



WHAT'S NEW

SOLIDWORKS Products 2027 PRERELEASE



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What's New for SOLIDWORKS Products 2027

This chapter includes the following topics:

- **For More Information**
- **AI in the 3DEXPERIENCE Platform**



SOLIDWORKS® products for the 2027 release year contain user-driven enhancements that help streamline and accelerate your product development processes from concept to manufacturing.

This document covers all enhancements for SOLIDWORKS products 2027.

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 **3DEXPERIENCE**® platform (login required).

Release Notes

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

Legal Notices

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

AI in the 3DEXPERIENCE Platform

The **3DEXPERIENCE** platform provides a wide variety of tools to boost your productivity, some of which leverage artificial intelligence (AI) systems to enhance your experience.

What is AI?

AI refers to systems that mimic human thought processes, including learning, refining, and resolving problems through computer models. These systems can recognize patterns, predict data, and extract conclusions from large volumes of data. Generative AI creates new content, such as text, media, and other collateral, based on input data.

How is AI Used at Dassault Systèmes?

At Dassault Systèmes, AI tools are built into many **3DEXPERIENCE** apps. Depending on your particular needs and workflows, these tools leverage different technical approaches to enhance your productivity; including machine learning, neural networks, large language models, and natural language processing.

What are the Benefits of AI?

The **3DEXPERIENCE** AI tools provide you with support to leverage your data, communicate clearly, improve accessibility for other people, enhance your efficiency, and use data to enable reliable decision-making. You can apply such tools in a wide range of situations, from discovering new medicines to designing shoes or cities.

What Do You Need to Consider Before Using AI?

You must ensure that the use of **3DEXPERIENCE** AI tools conforms with the data policies of your organization.

When using **3DEXPERIENCE** AI tools (either on a public or private cloud **3DEXPERIENCE** platform), computations happen in environments that run with shared computing resources using only the data strictly necessary for that purpose.

When you use **3DEXPERIENCE** AI tools, always review and validate the output data to ensure reliability and accuracy.

Dassault Systèmes documents the purpose of the various **3DEXPERIENCE** AI tools of its solutions in the Documentation and more specifically in the user assistance, and you must use those AI tools in accordance with their intended purpose as outlined herein.

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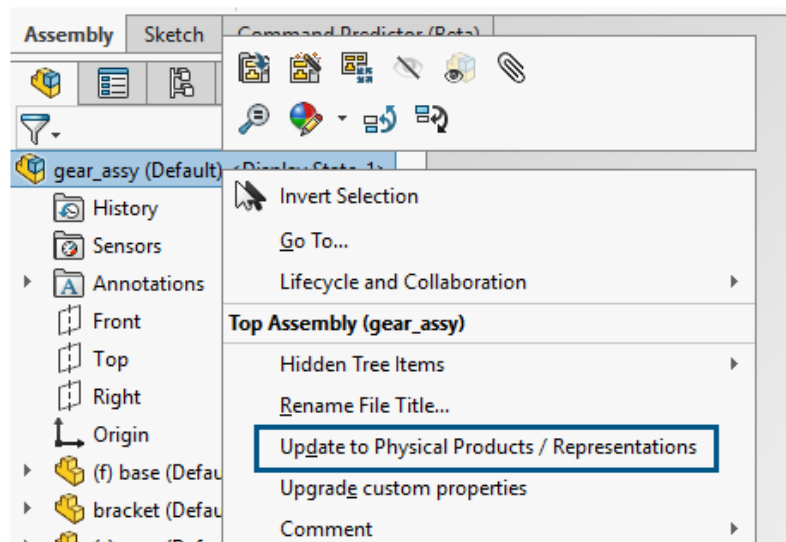
Using SOLIDWORKS Design on the 3DEXPERIENCE Platform

This chapter includes the following topics:

- **3DEXPERIENCE Physical Products and Representations**
- **Refreshing MySession Data in the Background**

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

3DEXPERIENCE Physical Products and Representations

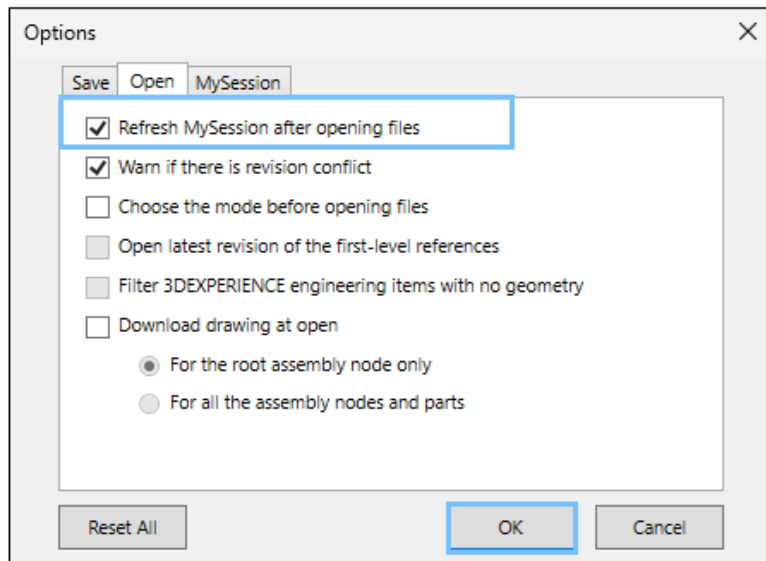


SOLIDWORKS Design for **3DEXPERIENCE** users is redesigned to better match how the **3DEXPERIENCE** platform handles physical products and representations.

The user interface is updated to replace the term **3DEXPERIENCE Compatibility** with the platform terms **Physical Products** and **Representations**. The **System Options** >

3DEXPERIENCE Integration page is renamed to **Data Management** and is redesigned to better communicate the benefits of physical products and representations.

Refreshing MySession Data in the Background



When you enable **Refresh MySession after opening files** in **Tools > Options > Open**, MySession data refreshes in the background while you work.

Benefits: This option keeps SOLIDWORKS Design responsive during refresh operations. You can continue saving, loading, updating, and reloading platform data while MySession refreshes in a separate process.

For example, if a MySession refresh is running after you open files from the **3DEXPERIENCE** platform, you can continue using PLM commands such as **Reload from Server** without waiting for the refresh to complete.

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Installation

SOLIDWORKS Treehouse Available with All SOLIDWORKS Design License Types

SOLIDWORKS Treehouse is available with the single-user license version of SOLIDWORKS Design as well as device-licensed versions.

To run Treehouse, click **Start > SOLIDWORKS Tools *version* > SOLIDWORKS Treehouse *version*.**

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SOLIDWORKS Fundamentals

This chapter includes the following topics:

- **Opening Referenced Display States of Models**
- **Changes to System Options and Document Properties**
- **User-Defined Variability**
- **Variability Tables**

Opening Referenced Display States of Models

When you open a model from an assembly or drawing view, the model opens in the same display state shown.

Changes to System Options and Document Properties

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

System Options

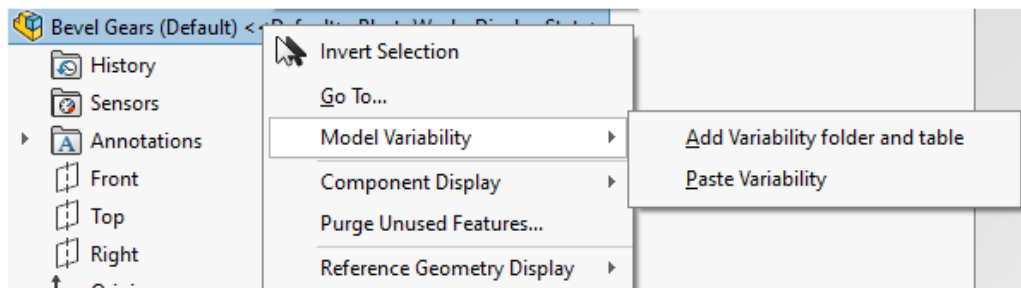
Option	Description	Access
What's New	Opens a help topic that shows changes to options in the current version.	All pages in the dialog box.
Disable sheet format edits	Locks drawing sheet formats to prevent edits to borders and title blocks.	Drawings
FeatureManager panel width	Control the width of the FeatureManager design tree across all models or individually per model.	FeatureManager

Option	Description	Access
Import Identical Bodies as Instances	For components of imported Parasolid® assemblies. Imports duplicated identical bodies as instances of the same component.	Import > File Format > Parasolid

Document Properties

Option	Description	Access
What's New	Opens a help topic that shows changes to options in the current version.	All pages in the dialog box.

User-Defined Variability



You can create user-defined variabilities. This provides more ease and flexibility to work with different variants.

To know more about variants and variability options, see the *Introduction* section of the **About Variants and Variability** topic in Dassault Systèmes User Assistance. Access to Dassault Systèmes User Assistance requires **3DEXPERIENCE** credentials.

You can create user-defined variabilities in two ways:

- Copy and paste a variability that has been defined on the platform. You can copy **Variability**, **Variant**, **Options**, or **Numerical Parameters** folder. You can select multiple folders for copy. To copy, select the folder and press **CTRL+C** or right-click

and select **Copy**. Paste the copied object in another assembly in the FeatureManager design tree.

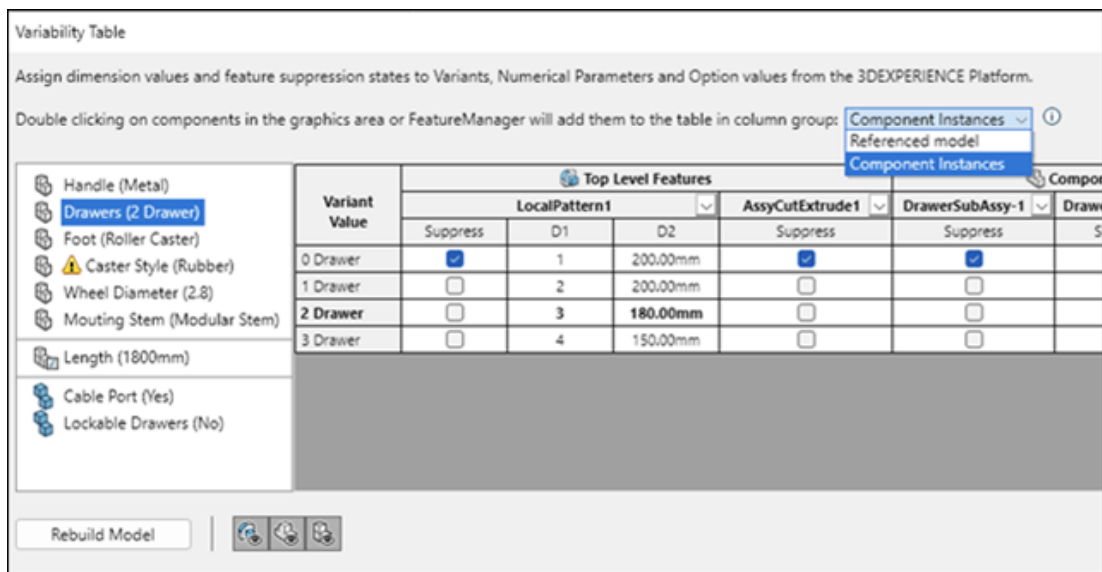
Copied variability objects are not linked to the original source.

- In the FeatureManager design tree, right-click an assembly and select **Model Variability > Add Variability folder and table**. This creates a **Variability** folder under which you can define variability options.

You can modify a user-defined variability in the following ways. You can:

- Add values to a variant.
- Add new options to an option group.
- Rename variants, numerical parameters, option groups, and options.
- Modify the edit definition of a numerical parameter.
- Delete a variability.

Variability Tables



You can define **Variability Tables** in parts and assemblies for a SOLIDWORKS model to create flexible designs.

A **Variability Table** provides a single view of **Variants**, **Numerical Parameters**, and **Options** in one place. You can link SOLIDWORKS model features directly to the Variability Table, and automatically update the design. For more information about variability options, see [User-Defined Variability](#) on page 13.

Benefits: You can modify a complex model in a single, interactive grid. You can modify any feature, dimension, or component with one click which reduces effort and time.

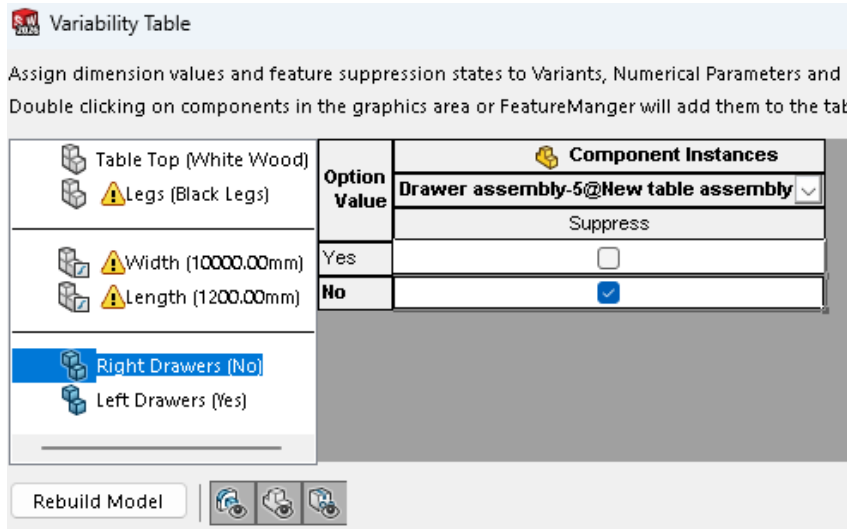
To update a Variability Table:

1. Once you have variability features assigned in your model, in the FeatureManager design tree, right-click **Variability Table**.
2. Select **Edit Variability Table**.

3. Select the **Variants**, **Numerical Parameters**, or **Options** that you want to link to the model.

For **Variants**, **Numerical Parameters**, and **Options**, you can modify dimensions (length, width, height, diameter), suppress, unsuppress, or add any conditions ($>$, $=$, or $<$) in the Variability Table.

Example: You can suppress the drawers of a model.



To suppress Right Drawers:

1. Double-click on the drawer assembly to add it to the Variability Table.
2. Select **No** for **Suppress**.
3. Click **Rebuild Model** and select **OK**.

The model updates automatically based on the selected parameters and variant settings in the **Variability Table**.

You can also modify the variability feature values of a component one level below directly from the **Variability Table**. This lets you control subcomponent behavior without opening individual part files.

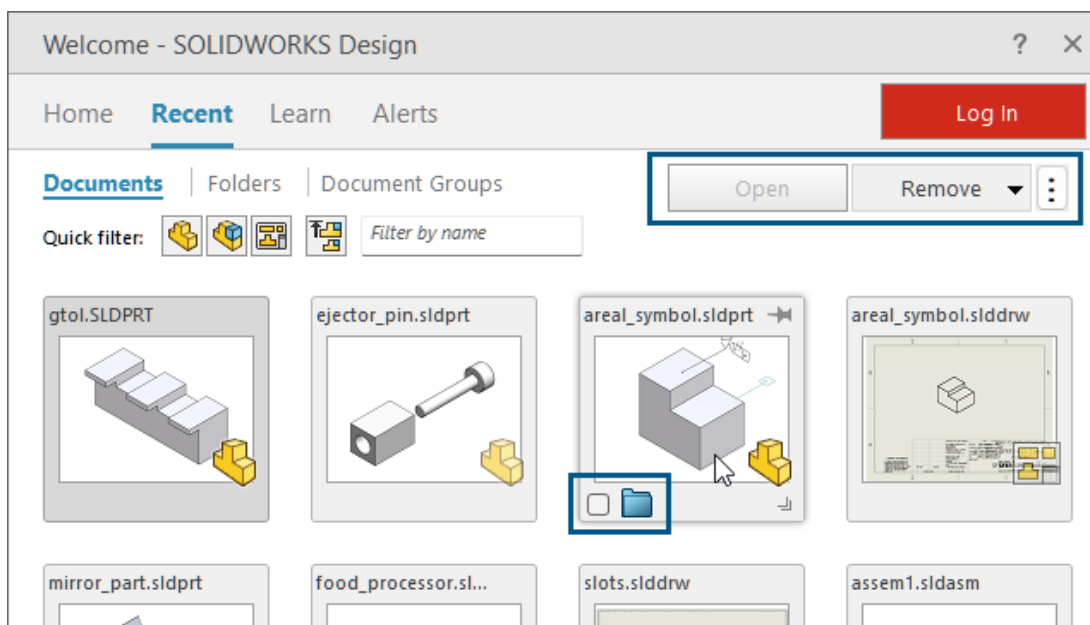
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User Interface

This chapter includes the following topics:

- **Welcome Dialog Box - Recent Documents Tab**
- **What's New in the Options Dialog Box**
- **Controlling the Width of the FeatureManager Design Tree**
- **Controlling the Open Recent Menu**
- **Component Status Visibility**
- **Resizing Dialog Boxes**
- **Tab Navigation**

Welcome Dialog Box - Recent Documents Tab



In the Welcome dialog box, the Recent Documents tab is redesigned for ease of use. The dialog box includes these changes:

Item	Description
Thumbnails	No check boxes are visible.

Item	Description
Open	Replaces Browse .
Remove	Includes options to remove: • Selected • All Unpinned Items • All Items
 Show in Folder	(Appears when you hover over documents.)
	(Appears when you hover over documents.) Selects the document without opening it. You can select multiple documents.

What's New in the Options Dialog Box

In the Options dialog box, you can click **What's New** so you can see what has changed in **System Options** and **Document Properties**.

A help topic shows changes to options in the current version.

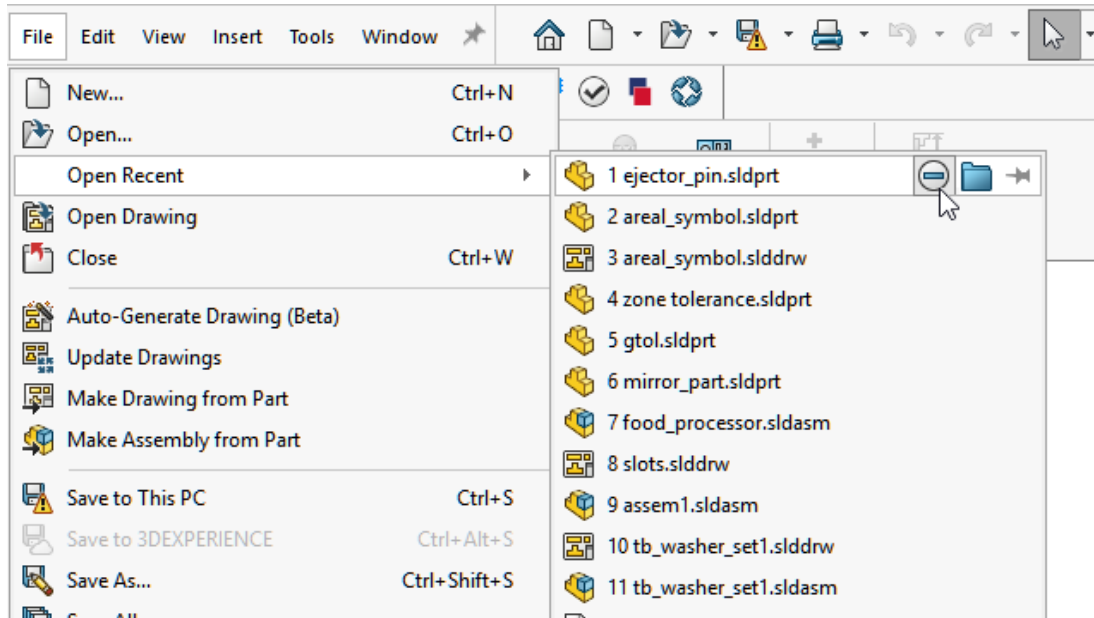
Controlling the Width of the FeatureManager Design Tree

You can specify an option to control the width of the FeatureManager design tree across all models or individually per model.

Click **Tools > Options > System Options > FeatureManager** and specify an option for **FeatureManager panel width**:

- **Apply width at document level.** Saves and restores the tree width on a per-model basis.
- **Apply width at the system level.** Applies the same width to every model. The app uses the current width at the time of the last resize.

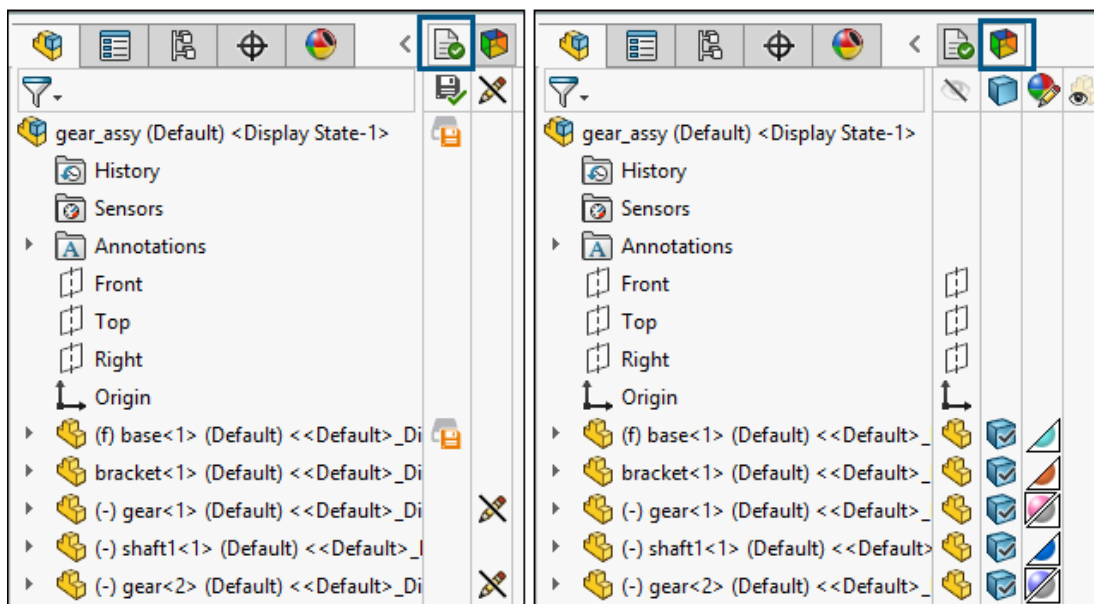
Controlling the Open Recent Menu



You can remove documents from the **Open Recent** menu to reduce clutter and find documents more easily.

1. Click **File** > **Open Recent**.
2. Hover over a document and click **Remove**

Component Status Visibility



The new File Status and Display Pane tabs in the FeatureManager design tree let you view the save status, read-only status, and visual display settings.

Benefits: You have instant insight into component changes and visibility states in large assemblies. You can manage assembly health and visual configurations faster, reducing time spent locating modified parts, and simplifying troubleshooting.

To view these changes, in an assembly file, open the FeatureManager design tree and click > to expand the tabs.

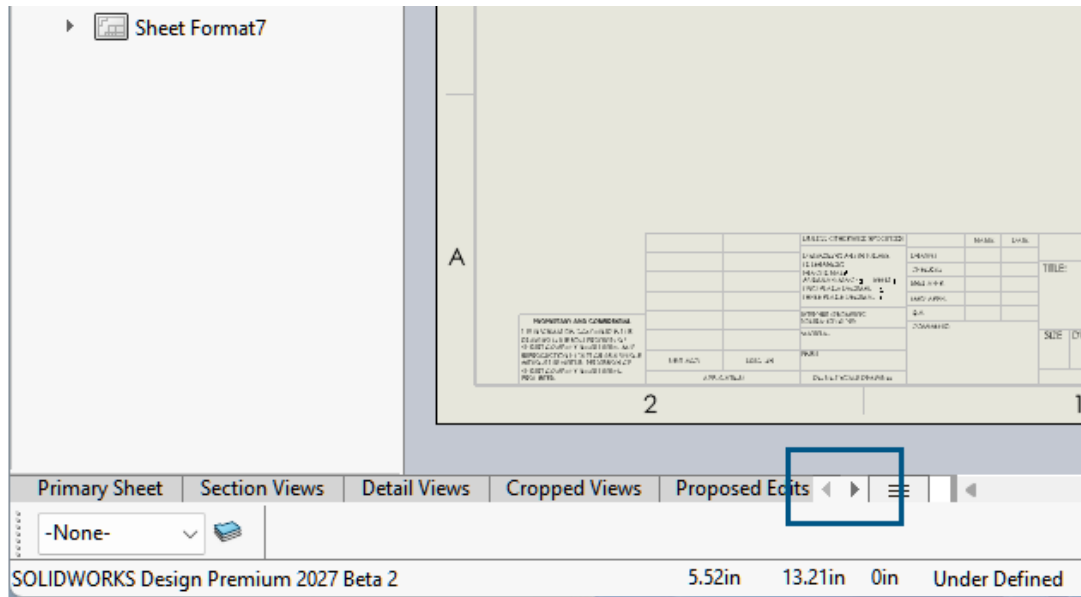
Item	Description
File Status  tab	Contains the Save Status column  and the Read-Only Status column . These show the status for every component in the assembly. This way, parent parts reflect changes in nested components. Modified  The component has been modified. If the component has no icon, it is unmodified. Read-Only  The component is read-only. If the component has no icon, there are no usage restrictions.
Display Pane  tab	Consolidates all existing display state controls, such as light, hidden, dimmed, and transparent, into a single collapsible tab. This keeps the tree view clean and makes it easier to visually find objects. There are no changes to the existing display state functionality.

