3. Click **Submit Feedback**.

4

SOLIDWORKS Fundamentals

This chapter includes the following topics:

Changes to System Options and Document Properties (2026 SP1/FD01)

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

System Options

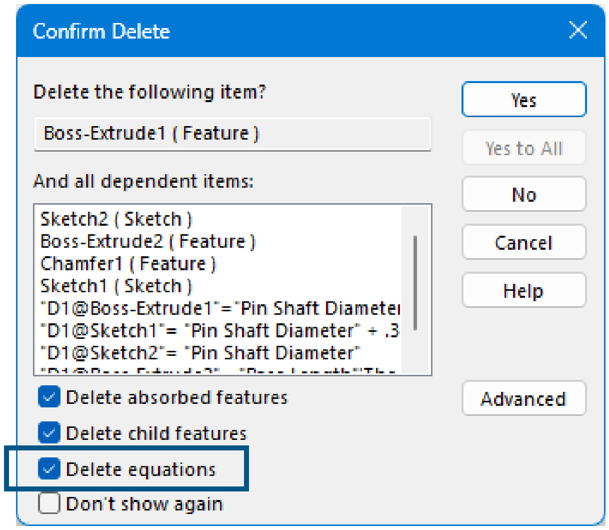
Option	Description	Access
View export options	(2026 SP1/FD01) Renamed to **Geometry export options**. Two new options added: **Export top-level components as blocks** and **Export polylines as geometry**.	**Export** > **DXF/DWG**
Mark assembly as modified when cosmetic changes are made to referenced documents	Specify if a rebuild is required for nonessential changes.	**Performance**
Enhanced graphics performance (requires SOLIDWORKS Design restart)	Required for SOLIDWORKS Design to use the DSPBR model.	**Performance**

Option	Description	Access
DSPBR (Dassault Systèmes Physically Based Rendering)	Renders models with appearances from DSPBR. These appearances are consistent with SOLIDWORKS Visualize and the **3D**EXPERIENCE platform and result in more realistic visuals. When selected, the Appearances PropertyManager includes a DSPBR tab. This tab ensures direct material-to-material translation. All parameters from SOLIDWORKS Design map directly to Visualize.	**Display**
Legacy Appearances	Uses older rendering functionality prior to SOLIDWORKS Design 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**

Document Properties

Option	Description	Access
	(2026 SP1/FD01) Shows full hole thread descriptions in SOLIDWORKS Model Based Definition (MBD) and drawings.	**Drafting Standard** > **Annotations**

Deleting Equations for Sketches and Features



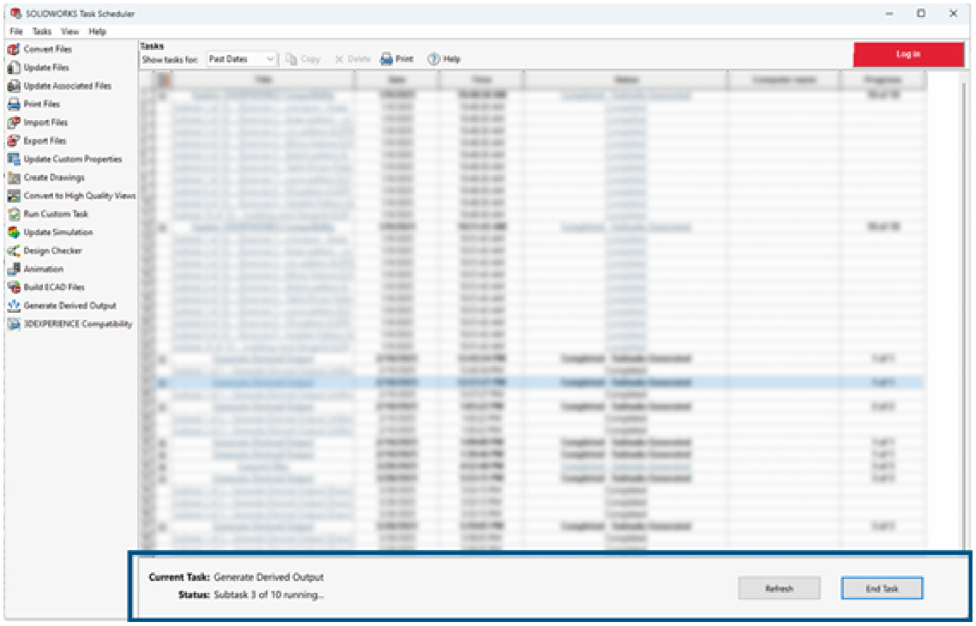
For sketches and features that contain equations, if you delete the sketch or feature, you can delete the equation directly from the Confirm Delete dialog box. Previously, you had to manually delete the equation using the Equations, Global Variables, and Dimensions dialog box.

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.

Ending a Running Task in the SOLIDWORKS Task Scheduler (2026 SP1/FD01)



You can end a task in the SOLIDWORKS Task Scheduler while it is running. When you end a running task, SOLIDWORKS Task Scheduler completes the active subtask and cancels any remaining subtasks.

Benefits: Ending a running task saves time if you need to stop or rerun the task.

To end a running task in the SOLIDWORKS Task Scheduler:

1. In the bottom pane, click **End Task**.
2. In the confirmation dialog box, click **Yes**.

Model Display (2026 SP1/FD01)

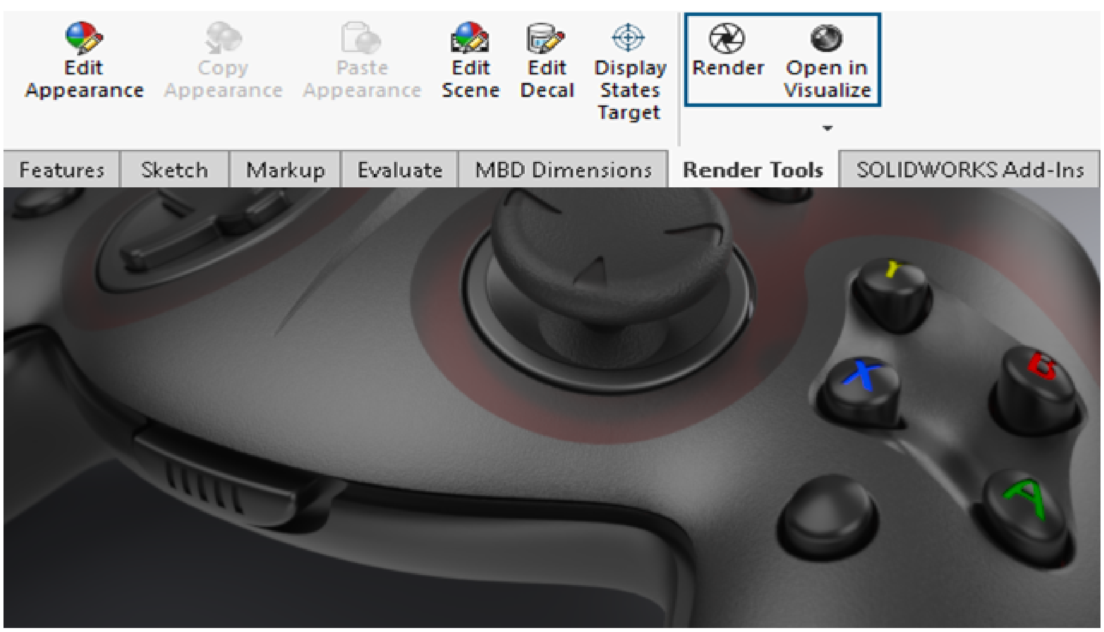
Flatten floor on

Flatten floor off

When displaying a model in an HDR (High Dynamic Range) scene, the **Flatten floor** functionality positions the model naturally on the ground plane, improving realism and overall visual quality.

In the Edit Scene PropertyManager, under **Floor**, select **Flatten floor (Background must be set to Use Environment)**.

Render with SOLIDWORKS Visualize from SOLIDWORKS Design



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

The RenderTools CommandManager includes the following tools.

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

Rendering with SOLIDWORKS Visualize from SOLIDWORKS Design

To render with SOLIDWORKS Visualize from SOLIDWORKS Design:

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.

SOLIDWORKS Appearances

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

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

- Select **Enhanced graphics performance (requires SOLIDWORKS Design 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.

Application Programming Interface

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

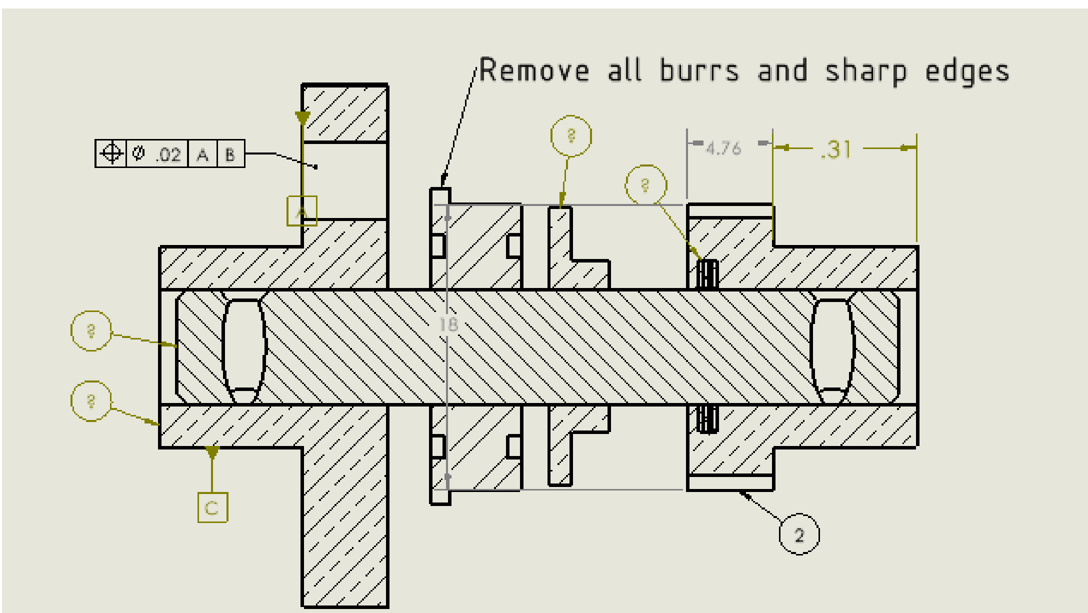
- SOLIDWORKS API includes the ability to:
  - Notify programs when you reorder cut list and solid body folder features
  - Add and edit family tables in drawings
  - Move drawing views independently of child views
- Simulation API. Support for cable connectors

5

User Interface

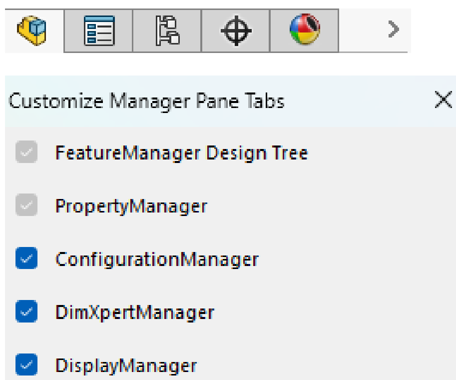
This chapter includes the following topics:

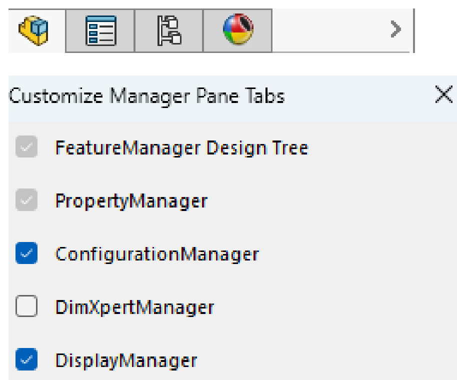
DS ISO Font (2026 SP1/FD01)



If you use a font that is unavailable, SOLIDWORKS Design uses the DS ISO font as the default.

Hiding and Showing Manager Pane Tabs



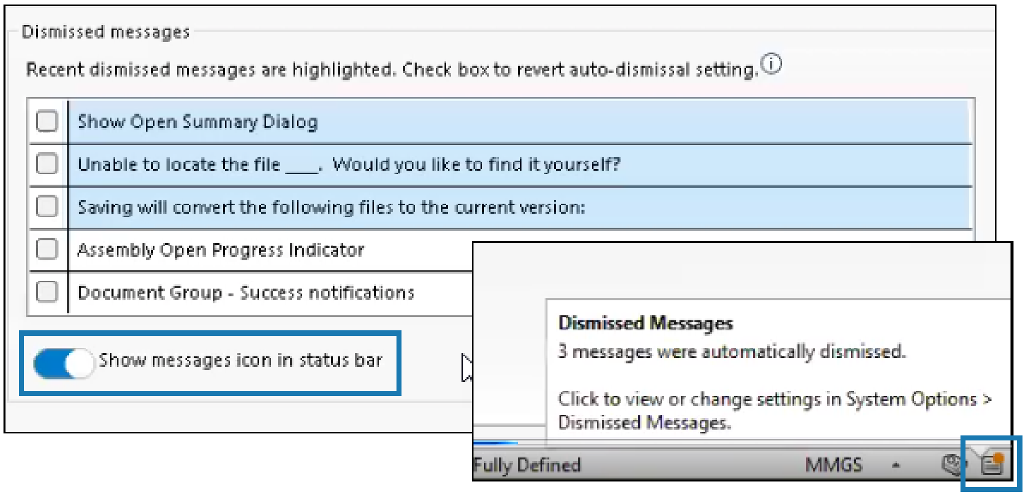


You can hide and show Manager Pane tabs to let you focus on the tabs that are most relevant to your work.

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

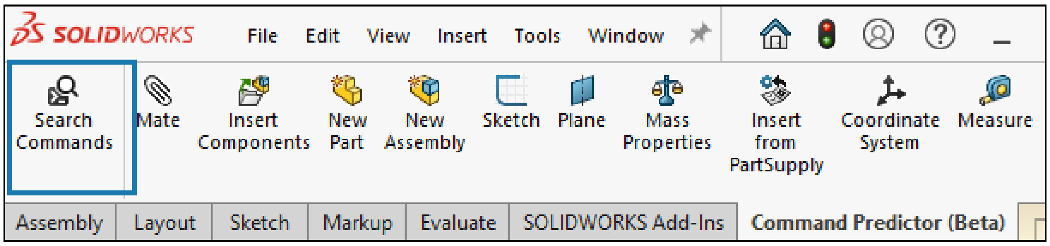


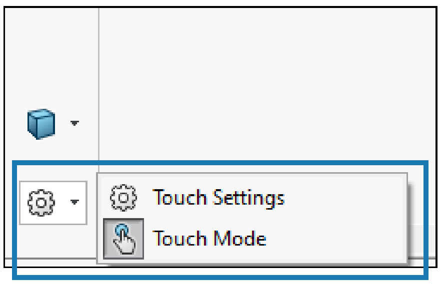
The status bar in the SOLIDWORKS Design window has a **Dismissed Messages** icon that lets you quickly view the messages that you have dismissed earlier and define the settings to view those messages again.

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 Beta tab, click **Search Commands** to search for a SOLIDWORKS Design tool.

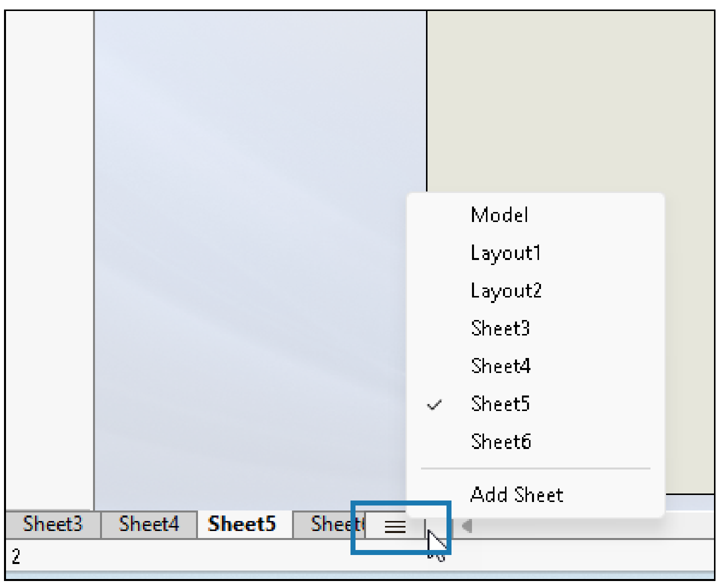
> THIS IS A BETA FEATURE UNDER EVALUATION. Any decision to use it is subject to important terms and conditions that the Customer understands and accepts by using it. Refer to the Offering Specific Terms available at **www.3ds.com/terms** for these terms and conditions.
>
> For information on how Dassault Systèmes uses AI technology, see **AI in the 3DEXPERIENCE Platform**.

Selection Filters

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

- **Filter Features** . 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.
- **Filter Components** . 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.

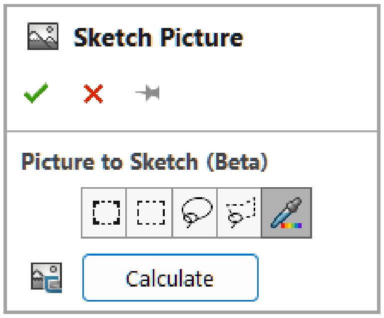
- Limitations while moving the folders below the **Origin** and above **Mates** feature are resolved. You can move components and folders up or down in the FeatureManager design tree.
- While working with multiple monitors, when you have a number of files open on a single monitor and you click **Window** > **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

Sketching

This chapter includes the following topics:

Converting Images to Sketches (Beta) (2026 SP1/FD01)



You can use **Picture to Sketch (Beta)** to automatically convert images to sketch entities.

To convert an image to a sketch:

1. Verify that the Autotrace tool is disabled:
   a. Click **Tools** > **Add-Ins**.
   b. In the dialog box, clear **Autotrace** if the app is selected.

   Picture to Sketch (Beta) is not available when Autotrace is enabled.

2. Edit a sketch and click **Tools** > **Sketch Tools** > **Sketch Picture**.
3. Select a picture to insert.

4. In the PropertyManager, under **Picture to Sketch (Beta)**, select an option:

	Crop Select	Reduces the area of the picture.
	Rectangle Select	Selects the part of the picture that you draw a rectangle around.
	Lasso Select	Selects the part of the picture that you draw a free hand loop around.
	Polygon Select	Selects the part of the picture that you draw a free hand ploygon around.
	Color Select	Selects an area based on color.

5. In the graphics area, use the selected tool to trace a region of the image.
6. Click **Calculate** to generate a preview of the sketch entity.
7. Click **OK** to accept the sketch entity.

THIS IS A BETA FEATURE UNDER EVALUATION. Any decision to use it is subject to important terms and conditions that the Customer understands and accepts by using it. Refer to the Offering Specific Terms available at **www.3ds.com/terms** for these terms and conditions.

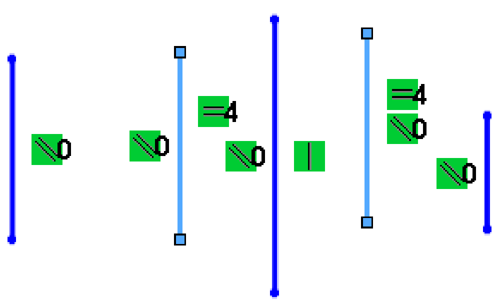
Slot Dimensions (2026 SP1/FD01)

You can dimension slots between arc centers.

In the Dimension PropertyManager, under **Dimension Text**, select **Slot length from arc center to arc center**. This changes the length display, not the slot parameters.

When cleared, the dimension shows the length of the arc tangents.

Relation Groups (2026 SP1/FD01)

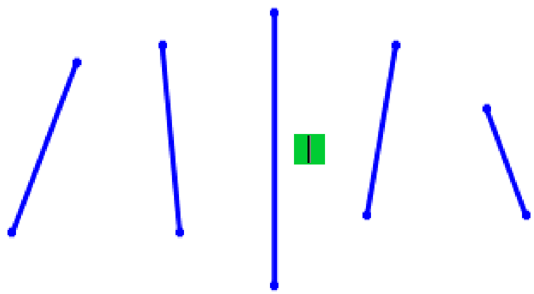


You can create relations to multiple sketch entities as a single relation group.

Entities can be in multiple relation groups. Relation groups are available for the following relation types: **Parallel**, **Equal**, **Collinear**, and **Coradial**.

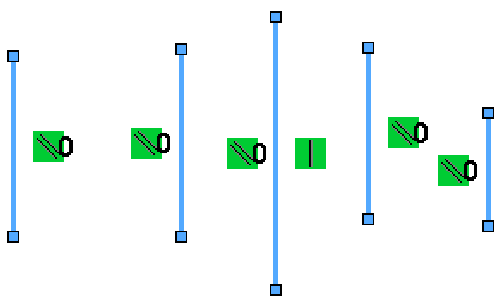
To create relation groups:

1. Open a sketch that has multiple entities.

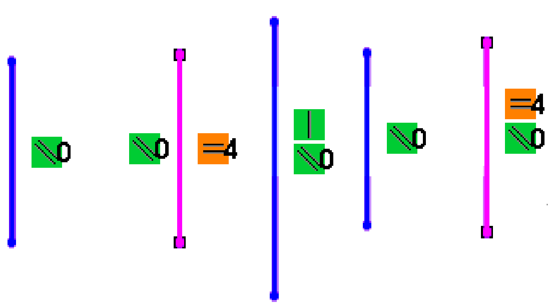


2. Select the entities and click **Add Relations** (Dimensions/Relations toolbar).
3. In the PropertyManager, under **Add Relations**, select **Parallel**.

   Entities in the relation group have the same number.



4. Add a second relation group by selecting two entities and selecting **Equal**.
5. Hover over an icon to highlight the entities in the relation group.

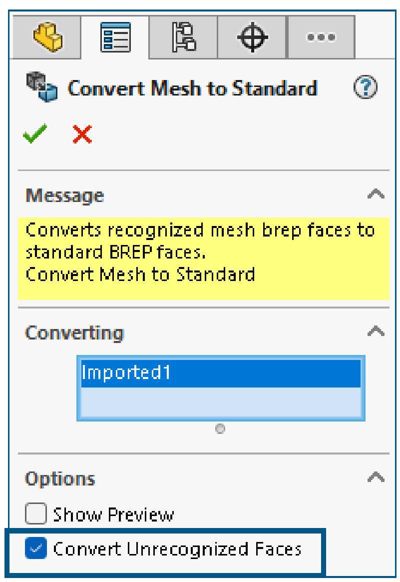


7

Parts and Features

This chapter includes the following topics:

Forcing Conversion to Standard BREP (2026 SP1/FD01)

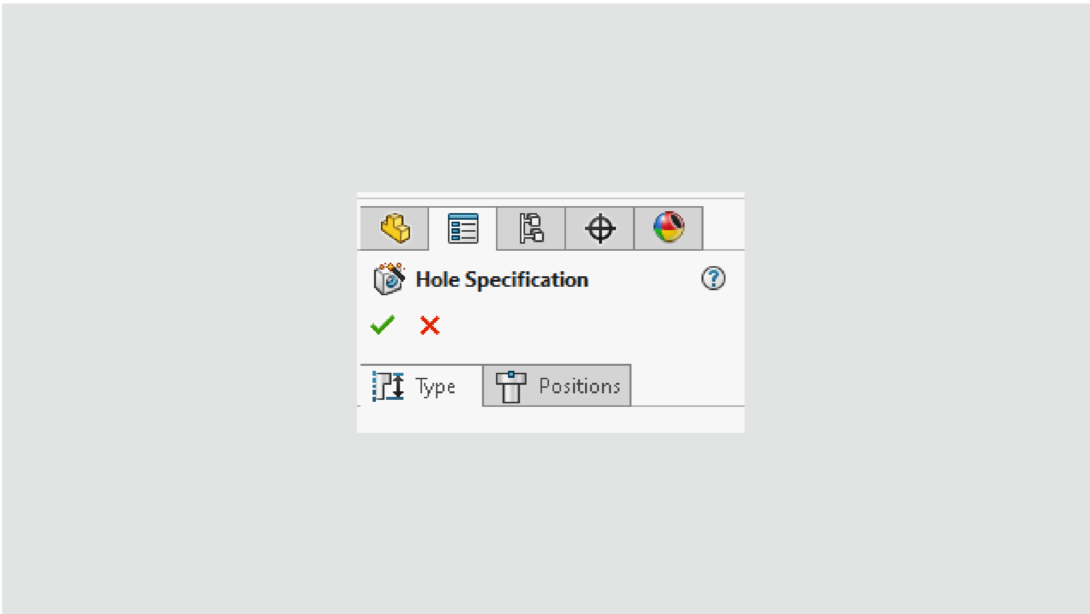


When you convert a mesh body to standard BREP geometry, you can force unrecognized mesh BREP faces to convert to standard BREP faces.

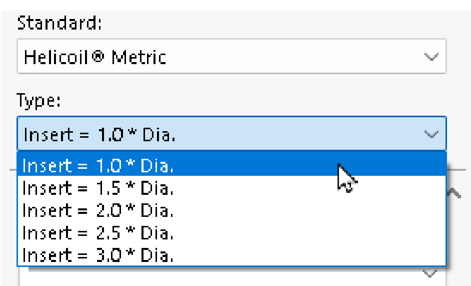
After you segment the mesh, you use the **Convert Mesh to Standard** command to convert the mesh faces to standard BREP faces. If there are faces that the command cannot convert, in the Convert Mesh to Standard PropertyManager, under **Options**, select **Convert Unrecognized Faces**, and run the command again. This command attempts to convert into standard BREP any mesh faces in the body that were not recognized the previous time you ran the **Convert Mesh to Standard** command.

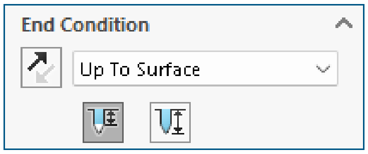
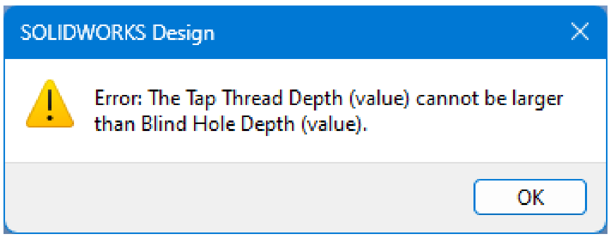
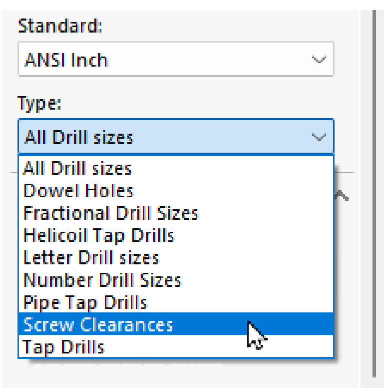
A message alerts you if there are still faces that this option cannot convert into standard BREP.

Hole Wizard (2026 SP1/FD01)

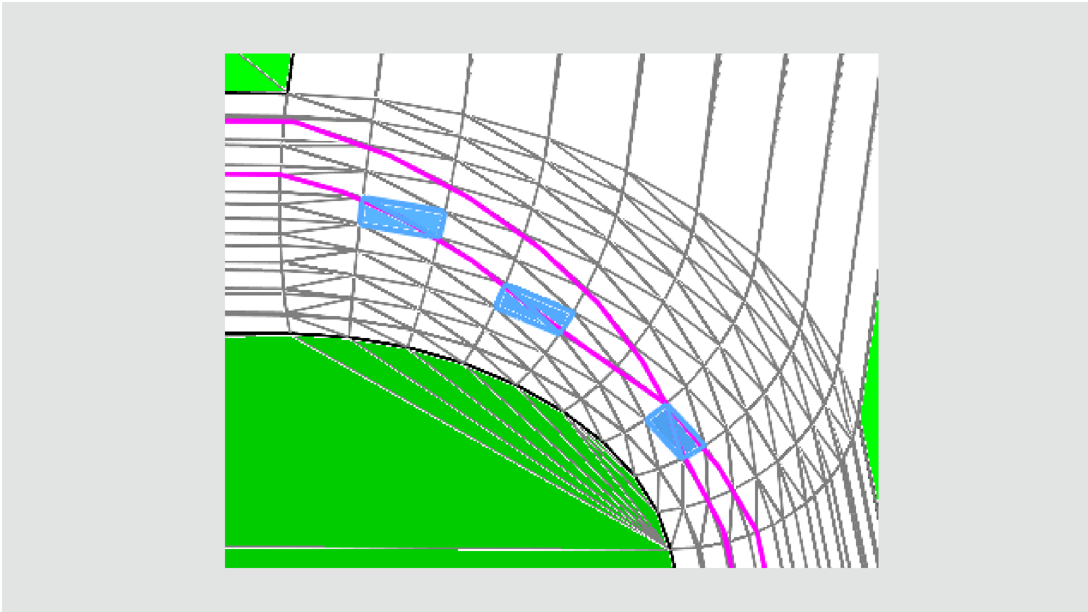


There are several enhancements to the Hole Wizard feature that improve accuracy, reliability, and performance.

Item	Enhancement
Inch and Metric Helicoil hole values stored in the Toolbox database  	The Helicoil Inch standard is updated with the values from the National Aerospace Standard – NASM33537 The Helicoil Metric standard is updated with the values from the Metric Aerospace Standard – MA1567. Previously, some values did not conform to these standards. Also, the software no longer calculates the values internally. These standard values ensure accuracy for Hole Wizard features. When you edit legacy Hole Wizard features, a dialog box prompts you to update Helicoil hole data to the latest standard.

Item	Enhancement
Up to Surface with **Depth up to Shoulder** enabled 	All holes that you create using these settings create a conical hole tip cut on the reference surface or plane.
Blind Hole Depth and **Tap Thread Depth** synchronization 	When you create blind holes, the **Tap Thread Depth** value cannot exceed the **Blind Hole Depth** value. If you specify invalid values, when you try to position a hole, an error message warns you about this requirement. Click **OK** to return to the PropertyManager and modify the values.
ANSI Inch holes using **Screw Clearances** for **Type** 	If you edit legacy models that use this hole type, a warning notifies you if the values are out of date. Click **Yes** to update the values to the latest standard (recommended) or **No** to make no changes. This **Type** is available for **Hole Types** of **Hole**  or **Straight Tap** .

Segmentation Guidelines (2026 SP1/FD01)

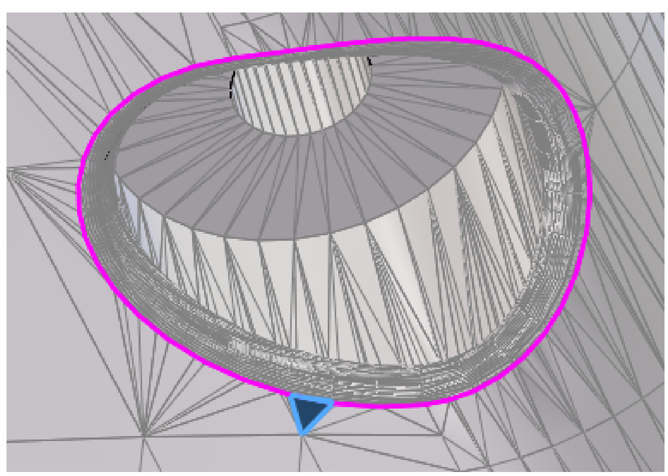


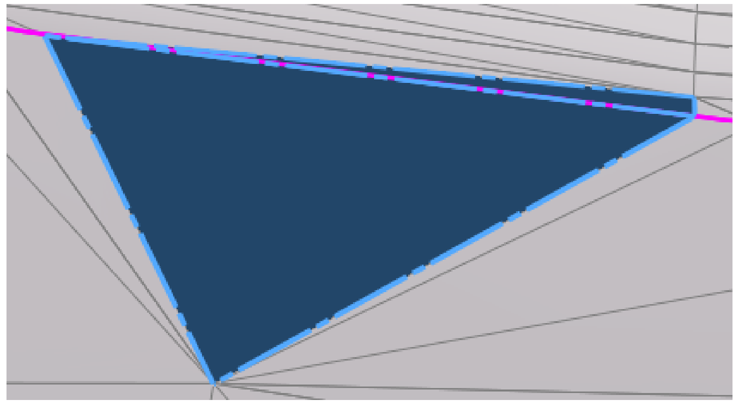
When you segment mesh faces using the **Crease Angle** and **Select Seed Facets** options, segmentation guidelines appear and help create segmented mesh faces that you can then convert to standard BREP faces.

In the Segment Mesh PropertyManager, under **Segmenting**, select the **Crease Angle** and **Select Seed Facets** options. When you select pairs of facets, segmentation guidelines appear between the pairs of facets. You can create multiple segmentation guidelines at the same time. The guidelines extend to the border of the segmented face.

Example of loop segmentation guideline formed from facet selection:

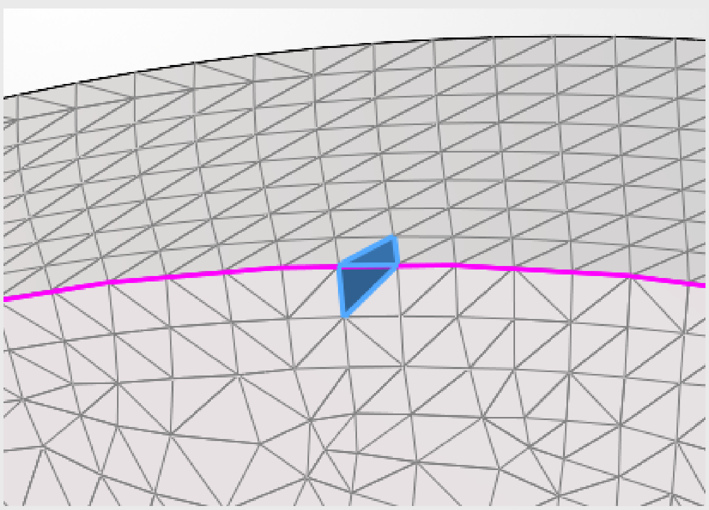
Full loop



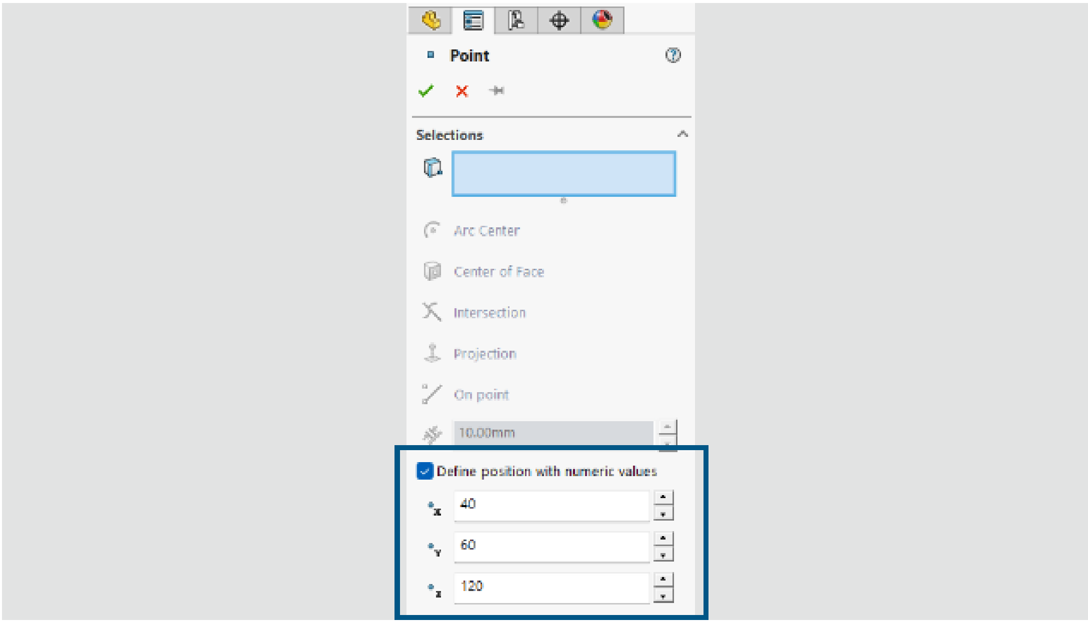
Close-up of selected facets	

You can use the **Crease Angle Tolerance** slider to control the maximum allowable angle between neighboring fins in the guidelines. This impacts the number of segmentation guidelines shown on the model.

