

WHAT'S NEW

SOLIDWORKS 2017



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1

Welcome to SOLIDWORKS 2017

This chapter includes the following topics:

- **Top Enhancements**
- **Sample Files**
- **For More Information**

SOLIDWORKS® 2017 delivers many enhancements and improvements, most created in direct response to customer requests. This release focuses on helping you get your work done faster and more easily than ever before by enabling you to:



- **Design.** Drive innovation with precise, advanced modeling capabilities. Work faster with more control and less overhead.
- **Validate.** Solve complex problems and validate your designs with confidence and ease. Increase productivity while gaining greater insights into design performance.
- **Collaborate.** Accelerate new product innovation with efficient content sharing across the complete supply chain. Leverage existing content and engage all team members more effectively.

- **Build.** Implement a fully integrated model-based manufacturing strategy. Use your 3D CAD models, including dimensions, tolerances, annotations, tables, and electrical information to create all your manufacturing deliverables.
- **Manage.** Execute a multi-disciplinary product data management workflow. Collect and manage all design data across your entire enterprise.

Top Enhancements

The top enhancements for SOLIDWORKS 2017 provide improvements to existing products and innovative new functionality.

Throughout this guide, look for the  symbol in these areas:

- User Interface**
 - [Customizing Configuration Sort Order](#) on page 18
- Assemblies**
 - [Distance Mates to Cylindrical Components](#) on page 49
 - [Facility Layout](#) on page 35
 - [Assembly Performance Improvements](#) on page 34
 - [Preserving Drawing References](#) on page 52
 - [SpeedPak Enhancements](#) on page 54
 - [Treehouse Improvements](#) on page 56
- SOLIDWORKS Costing**
 - [Common Rules for Sheet Metal and Machining Parts](#) on page 62
 - [Evaluating the Cost of Assemblies in Nested Tree Mode](#) on page 65
 - [Feature Recognition for Operations](#) on page 60
 - [Rules-based Costing for Parts](#) on page 62
 - [Subassembly Costing](#) on page 64
 - [Unique Rules for Sheet Metal Templates](#) on page 64
- DimXpert**
 - [Multiple Edge Selection](#) on page 70
- Drawings and Detailing**
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- Import/Export**
 - [IFC Entity Classification and Custom Property Export](#) on page 104
 - [Exporting SOLIDWORKS files as 3MF Files](#) on page 106
- Model Display**
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 - [Creating Transparent Section Views](#) on page 132
 - [PhotoView 360 Network Rendering Supports Frame-by-Frame Rendering](#) on page 132

- Parts and Features**
 - [Bidirectional Circular Patterns](#) on page 141
 - [Chamfer Enhancements](#) on page 143
 - [Converting Features to Bodies and Surfaces](#) on page 144
 - [Advanced Holes](#) on page 136
 - [Derived Part References are Retained](#) on page 148
 - [Sweep Profile Selection of Faces, Edges, and Curves](#) on page 150
 - [Window Selection for Instances to Skip](#) on page 154
 - [Wrap Creates Geometry on Any Face](#) on page 155

- Routing**
 - [Enhancements in Edit Flattened Route](#) on page 192
 - [Securing Distinct Routes Through Clips](#) on page 196

- Sheet Metal**
 - [Creating Three Bend Corner Reliefs](#) on page 198

- Sketching**
 - [Creating Sketch Offsets on 3D Geometry Surfaces](#) on page 216
 - [Prevention of Accidental Micro Lines](#) on page 219
 - [Shaded Sketch Contours](#) on page 219
 - [Extruding Shaded Sketch Contours](#) on page 220

- SOLIDWORKS MBD**
 - [Comparing 3D PMI Between Parts](#) on page 123

- SOLIDWORKS PDM**
 - [Generating SOLIDWORKS MBD 3D PDFs](#) on page 160
 - [Latest Version Overwrite](#) on page 161
 - [Rolling Back a File with References](#) on page 166
 - [SOLIDWORKS PDF Task Add-in](#) on page 171
 - [SOLIDWORKS PDM Web2 Viewer Enhancements](#) on page 181
 - [Vault Database Replication](#) on page 183

- SOLIDWORKS Simulation**
 - [Improved Arc Length Control Method for Nonlinear Contact Analysis](#) on page 207
 - [Editing Multiple Contact Sets](#) on page 206
 - [Mass Properties in Simulation](#) on page 208
 - [Offloaded Simulation](#) on page 209
 - [Remote Loads and Mass for Beams](#) on page 214

- SOLIDWORKS Toolbox**
 - [Purging Inactive Toolbox Configuration Data](#) on page 222

All features are available in SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium unless otherwise noted.

Sample Files

Beginning in SOLIDWORKS 2017, sample files are stored in a new location.

The previous location was in the SOLIDWORKS installation folder. The **Program Files** folder is restricted from write-access by default on all supported operating systems. Many users were unable to complete tutorials due to these restrictions. The **Public Documents** folder, by default, is writeable and visible to all users.

To open sample files for this book, go to *drive letter:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\chapter name\filename*.

For example, C:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\sheet metal\bracket.sldprt.

For More Information

Use the following resources to learn about SOLIDWORKS:

What's New in PDF and HTML This guide is available in PDF and HTML formats. Click:

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

Interactive What's New In SOLIDWORKS, click the  symbol to display the section of this manual that describes an enhancement. The symbol appears next to new menu items and the titles of new and changed PropertyManagers.

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

What's New Examples What's New Examples are updated at every major release to provide examples of how to use most top enhancements in the release.

To open What's New Examples click **Help > What's New > What's New Examples**.

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

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

2

User Interface

This chapter includes the following topics:

- [Customizing Configuration Sort Order](#)
- [Disabling Selection Accelerator Toolbars](#)
- [Dynamic Reference Visualization](#)
- [Comment in FeatureManager Design Tree](#)
- [Hide All Types](#)
- [Interface Updates for SOLIDWORKS Rx, Task Scheduler, and Sustainability](#)
- [Property Tab Builder Redesign](#)
- [Selection Breadcrumbs](#)
- [Shortcut Menus](#)
- [Suppressing Configuration and Display State Names in the FeatureManager Design Tree](#)
- [Touch-based Interactions](#)

Customizing Configuration Sort Order

You can specify the order in which configurations are listed in the ConfigurationManager.

To customize configuration sort order:

1. In the ConfigurationManager, right-click the top-level configuration and click **Tree Order**.
2. Select one of the following:

Option	Description
Numeric	Sorts by ascending alpha or numeric value.
Literal	Sorts by alphabetical order.
Manual (Drag and Drop)	Lets you drag and drop configurations in the configuration tree. As you drag up or down the tree, each configuration that you drag over is highlighted. The configuration that you are moving drops immediately below the currently highlighted item when you release the pointer.
History-Based	Sorts by the date the configuration was created, from earliest created at the top of the list to most recently created at the bottom of the list.

Disabling Selection Accelerator Toolbars

You can disable the selection accelerator toolbars that appear when you select entities for the **Fillet**, **Move Face**, and **Delete Face** tools.

To disable selection accelerator toolbars:

Do one of the following:

- In the PropertyManager, clear **Show selection toolbar**.
- Click  at the end of the selection toolbar.

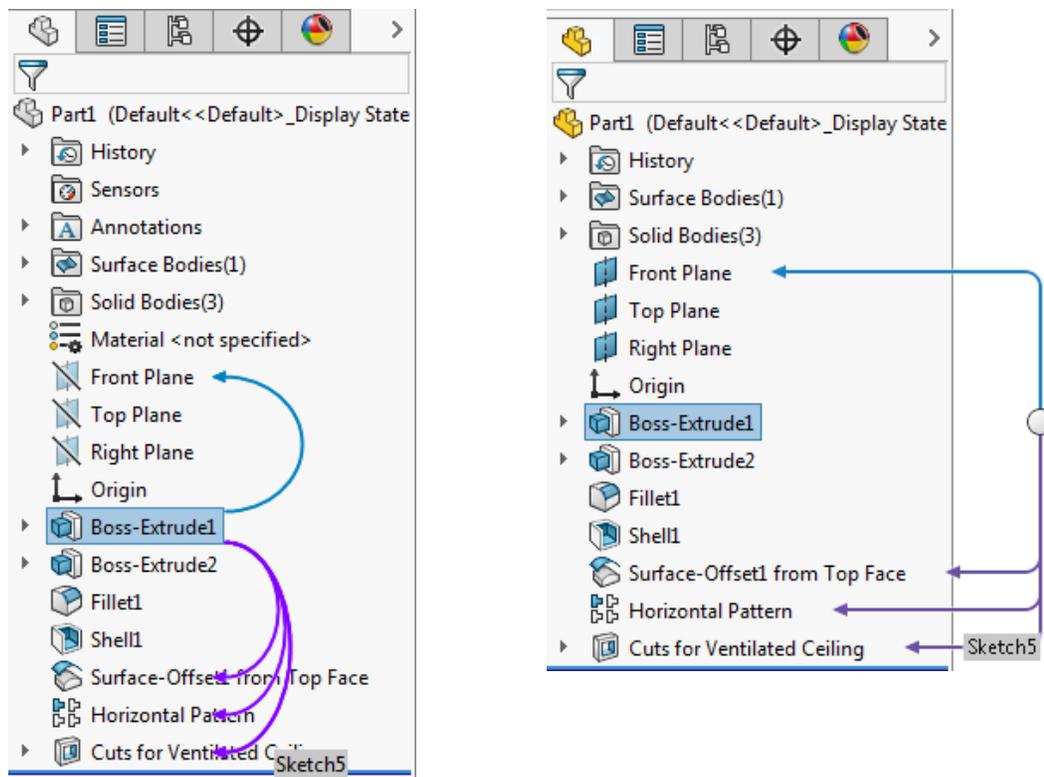
The selection is stored for the next feature in the PropertyManager. For example, if you clear a check box for a fillet, it remains cleared for the next fillet in the PropertyManager.

To enable accelerator toolbars, select **Show selection toolbar**.

Dynamic Reference Visualization

Dynamic reference visualization showing parent and child relationships is enhanced to avoid overlapping text and lines.

Arrows begin from the circle to indicate the feature whose parent and child relationships you are viewing.



Comment in FeatureManager Design Tree

The existing ability to add comments to models has been enhanced. It is easier to view and edit models based on the comments over the lifecycle of a product's design.

- You can add comments to almost all nodes in the FeatureManager design tree. The comments extends to the following nodes:
 - Mates (all types)
 - Mate folder or Mate group
 - Folders
 - Sensors
 - Material
 - Annotation views
 - Blocks
 - Solid bodies folder
 - Surface bodies folder
 - Sheet metal nodes
 - Mate references
- To locate comments more easily, you can turn on the comment indicators.
- You can add standalone comments that are not attached to a feature or node in the FeatureManager design tree.
- When you create or edit a comment, you can attach an image or a screen capture of the model.
- You can view feature comments in the PropertyManager.
- When you add a comment to a feature, a timestamp is added automatically to the comment.
- You can view all comments in one dialog box and export the comments to a Microsoft Word document.

The ConfigurationManager also shows similar enhancements for comments.

Turning on Comment Indicators

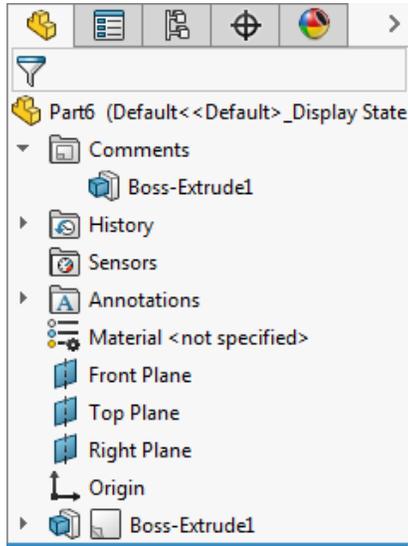
You can turn on comment indicators in the FeatureManager design tree display to locate the comments more easily.

To turn on comment indicators:

1. In the FeatureManager design tree, right-click the top node.

2. Click **Tree Display > Show Comment Indicator**.

Visual indicators  appear in the FeatureManager design tree to indicate if a comment exists.



Adding Standalone Comments

You can add standalone comments that are not attached to a feature or node in the FeatureManager design tree.

To add standalone comments:

1. Click **Add Comment**  (Standard toolbar).
2. In the Comment box, type the comment and click **Save and Close**.

The standalone comment is added in the **Comments**  folder in FeatureManager design tree.

You can also right-click the **Comments** folder and click **Add Comment** to add a standalone comment.

Adding Images and Screenshots to Comments

You can add images and screenshots to comments. When you add a comment to a feature, a timestamp is added automatically to the comment.

To add images and screenshots to comments:

1. Click **Add Comment**  (Standard toolbar).

2. In the Comment box, click one or both of the following:
 - **Insert image** . Lets you browse for an image and adds it in the comment with a preview.
 - **Insert screenshot** . Adds an image of the graphics area in the comment with a preview.
3. Click **Save and Close**.

Viewing and Editing Comments in PropertyManagers

You can view and edit the comments of a feature in PropertyManagers.

To view and edit comments in PropertyManagers:

1. Click **Tools > Options > System Options > Collaboration**.
2. In the dialog box, select **Show Comments in PropertyManager**.
3. In the graphics area, select a feature that has a comment attached to it and click **Edit Feature** .
4. In the PropertyManager, under Comment, click **Edit**.
5. Modify the comment and click **Save and Close**.
6. Click .

Hide All Types

You can use the **Hide/Show Items** tool in the Heads-up View toolbar to hide all items in one click without leaving the graphics area.

The tool is split into two: . On the left side, you can toggle between hide all types and show the previous visibility state. On the right side, you can view the existing list of types to change visibility of one item at a time.

Interface Updates for SOLIDWORKS Rx, Task Scheduler, and Sustainability

The user interface for SOLIDWORKS Rx, Task Scheduler, and Sustainability have been updated for consistency with the user interface redesign in SOLIDWORKS 2016.

Improvements include an updated color scheme, icons, tabs, and more readable text.

Property Tab Builder Redesign

The user interface for the Property Tab Builder has been updated to be easier to use.

Improvements include an updated color scheme and icons, and an improved layout of the areas for creating group boxes, textboxes, lists, and other features of custom tabs.

The installed version of the SOLIDWORKS software displays in the Property Tab Builder header. This label is useful when you have users running different versions of the software, and you want to make sure that you are creating a template that matches what a given set of users is running on their machines.

You can also drag the border of the **Custom Properties** panel to make it wider or narrower relative to the **Control Attributes** panel.

Selection Breadcrumbs

Selection breadcrumbs have better feedback. When you hover over the items, breadcrumbs display information only for the mates that are applicable for the selected face, edge, vertex, feature, or body.

Enhancements include additional feedback for:

- Assembly mates. Mate breadcrumbs are better aligned so you know which mates apply to the selected parts or subassemblies.



- Failures and errors. Mates with failures or errors appear in breadcrumbs.



- Suppressed states. The suppression state of entities are reflected in breadcrumbs.



When you click a breadcrumb, only the mates that are associated with the entity appear.

Subassembly selected	Part selected

In addition, breadcrumbs provide easy access to a part's reference planes to let you:

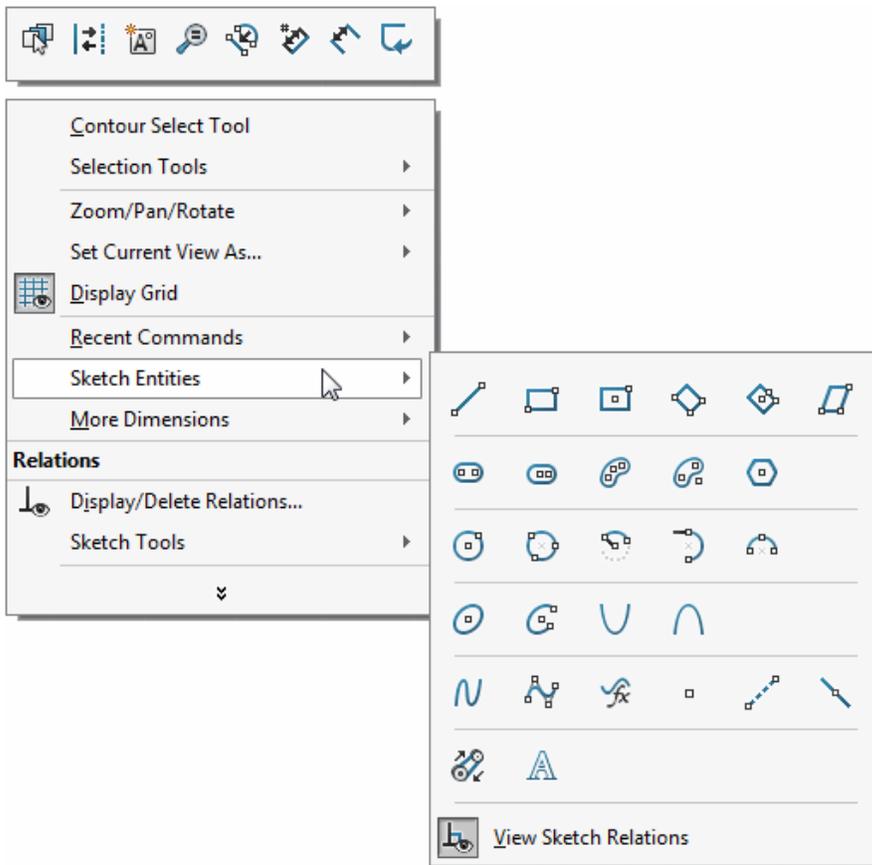
- Control the reference plane visibility.
- Pre-select reference planes for use as sketch planes, use in a feature, or creating an assembly mate.



Shortcut Menus

The shortcut menus are enhanced and standardized to provide quick access to the most commonly used tools.

To increase efficiency, duplicate tools are removed and related tools are moved to flyout menus. The long **Sketch Entities** flyout menu uses the compact grid format to avoid unnecessary mouse travel. You can scroll through the menu using the middle mouse wheel if the menu runs off the screen.



Suppressing Configuration and Display State Names in the FeatureManager Design Tree

In models that contain only one configuration, you can suppress the configuration and display state information that is appended to assembly and part names in the FeatureManager design tree.

To suppress configuration and display state names in the FeatureManager design tree:

In the FeatureManager design tree, right-click the top node and click **Tree Display > Do not show Configuration/Display State Names if only one exists.**

If there is only one configuration, the configuration and display state names are suppressed. You can still access the information in the FeatureManager design tree by hovering over the component.

If there is more than one configuration, the configuration and display state names appear after the component name.

Touch-based Interactions

Touch gestures are revised to make the user experience easier and more intuitive. The software is enhanced to support controls for basic view manipulation on touch screen devices.

You can work on models using the touch gestures and perform actions like a finger drag to rotate and a two finger drag to pan. Also, you can use **Pan**, **Roll**, and **Zoom In/Out** actions concurrently. These updated controls apply to Microsoft Windows 8 and later versions.