Resizing Dialog Boxes

The Cut-List-Properties and Pack and Go dialog boxes are enhanced for ease of use.

If you resize the column widths or the dialog boxes, the app uses the same sizing when you reopen the dialog boxes.

Tab Navigation



Documents with overflowing tabs, as drawing sheets or the MotionManager, include arrows to help you navigate to hidden tabs.

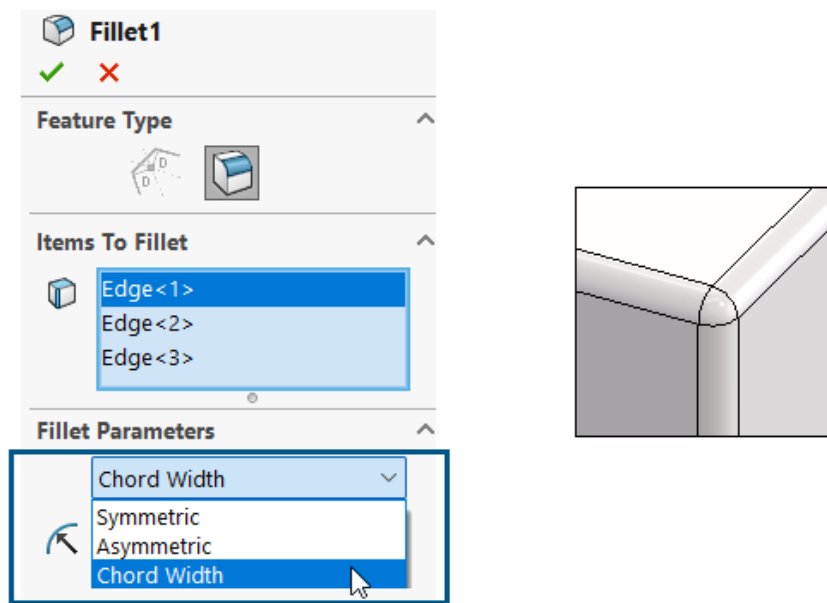
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Parts and Features

This chapter includes the following topics:

- **Chord Width Option for Constant Size Fillets**
- **Split and Save Bodies Enhancements**

Chord Width Option for Constant Size Fillets

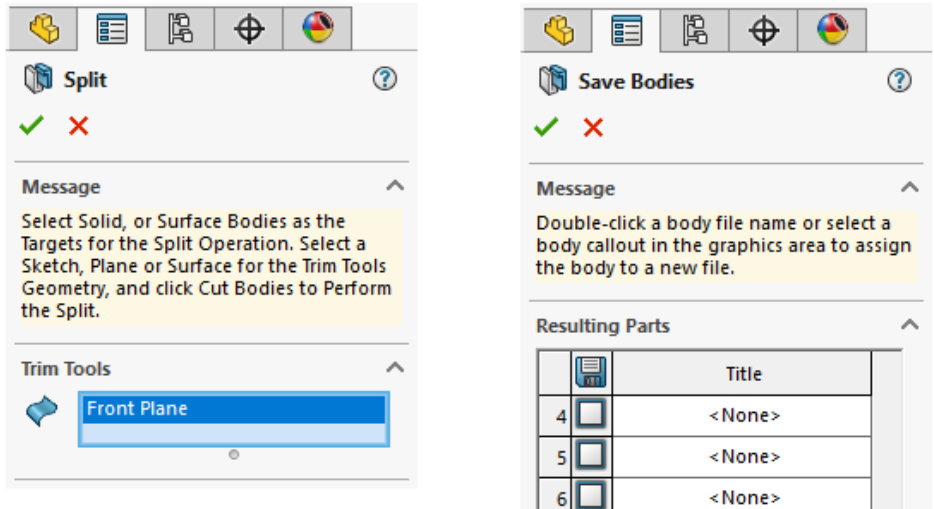


You can apply **Constant Size** fillets with the **Chord Width** parameter directly to edges. Previously, you had to use **Face** fillets to apply chord width parameters.

Benefits: You can apply the **Chord Width** parameter to constant size fillets without switching between fillet modes, speeding up your workflow.

You can also create more complex **Constant Size** fillets, such as three edges meeting at a vertex. This is not feasible with **Face** fillets.

Split and Save Bodies Enhancements

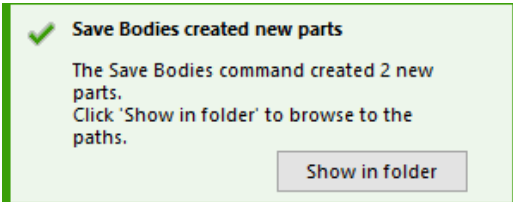
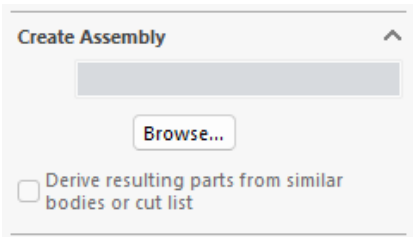
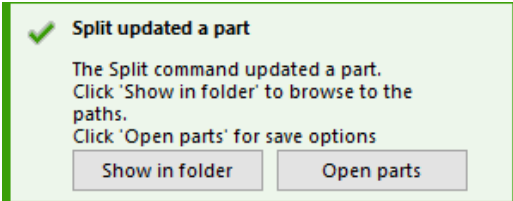


All **3DEXPERIENCE** users can now save split bodies and saved bodies locally before upload, ensuring their part remains on disk even if the app loses the **3DEXPERIENCE** connection. The app provides a single notification, restores the **Derive resulting parts from similar bodies or cut list** check box, and streamlines the **Create Assembly** workflow.

Previously, the functionality differed based on whether you used a connector app or installed **SOLIDWORKS Design** from the **3DEXPERIENCE** platform.

Benefits: You can create assemblies more efficiently, get instant local copies that safeguard your data, and have a single notification confirming uploads, all within familiar workflows.

Enhancement	Description
Local save with Create Assembly selected (New for users who install SOLIDWORKS Design from the 3DEXPERIENCE platform)	The Save Bodies command with Create Assembly selected first saves the models locally. The Browse button is restored, allowing you to optionally navigate to a local folder to save the assembly. Previously, a separate Save to 3DEXPERIENCE dialog box opened for each part and the assembly.
Local save without Create Assembly selected (New for users who install SOLIDWORKS Design from the 3DEXPERIENCE platform)	When you run the Save Bodies command without Create Assembly selected, the app saves the models locally before uploading them to the platform.

Enhancement	Description
Notification of part creation or update for Save Bodies (New for users who install SOLIDWORKS Design from the 3DEXPERIENCE platform)  A notification box with a green border and a green checkmark icon. The title is "Save Bodies created new parts". The text says: "The Save Bodies command created 2 new parts. Click 'Show in folder' to browse to the paths." There is a button labeled "Show in folder".	After you run the Save Bodies command, a notification indicates whether the command created new parts or updated existing parts, and offers to open the resulting location in File Explorer.
Derive parts option (New for users who install SOLIDWORKS Design from the 3DEXPERIENCE platform)  A screenshot of the "Create Assembly" dialog box. It has a title bar "Create Assembly" with a collapse icon. Below the title bar is a text input field. Below the input field is a "Browse..." button. At the bottom, there is a checkbox labeled "Derive resulting parts from similar bodies or cut list".	If you choose to create an assembly, the app now provides the Derive resulting parts from similar bodies or cut list option in the Create Assembly section.
Split body local save (New for users who install SOLIDWORKS Design from the 3DEXPERIENCE platform)	The Split Body command now saves the split bodies locally first, then uploads them to the 3DEXPERIENCE platform. Previously, a Save to 3DEXPERIENCE dialog box appeared for each resulting part.
Notification of part creation or update for Split Bodies (New for users who install SOLIDWORKS Design from the 3DEXPERIENCE platform)  A notification box with a green border and a green checkmark icon. The title is "Split updated a part". The text says: "The Split command updated a part. Click 'Show in folder' to browse to the paths. Click 'Open parts' for save options". There are two buttons: "Show in folder" and "Open parts".	After you run the Split Bodies command, a notification indicates whether the command created new parts or updated existing parts, and offers to open the resulting location in File Explorer.

7

Sheet Metal

This chapter includes the following topics:

- **Upgrading Sheet Metal Architecture**
- **Stamps Across Bends**

Upgrading Sheet Metal Architecture

If you created sheet metal models with legacy architecture, you can convert them to the current architecture. After conversion, you can use all sheet metal tools with the models, but you cannot revert to the legacy architecture.

In the FeatureManager design tree, right-click the top-level part or assembly and select **Upgrade Sheet Metal features**. After the conversion, the **Sheet-Metal**  and **Flat-Pattern**  features appear in the FeatureManager design tree.

For derived parts, the tool upgrades only the derived part, not the parent part. For assemblies, the tool upgrades all components.

Upgrading improves performance and stability and to gives you access to all new sheet metal functionality.

Stamps Across Bends

You can use the **Stamp** tool to create stamps across sheet metal bends (multiple faces).

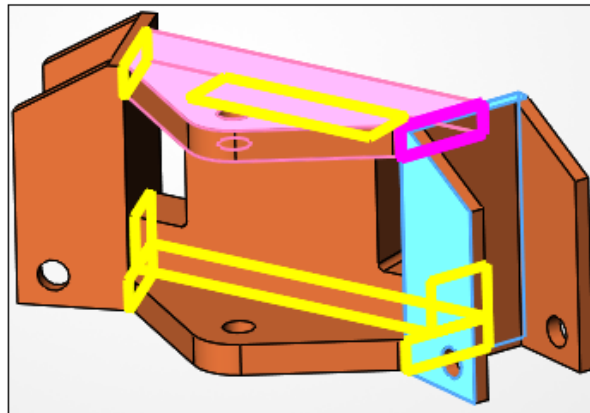
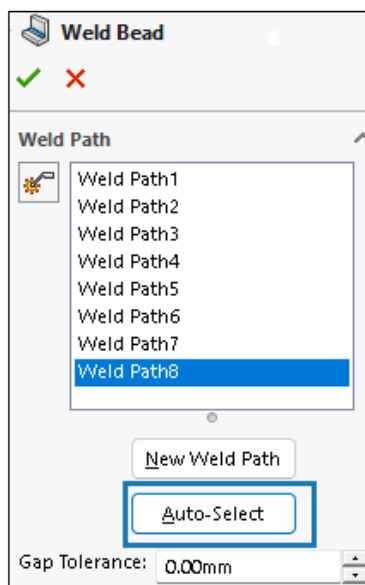
Limitations:

- Only planar faces are supported.
- You cannot create multiple-level stamps, otherwise the app cannot fold the part.
- After folding the sheet metal part, if you modify the stamp faces (such as cuts or fillets), those changes do not appear on the flattened part.

8

Structure System and Weldments

Weld Bead PropertyManager



You can use **Auto-Select** to select all weld faces and add weld beads for multibody parts and assemblies.

Gap Tolerance lets you add weld beads if the gap between two faces is less than or equal to the specified gap.

Benefit: Saves time by automatically selecting weld faces.

To open this PropertyManager:

In a part, click **Weld Bead**  (Weldments toolbar) or **Insert** > **Weldments** > **Weld Bead**. In an assembly, click **Insert** > **Assembly Feature** > **Weld Bead**.

9

Assemblies

Preventing Rotation in Coincident Mates and Mated Cones

You can prevent the rotation of mates in coincident mates between pairs of linear entities, coincident cones, and distance and limit distance between cones.

To prevent rotation of a coincident mate or mated cone in an assembly:

1. Open a model that has mated cones or a coincident mate between pairs of linear entities.
2. In the FeatureManager design tree, select the mate and click **Edit Feature** .
3. In the PropertyManager, select **Lock rotation**.

If you select **Lock rotation** on a component that is constrained with no remaining degrees of freedom and no mate conflicts, a tooltip indicates that you cannot lock the rotation.

4. Click .

In the FeatureManager design tree, right-click the **Mates**  folder and click **Lock rotation** to prevent the rotation of all mates in the model.

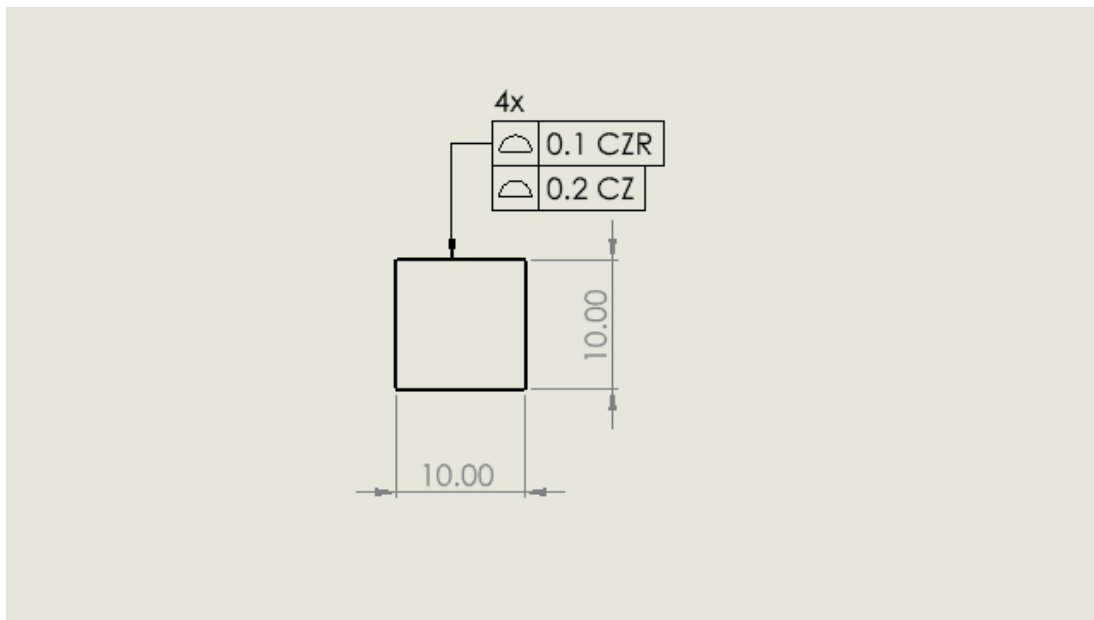
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Detailing and Drawings

This chapter includes the following topics:

- **Zone Tolerances in Geometric Tolerance Symbols**
- **Areal Symbols in Surface Finish Symbols**
- **Exclude Drawing Views from Rebuilding**
- **Locking Sheet Formats**
- **General Tolerance Tables**
- **Centerline Font**

Zone Tolerances in Geometric Tolerance Symbols



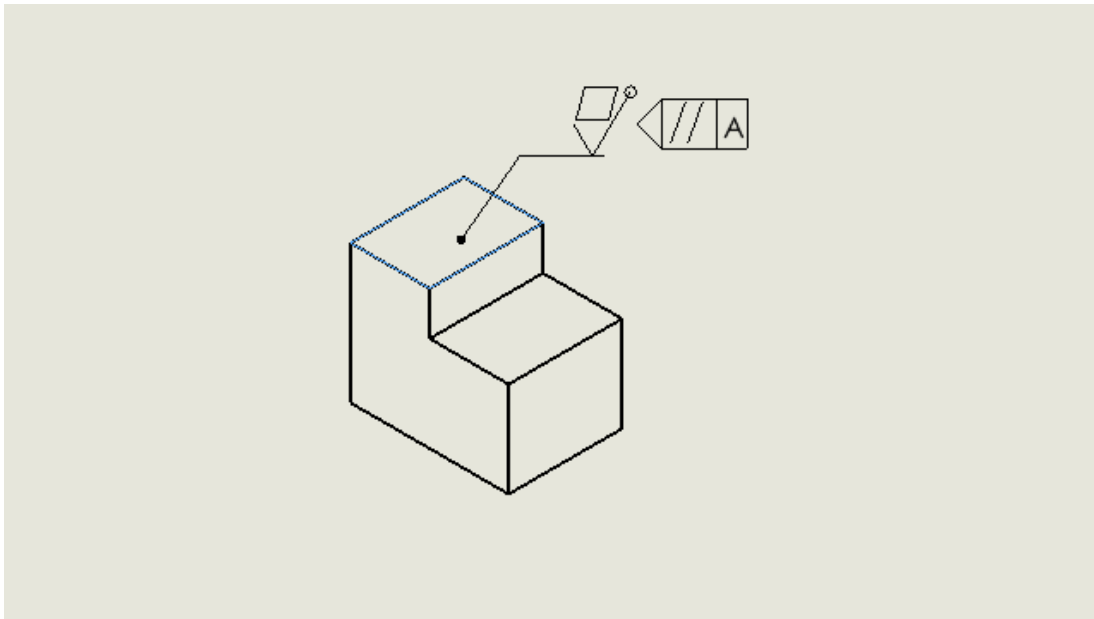
For drawings in the ISO or DIN standard, you can specify a CZR (combined zone rotational) tolerance zone. You can use the CZ, SZ, and CZR tolerance zones in combination together. Previously, the CZ and SZ zones were mutually exclusive.

To add zone tolerances in geometric tolerance symbols:

1. In a drawing in the ISO or DIN standard, click **Geometric Tolerance**  (Annotation toolbar) or **Insert > Annotations > Geometric Tolerance**
2. In the Geometric Tolerance PropertyManager, specify options.

3. On the drawing sheet, click to place the symbol.
4. In the Tolerance dialog box:
 - a. Click a symbol.
 - b. Select **Offset Zone**.
 - c. For **Combination**, select **CZ**, **SZ**, or **CZR**.
 - d. Click **Done**.

Areal Symbols in Surface Finish Symbols

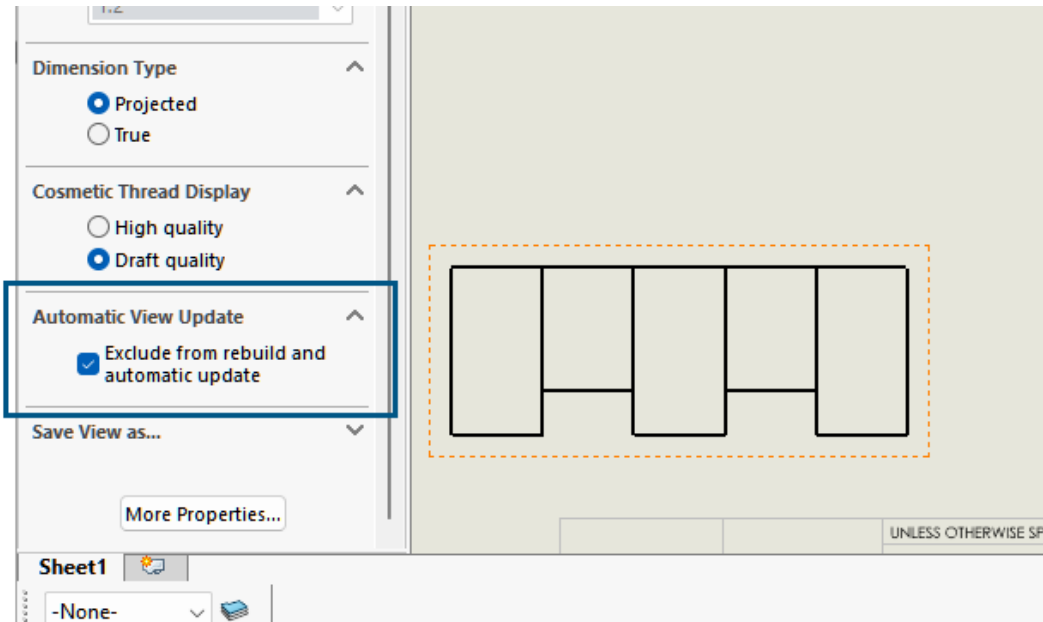


To comply with the ISO 25178 standard, you can display areal symbols in surface finish symbols. Areal symbols indicate 3D surface texture.

In the Surface Finish PropertyManager, under **Symbol**, select **Show Areal Symbols** and specify the **Symbol** and **Symbol Layout**:

	Basic
	Machining Required
	Machining Prohibited
	Local
	All Around

Exclude Drawing Views from Rebuilding



You can exclude drawing views from rebuilding when you click **Rebuild** or **Ctrl+B**.

In a drawing view PropertyManager, under **Automatic View Update**, select **Exclude from rebuild and automatic update**.

This prevents specified drawing views from updating if you rebuild the drawing.

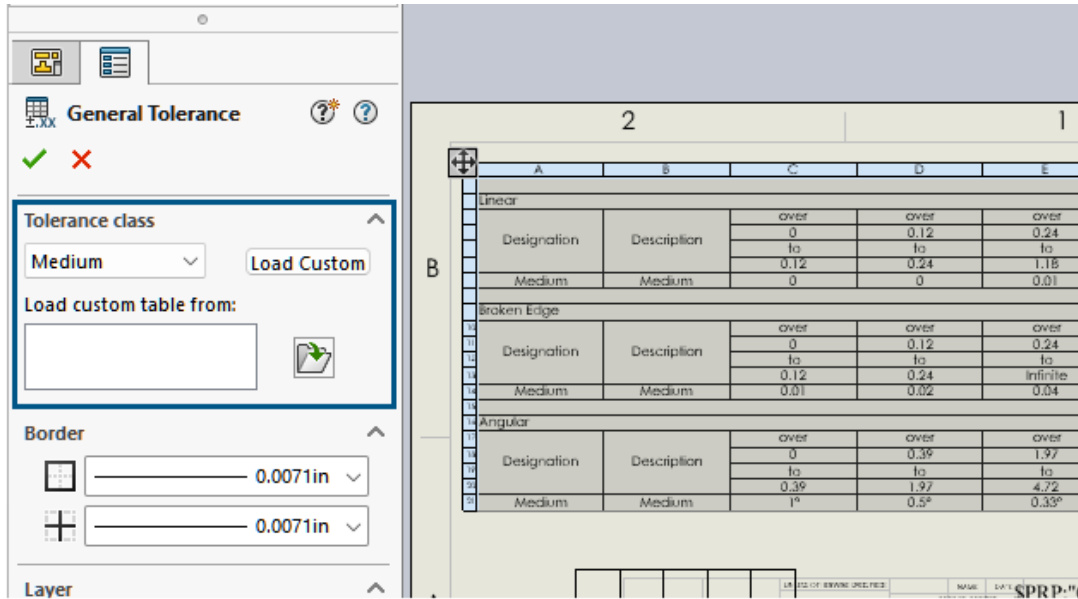
Locking Sheet Formats

You can lock drawing sheet formats to prevent edits to borders and title blocks.

In **Tools > Options > System Options > Drawings**, select **Disable sheet format edits**.

If you are connected to the **3DEXPERIENCE** platform, administrators can use the Settings Administrator Tool to specify the option across all users.

General Tolerance Tables



You can add columns and rows to general tolerance tables, and customize the tolerance designation and description.

Previously, general tolerance tables had only nine columns by default. Certain standards require additional rows, so you can add rows and columns to ensure that the table complies with standards.

In the General Tolerance PropertyManager, under **Tolerance Class**, you can specify table parameters. Previously, this was available in the Document Properties - DimXpert dialog box only. The PropertyManager includes these parameters:

- **Tolerance Class**
- **Load Custom**
- **Load custom table from**
- **Open**

Centerline Font

You can specify a centerline font for centerlines in drawings, similar to using the font in center marks. The option makes the centerline extension lines follow the font selected for centerlines.

In **Tools > Options > Document Properties > Centerlines/Center Marks**, under **Centerlines**, select **Centerline font**.

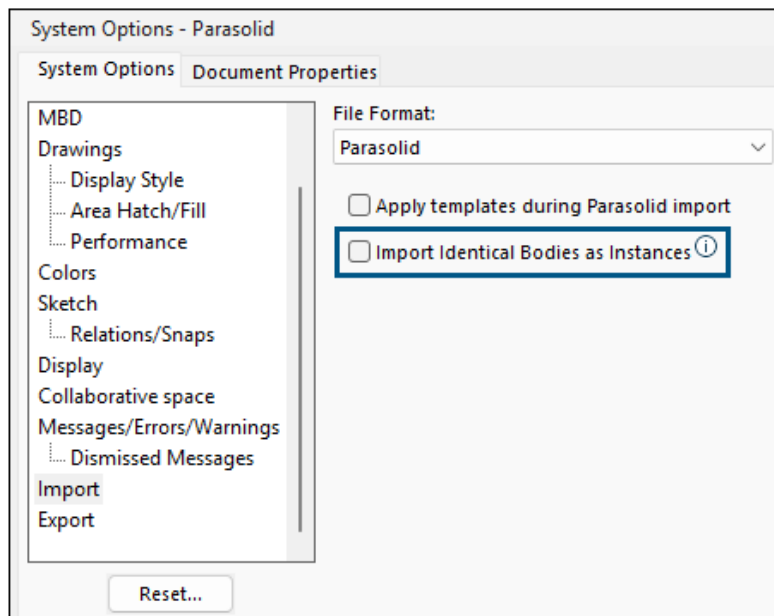
11

Import/Export

This chapter includes the following topics:

- **Duplicate Body Detection for Parasolid Assembly Import**
- **Importing E57 Point Clouds**

Duplicate Body Detection for Parasolid Assembly Import



You can achieve a more complete data interchange between Parasolid assemblies (X_B and X_T) that you import into SOLIDWORKS Design.