You must select pairs of facets. The facet pairs can be on the same face or you can select multiple pairs that are on different faces.



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.

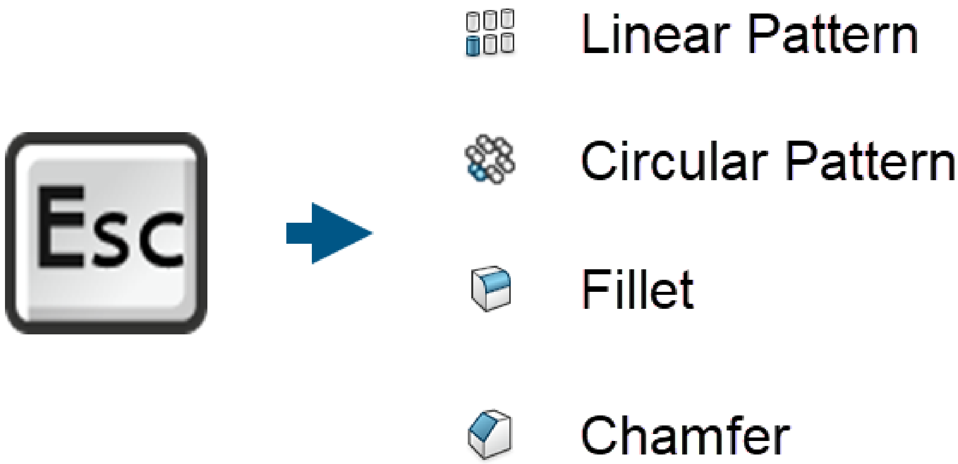
To create reference points by specifying XYZ values:

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.

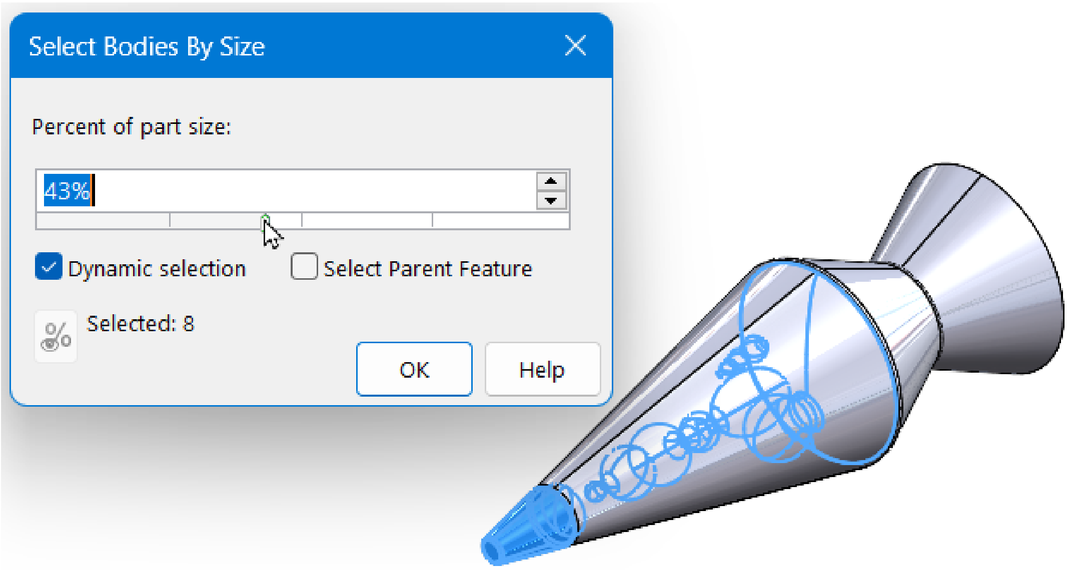
Benefits: You can exit processes that may take a long time to complete or that you started by mistake.

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**	• 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**.
Fillet and **Chamfer**	• 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® Design, 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

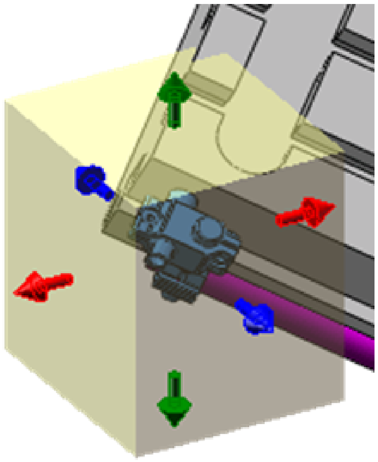
Specify a percentage of the part size that you want to select. Specify these options:

- **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.

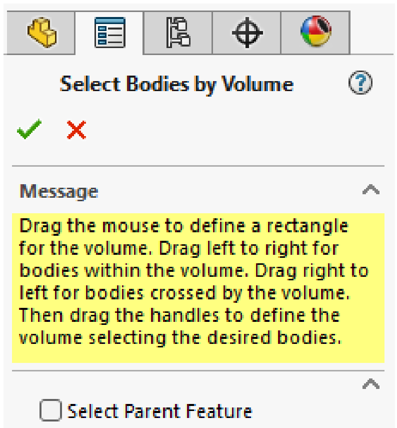
> If you import neutral-format multibody parts, to create **Imported** parent features, in the FeatureManager design tree, right-click the Imported part icon and click **Break Link**. You cannot undo this action.

Select Bodies By Volume

Select bodies based on a temporary volume that you define.



Follow the directions in the PropertyManager.

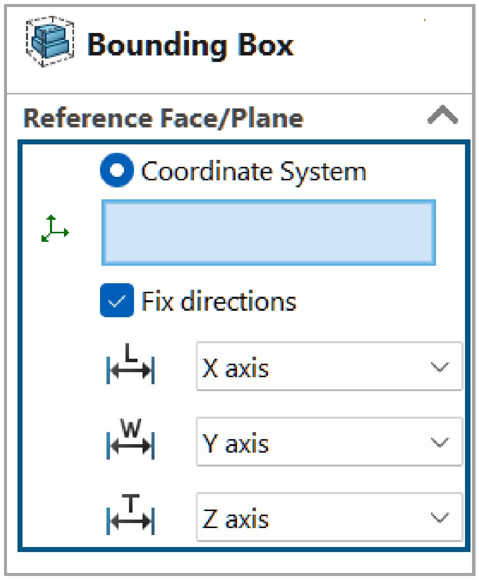


Drag to define a rectangle for the volume.

- 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.

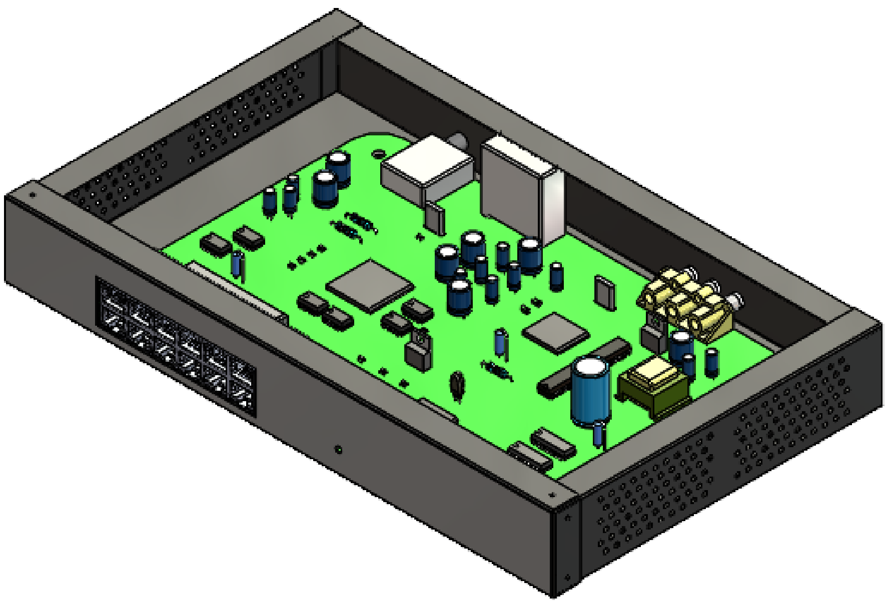
3. To change the axis direction, select **Fix directions** and select an axis:

	**Axis**	Specifies the axis for the cylindrical bounding box.

8

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

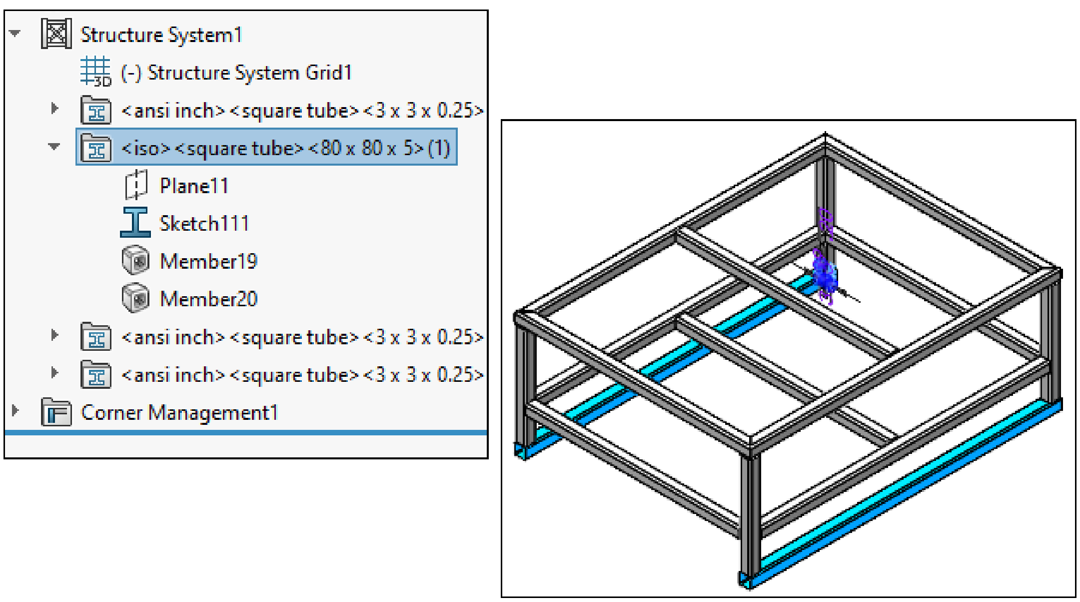
9

Structure System and Weldments

This chapter includes the following topics:

- **Structure System User Interface (2026 SP1/FD01)**
- **Accessing Cut List Properties from File Properties**
- **Enhanced Corner Treatments**

Structure System User Interface (2026 SP1/FD01)



Enhancements to the user interface help improve productivity.

When you select a folder in the FeatureManager design tree, the members of that folder highlight in the graphics area.

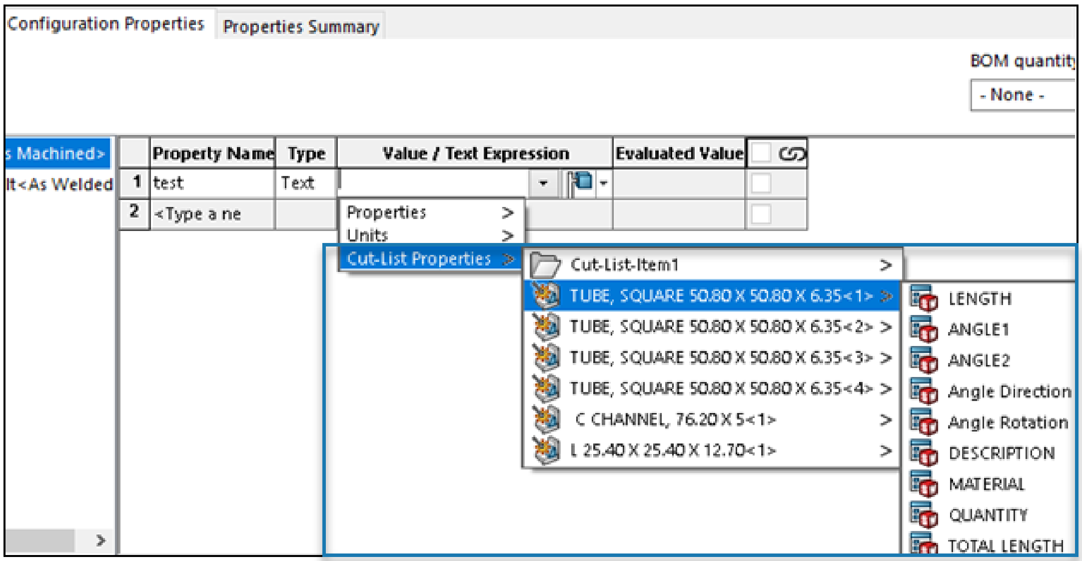
In the Secondary Member PropertyManager, for **Support Plane Member** and **Between Points Member** member types, the **Align Member** functionality is enhanced. When you edit a secondary member and click **Align Member**, SOLIDWORKS Design highlights the:

- Cross-section of both primary members.
- Line that joins the piece points of the secondary member at both ends.

Align Member is available for primary members with similar and nonsimilar profiles.

When you select **Align Both Ends** and select a pierce point of the primary member, both ends of the secondary member shift equally to align the pierce points with primary members. When you clear the option, only one end of the secondary member shifts.

Accessing Cut List Properties from File Properties



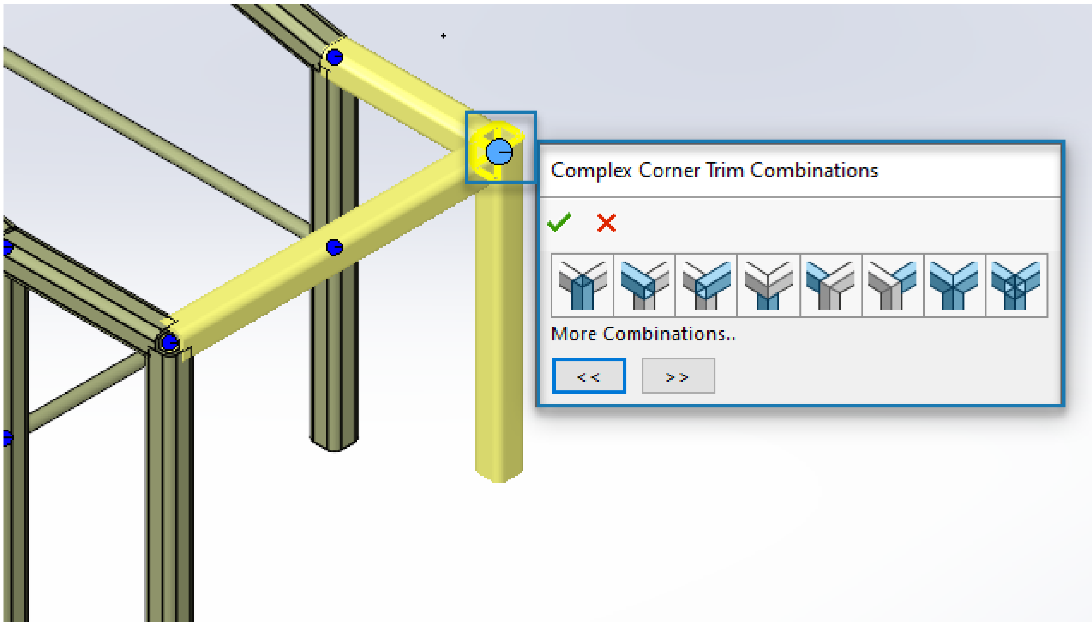
You can access cut list properties from the Configuration Properties tab of the Properties dialog box.

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: `$PRPWLD:"*`*Cutlist Item name*`*:*`*Cutlist Property Name*`*"`. SOLIDWORKS Design links the selected cut list property to the formula.

Enhanced Corner Treatments



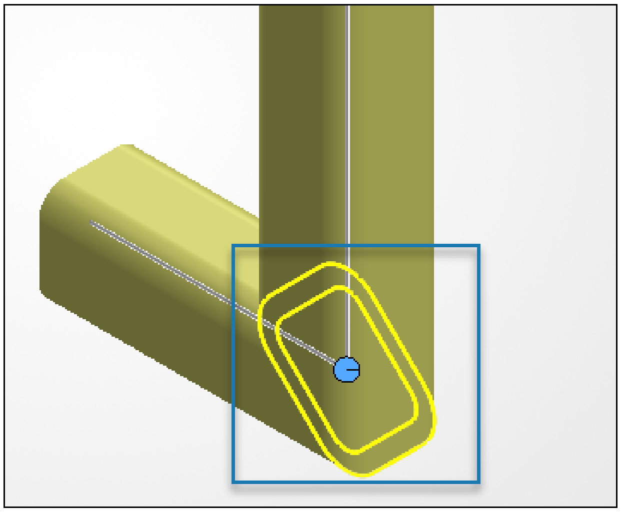
You can select the appropriate combination of trim type and trim order for a complex corner from the graphics area.

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.



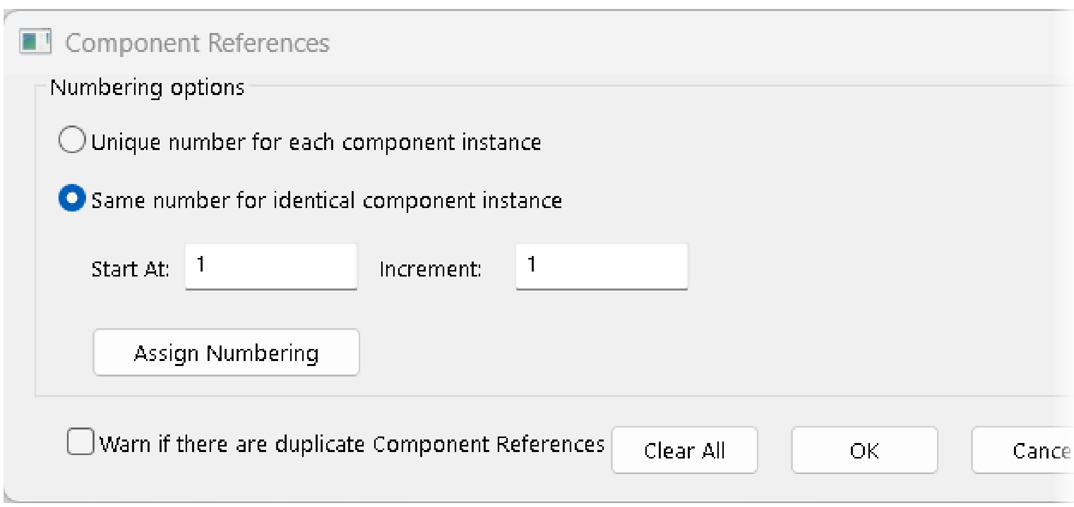
10

Assemblies

This chapter includes the following topics:

- **Assigning the Same Component Reference Number (2026 SP1/FD01)**
- **Pattern Assistant (2026 SP1/FD01)**
- **Specifying Rebuild Requirements for Cosmetic Changes**

Assigning the Same Component Reference Number (2026 SP1/FD01)



You can assign the same component reference number to identical component instances.

Use the following options in the Component References dialog box to assign component references:

Unique number for each component instance	Assigns a different reference number to each component instance.
Same number for identical component instances	Assigns the same reference number to duplicate component instances. Assigns a different reference number to components that are not duplicates.

Start At	Specifies the first number in the sequence of component reference numbers.
Increment	Specifies the increment between component reference numbers.
Warn if there are duplicate Component References	Shows a message when more than one component has the same component reference number. This option is cleared when **Same number for identical component instance** is selected.
Assign Numbering	Assigns a number based on the component position in the FeatureManager design tree and the numbering option selected. Existing component references are overwritten.

These options replace the **Use Tree Order** option.

To assign the same number to identical component instances:

1. Open an assembly that contains multiple instances of a component.
2. Right-click the assembly name in the FeatureManager design tree.
3. Click **Edit Component References**.
4. In the dialog box, select **Same number for identical component instance**.
5. Click **Assign Numbering**.

Component Name	Component Description	Component Reference
chair legs<1>	chair legs	1
chair legs<3>	chair legs	1
chair cushion<1>	chair cushion	2
chair strength bar<1>	chair strength bar	3
chair strength bar<3>	chair strength bar	3
chair back support<1>	chair back support	4
chair back support cushion<1>	chair back support cushion	5

6. Optional: Set the **Start At** and **Increment** values and click **Assign Numbering**.

Component Name	Component Description	Component Reference
chair legs<1>	chair legs	10
chair legs<3>	chair legs	10
chair cushion<1>	chair cushion	12
chair strength bar<1>	chair strength bar	14
chair strength bar<3>	chair strength bar	14
chair back support<1>	chair back support	16
chair back support cushion<1>	chair back support cushion	18

Numbering options

Unique number for each component instance

Same number for identical component instance

Start At: 10 Increment: 2

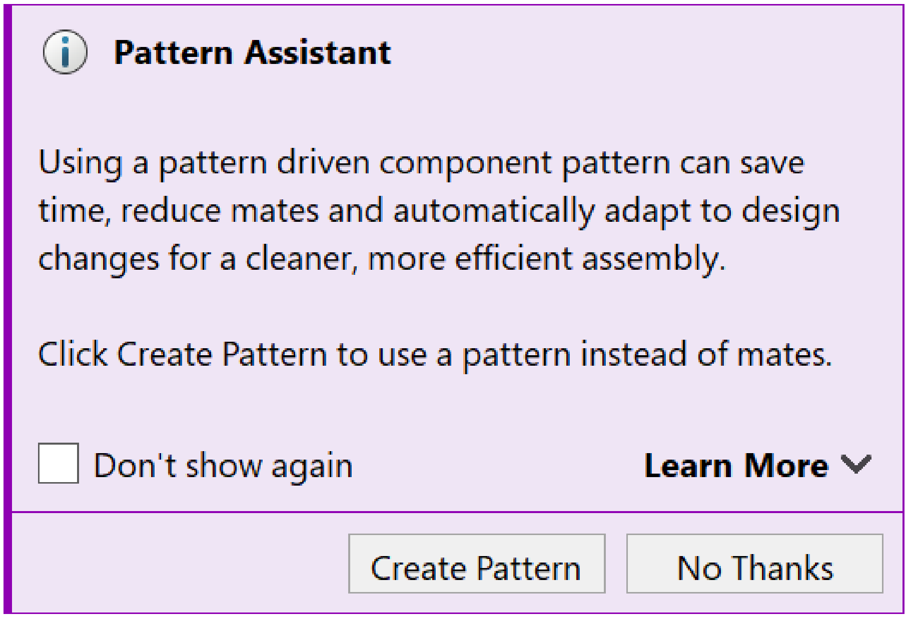
7. Select **Unique number for each component instance**.
8. Optional: Click **Warn if there are duplicate Component References**.

Component Name	Component Description	Component Reference
chair legs<1>	chair legs	10
chair legs<3>	chair legs	10
chair cushion<1>	chair cushion	12
chair strength bar<1>	chair strength bar	14
chair strength bar<3>	chair strength bar	14
chair back support<1>	chair back support	16
chair back support cushion<1>	chair back support cushion	18

9. Click **Assign Numbering**.

Component Name	Component Description	Component Reference
chair legs<1>	chair legs	10
chair legs<3>	chair legs	12
chair cushion<1>	chair cushion	14
chair strength bar<1>	chair strength bar	16
chair strength bar<3>	chair strength bar	18
chair back support<1>	chair back support	20
chair back support cushion<1>	chair back support cushion	22

Pattern Assistant (2026 SP1/FD01)

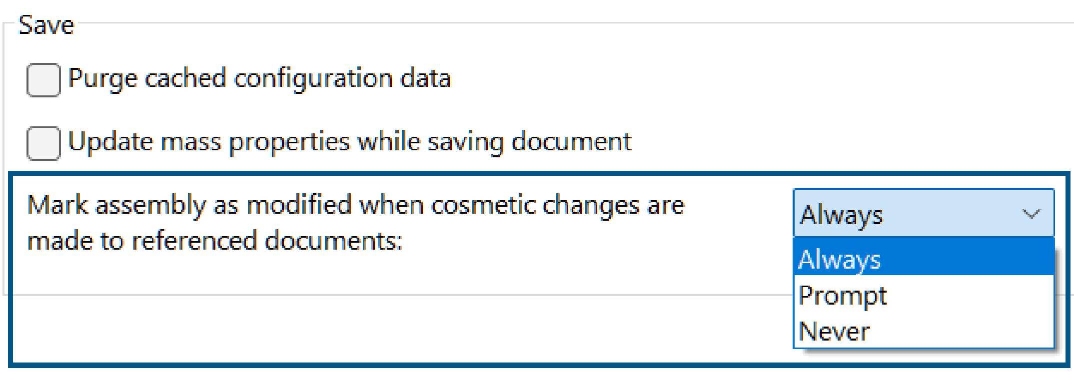


You can use Pattern Assistant to guide you when to use pattern driven component patterns instead of individual mates.

Pattern Assistant displays when creating a mate to a component that has geometry that can drive a Pattern Driven Component Pattern and there is another component instance that is mated to the same geometry.

Click **Learn More** to display more content. Click **Don't show again** so the message does not reappear. To reactivate the dismissed message, see **Tools** > **Options** > **System Options** > **Messages/Errors/Warnings** > **Dismissed Messages**.

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.

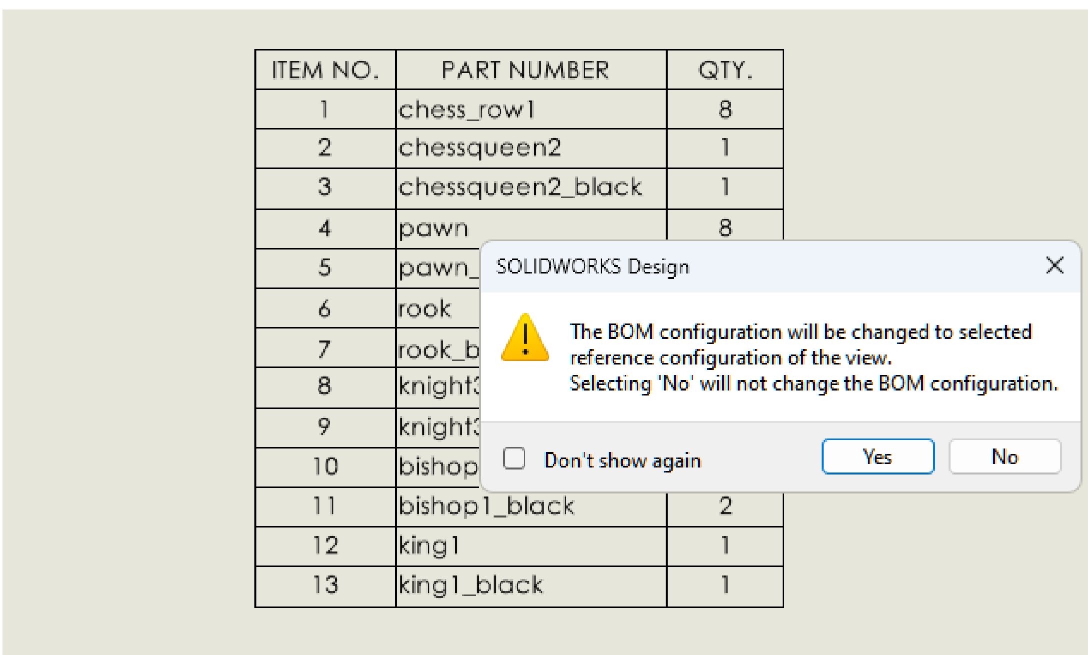
11

Detailing and Drawings

This chapter includes the following topics:

- **Linking BOM Configurations to Drawing View Configurations (2026 SP1/FD01)**
- **Eliminating Duplicate Annotations in Model Items (2026 SP1/FD01)**
- **Exporting Drawing Views as Sketch Blocks (2026 SP1/FD01)**
- **Weld Symbols (2026 SP1/FD01)**
- **Hole Thread Descriptions (2026 SP1/FD01)**
- **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**

Linking BOM Configurations to Drawing View Configurations (2026 SP1/FD01)

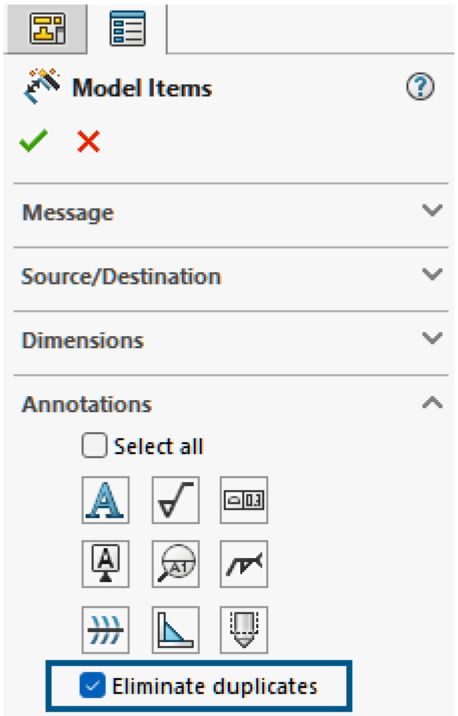


If you change the configuration of a drawing view, you can update the bill of materials (BOM) to reference the new configuration.

To link BOM configurations to drawing view configurations:

1. In a drawing with a BOM, select a drawing view.
2. In the Drawing View PropertyManager, under **Reference Configuration**, select a different configuration.
3. In the dialog box, click **Yes** to change the BOM to reflect the drawing view configuration.

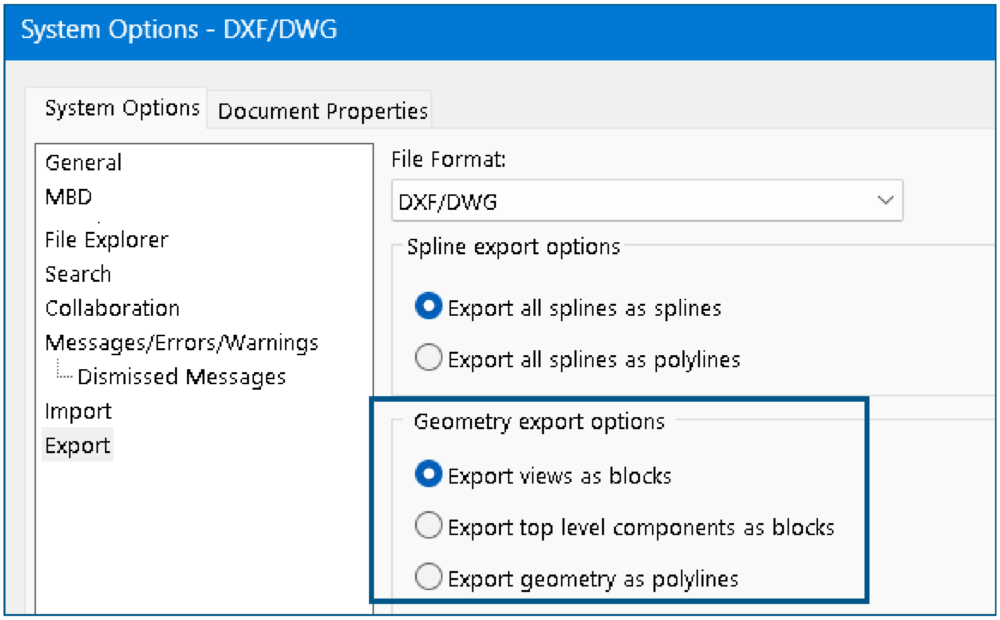
Eliminating Duplicate Annotations in Model Items (2026 SP1/FD01)



When you insert annotations with the Model Items PropertyManager, you can eliminate duplicate annotations.

In the Model Items PropertyManager, under **Annotations**, select **Eliminate duplicates**. The software inserts unique annotations only.

Exporting Drawing Views as Sketch Blocks (2026 SP1/FD01)

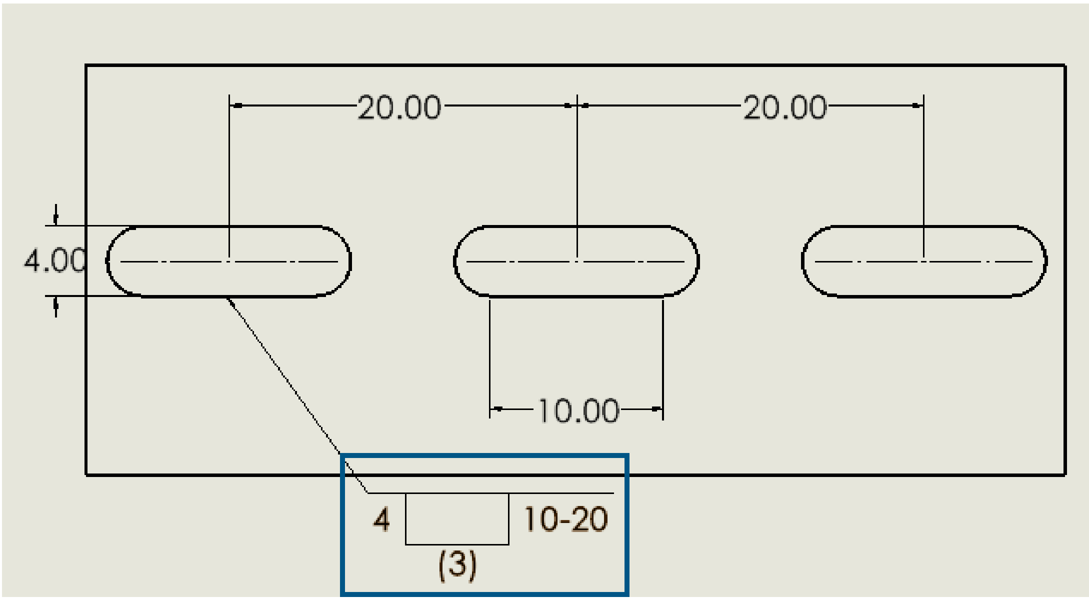


When you export drawings as DXF or DWG files, two additional options let you export top-level components as blocks or export geometry as polylines. In the **Tools** > **Options** > **System Options** > **Export** DXF/DWG dialog box, the subsection **View export options** is renamed to **Geometry export options**.