3

SOLIDWORKS Fundamentals

This chapter includes the following topics:

- **Application Programming Interface**
- **Improved Browse for Folder Dialog Box**

Application Programming Interface

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

SOLIDWORKS 2017 API includes the ability to:

- Create and access selection sets.
- Traverse the Manager Pane and activate the FeatureManager design tree, PropertyManager, ConfigurationManager, DimXpertManager, DisplayManager, or custom tab. Also, fire a pre-notification and notification when activating a tab in the Manager Pane.
- Get whether an axis is a temporary axis and get its reference face.
- Set the properties of a selected component in a specific configuration, including whether the component is an envelope.
- Get the parameterization data of a specific spline in a sketch.
- Get or set the state of a check box on a PropertyManager page.
- Create sweep features using a circular profile or tool body.
- Create bidirectional sweeps.
- Import tables from and export tables to Microsoft Excel files for variable patterns.
- Modify variable patterns.
- Get or set the width or height at which to lock a column or row in a table annotation.
- Offset selected edges to create a 3D sketch on a face or surface.
- Zoom a drawing sheet to its maximum size within a window.
- Make a selected component independent.
- Replace components either by letting SOLIDWORKS attempt to match the configuration of the old components with a configuration in the replacement component or by using the specified configuration in the replacement component.
- Isolate components.
- Get or set the routing component grouping options for a BOM table in a drawing of an assembly containing routing components.
- Get the names of the exploded views for a specific configuration, get the number of exploded views in a specific configuration, and get the name of the configuration for a specific exploded view in an assembly.
- Get the name of the exploded view currently shown in the model.

- Get the collapsed or exploded transform of a component when the assembly is exploded.
- Expand the specified component in the specified FeatureManager design tree pane.
- Delete either the selected components of a subassembly or the subassembly of the selected component.
- Get or set the display state of a drawing view.
- Group selected edges or sketch segments into one object for the path for a sweep feature.
- Get the persistent reference ID of the configuration referenced in a drawing view.
- Get or set whether to display the description of a configuration in a bill of materials.
- Get or set whether to enable cell drop-down lists in design tables.
- Get or set the vertical justification of a standard note.
- Get or set the length of a bent leader.
- Get or set whether to propagate visual properties in a table-driven pattern.
- Synchronize flexible subassemblies when creating a linear or circular pattern of components in an assembly.
- Get or set whether to propagate visual properties in a derived pattern feature.
- Get or set whether to disable an equation and get the number of disabled equations in a model.
- Specify a sketch plane, surface/face/plane, vertex, or offset as a start condition when creating an extruded surface.
- Draft angles, cap ends, delete original faces, and knit the bodies created by deleting original faces in extruded surfaces.
- Specify multiple drawing sheets whose setups to modify.
- Update SOLIDWORKS Toolbox components in the specified assembly level using the current information in Toolbox settings.
- Create SpeedPak configurations for subassemblies in an assembly.
- Rebuild all features or rebuild only those features that need to be rebuilt in all configurations without activating each configuration in a model.
- Create a STEP 242 file from SOLIDWORKS MBD.
- Create a derived pattern feature, get or set which pattern instance to use as the seed feature for a derived pattern feature, and get or set whether to propagate visual properties in a derived pattern feature.
- Get the DimXpert name for an annotation.
- Fire a notification when publishing an assembly or part document to SOLIDWORKS MBD 3D PDF.

Improved Browse for Folder Dialog Box

The Browse for Folder dialog box has been renamed to Select Folder and has been enhanced for ease of use.

The Browse for Folder dialog box had limited functionality, which made it more cumbersome to use. In the Select Folder dialog box, you can:

- See the **Date created** and **Date modified** information.
- Search for folders.
- Sort folders.
- Select virtual (non-file system) folders such as **My Computer**.

The Select Folder dialog box is available in many areas, such as the System Options - File Locations dialog box when you click **Add** and in the Design Library when you click **Add File Location** .

4

Installation

This chapter includes the following topics:

- **Installation Manager**
- **Previous Toolbox Detected When Installing or Upgrading**
- **Uninstall Completes Even When DLLs Cannot Be Unregistered**

Installation Manager

The following products are available in the Installation Manager:

- SOLIDWORKS PCB and SOLIDWORKS PCB Services
SOLIDWORKS PCB is part of the **Individual** installation. On the Serial Number page, you must enter a serial number for SOLIDWORKS PCB in the Electrical Design section.
SOLIDWORKS PCB Services is part of the **Server** installation. On the Welcome page, select Install SOLIDWORKS PCB Services on this computer. SOLIDWORKS PCB Services requires you to assign a port number. The default port number is 9780.
- SOLIDWORKS Visualize and SOLIDWORKS Visualize Boost
Both products can be installed as part of the **Individual** installation. For more information about the Visualize products, see the SOLIDWORKS Visualize section.

Previously, you could only install these products from separate installation packages.

Previous Toolbox Detected When Installing or Upgrading

If you are upgrading to SOLIDWORKS 2017 or installing SOLIDWORKS 2017 side-by-side with an earlier major version, the Installation Manager prompts you to decide how to handle your existing Toolbox/Hole Wizard data.

You can choose to:

- Create a new SOLIDWORKS 2017 Toolbox.
- Use an existing Toolbox with SOLIDWORKS 2017. If the existing data is an older version, it will be upgraded in-place.
- Create a copy of the existing Toolbox/Hole Wizard and upgrade the copy to 2017. This option allows you to use a copy for the newer version of the data while preserving the original for use with earlier versions.

In previous versions, users had to manually copy their Toolbox folders to preserve the older Toolbox/Hole Wizard data.

Uninstall Completes Even When DLLs Cannot Be Unregistered

You can uninstall the SOLIDWORKS software even if some DLLS cannot be unregistered. At the end of the uninstall process, a dialog box lists which DLLs failed to unregister. Typically, no further action is necessary.

Previously, uninstalling SOLIDWORKS failed if one or more DLLs could not be unregistered.

5

Administration

This chapter includes the following topics:

- **Pack and Go Improvements**
- **Preventing Users from Changing System Options**
- **Reliability Report and Reboot Check in SOLIDWORKS Rx**
- **Repairing Corrupted Files**
- **Running Selected Simulation Studies in Task Scheduler**

Pack and Go Improvements

If the **Flatten to single folder** option is not selected, Pack and Go now uses short, relative folder paths.

Previously, the entire path was used causing problems with the Windows 256 character limit.

The state of checkboxes in Pack and Go are now also retained. Previously, all checkboxes were cleared each time you ran Pack and Go; the software required that you check them again.

To access Pack and Go, in SOLIDWORKS, click **File > Pack and Go**.

Preventing Users from Changing System Options

You can use the Settings Administrator Tool to set system options that are applied when deploying or upgrading the SOLIDWORKS software, and to prevent users from changing those options. This tool replaces the Copy Settings Wizard for deploying settings, although that wizard is still available for copying settings.

You can start with:

- The default set of options for that release of SOLIDWORKS.
- Options previously applied on that computer.
- Options specified in an existing `.sldreg` or `.sldSettings` file.

Two columns of check boxes let you select which of these options to apply, and which of the applied options to lock so users cannot change them.

When users access the System Options dialog box, the locked options appear unavailable with a lock icon . However, you can create a password that allows users to unlock

options. When users hover over a locked option, a tooltip displays your name and email, so they can contact you and request that password.

You select whether the system options apply only the first time a user starts the software, or every time the SOLIDWORKS software starts, and whether customizations apply at the same time. You also select whether the options remain locked when users cannot access the network location with the `.sldSettings` file where the lock definitions are stored.

The Settings Administrator Tool is installed with a SOLIDWORKS administrative image. You can run the tool from the location where you stored the image.

The Settings Administrator Tool is stored in the following default location: `C:\SolidWorks Admin\SOLIDWORKS <version>\64bit\SOLIDWORKS\Program Files 64\SOLIDWORKS\sldSettingsAdmin.exe`.

Reliability Report and Reboot Check in SOLIDWORKS Rx

SOLIDWORKS Rx provides more data about events causing SOLIDWORKS to terminate on your machine. This information can help you or your VAR diagnose the root cause of any persistent reliability problem.

The **Diagnostics** tab displays the time since the last reboot and whether a reboot is pending.

The **Reliability** tab includes a chart listing sessions that have terminated in the past 60 days. The chart indicates whether a session ended normally or with a crash. For crash events, the call stack displays, along with a count of how many times that call stack appeared during the past 60 days. The **Reliability** tab also displays Windows events for the five minutes preceding each crash, including session termination and installation events.

When you generate a SOLIDWORKS Rx zip file, information from the **Reliability** tab is automatically included in the zip file, so you can share this information with your VAR.

To access the tabs in SOLIDWORKS Rx, from the Windows **Start** menu, click **All Programs > SOLIDWORKS version > SOLIDWORKS Tools > SOLIDWORKS Rx**.

Repairing Corrupted Files

The SOLIDWORKS software provides an option for repairing corrupted files. The auto repair function attempts to identify which segments are damaged from the corrupted files and removes them if possible.

When you encounter a file that is corrupted, a dialog box asks if you want to repair the file. If you answer **Yes**, the software automatically copies the corrupted file and attempts to repair the copied file rather than work on the original. If the copied file opens, the software notifies you that the repair is successful and reports which segment of the damaged file was removed and the impact that the segment has on the file. You can then save the repaired file if the result is acceptable.

Assemblies or drawings which reference damaged parts will open without the part files. To repair these files, you must open the parts in their own window. The original assembly

or drawing remains unchanged by the repair process. After the repair, you can substitute the repaired files for the original damaged files.

If a file fails to open or the result is not acceptable, you should retrieve a recent back up of the file. If a back up is not available or if file damage is a repeated problem, contact your local support representative for assistance.

Running Selected Simulation Studies in Task Scheduler

In the **Update Simulation** task, you can specify selected simulation studies to run for a file or file type. Previously, you had to run all studies for a file or file type.

To access simulation studies in Task Scheduler, from the Windows **Start** menu, click **All Programs > SOLIDWORKS version > SOLIDWORKS Tools > SOLIDWORKS Task Scheduler > Update Simulation**.

6

Assemblies

This chapter includes the following topics:

- **Assembly Performance Improvements**
- **Automatic Browsing when Inserting Components**
- **Disabling Previews of Hidden Components**
- **Facility Layout**
- **Support for Path Mates**
- **Grouping Components**
- **Improved Shortcut Menu for Configurations**
- **Loading Documents in Memory Only**
- **Mate Controller Enhancements**
- **Mates**
- **Options to Improve Performance in Large Assembly Mode**
- **Preserving Drawing References**
- **Searching for Externally Referenced Documents**
- **Sorting Components by Their Opening Time**
- **SpeedPak Enhancements**
- **Treehouse Improvements**

Assembly Performance Improvements

Assembly performance has improved when you drag components, rebuild assembly features, and show hidden components.

- Dragging components. In assemblies that contain hundreds or thousands of mates, the performance when you drag components is faster and smoother. Additionally, performance when dragging components in an assembly that has a center of mass feature is improved. After you drag the components, you need to update the center of mass. Click **Rebuild**  (Standard toolbar) or **Edit > Rebuild**.
- Rebuilding assembly features. Performance while rebuilding assembly features has improved.
- Showing hidden components. In large assemblies, when you use **Show Hidden Components** to toggle the display of shown and hidden components, the time to complete the process is reduced.

Automatic Browsing when Inserting Components

When you create a new assembly or open the Insert Components PropertyManager and no components are available under **Open documents**, the Open dialog box opens automatically so that you can browse for components to insert.

Previously, you had to click **Browse** to open the dialog box.

To disable this behavior for the Begin Assembly PropertyManager, clear the **Automatic browse when creating new assembly** option.

Disabling Previews of Hidden Components

You can turn off the functionality that causes previews to appear when you select hidden components in the FeatureManager design tree.

The previews can cause performance issues in large assemblies, especially when previewing complex components.

To turn off the preview behavior, click **Tools > Options > System Options > FeatureManager** and select or clear **Enable preview of hidden components**.

Facility Layout

You can use special tools and workflows when you work with very large models such as plant layouts.

You can define connection points and a ground face in part and assembly models and publish the models as assets. When you insert an asset into an assembly, magnetic mates snap the asset into position relative to other assets.

You can create a SpeedPak configuration of an asset directly in the Asset Publisher PropertyManager. Using SpeedPak configurations can greatly enhance performance, especially in very large assemblies. However, you cannot use entities from an existing SpeedPak when creating an asset.

Publishing an Asset

You can publish a model as an asset. You define connection points that enable the asset to snap into position relative to other assets in an assembly. Optionally, you define a ground plane, and you can create a SpeedPak configuration.

To publish an asset:

1. In a part or assembly, click **Tools > Asset Publisher**.

2. (Optional.) In the Asset Publisher PropertyManager, under **Ground Plane**, set the following:

Option	Description
Ground Plane	Defines which face of the model to attach to the ground plane when you insert the asset into an assembly. Select a model face.
Ground Plane Distance	(Optional.) Defines the offset distance for the selected model face from the ground plane. Enter a value.
Reverse Direction	(Optional.) Controls the alignment of the ground face relative to the ground plane. Click Reverse Direction  , or click the handle in the graphics area.

3. Under **Connecting Points**, set the following:

Option	Description								
Connect Reference Name	Defines the name of the connecting point you are creating. Enter a name or use the default.								
Connect Point	Defines a point for snapping to other assets in an assembly. Based on your selection the point is positioned as follows: <table border="1" data-bbox="641 1045 1417 1306"> <thead> <tr> <th>Selection</th> <th>Point Position</th> </tr> </thead> <tbody> <tr> <td>Vertex</td> <td>At the vertex</td> </tr> <tr> <td>Linear edge</td> <td>At the midpoint</td> </tr> <tr> <td>Circular edge</td> <td>At the center point</td> </tr> </tbody> </table>	Selection	Point Position	Vertex	At the vertex	Linear edge	At the midpoint	Circular edge	At the center point
Selection	Point Position								
Vertex	At the vertex								
Linear edge	At the midpoint								
Circular edge	At the center point								
Connect Direction	Defines the direction in which to snap when you drag the asset near another asset's connection point in an assembly. Select a face.								

4. Click **Add Connector**.

The connection point and direction are indicated on the model.

5. Repeat steps 3 and 4 to add more connection points.
6. (Optional.) Under **Option**, select **Create SpeedPak**.

A SpeedPak configuration is created as a child of the original configuration. No components appear in the FeatureManager design tree. In the graphics area, when you move the pointer over the asset, only the connection points, their direction faces, and the ground face are visible in the region surrounding the pointer.

7. Click .

Defining a Ground Plane

You can define a ground plane in an assembly. When you insert a published asset into the assembly, the asset's ground face snaps to the assembly's ground plane.

To define a ground plane:

1. In an assembly, click **Insert > Reference Geometry > Ground Plane**.
2. In the PropertyManager, in **Ground Plane**, select a face in the assembly.
3. (Optional.) Click **Reverse Direction** , or click the handle in the graphics area.
4. Click .

Ground Plane  appears in the FeatureManager design tree. When you insert an asset, the asset's ground face snaps to the assembly's ground plane.

Using Magnetic Mates

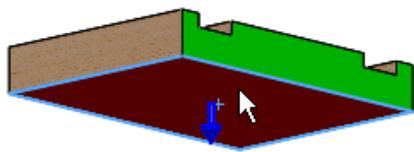
In this exercise, you publish two models as assets by setting their ground planes and adding connecting points for the magnetic mates. Then, in an assembly, you create a ground plane and insert the assets into the assembly. Finally, you use magnetic mates to connect the assets.

Publishing the First Asset

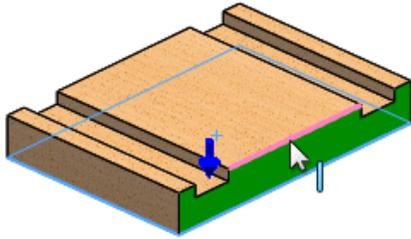
In this step, you use a straight train track to define the ground plane and add connection points for the magnetic mates.

To publish the first asset:

1. Open *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\assemblies\straight_track.SLDPRT.
2. Click **Tools > Asset Publisher** .
3. In the graphics area, select the face shown for **Ground Plane**.

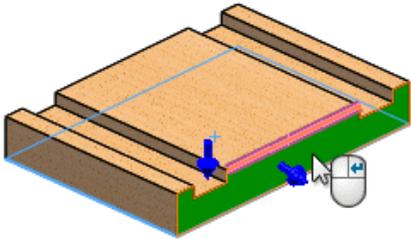


4. Select the edge shown for **Connect point** under **Connecting Points**.



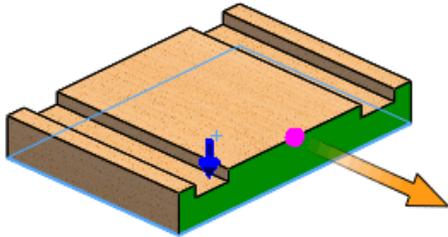
When you select a linear edge, the midpoint is chosen.

5. Select the face shown for **Connect direction**.

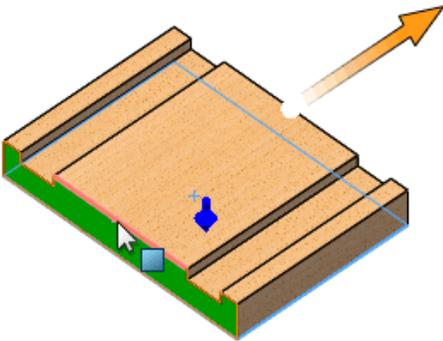


You can improve workflow by right-clicking in the graphics area.

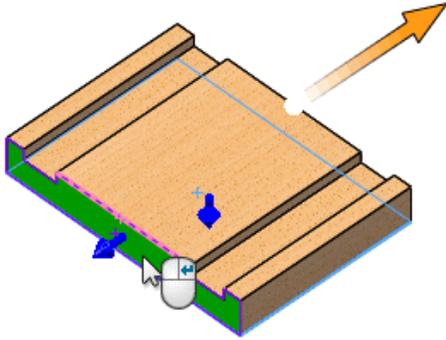
6. Right-click to add the connector to **Connecting Points** or click **Add Connector**.



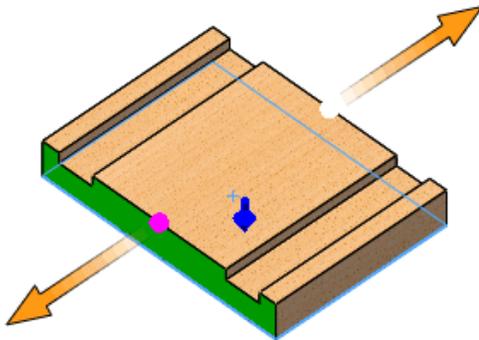
7. Rotate the part and select the edge shown for **Connect point**.



8. Select the face shown for **Connect direction**.



9. Click **Add Connector**.



10. Click ✓.

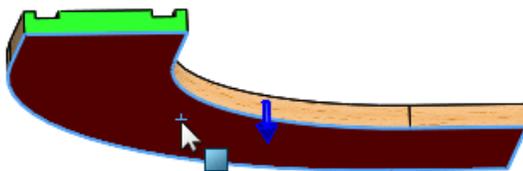
The FeatureManager design tree updates to include **Published References** containing **Ground Plane**, **Connector1**, and **Connector2**.

Publishing the Second Asset

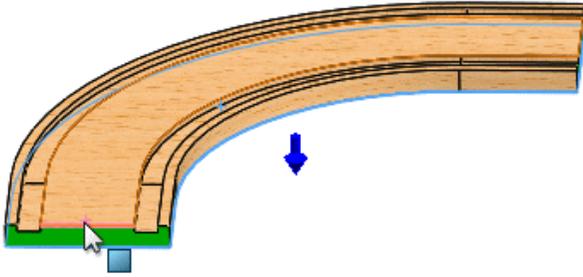
In this step, you use a curved train track to define the ground plane and add connection points for the magnetic mates. Additionally, you select a different configuration.