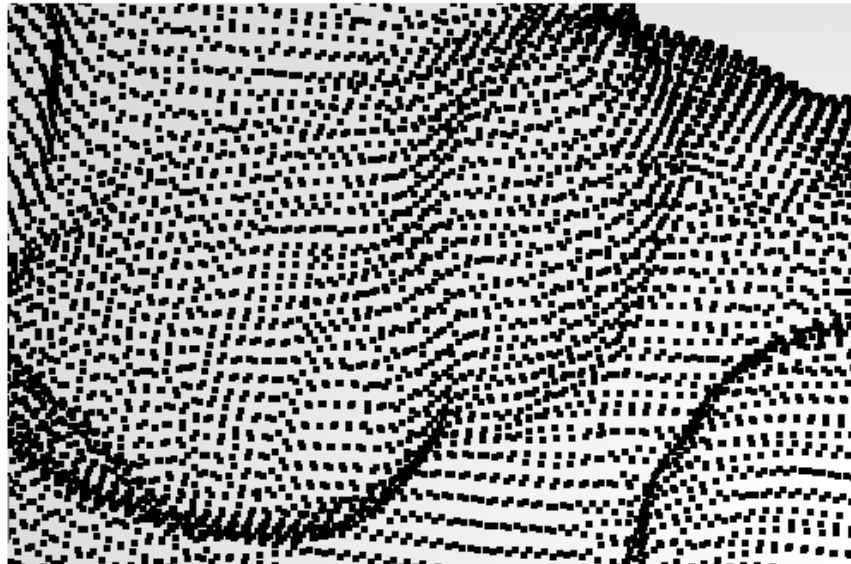
The software detects multiple instances of identical geometric bodies and imports the identical bodies as multiple instances of a single model instead of multiple components that have identical geometry. To enable this functionality, click **Tools > Options > System Options > Import > File Format > Parasolid** and select **Import Identical Bodies as Instances**.

This functionality supports identifying duplicated assembly components, not duplicated assemblies.

Benefits: Imported Parasolid assemblies keep the original copy relationships, store fewer separate solids, reduce RAM consumption during assembly operations, and improve performance.

Legacy	2027 and Later
<pre> TopAssy --- HexHeadBolt<1> --- HexHeadBolt_1<1> --- HexHeadBolt_2<1> </pre>	<pre> TopAssy --- HexHeadBolt<1> --- HexHeadBolt<2> --- HexHeadBolt<3> </pre>
Imports duplicated identical bodies as new components.	Imports duplicated identical bodies as instances of the same component.

Importing E57 Point Clouds



You can now import large E57 point cloud files into SOLIDWORKS Design that appear as point cloud features in the FeatureManager design tree.

The import shows information about the point cloud before it is imported. You can perform basic commands on the point clouds such as **Hide/Show**, **Delete**, and **Section View**.

You cannot reference any point cloud points in other features. For example, you cannot use a point cloud point as an **Up to Vertex** selection for **End Condition** in an extrude feature.

Benefits: You can avoid third-party scanner-management tools and refresh large point clouds instantly, keeping the data current without reimporting or recreating geometry.

You must have SOLIDWORKS Design Premium or higher to access this functionality.

To import point clouds:

1. In SOLIDWORKS Design, open a part and click **Insert** > **Point Cloud** .
2. In the Open dialog box, browse to select the point cloud file and click **OK**.

The Import Large Cloud of Points dialog box opens. It displays key information about the point cloud.

3. Click **OK**.

A status bar in the lower-left corner reports on the import process and the point cloud loads. The point cloud feature  appears in the FeatureManager design tree. The point cloud retains and displays its color information in the graphics area to improve clarity.

The app saves the point cloud data to a local SOLIDWORKS Design repository, whose location you can modify. There can be multiple repositories for different point clouds in the same model or in different models.

You cannot save the point cloud file to the **3DEXPERIENCE** platform.

4. You can interact with the point cloud:

- Hide and show
- Create section views
- Delete
- Zoom in and out

5. Save the model.

The model contains a reference to the locally saved point cloud (E57 file) so the model file size does not increase significantly. The reference to the point cloud persists across sessions. If you save the model to the **3DEXPERIENCE** platform, there is no relationship captured to the locally saved point cloud. Therefore, the point cloud is not visible in any downstream **3DEXPERIENCE** platform applications because the cloud is not stored on the platform.

If an assembly references the part with a point cloud, the point cloud is visible in the assembly, except in Large Design Review.

6. To use a different point cloud, right-click the point cloud feature and click **Edit Feature**.

If you choose a new point cloud, it completely replaces the previous point cloud.

The Open dialog box appears.

7. Browse to select a new point cloud E57 file and open it.

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SOLIDWORKS PDM

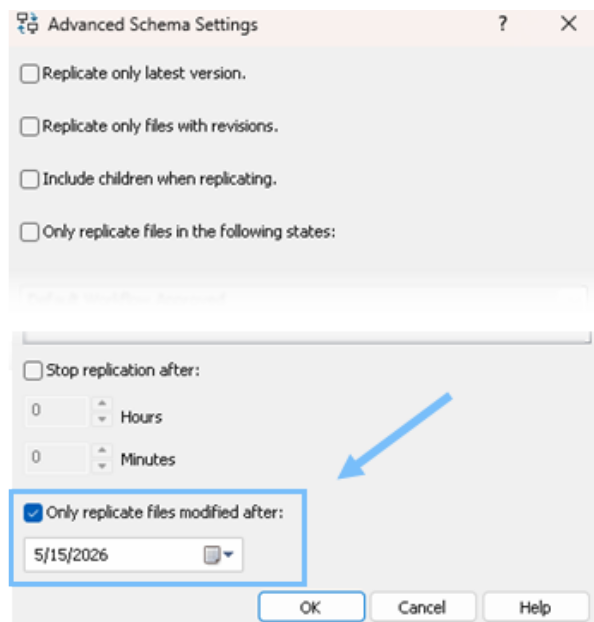
This chapter includes the following topics:

- **Replicating Archives by File Modification Date**
- **Single Sign-On (SSO) Authentication for Vault Views**
- **Custom Column Sets in the Web2 Client**
- **Days in State Column in Contains and Where Used Tabs**

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.

Replicating Archives by File Modification Date



You can schedule an archive to replicate only files that were modified after a certain date. This functionality does not apply to on-demand replication.

Benefits: Replicating by modification date saves the time and cost of replicating obsolete data.

To replicate archives by file modification date:

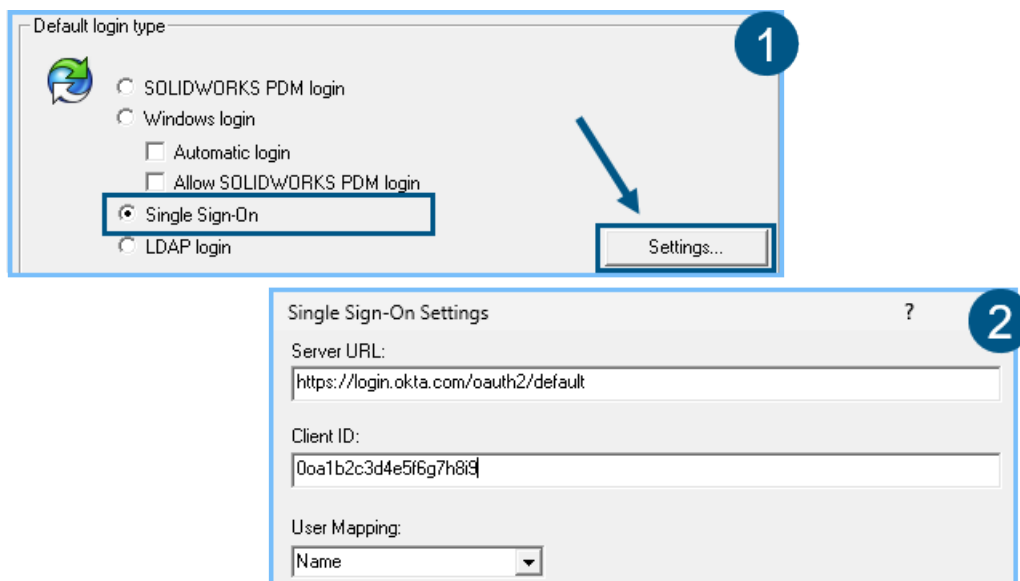
1. In the SOLIDWORKS PDM Administration Tool, expand a replicated vault and double-click **Replication**.
2. In Replication Settings, click **New Schema**.
3. Define the name and contents of the schema.

See [Defining Replication](#) in the *SOLIDWORKS PDM Replication Guide*.

4. Click **Advanced**.
5. In Advanced Schema Settings, select **Only replicate files modified after:**.
6. Click **Date** and select a date.
7. Click **OK**.

If you select **Include children when replicating**, all children will be replicated, even if they were not modified after that date.

Single Sign-On (SSO) Authentication for Vault Views



Administrators can configure single-sign-on (SSO) authentication for users to access their vault views. With SSO enabled, users can sign in with their organization's credentials and multi-factor authentication (MFA) methods instead of a PDM or Windows® user name and password.

Benefits: Using one credential improves security, simplifies user administration, and reduces password fatigue.

SOLIDWORKS PDM supports the following Identity Providers (IdPs):

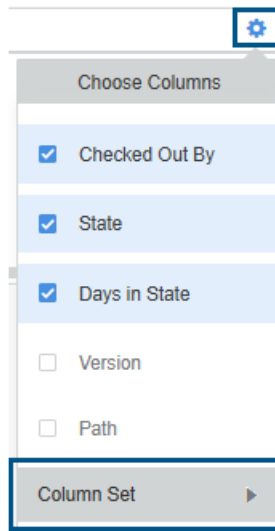
- Auth0®
- Okta®
- Keycloak™
- Microsoft® Entra® ID
- AWS® IAM Identity Center

SOLIDWORKS PDM does not support:

- More than one IdP per vault.
- Different IdPs for different vaults within one organization.
- SAML protocol.

To configure a vault for SSO and MFA, see [Single-Sign-On \(SSO\) Settings Dialog Box](#) in the *SOLIDWORKS PDM Archive Server Configuration Tool Guide*.

Custom Column Sets in the Web2 Client



Web2 users can use custom column sets configured by their vault administrators in the List View of the large screen layout.

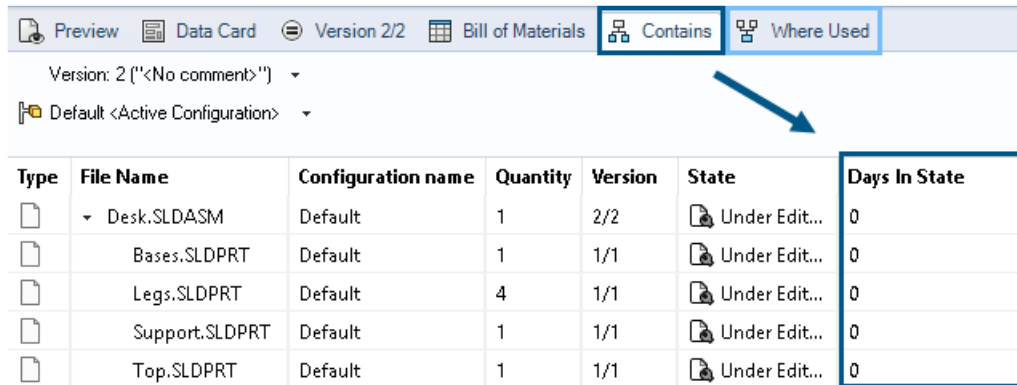
Benefits: Customizing column sets helps users find information faster.

To access custom column sets, the Web2 Client must be using the SOLIDWORKS PDM WebAPI.

To create a custom column set, see [Creating a Customized Column Set](#) in the SOLIDWORKS PDM Administration Tool Help.

To use a custom column set in the List View of the Web2 Client, click **Show More** . If your administrator has given you permission to view custom column sets, the **Column Set** option appears at the bottom of the menu. Click **Column Set** to expand the menu, and select an option from the list.

Days in State Column in Contains and Where Used Tabs



Type	File Name	Configuration name	Quantity	Version	State	Days In State
	▼ Desk.SLDASM	Default	1	2/2	 Under Edit...	0
	Bases.SLDPRT	Default	1	1/1	 Under Edit...	0
	Legs.SLDPRT	Default	4	1/1	 Under Edit...	0
	Support.SLDPRT	Default	1	1/1	 Under Edit...	0
	Top.SLDPRT	Default	1	1/1	 Under Edit...	0

You can view a file's **Days in State** in the Contains and Where Used tabs.

To view how many days a file has existed in its current workflow state, select the file and click **Contains**  or **Where Used** . The information appears in a column of the table.

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SOLIDWORKS Manage

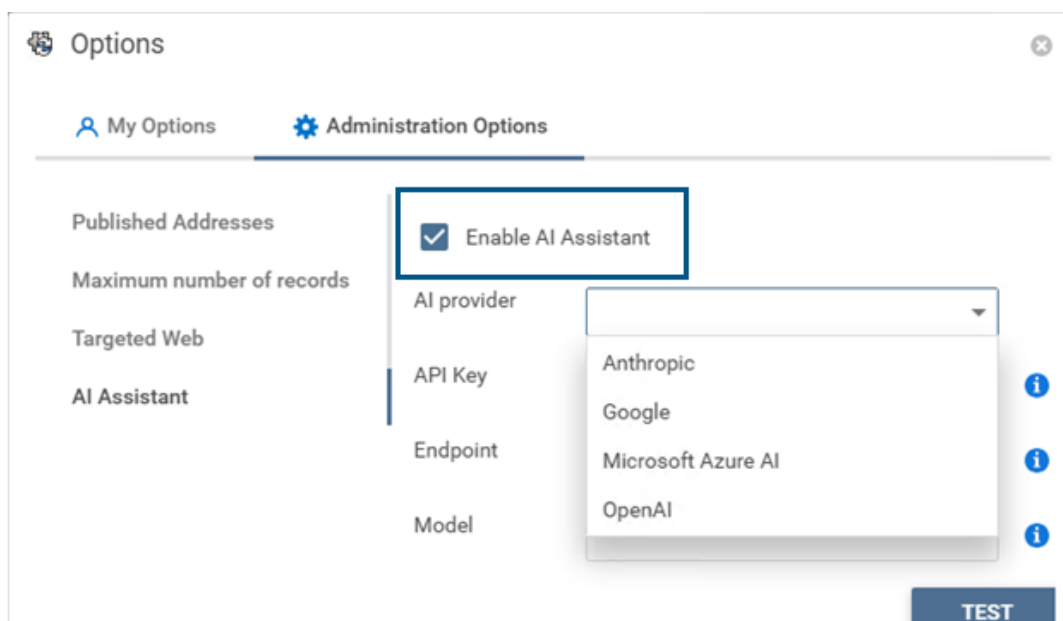
This chapter includes the following topics:

- **AI Functionality**
- **Bill of Materials Templates**
- **Column Sorting in BOM Compare**
- **Active Configuration Data Handling**
- **Microsoft Outlook Integration**
- **Linear Graphs for Process Records**

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.

AI Functionality



You can connect several AI platforms to SOLIDWORKS Manage Help, Web API, and demo API projects. This helps you ask questions and create applications using the API.

For information on how Dassault Systèmes uses AI technology, see [AI in the 3DEXPERIENCE Platform](#).

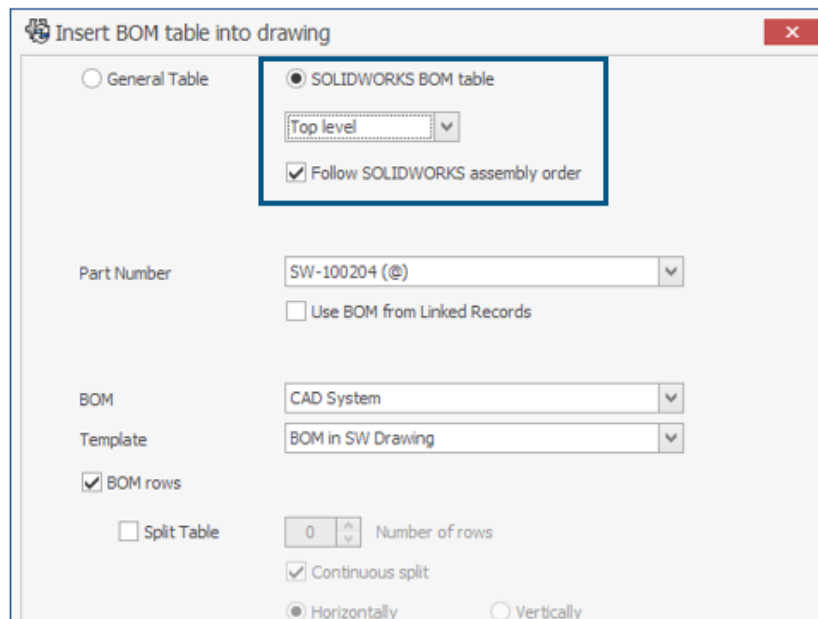
You can access the AI Assistant using the SOLIDWORKS Manage Plenary Web client.

To turn on the AI Assistant:

1. Log in to the Plenary Web client as an administrator.
2. In the Options dialog box, on the Administrative Options tab, select **Enable AI Assistant**.
3. Enter the required fields which you can get from the AI platform provider.
4. In **Permissions**, select which users and groups can access the AI Assistant.
5. Click **Save** and then **Close**.

The AI Assistant appears in a tab in the right-hand flyout panel.

Bill of Materials Templates

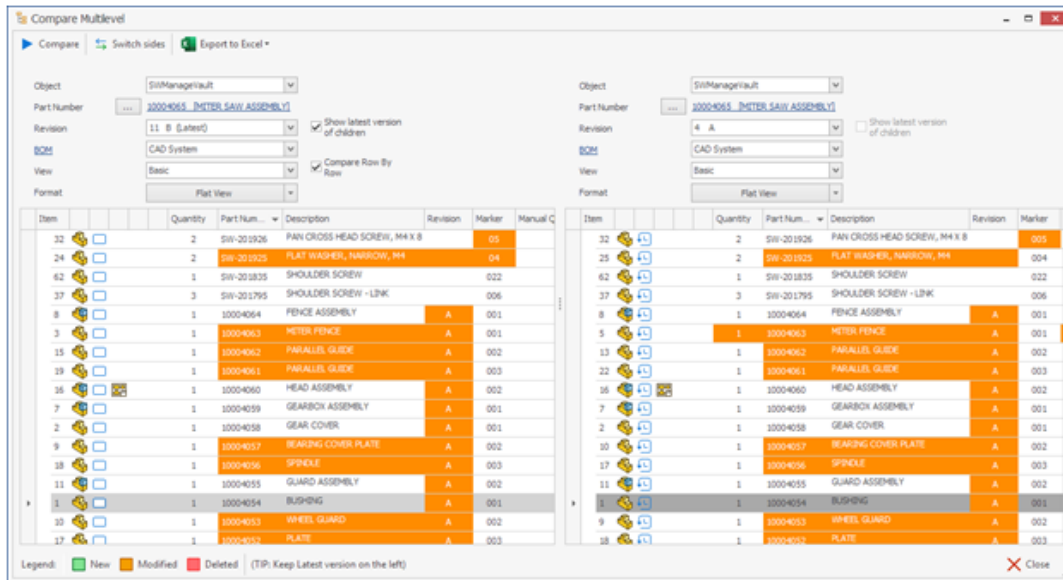


When you insert a bill of materials (BOM) from the structure in SOLIDWORKS Manage into a SOLIDWORKS drawing, SOLIDWORKS Manage uses a BOM template instead of a general table template.

The templates support automatic balloon numbering and BOM types of **Top-level only** or **Parts only** (**Indented** is not supported).

You can still use the general table template which includes **Flattened** BOMs.

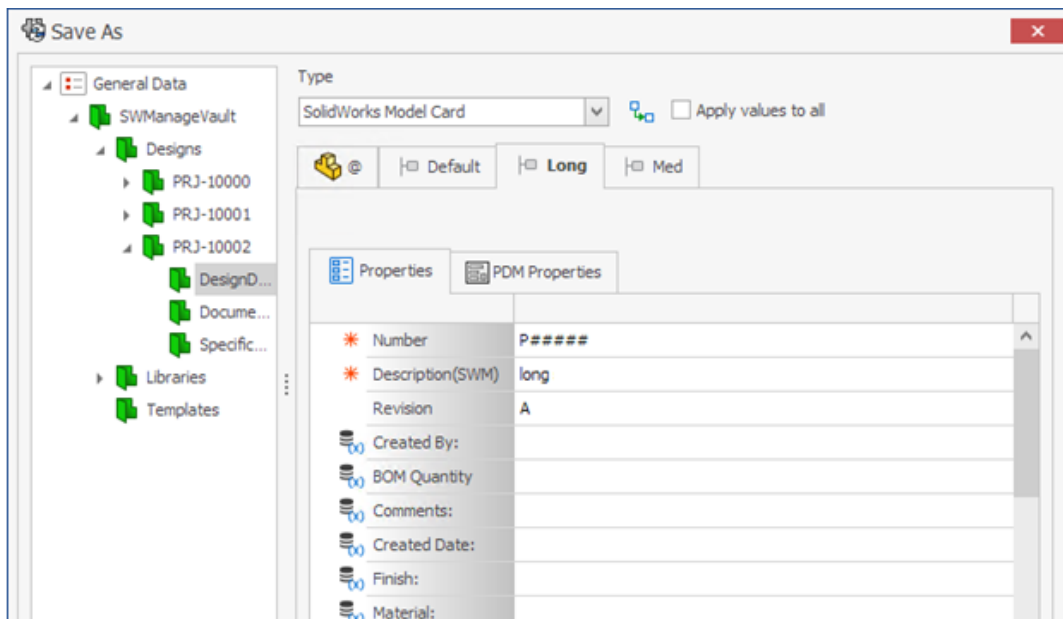
Column Sorting in BOM Compare



When comparing BOMs using the **Compare Row by Row** option, you can sort based on any column so that both BOM views update.

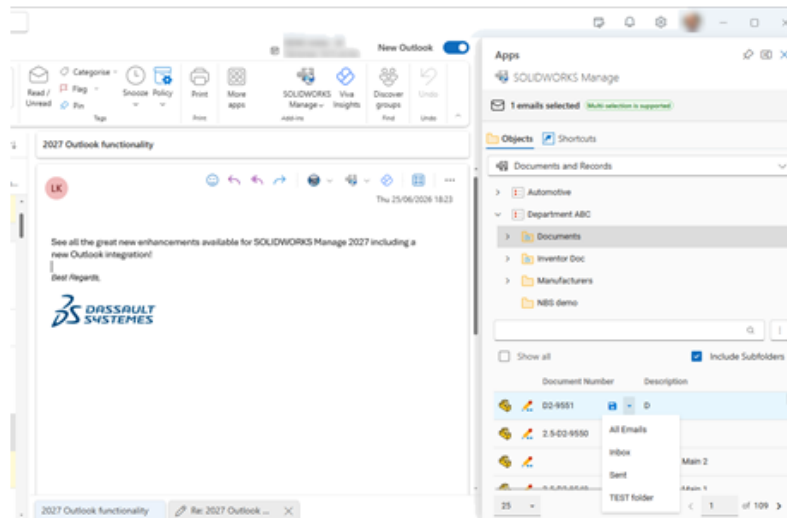
This helps you see differences more easily when the BOM order may be different between the two views. Row-by-row comparison checks the cell in each row in one view and compares it to the same row and cell in the other view.

Active Configuration Data Handling



When creating a new SOLIDWORKS record from an existing multiple configuration file with the SOLIDWORKS Manage **Save As** tool, you can use an editable data grid containing the configurations and associated fields. This helps you fill in field values faster than opening each configuration record separately.

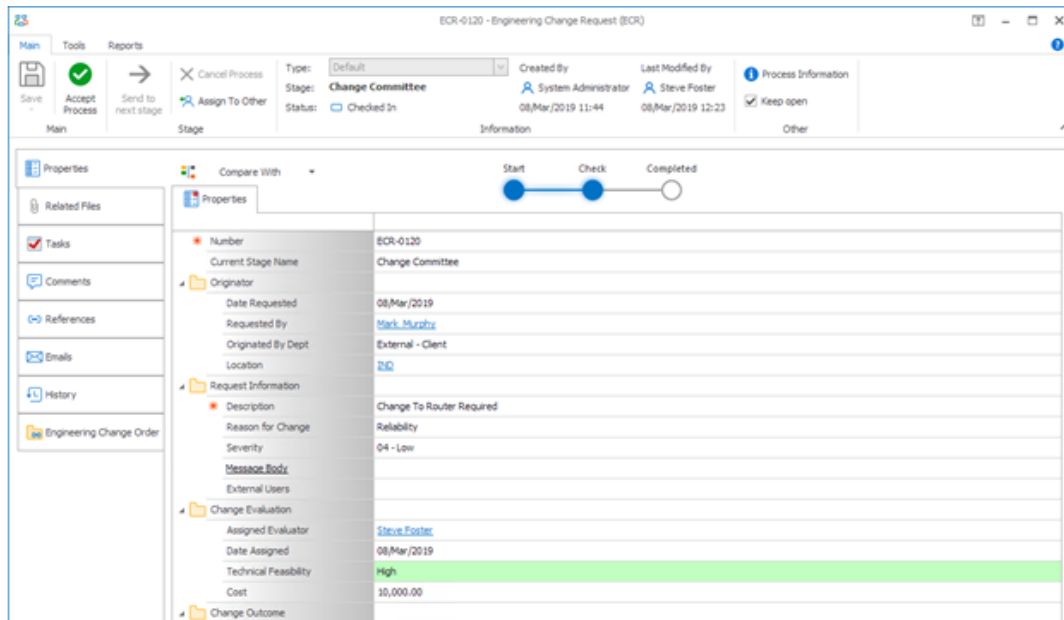
Microsoft Outlook Integration



The integration for Microsoft Outlook® supports the Classic Outlook and updated Outlook versions.