The two additional options under **Geometry export options** are:

Option	Description
Export top-level components as blocks (Default)	Exports all top-level components as DXF or DWG files. The blocks are listed based on their view names in the Drawings PropertyManager.
Export polylines as geometry	Exports the geometry of the entire drawing as polylines

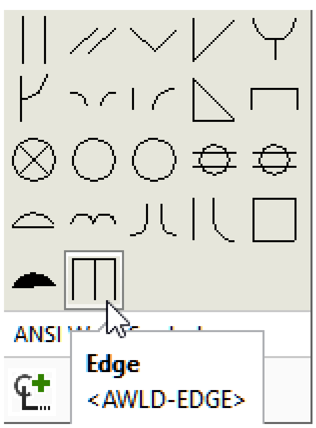
Weld Symbols (2026 SP1/FD01)



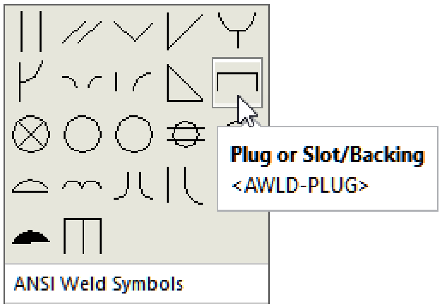
Weld symbols are enhanced to align with the AWS A2.4:2020 standard.

Click **Weld Symbol** (Annotation toolbar). In the Properties dialog box, for the ANSI drafting standard, weld symbols include:

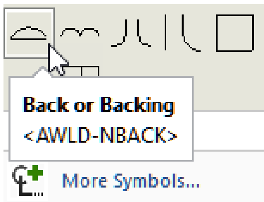
- **Edge** symbol.



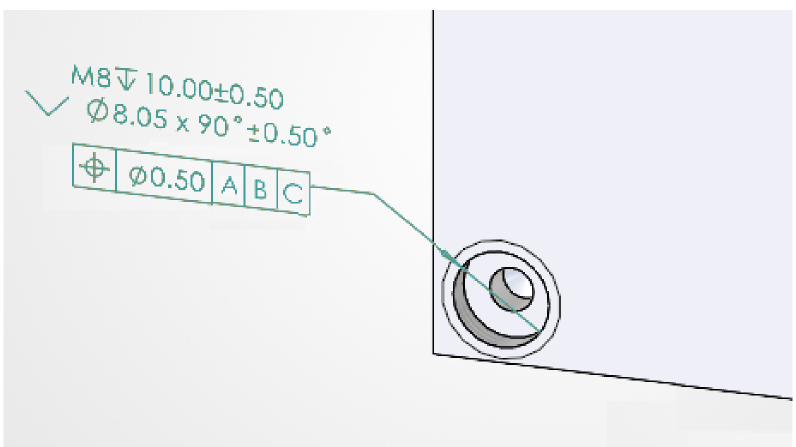
- **Plug or Slot/Backing** symbol. The **Plug or Slot** symbol has additional text to indicate that it is also a **Backing** symbol.



- **Back or Backing** symbol. The **Back Fill** symbol is renamed to **Back or Backing**.



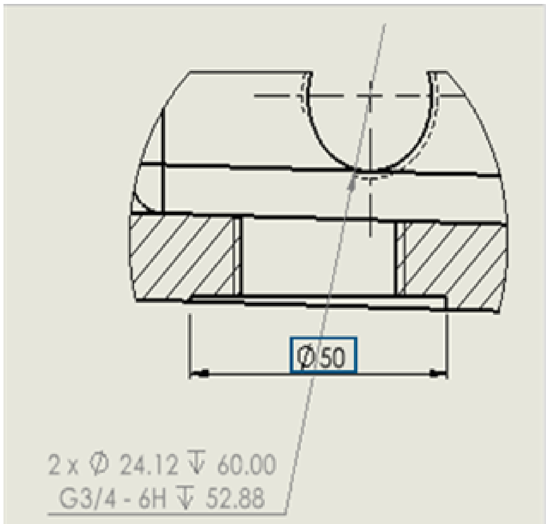
Hole Thread Descriptions (2026 SP1/FD01)



You can show full hole thread descriptions in Model Based Definition (MBD) models and drawings. This maintains a uniform display of thread descriptions.

In **Tools** > **Options** > **Document Properties** > **Drafting Standard** > **Annotations**, select **Show full thread description for all holes**.

Adding Breaks to Dimension Lines around Dimension Text



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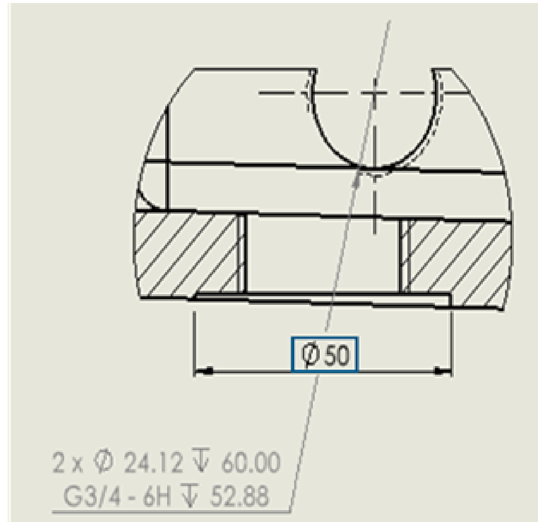

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You can add breaks to dimension lines to avoid overlapping dimension text.

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 the Customer understands and accepts by using it. Refer to the Offering Specific Terms available at **www.3ds.com/terms** for these terms and conditions.
>
> For information on how Dassault Systèmes uses AI technology, see **AI in the 3DEXPERIENCE Platform**.

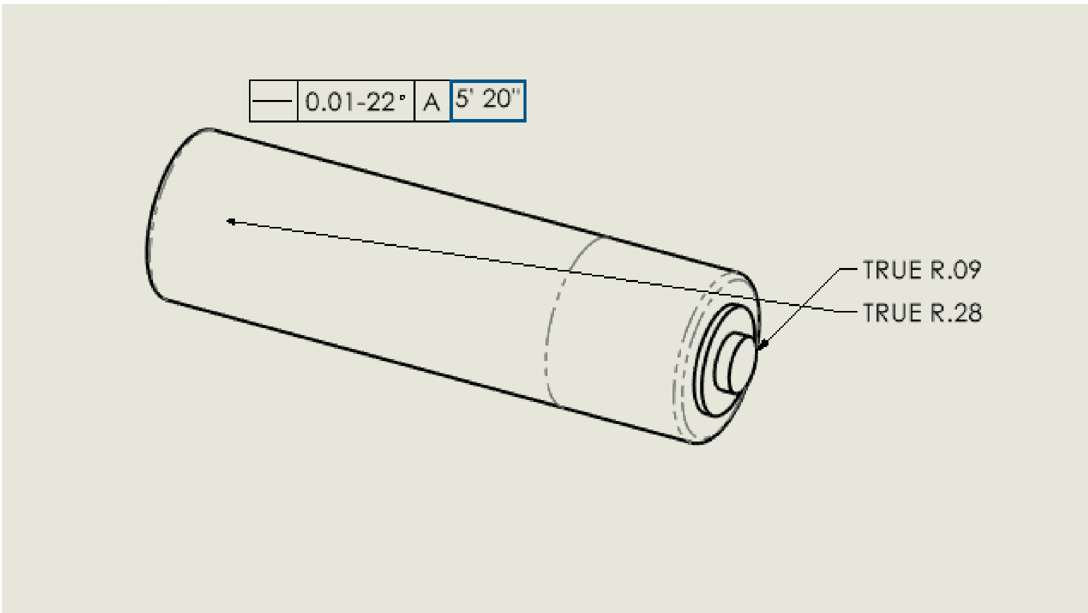
Auto-Generate Drawings (BETA) automatically creates:

- Section views, such as views with internal feature dimensions.

- Hole callouts for drawings generated for imported models, such as STEP.

> SOLIDWORKS Design 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** (Annotation toolbar) or **Insert** > **Annotations** > **Geometric Tolerance**.
2. In the graphics area, click to place the symbol.

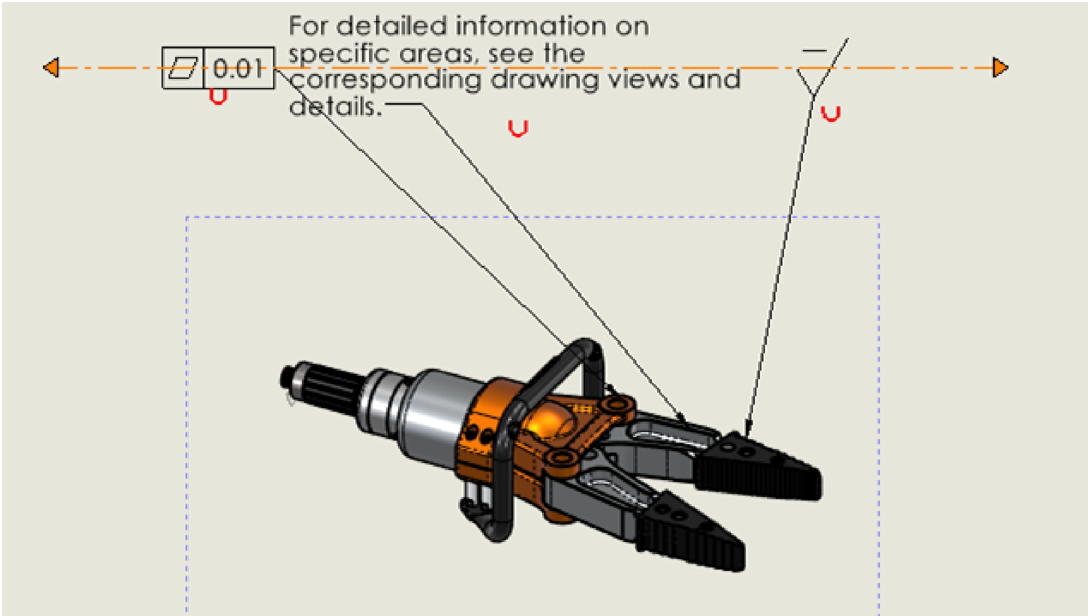
   > A feature control frame appears with handles and a Tolerance dialog box surrounding it.

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

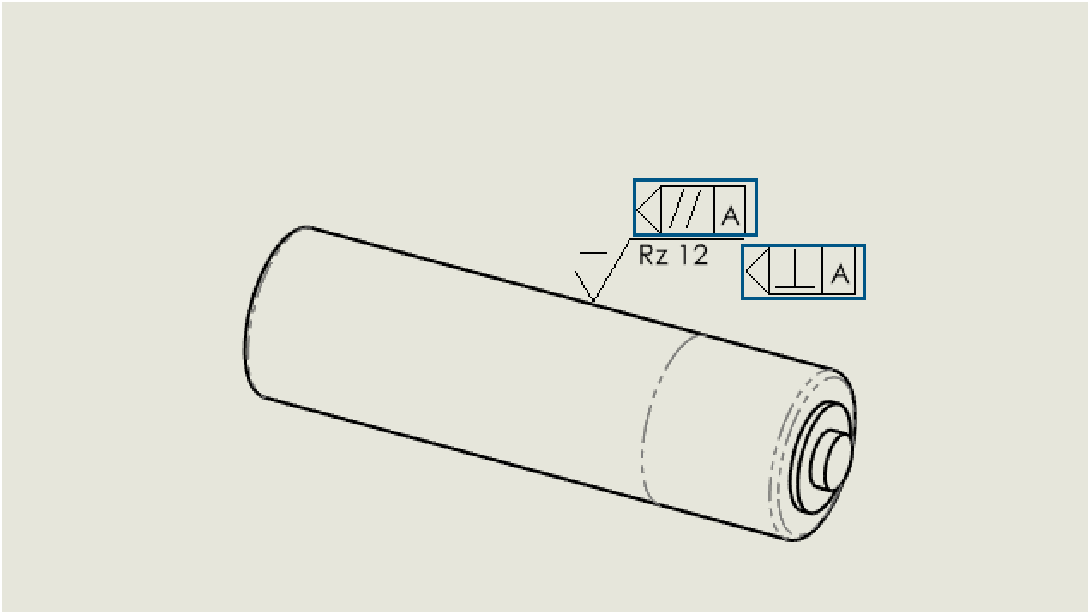


You can use magnetic lines to align annotations, such as notes, weld symbols, geometric tolerance symbols, surface finish symbols, and revision symbols, to improve drawing presentations.

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:
   a. Under **Tolerance Type**, specify **Parallel** ⫽ or **Perpendicular** ⊥.
   b. (Optional) Under **Datum Reference**, enter a datum.
   c. Click **Close**.
3. In the graphics area, click to place the symbol.

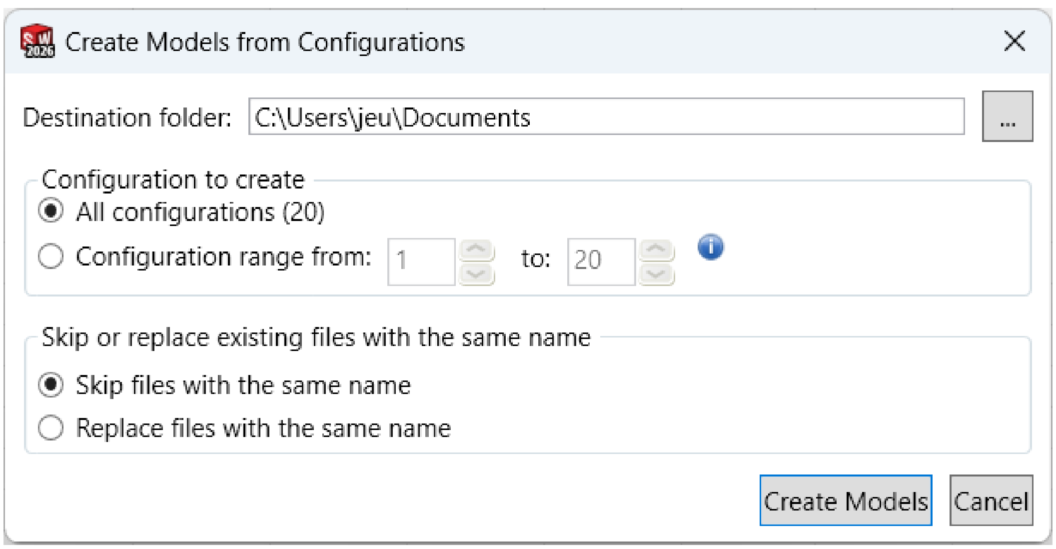
12

Configurations

This chapter includes the following topics:

- **Creating Models from the Configuration Publisher (2026 SP1/FD01)**
- **Configuration Tables and Display State Tables Usability**
- **Splitting Out Configurations Into Individual Files**

Creating Models from the Configuration Publisher (2026 SP1/FD01)



You can create unique single-configuration models using the Configuration Publisher.

To create models from the Configuration Publisher:

1. Open a model that has configurations that are driven by the Configuration Publisher.

   In the ConfigurationManager, the PropertyManager icon indicates that the model has a custom PropertyManager driven by the Configuration Publisher.

2. In the ConfigurationManager, right-click the PropertyManager icon and click **Edit Feature**.

   The Configuration Publisher appears.

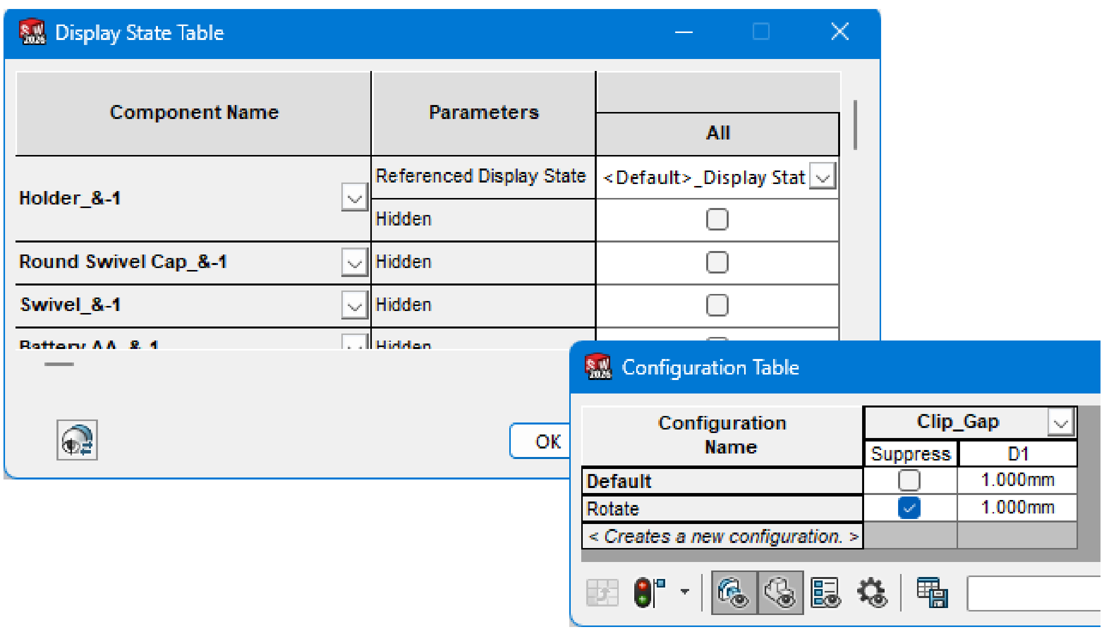
3. On the Preview tab, click **Create Models**.
4. In the Create Models from Configurations dialog box, specify the **Destination folder** and the options, and click **Create Models**.

   You can specify the configurations to create and whether to skip or replace existing files.
5. Click **Create Models**.

   The software creates the files and stores them in the **Destination folder** that you specified. A dialog box notifies you about the results.

   Each part or each assembly plus all its referenced components is created in a unique subfolder of the **Destination folder**. The naming convention is *<ModelName><PartNumber>*. If duplicate folder names occur, the app adds an incremental numerical suffix. The software deletes suppressed components from the created models.
6. To view the log file, in the dialog box, click **View Log File**.

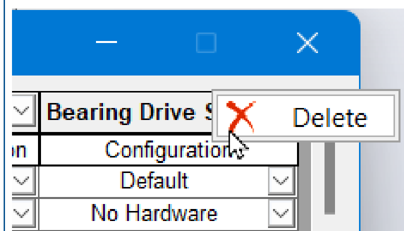
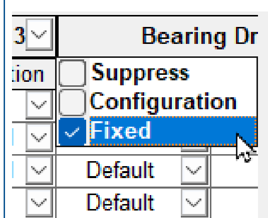
Configuration Tables and Display State Tables Usability



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

Configuration Table Improvements

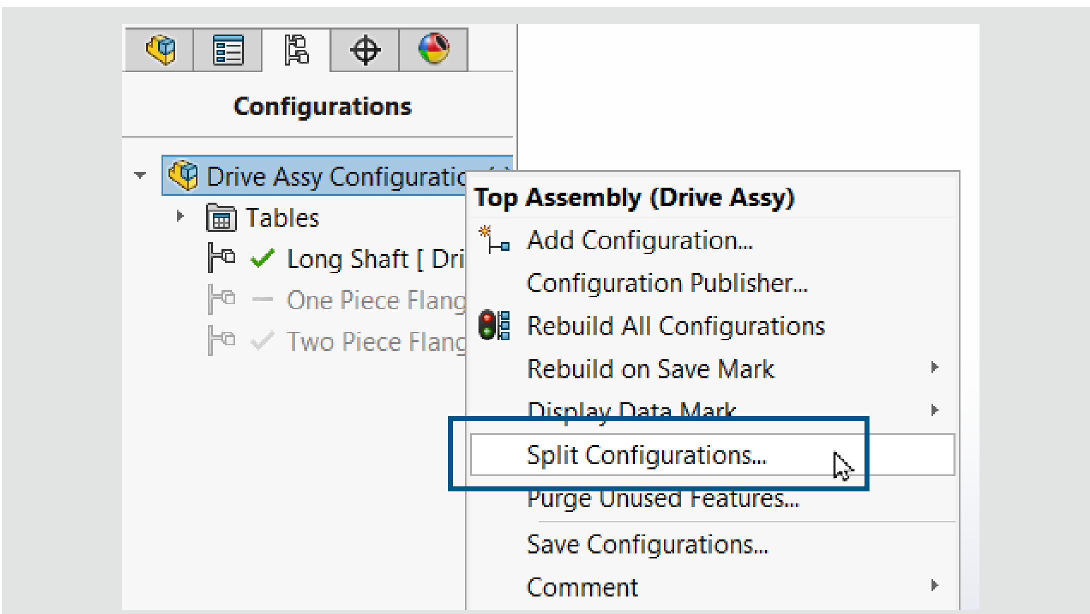
Area	Improvement
Suppress and **Fixed** columns	When you double-click a component in the graphics area to add it to the configuration

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

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**.

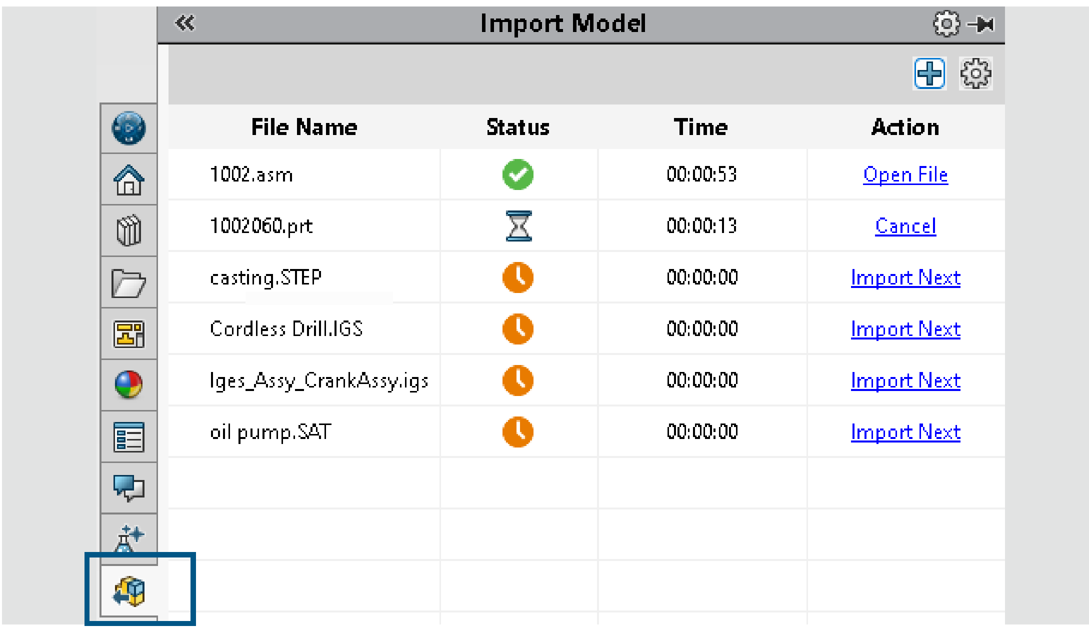
   All the configurations are split out into separate files that are in the same location as the original file. The file names are <original file name>.<configuration name>.SLDPRT or SLDASM. For example, for a part named BasePart.SLDPRT that has a split configuration named LongHandle, the split configuration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

13

Import/Export

This chapter includes the following topics:

Importing Models Using a Background Process (2026 SP1/FD01)



When you import large files into SOLIDWORKS Design, you can run the import process in the background so you can continue working in SOLIDWORKS Design. The software tracks the import process using the **Import Models** tab in the Task Pane.

This functionality is available for all file formats that 3D Interconnect supports. A separate asynchronous import process saves you time, especially when you import large files that you expect to take a long time to import.

> This functionality does not replace the instant import functionality using the **File** > **Open** command, which is still available.
>
> You can access the background import functionality from the Open dialog box. If you browse to select a supported file type, the **Import in Background** option appears under the selection area. Select **Import in Background** and click **Open** to open the Import Models tab in the Task Pane and import the file using the background import process.

To import models using a background process:

1. In the Task Pane, click the Import Models tab.
2. Click **Select Files to Import**.

   The first time you click **Select Files to Import**, a message prompts you to specify an **Import Queue location** to save the imported files.

3. After you specify the import location, click **Select Files to Import** again to display the **Open** dialog box.
4. In the **Files of Type** list, select a file type.
5. Browse to select a file and click **Open**.

   > You can select multiple files to import them at the same time.

   The import process starts. You can switch to the main window to continue working in SOLIDWORKS Design while the import process continues. A notification advises you when the import process is completed and in the Import Models tab, the columns show the **Status** and **Time** for each imported file.

   To reposition a model to be next in the import queue order, under **Action**, click **Import Next**. That model moves directly under the current import row and becomes the next model to be imported.

   You can click **Select Files to Import** as many times as required to put multiple files in the import queue for importing. The software imports the files sequentially.

6. To open an imported file, on the Import Models tab, under **Action**, click **Open File**.

   In the Import Models tab, you can right-click an imported file to open the file location or clear the imported files from the queue. The software empties the Task Pane queue when you end a session, however, the files remain in the import queue location.

   If you exit SOLIDWORKS Design while an import is in progress, a dialog box appears with the option to exit SOLIDWORKS Design and cancel the import or to keep the app open so the import continues.

Face and Edge Identifiers During Import

When you import certain source CAD files into SOLIDWORKS Design, SOLIDWORKS Design 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 Design:

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

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

If you update the source CAD file to a newer version that you reimport into SOLIDWORKS Design, SOLIDWORKS Design reimports these IDs, keeping their original values. SOLIDWORKS Design 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 Design may change to improve performance.

SOLIDWORKS Design strives to keep these IDs stable through all service packs of the current major release, for example, SOLIDWORKS Design 2026. SOLIDWORKS Design 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.

14

SOLIDWORKS PDM

This chapter includes the following topics:

SOLIDWORKS® PDM is offered in two versions. SOLIDWORKS PDM Standard is included with SOLIDWORKS Design Professional, SOLIDWORKS Design Premium, and SOLIDWORKS Design Ultimate, and is available as a separately purchased license for non-SOLIDWORKS Design 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.

Automatic Windows Login for Web2 (2026 SP1/FD01)

You can log in to the Web2 client automatically with your Windows credentials.

Your administrator must enable Automatic Login for any vault you want to access.

Benefits: Logging in automatically saves time and reduces administrative work.

To log in to Web2 automatically with Windows credentials, click **Log in using Windows User** on the Web2 login page.

Web2 verifies your account and opens the vault.

For best results:

- Use a computer that is on a domain and a domain user account.
- Verify that your browser is connected to your Windows account.
- Disable any private browsing settings on your browser (for example, Incognito Mode, Private Browsing, or Secret Mode).

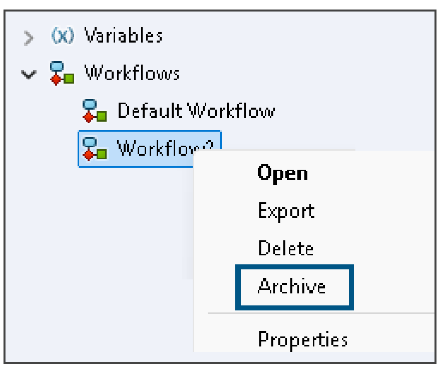
Refreshing the Check In and Change State Dialog Boxes (2026 SP1/FD01)

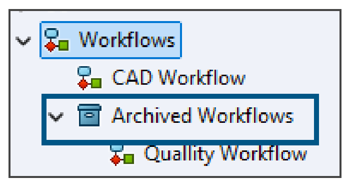
If you receive a warning when checking in or changing the state of files, you can address the related issue and then refresh the Check In or Change State dialog box. After you refresh the dialog box, all resolved warnings disappear, and you can proceed with the action.

Benefits: You can address issues without canceling and restarting the check in or change state action.

For example, if you try to check in a file that is open in another application, SOLIDWORKS PDM displays an error message in the Check In dialog box. In earlier releases, you needed to close the dialog box, close the file, and restart the check in operation. Now, you can close the file, click **Refresh** in the Check In dialog box, and resume checking in the file.

Archive Workflows

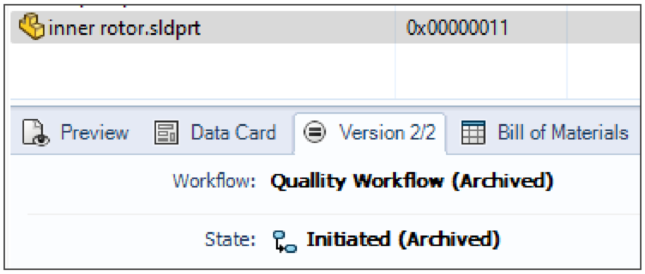




In the SOLIDWORKS PDM Administration tool, you can archive a workflow. You cannot transition or rollback the file to another state within the archived workflow.

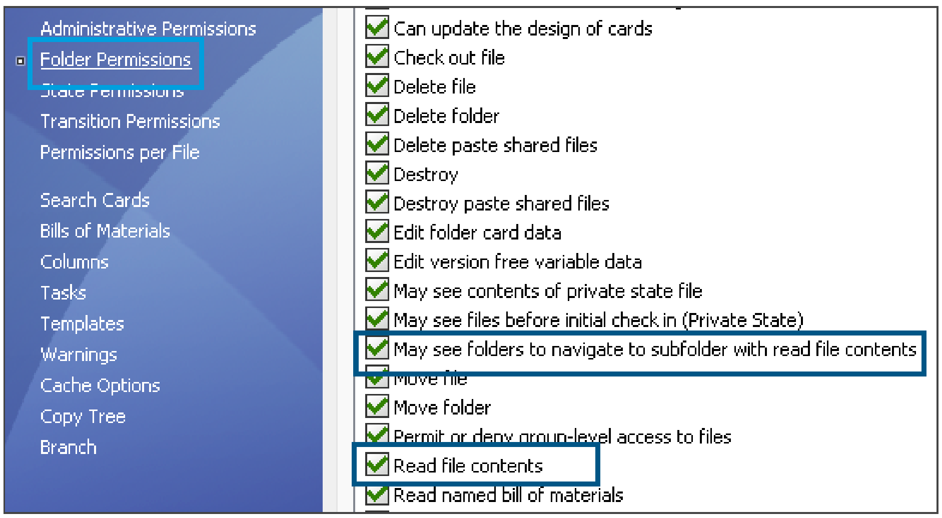
To archive a workflow, right-click a workflow and select **Archive**. The workflow then moves under the **Archived Worflows** sub folder.

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.

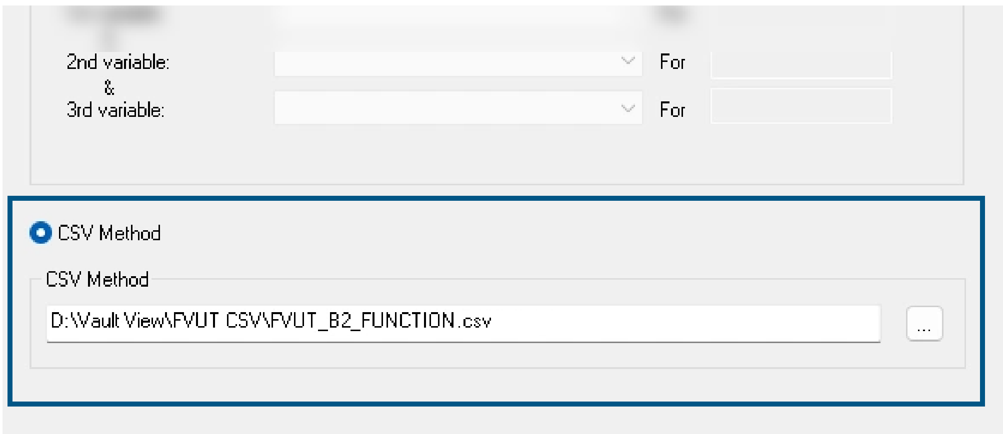
Lower Level Folder Access



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.

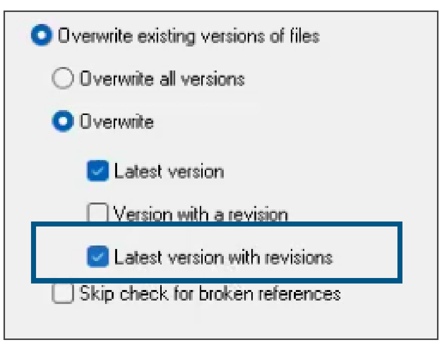
File Version Upgrade Tool



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**.

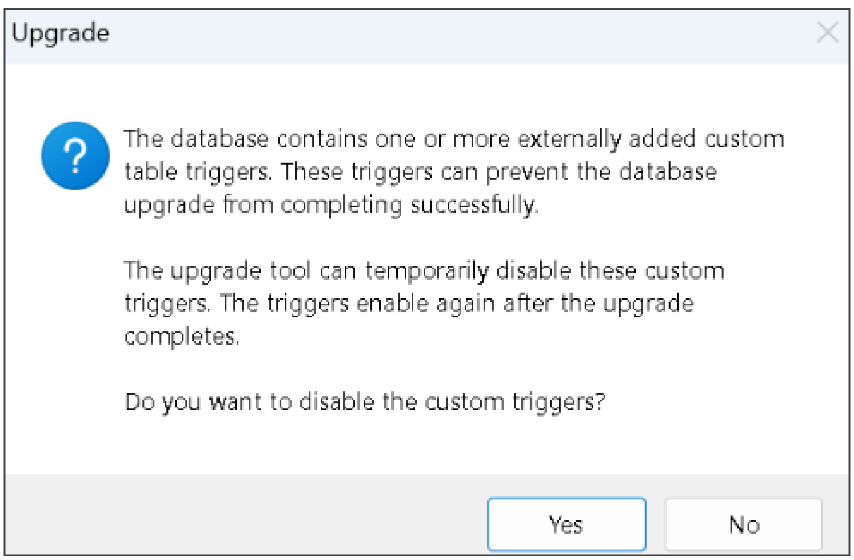
  

  With this option, you get a version attached with the latest revision. For example, if the following are the revisions and versions for a file, this option picks version 5 of the file to overwrite.

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.