To publish the second asset:

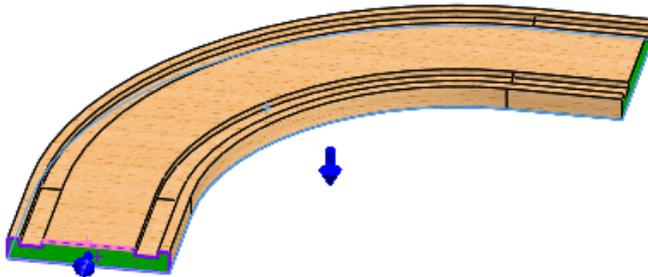
1. Open *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\assemblies\turn.SLDPRT.
2. Click **Tools** > **Asset Publisher** .
3. In the graphics area, select the face shown for **Ground Plane**.



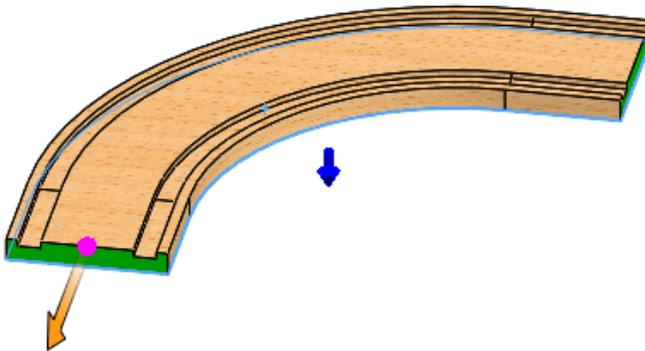
4. Select the edge shown for **Connect point** under **Connecting Points**.



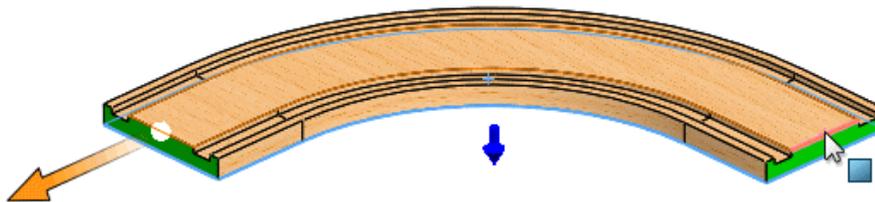
5. Select the face shown for **Connect direction**.



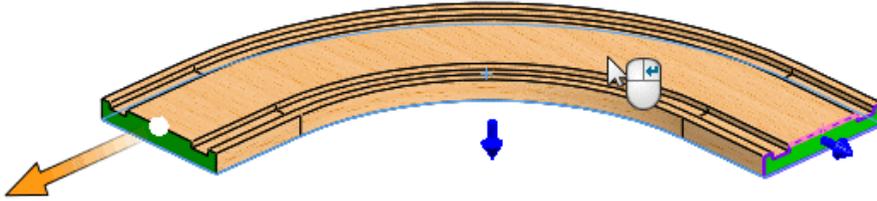
6. Right-click to add the connector to **Connecting Points**.



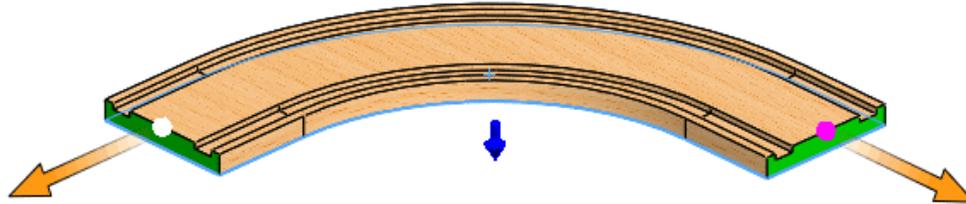
7. Rotate the part and select the edge shown for **Connect point**.



8. Select the face shown for **Connect direction**.



9. Click **Add Connector**.



10. Click ✓.

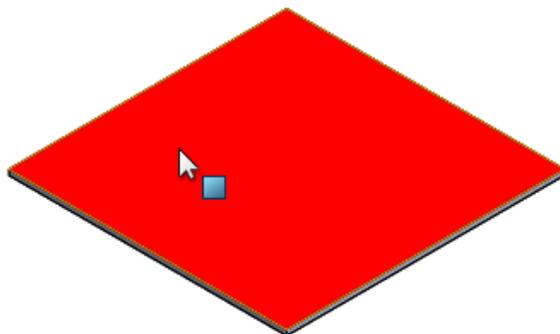
The FeatureManager design tree updates to include **Published References** containing **Ground Plane**, **Connector1**, and **Connector2**.

Creating a Ground Plane

In this step, you use an assembly that will be the floor for the train tracks and define its top face as the ground plane.

To create a ground plane:

1. Open *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\assemblies\playground.SLDASM.
2. Click **Insert** > **Reference Geometry** > **Ground Plane** .
3. In the graphics area, click the top face for **Ground Plane**.



4. Click ✓.

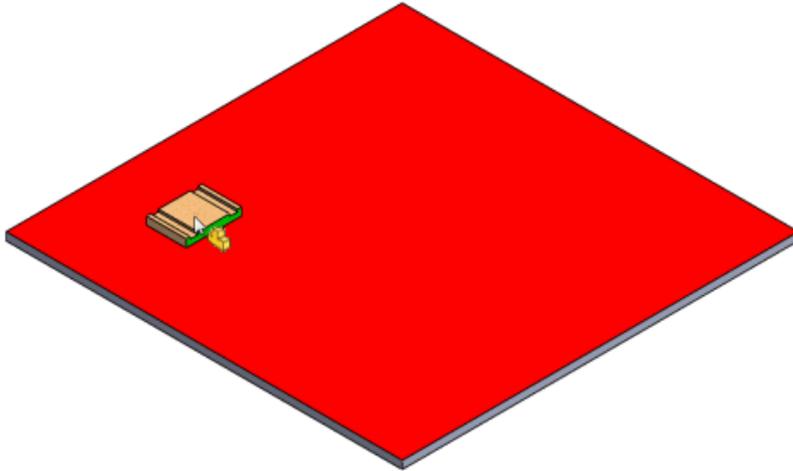
The FeatureManager design tree updates to include **Ground Plane**.

Inserting the Assets into the Assembly

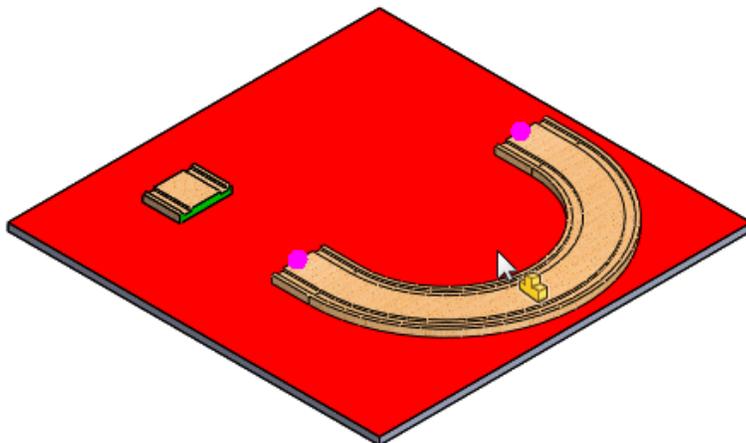
In this step, you insert the train tracks into the floor assembly and select an alternate configuration.

To insert the assets into the assembly:

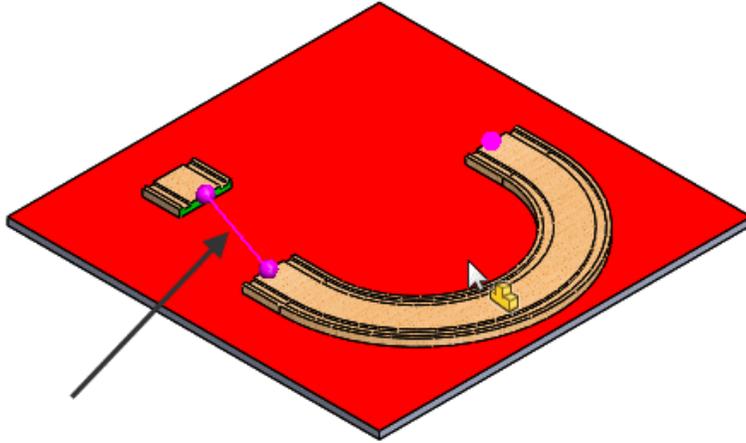
1. Click **Insert Components**  (Assembly toolbar) or **Insert > Component > Existing Part/Assembly**.
2. In the PropertyManager, in **Open documents**, select **straight_track** and drag the pointer with the attached track to the graphics area as shown.



3. Click to place the track on the floor.
The track mates to the floor because ground mates are included.
4. In the PropertyManager:
 - a) In **Open documents**, select **turn**.
 - b) Under **Configuration**, select **0 (U turn)**.
 - c) Drag the pointer with the attached track to the graphics area as shown, but do not release it.

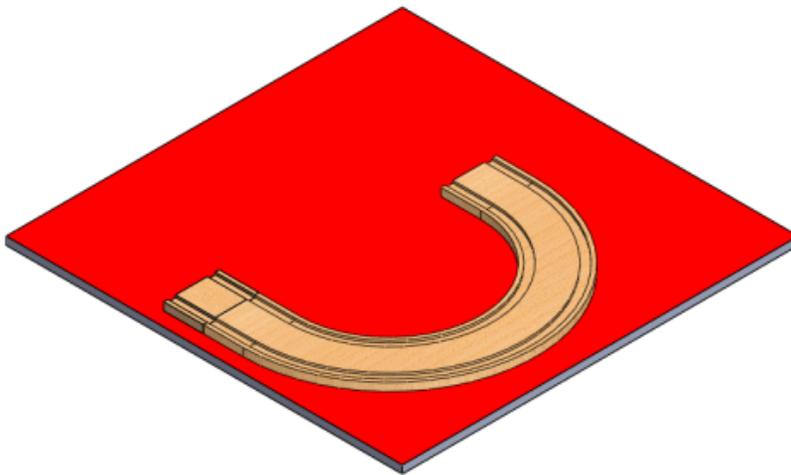


5. Drag the curved track toward the straight track and notice the magnetic mate snap line that appears as you drag the curved track closer to straight track.



If you release the curved track when the snap line is displayed, the tracks will mate.

6. Click to release the curved track so that it snaps to the straight track.

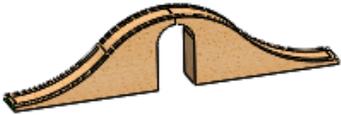
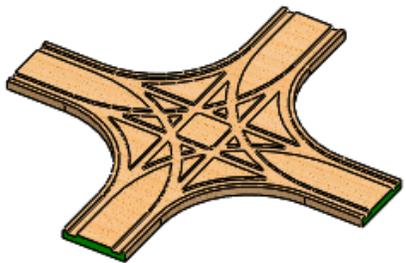
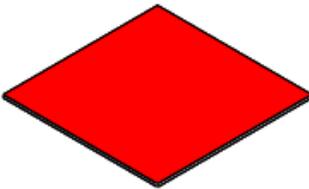


Magnetic Mates Toy Chest

Use this set of toy train components to experiment with magnetic mates.

In addition to the straight and curved tracks that you use in the magnetic mates example, there are additional parts and assemblies you can use to try the magnetic mates functionality. These components are located in *drive letter:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\assemblies*.

Component	Description	File Name
	Straight track Configurations: 50, 100, 150, 200, 250, 300	straight_track.SLDPRT

Component	Description	File Name
	Curved track Configurations: 90°, 120°, U turn	turn.SLDPRT
	Station	station.SLDPRT
	Bridge	bridge.SLDPRT
	Cross track	cross.SLDPRT
	Playground	playground.SLDASM
	Bridge layout	bridgelayout.SLDASM

Support for Path Mates

Path mate (Distance Along Path, Percent Along Path) is now a supported mate type.

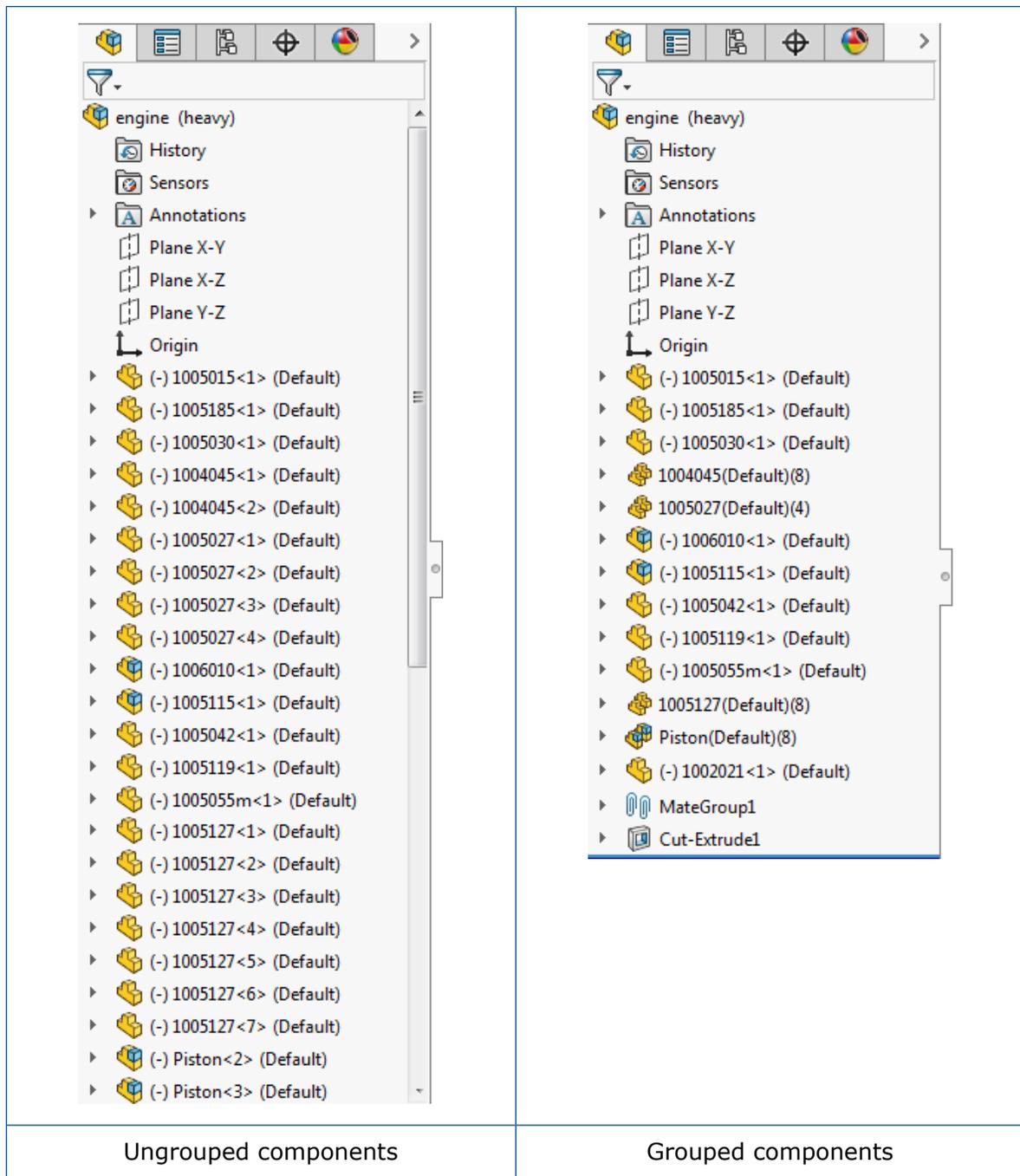
The Mate Controller does not support path, width, or slot mates created with the **Free** constraint.

Grouping Components

In the FeatureManager design tree, you can automatically group the same components with the same configuration into a folder-like structure.

Grouping can greatly condense the length of the FeatureManager design tree and make it easier to find components, especially in large assemblies.

In the FeatureManager design tree, right-click the top-level assembly and click **Tree Display > Group Component Instances**.



Improved Shortcut Menu for Configurations

In the ConfigurationManager, when you right-click the top node of a part or assembly, the shortcut menu lists only the most commonly used tools. Unnecessary tools have been removed.

The following tools are displayed:

- **Add Configuration...**

- **Configuration Publisher**
- **Purge Unused Features**
- **Comment**
- **Tree Display**
- **Tree Order**
- **Collapse Items**
- **Customize Menu**

Loading Documents in Memory Only

You can load referenced documents in memory only, rather than opening them in separate windows.

The option, **Load documents in memory only**, lets you keep references up to date without opening windows for documents that you do not need to see. This option is useful when you open an assembly containing many component parts that have external references.

Click **Tools > Options > System Options > External References** and select **Load documents in memory only**.

Mate Controller Enhancements

Creating Configurations from Positions

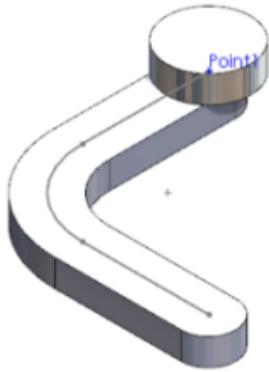
You can create a configuration from a position using the Mate Controller.

This example uses a path mate as the mate type.

To create configurations from positions in Mate Controller:

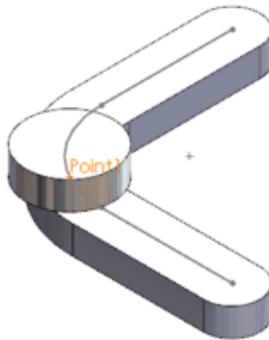
1. Open *drive letter:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\assemblies\mate_controller\path_mate_config.sldasm*.
2. Click **Mate Controller**  (Assembly toolbar) or **Insert > Mate Controller**.
3. In the PropertyManager, under **Mates**, click **Collect All Supported Mates** .

4. Leave **Position 1** at **0.00**mm and click **Add Configuration** .

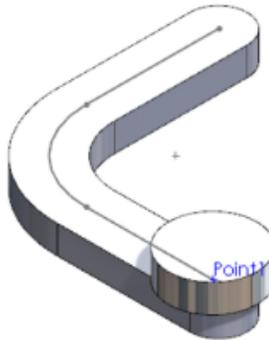


Add Configuration  automatically adds the model to the ConfigurationManager, without you having to add more information.

5. Add **Position 2** by doing the following:
 - a) In **PathMate1**, enter 70.00mm.
 - b) Under **Mate Positions**, click **Add Position**  and click **OK**.
 - c) Click **Add Configuration** .



6. Add **Position 3** by doing the following:
 - a) In **PathMate1**, enter 140.00mm.
 - b) Under **Mate Positions**, click **Add Position**  and click **OK**.
 - c) Click **Add Configuration** .



7. Adjust **Position 2** by entering **80.00**mm and click **Add Configuration** .
8. In the Mate Controller Notification dialog box, click **Update**.
Updates automatically propagate in the ConfigurationManager.
9. Click .

Mates

Better Component Placement While Mating

When adding concentric or coincident mates to components that have all degrees of freedom unconstrained, components are placed in more logical positions on the screen and relative to other components.

In previous releases, the coincident or concentric mate was added correctly, but one component was often placed off screen or hidden behind or within other components. Then, to continue your work, you needed to zoom in and out, rotate the view, or hide other components.

In SOLIDWORKS 2017, to improve the placement of the components being mated, the mate software takes into account where you click on each of the components to mate. The components are placed so that those clicked regions are near each other. The software also avoids or minimizes interference with other components.

Distance Mates to Cylindrical Components

When you add a distance mate between two cylindrical faces, four options are available for placement of the distance.

The new options are also available when you add a distance mate between a cylindrical face and an axis, edge, line, vertex, or point.

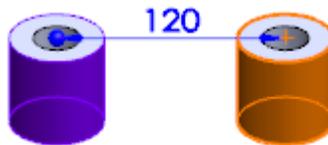
Initially, the default option is **Center to Center**. Subsequently, the software remembers your last used option from session to session.