The interface includes a side panel where you can choose the SOLIDWORKS Manage object to create records from selected emails in. You can select individual SOLIDWORKS Manage records to attach emails to.

Linear Graphs for Process Records



You can configure process records to show a linear graph depicting the major phases of a process. This helps you understand where a process is in its overall progress.

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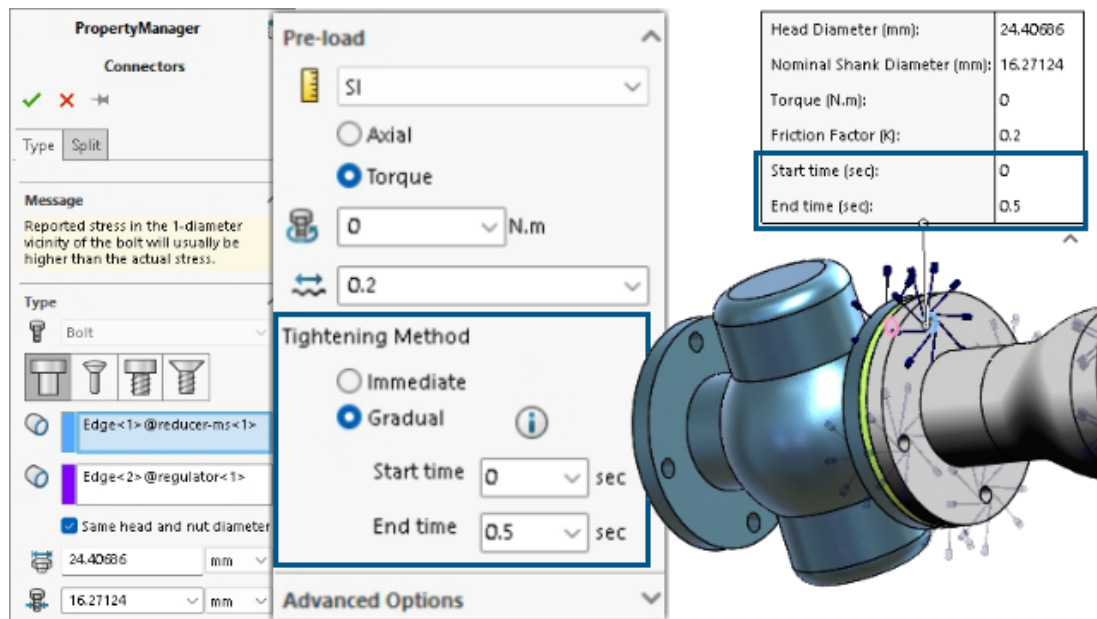
SOLIDWORKS Simulation

This chapter includes the following topics:

- **Preloading Bolts for Nonlinear Studies**
- **General Spring Connector in Nonlinear Studies**
- **Mesh Generation Quality Checks**
- **Improved Accuracy for Heat Power Calculations**
- **Viewing Interface Stresses for Bonded Interactions**
- **User Interface**

SOLIDWORKS® Simulation Standard, SOLIDWORKS Simulation Professional, and SOLIDWORKS Simulation Premium are separately purchased products.

Preloading Bolts for Nonlinear Studies



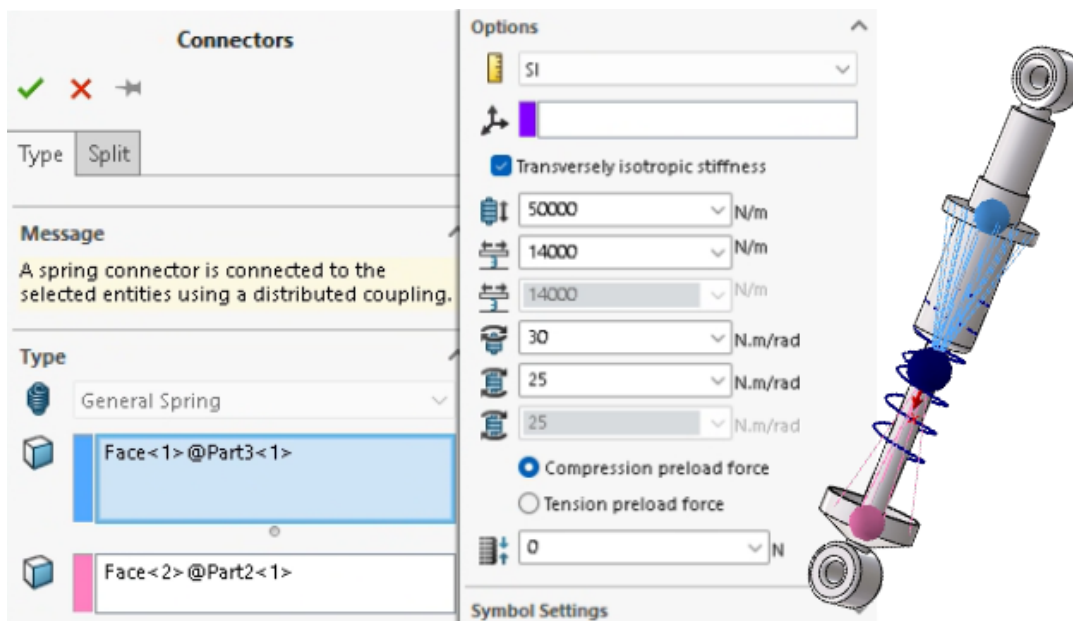
You can apply time-based preloads to simulate bolt connectors more accurately in nonlinear studies.

Two options are available for tightening bolt connectors in nonlinear studies.

Tightening Method	Description
Gradual	The bolts are gradually tightened with the given preload based on start time and end time settings.
Immediate	The bolts are preloaded before the start of the nonlinear analysis.

When you open an existing nonlinear study, bolt connectors use the immediate tightening method to preserve the behavior of the original study.

General Spring Connector in Nonlinear Studies



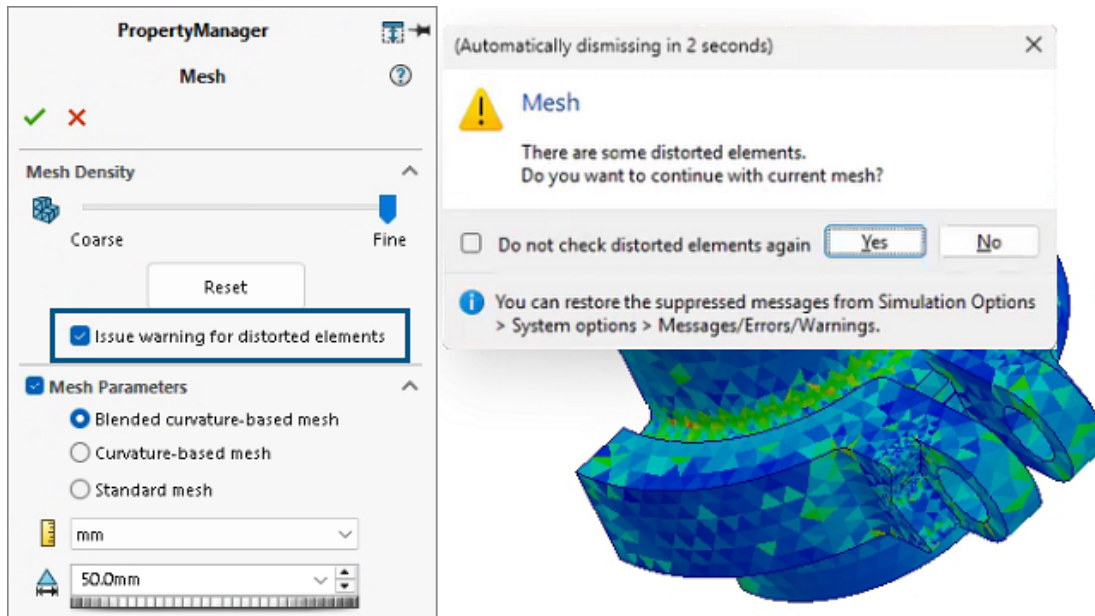
The application of **General Spring** connector is now extended to nonlinear static and nonlinear dynamic studies, enabling accurate representation of spring elements in simulations involving large displacement and other nonlinear effects.

To open the **General Spring** PropertyManager, right-click **Connections** in the simulation study tree and select **General Spring**.

You can specify a general spring connector between flat, nonflat, and concentric cylindrical surfaces. To accurately represent the general spring connector, you can define up to six stiffness parameters using a local coordinate system.

After running a simulation, you can extract the forces acting on the general spring connectors, such as shear force, axial force, bending moment, and torque by right-clicking **Results** and selecting **List Connector Forces**.

Mesh Generation Quality Checks



The mesh quality diagnostic tool that detects distorted elements with negative Jacobian ratios is enhanced to also detect elements with high aspect ratios, zero areas, or zero volumes.

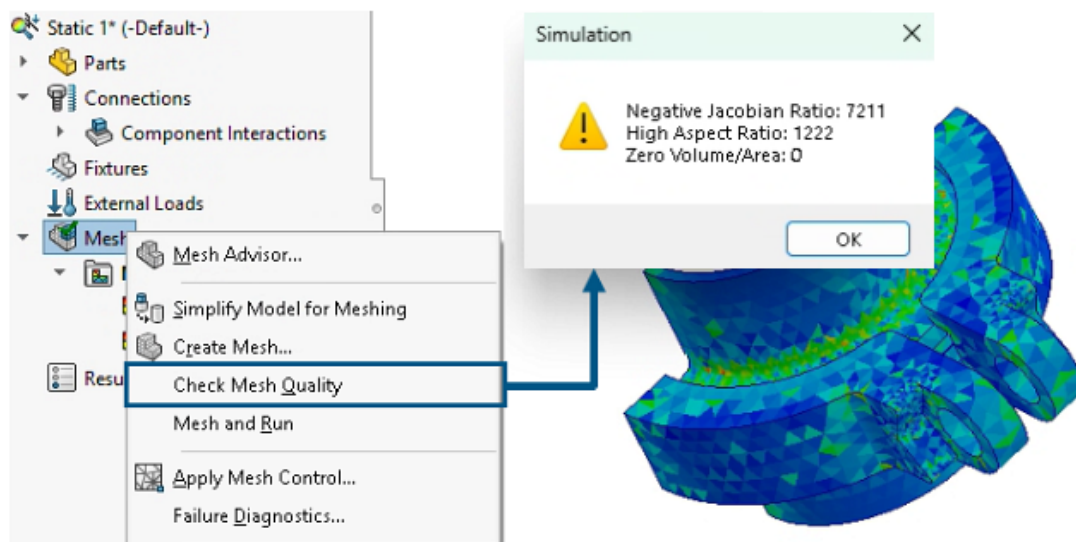
Select **Issue warning for distorted elements** in the Mesh PropertyManager to display a warning when the diagnostic tool detects poor-quality elements. You can continue with the current mesh, or cancel the mesh to check for geometry irregularities and change the mesh settings.

Refine the mesh in areas where distorted elements are detected to improve the mesh quality.

After mesh generation completes, SOLIDWORKS Simulation automatically displays a mesh quality report which displays the number of elements with issues such as:

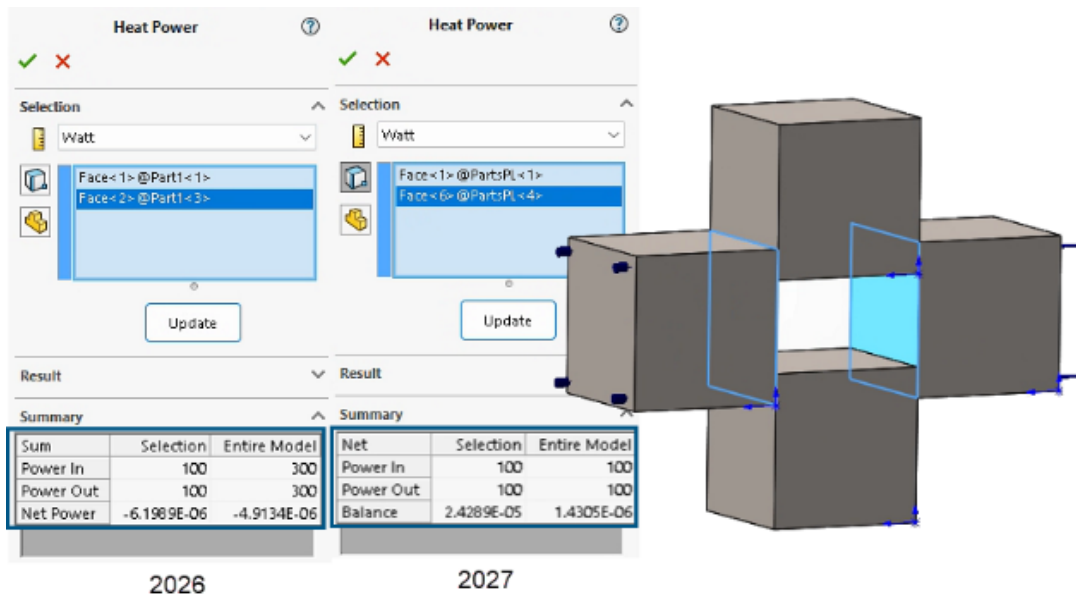
- Negative Jacobian ratios,
- High aspect ratios,
- Zero areas, or
- Zero volumes.

You can also review mesh quality, by right-clicking **Mesh** in the study tree and selecting **Check Mesh Quality**.



In addition to the mesh quality checks, the Standard, Curvature-based, and Blended curvature-based meshing algorithms now use the Converged Volume Mesher (CVM) as the default volume filler for solid meshing. Converged Volume Mesher creates high quality meshes with improved performance.

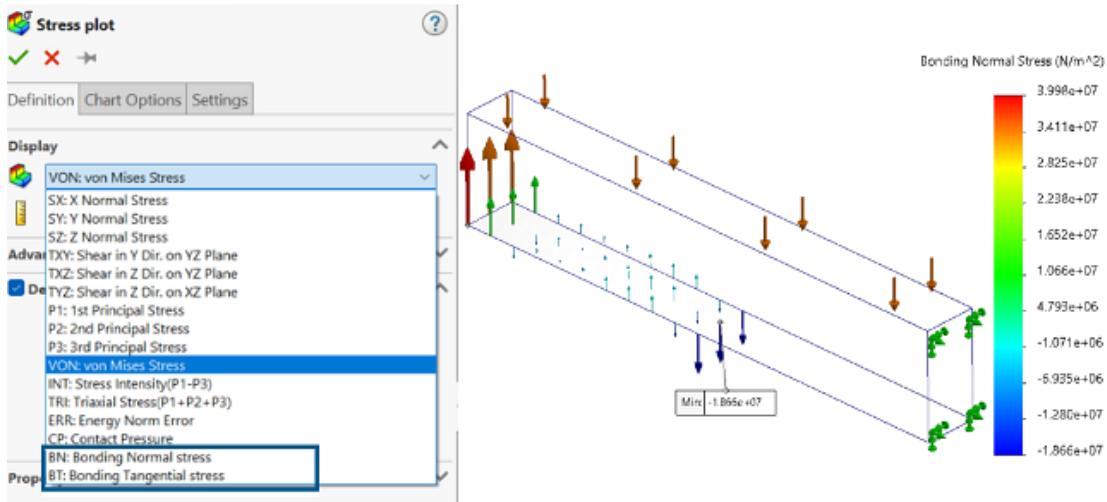
Improved Accuracy for Heat Power Calculations



Heat power results summary now correctly accounts for heat transfer between contacting or bonded bodies in thermal studies.

Internal heat flow between connected bodies is now automatically balanced when computing overall model heat power summaries, resulting in accurate total heat power inlet and outlet values for the entire structure as well as selected bodies or faces.

Viewing Interface Stresses for Bonded Interactions



You can now view normal and tangential interface stresses for bonded interactions with **Surface to Surface** bonding formulation.

Normal and tangential interface stress components help you identify regions with high tensile, compressive, or shear stresses at bonded interfaces.

To plot normal and tangential interface stress components:

1. In the simulation study tree, right-click **Results** and select **Define Stress Plot**.
2. From the **Component** options, select **BN: Bonding Normal stress** or **BT: Bonding Tangential stress** and click **OK** .

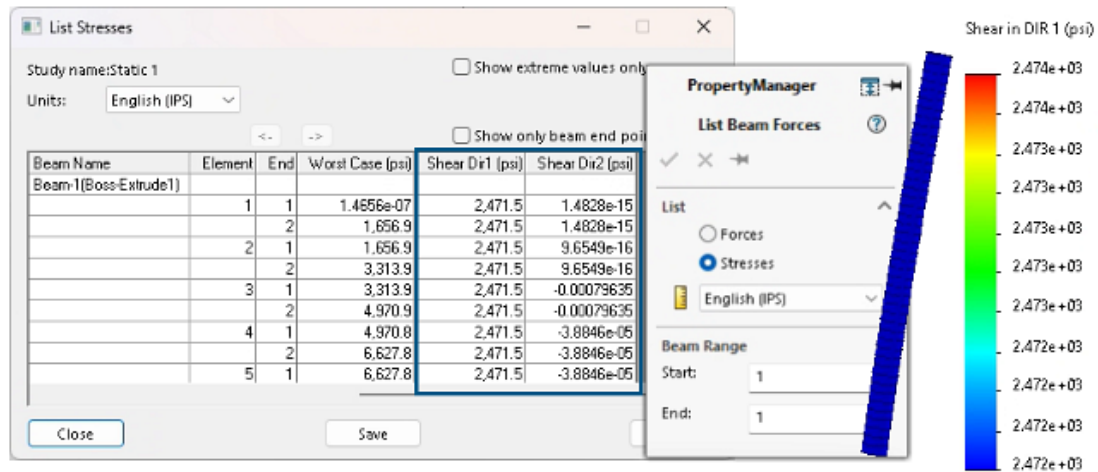
You can also list the numerical values of bonding normal and tangential stress components at each interface node from the **List Results** PropertyManager.

A vector plot helps visualize whether a bonded component is under compression or tension. Normal interface stresses act perpendicular to the bonded surface. The vector plot shows the direction of the normal stress. Vectors pointing in the positive normal direction indicates tension. Similarly, vectors pointing in the negative normal direction (opposite to the surface normal) indicates compression.

User Interface

Several user interface enhancements improve user experience.

List Shear Stress Values for Beams



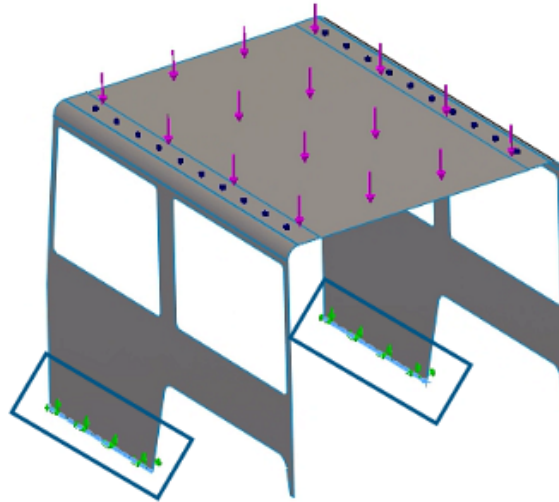
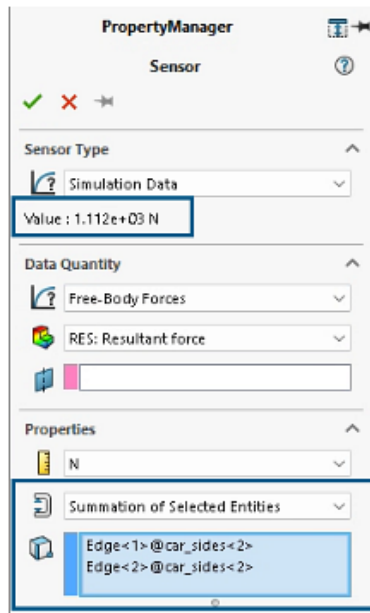
The List Beam Forces command has been enhanced to include dedicated columns for shear stresses acting on the beam.

In earlier versions of SOLIDWORKS Simulation, shear stresses could only be visualized in plots and were not available in the list output. Now, the list includes shear stress values for both local directions.

To list beam stresses in a tabular format:

1. In the simulation study tree, right-click **Results** and select **List Beam Forces**.
2. From the **List** options, select **Stresses** and click **OK** .

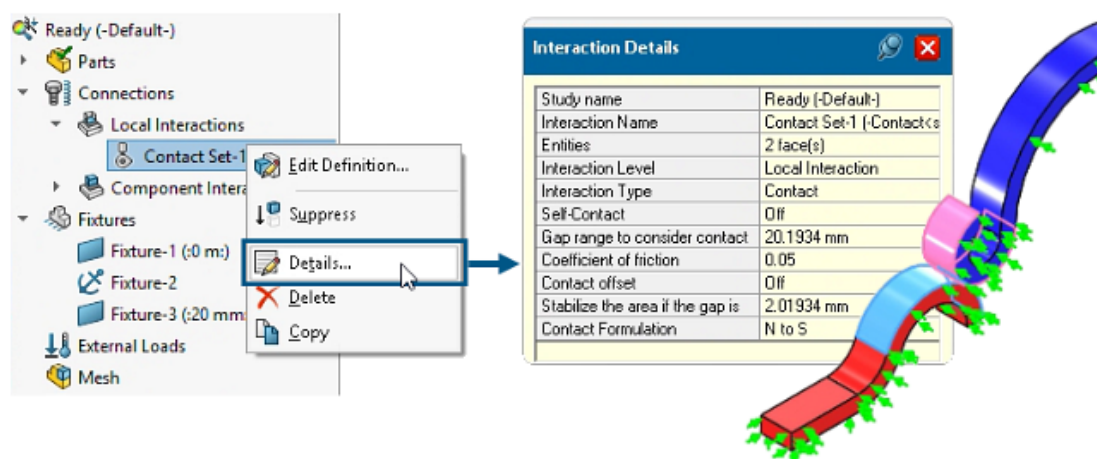
New Sensor Criterion for Summing Selected Entities



A new sensor criterion called **Summation of Selected Entities** is now available and allows you to calculate the sum of simulation data quantities on the selected entities.

In earlier versions of SOLIDWORKS Simulation, you had to extract results values individually and manually compute the summation, which was time-consuming. Now, selecting the required entities automatically displays the summation value directly in the sensor PropertyManager.

Viewing Interaction Details



You can now view information about defined interactions using the **Interaction Details** dialog box.

In the simulation study tree right-click an interaction that you have already defined and select **Details** . The **Interaction Details** dialog box appears and shows all the information about the defined interaction.

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SOLIDWORKS Visualize

SOLIDWORKS® Visualize is a separately purchased product.

Using VDB Textures in Volume Rendering

You can import OpenVDB (.vdb) files and file sequences to create realistic volumetric effects such as smoke, clouds, fog, and atmospheric effects.

To use VDB textures in volume rendering:

1. Click **Project** > **Models** > **New Volume** and select an option:

Uniform creates a volume object without importing a file. See [Creating Volume Objects](#).

From File (.vdb)

Imports a file to render a volume object with heterogeneous (spatially varying) volume density. Useful for realistic volumetric effects like clouds, smoke, fog, and fire.



From File Sequence (.vdb)

Imports a file sequence to render a volume object with heterogeneous (spatially varying) volume density that can vary from frame to frame in an animation sequence. Useful for realistic volumetric effects in motion studies.