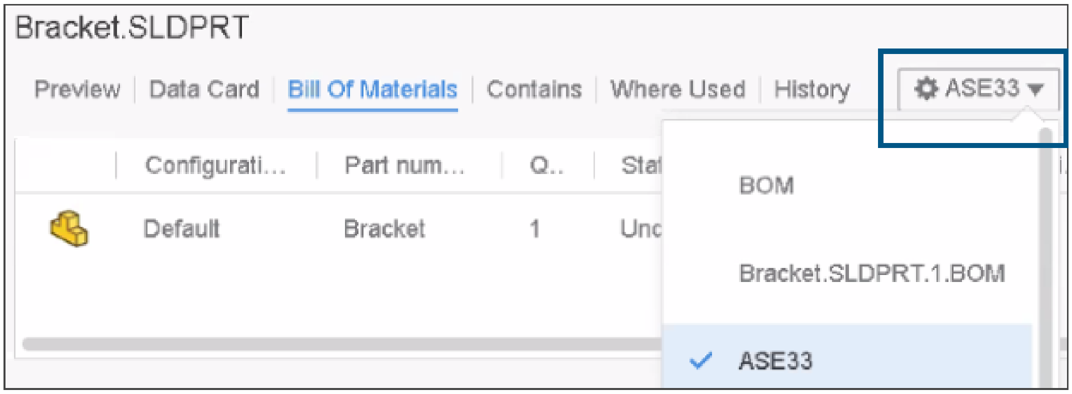
Disabling Custom Triggers before Database Upgrade



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

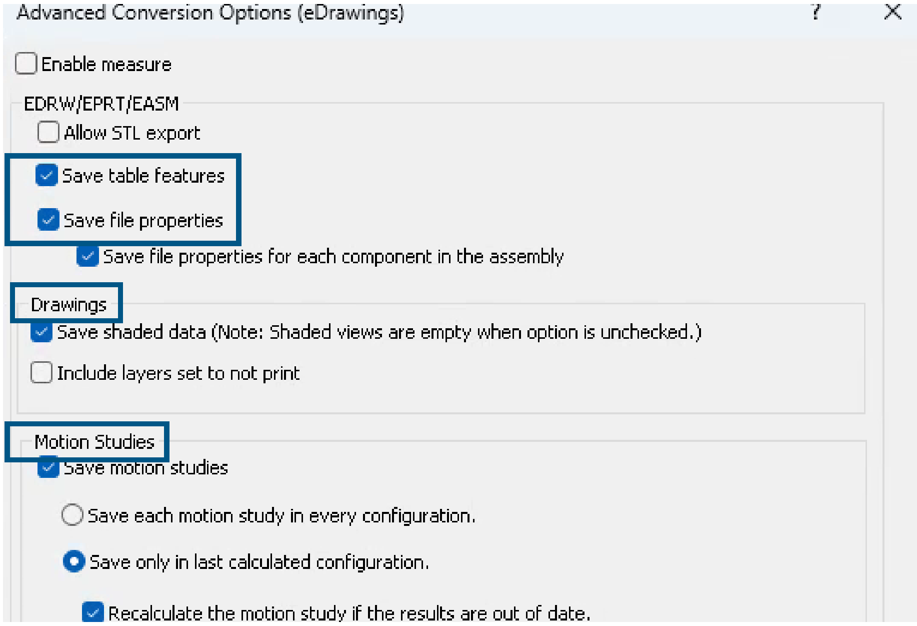
The Advanced Encryption Standard (AES) for data transfer between the archive server and the client has been upgraded from AES-128 to AES-256. This makes the data transfer more secure.

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.

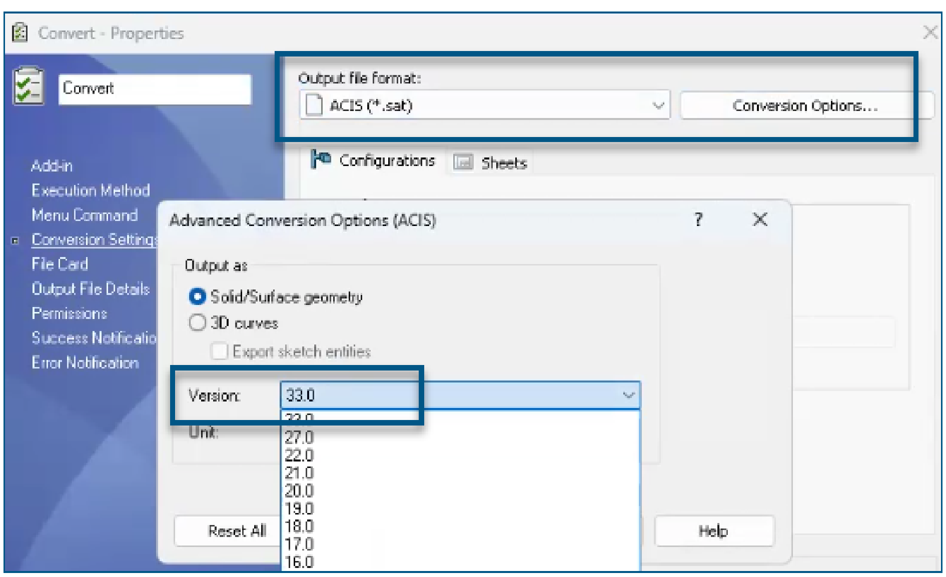
Convert Task Options



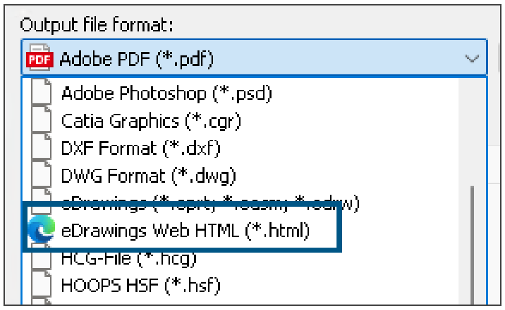
The SOLIDWORKS PDM Administration tool includes enhancements for the convert task options for Parasolid™, ACIS®, and eDrawings® file format.

The enhancements are:

- A modified user interface for the eDrawings file format similar to the SOLIDWORKS Design 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

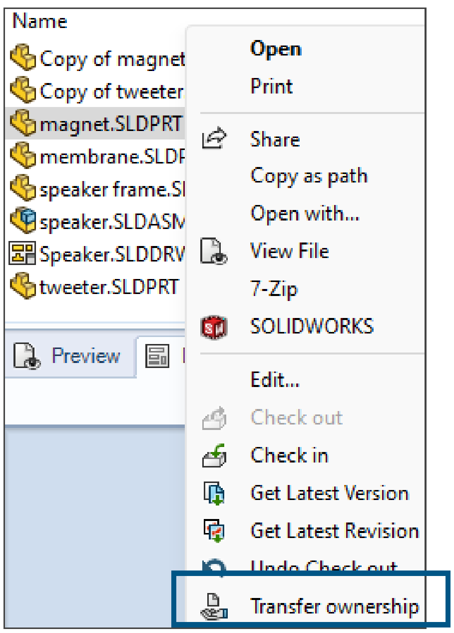


- Support of higher versions for `Parasolid` (up to 35.1) and `ACIS` (up to 33.0) file formats.



- 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**.

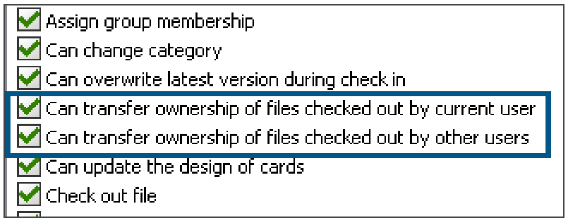
Automatic Synchronization of Vault Views



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**

15

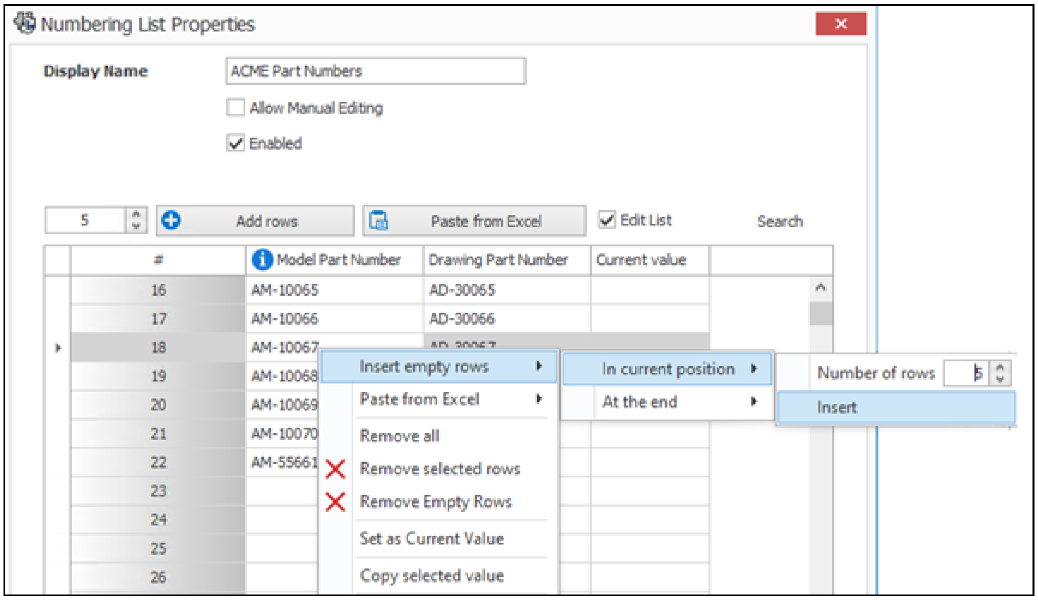
SOLIDWORKS Manage

This chapter includes the following topics:

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

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

Numbering Lists



Numbering Lists lets you assign part numbers to SOLIDIWORKS 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. If a value in the **Model Part Number** column is empty, SOLIDWORKS Manage removes the entire row when you save.
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

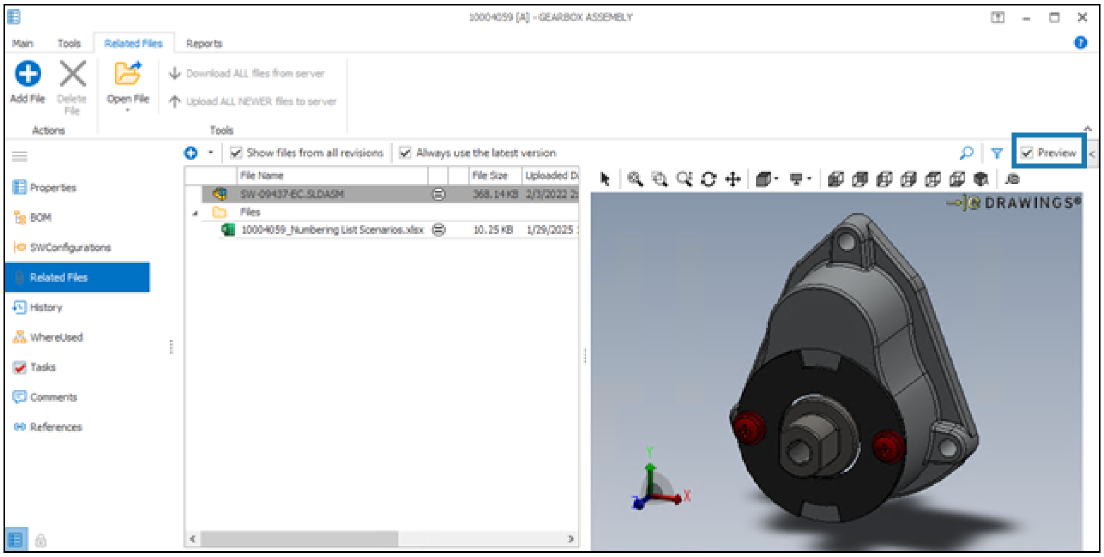
Model first, then drawing	Drawing first, then model
The model gets the current value **Model Part Number**, then the drawing gets the next **Model Part Number** in the list.	The drawing gets the current value **Model Part Number**, then drawings get the next **Model Part Number** in the list.

Applying a Number to a SOLIDWORKS File

To apply a number to a SOLIDWORKS file:

1. In SOLIDWORKS Design, 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



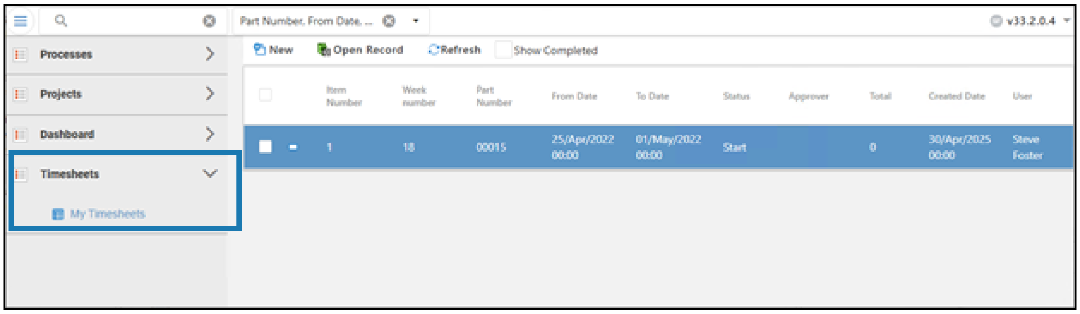
You can preview related files that include primary files for document and PDM objects in the Related Files tab.

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

To preview related files:

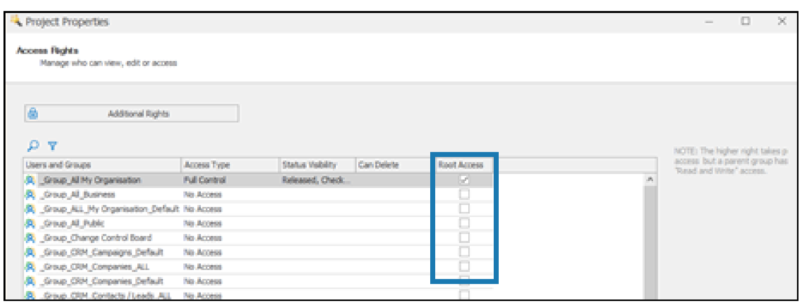
1. Click Related Files in the ribbon.
2. Select **Preview**.

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



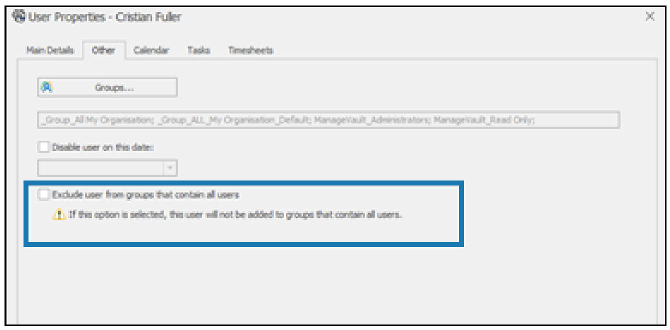
You can grant access to an object's root location for specific users and groups.

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



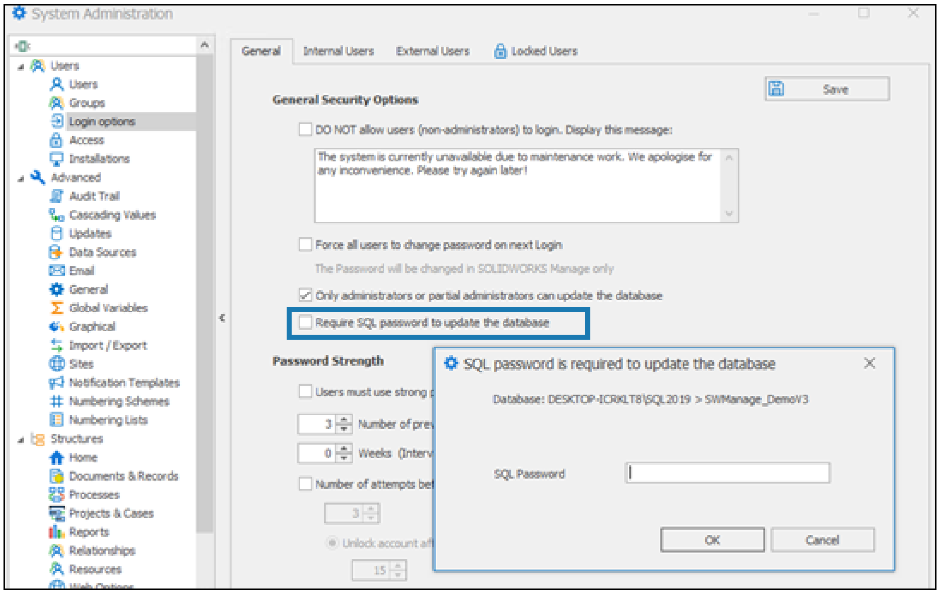
You can exclude new Manage-only users from groups that automatically include all users.

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

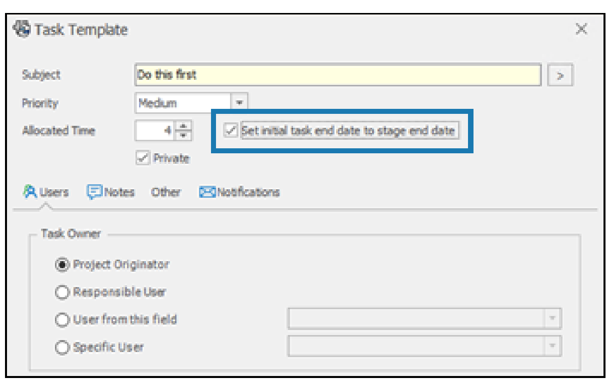


You can secure database updates with an SQL password.

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**.

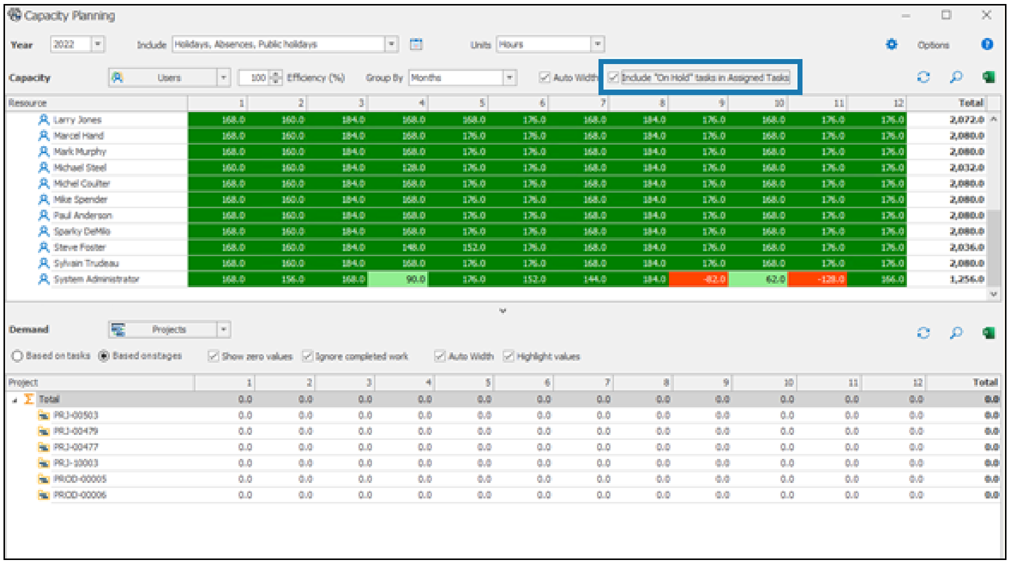
Set the End Date for a Task



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



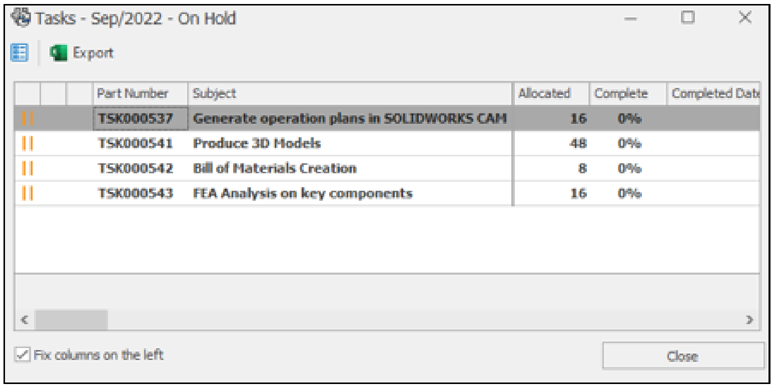
You can identify tasks with the status **On Hold**. This status indicates that you cannot work on the task yet.

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.

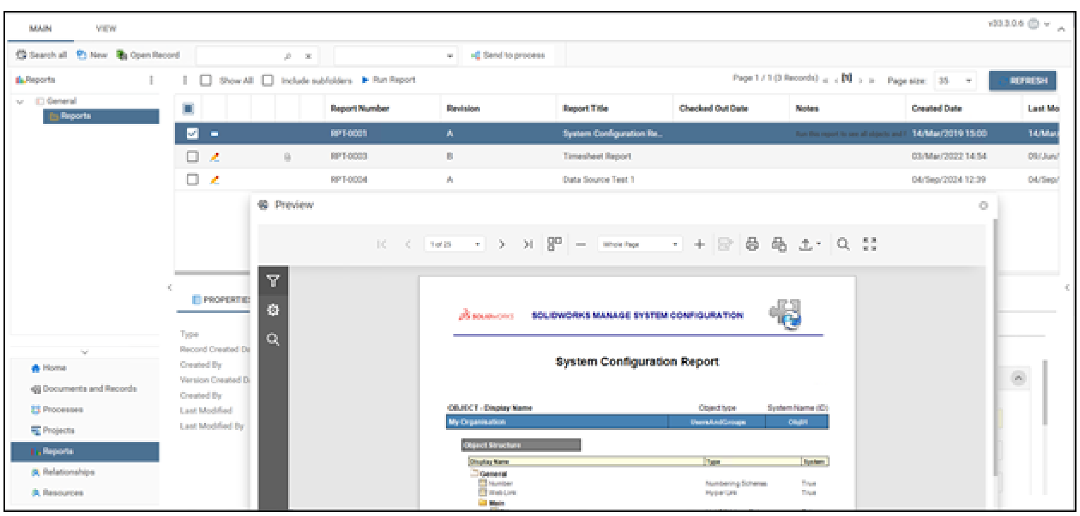
Viewing Task Details from the Capacity Planning Tool



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



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.

16

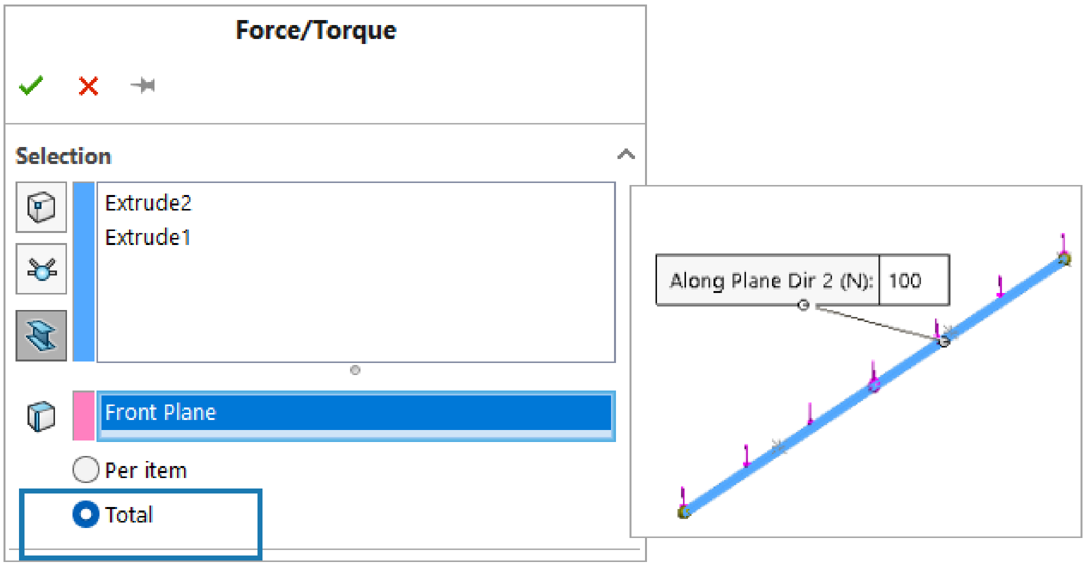
SOLIDWORKS Simulation

This chapter includes the following topics:

- **Applied Forces on Beams**
- **Buckling Studies**
- **Cable Connector (2026 SP1/FD01)**
- **Display of Angular Deformations**
- **Distributed Remote Load on Shell Edges**
- **Improved Accuracy for Gravity Loads (2026 SP1/FD01)**
- **Improved Accuracy for Free Body Forces (2026 SP1/FD01)**
- **Licensing Updates**
- **Enabling SIMULIA Compatibility for Simulation Workflows (2026 SP1/FD01)**
- **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.

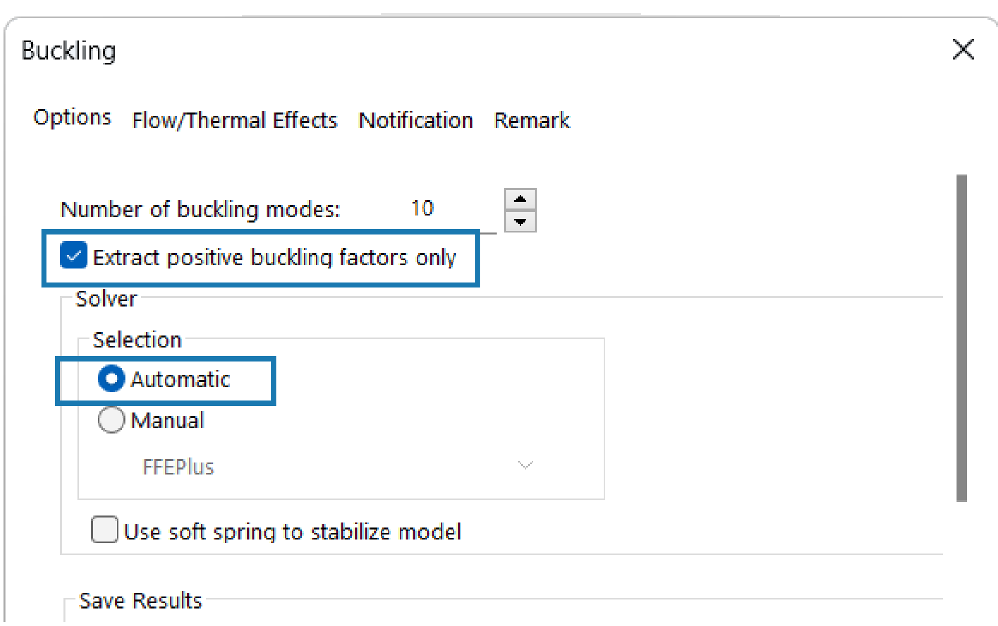
Applied Forces on Beams



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.

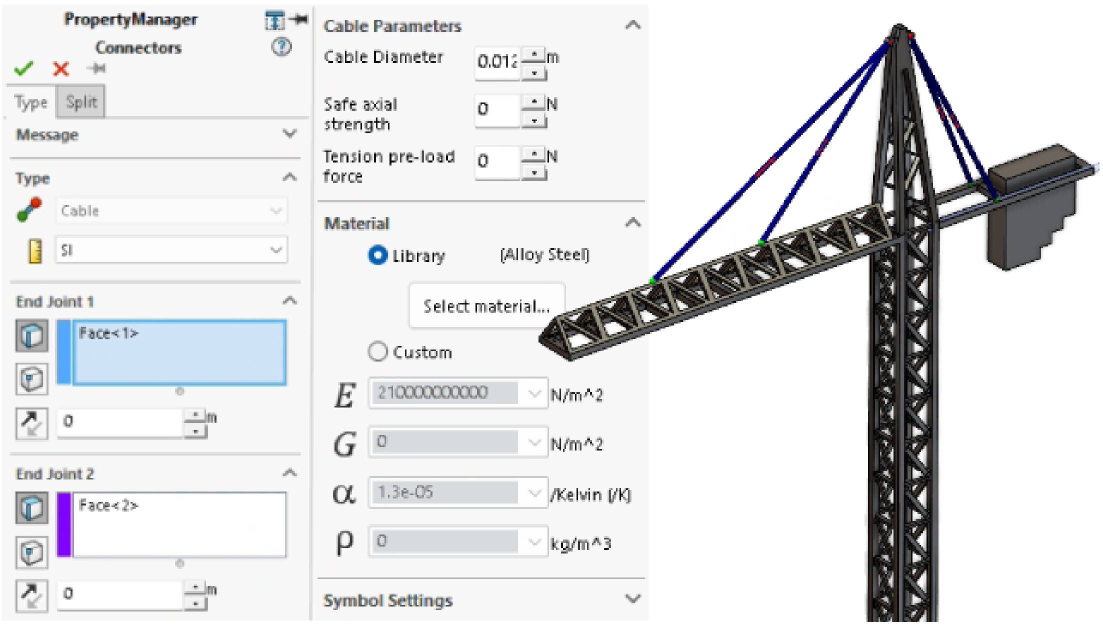
Buckling Studies



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.

Cable Connector (2026 SP1/FD01)



You can specify a cable in tension between faces (of solids), circular edges (of shells), or vertices (of solids or shells), or a combination of vertices to faces or vertices to circular edges or circular edges to faces.

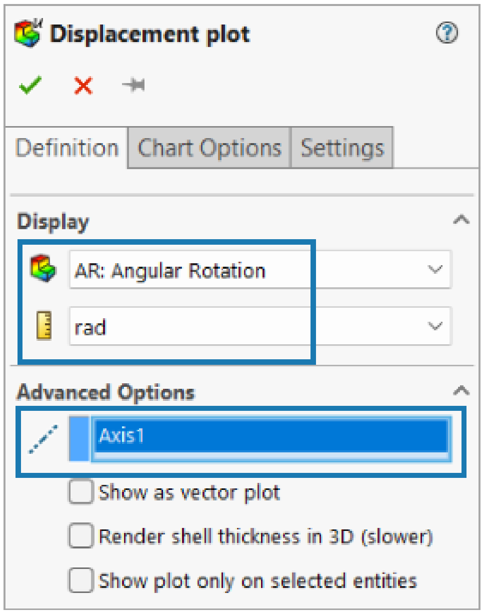
You can use a cable connector to accurately model cable elements, which are commonly used in cable-stayed bridges, cranes, and other load-bearing structural elements. To model a cable, you must specify the cable's diameter and material. You can also specify the **Safe axial strength** which is useful during post processing to determine whether the cable in tension can withstand the applied axial loads.

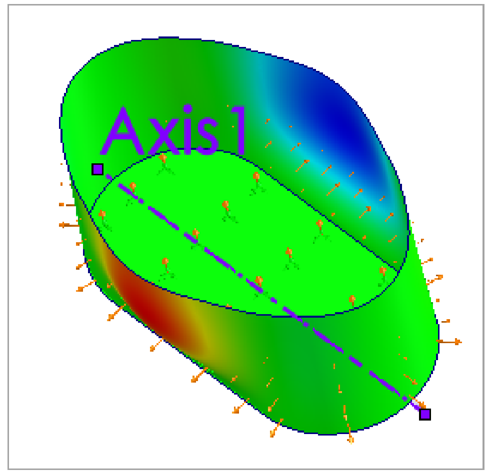
To define a cable connector, in the Simulation study tree, right-click **Connections** and click **Cable** .

To list the axial force acting on a cable connector after you run a simulation, right-click **Results** and click **List Connector Force**.

The cable connector is supported in linear static studies and is available with SOLIDWORKS Simulation Professional and SOLIDWORKS Simulation Premium.

Display of Angular Deformations



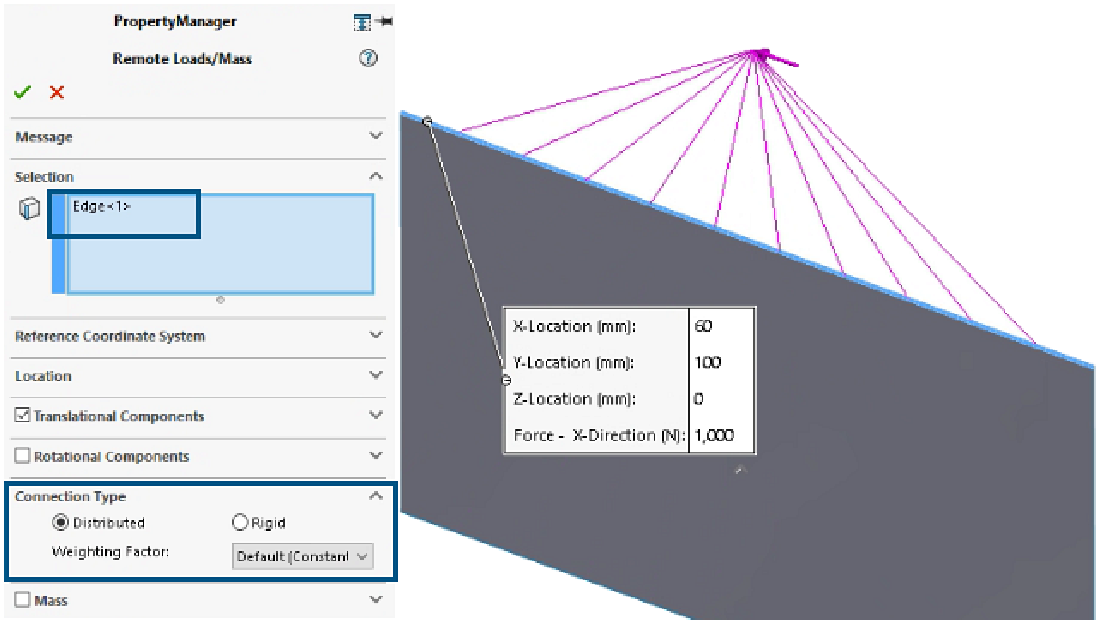


You can plot angular deformation results with respect to a given axis in units of degrees or radians.

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

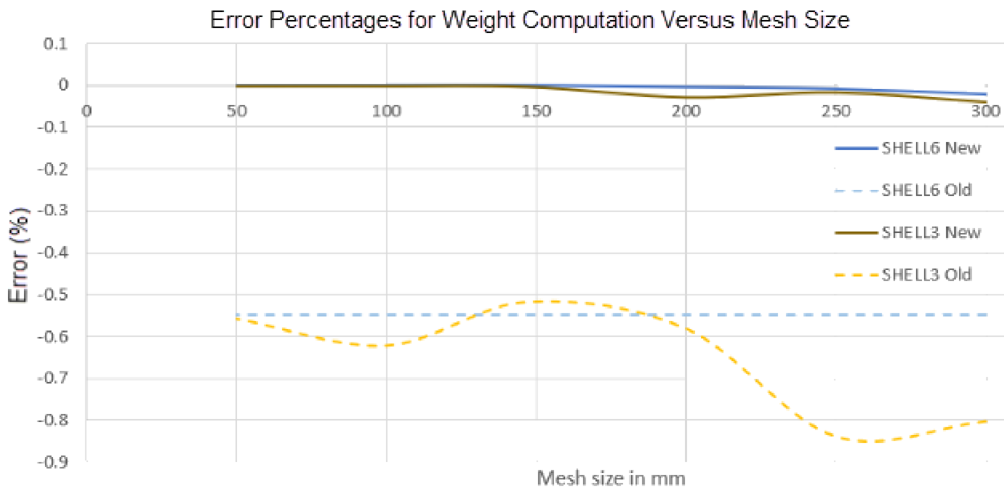


The distributed coupling formulation for remote load and remote mass now supports shell edges.

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.