Previously, the specified distance was applied at the minimum distance between the cylinders and no other options were available.

The distance placement options include the following:



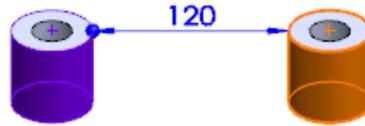
Center to Center Applies the distance between the cylinder axes.





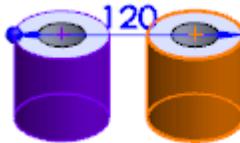
Minimum Distance

Applies the distance where the cylinders are closest to each other.



Maximum Distance

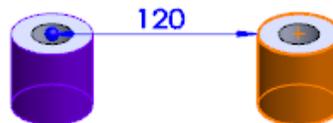
Applies the distance where the cylinders are farthest from each other. This option is not available if the distance is less than the sum of the radii of both cylinders.



Custom Distance Applies the distance to the combination of conditions you select. Specify a condition for each cylinder:

- **Center**
- **Min**
- **Max**

For example, you can select **Center** for the cylinder on the left and **Min** for the cylinder on the right.



The lower limit for the applied distance is calculated based on the selected options.

Max and Center The applied distance has a lower limit that is equal to the radius of the cylinder that has **Max** selected.

Max and Min The applied distance has a lower limit that is equal to one of the following radii:

- **Max** radius minus the **Min** radius if the **Max** radius is greater than the **Min** radius
 - **Min** radius
-

Driven Mate Dimensions

You can set mate dimensions as driven, so that the value is influenced by other moving components in the assembly.

By default, mate dimensions are driving dimensions. For the following mate types you can change the dimensions to driven:

- Distance and distance limit
- Angle and angle limit
- Width (Distance and Percentage options)
- Slot (Distance and Percentage options)
- Path (Distance and Percentage options)

To use driven mate dimensions:

Do one of the following:

- In the graphics area, right-click the mate dimension and click **Driven**.
- In the FeatureManager design tree, in the Mates folder, right-click the mate and select **Driven**.
- In the graphics area, select the mate dimension. Then in the PropertyManager, on the Other tab, select **Driven**.

You can change the mate dimension back to driving by repeating one of the methods above and clearing **Driven**.

Over Defined Mates

The dialog box for over defined mates has been redesigned for ease of use.

The options in the dialog box are better defined so you know exactly how the mate will behave:

- **Add this mate and break other mates to satisfy it**
- **Add this mate and over define the assembly**

If you press **ESC**, the mate is not created.

Options to Improve Performance in Large Assembly Mode

When you are working on large assemblies, you can select system options that reduce the number of rebuilds and other time-consuming operations.

Suspend automatic rebuild

You can control the existing option, **Suspend automatic rebuild**, for Large Assembly Mode (LAM).

To suspend automatic rebuilds for LAM:

1. Click **Tools > Options > System Options > Assemblies**.

2. Under When Large Assembly Mode is active, select **Suspend automatic rebuild**.

Verification on Rebuild

You can control the existing option, **Disable verification on rebuild**, for LAM.

To disable verification on rebuild for LAM:

1. Click **Tools > Options > System Options > Assemblies**.
2. Under When Large Assembly Mode is active, select **Disable verification on rebuild**.

Maximum Time to Update Center of Mass and Equations

In an active assembly, operations for the center of mass and equations linked to mass properties are updated only when they take 0.5 seconds or less. If the update takes more than 0.5 seconds, the center of mass and equations are marked with , and are not updated until you initiate a rebuild manually.

Preserving Drawing References

Drawing references (such as dimensions, notes, and annotations) are maintained when you replace a subassembly with a multibody part created from the subassembly.

Searching for Externally Referenced Documents

When you open assembly or drawing documents, new search options provide more control over where the software searches for missing referenced documents.

The options also help to reduce the opening time by reducing the number of folders to be searched.

To search for externally referenced documents:

1. Click **Tools > Options > External References**.

- Under **Search external references in**, select or clear options.

Option	Description
Referenced Documents specified in File Locations	Searches for missing referenced documents in the Referenced Documents folders specified in System Options > File Locations . Otherwise, the standard recursive search routine is used.
Include	Searches subfolders of the Referenced Documents folders.
Exclude active folders and recent save locations	Blocks the software from searching open folders and folders where you have recently saved items.
Go To Referenced Documents	Opens the File Locations section of the Options dialog box.

Sorting Components by Their Opening Time

A property stores the time taken to open each of an assembly's components the last time you opened the assembly. You access the property, **SW-Open Time**, in **Assembly Visualization**.

The property shows you which components take the longest to open. You could then simplify or remove those components to decrease the time needed to open the assembly.

To sort components by their opening time:

- In an assembly, click **Assembly Visualization**  (Tools toolbar) or **Tools > Evaluate > Assembly Visualization**.
- On the **Assembly Visualization** tab , click the arrow  to the right of a column header.
- On the flyout list, click **More**.
- In the **Custom Column** dialog box, in **Properties**, select **SW-Open Time**.
- Click **OK**.

SW-Open Time becomes the column header, and the opening time for each component appears.

- Click the column header to sort the components by opening time. Click again to reverse the sort order.

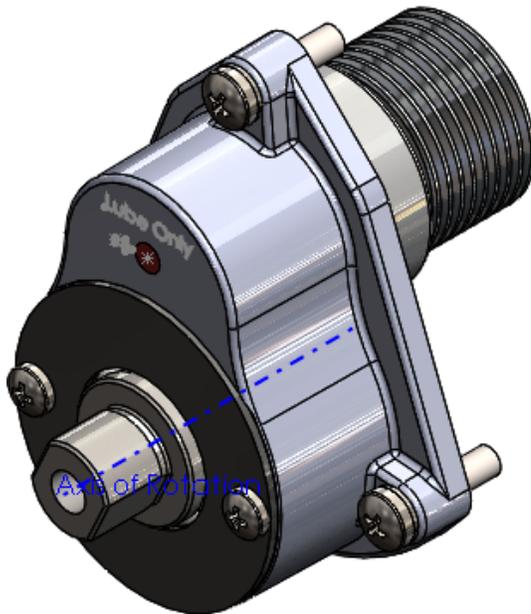
SpeedPak Enhancements

You can include reference geometry and curves in a SpeedPak. You can select sketches from any level of the assembly to include in a SpeedPak.

Previously, all top-level unabsorbed sketches were included in the SpeedPak. You had to suppress top-level sketches that you wanted to exclude, and you could not select sketches in subassemblies.

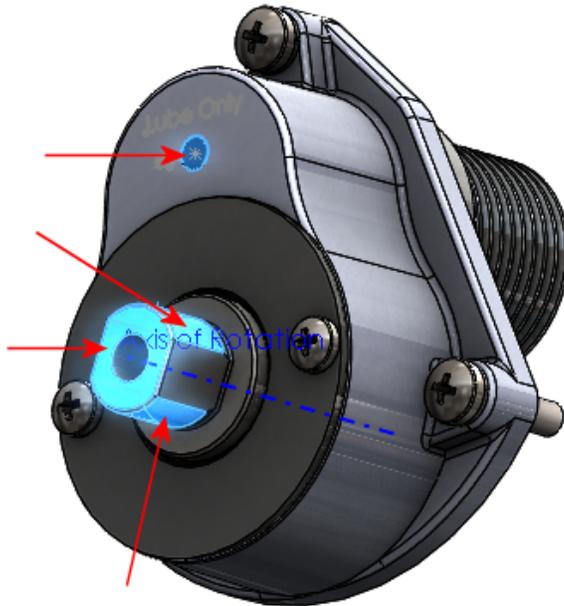
To include reference geometry and sketches in a SpeedPak:

1. Open *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\assemblies\speedpak\gearbox.sldasm.



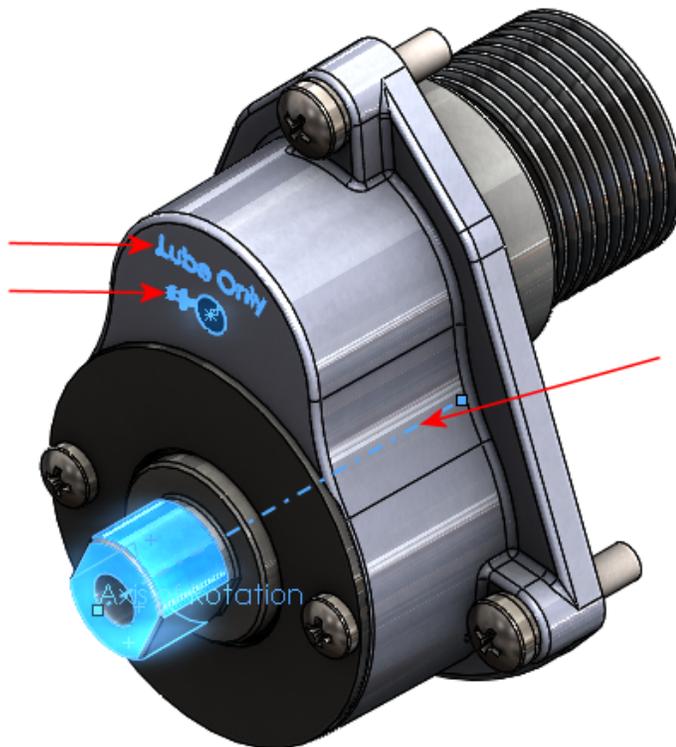
2. On the ConfigurationManager  tab, right-click the **Default** configuration and click **Add SpeedPak**.

3. In the graphics area, select the four faces shown for **Faces to Include** .



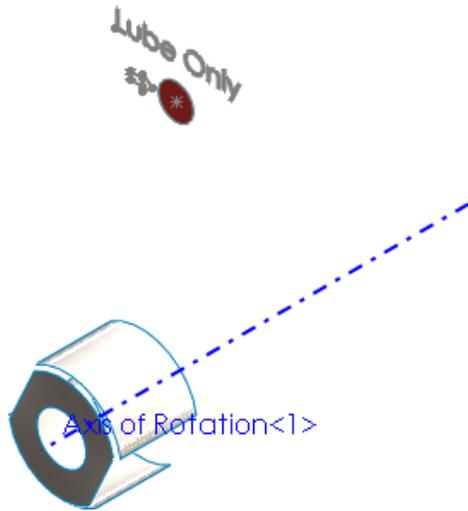
Three of the faces are on the **SHAFT - DRIVE** component.

4. In the PropertyManager, under **Include**, click in **Reference Geometries, Sketches, and Curves to Include**   .
5. In the graphics area, select the **Axis of Rotation**, **Lube Only** text, and **Lube Arrow** sketch shown.



6. In the PropertyManager, under **Options**, select **Remove ghost graphics**.

7. Click .
The sketches and axis are included in the SpeedPak.



Treehouse Improvements

Including Drawings When Importing Structures

When you import a structure into Treehouse, you can also import any associated drawing files.

To include drawings when importing structures:

1. In Treehouse, click **Treehouse Options**.
2. In the dialog box:
 - a) Under **Import options**, select **Import drawings with models**.
 - b) Under **Search in folder locations**, click **Add**, browse to a folder, and click **OK**.
 - c) Click **OK**.

The next time you import a Treehouse structure that contains a drawing, the drawing node is included in the structure.

Opening Structures in Microsoft Excel

You can open Treehouse structures in Microsoft Excel. This helps you to use a spreadsheet to capture the parts and assemblies that comprise a model.

To open structures in Microsoft Excel:

1. Open a structure in Treehouse.
2. Click **Open in Excel** .

3. In the dialog box, select an option:
 - **Open Structure.** Opens the entire Treehouse structure in Microsoft Excel. Parts and assemblies appear in an indented list.
 - **Open Parts Only.** Opens only parts from a Treehouse structure in Microsoft Excel.
4. Click **OK**.

Microsoft Excel opens with a spreadsheet of the entire structure or parts. The columns included in the spreadsheet are **Document Name**, **Active Configuration**, and **Quantity**. Each file property is displayed in its own column. The **Quantity** value for duplicate parts is shown for each instance in the structure.

Printing Structures

You can print Treehouse structures on paper or to PDF.

In Treehouse, click **Print**  and set options in the dialog box. The system prints to the extents of the structure.

User Interface

User interface improvements include:

- Redesigned interface consistent with the SOLIDWORKS software user interface.
- Removed animations when you click **Show Document Properties**  on a node.
- Changed icons for drawings. Click  to minimize a drawing. Previously you had to click  to minimize display of a drawing in the structure. This icon change eliminates confusion about what happens to the drawing reference.
- Editable document names in the structure. Double-click any document name to edit the text. Previously you had to click **Show Document Properties** to change a document name.

7

SOLIDWORKS Composer

This chapter includes the following topics:

- **SOLIDWORKS Composer**
- **SOLIDWORKS Composer Player**

SOLIDWORKS Composer

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

Advanced Features Available in Composer Built-in Samples

Advanced features are available in Composer built-in samples without the Player Pro license.

A Composer Player Pro license is necessary to use advanced features in published Composer files. However, if you are using a Composer built-in sample, advanced features are now available even without the Player Pro license. When opening a Composer HTML file without a Player Pro license, an information message appears to inform you about this. See *About the HTML Output* in the Composer help.

Enhancement of Translate and Scale Motions

The Translate and Scale motions of the Animation Library workshop have been enhanced to support mirroring of animations.

See *Animation Library Workshop* in the Composer help.

Mirror Animation

You can now mirror animations.

Technically, this reproduces in inverted order the keys of selected animation blocks. For example, mirroring a Screw step creates an Unscrew step. See *Creating Pre-defined Animations* and *Custom Animation XML Example* in the Composer help.

ProfilesLoadOnDemand

This new advanced preference lets you specify whether workshop profiles should be loaded on demand, or all at once.

Loading profiles on demand only can be useful when many profiles are defined, or for profiles stored on a network. See *Advanced Settings* in the Composer help.

Loading of profiles on demand was the existing behavior for all workshops that use profiles, except for the Technical Illustration and the Styles workshops (for which all profiles used to be loaded when selecting one in the Profile list). This new preference makes the loading of profiles homogeneous throughout all workshops.

Sorting Profiles Alphabetically or By Category

Two new options are now available for workshop profiles.

You can specify whether profiles should be sorted alphabetically or by category of location, that is Composer default path profiles, user-defined path profiles, or other path profiles (such as network locations). See *About Profiles for Workshop Settings* in the Composer help.

SOLIDWORKS Composer Player

Advanced Features Available in Composer Built-in Samples

Advanced features are available in Composer built-in samples without the Player Pro license.

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8

SOLIDWORKS Costing

This chapter includes the following topics:

- **Bounding Box Nesting Preview for Last Sheet**
- **Feature Recognition for Operations**
- **MySOLIDWORKS Costing Template Integration**
- **Rules-based Costing for Parts**
- **Subassembly Costing**
- **Template Enhancements**
- **Time-based Support for Costing**

SOLIDWORKS Costing is available in SOLIDWORKS Professional and SOLIDWORKS Premium. Costing for assemblies is only available in SOLIDWORKS Premium only.

Bounding Box Nesting Preview for Last Sheet

You can view the bounding box nesting layout on the last sheet when you have multiple sheets in the lot and you can change between the first and last page. You can also resize and scale the Bounding Box Nesting Preview while maintaining the original aspect ratio.

To change between the first and the last page when you have multiple sheets, in Bounding

Box Nesting Preview , click **First Page** or **Last Page**.

Feature Recognition for Operations

The performance of feature recognition to handle special geometry cases is enhanced. New tools and workflow improvements are introduced to streamline and speed up the costing process.

The following new tools are added:

- **Split operation.** To split a coaxial hole, right-click the entity in the CostingManager and click **Split Operation**.
- **Combine operations.** To rejoin split holes, right-click either of the resulting holes generated from the **Split Operation** in the CostingManager and click **Combine Operations**.
- Turning features. You can convert an ID Turn operation to a hole and automatically assign threading operations if the information exists in the template. No **Convert** operation is necessary.

- Facing operation. You can convert a face mill operation to an end cutting operation for block, cylinder, and plate stock bodies by right-clicking the entity and clicking the option.
- Boring operation. You can right-click a **Simple Hole** to add a boring operation and then select a tool.

Multi-selection capability is provided in the CostingManager for the following features and operations in single body mode:

- Hole features. Convert to circular pockets, splitting hole chains, and Move To.
- Hole operations. Select tools for drill, counter-bore, counter-sink, tap operations, and additional hole operations.
- Mill features. Convert to holes or volume features, and Move To.
- Mill operations. Select tools for mill operations, additional operations, and remove additional operations.
- Volume features. Change machines, processing, convert to mill, drill, or turn tools, and Move To.
- Volume operations. Select tools for drill, mill, or turn.
- Facing features. Convert facing to end cut features, and Move To.
- Facing operations. Select a common tool for facing operations.
- End cut features. Apply Run No Cost, Apply Cost Override, Edit Cost Override and Remove Cost Override. Convert end cut features to facing features, and Move To.
- Machining Cut Path features. Apply Run No Cost, Apply Cost Override, Edit Cost Override, and Remove Cost Override. Select cut path features and apply a cutting machine.
- Sheet metal bend features. Select edge and hem bend and apply a cutting feature.
- Turning features. Select turning features and apply a machine, select ID turning features and apply a drill tool, Move To.

You can add stock in **X**, **Y** and, **Z** directions of **Plate** stock body types similar to the **Block** stock body type.

You can apply part settings for an assembly in the CostingManager instead of modifying them in the Costing Task Pane.

Cost Estimation Progress Dialog Box Enhancements

You can view the Cost Estimation Progress dialog box while costing machining and sheet metal parts. You can click **Skip this part** button in the Progress dialog box while costing assemblies.

- Cost Estimation Progress dialog box and **Cancel** button. The Cost Estimation Progress dialog box is available for single body parts. When a single body part is costed, you can view the **Cost Estimation Progress** dialog box that includes a progress bar and a **Cancel** button. You can use the **Cancel** button to abort the Costing process for a single operation or multiple operations. When you cancel a process:
 - The Task Pane and CostingManager remain active.
 - You can return to the costing operation for the cancelled operation at a later stage.
- The canceling process for single body parts, multi-body parts, and assemblies is faster.
- **Skip this part**. This button is in the Cost Estimation Progress dialog box while costing assemblies. You can use this to skip the costing process for time-consuming components and move on to the next component in the queue. The skipped components remain

uncosted with no cost value assigned. This button provides a convenient way to skip or cancel one or more components as opposed to aborting the entire costing process for the assembly.

MySOLIDWORKS Costing Template Integration

Manufacturers on the MySOLIDWORKS Manufacturing Network can upload their sheet metal templates so that users can access them in the SOLIDWORKS software. Users seeking to cost their sheet metal parts can also view the manufacturer information and contact them.

The Costing Task Pane for sheet metal is modified to include two new options under **Costing Template**:

- **Local**. Displays templates saved on your system.
- **MySolidWorks**. Displays manufacturers' templates from MySolidWorks Manufacturing Network.

Costing with Manufacturer Information on MySolidWorks

To cost with manufacturer information on MySolidWorks:

1. In a sheet metal part, in the Costing Task Pane, under **Costing Template**, click **MySolidWorks**.
2. Under **Manufacturer**, select a manufacturer from the drop-down list.
3. Under **Template**, select a template from the drop-down list.
4. Click **Begin Cost Estimation**.

Rules-based Costing for Parts

You can customize machining and sheet metal templates to more accurately reflect the manufacturing process for creating parts. You can use unique sheet metal rules, unique machining rules, or common sheet metal and machining rules for handling special geometry cases.