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SOLIDWORKS CAM

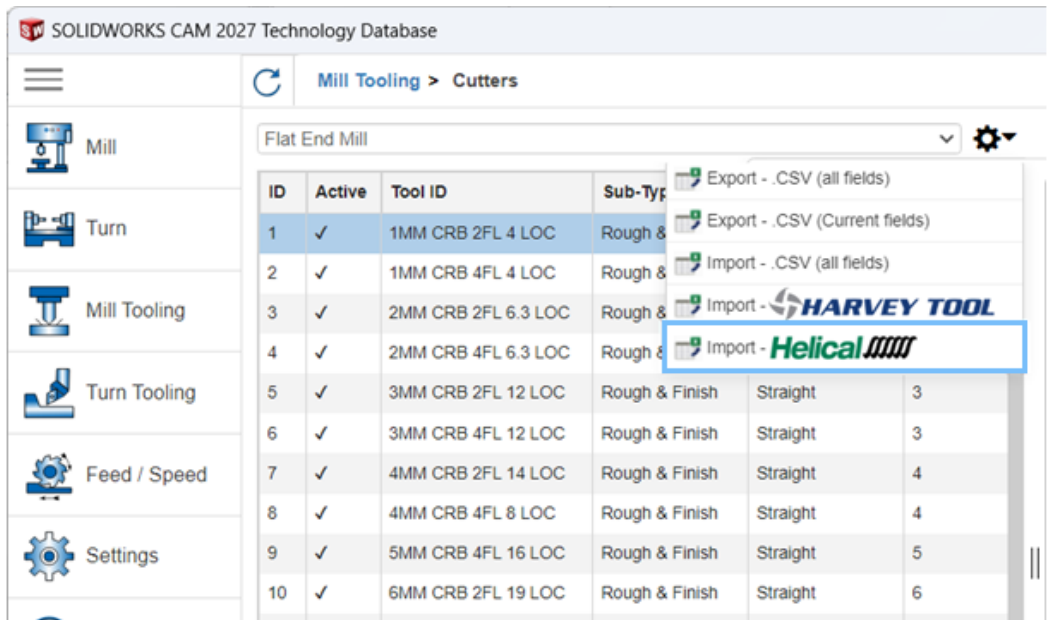
This chapter includes the following topics:

- **Tool Libraries for Helical Tools**
- **Tool Libraries for Harvey Tools**
- **Z Limits for Turn Thread Operations**
- **Z Length for Lead In/Out for Turn Thread Operations**
- **Expanded Link Operations Dialog Box**
- **Optimize Feedrate for Ramping and Helical Moves**
- **Duplicate Station Numbers in Tool Cribs**
- **Cutting Parameters for Turn Inserts**
- **Empty Tool Stations**
- **Chip Breaking for Turn Operations**
- **Folders in the SOLIDWORKS CAM Trees**

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.

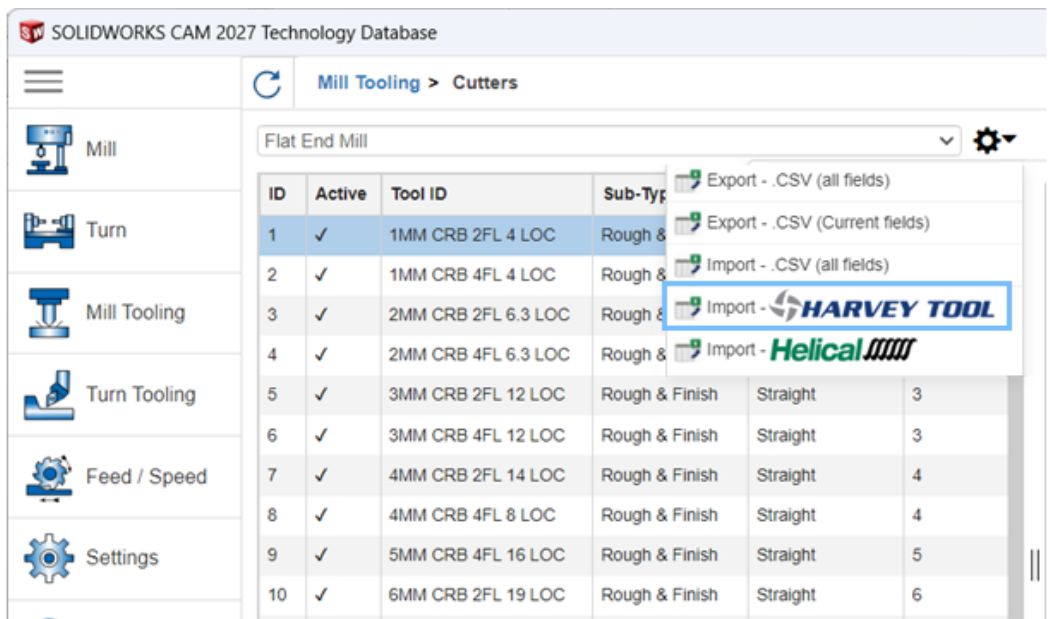
Tool Libraries for Helical Tools



The Technology Database (TechDB) has an updated and expanded library of Helical tools, offering a wider range of tool options for selection and use during machining operations.

Previously, the tool library for Helical tools contained approximately 6,000 tool entries. Now it includes approximately 10,550 tools. This gives you more comprehensive and up-to-date tools to match for machining requirements.

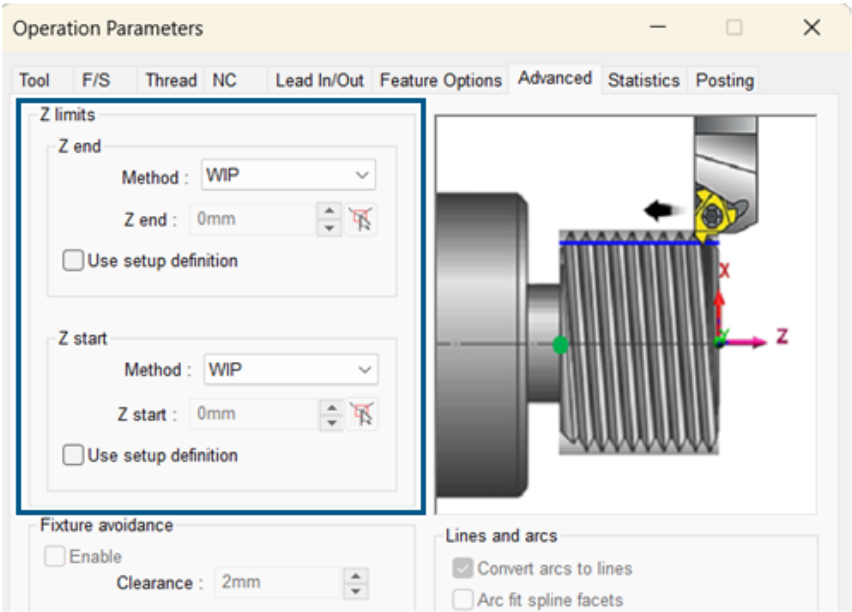
Tool Libraries for Harvey Tools



The TechDB has an updated and expanded library of Harvey tools, offering a wider range of tool options for selection and use during machining operations.

Previously, the tool library for Harvey tools contained approximately 13,000 tool entries. Now it includes approximately 17,485 tools. This gives you more comprehensive and up-to-date tools to match for machining requirements.

Z Limits for Turn Thread Operations



For Turn Thread operations, you can assign **Z start** and **Z end** limits.

The Advanced tab on the Operation Parameters dialog box contains the following parameters for Turn Thread operations.

Z limits

Parameter	Description
Method	• WIP. Calculates the Z end position of the Turn Thread toolpath based on the feature length and the work in process of the stock. • User Defined. Uses the Z end value for limiting the end position of the Turn Thread toolpath.
Z end	Specifies the Z value to limit the end position of the Turn Thread toolpath.

Parameter	Description
Method	• WIP. Calculates the Z start position of the Turn Thread toolpath based on the feature length and the work in process of the stock. • User Defined. Uses the Z start value for determining the start position of the Turn Thread toolpath.
Z start	Specifies the Z value for the start position of the Turn Thread toolpath.
 Pick point	(Unavailable if you select Use Segment Limits.) Opens the Define Point dialog box where you can specify the Z end or Z start values. You can select circular edges, vertices, origins, sketch end points, and coordinate systems.
Use setup definition	Globally defines the Z end and Z start limits. These settings apply automatically to all operations generated for Turn features under a Turn Setup. To use the global definitions, you must select Use setup definition for that specific operation.
 Apply to All	Applies global definitions of a Z end or Z start to all applicable operations under a Turn Setup. This ensures that the Use setup definition option associated with the Z end and Z start parameters under Advanced tab of all applicable Turn operations is checked by default.

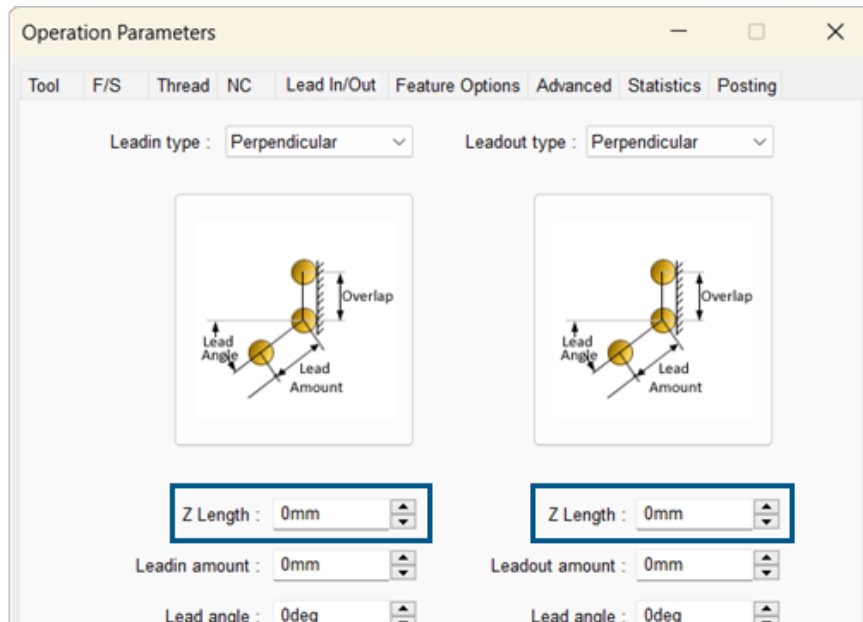
Fixture avoidance

These parameters do not apply to Turn Thread operations.

Lines and arcs

These parameters do not apply to Turn Thread operations.

Z Length for Lead In/Out for Turn Thread Operations



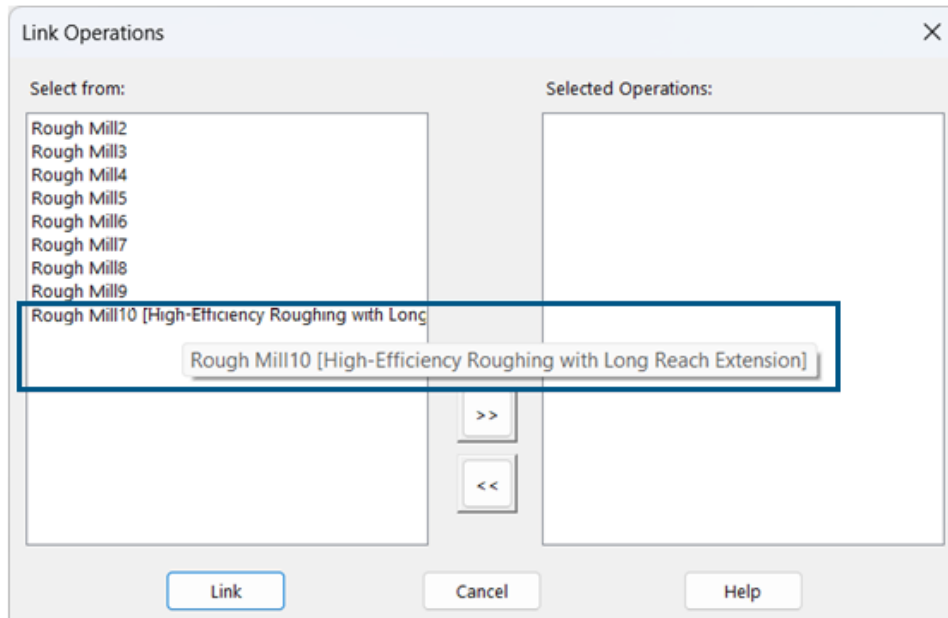
You can specify a **Z Length** parameter for lead ins and lead outs of Turn Thread operations to define the Z travel distance per the thread pitch.

For Turn Thread operations, you need to define the lead in/lead out values in the denominator of the thread pitch value. If you specify the lead in/lead out at an angle, then it is important to define the Z travel distance per the thread pitch.

The Lead In/Out tab of the Operation Parameters dialog box includes the **Z Length** parameter for **Leadin** and **Leadout types** of **Parallel** or **Perpendicular**.

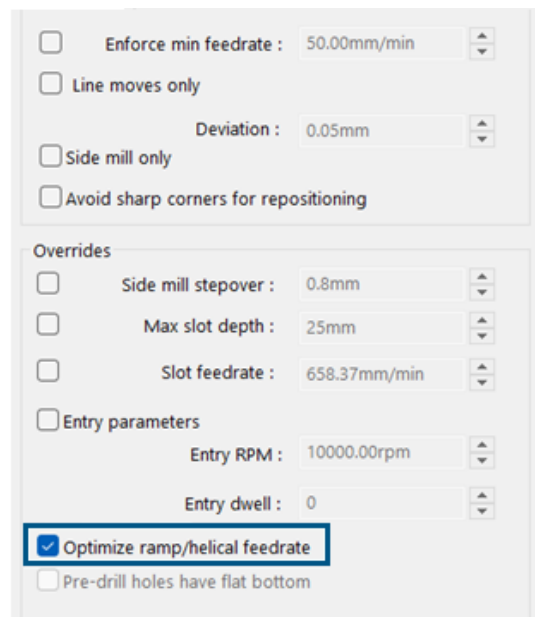
If you modify **Z Length**, the **Leadin** and **Leadout amount** change and vice versa. In both cases, the **Lead angle** does not change.

Expanded Link Operations Dialog Box



The Link Operations dialog box is expanded and includes tooltips, helping you identify and manage operations with long names.

Optimize Feedrate for Ramping and Helical Moves

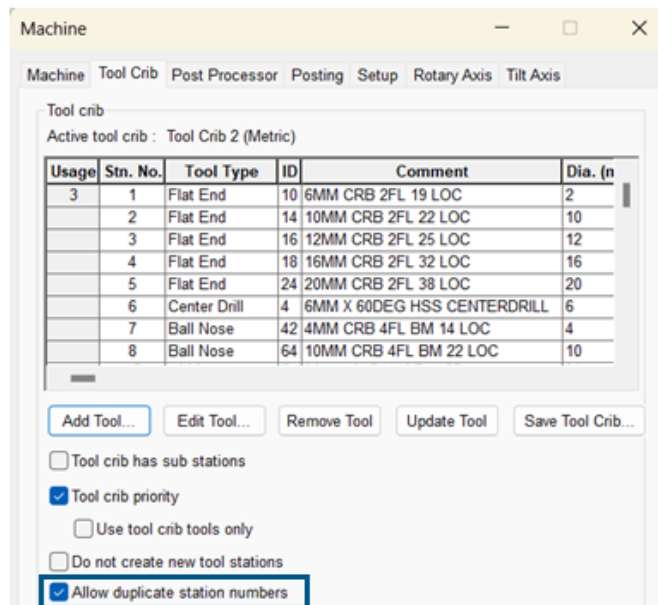


You can specify an option to optimize the feedrates for ramp and helical moves in VoluMill operations.

Previously, you could assign ramp or helical moves for VoluMill operations. However, you could not regulate the feedrate for the ramp and helical moves. This option puts less load on the cutting tool and prevents breakages.

In the VoluMill Settings dialog box, under **Overrides**, select **Optimize ramp/helical feedrate**. When cleared, the leadin feedrate applies to all ramp and helical moves.

Duplicate Station Numbers in Tool Cribbs



You can specify whether to allow duplicate station numbers in tool cribs associated with a selected machine.

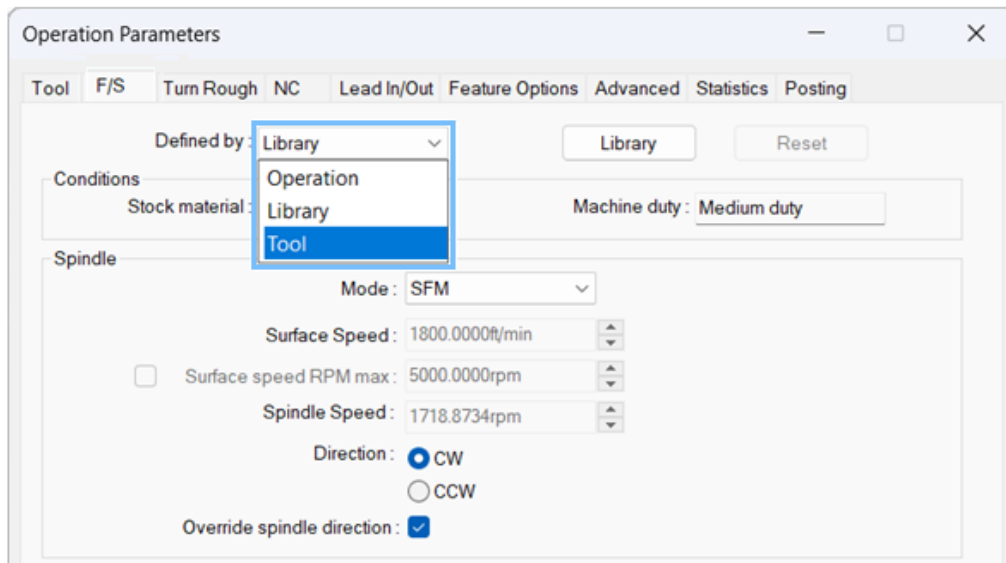
In the Machine dialog box, on the Tool Crib tab, select **Allow duplicate station numbers**. When selected, SOLIDWORKS CAM allows stations with duplicate station numbers, including substations in the tool cribs.

The default setting depends on your selection in the TechDB in:

- **Mill > Machines > Turret.**
- **Turn > Machines > Turret**

For models programmed before SOLIDWORKS CAM 2027, this option is selected.

Cutting Parameters for Turn Inserts



You can define cutting parameters for Turn Inserts based on the material of the machined stock.

For each Turn Insert, you can define one or more stock material groups and cutting parameters based on the stock material to machine. Every stock material group defined for a Turn Insert has a set of cutting parameters associated with it. You can define and edit them in the Cutting Parameters dialog box (in the TechDB and in SOLIDWORKS CAM).

In the Operation Parameters dialog box, on the F/S tab, you can specify **Defined by** as **Tool**. When selected, SOLIDWORKS CAM uses the feed and speed values defined in the Cutting Parameters dialog box. (You can specify the default setting for **Defined by** in the TechDB.)

Cutting Parameters Dialog Box (Turn Inserts)

You can define cutting parameters for all types of Turn Inserts.

In the Operation Parameters dialog box, on the F/S tab, you must specify **Defined by** as **Tool** for Turn operations.

To open this dialog box:

1. Do one of the following:
 - In the Operation Parameters dialog box, on the Tool tab, under **Turn Insert**, click **Cutting parameters**.
 - In the Edit Tool Parameters dialog box, on the Diamond Insert tab, under **Properties**, click **Cutting parameters**.

Stock Material

Stock Material Group	Specifies the stock material groups defined for the selected Tool in the TechDB.
Associate with stock material	Automatically selects the stock material group so it is in sync with the Material specified in the Stock Manager dialog box. The cutting parameters associated with the selected stock material group populate within their respective fields in the Cutting Parameters dialog box. Parameters for Surface Speed, Spindle Speed, and Feedrates are unavailable once populated.

Surface Speed

Specifies the SMM (surface meter per minute)/SFM (surface feet per minute) values to the active Turn Insert assigned for the stock material.

Spindle Speed

Displays a calculated value based on the RPM and the tool diameter.

Feedrates

FPR (feed per revolution)	Displays a calculated value based on the XY feedrate, the RPM, and the number of flutes of the selected tool.
FPM (feedrate per minute)	(For Probe Tools only.) Assigns the feedrate for the Probe movement of a Probing operation.

Parameters

Coolant	Specifies the coolant type.
----------------	-----------------------------

Defining Default Cutting Parameters for Turn Inserts in the TechDB

In the TechDB, you can define the default cutting parameter values to populate the Cutting Parameters dialog box in SOLIDWORKS CAM. These parameters apply when you specify the Turn Insert to machine a stock material.

Cutting Parameters

Stock material group name
Default

Add Delete

Stock material group name (ID: 0)
Default

☒ Surface speed
SFM: 100
Max rpm: 5000 rpm

☐ Spindle Speed
Spindle Speed: 5000 rpm

☒ Feed Rate
FPR: 0.0039 in
FPM: 3.93701 in/min

☐ Parameters
Coolant type: Flood

In the TechDB, click **Turn Tooling > Cutting Parameters**.

Empty Tool Stations

SOLIDWORKS CAM 2027 Technology Database

Mill > Tool Crib

New Copy Edit Delete

Select Tool Crib: Tool Crib 2 (Inch) Total stations: 20 Define mill tools using tool assembly: ☐

ID	Stn. No.	Sub. No.	Station...	Comb ID	Tool Type	Tool ID	Diams	
1	1	0	-	0	Flat End Mill	47	1	0
2	2	0	-	0	Flat End Mill	61	2	0
3	3	0	-	0	Flat End Mill	69	3	0
4	4	0	-	0	Flat End Mill	76	4	0
5	5	0	-	0	Center Drill	5	5	0
6	6	0	-	0	Counter Sink	1	6	0
7	7	0	-	0	Ball Nose Mill	101	7	0
8	8	0	-	0	Ball Nose Mill	115	8	0
9	9	0	-	0	Ball Nose Mill	121	9	0
10	10	0	-	0	Bore	2	10	0
22	11	0	-	0	Taper Ball Nose Mill	59	11	0
33	12	0	-	0	Face Mill	2	1	0
41	17	0	-	N.A.	Probe Tools	7	1	0
47	19	0	-	N.A.	None	-1	1	0

Add Save Copy Delete

Empty Station

Bore

Drill

Flat End Mill

Ball Nose Mill

Hog Nose Mill

Ream

Tap-Cutting

Tap - Rolling

Corner Round

Center Drill

Taper Flat End Mill

Taper Ball Nose Mill

Taper Hog Nose Mill

Counter Sink

Single-point Thread Mill

Multi-point Thread Mill

There are many options that you can specify to use empty tool stations in a tool crib.

Defining Empty Tool Stations

In the SOLIDWORKS CAM Tools Tree, right-click and select **Add Station**. The empty tool station displays as the last entry in the active tool crib in the SOLIDWORKS CAM Tools Tree.

In the TechDB, on the Tool Cribs form, click **Add > Empty Station**. The empty tool station displays as the last entry in the selected tool crib.

Removing Tools from Tool Station Nodes

You can remove tools that are unused in any operations. This removes the tool from the selected tool station node. The tool station node displays as empty in the SOLIDWORKS CAM Tools Tree and in the Tool Crib tab in the Operation Parameters dialog box.

In the SOLIDWORKS CAM Tools Tree, right-click a tool node and select **Remove Tool**.

In the TechDB, on the Tool Cribs form, in **Type of Tool**, select **None**. This specifies the **Tool ID** parameter as **-1**.

Creating New Tool Cribs with Empty Tool Stations

You can specify all tool stations in newly created tool cribs as empty.

In the TechDB, on the Tool Cribs form, click **New** and select **Create all stations as empty**. Enter a value for **Total stations** to define the number of empty tool stations to create.

Preventing New Tool Stations from Being Added

You can prevent SOLIDWORKS CAM from adding new tool stations to the active tool crib.

In the Machine dialog box, on the Tool Crib tab, select **Do not create new tool stations**. This option is unavailable when you select **Tool crib priority** and **Use tool crib tools only**.

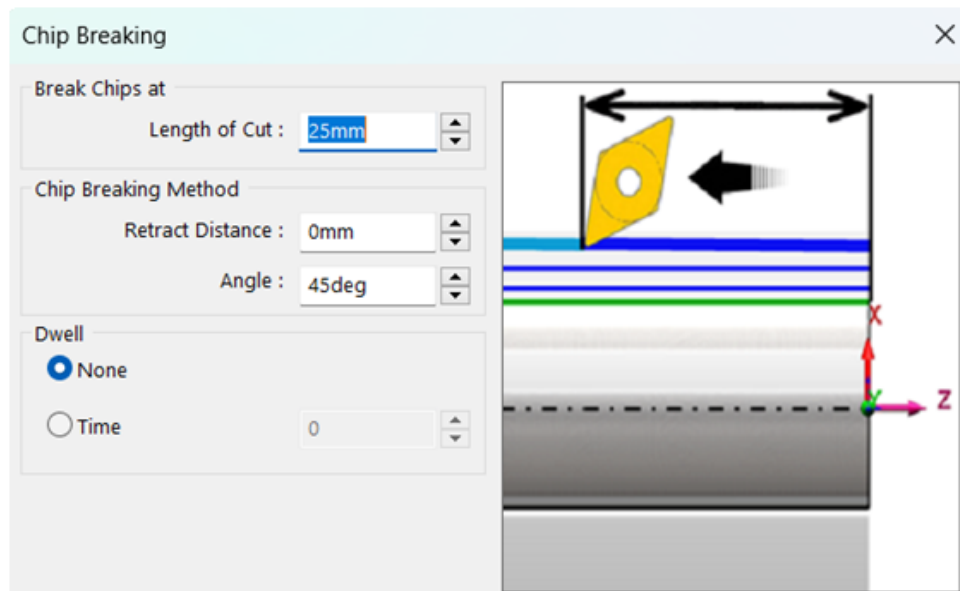
In the TechDB, on the Machines form, on the Turret tab, select **Do not create new tool stations**.

Adding New Tools to the Tool Crib during Generate Operation Plan

When you use the **Generate Operation Plan** tool, the app adds new tools to the tool crib from the tool library if they are required for machining the operations.

For Mill and Turn tool cribs, the app assigns the new tools to the empty tool stations if they are compatible with the tool type of the empty tool station.

Chip Breaking for Turn Operations



You can specify an option for chip breaking functionality for Turn operations to shorten chips generated during the machining process.