Improved Accuracy for Gravity Loads (2026 SP1/FD01)

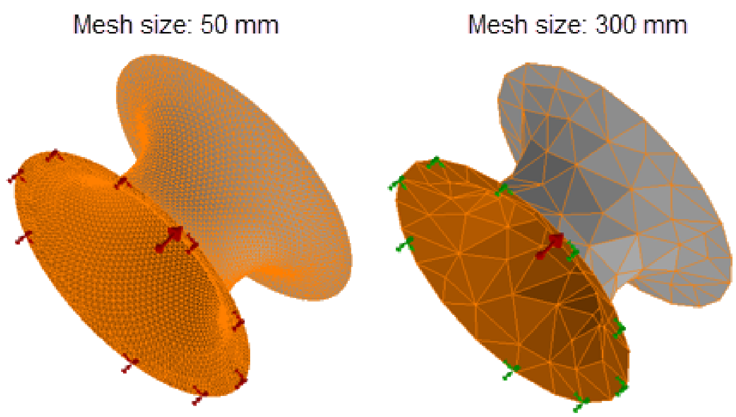


The calculation of gravity loads for curved edges and surfaces that are meshed with shells is more accurate.

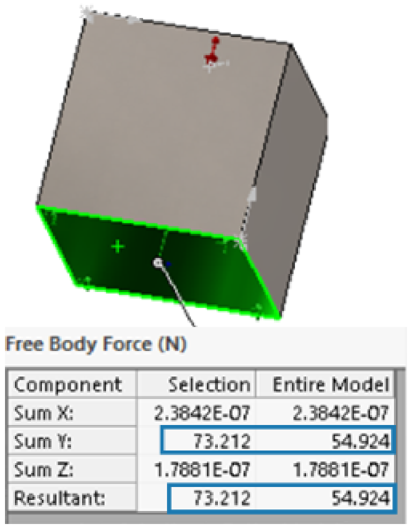
The algorithm for computing volumes of curved geometries that are meshed with shell elements has been enhanced, leading to higher fidelity in the evaluation of structural mass and gravity loads. The calculation of shell element volumes is now performed by generating wedge elements that incorporate both the shell thickness and the nodal connectivity of the shell elements.

The improvement in accuracy for weight calculations is most noticeable for double-curved geometries, as well as for curved geometries meshed with draft-quality shells, shells with offsets, and composite shells. This enhancement is available for linear static studies.

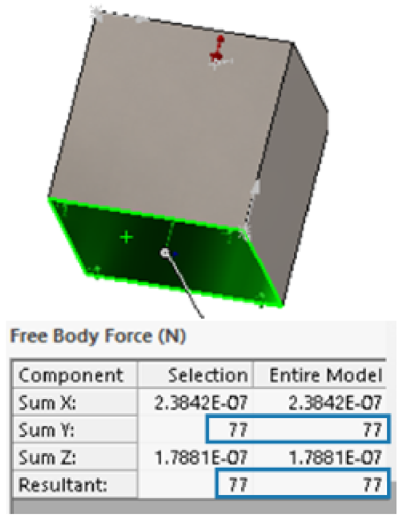
The image above shows the error percentages for the weight calculations of an anticlastic geometry meshed with shell elements, using the new (SHELL6 New and SHELL3 New) and old (SHELL6 Old and SHELL3 Old) algorithms. The new algorithm significantly reduces the error percentages for the weight calculations for the anticlastic geometry, which is shown in the image below.



Improved Accuracy for Free Body Forces (2026 SP1/FD01)



2025



2026

The calculation of free body forces for models with applied gravity and centrifugal loads is improved for linear static studies. In addition, you can list free body forces at the body level.

The image above compares the current and earlier free body forces for a 1 m^3 cube with a density of 7,700 Kg/m^3 and an applied gravity load of g = 10 N/s^2. The cube is constrained at its bottom face.

In SOLIDWORKS Simulation 2026, the free body forces for the bottom face and also the entire cube are shown correctly as 77 N, which is the cube's body weight.

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.

Enabling SIMULIA Compatibility for Simulation Workflows (2026 SP1/FD01)

Administrators can manage compatibility with Simulia apps when saving to the **3D**EXPERIENCE platform. This functionality is available in the MySession action bar under **Tools** > **Options** > **Save**.

Benefits: This lets SOLIDWORKS Design enable:

- Parameter publishing only when required for simulation workflows.
- A stable ID for the physical products that are interchangeable across a CAD family.

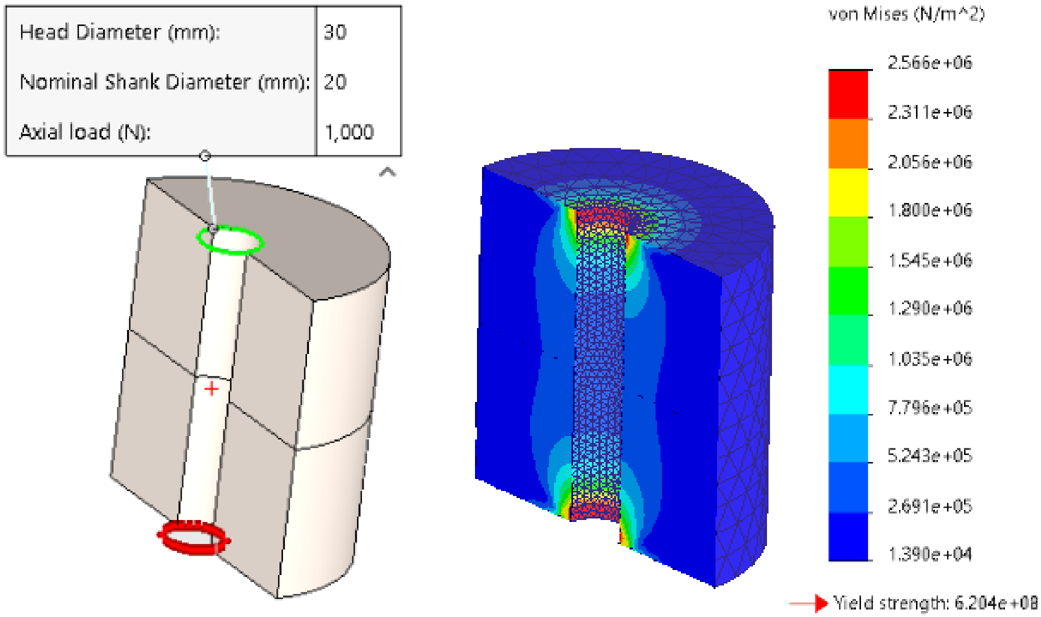
When turned off, SOLIDWORKS Design skips publishing certain PLM parameters to the **3D**EXPERIENCE platform and assigns a unique ID to each physical product in a CAD family. This reduces save time and improves performance for most users by skipping unneeded parameter publishing.

This functionality is necessary because certain simulation workflows depend on having SOLIDWORK Design parameters available in the platform. When turned on, SOLIDWORKS Design publishes parameters to the platform and assigns the same ID to all physical products in a CAD family. This allows SIMULIA tools to reuse parameters across configurations. Publishing parameters adds additional processing which can increase save time depending on the data size.

Limitations:

- Parameters published in the platform remain in the platform if you delete them in SOLIDWORKS Design.
- CAD families saved with the functionality off do not have stable IDs, even after you enable it.
- If you save data with the functionality off and later enable it, you must rebuild (for example, press **Ctrl + Q**) and resave to publish parameters.
- Save times vary with data size. Test large assemblies before using the functionality for all users.

Performance Improvement for Studies with Connectors



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

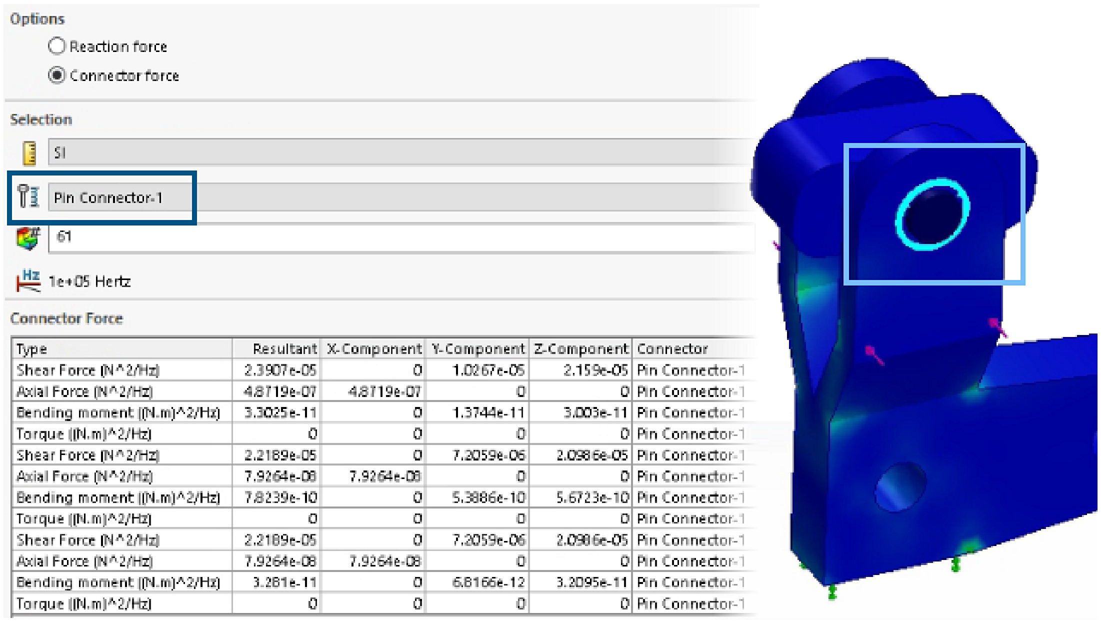
Expected solver enhancements include:

- 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



Type	Resultant	X-Component	Y-Component	Z-Component	Connector
Shear Force (N^2/Hz)	2.3907e-05	0	1.0267e-05	2.159e-05	Pin Connector-1
Axial Force (N^2/Hz)	4.8719e-07	4.8719e-07	0	0	Pin Connector-1
Bending moment ((N.m)^2/Hz)	3.3025e-11	0	1.3744e-11	3.003e-11	Pin Connector-1
Torque ((N.m)^2/Hz)	0	0	0	0	Pin Connector-1
Shear Force (N^2/Hz)	2.2189e-05	0	7.2059e-06	2.0986e-05	Pin Connector-1
Axial Force (N^2/Hz)	7.9264e-08	7.9264e-08	0	0	Pin Connector-1
Bending moment ((N.m)^2/Hz)	7.8239e-10	0	5.3886e-10	5.6723e-10	Pin Connector-1
Torque ((N.m)^2/Hz)	0	0	0	0	Pin Connector-1
Shear Force (N^2/Hz)	2.2189e-05	0	7.2059e-06	2.0986e-05	Pin Connector-1
Axial Force (N^2/Hz)	7.9264e-08	7.9264e-08	0	0	Pin Connector-1
Bending moment ((N.m)^2/Hz)	3.281e-11	0	6.8166e-12	3.2095e-11	Pin Connector-1
Torque ((N.m)^2/Hz)	0	0	0	0	Pin Connector-1

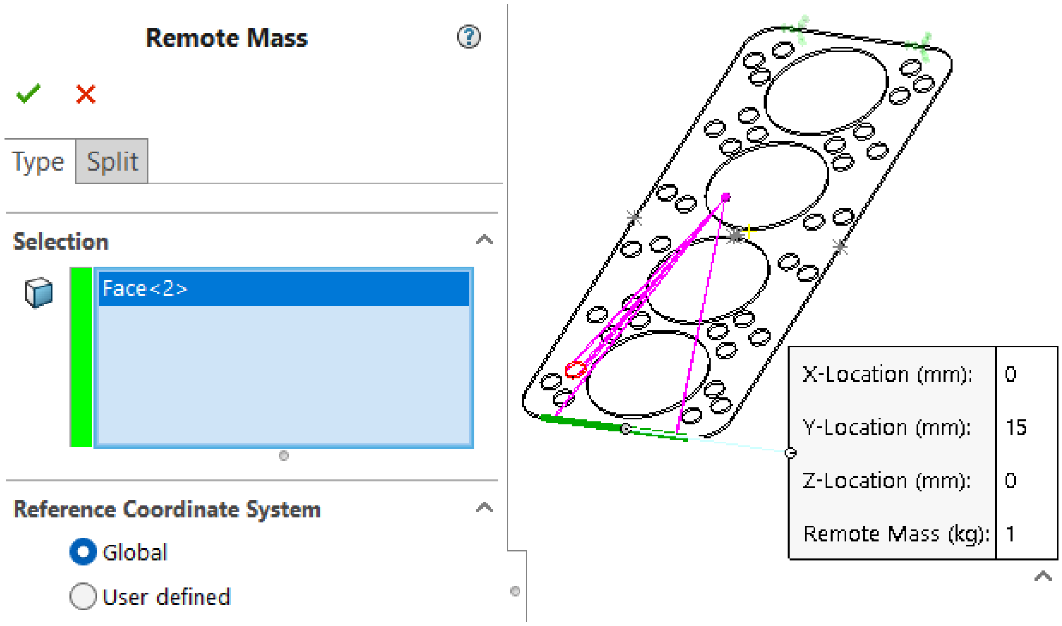
You can extract pin connector forces, including shear force, axial force, bending moment, and torque, in linear dynamic random vibration studies.

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



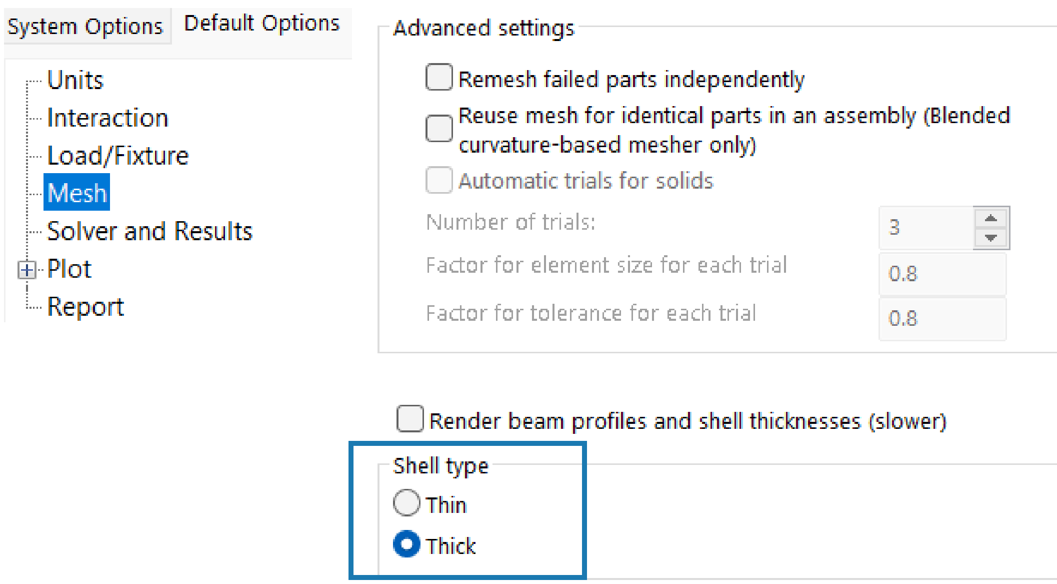
Response spectrum analysis studies support the application of remote masses.

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.

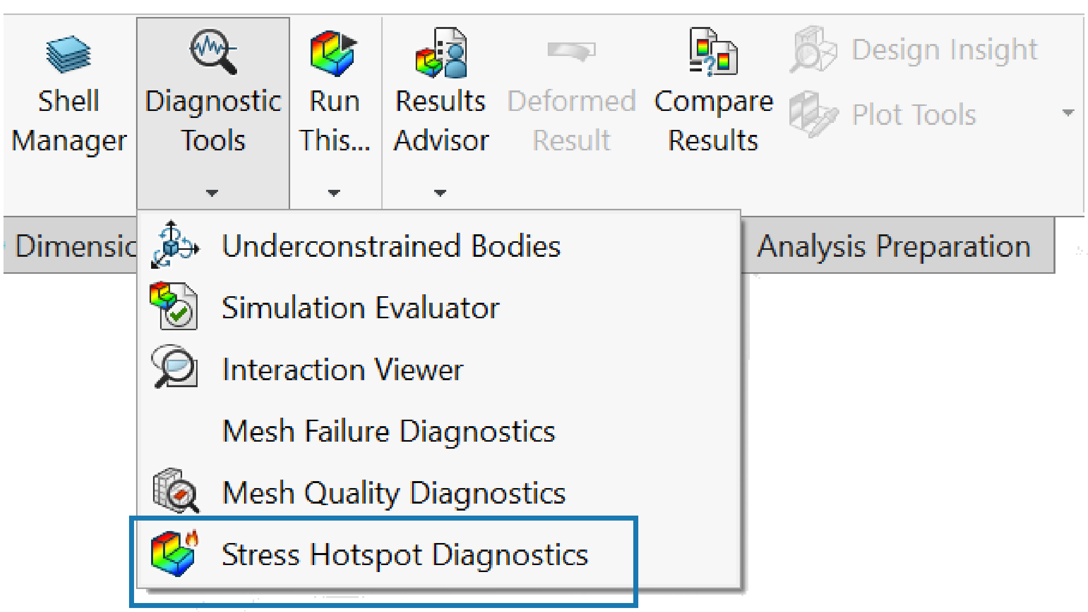
Shell Definitions



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.

17

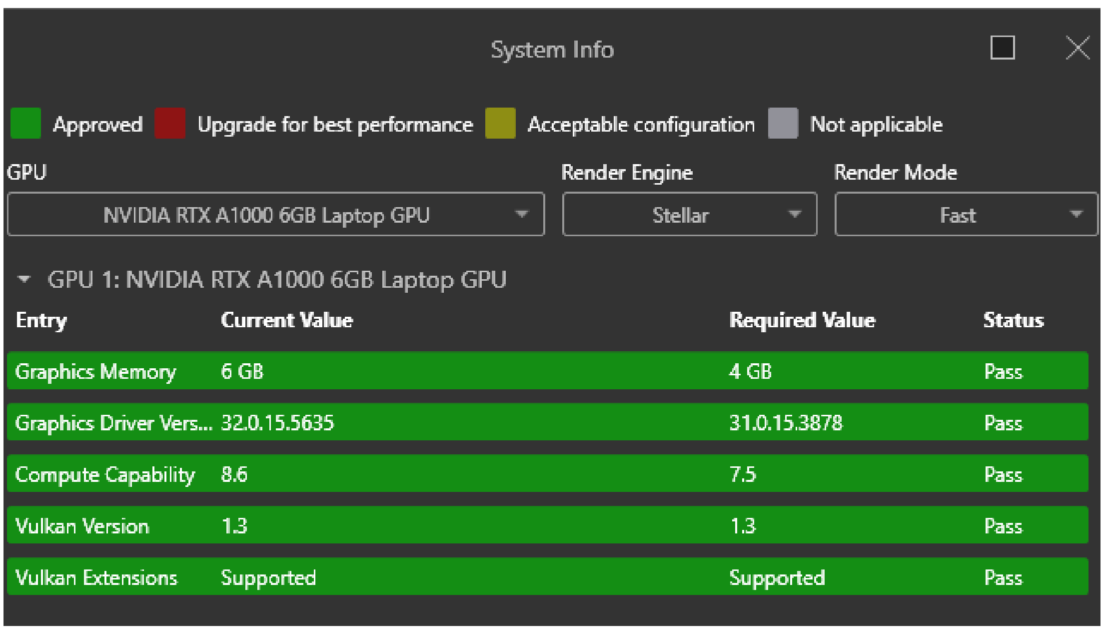
SOLIDWORKS Visualize

This chapter includes the following topics:

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

SOLIDWORKS® Visualize is a separately purchased product.

Support for AMD Hardware in Stellar Fast Render Mode



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).

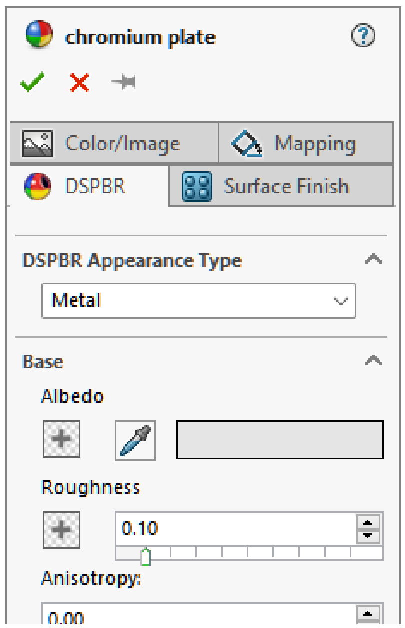
Different render engines and render modes require different minimum requirements and GPU compatibility. Therefore, the System Info dialog box is redesigned to reflect the support for additional hardware.

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 Design



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

In the Appearances PropertyManager, a DSPBR tab ensures direct material-to-material translation. All parameters from SOLIDWORKS Design 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 Design 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.

Additional DSPBR Controls (2026 SP1/FD01)

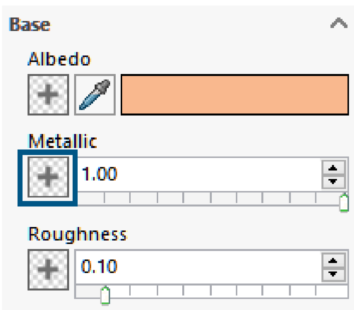
Additional parameters are available in the Appearances PropertyManager so you can fine-tune the DSPBR appearances you apply to models. These help you create richer, more physically accurate materials directly in SOLIDWORKS Design.

To open this PropertyManager:

1. In the DisplayManager, click **View Appearances**.
2. Right-click an appearance and select **Edit Appearance**.
3. In the PropertyManager, click the DSPBR tab.

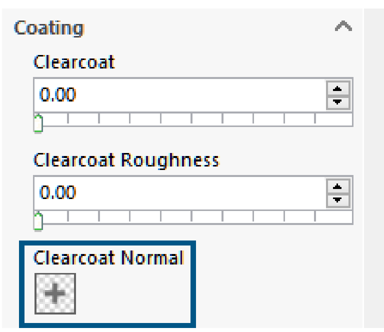
Metallic, Transparency, and Emission Color Textures

You can produce more realistic materials with additional controls. For the **Metallic**, **Transparency**, and **Emission Color** texture maps, you can browse, assign, or clear textures by clicking.



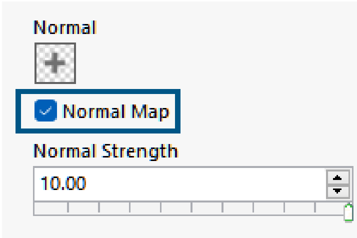
Normal Strength and Clearcoat Normal Strength Controls

You can adjust **Normal** strength and **Clearcoat Normal** strength to fine-tune surface detail intensity.



Normal Map Mode

Normal Map mode uses a non grayscale image describing normal red green blue channels (normal vector XYZ). When cleared, the app uses a greyscale heightmap as a quick normal approximation.


Normal
Normal Map
Normal Strength
10.00

18

SOLIDWORKS CAM

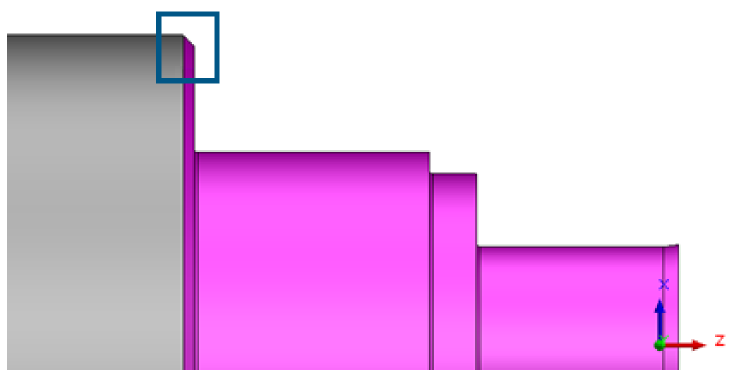
This chapter includes the following topics:

- **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 Design license that has SOLIDWORKS Subscription Service.

SOLIDWORKS CAM Professional is a separately purchased product.

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.

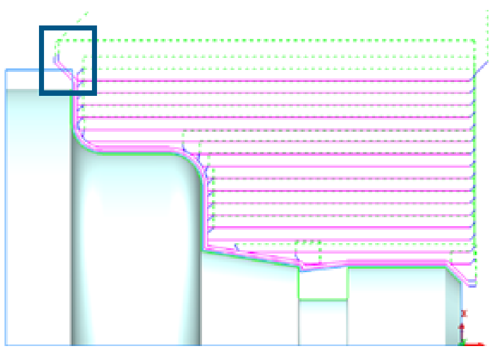
During the Turn toolpath machining process, burrs (unwanted sharp edges), can form near the tool inserts when the edges of the cylindrical stock are machined. Burrs can cause damage when the stock material slides through guide bushings. To eliminate burr formation, you can specify an option for bar break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

- Turn Rough
- Turn Finish

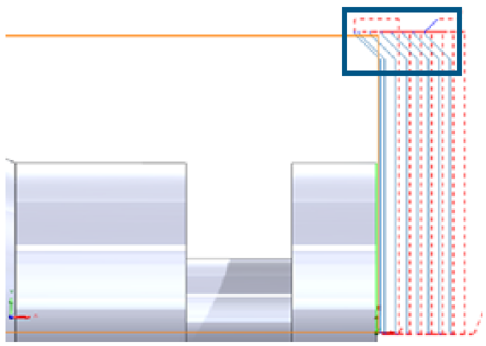
- Groove Rough (OD features only)
- Groove Finish (OD features only)
- Face Rough
- Face Finish

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.

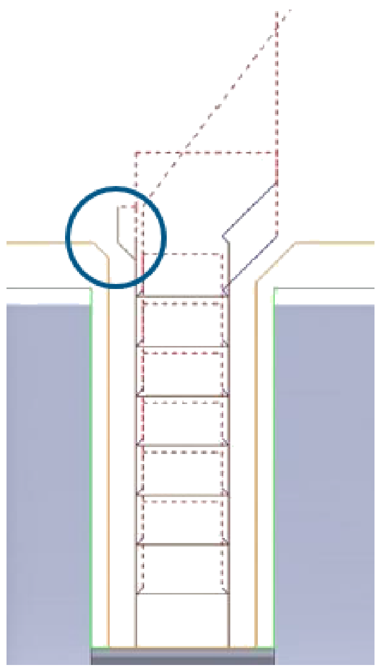


Turn Rough toolpath with bar break chamfers on first and final cut passes

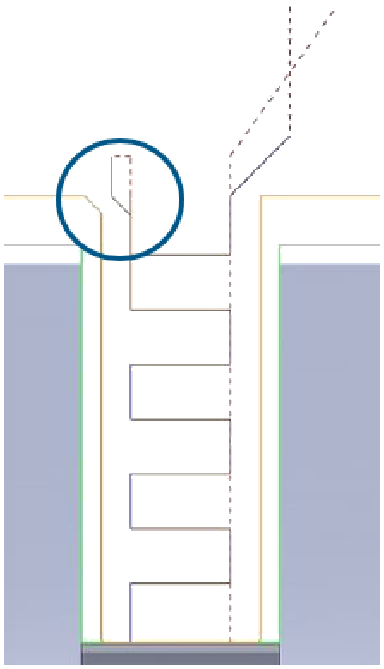


Face Rough toolpath with bar break chamfers for each cut pass

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.

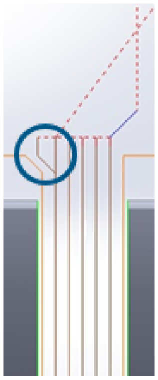


Groove Rough toolpath with zig groove style



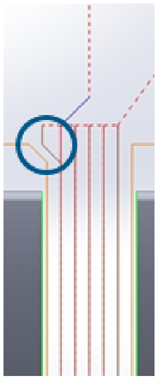
Groove Rough toolpath with zigzag groove style

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



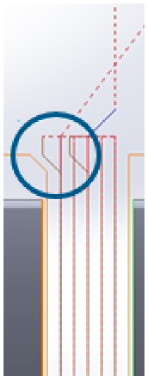
Order: S123

Bar break move added before final cut pass



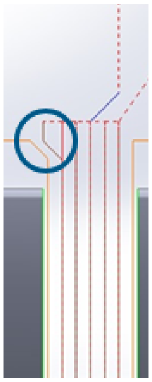
Order: S321

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

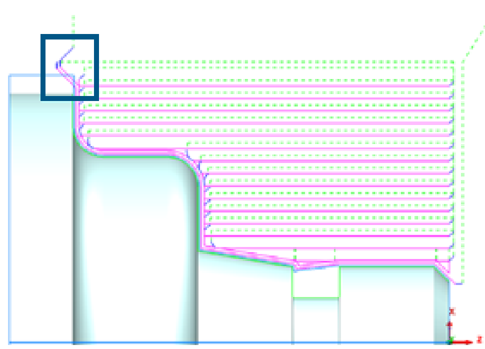
To specify bar break chamfering of stock in turn toolpaths:

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

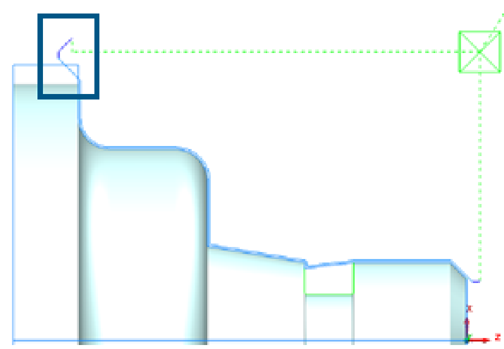
Method	Description
Chamfer	Chamfers the sharp edges of the stock along the OD feature. Specify the **Distance** and **Angle** of the chamfer.
Radius	Fillets the sharp edges of the stock along the OD feature. Specify the **Distance** and **Radius** of the fillet.
%	Specifies the chamfer distance as a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

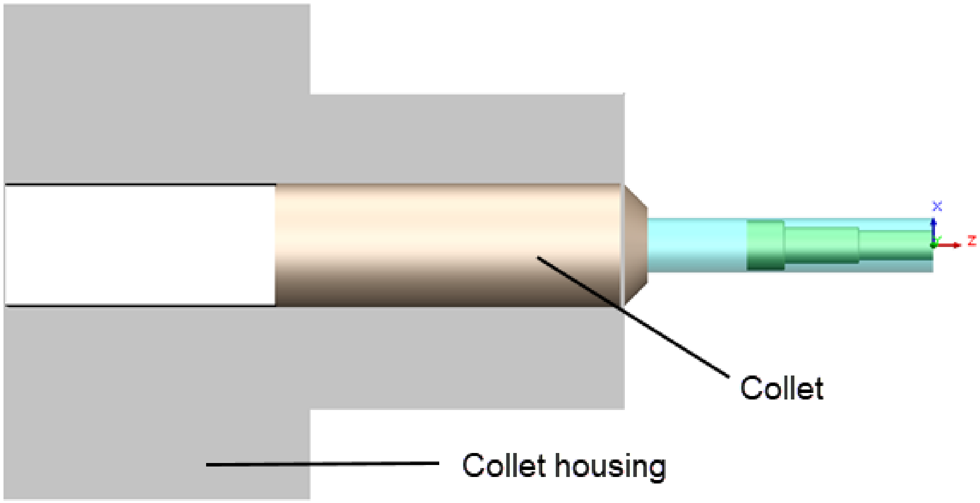


Turn Rough toolpath with **Reverse Cut type**. Bar break chamfers are added to the first and final cut passes.



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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CircuitWorks

CircuitWorks™ is available in SOLIDWORKS® Standard, SOLIDWORKS Professional, SOLIDWORKS Premium, and SOLIDWORKS Ultimate.

Build Model Performance for ECAD Files (2026 SP1/FD01)

You can build a 3D model faster from `ECAD` files (`IDF`, `PADS`, `IDX`).

> To improve **Build Model** performance for `ECAD` files, in **Tools** > **CircuitWorks** > **CircuitWorks Options** > **SOLIDWORKS Import**, clear **Show part creation in SOLIWORKS**. By default, **Show part creation in SOLIWORKS** is clear.

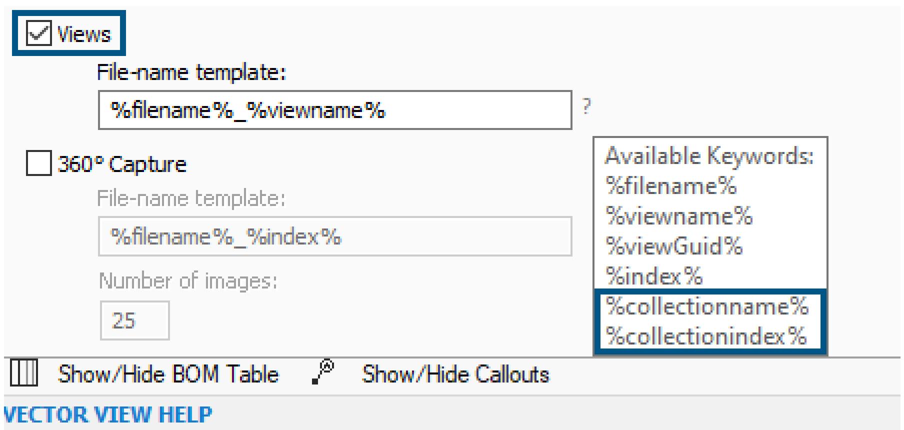
20

SOLIDWORKS Composer

This chapter includes the following topics:

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

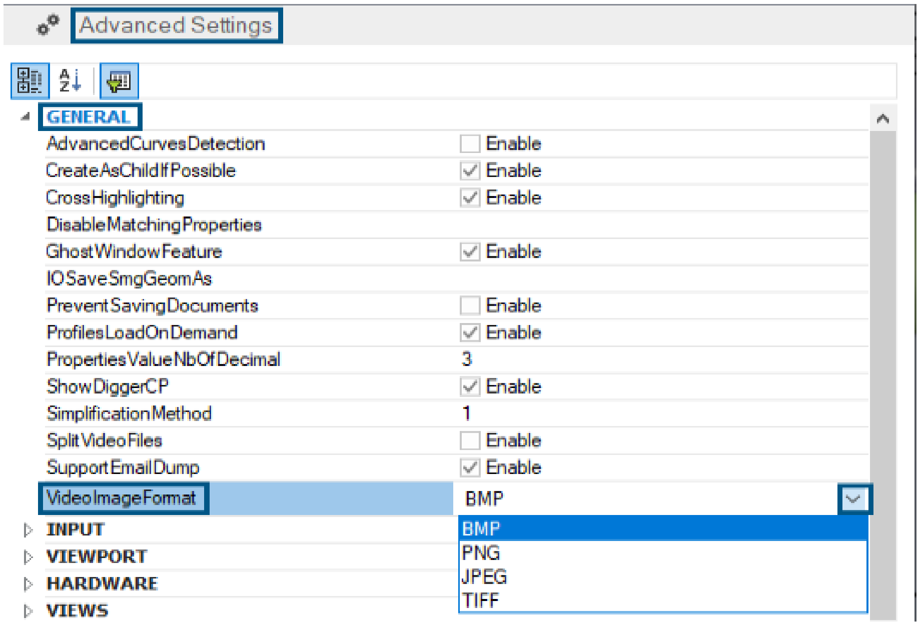
Filename Template Options for Workshops



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

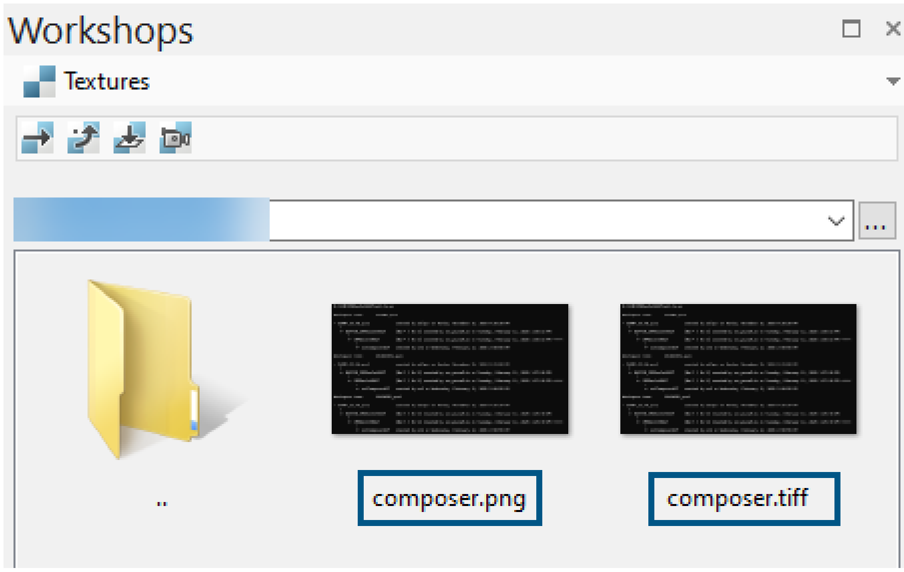


You can use BMP, PNG, JPEG, and TIFF image formats for animation frames while creating videos. You can create videos using the Video Workshop.