Previously, you could only customize machining templates for rules-based Costing.

You can use additional rules for machining and also customize sheet metal templates for special geometry cases.

Common Rules for Sheet Metal and Machining Parts

The list of Rules visible under the Rules tab common to sheet metal templates and machining templates are:

Option	Description
IF a material is chosen, THEN add a cost	Sets a predefined cost for the chosen material.

Option	Description
IF a material is chosen, THEN add a cost as % of calculated cost	Sets a cost as a percentage of the total, material, or manufacturing cost for the selected material.
IF a material is chosen, THEN add a markup/discount to the total/material cost	Sets a markup/discount as a percentage of the total or material cost for the selected material.
IF a material is chosen, THEN add a custom operation	Sets a custom operation for the selected material.
IF model weight is a certain size, THEN add a cost	Sets a cost if the model weight is between two predefined weight values.
IF model weight is a certain size, THEN add a cost as % of calculated cost	Sets a cost as a percentage of the total, material, or manufacturing cost, if the model weight is between two predefined weight values.
IF model weight is a certain size, THEN add a markup/discount to the total/material/manufacturing cost	Sets a markup/discount as a percentage of the total or material cost, if the model weight is between two predefined weight values.
IF the longest edge of the stock bounding box is a certain size, THEN add a custom operation	Sets a custom operation if the longest edge of the stock machining bounding box (X, Y, or Z axis) is between two predefined length values.
IF the longest edge of the stock bounding box is a certain size, THEN add a cost	Sets a cost if the longest edge of the stock bounding box (X, Y, or Z axis) is between two predefined length values.
IF the longest edge of the stock bounding box is a certain size, THEN add a cost as % of calculated cost	Sets a cost as a percentage of the total, material, or manufacturing cost, if the longest edge of the stock bounding box (X, Y, or Z axis) is between two predefined length values.
IF the longest edge of the stock bounding box is a certain size, THEN add a markup/discount to the total/material cost	Sets a markup/discount as a percentage of the selected total or material cost, if the longest edge of the stock bounding box (X, Y, or Z axis) is between two predefined length values.
IF a part weight is between X and Y Weight, THEN add a custom operation	Sets a custom operation, if a part weight is between two predefined values.

Unique Rules for Machining Templates

The list of Rules unique to machining templates are:

Option	Description
IF milling operation is selected THEN multiply Step Over Percentage of feedrate (f)	Multiplies the step-over percentage of feedrate by a predefined value, if you select a milling operation.
If DimXpert Hole tolerance is in certain range then add additional operation	Performs an additional operation if a DimXpert Hole tolerance is between two predefined tolerance values.
USE the plate stock body type IF the bounding box is a certain size	Uses the plate stock body type if the smallest dimension of a part bounding box is less than a predefined weight value.

Unique Rules for Sheet Metal Templates

The Rule unique to sheet metal template is:

Option	Description
IF model weight is a certain size, THEN use this bend operation	Sets a bend setup operation from the available bend setup operations for the part, if its weight is between two predefined values.

Subassembly Costing

While costing an assembly, you can either use a **Flat Tree** mode or a **Nested Tree** mode. You can also cost subassemblies or apply purchased cost to them.

The two modes of Costing are:

Flat Tree	Costs top assemblies without costing subassemblies. The Flat Tree option costs assemblies as per the existing behavior.
Nested Tree	Costs the top assembly and its subassemblies or applies purchased cost to subassemblies.

With assembly costing, you can:

Apply Purchased Cost	You can cost the assembly in Nested Tree mode and apply cost overrides to individual subassemblies by using the option Use Purchased Cost on the shortcut menu. If you apply purchased cost to any subassembly then all components in this subassembly are greyed out in the CostingManager.
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Apply component cost changes to multiple instances across the whole assembly.

When you change the cost of a component having multiple instances within or outside of subassemblies, the cost change is applied across all instances in the assembly.

Costing preference. Following is the order of preference for Costing for an uncosted assembly:

1. **Custom property cost.** If a custom property cost is defined for a subassembly or component and the custom property is defined in the Costing template, then this cost is considered over the cost defined in the template.
2. **Calculated cost.** If both, custom property cost and cost in template are not defined, then the costs are calculated individually. If a subassembly is already saved with Costing data, then saved cost is considered over other type of costs.

Options in CostingManager:

Option	Description
Use Cost from Template	Applies cost defined in template
Use Purchased Cost	Applies cost override on existing cost
Use Custom Property Cost	Applies cost defined as a Custom Property.
Use Calculated Cost	Applies calculated cost of components and subassemblies

Evaluating the Cost of Assemblies in Nested Tree Mode

To evaluate the cost of subassemblies in an assembly:

1. Open *drive Letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\Costing\PunchingMachine\PunchingMachine.sldasm.
2. Copy the template `Punch data.sldctc` from `\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\Costing\PunchingMachine` to `C:\ProgramData\SOLIDWORKS\SOLIDWORKS 2017\lang\english\Costing templates`.
3. Click **Tools > Options > File Locations** and do the following:
 - a) Under **Show folders for** select **Costing Templates** from the drop-down list.
 - b) Click **Add** and provide the file path `C:\ProgramData\SOLIDWORKS\SOLIDWORKS 2017\lang\english\Costing templates`.
4. Click **Tools > SOLIDWORKS Applications > Costing** .
5. In the Costing Task Pane, under **Costing Templates**, in **Main template**, select the template `Punch data`.

6. Under **Part List**:
 - a) Click **Nested Tree**.
 - b) Select parts for:
 - **Costs to be calculated**
 - **Cost Defined**

You can view only parts and not assemblies in the above sections.

7. To set or change a predefined cost, select the part in **Cost defined** and type the cost.
8. Under **Toolbox Parts**, select parts and set one of the following:
 - **Exclude**. Excludes the selected bodies from cost calculations.
 - **Include**. Includes the selected bodies for cost calculations.
9. Click **Begin Cost Estimation**.
10. Click **Yes**.

When you click **Yes**, the subassemblies that are saved with a cost calculated with a flat tree method may change.

11. In the CostingManager, right-click the subassembly cost and click one of these options on the shortcut menu:
 - **Use Cost from Template**
 - **Use Purchased Cost**
 - **Use Custom property Cost**
 - **Use Calculated Cost**

You can change between **Use Cost from Template/ Use custom property Cost**, **Use Purchased Cost** and **Use Calculated Cost** options. If a custom property is defined for a subassembly, then the option **Use Cost from Template** is not visible on the shortcut menu.

The assembly cost is updated.

Template Enhancements

Microsoft Excel Template for Reports

You can customize the Costing Excel report template to rearrange the position of specific input tags in the report. You can also create multiple Microsoft Word or Excel templates and store them in one location.

Previously, you could customize only the Word report template. You can customize the Excel report template and also add your own styling.

You can also:

- Select a template in the defined location from **Report Options**.

- Open an Excel report template when:
 - A Word report template is open.
 - Another Excel template is open.
 - Both Word and Excel templates are open.

Multi-quantity Quote Support in Report Templates

You can provide five alternate quantities of cost values in a Microsoft Excel or Word report while costing a part to show the customer how the cost per part may reduce as the manufacturing quantities are increased.

You can choose between two report types:

- **Simple Estimate Report.** Covers the following information:
 - Model name
 - Company information
 - Quote information
 - Estimated time per part
 - Estimated cost per part
 - Alternate quantities
- **Detailed Report.** Covers the information covered in a Simple Quote Report and also the following:
 - Cost breakdown
 - Manufacturing cost breakdown - Setup Costs and Setup Operations Costs (parts only)
 - Setup operations (parts only)
 - No cost features (parts only)
 - Baseline sections (parts only):
 - Manufacturing cost breakdown - Setup Costs and Setup Operations Costs
 - Setup operations
 - No cost features

Multiple Instances of the Costing Template Editor

You can open a second instance of the same template in the Costing Template Editor in read-only mode.

Previously, you received a warning message if you tried to open another instance of the same template in the Costing Template Editor.

This functionality is useful when you modify information in a tab in the Costing Template Editor and you want to compare the information.

Sorting Tables in the Costing Template Editor

You can sort tables alphabetically in the Costing Template Editor by clicking the column headers once.

Previously you had to double-click column headers to sort the tables.

To sort tables in the Costing Template Editor:

1. In a sheet metal or machined part, click **Costing**  (Tools toolbar) or **Tools > SOLIDWORKS Applications > Costing**.
2. In the Task Pane, under **Costing Template**, click **Launch Template Editor**.
3. In the Costing Template Editor, click any tab with columns and click any column header.

The table is alphabetically sorted according to the selected column.

Using Copy As and Modify in Costing Templates

You can use the **Copy As** and **Modify** tools for some Operations tabs in sheet metal and machining templates, for copying and reusing multiple rows of information.

To access Copy As and Modify in Costing templates:

1. In a sheet metal or machining part, click **Costing**  (Tools toolbar or **Tools > SOLIDWORKS Applications > Costing**).
2. In the Costing Task Pane, under **Costing Template**, click **Launch Template Editor**.
3. In the Costing Template Editor, under Operations:
 - For sheet metal parts, click Cut or Bend.
 - For machined parts, click Cut, Mill, Drill, or Turn.
4. Select rows, right-click the selected rows and click **Copy As** or **Modify**.

You can manage options from the cell menus in the Operations tabs.

Time-based Support for Costing

You can switch to a time-based Costing method in the sheet metal template for the cut operations. You have more flexibility for data input and you can also calculate the time taken to manufacture a sheet metal part using the input parameters.

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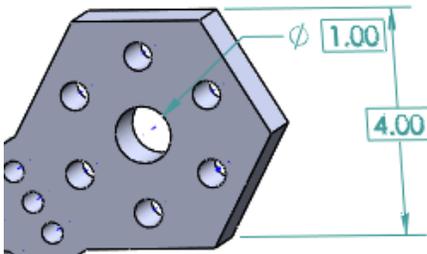
DimXpert

This chapter includes the following topics:

- **Creating Basic Size Dimensions**
- **General Block Tolerance**
- **Multiple Edge Selection**
- **Use of Reference Geometry with DimXpert Dimensions**
- **Using Datums in Auto Dimension Schemes**
- **Using Polar Dimension Schemes for Geometrically Toleranced Parts**

Creating Basic Size Dimensions

You can create basic size dimensions manually.



To create basic size dimensions:

1. In a part, click **Basic Size Dimension**  (DimXpert toolbar) or **Tools > DimXpert > Basic Size Dimension**.
2. Select entities in the graphics area.
3. Place the dimension.
4. In the PropertyManager, set options.
5. Click .

General Block Tolerance

You can include a default tolerance for dimensions when insignificant decimal places are not used to determine tolerance, such as when the unit of measure is metric.

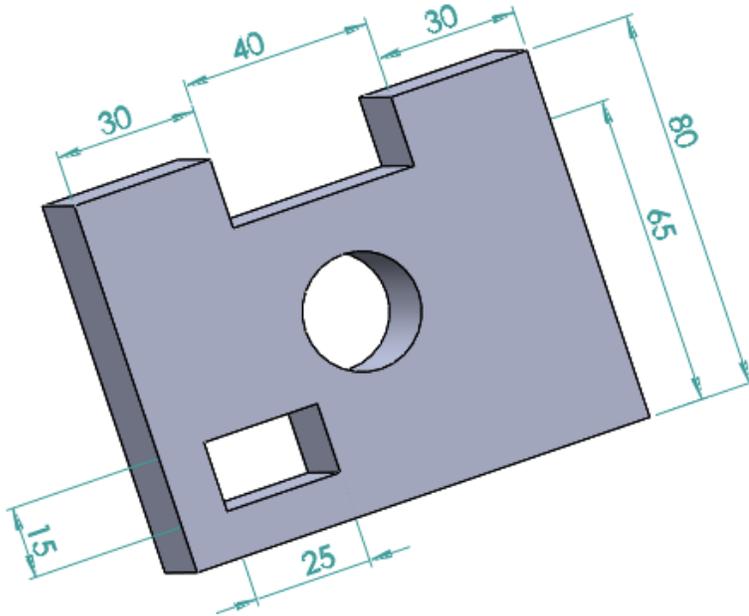
To set the General Block Tolerance option:

1. In a part or assembly, click **Tools > Options > Document Properties > DimXpert**.

2. Under **Methods**, select **General Block Tolerance**.
3. Under **General block tolerance**, set **Length tolerance** and **Angle tolerance**.
4. Click **OK**.

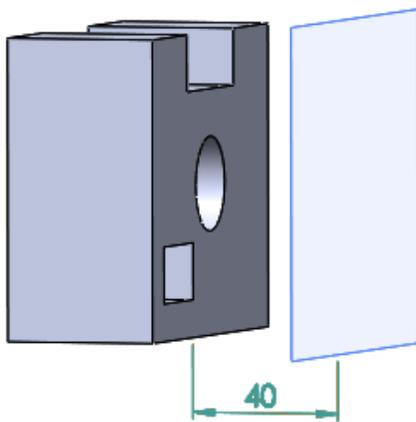
Multiple Edge Selection

You can create dimensions between two planes by selecting two edges when the corresponding planes are parallel.



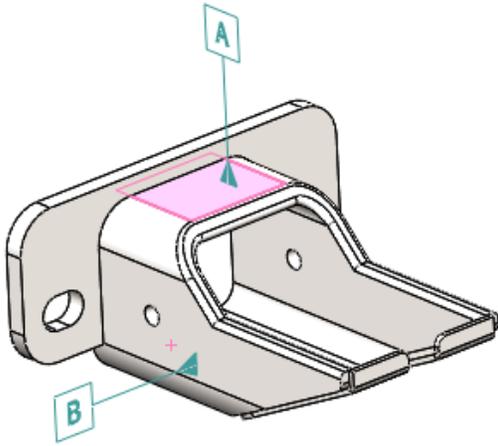
Use of Reference Geometry with DimXpert Dimensions

You can create DimXpert dimensions using reference planes, axes, points, and center of mass symbols.



Using Datums in Auto Dimension Schemes

The DimXpert **Auto Dimension Scheme** tool lets you select existing datums for the reference features.



To use datums in Auto Dimension Schemes:

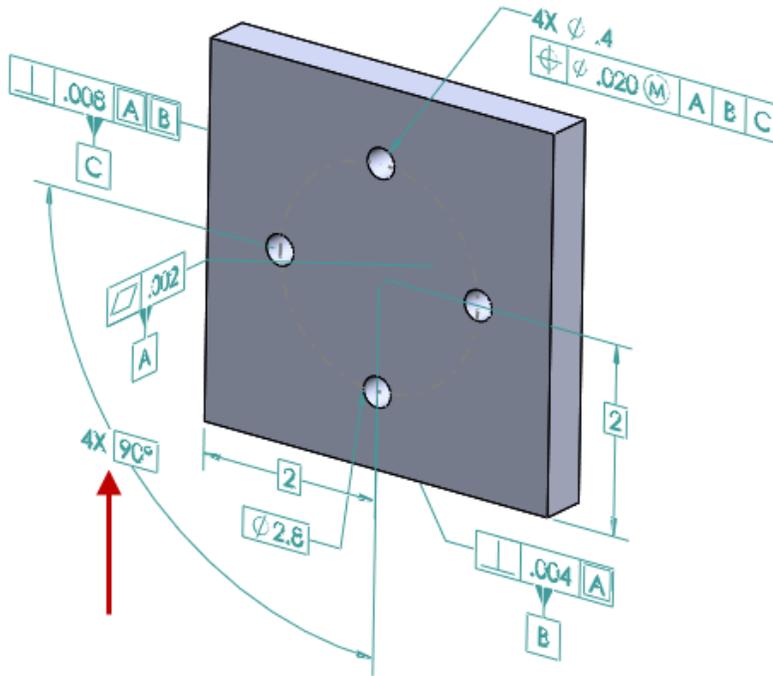
1. Click **Auto Dimension Scheme**  (DimXpert toolbar) or **Tools > DimXpert > Auto Dimension Scheme**.
2. In the PropertyManager, under **Datum Selection**, select the primary datum symbol and optionally select the secondary and tertiary datum symbols to use when generating geometric tolerances.

You can select existing datums.

3. Set options.
4. Click .

Using Polar Dimension Schemes for Geometrically Toleranced Parts

The DimXpert **Auto Dimension Scheme** tool supports polar dimensioning schemes.



To use polar dimension schemes for geometrically toleranced parts:

1. Click **Tools > Options > Document Properties > DimXpert > Geometric Tolerance**.
2. In the dialog box, under **Basic dimensions**, select **Create basic dimensions** and **Polar**.
3. Click **OK**.
4. Click **Auto Dimension Scheme**  or **Tools > DimXpert > Auto Dimension Scheme**.
5. In the PropertyManager, in **Settings**, under **Pattern Dimensioning**, select **Polar**.
6. Set options.
7. Click .

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Detailing and Drawings

This chapter includes the following topics:

- **Dimensions**
- **Drawing Views**
- **Model Detailing**
- **Notes and Balloons**
- **Performance in Drawings**
- **Sheets**
- **Tables**

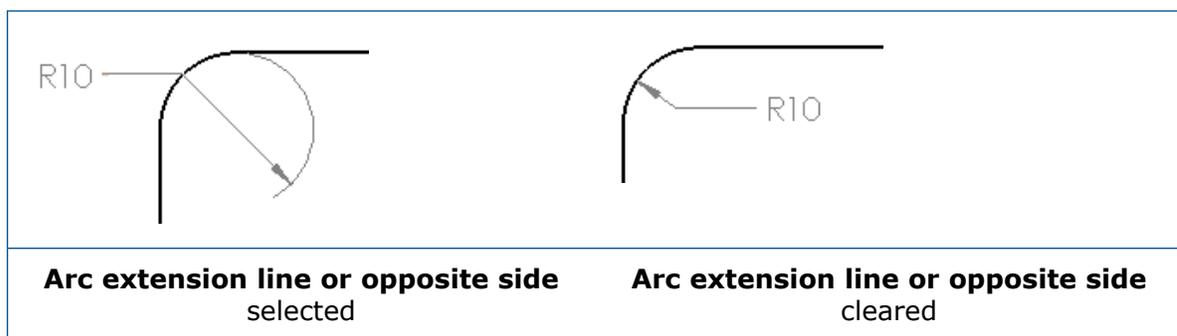
Dimensions

Arc Extension Line or Opposite Side

The **Arc extension line or opposite side** setting lets you control the radial dimension leader.

The **Dimension to inside of arc** option is removed from **Witness/Leader Display** on the Leaders tab of the Dimension PropertyManager. It is replaced by **Arc extension line or opposite side**, which lets you control the radial dimension leader:

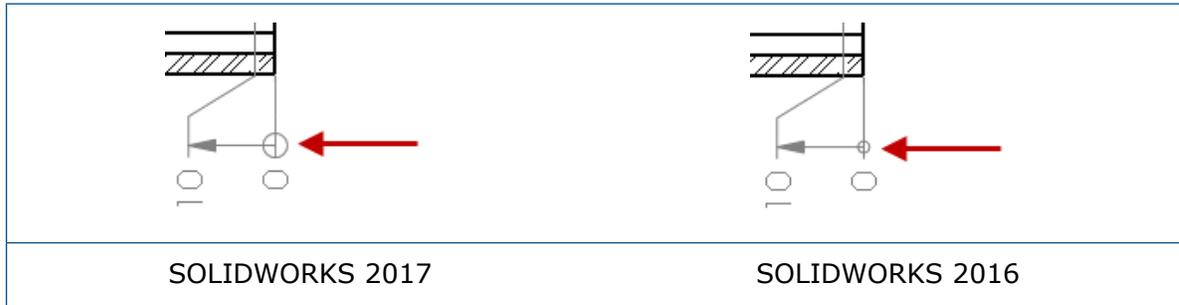
- Where possible, it is applied to an arc extension line.
- If not possible, it is attached to the opposite side of the arc.