During the Turn machining process, the machining of metals can result in long chips which can reduce efficiency and tool life. The Chip Breaking option breaks these chips to smaller, manageable pieces. This option modifies the resulting toolpath to form only short chips.

To specify chip breaking for turn operations:

1. Navigate to one of the following tabs:
 - Turn Rough
 - Turn Finish
 - Bore Rough
 - Bore Finish
 - Face Rough
 - Face Finish
2. Under **Cut Type**, select the check box next to **Chip Breaking**.
For Turn Rough, Bore Rough, and Face Rough operations, the check box is available if you cleared **Canned Cycle Output**.
3. Click **Chip Breaking**.
4. In the dialog box, specify options and click **OK**.

Chip Breaking Dialog Box

You can specify the default values for this dialog box in the TechDB. In the corresponding Operation forms supported for Turn operations, under **Cut type**, select **Chip breaking** and specify values.

Break Chips at

Length of Cut	Specifies the distance along the cut at which a chip breaking point occurs in the toolpath. (The distance between consecutive chip breaking points in the toolpath.) The value must be greater than zero.
----------------------	---

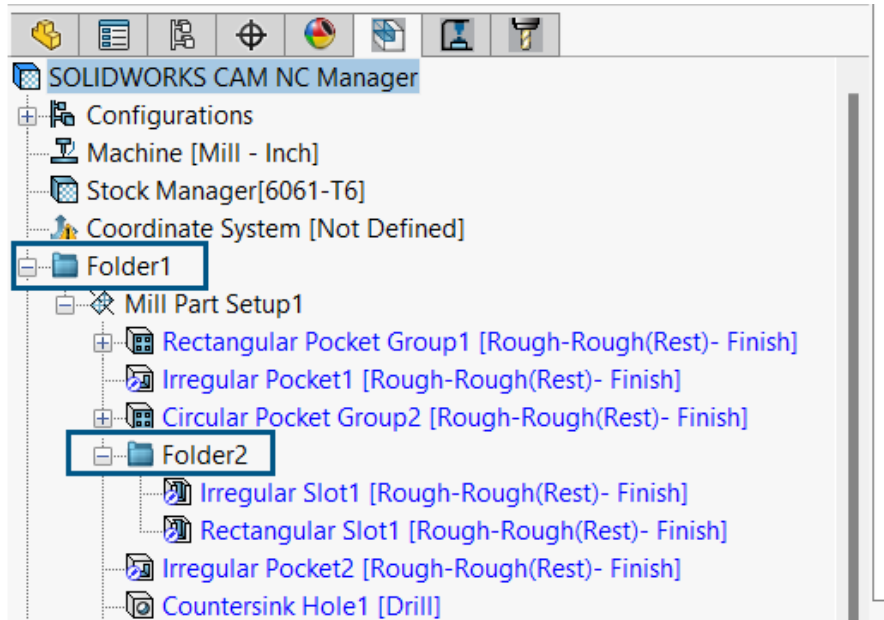
Chip Breaking Method

Retract Distance	Specifies the distance that the tool must traverse in the reverse direction from the chip break point. The value can be zero or greater.
Angle	Measures the angle for the Retract Distance with respect to the segment of the toolpath where the chip breaking point is located. You can specify 0° to 90°.

Dwell

None	Specifies that the tool does not dwell after traversing the retract distance. It immediately restarts the cutting moves on the toolpath.
Time	Specifies the seconds that the tool dwells after traversing the retract distance. The tool remains at the end of the retract distance for the time specified.

Folders in the SOLIDWORKS CAM Trees



You can create folders in various trees in the app. This eases navigation when working with complex parts that contain multiple features, operations, or tools within the trees. Grouping features, operations, part setups, and tools with similar characteristics together enhances visual identification and management in the trees.

You can create custom folders and add items to them in the Feature, Operation, and Tools trees. This helps you group and manage features, operations, part setups, and tools.

Right-click a feature, operation, part setup, or tool node in a tree and select **Folder >**:

- **Create new folder.** Creates a new folder above the selected node.
- **Add to new folder.** Creates a new folder above the selected node and moves the selected node in the folder.

In the Feature tree, you can:

- Create folders at the part setup and feature node levels
- Group part setups by dragging
- Group features with the same machining direction by dragging
- Relocate user-defined folders by dragging

In the Operation tree, you can:

- Create folders at the part setup and operation node levels
- Group part setups by dragging
- Group operations with the same machining direction by dragging
- Relocate user-defined folders by dragging

In the Tools tree, you can:

- Create folders at the tool node level
- Group tool nodes by dragging
- Relocate user-defined folders by dragging

CircuitWorks

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

Building Board Models from ECAD Files

Performance is noticeably faster when building board models from ECAD files that contain a high number of components and high numbers of defined areas, such as keep-in and keep-out.

18

SOLIDWORKS Electrical

This chapter includes the following topics:

- **Defining Wire Connections in Line Diagrams**
- **Export Electrical Projects to ControlBuild**
- **Obsolete Flag for ECP Data**

SOLIDWORKS® Electrical is a separately purchased product.

Defining Wire Connections in Line Diagrams

You can now fully define harness connections directly within line diagrams without switching to schematic views. You can use the Detailed connection dialog box (formerly Detailed Cabling) to support both individual wire-based connections and traditional cables.

This update allows you to manage pure wires, mixed cable and wire configurations, or complete harnesses using standard schematic workflows.

The Detailed connection dialog box now includes full harness information, enabling you to build a complete harness ready for 3D modeling directly from the line diagram-without compromising performance.

Previously, the Detailed Cabling dialog box supported only cables and cable cores. You had to use schematic views to define individual wire connections and integrate them into a harness.

To access the Detailed connection dialog box:

1. Open the Line Diagram workspace.
2. Right-click a connection line and select **Detailed connection**.

You can use the options in the **Wire Management**, **Harness**, and **Cable Management** groups to modify a harness.

Export Electrical Projects to ControlBuild

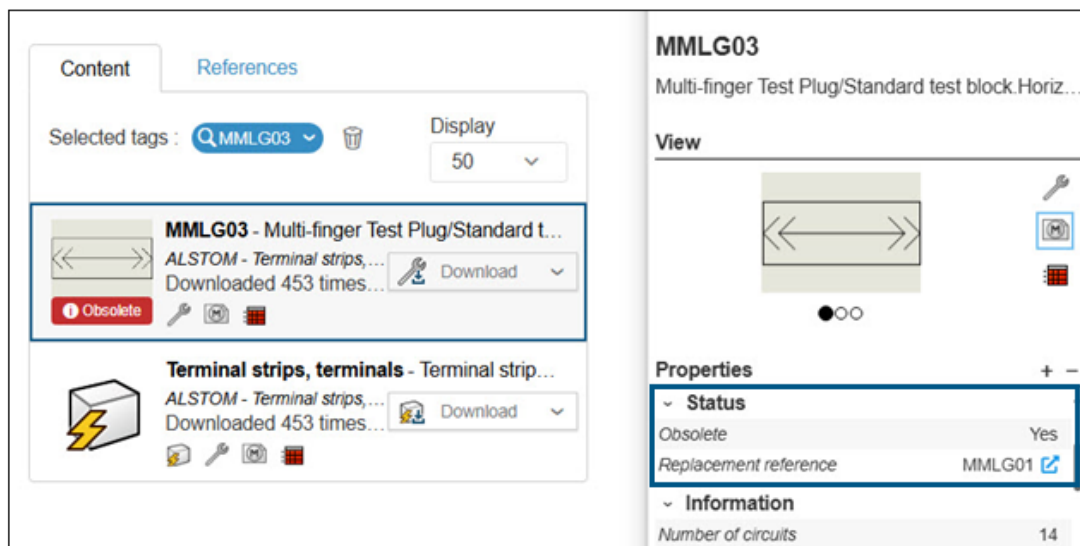
SOLIDWORKS Electrical Schematic now integrates with ControlBuild, a development tool for designing, simulating, and testing control systems. This allows you to export electrical schematics directly to ControlBuild for advanced validation, simulation, and error detection, streamlining the development process for industrial and railway domains.

Key features:

- **Seamless export:** Schematic files (including schemes and mixed schemes) are exported file-by-file to ControlBuild for logic validation, dynamic simulation, and error detection.
- **Component mapping:** A default-mapping configuration automatically associates electrical components with their ControlBuild equivalents. You can edit this configuration or create new ones for future exports, ensuring flexibility across different project setups.
- **Optimized workflow:** Once exported to ControlBuild for testing, any issues identified are corrected directly in SOLIDWORKS Electrical Schematic, keeping the schematic as the single source of truth and reducing overall project development time.

Benefit: Traditionally, electrical schematics are validated late in the process, during wiring or commissioning, when changes are expensive. With ControlBuild simulation, teams can identify missing components, verify control logic and timing, and uncover hard-to-detect design flaws (misplaced contactors, inadequate motor protection, or overly disruptive faults) before physical wiring begins. This ensures a safer, more reliable design that arrives on the shop floor prevalidated.

Obsolete Flag for ECP Data



In the Electrical Content Portal, the **Obsolete Flag** feature is introduced to identify outdated or incorrect data for manufacturer parts and cable references. This helps prevent the use of outdated or incorrect components in designs while maintaining traceability.

Enhancements:

- **Replacement Reference:** Displays a link to navigate to the approved alternative reference page, if the replacement parts exist in the ECP.
- **Status** section: Displays the obsolete status in the **Properties Panel**.
- **Obsolete** badge:
 - Visible at the bottom of the manufacturer part or cable reference thumbnail.

- Visible at the bottom of the electrical package thumbnail when all the content is obsolete.
- Visible at the end of the suggestions displayed when search is entered in the search bar.
- **Filtering** options: Use the **Show Obsolete** tag available under the Tags filter to toggle visibility.

Obsolete items are hidden by default in catalogs and can be automatically replaced during imports, helping maintain design accuracy while preserving traceability.

19

DraftSight

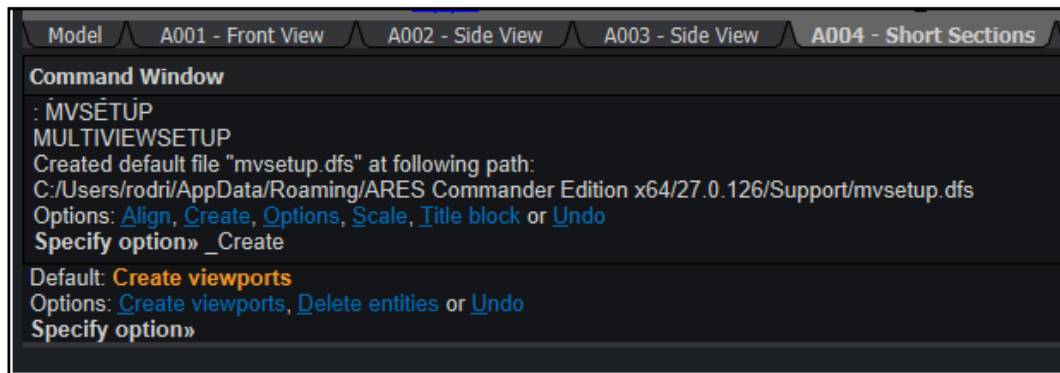
This chapter includes the following topics:

- **CAD Features**
- **BIM Features**
- **Selection and Copy Commands**
- **Zoom and Pan Commands**
- **Redraw-heavy Operations**
- **EXPERT Variable Value**
- **DraftSight Customer Experience Improvement Program**
- **Dimension Overdrawn Dialog Box (DraftSight Mechanical Only)**
- **Fits List Command (DraftSight Mechanical Only)**
- **Power Dimensioning Dialog Box (DraftSight Mechanical only)**
- **Support for Drawing 2022 Files (DraftSight Mechanical Only)**

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

CAD Features

MVSETUP Command



You can use the `MVSETUP` command to define drawing units, set scales, insert title blocks, and create viewports through a guided setup.

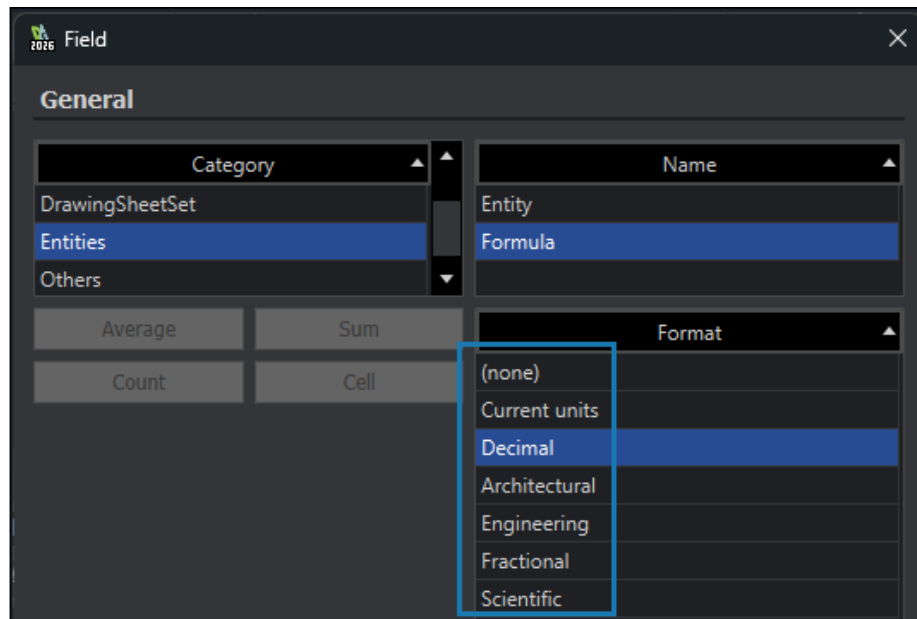
This command streamlines the process of specifying up a drawing for plotting and results in consistent and professional layouts.

You can use the command in a model space to create the drawing by specifying units and a scale, but it does not create viewports or title blocks. When you use the command in layouts, it provides a setup that includes sheet size selection, viewport creation, viewport scaling, and view alignment.

To access this command:

- In the command window, type `MVSETUP`.
- On the ribbon, click **View** > **Viewports tools** > **Multi View setup**.

Formulas and Position in FIELD

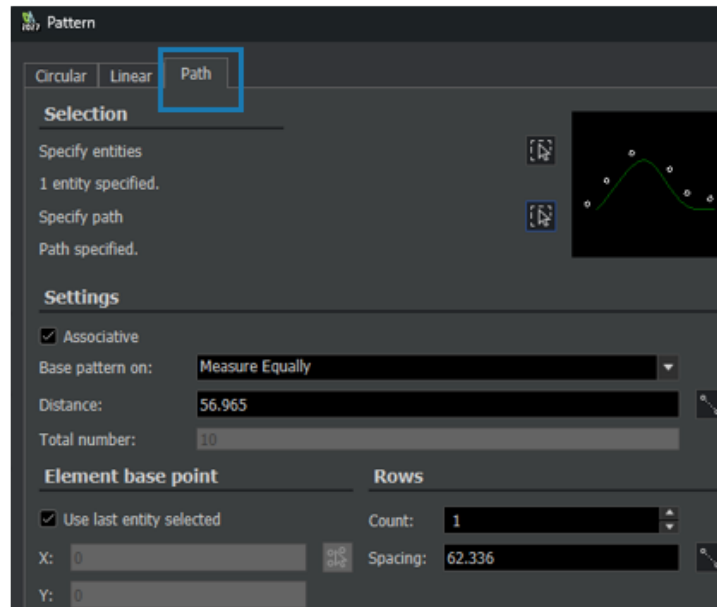


You can use the **Formula** option in the `FIELD` command to insert field values into mathematical expressions. You can also display the coordinates of points in drawings.

The formula lets you refer to the previously defined field values in custom calculations. Formulas dynamically update based on changes to other fields, offering powerful annotations in drawings.

Displaying coordinates of points in drawings shows the exact position of any point within the CAD environment, improves accuracy, and simplifies tasks that require positional reference.

Path-based Associative Arrays



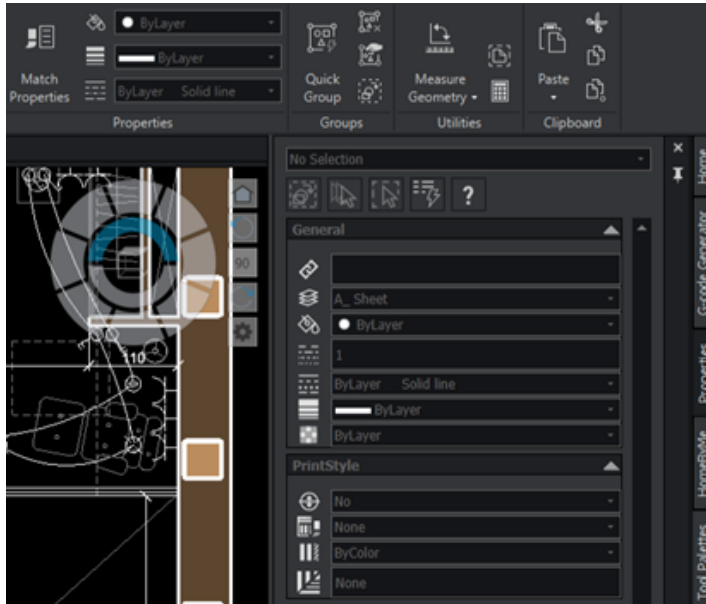
You can use the **Path** option in `PATTERN` to distribute objects along a straight or curved path while maintaining full associativity.

To access this command:

- In the command window, type `PATTERN`.
- On the ribbon, click **Home** > **Modify** > **Pattern**.
- On the menu, click **Modify** > **Pattern**.

Previously, you had only rectangular and polar arrangements. You can place objects along arcs, poly lines, or splines, and adjust the array by modifying the path or the array parameters.

PROPERTYPainter Command



The PROPERTYPainter command lets you copy properties from one drawing to another open drawing and supports properties.

To access this command:

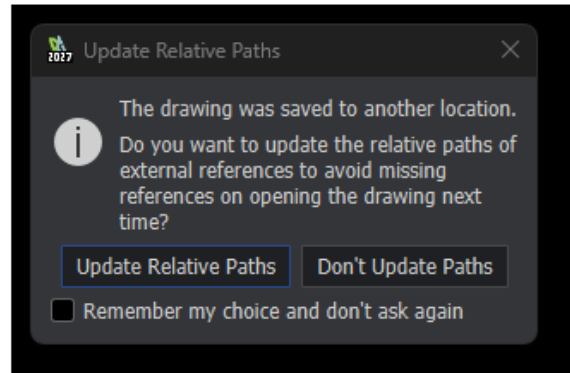
- In the command window, type PROPERTYPainter.
- On the ribbon, click **Home > Properties > Properties Painter**.
- On the menu, click **Modify > Properties Painter**.

Previously, the PROPERTYPainter command let you copy properties within the same drawing.

You can copy:

- Dimension styles
- Text styles
- Plot styles
- Materials
- Transparency
- Hatch properties

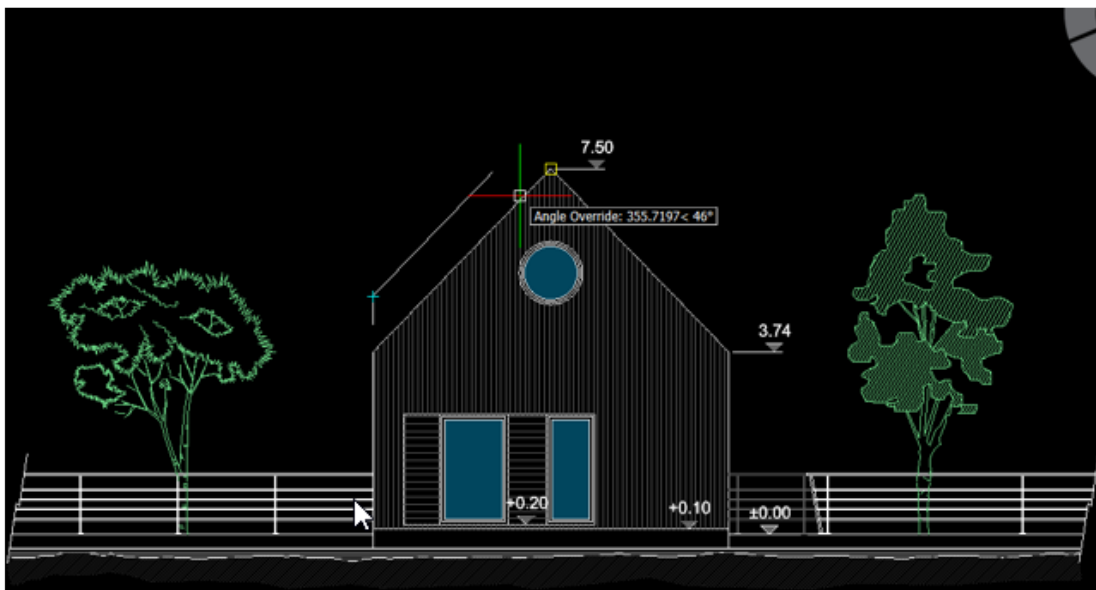
Saving a File with External References



When you save a drawing file with external references to a new location, DraftSight displays a warning about loading references.

The warning prevents broken links and provides a smooth workflow.

Angle Input Override



Angle Input Override lets you lock an angle when entering points using the polar coordinate system.

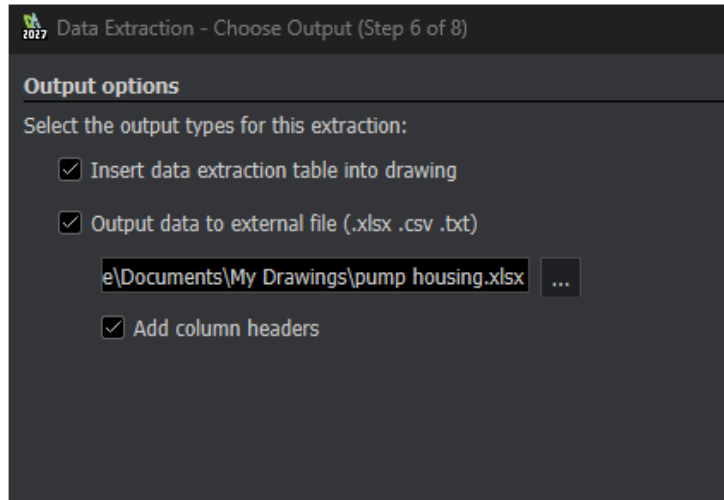
It improves control and precision in commands that require multiple points.

EXPERT Variable Value

When you specify the **EXPERT** system variable to 6 and open a file in read-only mode, DraftSight suppresses the read-only notification.

This setting is temporary and **EXPERT** resets to its default value in new sessions.

XLSX Export Support for Data Extraction



You can use the `DATAEXTRACTION` command to extract specific information from drawing entities and use it to create schedules, parts lists, or bills of materials.

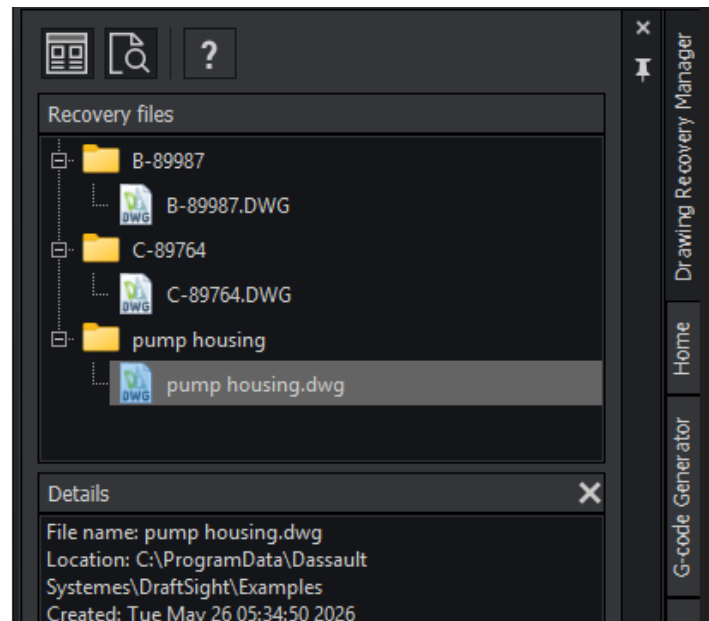
To access this command:

- In the command window, type `DATAEXTRACTION`.
- On the ribbon, click **Insert** > **Data** > **Data Extraction**.
- On the menu, click **Insert** > **Data Extraction**.

You can extract the data and export it to the `XLSX` format so that it is easy to open, review, format, and share the extracted drawing data.

Previously, you could export the extracted data to `CSV` or `TXT` files.

DRAWINGRECOVERY Command



The `DRAWINGRECOVERY` command opens the Drawing Recovery Manager palette and lets you recover drawing files if the software terminates unexpectedly.

To access this command:

- In the command window, type `DRAWINGRECOVERY`.
- On the ribbon, click **Manage** > **Drawing Recovery Manager**.
- On the menu, click **File** > **Drawing Recovery Manager**.

If the Drawing Recovery Manager palette does not appear after software termination, you can start it by entering `DRAWINGRECOVERY` in the command window.