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.

21

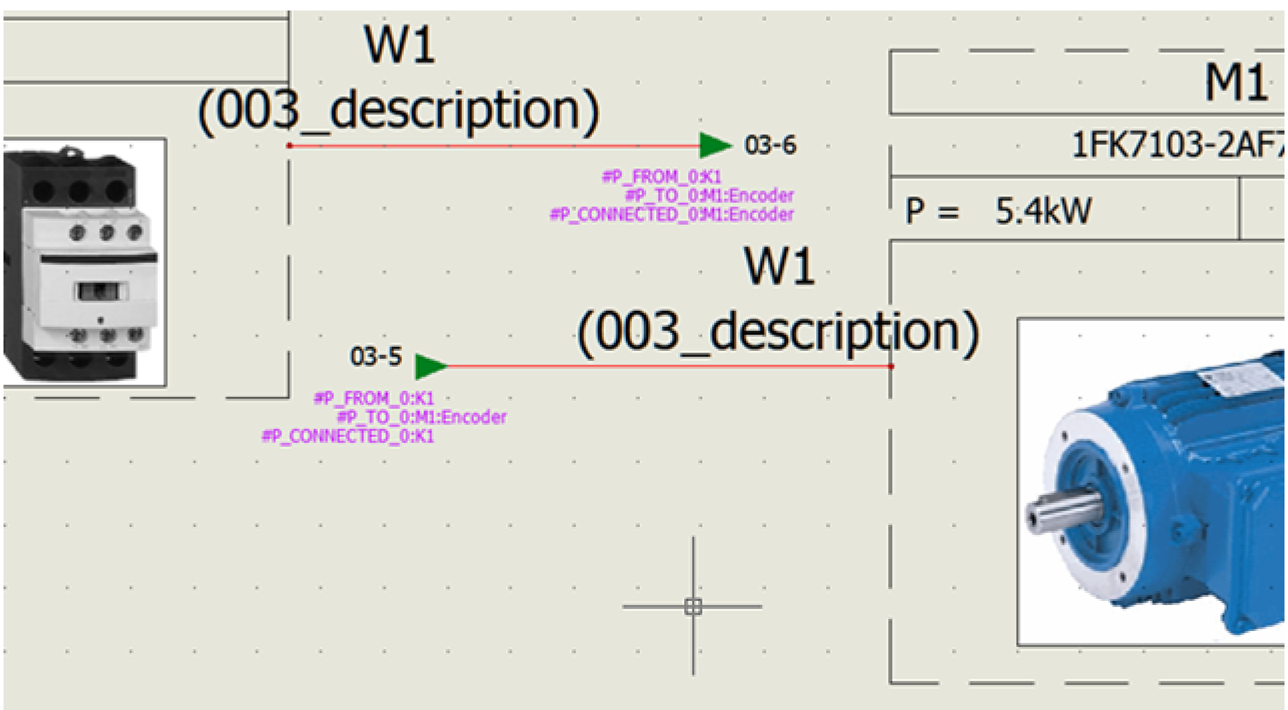
SOLIDWORKS Electrical

This chapter includes the following topics:

- **Project Management Productivity (2026 SP1/FD01)**
- **Draw Multiple Terminal Strips Side by Side (2026 SP1/FD01)**
- **Update Dynamic Connector after Insertion (2026 SP1/FD01)**
- **TraceParts Publisher in the Electrical Content Portal (2026 SP1/FD01)**
- **Cable Management**
- **Hiding System Classes**
- **Routing Selected Wires Separately**
- **Connector Dynamic Insertion**
- **Update and Replace Project Data**

SOLIDWORKS® Electrical is a separately purchased product.

Project Management Productivity (2026 SP1/FD01)

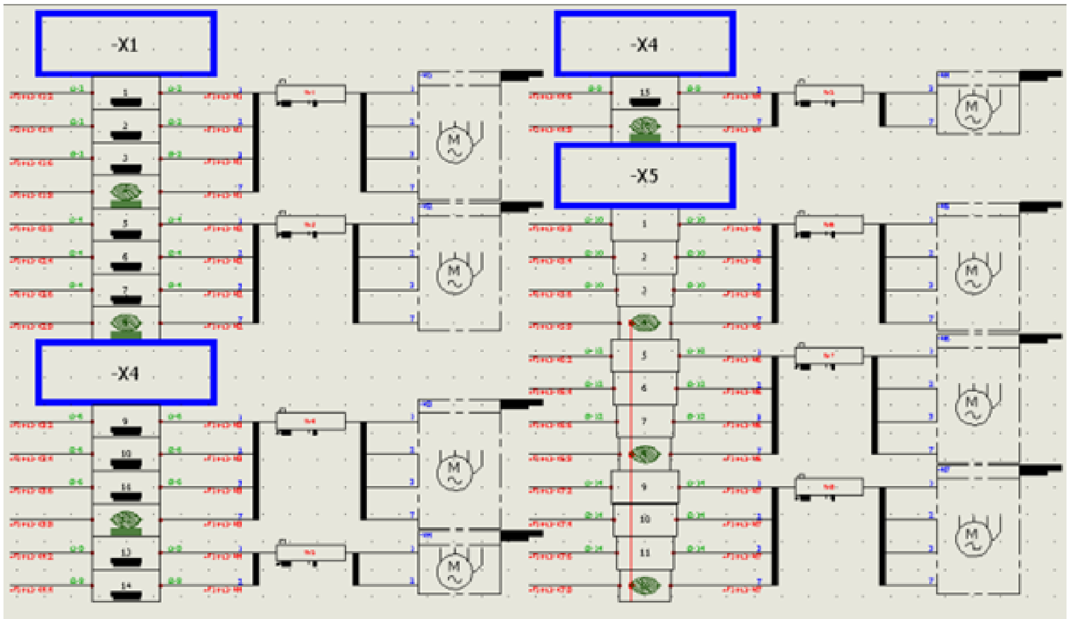


Project management productivity is enhanced to complete project tasks faster with fewer clicks and by eliminating previous workflow restrictions.

Enhancements:

- You can insert origin-destination arrows for wires with different styles without manual wire style changes. In the Origin-destination management dialog box, a new option **Insert for identical wire style only** is added to **Single insertion** tool. Select to insert origin-destination arrows from one wire to another with or without the same style.
- Ensured consistent behavior of origin–destination arrows between multiwire and single-line diagrams. Single-line diagrams display connected component marks and connection details, matching the functionality of multiwire diagrams.
- You can duplicate a project while preserving the original book and drawing revisions, so the entire version history remains intact.
- In the Select circuit from component manufacturer part dialog box, a new option **Repeat until last symbol inserted** option is added. Select to add all the remaining symbols at once, eliminating repeated dialog openings.

Draw Multiple Terminal Strips Side by Side (2026 SP1/FD01)



You can generate multiple parallel terminal strip drawings side by side on a single page. You can also have multiple terminal strips on drawings horizontally and vertically and define its drawing space.

The **Layout** tab in the **Terminal strip drawing configuration** dialog box is enhanced to include new tools.

New Tools:

- **Drawing space**. Determines the available area for the terminal strips.
- **Parallel**.

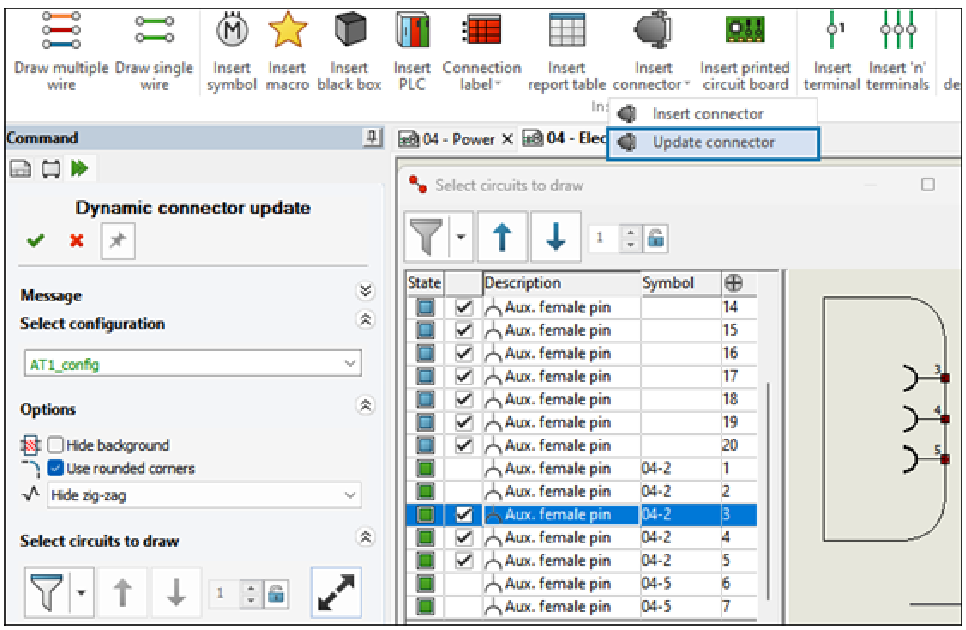
- **Activate parallel**. Arranges several terminals in multiple columns or lines on the same page.
- **Parallel spacing**. Defines the spacing between parallel terminal strips.
- **Parallel Terminal Strips count**. Displays how many strips are placed side by side in parallel.
- **New terminal strip in next parallel**. Controls whether the next terminal strip continues in the same parallel position or starts in a new one.

With the four layout options **Terminal strip orientation**, **Activate multiple**, **Activate parallel**, and **New terminal strip in next parallel**, you have up to 10 possible layout combinations. This flexibility lets you choose the configuration that best matches your drawing style and project requirements.

You can also insert manual breaks in the **Terminal Strip Editor** dialog box.

- Click **Insert or remove page break** to add the page break and force the next strip to continue on a new page.
- Click **Insert or remove parallel break** to force the next strip to continue in the next line.

Update Dynamic Connector after Insertion (2026 SP1/FD01)



The **Update connector** command is now available. It opens the new Dynamic connector update tab, which works similarly to the **Insert connector** command with some adaptations.

You can modify an existing connector symbol directly in the schematic. You can add, remove, or rearrange circuits, change options as required, and update visual configurations without deleting or reinserting the symbol.

Benefits: You can save time by avoiding delete-and-recreate cycles, reduce errors, maintain a cleaner schematic, and get instant visual preview with flexible circuit organization.

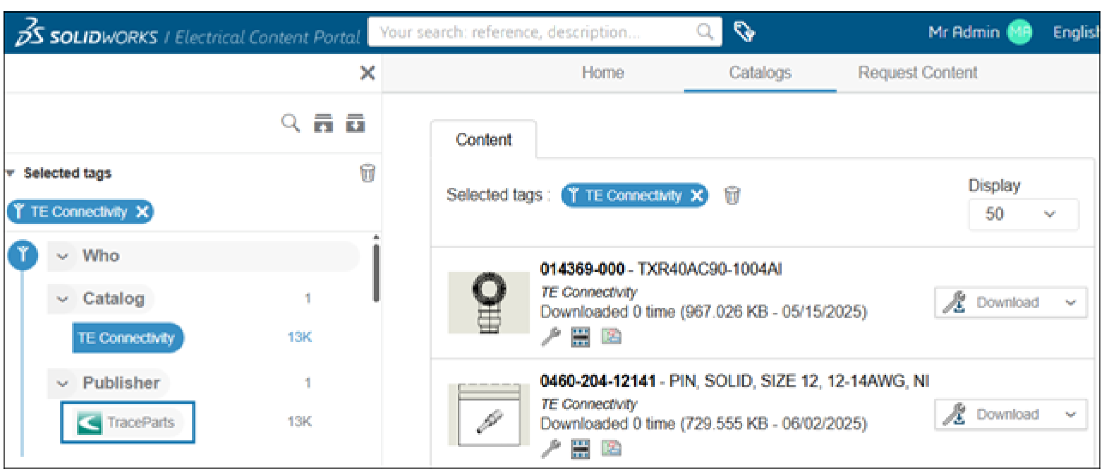
To access the Update connector, do one of the following:

- Click **Schematic** > **Insert connector** > **Update connector**.
- Right-click a connector symbol, and select **Component** > **Update connector**.

The Dynamic connector update tab contains the following sections.

- **Message**. Displays instructions to follow.
- **Select Configuration**
- **Options**. Contains the following options for the visual representation of the connector symbol:
  - **Hide background**
  - **Use rounded corners**
  - **Zig-zag option**. Lets you select the options which control the display of the connector zig-zag feature.
- **Select circuits to draw**. Contains options for modifying the circuit selection, such as applying filters and using control to reorder the circuits.
  - Selecting **Select circuits** opens the Select circuit to draw dialog box, where you can modify circuit selection and view an instant preview for selected connector.

TraceParts Publisher in the Electrical Content Portal (2026 SP1/FD01)



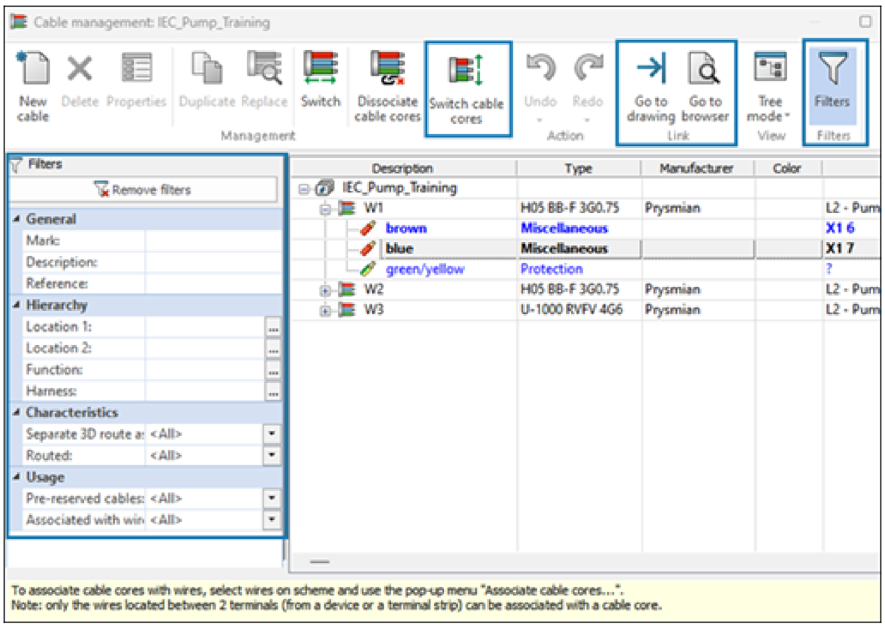
TraceParts is a new **Publisher** in the 6WTAGs filter. TraceParts is one of the world's leading CAD content platforms for engineering and a trusted manufacturer for delivering high-quality CAD data.

First large-scale project with TraceParts is delivered in collaboration with TE Connectivity, a global leading manufacturer in connectivity and sensor solutions. Through this partnership, you now have access to tens of thousands of fully detailed electrical components in the Electrical Content Portal (ECP).

The new content covers a wide range of product classes, including Connectors, Terminal strips, and Contactors.

Use the **6WTAGs** filter to select **TraceParts** as the Publisher and explore the new electrical components available in ECP.

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**

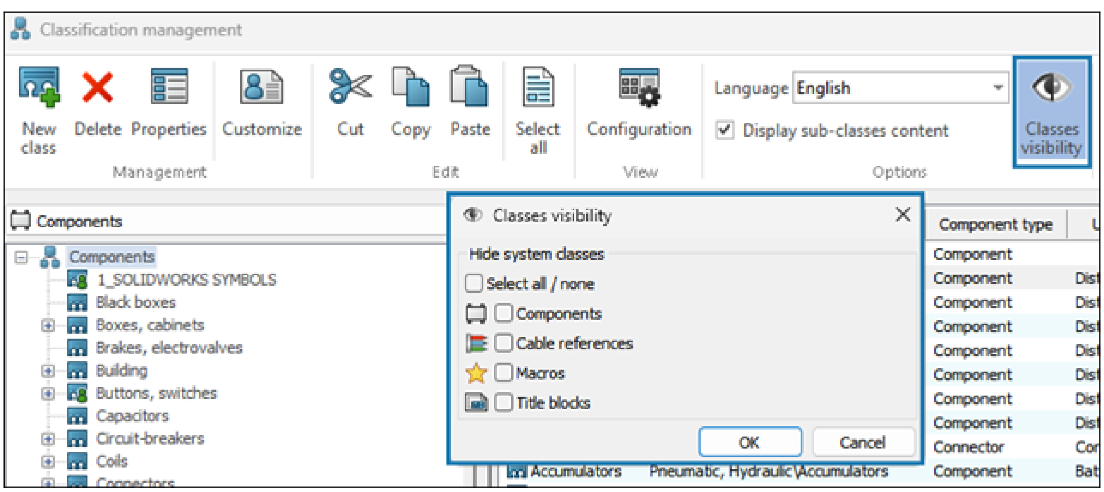
- **Pre-reserved cables**
- **Associated with wire**

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.

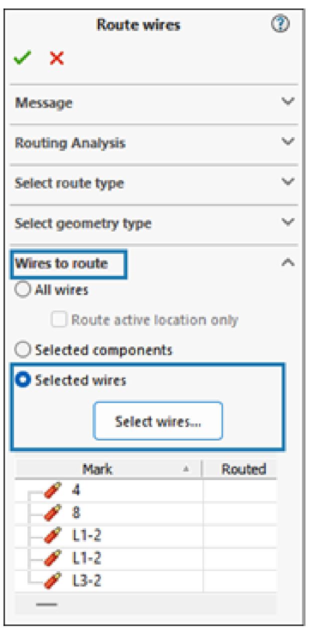
Hidden classes remain visible in the Classification Management dialog box but are removed from library and selection interfaces. Custom subclasses under system classes stay visible to preserve the hierarchy.

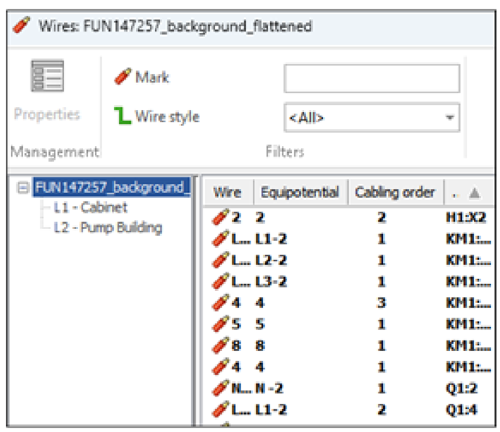
Benefits: You can find relevant components and symbols more easily.

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.

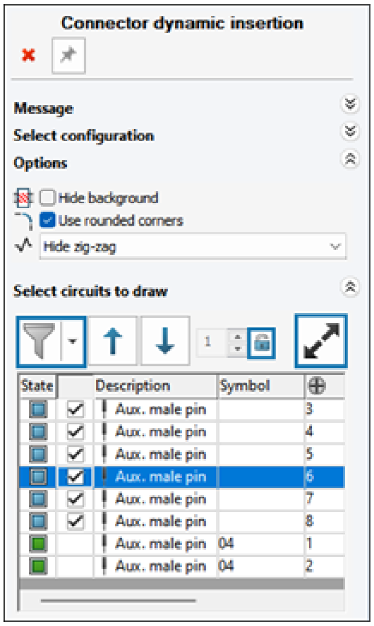
To route selected wires separately:

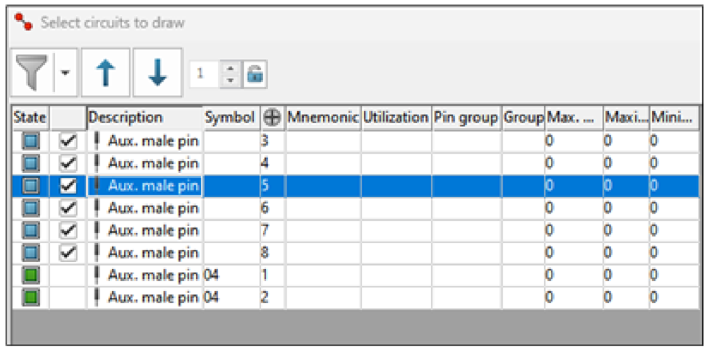
1. In the ribbon, click **Route Wires**.
2. In the **Route wires** PropertyManager, click:
   a. Selected wires.
   b. Select wires.
3. In the Wires dialog box:

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.

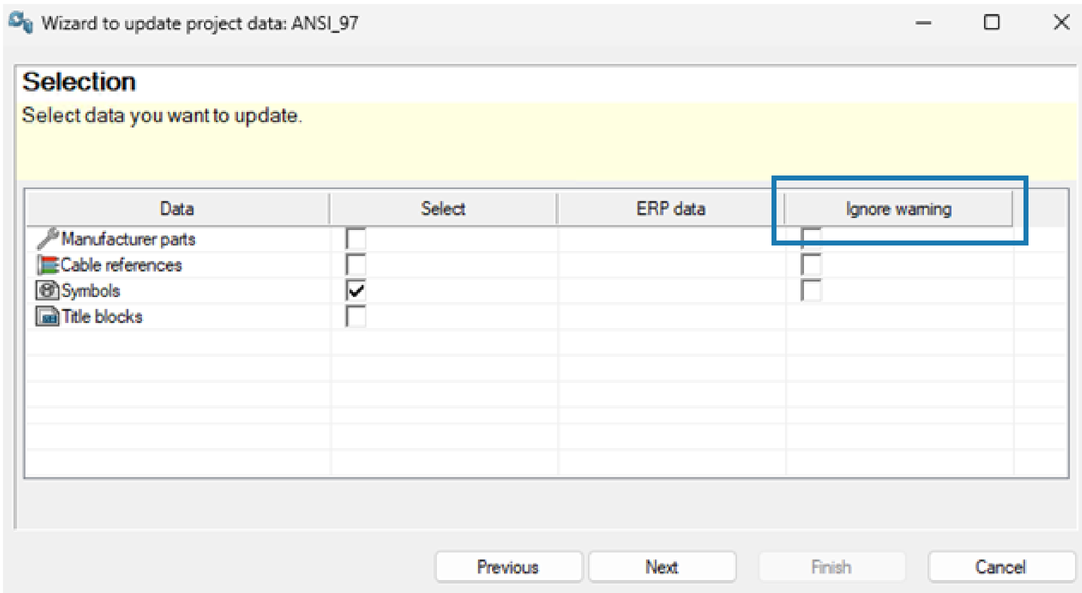
To open the Select circuits to draw dialog box:

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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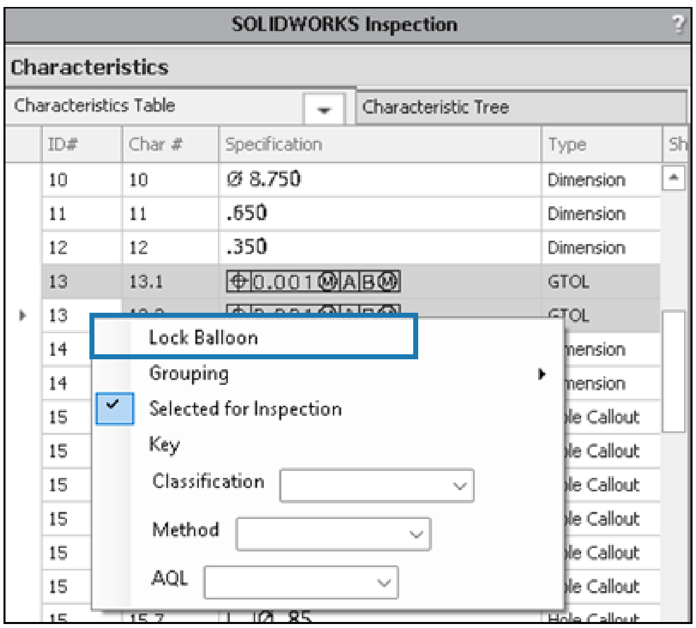
SOLIDWORKS Inspection

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

Reorder and Lock Balloons (2026 SP1/FD01)



In SOLIDWORKS Inspection Add-in, you can now reorder balloons across views and sheets. Previously, you could reorder balloons only within a view.

You can also lock balloons to existing characteristics. To lock a balloon, in the SOLIDWORKS Inspection Manager, on the Characteristic Table tab, right-click a characteristic and select **Lock Balloon**.

- If you Lock Balloon on a characteristic that is a part of group characteristics, all characteristics in that group get locked to that balloon.
- If you want to merge, unmerge, group, delete, or modify the balloon settings, unlock the balloon first.
- You cannot reorder or renumber a locked balloon.

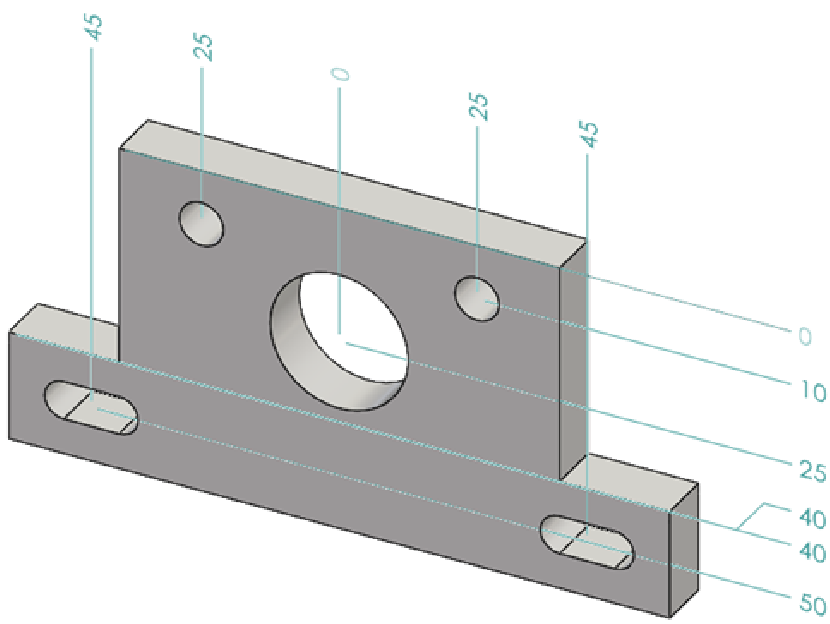
23

SOLIDWORKS MBD

This chapter includes the following topics:

SOLIDWORKS® MBD is a separately purchased product.

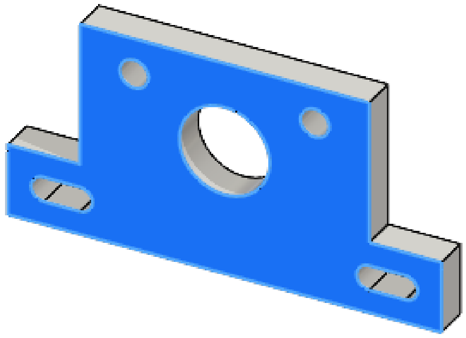
Ordinate Dimensions (2026 SP1/FD01)



You can use ordinate dimensions in parts.

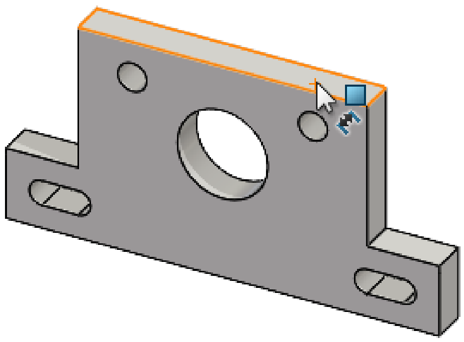
To use ordinate dimensions:

1. In a part, click **Ordinate Dimensions** (MBD or MBD Dimensions toolbar) or**Tools** > **MBD Dimension** > **Ordinate Dimensions**.
2. In the graphics area:
   a. Click the annotation view/annotation plane as the plane on which the dimensions reside.



The annotation view/annotation plane can be a plane or planar face or plane. The annotation view can be the active view, an existing view, or a new view.

b. Click the base feature.

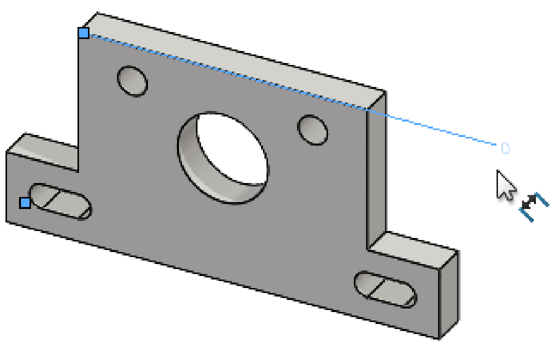


> If the base feature also defines the dimension direction (such as a plane), go to step 2d.

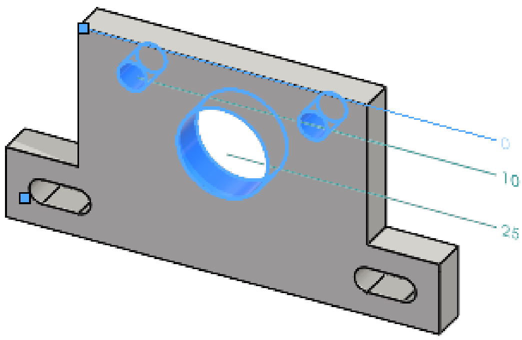
c. Specify the dimension direction, which is perpendicular to the annotation plane.

   The dimension direction can be a plane, planar face, edge, or axis.

d. Click to place the 0.0 dimension.



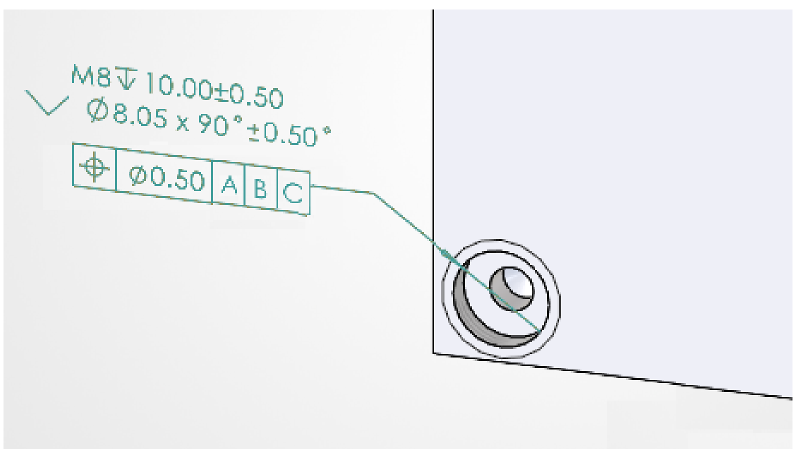
e. Specify features to dimension.



You can select any supported base feature entities. Vectors for axial features must be perpendicular to the dimension direction. Vectors for planar features must be parallel to the dimension direction.

3. Click .

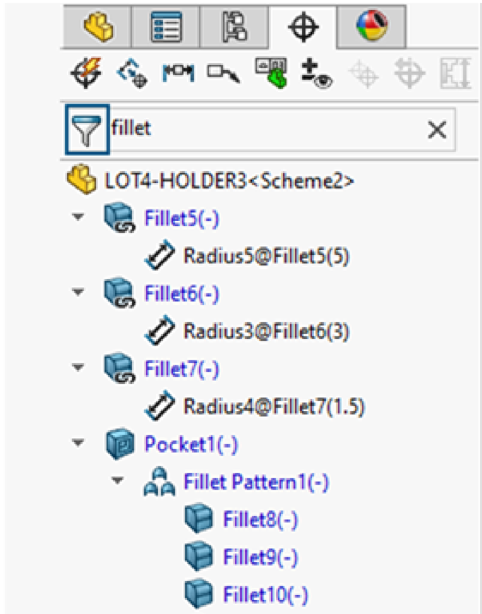
Hole Thread Descriptions (2026 SP1/FD01)



You can show full hole thread descriptions in Model Based Definition (MBD) models and drawings. This maintains a uniform display of thread descriptions.

In **Tools** > **Options** > **Document Properties** > **Drafting Standard** > **Annotations**, select **Show full thread description for all holes**.

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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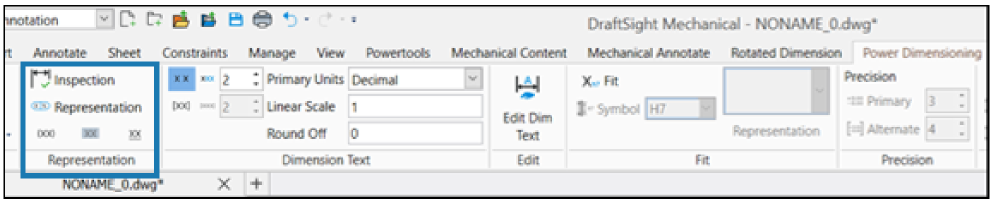
DraftSight

This chapter includes the following topics:

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. DraftSight is a combined solution of DraftSight with the power of the **3D**EXPERIENCE platform.

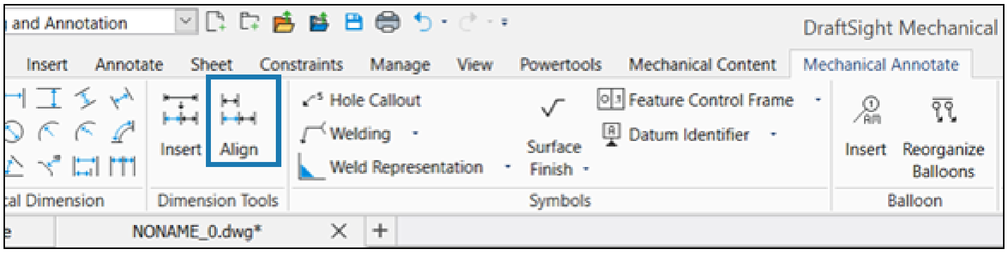
Representation Panel in Power Dimensioning Contextual Ribbon Tab (DraftSight Mechanical Only) (2026 SP1/FD01)



The **Representation** panel on the Power Dimensioning contextual ribbon lets you apply annotations to dimensions faster. This eliminates manual text overrides or complex property adjustments.