Changing the Origin Circle Size in DIN Ordinate Dimensions

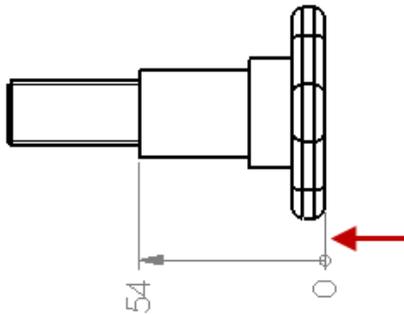
You can control the origin circle size for ordinate dimensions when using the DIN standard.

To support the current DIN 406 standard and previous DIN standards, you can set the origin circle size in ordinate dimensions.



To change the origin circle size in DIN ordinate dimensions:

1. Open *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\drawings\knob-1.SLDDRW.
2. Select the ordinate baseline that contains the origin circle.

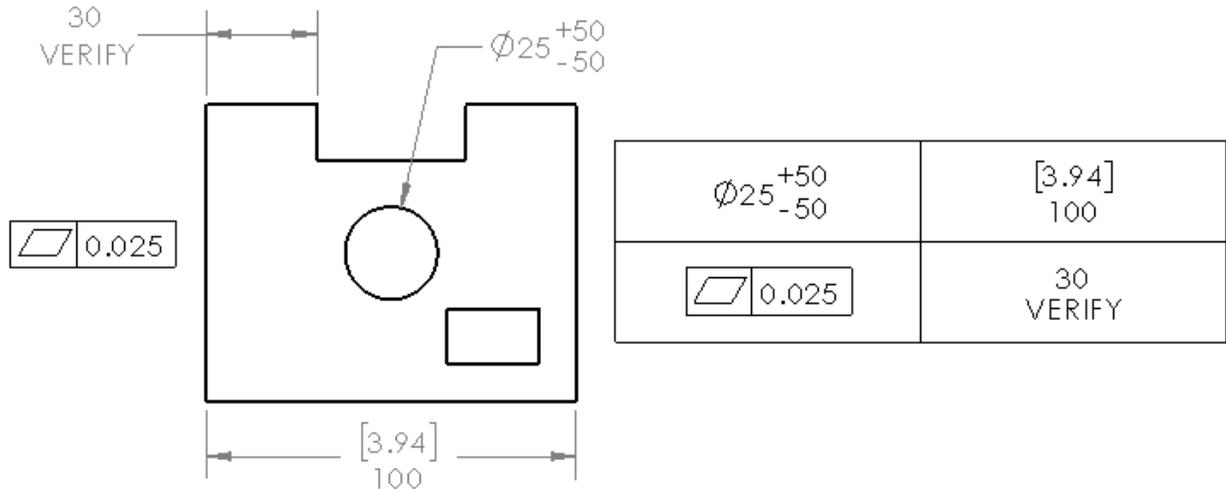


3. In the PropertyManager, click the Leaders tab.
4. Under **Witness/Leader Display**, click **Size**.
5. In the Circle Size dialog box, clear **Use document size**.
6. Type 5 for the diameter of the origin circle and click **OK**.
7. Click .

To set a document property for the origin circle size in a DIN ordinate dimension, click **Tools > Options > Document Properties > Dimensions > Ordinate** and click **Size**.

Dimension and Geometric Tolerances in General Tables

When linking the value of a dimension or geometric tolerance to a cell in a general table, the dimension or geometric tolerance in the cell displays the same elements as the original annotation, including tolerance, dual dimensions, and additional text.

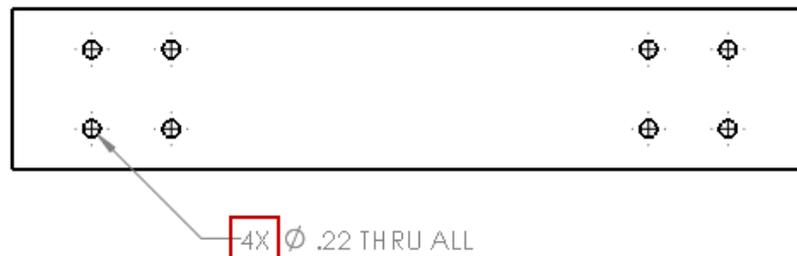


Hole Callout Instance Count for Mirrored Holes

When you use the **Hole Callout** tool on a hole pattern that includes a mirror on the same body, the instance count includes the mirrored holes.



SOLIDWORKS 2017



SOLIDWORKS 2016

Short Leader Radius Dimensions

The **Smart arrows attach to arcs and point away from dimension** option supports radial dimensions with short leaders.

To set the option, click **Tools > Options > Document Properties > Dimensions > Radius** and under **Arrow placement**, select **Smart arrows attach to arcs and point away from dimension**.

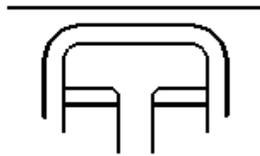
Tolerance Font Size

You can set the tolerance font size for individual dimensions using options in **Text Font** on the Other tab in the Dimension PropertyManager.

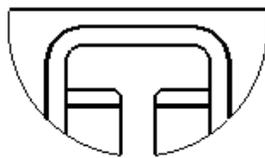
Drawing Views

Creating Crop and Details Views with No Outline

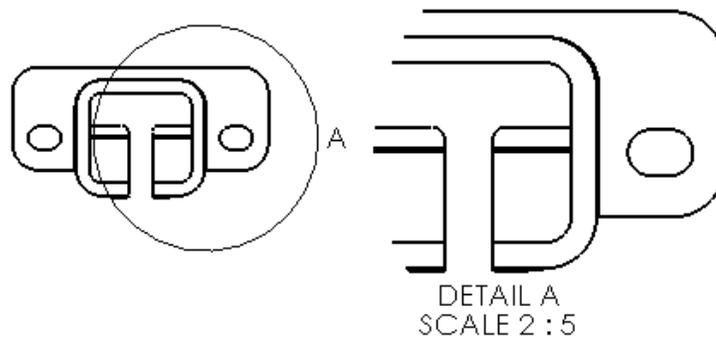
Crop and detail views support no outline.



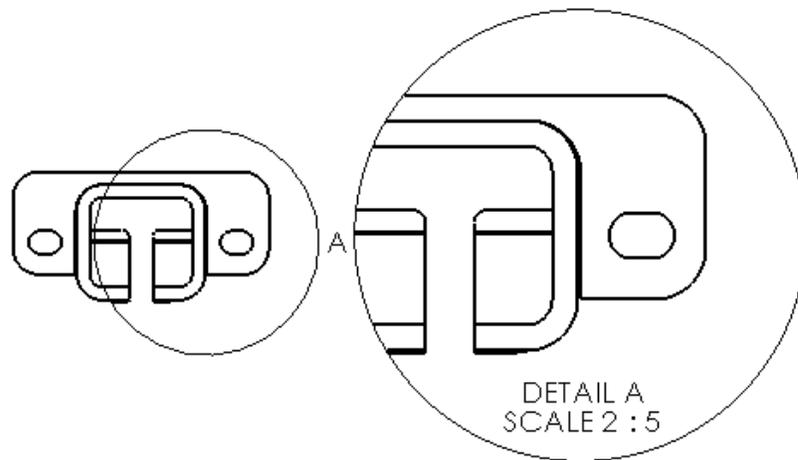
Crop view with no outline



Crop view with full outline



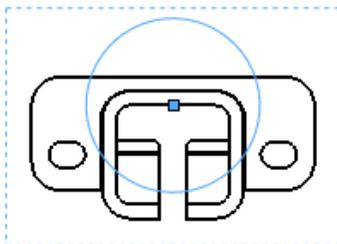
Detail view with no outline



Detail view with full outline

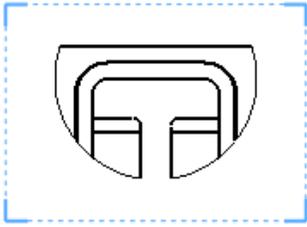
To create a crop view with no outline:

1. In a drawing view, sketch a closed sketch profile, such as a circle.



2. Select the circle.

3. Click **Crop View**  (Drawing toolbar) or **Insert > Drawing View > Crop**.



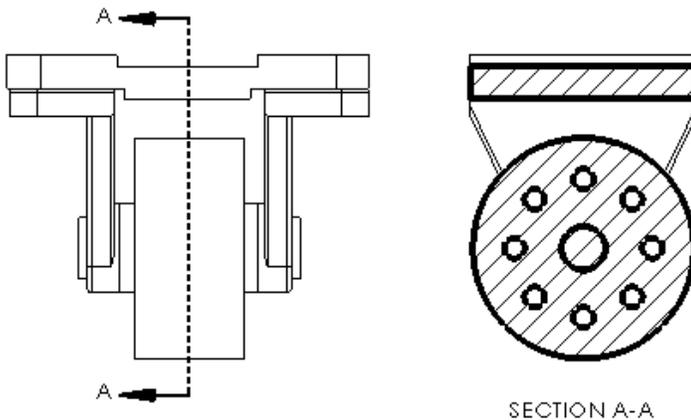
4. Select the drawing view.
5. In the PropertyManager, under **Crop View**, select **No outline**.



6. Click .

Emphasizing Outlines

In section views, you can emphasize the outline of cut faces in compliance with ISO 128-50. Typically, this means setting the outline of the faces thicker than other model lines in the section view.



Emphasizing the Outline in a Section View

To emphasize the outline in a section view:

1. Select a section view in a drawing.
2. In the PropertyManager, under **Section View**, select **Emphasize Outline**.
3. Click .

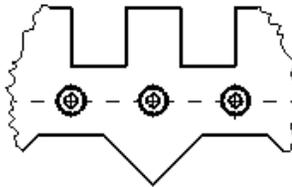
Setting the Line Font for Emphasized Outlines

To set the line font for emphasized outlines:

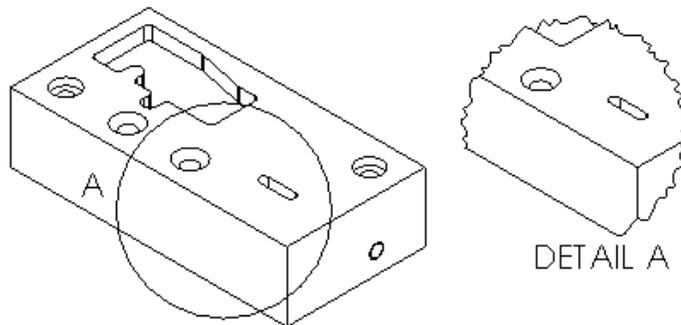
1. In a drawing, click **Tools > Options > Document Properties > Line Font**.
2. In the dialog box, in **Type of edge**, select **Emphasized Section Outline**.
3. Set options.
4. Click **OK**.

Jagged Outlines

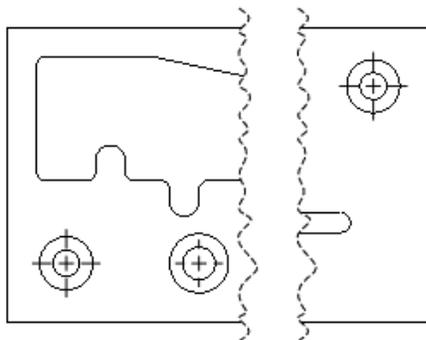
Crop and detail views support jagged outlines. Break views support jagged cut break line styles.



Crop view with jagged outline



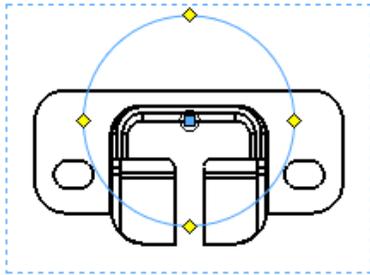
Detail view with jagged outline



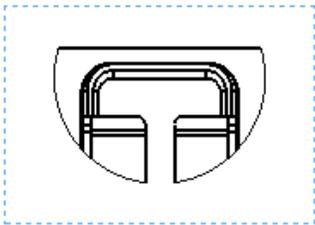
Creating Crop Views with Jagged Outlines

To create crop views with jagged outlines:

1. In a drawing view, sketch a closed profile for the crop view, for example, a circle.

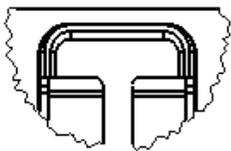


2. Click **Crop View**  (Drawing toolbar) or **Insert > Drawing View > Crop**.



The view outside the profile disappears.

3. Select the drawing view.
4. In the PropertyManager, under **Crop View**, select **Jagged outline**.
5. Under **Shape Intensity**, move the slider to define the intensity of the jagged outline.



6. Click .

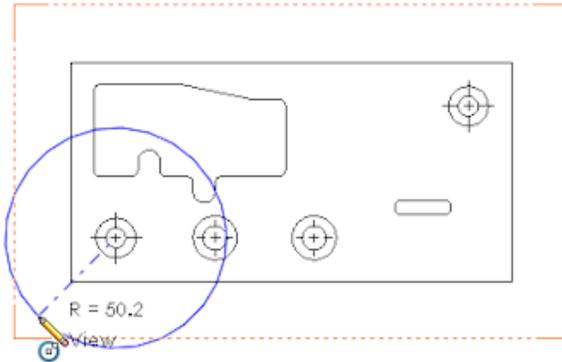
Creating Detail Views with Jagged Outlines

To create detail views with jagged outlines:

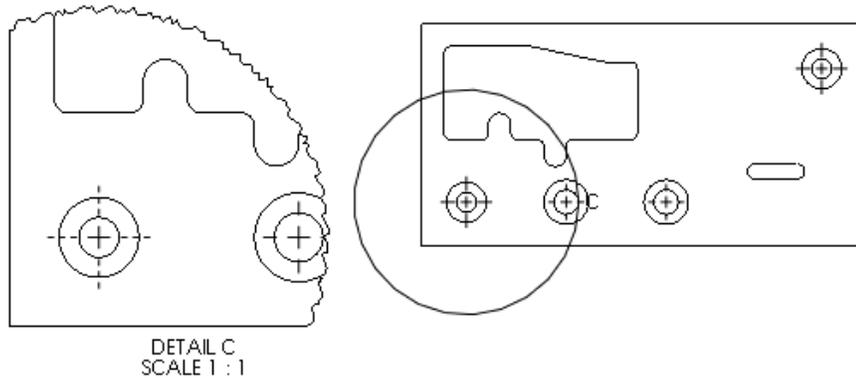
1. In a drawing view, click **Detail View**  (Drawing toolbar) or **Insert > Drawing View > Detail**.

The **Circle** tool  is active.

2. Sketch a circle.



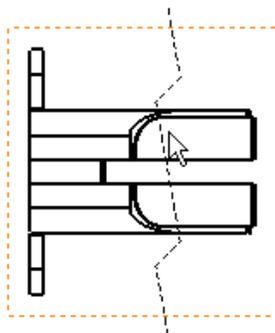
3. In the PropertyManager, under **Detail View**, select **Jagged outline**.
4. Under **Shape Intensity**, move the slider to define the intensity of the jagged outline.
5. Click to place the view.



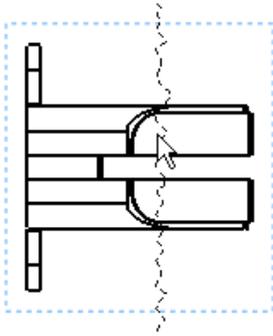
Creating Break Views with Jagged Cut Break Line Styles

To create break views with jagged cut break line styles:

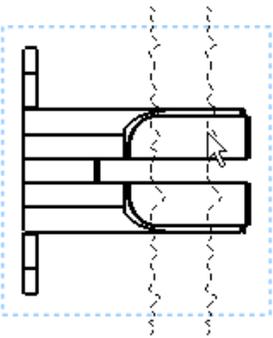
1. In a drawing view, click **Break View**  (Drawing toolbar) or **Insert** > **Drawing View** > **Break**.



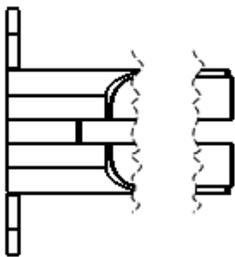
2. In the PropertyManager, in **Break line style**, under **Break View Settings**, click **Jagged Cut** .



3. Under **Shape Intensity**, move the slider to define the intensity of the jagged cut.
4. Click to place the first break line.
5. Click to place the second break line.

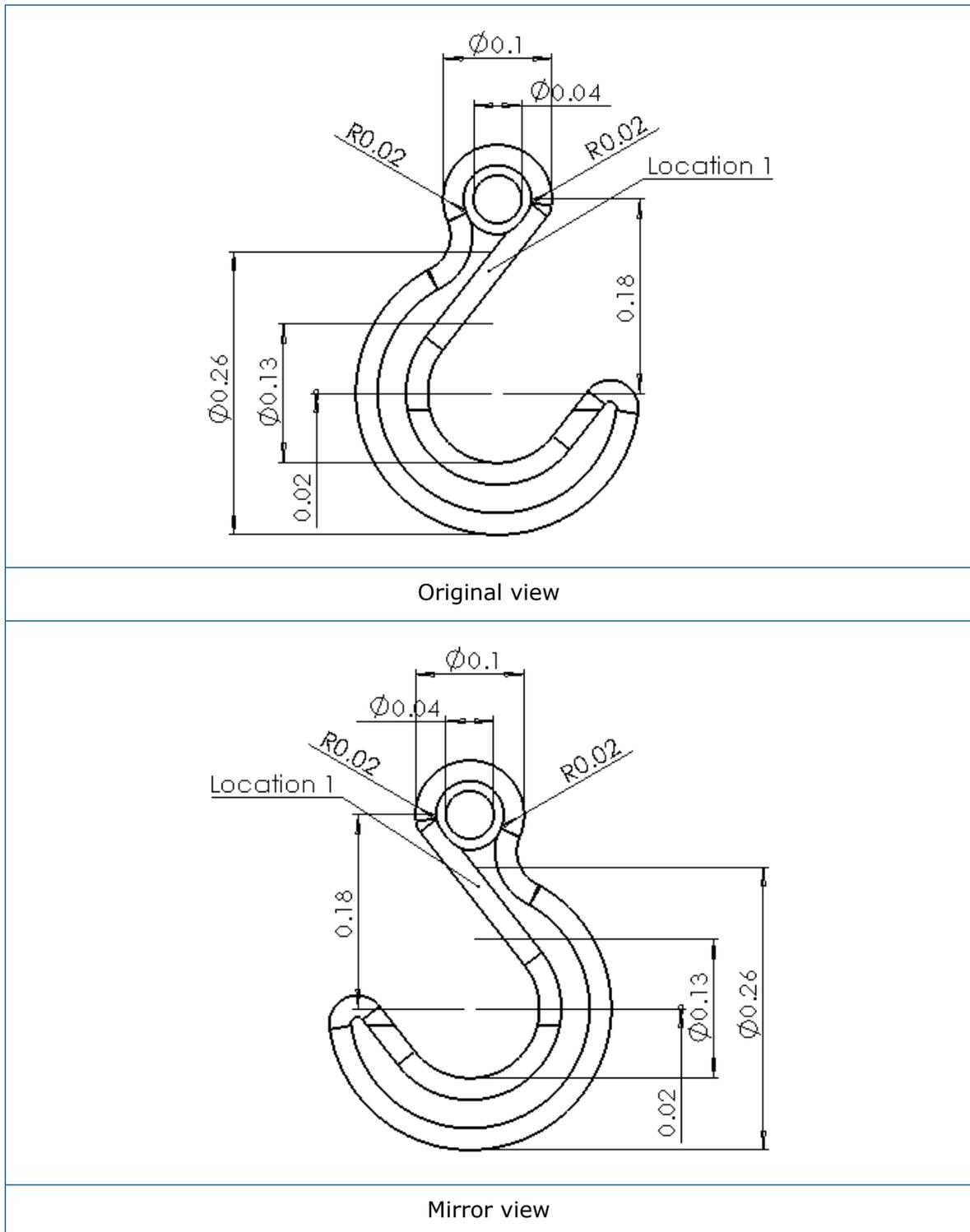


6. Click .



Mirror Drawing Views ★

You can display model, relative to model, and predefined drawing views as mirror views without creating the mirror components. Projected and other derived views use the mirror state from their parent view.