Smart Calculator Palette



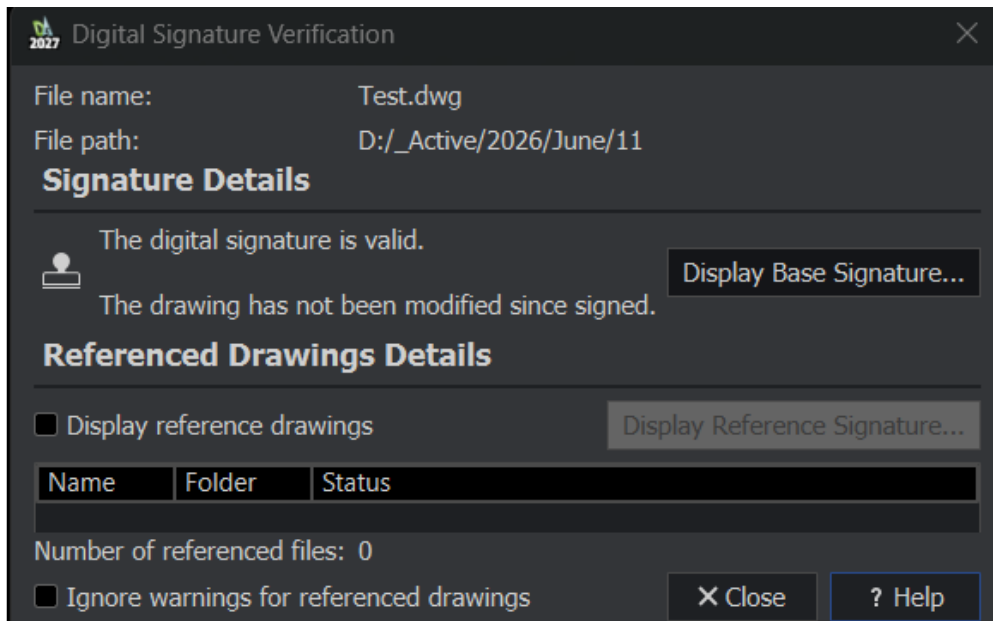
The smart calculator palette opens in a side palette and lets you calculate without leaving the drawing workspace.

To access this command:

- In the command window, type `SMARTCALCULATOR`.
- On the ribbon, click **Manage > Utilities > Smart Calculator**.
- On the menu, click **Tools > Smart Calculator**.

This makes it easier to calculate values, verify measurements, and continue editing the drawing at the same time.

Digital Signature for Drawings



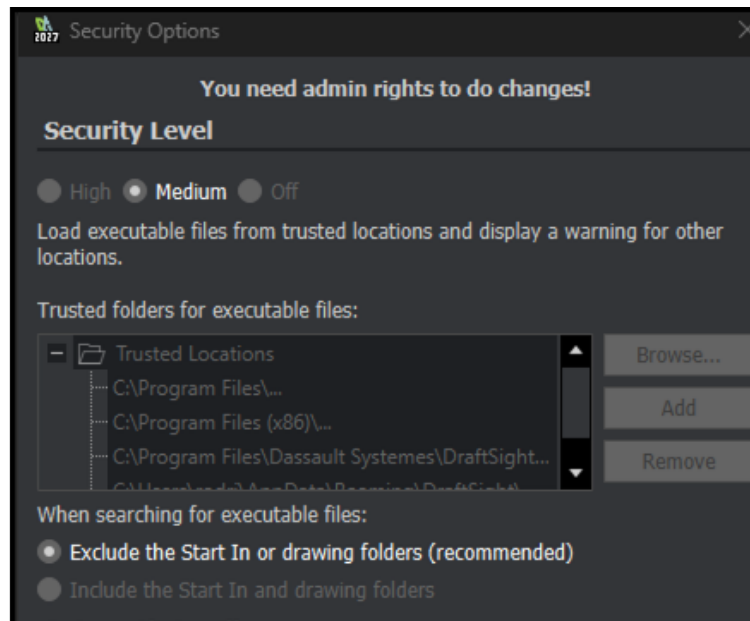
You can use a digital signature to prove the identity, intent, and integrity of electronic documents.

It ensures that the file remains unchanged after you sign it.

To access this command:

- In the command window, type `SIGNFILE`.
- On the ribbon, click **Manage** > **Digital Signature** > **Sign File**.
- On the menu, click **Tools** > **Digital Signature** > **Sign File**.

SAFETYOPTIONS Command



You can use the `SAFETYOPTIONS` command to open the Safety Options dialog box that lets you control the safety of executable files that are at risk.

In the command window, type `SAFETYOPTIONS`.

The settings restrict specific types of executable files from running and protect the software and the computer. You can also define a safe location to save the files.

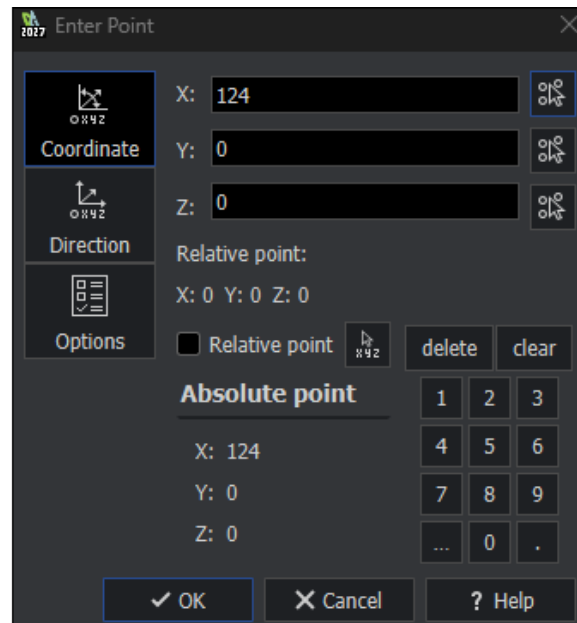
EXPORTSTL Command

The `EXPORTSTL` command lets you save 3D solid objects as `STL` files.

In the command window, type `EXPORTSTL`.

This helps you create models for 3D printing, prototyping, and sharing with applications that support the stereo lithography format.

ENTERPOINT Command



You can use the `ENTERPOINT` command to enter drawing coordinates to place a point.

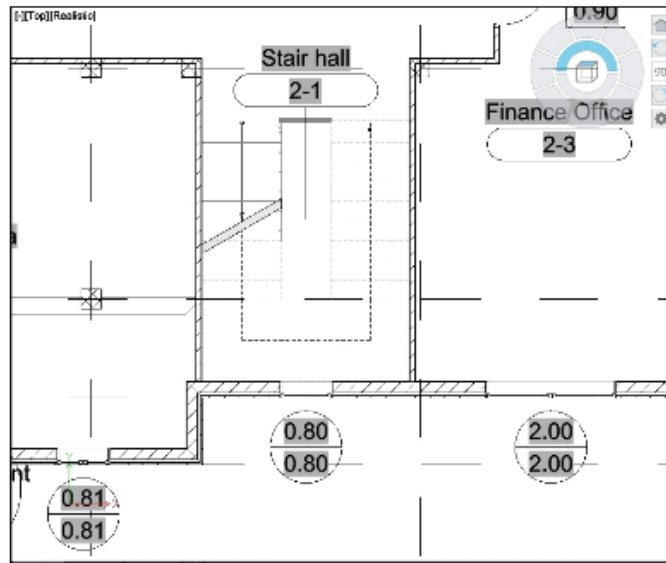
To access this command:

- In the command window, type `ENTERPOINT`.
- On the ribbon, click **Manage > Utilities > Enter Point**.
- On the menu, click **Tools > Enter Point**.

The command opens a dialog box with a numeric keypad for coordinate input, options to define the point relative to another point, and shows the equivalent polar coordinate. This helps you place points accurately and efficiently during drafting.

BIM Features

Stairs 2D Representation in IFC Files



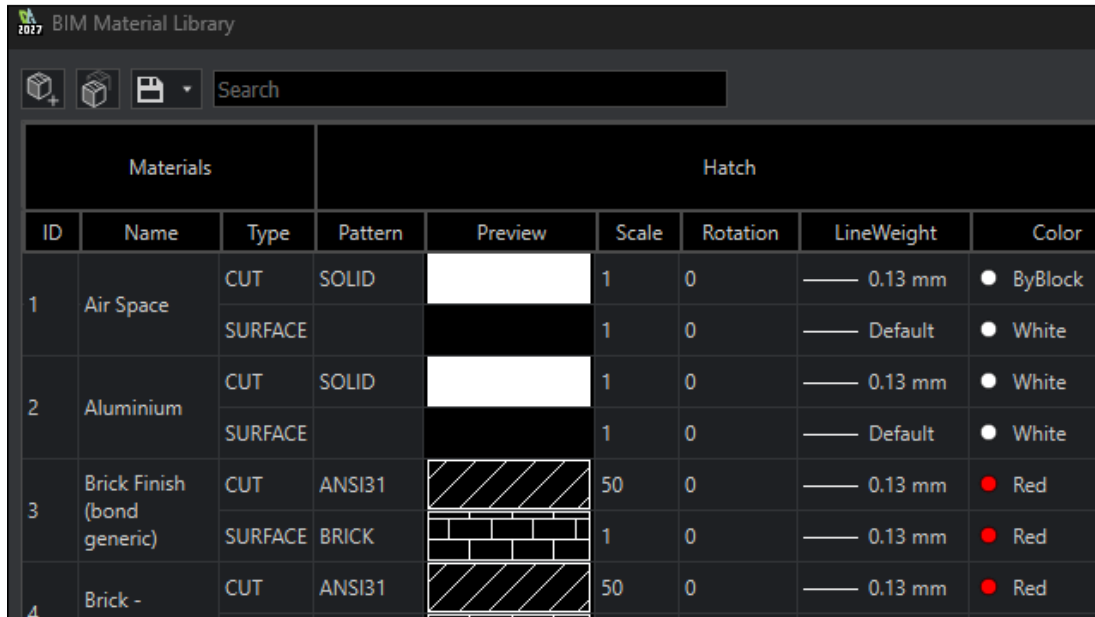
You can use the `BIMSECTION` command for IFC models for 2D representation of stairs.

To access this command:

- In the command window, type `BIMSECTION`.
- On the ribbon, click **BIM > Drawings > Plan**.
- On the menu, click **BIM > Drawings > Plan**.

2D representation provides clearer drawings and better compliance.

BIM Material Library Editor



Materials			Hatch					
ID	Name	Type	Pattern	Preview	Scale	Rotation	LineWeight	Color
1	Air Space	CUT	SOLID		1	0	0.13 mm	ByBlock
		SURFACE			1	0	Default	White
2	Aluminium	CUT	SOLID		1	0	0.13 mm	White
		SURFACE			1	0	Default	White
3	Brick Finish (bond generic)	CUT	ANSI31		50	0	0.13 mm	Red
		SURFACE	BRICK		1	0	0.13 mm	Red
4	Brick -	CUT	ANSI31		50	0	0.13 mm	Red

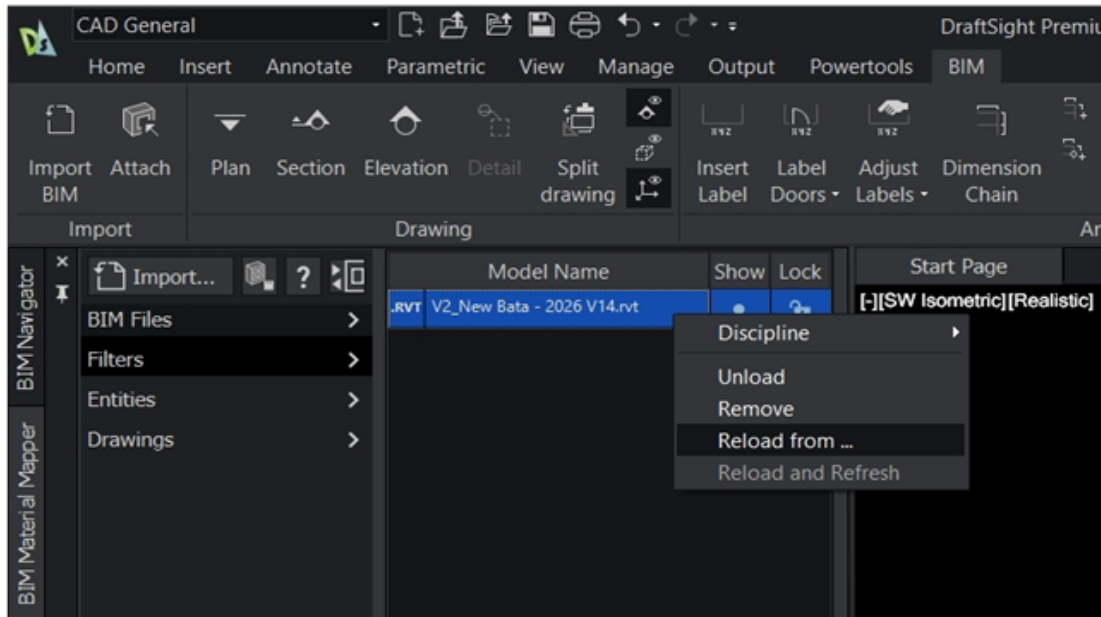
You can use the `BIMMATERIAL` command to customize the materials library that provides you with greater control over designs.

To access this command:

- In the command window, type `BIMMATERIAL`.
- On the ribbon, click **BIM** > **Manage** > **Material Library**.
- On the menu, click **BIM** > **Manage** > **Material Library**.

You can modify colors, hatch patterns, boundary edges, and other properties for cut elements and projected elements. This supports BIM integration and helps you produce professional and precise documentation.

Optimizing RELOAD-FROM



You can use the **RELOAD-FROM** command to reload geometry of BIM drawings.

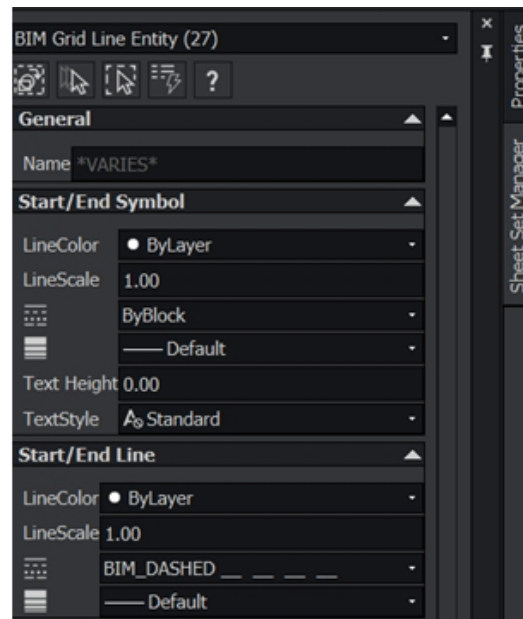
To access this command:

- In the command window, type `BIMNAVIGATOR`.
- On the ribbon, click **BIM** > **Palettes** > **Navigator**.
- On the menu, click **BIM** > **Navigator**.

The reload occurs while retaining any additional entities (such as annotations, dimensions, or title blocks) that added to the original geometry extracted from the BIM model.

Labels, dimensions, and room information adjust so that the added or deleted entities, or modified room layouts, reflect consistently in drawings.

Grid Lines for IFC Files



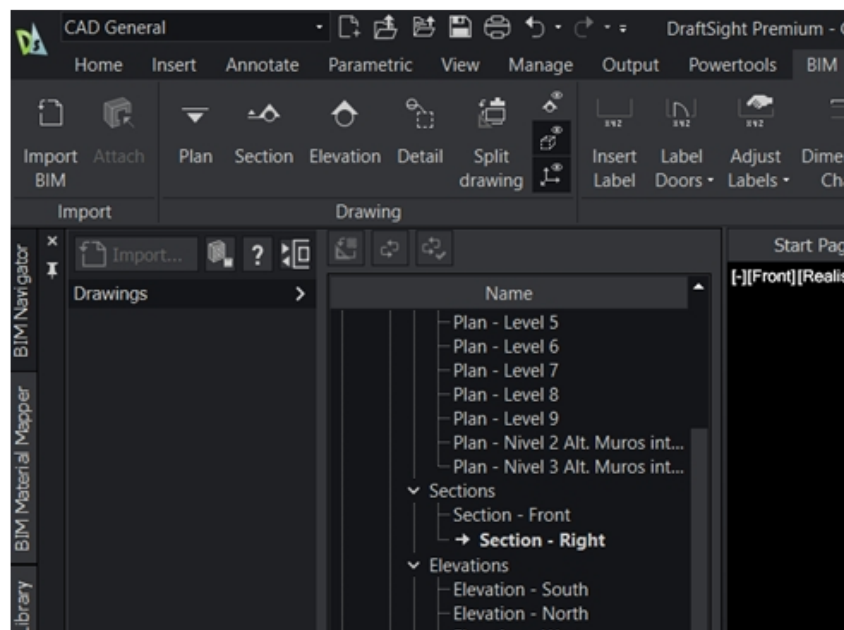
You can use the `BIMIMPORT` command for grid lines of IFC models.

To access this command:

- In the command window, type `BIMIMPORT`.
- On the ribbon, click **BIM** > **Import** > **Import BIM**.
- On the menu, click **BIM** > **BIM Navigator**.

It reads and displays the grid data and enhances drawing quality and details.

References in BIM Drawings



BIM drawings support external and internal references for detail drawings in large or multibuilding projects.

To access this command:

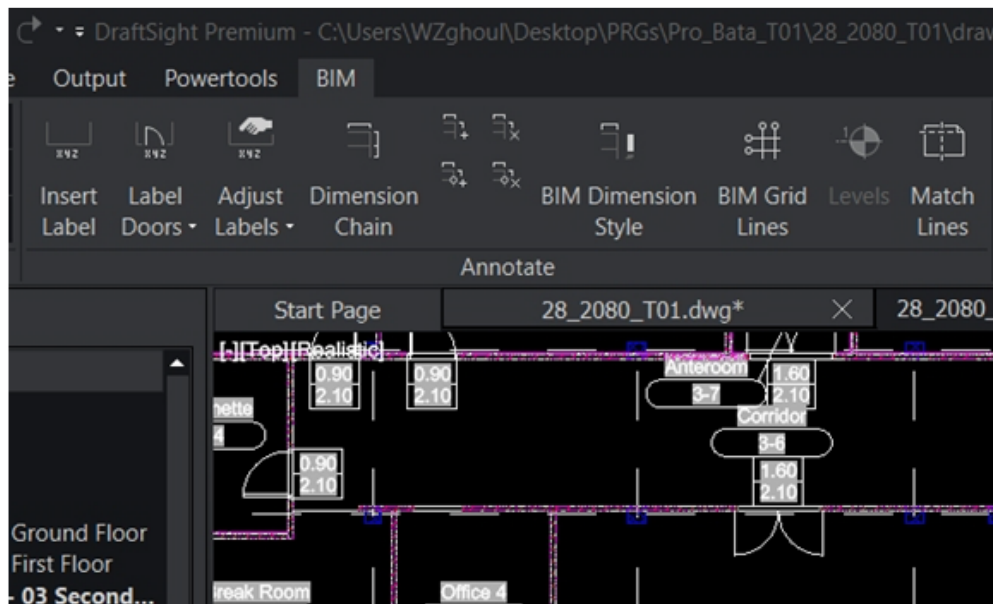
- In the command window, type `BIMATTACH`.
- On the ribbon, click **BIM** > **Drawing** > **Attach**.
- On the menu, click **BIM** > **Drawing** > **Attach**.

You can create a detail drawing once and reference it across multiple building models or plan drawings.

External references let you store drawings outside the project to link and reuse them. Internal references let you reference drawings created within the project by adding a call out at the required location.

This functionality reduces drawing duplication, minimizes the risk of errors, and improves project performance by keeping the main drawing lighter and easier to manage.

External Dimensions Support for Curtain Walls



External dimensions support curtain walls to recognize and include when generating dimensions for building facades.

To access this command:

- In the command window, type `BIMDIMENSIONCHAIN`.
- On the ribbon, click **BIM** > **Annotate** > **Dimension Chain**.
- On the menu, click **BIM** > **Dimension** > **Dimension Chain**.

Curtain walls are non-load-bearing exterior wall systems, commonly composed of mullions, panels, and grids, and are often used for complex glass or modular facade designs.

You can process the curtain wall information imported from BIM models as part of the external wall system. This produces more complete and accurate exterior dimensions, especially for projects with modern facades, angled walls, curved surfaces, or other nonstandard building envelopes.

Selection and Copy Commands

The speed and responsiveness of selection and copy commands in large drawings is improved.

The Copy command avoids recalculations and has improved performance with multiple repeated entities or xrefs.

Zoom and Pan Commands

The enhancements reduce CPU load during rendering preparation and improve smoothness during interactive operations like panning and zooming through sheet views.

It also makes the rendering engine more consistent across angle and core profiles and improves performance on both integrated and dedicated GPUs.

Redraw-heavy Operations

DraftSight provides faster command execution during redraw-heavy operations, especially in layouts with complex visual styles.

It reduces redundant **OpenGL** state changes by unifying shared uniform values across rendering passes.

Noticable improvement happens when you switch between model/layout tabs or the active viewport.

EXPERT Variable Value

You can specify the value of the *EXPERT* variable to 1 for the current session.

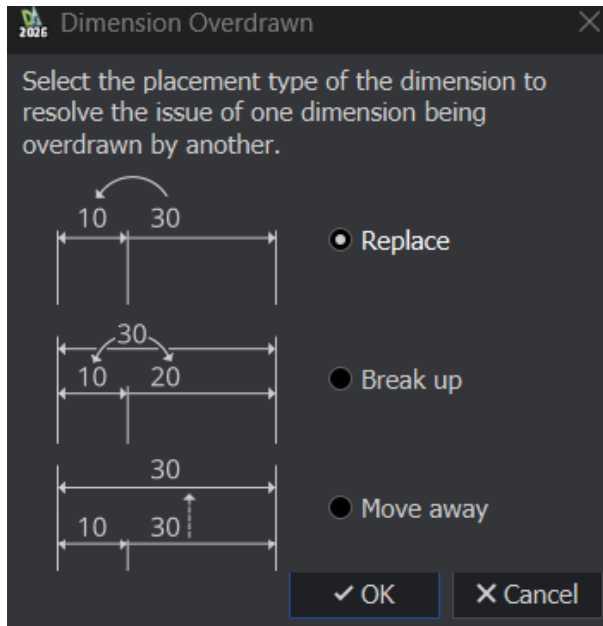
With the variable value as 1, when you open a *DWG* file in read-only mode, the notification does not appear. In a new session, *EXPERT* value resets to the default.

DraftSight Customer Experience Improvement Program

The DraftSight Customer Experience Improvement Program automatically sends diagnostic log files to Dassault Systèmes SolidWorks Corporation.

These logs provide information about product behavior, stability, and potential issues, helping the development team to understand how DraftSight performs.

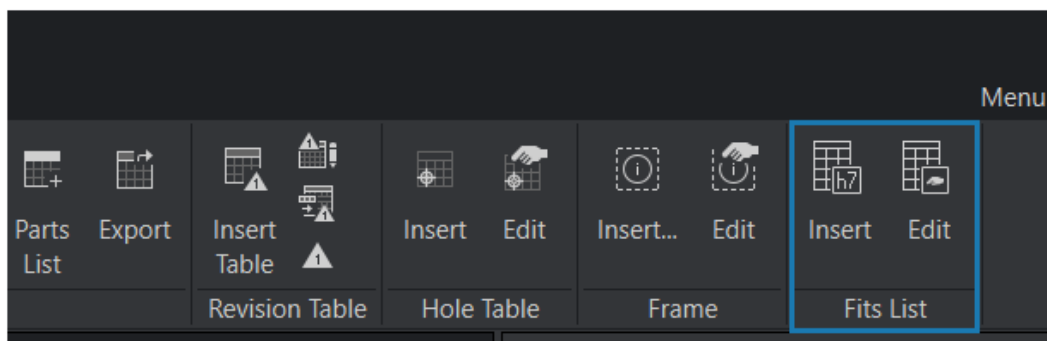
Dimension Overdrawn Dialog Box (DraftSight Mechanical Only)



You can use the Dimension Overdrawn dialog box to fix linear dimensions.

The dialog box appears when a new linear dimension overlaps an existing dimension. It lets you replace, split, or move the dimension while keeping all of its original properties. The tool arranges the lines into the baseline layout to avoid deleting and recreating dimensions.

Fits List Command (DraftSight Mechanical Only)



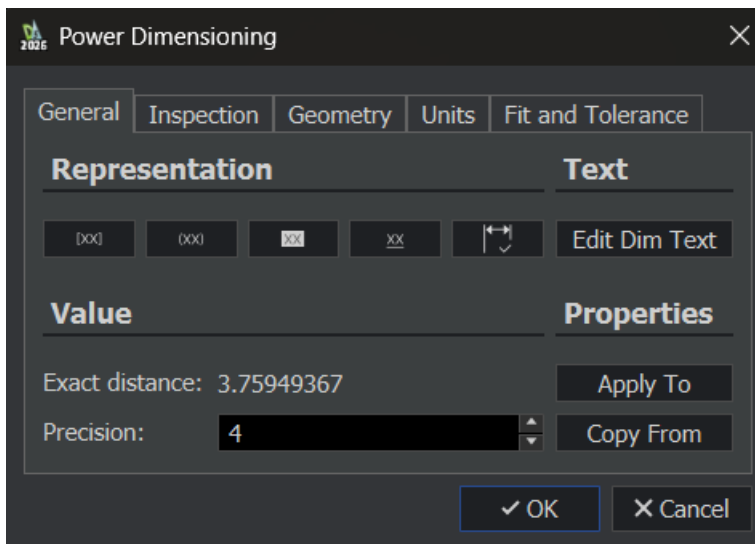
You can use the `AM_FITSLIST` command to pull every fit dimension from an open drawing, build a table, and insert it.