It provides quick visual tags for special dimension types to convey the design content, inspection frequency, and measurement precision. The formatting options and support for inspection dimensions offer precision and flexibility.

Align Dimension Command (DraftSight Mechanical Only) (2026 SP1/FD01)



You can use the `AM_DIMALIGN` command to align dimensions precisely.

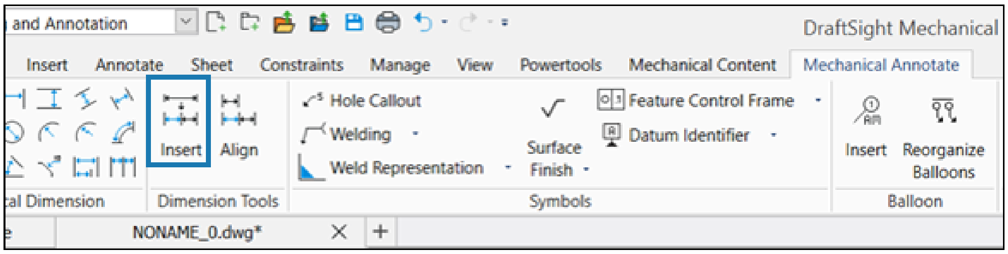
To access the command:

- In the command window, type `AM_DIMALIGN`.
- On the ribbon and menu, click **Mechanical Annotate** > **Dimension Tools** > **Align**.

This command lets you line up linear, ordinate, or angular dimensions with a selected base dimension. You can create cleaner and more readable technical drawings with minimal adjustments.

The `AM_DIMALIGN` command validates and aligns dimensions of the same type to ensure consistent orientation and spacing. It provides detailed feedback when it is unable to align dimensions because of mismatches in type, orientation, or geometry. The command enhances visual clarity and drawing accuracy in complex mechanical layouts.

Insert Dimension Command (DraftSight Mechanical Only) (2026 SP1/FD01)



You can use the AM_DIMINSERT command to split a linear or angular dimension into two separate dimensions at a specified point.

To access this command:

- In the command window, type AM_DIMINSERT.
- On the ribbon and menu, click **Mechanical Annotate** > **Dimension Tools** > **Insert**.

This command lets you line up linear, ordinate, or angular dimensions with a selected base dimension. You can create cleaner and more readable technical drawings with minimal adjustments.

You can break long dimension lines or angle spans into readable segments without manually recreating them. When you split the dimension, the app creates the new dimension from the second extension line of the original dimension. The new dimension inherits properties such as tolerance, fit, and layer assignment.

Automatically Replicating Windows Folder Structure for Bookmarks (3DEXPERIENCE Users Only) (2026 SP1/FD01)

Users who install DraftSight from the **3D**EXPERIENCE platform can use **Batch Save to 3DEXPERIENCE** to automatically create a bookmark structure that replicates the Windows folder structure.

To automatically replicate the Windows folder structure for bookmarks:

1. On the ribbon, click **DraftSight** > **Batch Save to 3DEXPERIENCE**.
2. In the Batch Save to 3DEXPERIENCE dialog box, click **Add Folder**.
3. Select a folder to upload.
4. In the Batch Save to 3DEXPERIENCE dialog box, click **Bookmark**.

5. In the Select a Bookmark dialog box, select the bookmark to which you want to upload the folder and click **Select**.
6. In the Batch Save to 3DEXPERIENCE dialog box, click **Save**.

DraftSight uploads all subfolders and `DWG` files of the selected folder to the bookmark in the same hierarchy as the Windows folder structure.

If the folder or subfolders include hidden files, the upload process stops and you get an error message.

Start Page Tab (For DraftSight Premium, Enterprise Plus, and Mechanical)

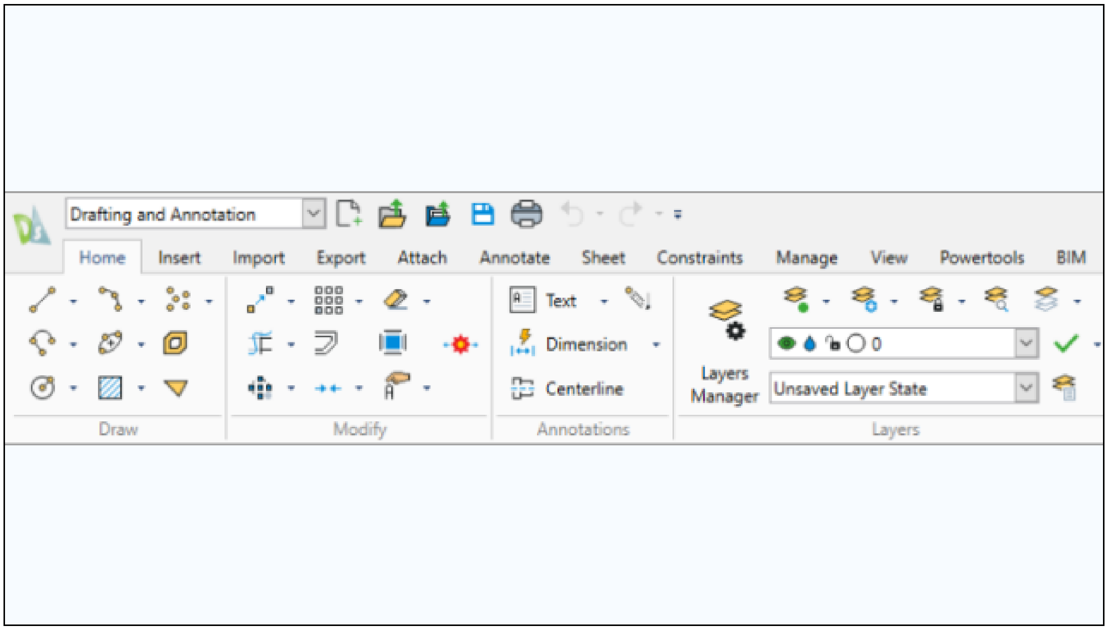
The Start Page tab centralizes key operations and creates a smoother user experience when you open DraftSight.

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

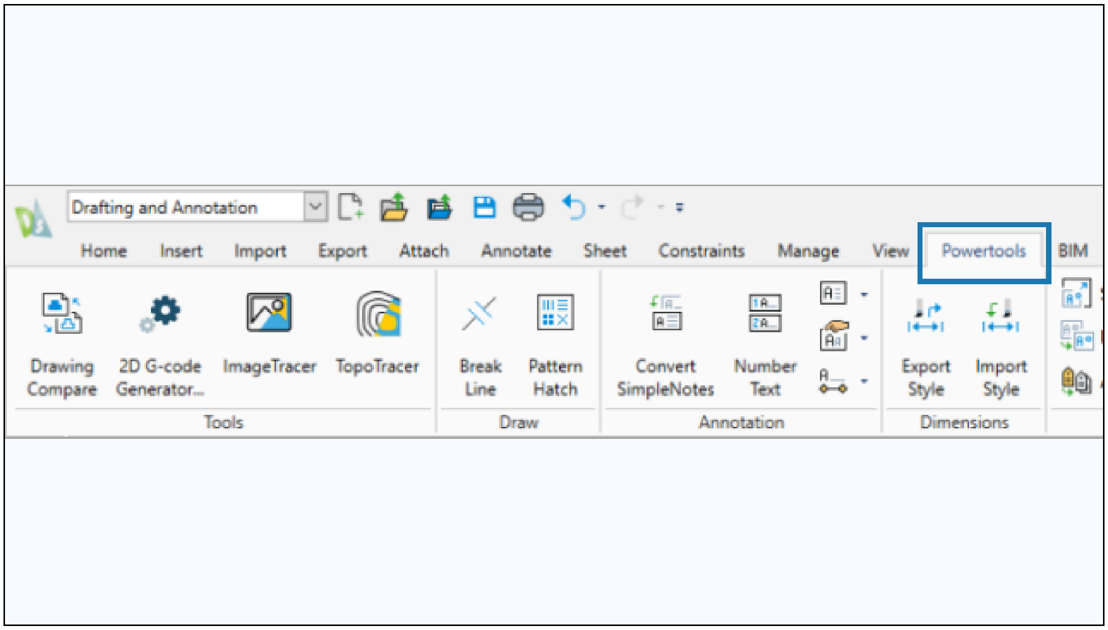


DraftSight has an updated ribbon layout and workspace to improve usability. The ribbon tabs have reduced clutter and distributed commands across additional tabs for better accessibility.

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 (For DraftSight Premium, Enterprise Plus, and Mechanical)



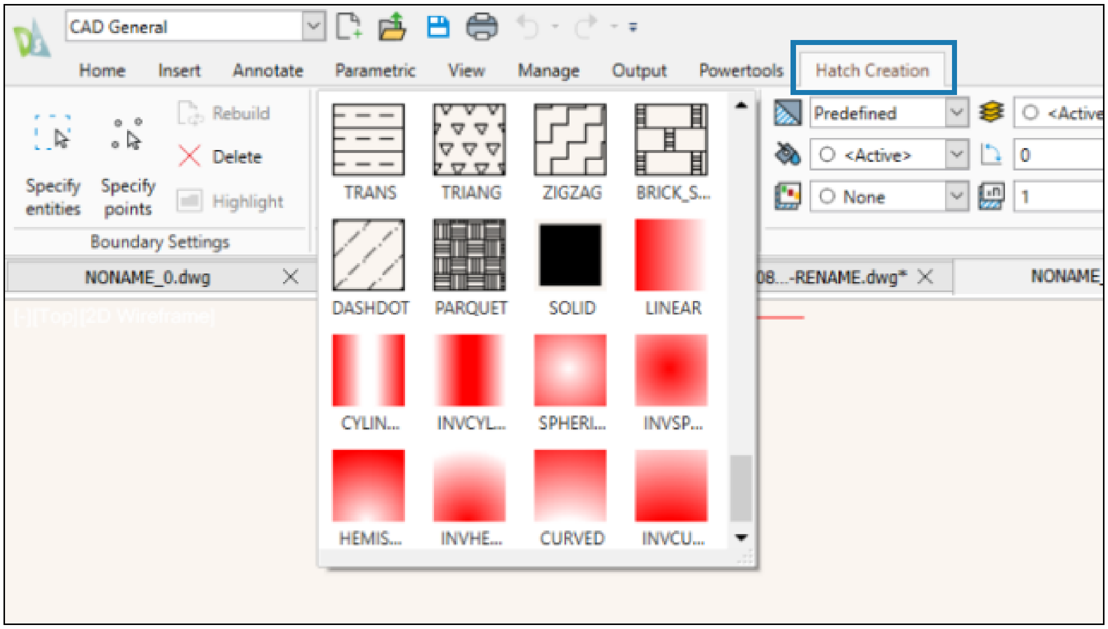
The Powertools ribbon tab has additional tools on it to streamline your workflow.

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

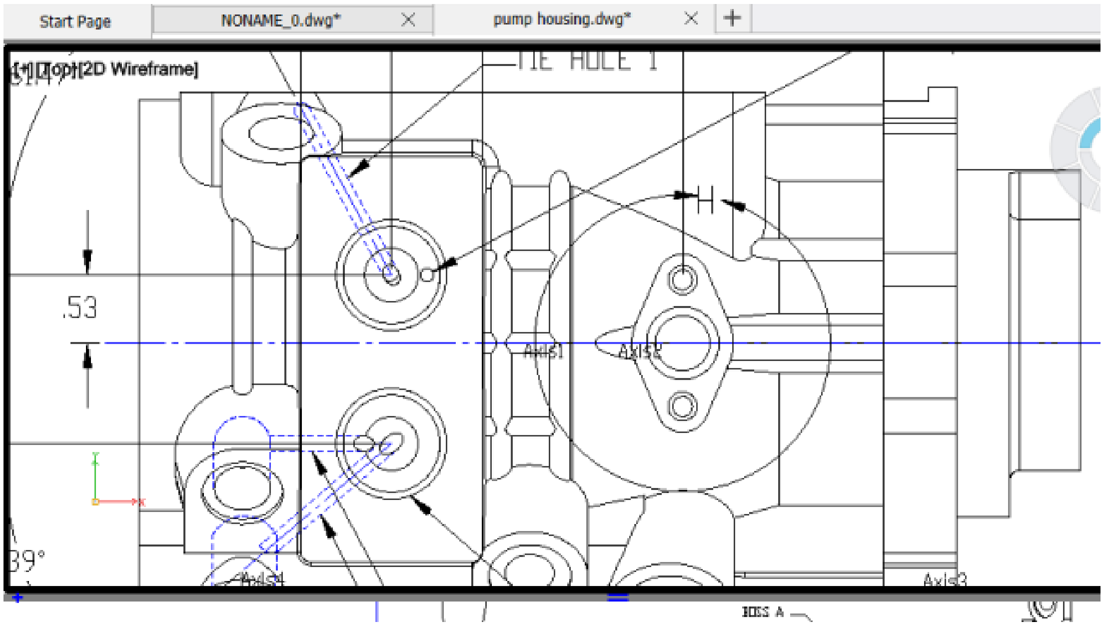


The contextual ribbon has tabs for the `HATCH` and `PATTERN` commands. This enhances productivity by reducing the time spent on finding the commands.

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.

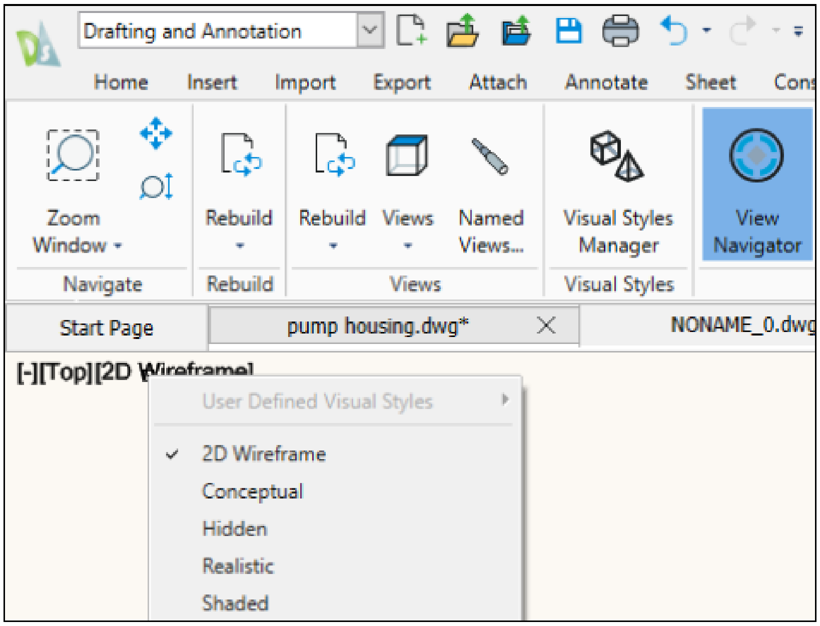
Manipulating ViewTiles (For DraftSight Premium, Enterprise Plus, and Mechanical)



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.

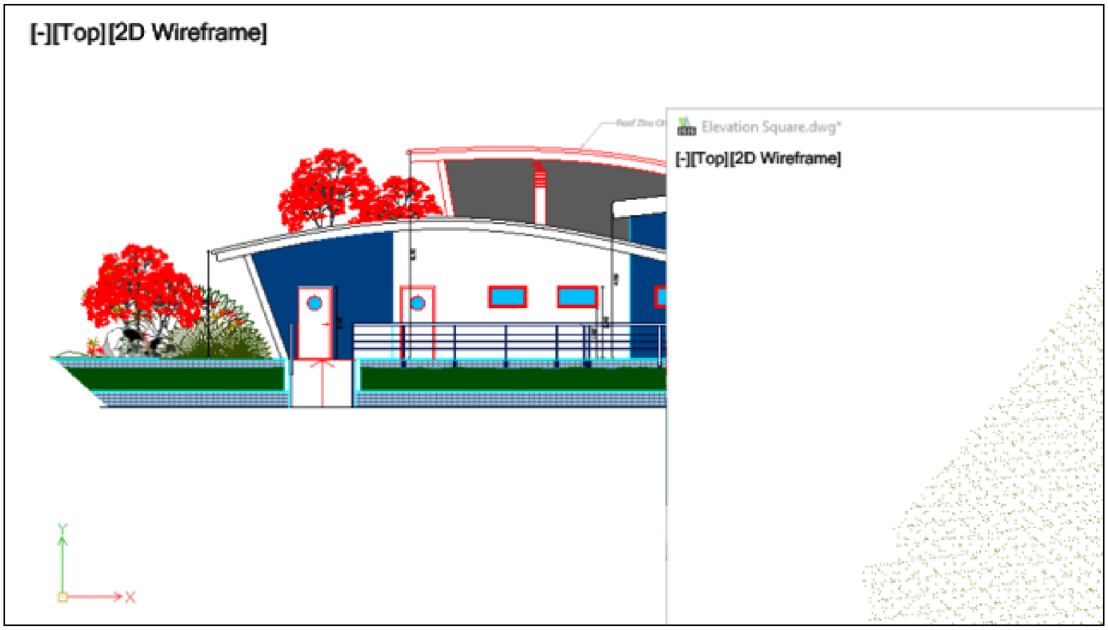
ViewTiles Controls



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 (For DraftSight Premium, Enterprise Plus, and Mechanical)



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

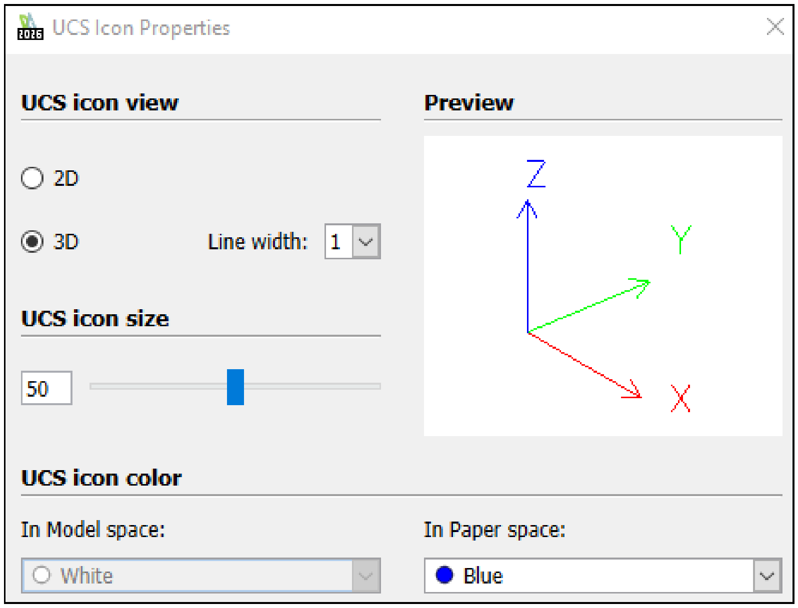
DraftSight supports the Hexagon Enhanced Compression Wavelet (`.ECW`) image format. `.ECW` files use efficient compression, making them ideal for large-scale images such as aerial and satellite photography. The files can include embedded map projection data, enhancing their usefulness in geographic information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



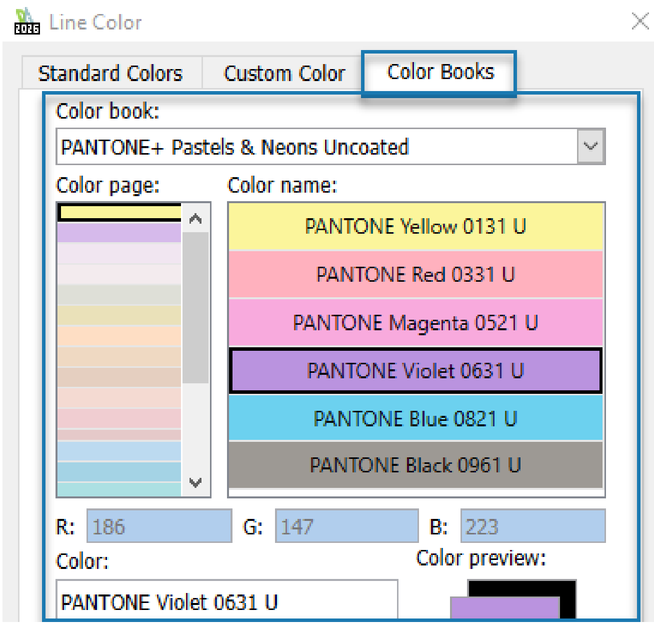
The CCS icon represents the current orientation of the coordinate system. You can adjust this visual indicator to suit your preferences or the needs of a particular project.

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 (For DraftSight Premium, Enterprise Plus, and Mechanical)



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.

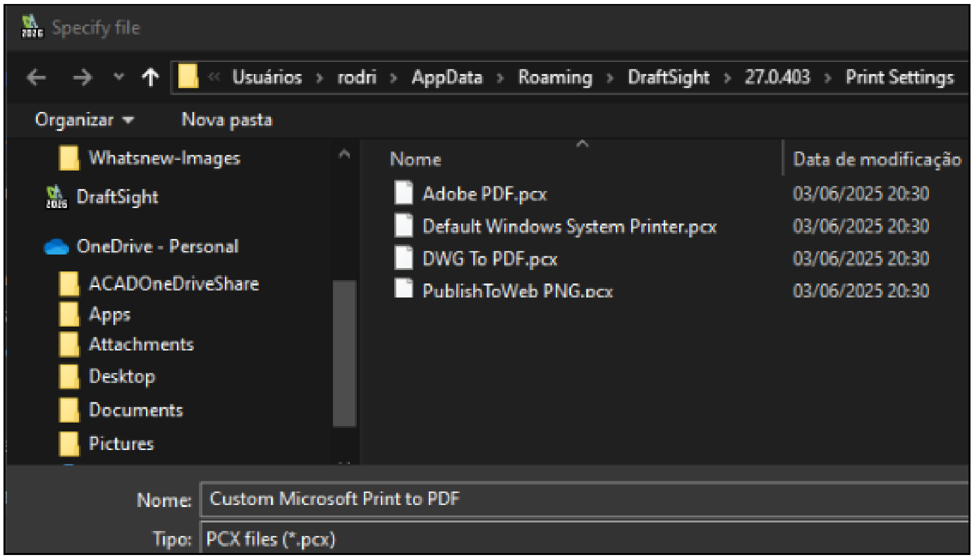
With color books, you can quickly select and apply exact shades from popular color systems such as Pantone® and RAL™, ensuring visual accuracy and consistency across designs.

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 (For DraftSight Premium, Enterprise Plus, and Mechanical)



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

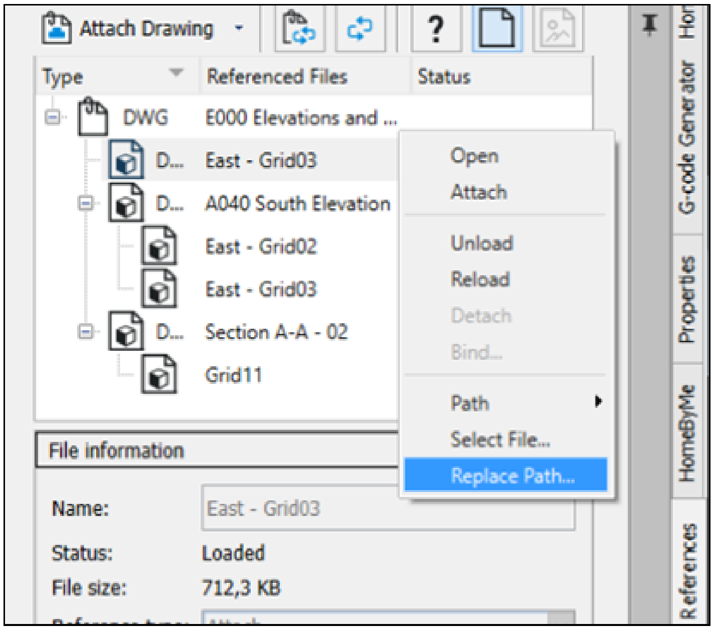
You can import `.PC3` files or create and save new print configuration files in the `.PCX` format. The `.PCX` format makes it easier to reuse and share print settings for consistent output across multiple print jobs. This improves workflow efficiency by ensuring standardized print configurations.

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



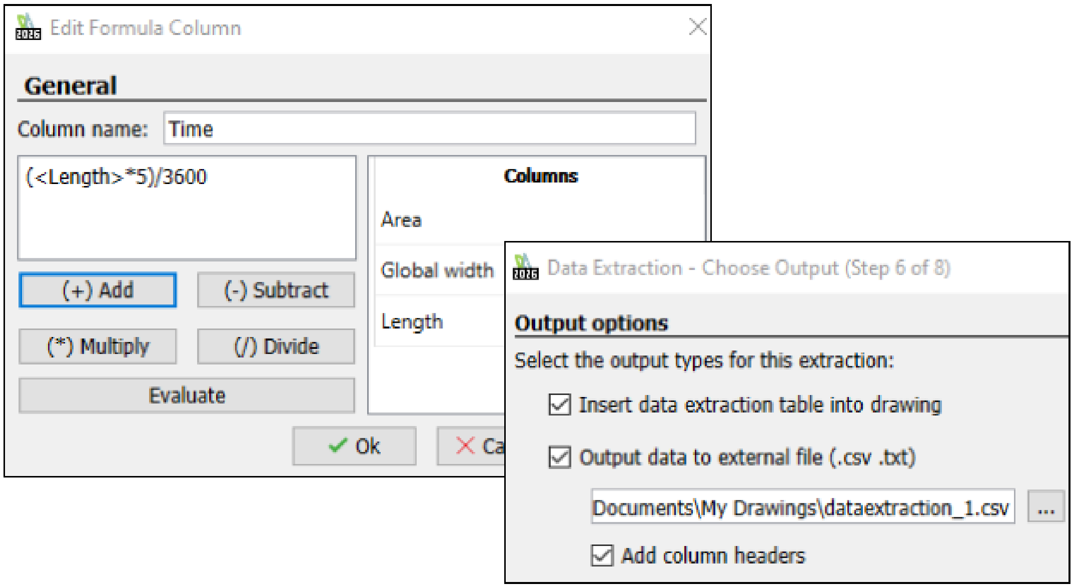
You can manage missing external references more efficiently with tools in the **References** palette. When a referenced file is moved or renamed, you can update the file path once. DraftSight offers to use the same path for other missing references if it finds matching files.

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.

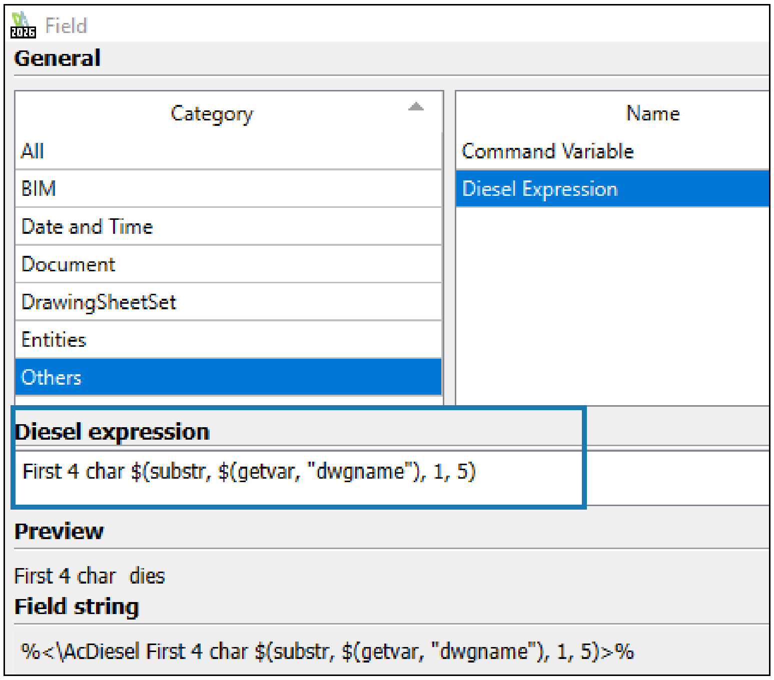
The DATAEXTRACTION wizard includes an **Insert Formula Column** option where you can define custom fields with mathematical expressions. This provides flexibility when structuring data extraction tables because you can make calculations directly in the wizard. You can create more refined and tailored 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 `FIELD` command supports Diesel Expressions, enabling dynamic text that automatically updates based on drawing properties or user inputs.

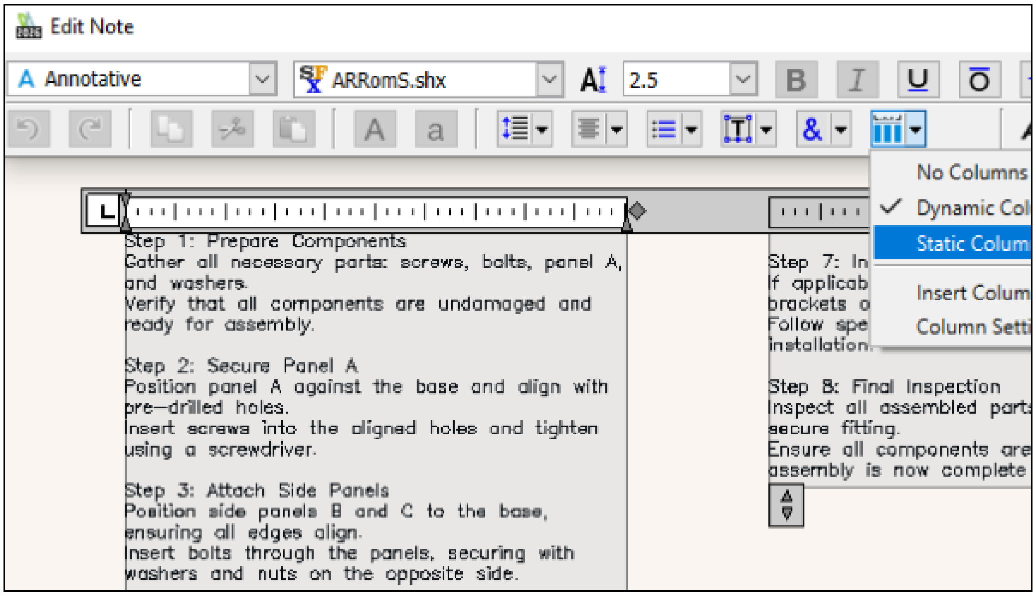
You can create custom text fields that concatenate strings, perform calculations, and apply conditional formatting without needing external scripts or complex programming. This improves automation and flexibility when working with text data in drawings.

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.

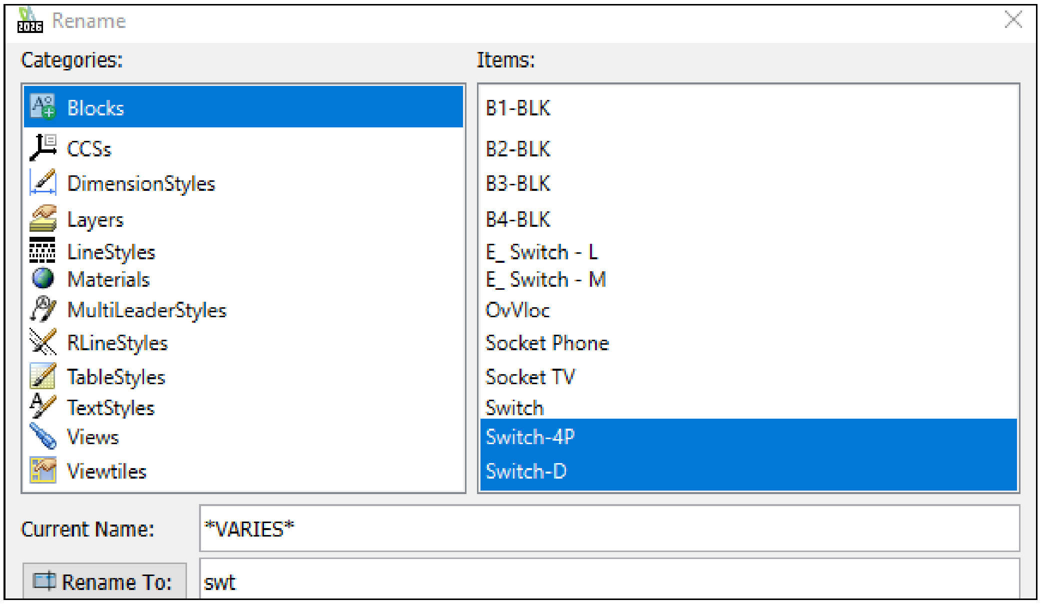
Columns are beneficial for complex documentation or design notes because they help you organize multicolumn text into clearly structured, easy-to-read layouts. They also maximize the available workspace by efficiently managing text within drawings.

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



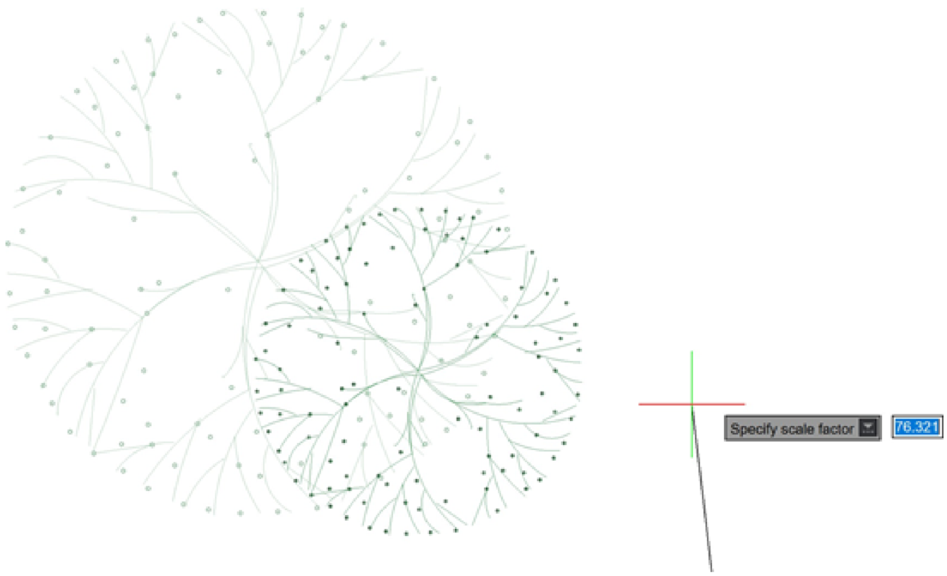
The RENAME command supports wildcard characters, making it easier to find and rename multiple named elements at once. You can filter matching items in the RENAME dialog box and rename multiple objects more efficiently.

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.

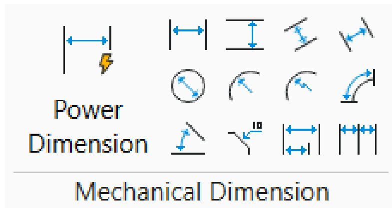
The **Copy** option provides a more streamlined and efficient process. Previously, you had to use the COPY command before the SCALE command to copy entities.