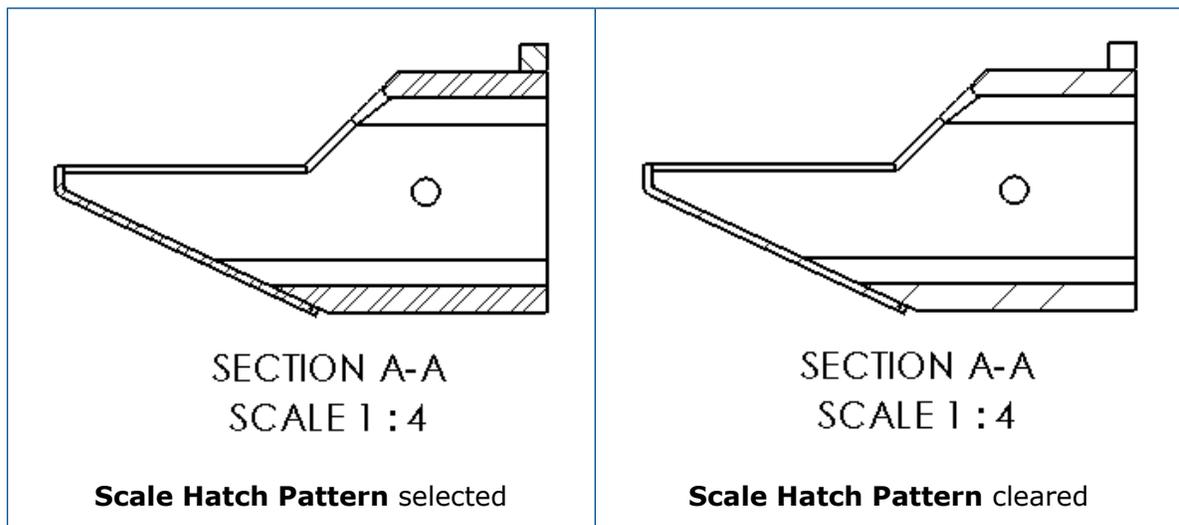


To mirror drawing views:

1. In a drawing, select a model, relative to model, or predefined drawing view.
2. In the PropertyManager, under **Orientation**, select **Mirror view**.
3. Under **Mirror view**, select **Horizontal** or **Vertical**.
4. Click .

Scale Hatch Pattern

Scale Hatch Pattern is available for section views and drawing views that include broken-out section views.



When you select **Scale Hatch Pattern** in the Section View PropertyManager, hatches within this view apply the view's scale to the hatch pattern. Previously, only detail views had this option.

Model Detailing

Custom Text Scale

You can set a custom text scale for text annotations in parts and assemblies.

To set a custom text scale:

1. Click **Tools > Options > Document Properties > Detailing**.

Alternatively, in the FeatureManager design tree, right-click the **Annotations**  folder and click **Details** to display the Annotation Properties dialog box

2. In the dialog box, clear **Always display text at the same size**.
3. In **Text scale**, select **Custom**.

4. In the two fields, enter the first and second value of the custom scale. For example, enter 3 and 10 to set the scale to 3:10.

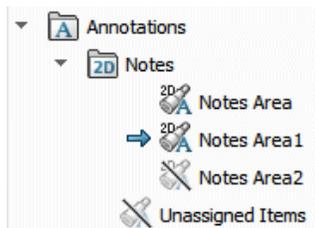
Drawings Maintained When Parts are Converted

When you purge a part of its features using **Convert to Bodies**, the original drawing file still references the part's geometry.

Managing Notes Areas

You can create multiple notes area annotation views in parts and assemblies, and you can delete and rename notes areas. You can also specify which notes area to display when opening the document.

The notes areas are placed in the **Notes** folder, which is under the **Annotations** folder in the FeatureManager design tree. Multiple notes areas let you create various sets of notes and place them on specific note areas that you can activate.



To create a notes area:

- In the FeatureManager design tree, in the **Annotations** folder, right-click **Notes** and click **Insert Notes Area**.

A notes area is added to the **Notes** folder with a sequential number following the name, for example, **Note area3**.

To delete a notes area:

- In the FeatureManager design tree, in the **Annotations** folder, right-click a **Note area** and click **Delete**.

To rename a notes area:

1. In the FeatureManager design tree, in the **Annotations** folder, click-pause-click the **Note area** to rename.
2. Type the new name.
3. Press **Enter**.

To specify a notes area to display when opening a part or assembly:

- In the FeatureManager design tree, in the **Annotations** folder, right-click the **Note area** and click **Show on Open**.

To change the notes area to display on open, you must first clear the selected notes area to display. Right-click the notes area and clear **Show on Open**.

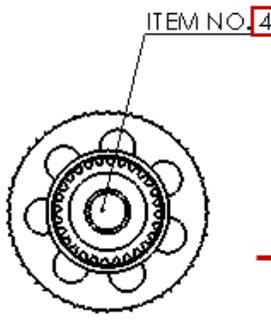
Reorder Annotation Views

Using drag and drop, you can reorder annotation views, and you can reorder notes area annotation views within the Notes folder.

Notes and Balloons

Linking a Note to a Table Cell

You can link a note to the contents of any BOM or hole table cell.

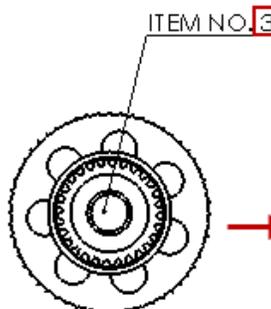


ITEM NO.	PART NUMBER	QTY.	DESCRIPTION
1	NOA_O_Spur_gear_A_1_INCH straight teeth	1	Inch - Spur gear 48DP 5.6T 20 PA .1875FW ---S5.6O.4375H.5L0.25N
2	NOA_O_Spur_gear_A_1_INCH straight teeth	1	Inch - Spur gear 48DP 28T 20 PA .1875FW ---S28O.4375H.5L0.25N
3	99040A510	4	.010 THICK X.250 ID X .375 OD SHIM (STAINLESS)
4	89-NOA O-4200-1233	1	SHA FT
5	89-NOA O-4200-1236	1	LABYRINTH RING
6	SSRF4ZZKCA5	1	SSRF4ZZKCA5

To link a note to a table cell:

1. In a drawing with a table, insert a note.
 - a) Click **Insert > Annotation > Note**.
 - b) Click to place the note.
 - c) Click to edit the note.
2. In the PropertyManager, under **Text Format**, select **Link Table Cell** .
3. In the table, select a cell.
4. Click .

Because the note is linked to a table cell, when the cell value changes, the note updates.



ITEM NO.	PART NUMBER	QTY.	DESCRIPTION
1	NOA_O_Spur_gear_A_1_INCH straight teeth	2	Inch - Spur gear 48DP 28T 20 PA .1875FW ---S28O.4375H.5L0.25N
2	99040A510	4	.010 THICK X.250 ID X .375 OD SHIM (STAINLESS)
3	89-NOA O-4200-1233	1	SHA FT
4	89-NOA O-4200-1236	1	LABYRINTH RING
5	SSRF4ZZKCA5	1	SSRF4ZZKCA5

Linking Part Numbers in Drawings and the Property Tab Builder

You can create a note in a drawing that links to a part number. You can place this note typically in the title block of a drawing sheet. You can also make a part number display in the Property Tab Builder. When you click on a part or component of an assembly, the part number appears in the Property Tab.

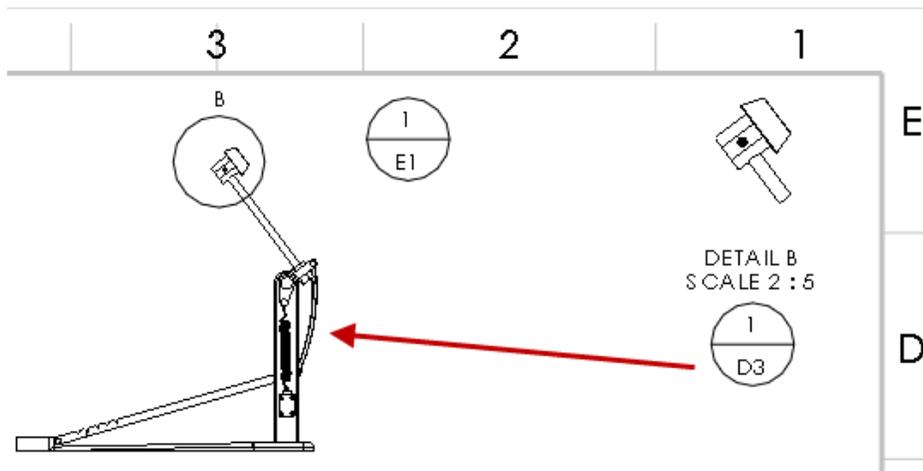
To create this note in a drawing sheet, in the Notes PropertyManager, under **Text Format**, click **Link to Property** . Then in the Link to Property dialog box, for **Property name**, select **SW-BOM Part Number(BOM Part Number)**. If this note is in the title block, when you update the part number in the drawing, the model automatically updates the part number, and vice versa.

Previously, this feature was only available in balloons.

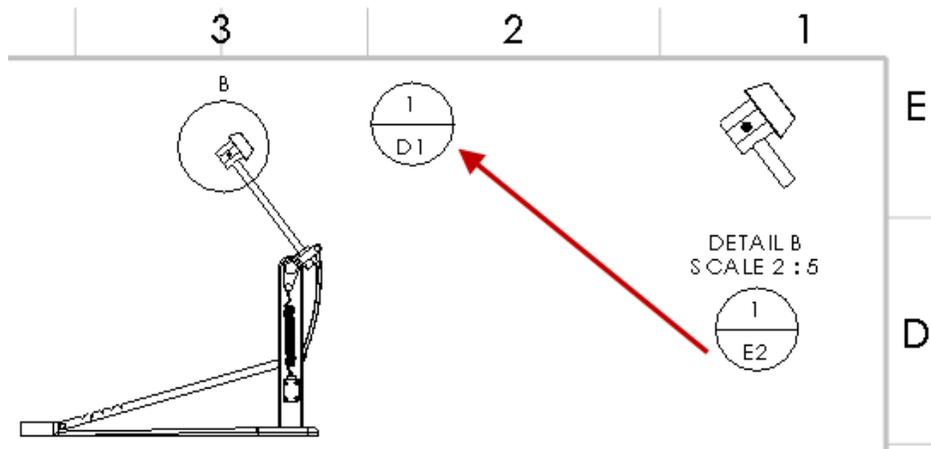
To display a part number in the Property Tab Builder, you need to add a textbox control and set the value to **SW-BOM Part Number**. If you already have a template, you can reference the file in **Tools > Options > File Locations > Custom Property Files**.

Location Label Zone

The **Display zone of counterpart location label** option displays the zone of the counterpart location label of the associated parent or child view.



Location label displaying the zone of the associated view's centroid



Location label displaying the zone of the counterpart location label of the associated parent or child view

To set this option, click **Tools > Options > Document Properties > Annotations > Location Label** and select **Display zone of counterpart location label**.

New Symbols for Tolerancing

New symbols are available for tolerancing.

To access the tolerancing symbols:

1. • For parts, click **Geometric Tolerance**  (DimXpert toolbar) or **Tools > DimXpert > Geometric Tolerance**.
 - For drawings, click **Geometric Tolerance**  (Annotation toolbar) or **Insert > Annotations > Geometric Tolerance**.
2. In the PropertyManager, under **Text**, click **More**.
3. In the Symbol Library, in **Categories**, select **ISO Tolerancing**.

New Tolerancing Symbols

	Two-point size		Average size
	Spherical size		Median size
	Least-squares		Mid-range size
	Maximum inscribed		Range of sizes
	Maximum circumscribed		Envelope requirement

	Circumference diameter		Free state
	Area diameter	ACS	Any cross section
	Volume diameter	SCS	Specified cross section
	Maximum size	CT	Common tolerance
	Minimum size		

To access the new modifying symbol, in the Symbol Library, in **Categories**, select **Modifying symbols**.

New Modifying Symbol

	Conical taper
---	---------------

Symbol Kern Spacing

BOUND is a control variable for symbols in the `GTOL.SYM` file. BOUND controls horizontal spacing of a symbol within a row of text in notes and dimensions.

Performance in Drawings

Auto Insertion of Center Marks Optimization

Creating drawing views with auto insertion of center marks is optimized for performance.

Automatically Populate View Palette

You can use the **Automatically populate View Palette with views** option to control whether time-consuming View Palette operations are initiated.

To turn off the automatic population of the View Palette, in **Tools > Options > System Options > Drawings**, clear **Automatically populate View Palette with views**.

Lightweight Drawing Open Time

The software no longer performs unnecessary operations associated with appearances when opening a drawing as lightweight.

Sheets

Change Sheet Format for Multiple Sheets at the Same Time

You can set sheet properties, including sheet formats and zone parameters, for multiple drawing sheets at the same time.

To change the sheet format for multiple sheets at the same time:

1. In the FeatureManager design tree, right-click a drawing sheet and click **Properties**.
2. In the Sheet Properties dialog box, on the Sheet Properties tab, click **Select Sheets to Modify**.
3. In the Sheet Selection dialog box, select the sheets to change.
4. Click **OK**.
5. On the Sheet Properties and Zone Parameters tabs, set the properties.
6. Click **Apply Changes**.

Tables

Lock Column Width in Table Templates

Table templates properly store and apply the **Lock Column Width** setting for entire tables and individual columns within a table.

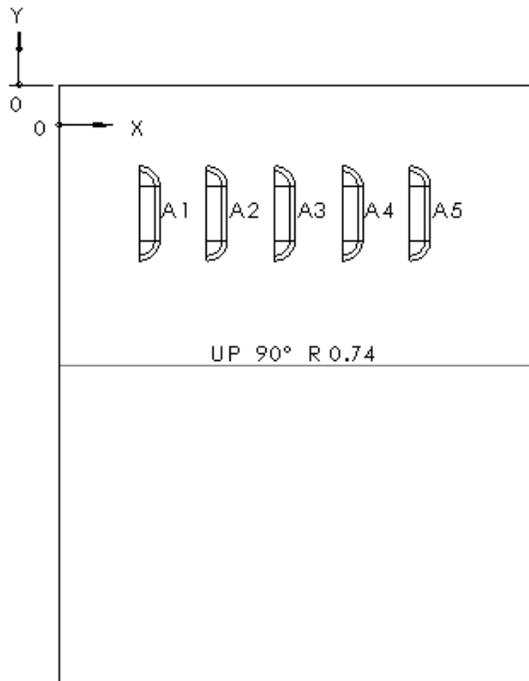
Replaced Component Item Numbers in BOMs

When replacing one component with another in an assembly, in the BOM you can assign a new item number to the replaced component while keeping it in the same row as the removed component.

To assign a new item number to the replaced component and keep it in the same row, in the Bill of Materials PropertyManager, select **Keep Missing Item** and **Assign new item number and do not keep replaced component**.

Retrieval of Sheet Metal Components on Drawings

In drawings, you can retrieve sheet metal information from the parent component in mirrored or derived sheet metal components.



TAG	PUNCH ID	X LOCATION	Y LOCATION	ANGLE(X AXIS)	QUANTITY
A1		0.03	-0.05	180.00°	1
A2		0.06	-0.05	180.00°	1
A3		0.08	-0.05	180.00°	1
A4		0.11	-0.05	180.00°	1
A5		0.13	-0.05	180.00°	1

For example, if you create a drawing view of a mirrored sheet metal part, any forming tools or library features in the mirrored part are available in punch tables.

When creating a derived part, you need to select **Sheet metal information** in the Insert Part PropertyManager.

11

eDrawings

This chapter includes the following topics:

- **Opening Files as Read-Only**
- **Password-protected Files**
- **Supported File Types in eDrawings**

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

Opening Files as Read-Only

You can set an option to always open files in eDrawings as read-only.

In **Tools > Options**, on the General tab, select **Open files read-only**.

When selected, this option also selects the **Open as read-only** option in the Open dialog box. Clearing the option in the Open dialog box does not affect the option in the Options dialog box.

Password-protected Files

The free version of eDrawings supports password-protected files.

When you publish an eDrawings file from your CAD application, you can assign a password to protect the eDrawings file. Previously this functionality was limited to eDrawings Professional.

Supported File Types in eDrawings

eDrawings supports additional file types.

Additional file types you can open in eDrawings include:

- STEP AP242
- IGES
- CATIA® V5
- Autodesk® Inventor™
- 3DXML

12

SOLIDWORKS Electrical

This chapter includes the following topics:

- **Associating a Data File Shortcut to a Project**
- **Associating External SOLIDWORKS Assemblies**
- **Automatically Update Project Level Reports**
- **Copying and Pasting with Anchor Points**
- **Creating Links in Design Rule Check Reports**
- **Changing Attribute Visibility**
- **Creating Cable Mark Formulas**
- **Importing and Exporting Drawings with Microsoft Excel**
- **Mark Wires with the Same Value as I/O Address Connected**
- **Other Enhancements for SOLIDWORKS Electrical**
- **Properties Panel Graphical Enhancement**

SOLIDWORKS Electrical is available as a separately purchased product.

Associating a Data File Shortcut to a Project

You can maintain folder structures according to your company standards by creating shortcuts to data files inside the SOLIDWORKS Electrical project folders.

When you associate an external data file, a shortcut is added in the document tree and project folder. The shortcuts created in the SOLIDWORKS Electrical project folders let you maintain company folder structures. The data files are not copied to the project folder.

To associate a data file shortcut to a project:

1. Do one of the following:
 - Right-click **Document book** > **New** and click **Data file shortcuts** .
 - Click the **Project** tab > **New**  and click **Data file shortcuts** .
2. In the dialog box, select one or more files to add to the project.

The software creates a shortcut in the document tree and project folder. If you delete the data file shortcut, the shortcut is removed from the project, and the data file remains untouched in its current location.

3. Do one of the following:
 - Double-click the shortcut to open it.
 - Right-click the shortcut and click **Properties** . The target file has an internal ID name; the path is grayed out in the property dialog boxes and cannot be changed.

If you move the target file, the database path updates. When you include shortcuts in an archive, the file targeted is copied into the project folder and the existing link is replaced.

Associating External SOLIDWORKS Assemblies

You can add existing SOLIDWORKS assemblies to SOLIDWORKS Electrical projects and create shortcuts to the files in the SOLIDWORKS Electrical project folder.

When you associate external assemblies, the software creates a shortcut to the assembly in the document tree, the SOLIDWORKS Electrical project folder, and the **SOLIDWORKS assembly** dialog box. These shortcuts let you maintain company folder structures. The associated files are not copied to the project folder.

The **SOLIDWORKS cabinet layout** tool in the **Process** tab is renamed to **SOLIDWORKS assembly**.