To access this command:

- In the command window, type `AM_FITSLIST`.
- On the ribbon, toolbar, and menu, click **Mechanical Annotate** > **Sheet** > **Fits List** > **Insert/Edit**.

Whenever you edit a dimension and run the command, the table updates automatically, keeping the fit data in sync with the drawing.

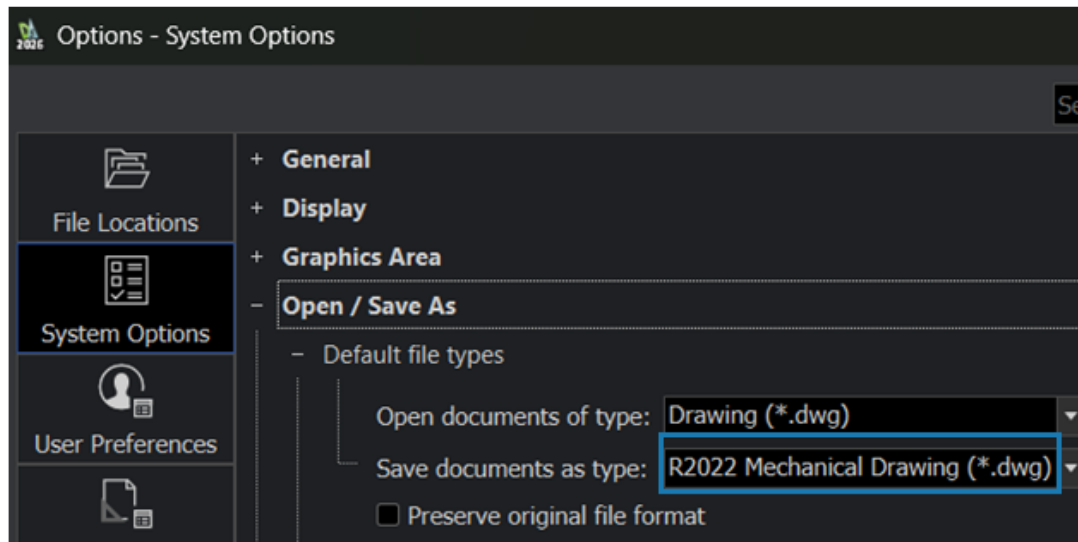
Power Dimensioning Dialog Box (DraftSight Mechanical only)



You can use the Power Dimensioning dialog box to create or edit dimensions with an easy workflow.

To use this dialog box instead of the contextual ribbon, use the `AMPOWERDIMEDITOR` system variable. By moving all the options into a tab-based interface, you save time switching between the ribbon and properties window.

Support for Drawing 2022 Files (DraftSight Mechanical Only)



DraftSight Mechanical opens DWG™ 2022 files that contain part references, part lists, BOMs, and annotations, and lets you modify that data inside the drawing.

This lets you work with the latest DWG file format without any file conversion steps or data loss, and maintain continuity between CAD and manufacturing workflows.

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eDrawings

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

Supported File Types

eDrawings has updated the supported versions for several file types.

Format	Version
ACIS® (.sat, .sab)	From 1.3 (103) to 35.0 (3500)
Autodesk® Inventor® (.ipt, .iam)	Up to 2026
CATIA® V5 (.CATPart, .CATProduct)	Up to V5-6R2026
CATIA V6 / 3DEXPERIENCE ® .3DXML	3DXML 4.3
Creo® - Pro/Engineer® (.ASM, .NEU, .PRT, .XAS, .XPR)	Pro/Engineer 19.0 to Creo 12.4.0.0
NX™ (Unigraphics®) (.prt)	UG11 to UG18, UG NX, NX5 to NX12, NX1847 to NX2512
Parasolid® (.x_b, .x_t, .xmt, .xmt_txt)	Up to 38.0
Solid Edge® (.asm, .par, .pwd, .psm)	1 to 20, ST1 - ST10, 2019 to 2026

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SOLIDWORKS Plastics

This chapter includes the following topics:

- **Materials Database**
- **Custom Project Folder**
- **Reynolds Number Result in Cool Simulations**

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.

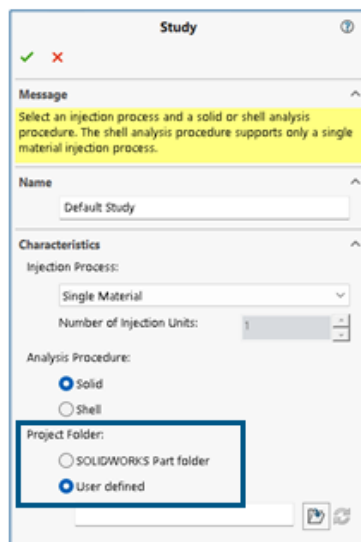
76 new material grades are added, 27 grades are updated, and 28 obsolete grades are removed from the database.

Manufacturer	Number of New Material Grades
Covestro®	29
MOCOM®	24
Solvay Specialty Polymers®	20
EMS-GRIVORY®	2
Trinseo®	1

Manufacturer	Number of Updated Material Grades
Covestro	22
Solvay Specialty Polymers	5

Manufacturer	Number of Removed Material Grades
Solvay Specialty Polymers	19
Covestro	7
EMS-GRIVORY	2

Custom Project Folder



You can specify custom project folders for storing study results, providing greater flexibility in file management. This helps separate simulation data from CAD files for improved project organization and reduced file clutter.

Custom folders simplify synchronization with PLM and PDM systems, making it easier to manage and track simulation results in enterprise workflows.

Reynolds Number Result in Cool Simulations

You can obtain Reynolds number results in cool simulations to help validate turbulent flow conditions in injection mold cooling circuits. This helps you evaluate flow behavior and confirm whether cooling channels operate within the desired flow regime.

By analyzing Reynolds number results, mold design engineers can optimize flow rates, pressure drop, and heat transfer efficiency to improve overall cooling performance and mold design.

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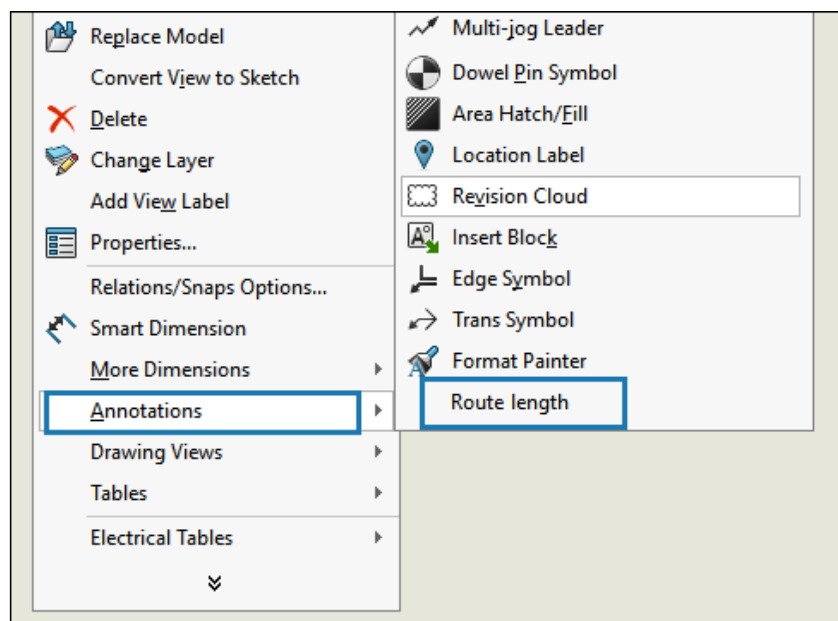
Routing

This chapter includes the following topics:

- **Adding Route Length Annotations in Flattened Route Drawings**
- **Creation of Individual Parts for Wires and Cables**
- **Exporting Pipe and Tube Data in a Single Operation**
- **In-Place Spline Conversion for Electrical Routes**
- **Make Assembly from Part Supports Routing Components**
- **Resizable From-To List Wizard Dialog Box**

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

Adding Route Length Annotations in Flattened Route Drawings



You can use the **Route Length** command to manually add or restore length annotation in flattened route drawings.

Benefits: You no longer need to recreate flattened route drawing views when route length annotations are deleted. It reduces rework, improves detailing efficiency, and gives you greater control over annotation management in both small and large routing designs.

If you accidentally delete route length annotations while cleaning a drawing, you can quickly restore them individually or all at once without recreating the flattened route drawing view.

To add or restore a route length annotation:

- Right-click a route segment, and select **Annotation > Route Length**.

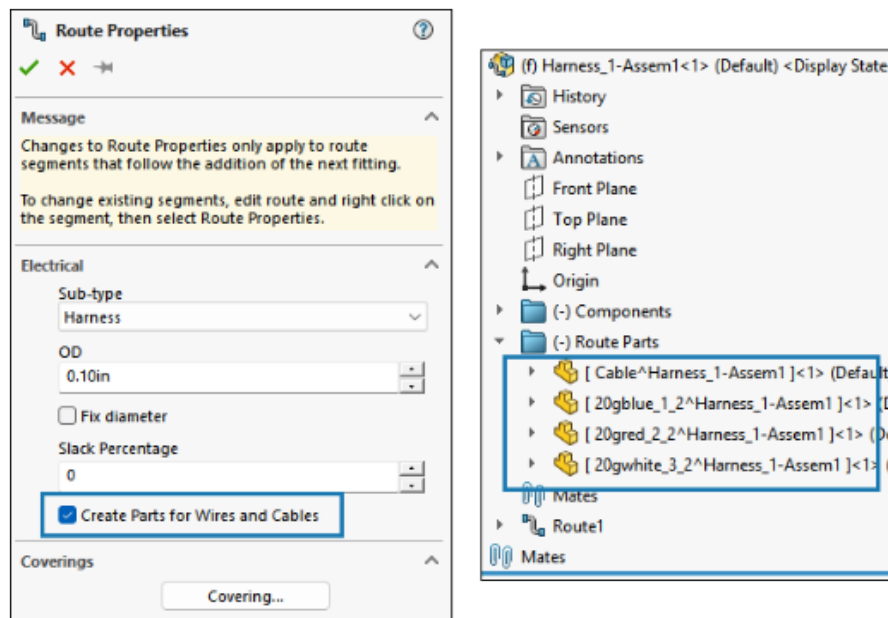
To restore all missing annotations:

- Right-click drawing view, and select **Annotation > Route Length**.

Enhancements include:

- Maintain full associativity between annotations and route geometry, ensures annotations update automatically when the route geometry changes.
- Supports selecting multiple route segments to create an annotation in a single operation.
- Prevent duplicate annotations from being added to route segments that already contain a length annotation.

Creation of Individual Parts for Wires and Cables



You can use **Create Parts for Wires and Cables** to treat each wire, cable, and harness as individual parts in the assembly instead of grouped features.

Treating each wire and cable as an individual part in the assembly using the **Create Parts for Wires and Cables** helps you locate, identify, and track specific wire components within the routing assembly. It also improves navigation, traceability, and visualization workflows in eDrawings and SOLIDWORKS Composer.

Previously, all wires and cables of a particular style are grouped together as features under a single part in the Route Parts folder.

Benefits: The **Create Parts for Wires and Cables** improves the traceability and manageability of individual wire and cable components, streamlines manufacturing

workflows. It aligns with the capabilities of the **3DEXPERIENCE** platform for enhanced data management, visualization, and analysis.

To create individual part files for wires and cables:

1. Under the **Electrical** tab, click **Route Properties**.
2. Select **Create Parts for Wires and Cables**.

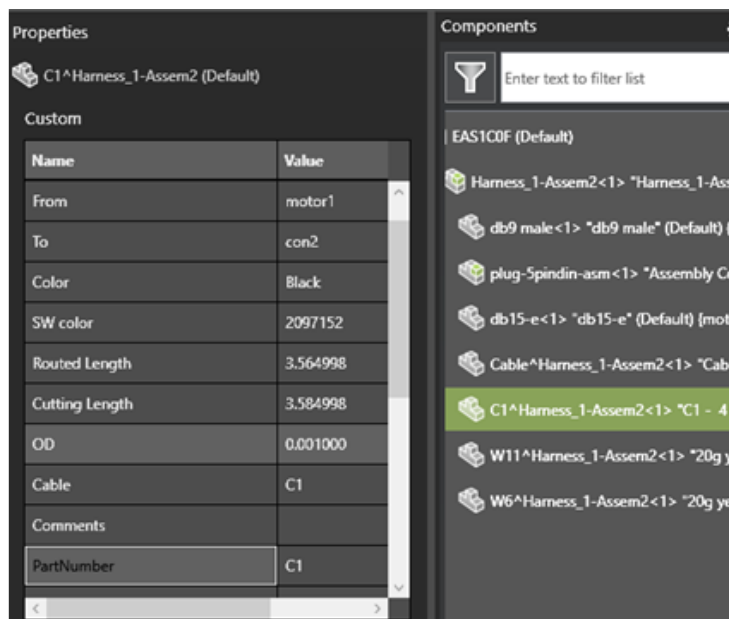
When selected, each wire and cable in the existing route assembly is created as a separate part file. The setting also applies to new routes added to the assembly.

Changes apply only after exit **Edit Route** CommandManager. A notification appears if unsaved changes exist.

The option works for the following cases:

Cases	Description
Wires in a route assembly	Each wire style is associated with a single route subassembly, and every wire within that subassembly is represented as an individual part.
Cables in a route assembly	The cable is represented as a single part instead of multiple separate features. The part acts as a container for the complete cable, while each cable core within the part is represented as an individual solid-body feature.
Wires in a harness assembly	Each wire is represented as a single part in the FeatureManager design tree, representing the complete wire instead of multiple separate features.
Cables in a harness assembly	The cable is represented as a single part that acts as a container for the entire cable. Within the cable part, each cable core is represented by its own solid-body feature.

Viewing Electrical Information for Wires and Cables in eDrawings



Electrical information for wires and cables is available through the custom properties of the generated part files in eDrawings.

When wires and cables are treated as individual parts, you can view detailed electrical properties for each wire and cable directly from the corresponding part file. This enables efficient searching, identification, and retrieval of electrical information using custom properties.

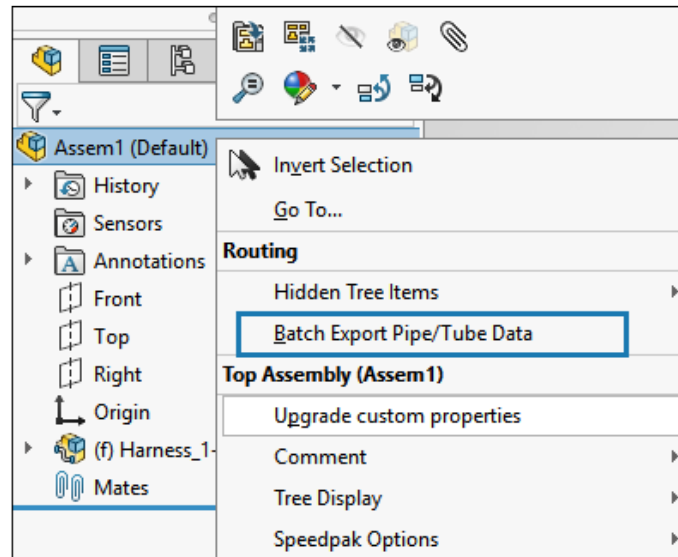
Previously, since wires and cables were represented only as features or solid bodies, eDrawings only displayed custom properties for parts and assemblies. You could not search for individual wires or cables or access their associated electrical information through custom properties.

To view the electrical properties of a wire or cable:

1. Open the assembly in eDrawings.
2. In the lower-right corner of the window, click **Components** to open the component tree.
3. From the tree, select the specific wire or cable.
4. Click **Properties** to view the associated electrical information.

The Properties pane displays electrical details such as wire name, wire size/gauge, insulation type, voltage rating, To/From component information, and pin termination details.

Exporting Pipe and Tube Data in a Single Operation



The **Batch Export Pipe/Tube Data** command lets you export routing data for all pipe and tube subassemblies directly from the top-level assembly in a single operation. You can save processing time for large routing projects as you no longer need to manually open and export each route assembly individually.

Benefits: This automation ensures consistent and organized exports while maintaining folder-level separation for manufacturing readiness. By applying uniform export settings across all routed assemblies, the command can reduce export processing time from minutes to seconds.

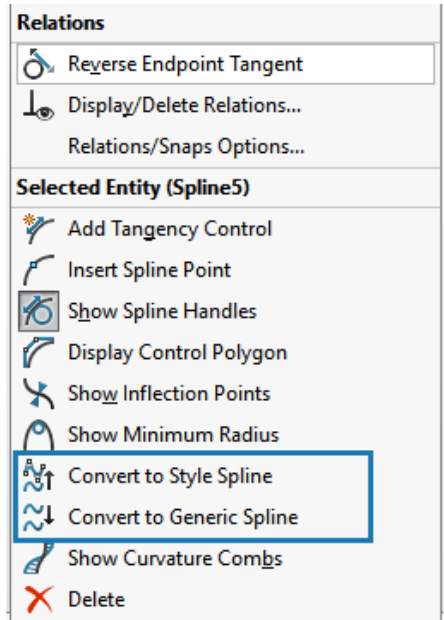
To access the command, do one of the following:

- Right-click the top-level assembly, and select **Batch Export Pipe/Tube Data**.
- Click **Tools > Routing > Piping > Batch Export Pipe/Tube Data** .
- Click **Tools > Routing > Flexible Tubing > Batch Export Pipe/Tube Data** .

Key Features:

- Detects all routed pipe and tube subassemblies in the active top-level assembly.
- Exports routing data for all detected assemblies in a single operation.
- Lets you define a target export folder for centralized output management.
- Creates separate folders and export files for each routed assembly.
- Applies consistent export settings across all routed assemblies.
- Generates a summary report of successful and failed exports.

In-Place Spline Conversion for Electrical Routes



While editing electrical routes, you can convert Default Spline (B-splines) into Style Splines and Generic Splines directly within the **Edit Route** environment using the **Convert to Style Spline** and **Convert to Generic Spline** options, without deleting or recreating route paths.

These conversion options provide faster and more flexible route editing with nondestructive spline controls for smoother and more accurate wire path adjustments.

Benefits:

- Eliminates the need to recreate wire paths during design changes.
- Enables faster route refinement and iteration.
- Improves manufacturable wire path creation.
- Provides better control over spline geometry.
- Preserves electrical connectivity and routing intelligence.

About the Splines:

Spline	Description
Default Spline (B-Spline)	Standard spline behavior for general route editing.
Generic Spline	Provides control polygon editing for precise local adjustments in constrained areas.
Style Spline	Uses handle-based controls to create smooth, manufacturable wire paths that better represent real-world wire behavior.

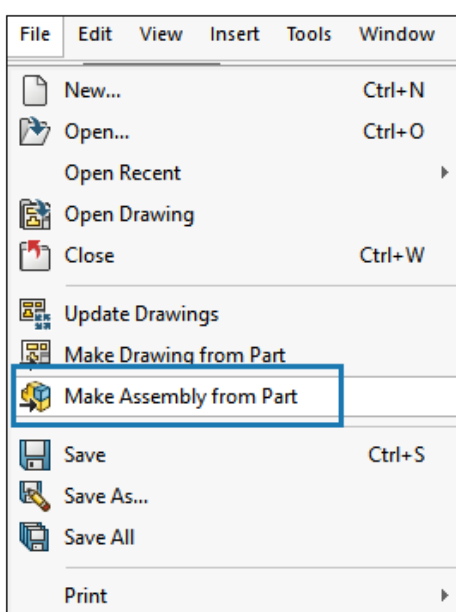
To switch the spline style type:

1. Enter **Edit Route** mode.

2. Right-click a spline segment.
You can select multiple spline segments to convert them in a single operation.
3. Select one of the following:
 - **Convert to Style Spline** 
 - **Convert to Generic Spline** 

These conversion options are not supported for ribbon and flex cables.

Make Assembly from Part Supports Routing Components



You can create routing-compatible assemblies directly from routing components such as flanges, tees, and crosses using the existing **Make Assembly from Part**  command. This speeds your workflow by eliminating manual routing and workaround steps. This feature automatically recognizes routing components and starts the appropriate workflow.

Routing components	Initializes a routing-compatible subassembly.
--------------------	---

Nonrouting components	Creates a standard assembly.
-----------------------	------------------------------

Benefits:

- Eliminates manual routing assembly setup
- Reduces errors when creating routing assemblies.
- Preserves standard SOLIDWORKS workflows.
- Simplifies electrical and piping routing preparation.

- Enables faster reuse of routing-ready components.

Previously, you could not use this command with routing components.

To access the command:

1. Open a part file.
2. Click **File** > **Make Assembly from Part** .

New Functional Behavior for Routing Components (Parts with CPoints)

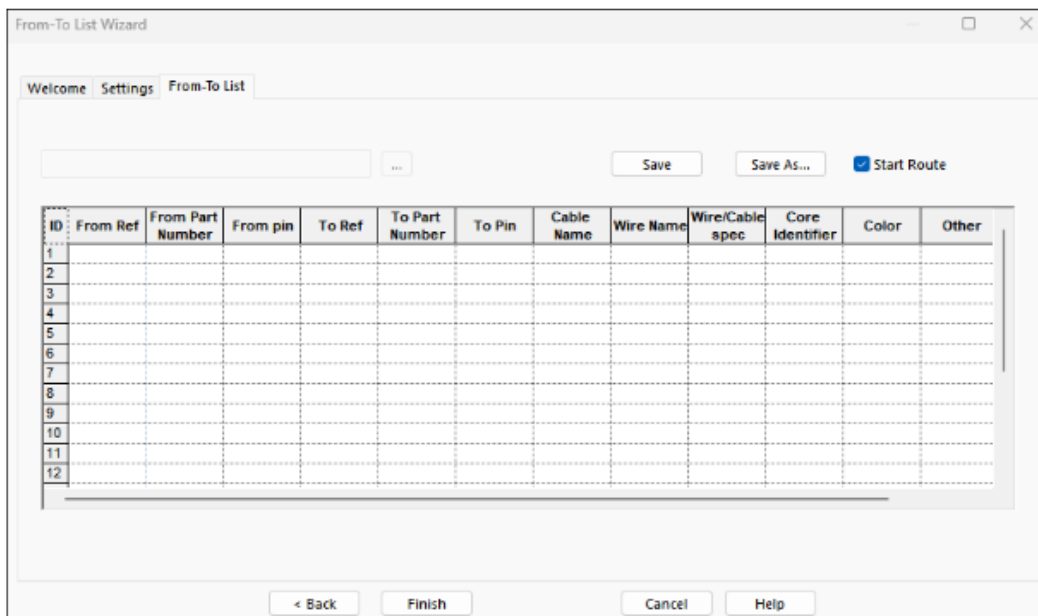
When the selected part contains at least one CPoint:

- If multiple configurations exist, you are prompted to select a configuration.
- If the Routing add-in is loaded:
 1. SOLIDWORKS opens Route Properties.
 2. Creates a routing-compatible subassembly.
 3. Inserts the component into the routing context.
 4. Automatically activates Auto Route for electrical components.
- If Route Properties is canceled:
 - A standard assembly is created instead.
- If the Routing add-in is not loaded:
 - A standard nonrouting assembly is created.

Nonrouting Components:

Behavior remains unchanged of the command and creates a standard assembly.

Resizable From-To List Wizard Dialog Box

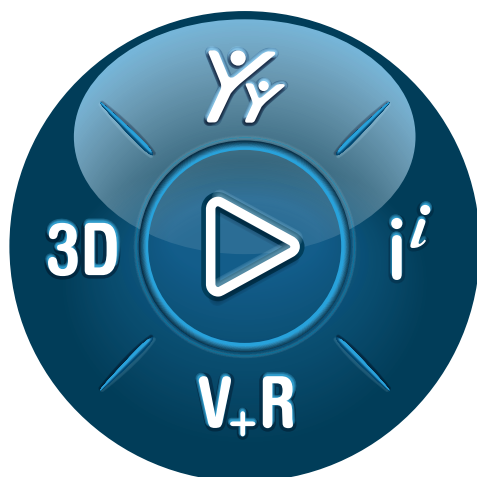


ID	From Ref	From Part Number	From pin	To Ref	To Part Number	To Pin	Cable Name	Wire Name	Wire/Cable spec	Core Identifier	Color	Other
1												
2												
3												
4												
5												
6												
7												
8												
9												
10												
11												
12												

You can resize or maximize the From-To List Wizard dialog box, and the grid adapts automatically to fit more rows, columns, and modern screen resolutions. It improves usability for large data sets (Example: wire lists, harness connectivity, routing workflows).

Benefits: This capability lets you efficiently view, review, edit, and validate large From-To connectivity data sets within the From-To List Wizard without excessive scrolling or navigation effort. It is especially beneficial for handling complex electrical projects where connectivity tables contain large amounts of data and multiple attributes that you must review together.

To access From-To List Wizard dialog box, click **Tools > Routing > > Electrical > From-To List Wizard**.



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Europe/Middle East/Africa

Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

Asia-Pacific

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

Americas

Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
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