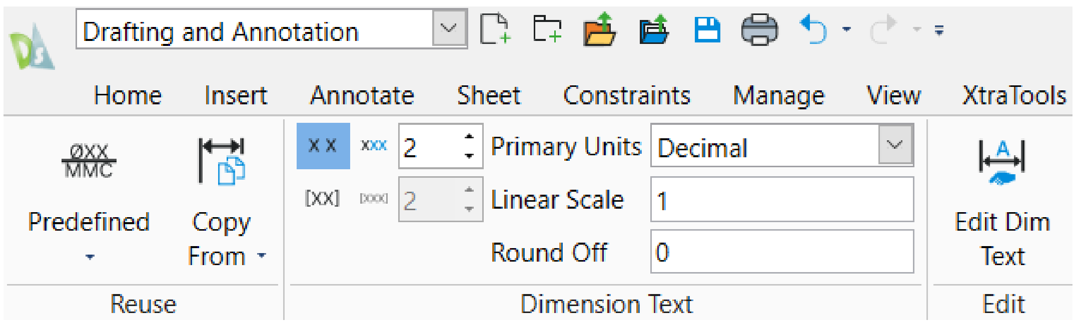
To use the SCALE command:

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.

25

SOLIDWORKS Flow Simulation

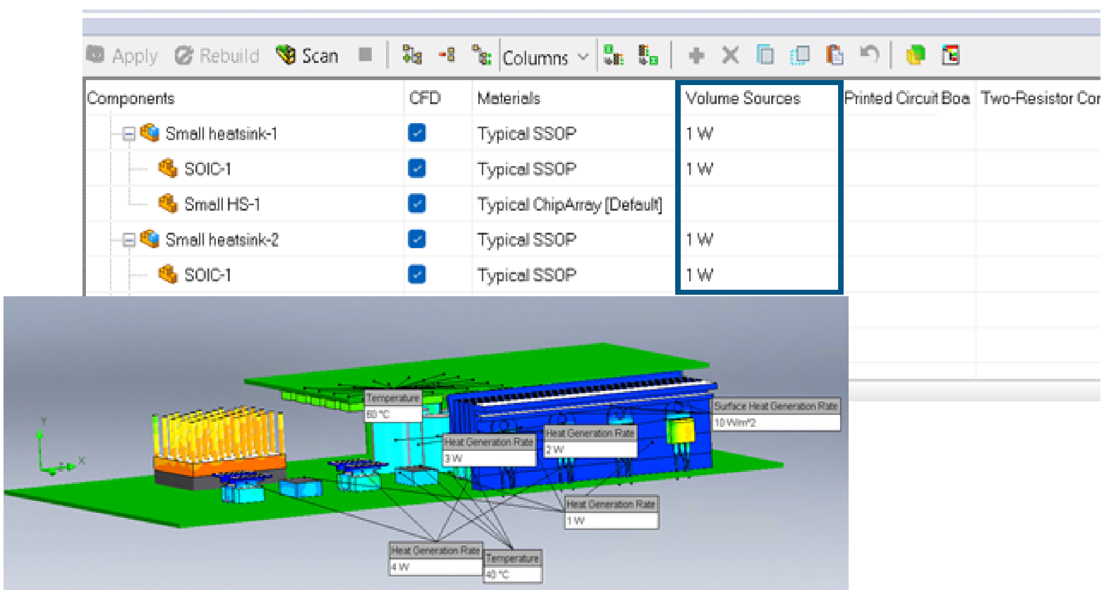
This chapter includes the following topics:

- **Component Explorer**
- **Fill Thin Slots**
- **Minimum and Maximum Goal Locations**
- **Bubble Charts for Parametric Studies**
- **Project Parameters from Components**

SOLIDWORKS® Flow Simulation is a separately purchased product.

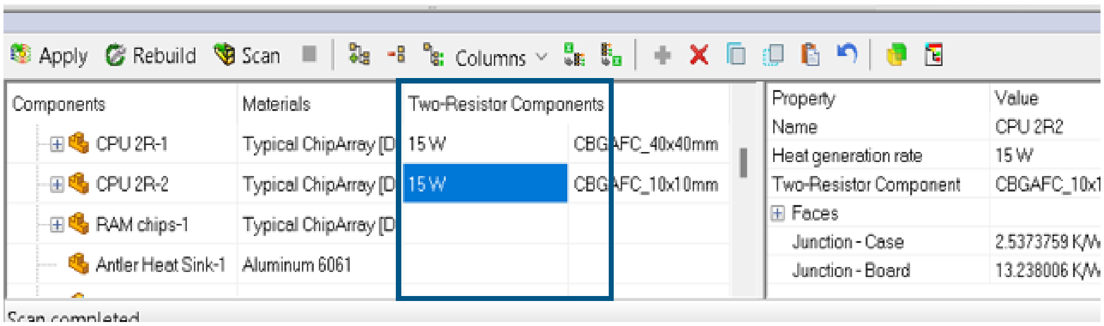
Component Explorer

Total Power of Sources



The Component Explorer includes a column for **Volume Sources** or **Surface Sources**. This helps you identify all sources and their power budgets.

Creating Two-Resistor Components



You can create two-resistor components in the Component Explorer. This reduces time and potential mistakes.

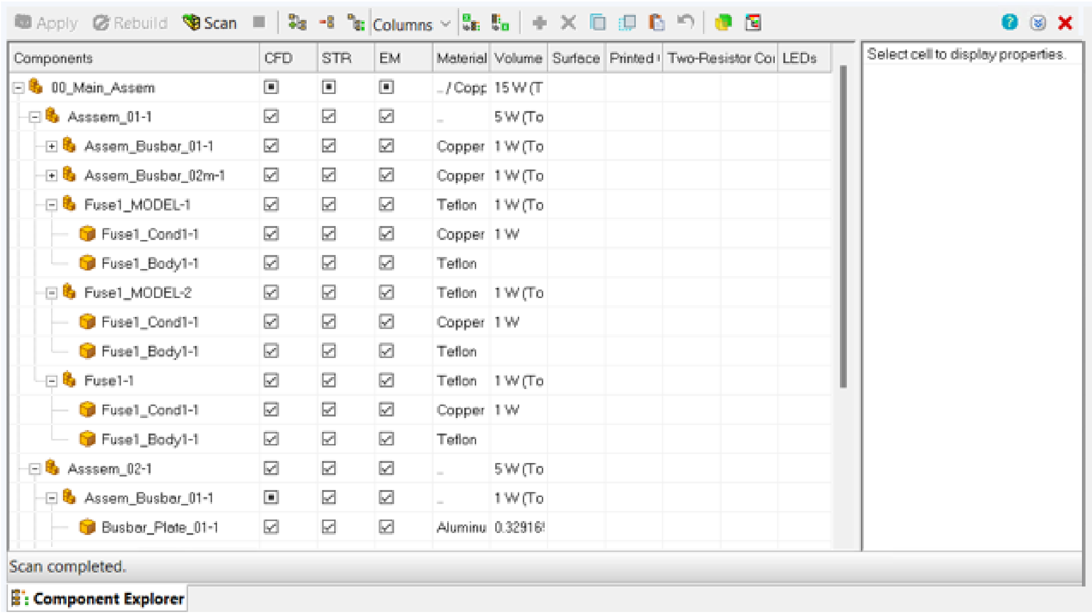
To create two-resistor components:

Do one of the following:

- In the Component Explorer, under **Two-Resistor Components**, specify the power or EDB item and click **Apply**.
- In Microsoft® Excel®, fill in a component list spreadsheet with two-resistor parameters, then import it into the Component Explorer.

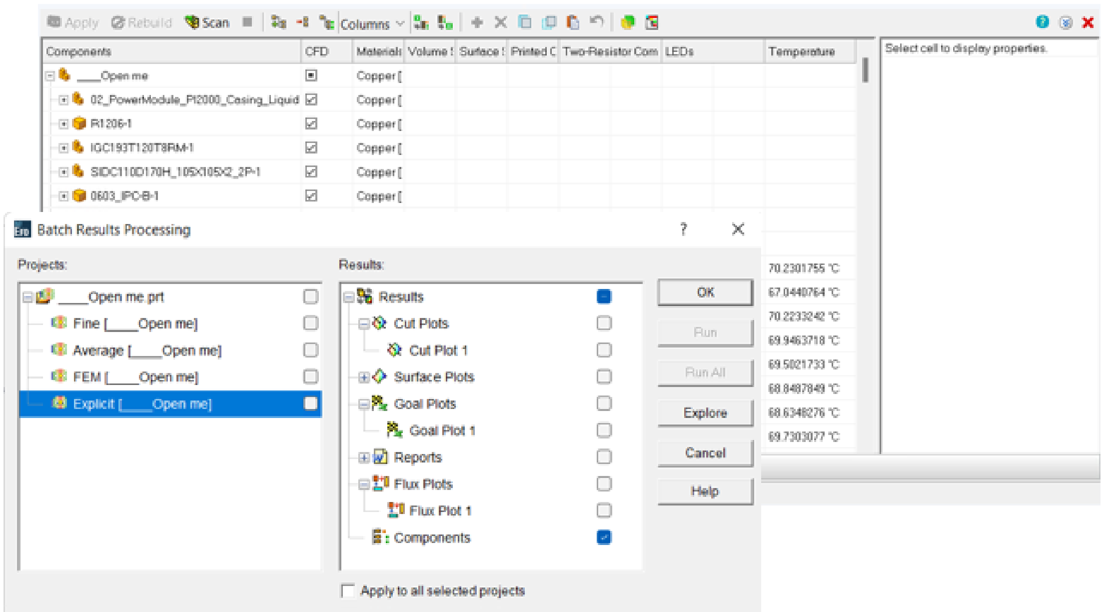
Flow Simulation finds the top and bottom faces automatically. A planar face in contact with another body (presumably PCB) is the bottom face and the opposite planar face is the top face. If Flow Simulation cannot find the faces, it creates an invalid feature. You can edit the feature in the tree and select faces manually.

Component Status



In the Component Explorer, you can manage component features and statuses. You can also remove volume features applied to components.

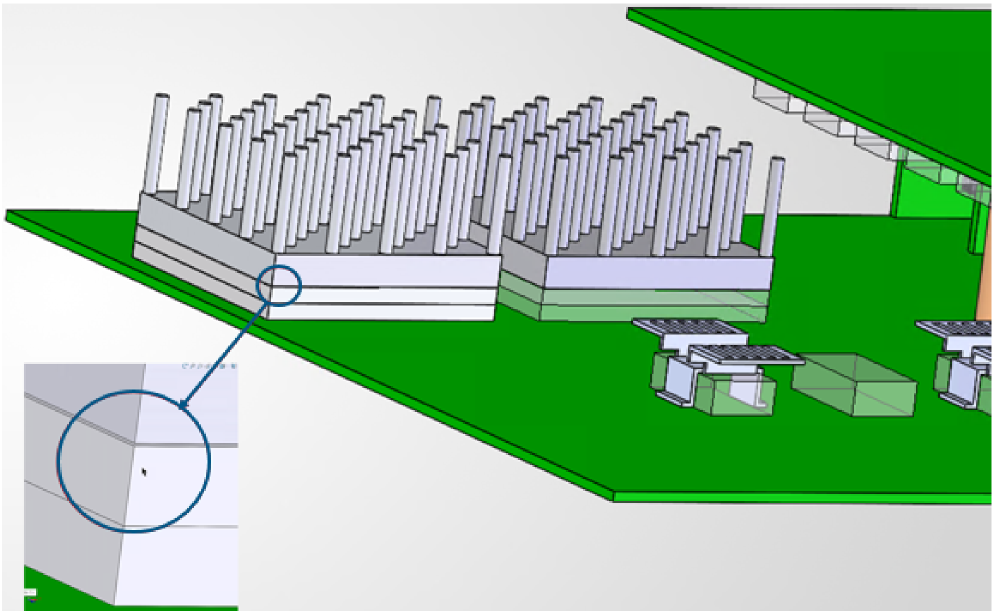
Temperature Column



The Component Explorer includes a **Temperature** column where you can export the values to Microsoft Excel.

This column provides temperatures of all solids without slowing down the solver. You can export a matching table of component names, input data, and resulting temperatures automatically through **Batch Results Processing**.

Fill Thin Slots



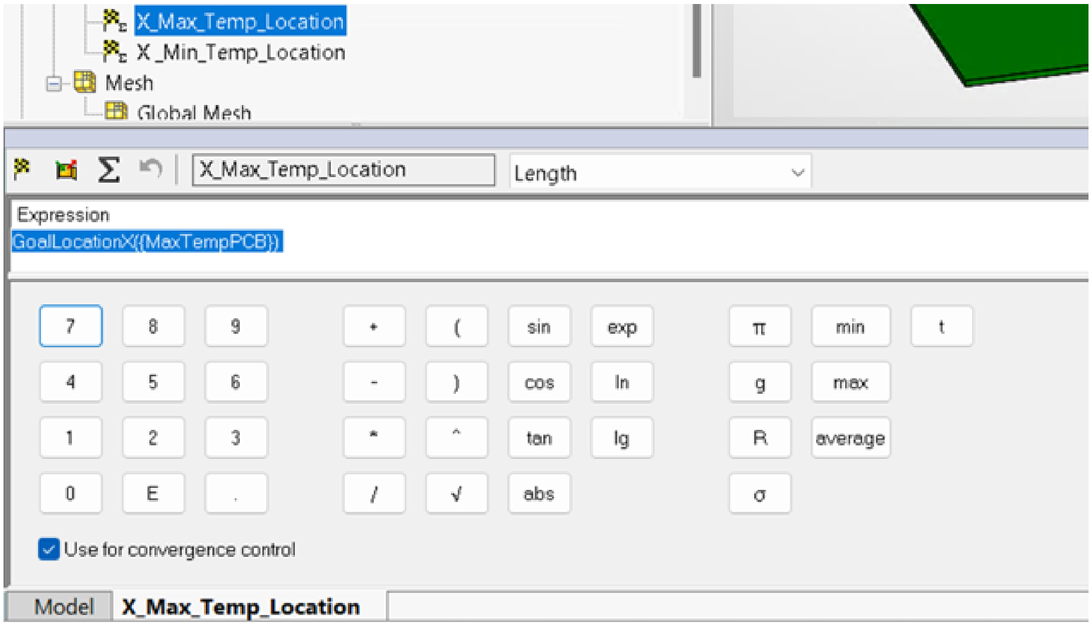
You can fill thin gaps in models for improved thermal accuracy.

Models can have thin gaps between heated components and heatsinks or gaps between glued parts. These gaps are meant to be filled with thermal interface materials (TIMs) or glue during assembly. However, simulation models with unfilled gaps give incorrect thermal results.

With the **Fill Thin Slot** tool, you can fill mesh cells inside gaps with specific solid materials corresponding to the thickness criteria.

Flow Simulation applies the material to mesh cells without creating a CAD body. You can see the resulting shape of filler material with postprocessing tools and the **Use CAD geometry** option cleared.

Minimum and Maximum Goal Locations



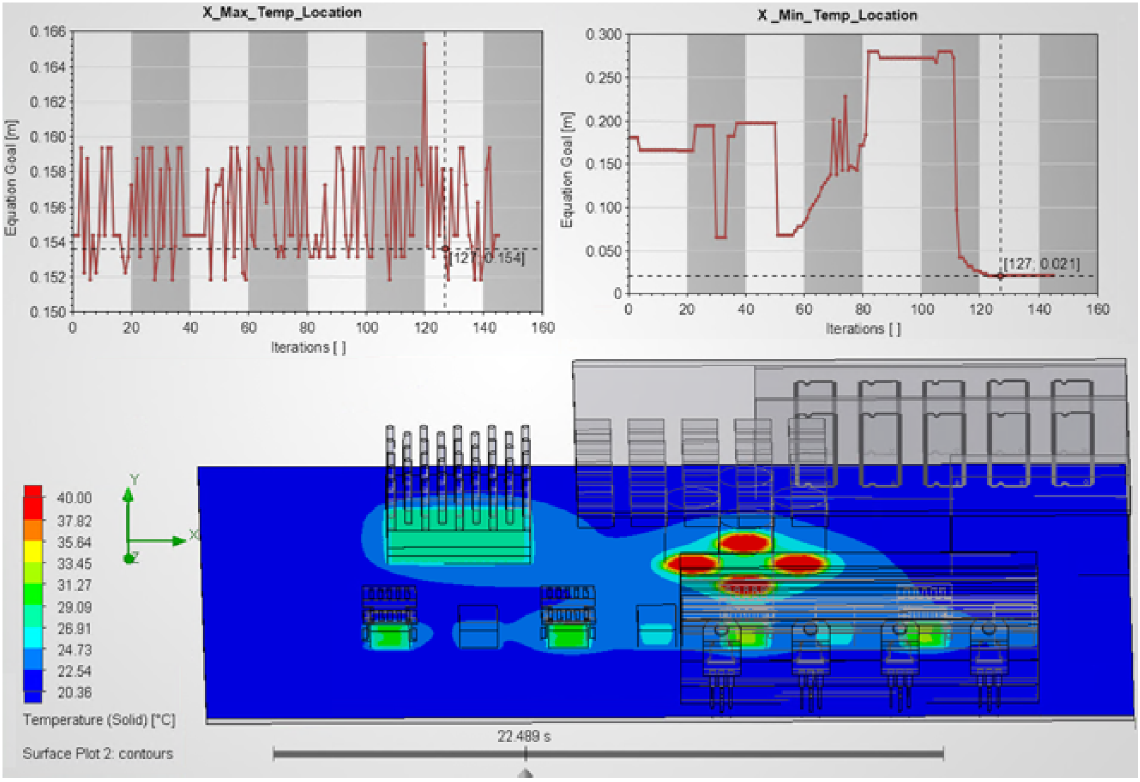
You can specify the minimum or maximum locations as an objective function.

When running simulations, you may need the coordinates of a point when a minimum or maximum parameter value is reached for simulation results or as an objective function.

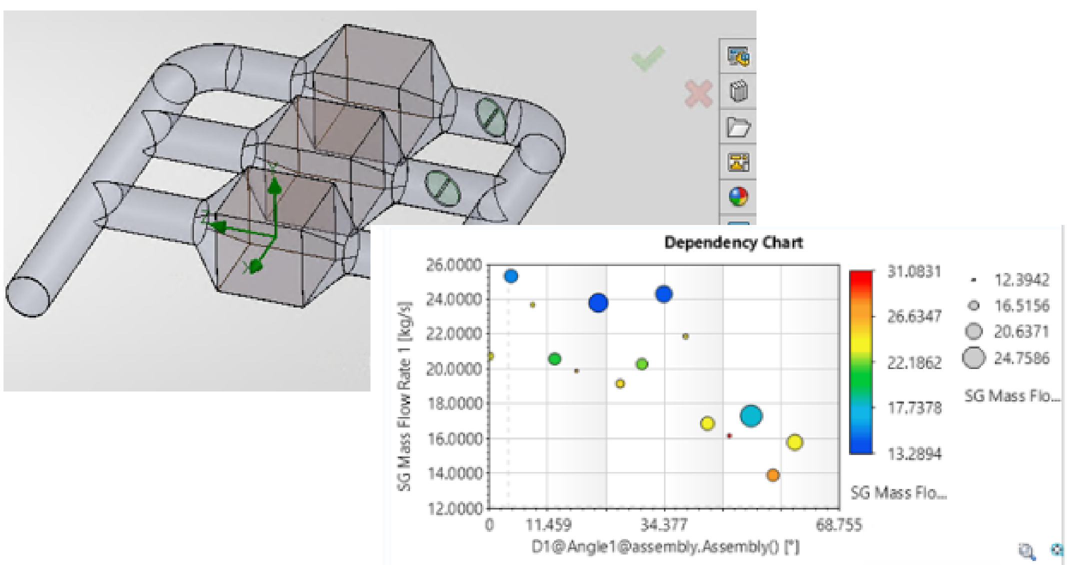
Functions to locate the point of volume or surface goal are in the equation goal expression, where *Goal Name* is the name of the minimum or maximum surface or volume goal:

- **GoalLocationX**({*Goal Name*})
- **GoalLocationY**({*Goal Name*})
- **GoalLocationZ**({*Goal Name*})

Example of minimum and maximum temperatures on a simulation:

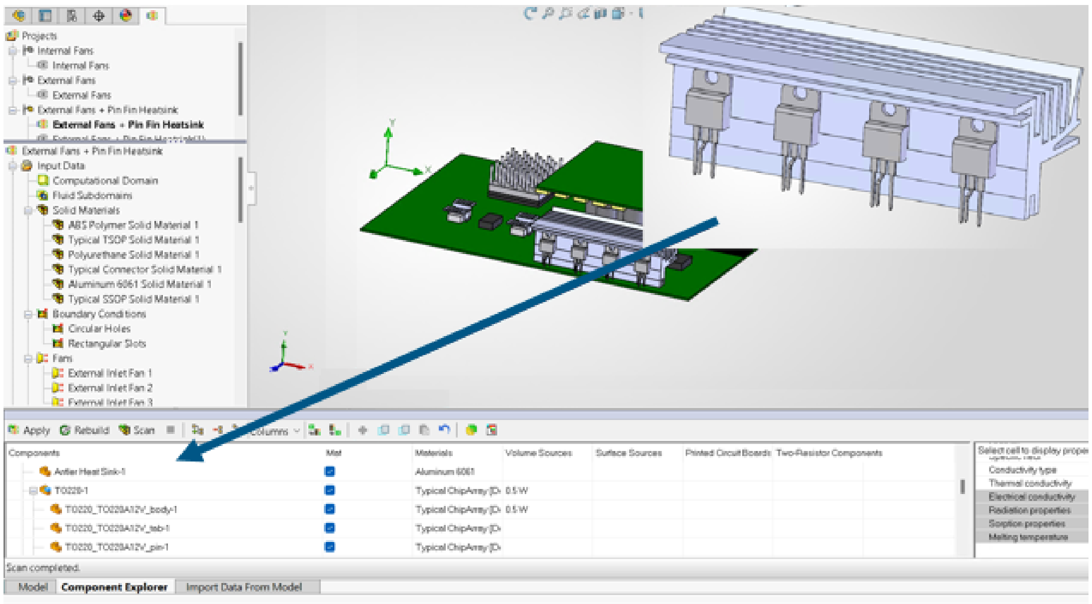


Bubble Charts for Parametric Studies



You can display up to three parameters on a bubble chart to compare design points of resulting parametric studies for multiparameter optimization.

Project Parameters from Components



You can manage embedded components more efficiently.

You can:

- Control multiple submodels embedded in the current analysis project individually
- Define project parameters in submodels and propagate them upward
- Control sophisticated dependencies defined in embedded components
- Create elaborate library components and reuse submodels
- Adjust parameters of individual components within a group of identical embedded components

26

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.

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

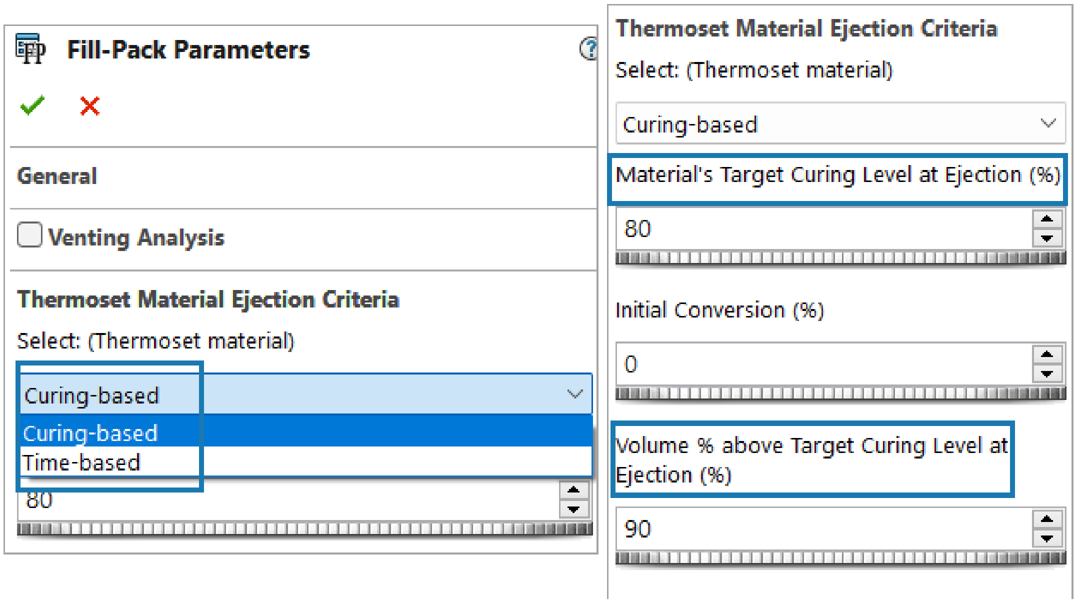
Manufacturer	Number of Removed Material Grades
Rohm GmbH and Company KG	27
Rohm and Haas	19
ICI	3
Mitsubishi Chemical®	1

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 • **Conversion** • **Time**	**Thermoset Material Ejection Criteria** • **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.

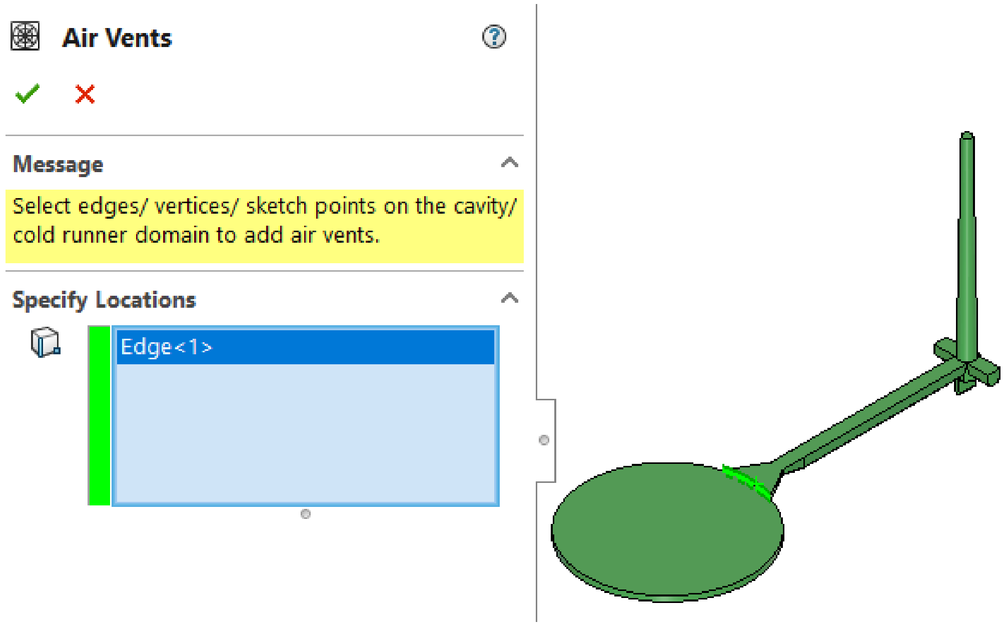
Results Related to Thermosets - SOLIDWORKS Plastics 2025	Results Related toThermosets - SOLIDWORKS Plastics 2026
1. **Curing Time at End of Fill** 2. **Material Reactive Conversion at End of Fill** 3. **Curing Time at Post-Filling End** 4. **Material Reactive Conversion at Post-Filling End**	1. **Time 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:

Routing is available in SOLIDWORKS® Design Premium and SOLIDWORKS Design Ultimate.

Adding Coverings over Inline Fittings (2026 SP1/FD01)

You can apply coverings over inline fittings such as tees, crosses, and reducers using the Covering PropertyManager.

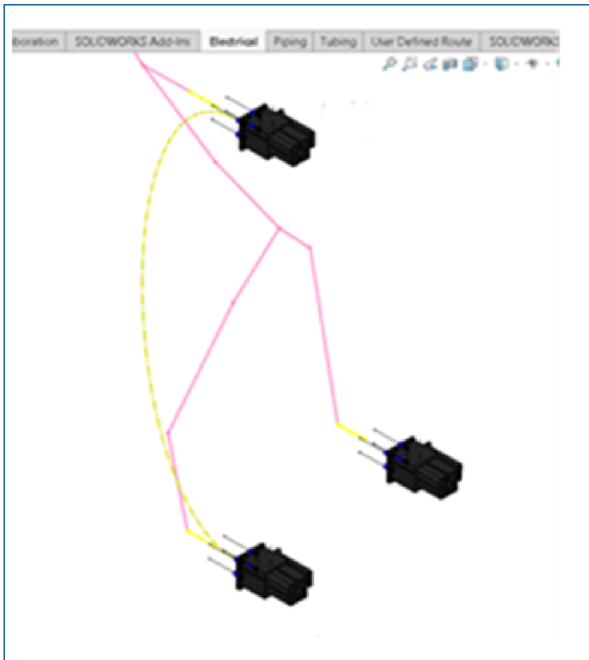
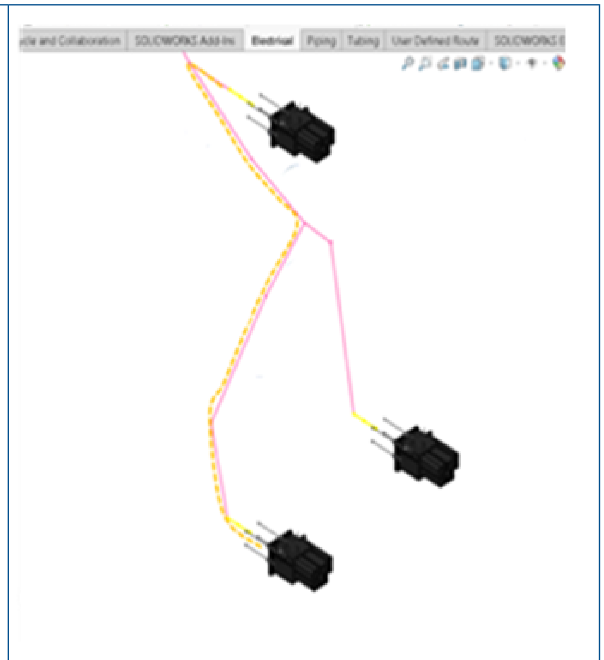
Benefits: You can represent insulation or protective layers more accurately and completely throughout the route.

To add coverings over inline fittings:

1. In the Covering PropertyManager, click **Segments and Fittings**.
2. Select the fittings that you want to cover.

The coverings appear across all views, including shaded, wireframe, and drawing modes. When you modify fittings or adjust routes, the coverings update automatically to reflect your changes.

Redirecting Guidelines to Follow a Route Path

	
Automatically generated guidelines following the shortest possible path between end connectors	Redirected guidelines following a route path

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.

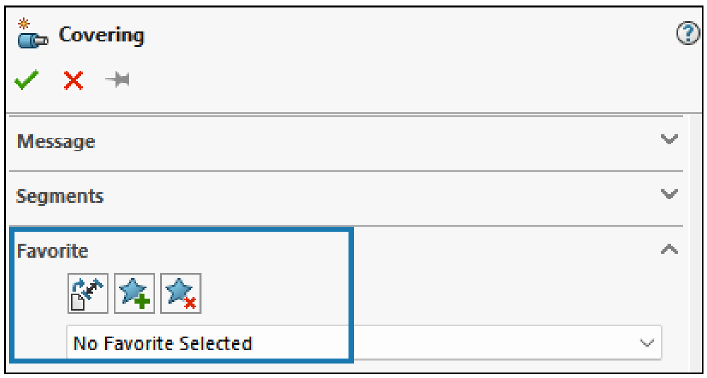
   A preview of the guidelines, aligned with the selected sketch, displays in the graphics area.

6. Click **Done**.

   The sketches representing the route path appear in the FeatureManager® design tree as **Routing_path***n*.

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.

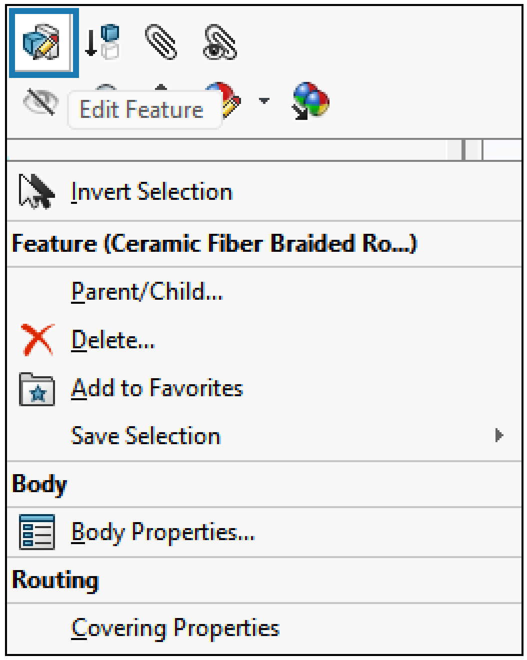
Benefits: This section enhances user efficiency, improves time management by providing quick access to commonly used covering style.

To mange 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. • 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.

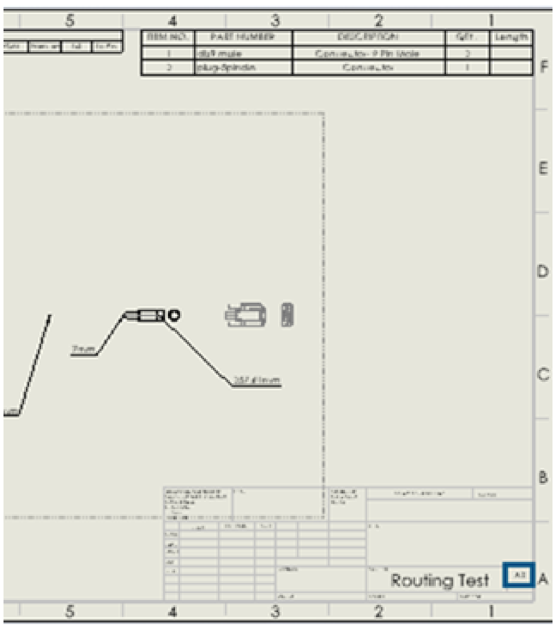
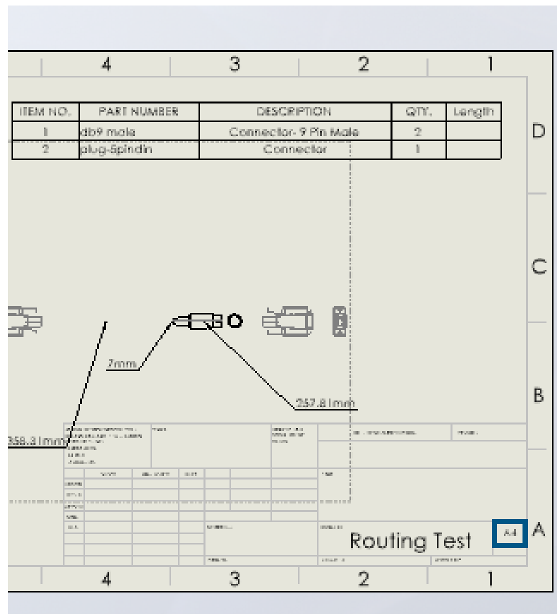
Benefits: These enhancements simplify connector table management. With more informative table names and batch selection tools, you spend less time sorting through connectors.

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 mutli-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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