To associate external SOLIDWORKS assemblies:

1. Click the **Process** tab and click **SOLIDWORKS assembly** .
2. In the dialog box, in **Select existing file**, select the assembly to add to the project. A shortcut is created in the document tree and project folder.
3. After the associated assembly shortcut is created:
 - Double-click the shortcut in the document tree to open it.
 - Right-click the shortcut and click **Properties** . The external assembly file has an internal ID name; the path is grayed out in the property dialog boxes and cannot be changed.

If you move the external assembly file, the database path updates. When you include shortcuts in an archive, the external assembly file is copied into the project folder and the existing link is replaced.

Automatically Update Project Level Reports

When printing, exporting, or archiving reports and terminal strips, you can ensure that the project data is up to date automatically.

Use **Update generated drawings** in the **Projects configuration** dialog box lets to automatically update data in reports and terminal strip drawings before printing, exporting, or archiving.

To automatically update project level reports:

1. Click **Project** tab > **Configurations**  > **Projects** .
2. In the dialog box, click the **General** tab > **Options** section.
3. From the **Update generated drawings** menu, select an option.

Never	Processes reports, design rules, and terminal strip drawings in the current state. Any recently made changes might be absent from the output.						
Ask me (default)	Displays a message with the following options: <table border="1" data-bbox="876 651 1412 1102"> <tr> <td>Update drawings</td> <td>Updates all reports, design rules, and terminal strip drawings before running this command.</td> </tr> <tr> <td>Continue action without updating</td> <td>Runs the command without updating drawings.</td> </tr> <tr> <td>Cancel</td> <td>Exits the current command.</td> </tr> </table>	Update drawings	Updates all reports, design rules, and terminal strip drawings before running this command.	Continue action without updating	Runs the command without updating drawings.	Cancel	Exits the current command.
Update drawings	Updates all reports, design rules, and terminal strip drawings before running this command.						
Continue action without updating	Runs the command without updating drawings.						
Cancel	Exits the current command.						
Always	Automatically updates the files before processing the command (no message is displayed).						

Copying and Pasting with Anchor Points

When copying a set of objects in a drawing, you can define an anchor or reference point and use that reference point to paste the objects into a new drawing.

To copy and paste with anchor points:

1. Do one of the following:
 - On the Edit tab, select **Copy with insertion point** .
 - In the graphics area, right-click and click **Copy with insertion point** .
 - Press **Ctrl + Shift + C**.
2. Select the objects to copy.

3. To enter a reference point, do one of the following:
 - Click a location in the graphics area to be the reference point. This sets the X and Y coordinates and closes the dialog box.
 - In Enter Coordinates, type the X and Y coordinates and click . On the Command tab, click .
4. Open a drawing where you want to paste the contents.
5. Do one of the following:
 - On the Edit tab, click **Paste** .
 - Right-click in the graphics area and click **Paste** .
 - Press **Ctrl + V**.
6. Position the pointer where you want to insert the content and click.
The content is inserted away from the pointer at the distance specified by the reference point.

Creating Links in Design Rule Check Reports

You can navigate to any entity in the design rule check (DRC) reports or normal reports using the Report Manager dialog box.

Contextual menus in the table cells of the Design rules and Report dialog box link to data in the Report Manager.

To create links in design rule check reports:

1. Do one of the following on the **Project** tab:
 - Click **Configurations** and select **Design rule check** or **Report**.
 - Click **Design rule check** or **Report**.

You need to go to Project > **Configuration** > **Design rule check** or **Report** to access all the reports, or go to Project > **Design rule check** or **Report** to access the selected reports.
2. Select a particular report.
3. Click **Properties**.
4. On the Columns tab, click **Go to** link.
5. Select **Field name** and **Link type** from the list.
IDs under **Field name** correspond to the property for which you set the **Go to** link.
6. Click **OK** and **Apply**.

7. In the Report manager, right-click in a cell to view the **Go to** link result(s).

Report Entry	Link
Document	Links to the drawing.
Component	Displays the component with links to the component tree. Displays all associated symbols with links to the drawing.
Symbol	Links to the symbol in the drawing, zooming around the symbol.
Cable	Displays a line diagram, where reserved, and links to the cable entity in the line diagram. Displays list of cable cores in the scheme and links to the text entity in the scheme.
Cable core	Links to text entity in the scheme.
Equipotential	Links to the start of the equipotential in the scheme.
Wire	Displays the wire in the scheme and links to the text entity. Displays from and to symbols, and links to symbols in the scheme.
Wire style	No link.
Manufacturer parts	Displays the referenced symbol and links to the 2D footprint, wiring label, and 3D component.
Harness	No link.

Changing Attribute Visibility

You can change the attribute visibility while defining or editing drawings symbols or title blocks.

To change attribute visibility:

1. On the Documents browser tab, under **Document book**, right-click any document and click **Title block > Open**.

2. On the Edit title block tab, click **Show attributes** and select an option.
You can also access **Show attributes** by right-clicking a symbol in the graphics area and clicking **Symbol > Open symbol > Edit symbol > Show attributes**.

Creating Cable Mark Formulas

You can define cable mark formulas through project configuration.

To create cable mark formulas:

1. On the Project tab, click **Configurations** and select **Project**.
2. In the dialog box, on the Font tab, click **Edit formula** to edit the cable mark formula.
You can type the formula under in **Formula: Scheme cable core**.
3. Click **OK** twice.

Importing and Exporting Drawings with Microsoft Excel

You can use the **Excel Automation** option to generate schematics automatically from a Microsoft Excel file by adding data and macros.

The contents of the Microsoft Excel file needs to be in a specific SOLIDWORKS Electrical format. The Excel Automation option lets you:

- Automatically create and insert a drawing from a Microsoft Excel file. You can redefine any macros later.
- Provide information on where to insert macros. For example, you may want to insert a macro in one or multiple drawings, or insert various macros in several drawings.
- Insert file and book descriptions, define the location and function of each macro.
- Define variables such as symbols or new marks. The respective properties of these symbols or marks are part of the macros and are associated with the drawings.

To import or export drawings with Microsoft Excel:

1. Click **Excel Automation**  (Export/Import toolbar).

Alternatively, you can click **Excel Automation > Create new Excel file for automation** to open a template and make a new file.

2. In the dialog box, select the custom Excel file and click **Open**.
3. Select an option:
 - **Change book association.** Creates a new association as mentioned in the selected Microsoft Excel file. For example, if the Excel file creates a new location, a new book is created and appears in the document tree. The drawings are associated with the newly created book.
 - **Do not change book association.** Keeps the association with the current book in the project.
 - **Cancel.** Exits the current command.

Selecting either of the first two options creates the drawings and displays a summary dialog box.

For additional information, see *SOLIDWORKS Electrical Help*.

Mark Wires with the Same Value as I/O Address Connected

You can set the same values/tags between various connected objects like wire, equipotentials; terminal and PLCs.

To set values, click **Process** > **Propagate data to connected objects**.

You can update these marks for the **Whole project**, **Current book**, **Current scheme**, **Current folder**, **Objects selected in the scheme**, and **Choose scheme**.

Other Enhancements for SOLIDWORKS Electrical

For additional information on these enhancements, see *SOLIDWORKS Electrical Help*.

SOLIDWORKS Electrical enhancements include:

- You can use the Super Part tool to create catalog parts using several parts/assemblies and several manufacturer parts.
- You can create the Super Part from the **Manufacturer parts manager** and add properties/information from the properties dialog box. The Super Part is then added to the database.
- Report enhancements:
 - You can generate reports specific to each Project book.
 - You can display the description of a wire group on top of the report table.
 - You are notified to update outdated drawing reports when you print them.
 - You can include the location and function descriptions in reports.
 - You can use the '|' character as a line breaker in reports.

To access the line breaker click **Project** > **Configuration** > **Report** > **Properties** > **Layout**.

- You can show connection dots on symbols as **Always**, **Never**, or only **When connected**.
- You can view current, selected, and opened document highlighted in the document browser.

Properties Panel Graphical Enhancement

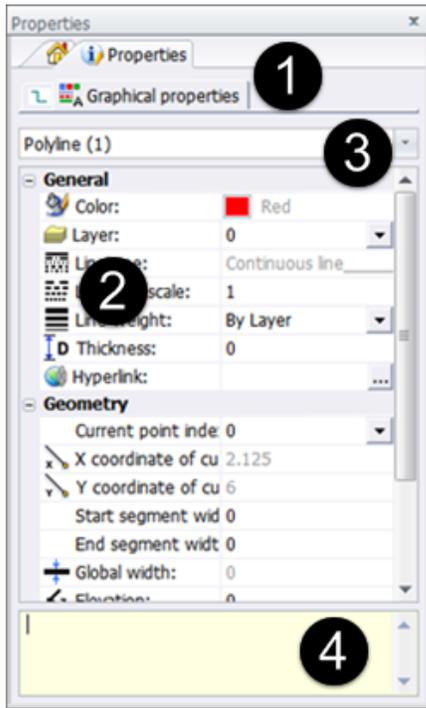
The Graphical Properties tab shows the graphical properties of any CAD entity, symbol, or wire element.

The **Graphical properties**  tab in the **Properties Panel** lets you segregate the contents of a selection. When you select a CAD entity, symbol, or wire element, all of its associated properties are listed with its graphical entity.

The **Properties Panel** is dockable on the side panels.

When you select a group of entities, all common properties are displayed. When you select specific entities, the related properties are displayed. Uncommon properties are hidden. Changing uncommon properties does not affect the CAD entity because it is driven at the application level.

The **Graphical properties** tab includes:



Item	Description
1	Graphical properties tab in the Properties panel.
2	Graphical icons corresponding to the listed property.
3	Drop-down menu listing the segregate contents or various entities of the selection, letting you choose a group of objects to edit.
4	Tooltip information such as why some properties are grayed out or cannot be edited.

13

SOLIDWORKS Flow Simulation

This chapter includes the following topics:

- **Korean Language Support**
- **Multiparameter Optimization**
- **Postprocessing Performance**
- **Transient Simulation Performance**
- **Using Conditions from Low-level Components**
- **Mesh Settings for Flow Simulation**

Available as a separately purchased product that can be used with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium.

Korean Language Support

SOLIDWORKS Flow Simulation is available in Korean.

Multiparameter Optimization

Multiparameter Response Surface-based optimization is added to the Parametric Study. You can conduct an optimization study for more than one input variable. You can select geometry or simulation parameters as variables.

The target optimization function can be maximizing, minimizing, or matching a value of a goal or a sum of goals. You can set different weighting factors for each goal to consider the importance of each goal for the objective function.

To conduct multiparameter optimization, create a **Design of Experiments and Optimization** parametric study, run calculation of design points, and click **Find Optimum**.

Postprocessing Performance

Postprocessing performance has been improved for surface plots and loading results.

Surface plots

Surface plots of a very complex geometry shape displayed with the **Use CAD geometry** option or comprising hundreds of faces are created much faster.

Goal plot and surface plot widgets

Goal plots and XY plots can now be displayed in the graphics area. The plots are updated in accordance with the loaded

transient results so you can track how the goal and XY plot parameters are changing in time.

Transient Simulation Performance

Transient simulation performance has been improved for saving data and viewing transient results.

Saving data

You can minimize transient data saved by saving selected parameters only instead of saving the whole field. In the Calculation Options dialog box, on the Saving tab, you can define when to save **Full Results**, and you can define time moments to save as **Selected Parameters**.

Instant switching between transient time moments

The **Transient Explorer** mode available for selected saved parameters lets you see transient results without waiting as long to load. Move a slider and results update immediately.

Using Conditions from Low-level Components

You can import conditions from a model for other projects (such as material properties and heat power). This can simplify the project definition by creating a library of models with predefined conditions.

A Flow Simulation project is saved in the model. If the model is used in an assembly, you can apply the conditions defined in this low-level component to the higher level assembly without recreating the condition.

For example, you may want to analyze a printed circuit board (PCB), then analyze the full electronics enclosure with fewer parts of the PCB installed. Instead of re-creating conditions for all parts inside the assembly, you can link the assembly's project to the conditions defined in the PCB.

By default, the conditions are linked, so changing conditions in the component automatically updates conditions in the assembly. However, you can break the link so imported conditions are not linked to the original component.

To use conditions from low-level components:

1. In the Import from Model dialog box, select the project from which you want to use data.
2. Import the project.

By default, conditions appear in the Analysis tree linked to the conditions in the component.

You can break the link and move conditions to the main assembly's project.

Mesh Settings for Flow Simulation

The new **Dependent on Channel Height** option allows you to easily achieve necessary mesh in channels of different height by setting the refinement level depending on channel

height. The option is available under the **Channels** group of the mesh settings dialog box.

14

Import/Export

This chapter includes the following topics:

- **IFC Entity Classification and Custom Property Export**
- **Importing and Exporting 3MF Files**
- **Importing Mesh Files**
- **SOLIDWORKS 3D Interconnect**
- **System Options for Interoperability**

IFC Entity Classification and Custom Property Export



You can save a SOLIDWORKS file as an IFC file with IFC entity classification. You can then export it with custom properties that include the IFC metadata along with other properties.

Classifying SOLIDWORKS files as IFC Entities

You can save a SOLIDWORKS file as an .ifc file with .ifc entity classification.

To classify SOLIDWORKS files as .ifc entities:

1. Open a SOLIDWORKS part or assembly file.
2. Click **File > Properties**.
3. In the dialog box, on the **Custom** tab:
 - a) In **Property Name**, select IFC.
 - b) In **Value/Text Expression**, select the appropriate taxonomy.
 - c) Click **OK**.

Exporting to IFC Format

You can export an IFC file with custom properties that include the IFC metadata along with other properties.

To export to IFC format:

1. With a model open, click **Save As**  (Standard toolbar) or **File > Save As**.

2. In the Save As dialog box:
 - a) In **Save as type**, select **IFC 2x3** (*.ifc) or **IFC 4** (*.ifc).
 - b) Click **Options**.
3. In the System Options dialog box, set options:

Option	Description
OmniClass™ or UniClass2015	Exports the classification metadata with the IFC entity if it is specified in the file.
Custom Properties or Material and Mass Properties	Exports properties for the SOLIDWORKS file.

4. If you are exporting as IFC 4, save as one of the following:
 - BREP
 - BREP and Tessellation
 - Tessellation
5. Click **OK**.
6. Click **Save**.

Importing and Exporting 3MF Files

You can import 3D Manufacturing Format (3MF) files and export SOLIDWORKS part and assembly files as 3MF files. Dassault Systemes is a founding member of the 3MF industry consortium, whose goal is to eliminate interoperability issues with 3D printing formats.

Importing 3MF files in SOLIDWORKS

To import 3MF files in SOLIDWORKS:

1. Click **Open**  (Standard toolbar) or **File > Open**.
2. In the dialog box, in **Files of type**, select **3D Manufacturing Format (*.3mf)**.
3. Browse to a file to open.
4. Click **Options**.
5. In the dialog box, set **Import as** to one of the following and click **OK**.
 - Graphics Body
 - Solid Body
 - Surface Body
6. Click **Open**.

The file is imported in SOLIDWORKS.

Exporting SOLIDWORKS files as 3MF Files

You can export SOLIDWORKS part and assembly files as 3MF files.

To export SOLIDWORKS files as 3MF files:

1. With a model open, click **Save As**  (Standard toolbar) or **File > Save As**.
2. In the dialog box:
 - a) Set **Save as type** to **3D Manufacturing Format (*.3mf)**.
 - b) Click **Options**.

3. In the dialog box, set these options:

Option	Description
File Format	Displays the selected file format.
Output as	For Unit , select the unit of measure.
Resolution	<p>Controls the tessellation of non-planar surfaces for 3MF output. A lower deviation setting produces a more-finely-tessellated 3MF file. Files generated with higher accuracy settings (increased tessellation) are larger in size and slower to generate.</p> <p>Coarse or Fine Preset resolutions.</p> <p>Custom</p> <ul style="list-style-type: none"> • Move the Deviation slider to adjust the deviation Tolerance, which controls whole-part tessellation. Lower numbers generate files with greater whole-part accuracy. • Move the Angle slider to adjust the angle Tolerance, which controls small-detail tessellation. Lower numbers generate files with greater small-detail accuracy, but those files take longer to generate. <div style="border: 1px solid gray; padding: 5px; margin-top: 10px;"> <p>As you move the two sliders, the corresponding concentric circles adjust accordingly. The circles show approximately how the tessellation will vary as a result of the settings. (See Preview below.)</p> </div> <p>Show 3MF info before saving file Displays this information: Triangles (number), File Size, File Format, and the directory path and file name.</p> <p>Preview before saving file Displays a faceted model preview and displays the Triangles (number) and File size information in the System Options dialog box.</p>
Include materials	Includes base materials with the exported file.
Include appearances	Includes appearances with the exported file. Only surface-mapped textural appearances are exported to 3MF.
Output coordinate system	Select a coordinate system to apply for export. If you select -- default -- , no transformation matrix is applied.

4. Click **OK**.

5. Click **Save**.

Importing Mesh Files

You can import a scanned file with mesh data into the SOLIDWORKS software without using the ScanTo3D add-in. The software imports the file and associated mesh data into a solid or surface body. You can select and reference the converted facets, verticals, and facet edges. You can also maintain the mesh as a mesh model in SOLIDWORKS that allows graphical viewing of the mesh model.

To import mesh files:

1. Click **Open**  (Standard toolbar) or **File > Open**.
2. In the dialog box, in **Files of type**, select **Mesh Files (*.stl, *.obj;*.off;*.ply;*.ply2)**.
You can also import ***.3mf** and ***.wrl** file formats as graphics bodies, solid bodies, and surface bodies.
3. Browse to a file to open.
4. Click **Options**.
5. In the dialog box, set **Import as** to one of the following and click **OK**.
 - Graphics Body
 - Solid Body
 - Surface Body

The software can import and create multiple solid bodies from the mesh file, provided the mesh is closed. If the mesh is not closed, the Import as Solid Body option automatically creates surface bodies for all bodies that cannot be imported as solid bodies.

6. Click **Open**.
The file is imported.

SOLIDWORKS 3D Interconnect

3D Interconnect replaces SOLIDWORKS current translation capabilities with new technology and workflows for working with third-party native CAD data in SOLIDWORKS 2017.

You can perform the following:

- Insert proprietary CAD data directly into a SOLIDWORKS assembly without converting it to a SOLIDWORKS file.
- Open the proprietary 3D CAD format in the SOLIDWORKS software with its associative link to the original part.
- Update changes in the SOLIDWORKS file if you update the proprietary CAD data in its authoring application by maintaining all downstream features created in SOLIDWORKS.

The following formats and their versions are supported in 3D Interconnect:

- CATIA® V5: .CATPart, .CATProduct for V5R8 – 5–6R2016
- Autodesk® Inventor: .ipt for V6 – V2016, .iam for V11 – V2016
- PTC®: .prt, .prt.*, .asm, .asm.* for Pro/ENGINEER® 16 – Creo 3.0
- Solid Edge®: .par, .asm, .psm for V18 – ST8
- NX™ software: .prt for UG 11 – NX 10

CATIA® V5-SOLIDWORKS Translator is available with SOLIDWORKS Premium license.

Turning 3D Interconnect On or Off

You can turn on and turn off the 3D Interconnect functionality in the SOLIDWORKS software. It is turned on by default.

To turn 3D Interconnect on or off:

1. Click **Tools** > **Options** > **System Options** > **Import**.
2. In the dialog box, set **File Format** to **Inventor/Catia V5/Creo/NX/Solid Edge**.
3. Select or clear **Enable 3D Interconnect**.
4. Click **OK**.

Inserting a Third-Party Native CAD File into a SOLIDWORKS Assembly

You can insert a third-party native CAD file into a SOLIDWORKS assembly using the **Insert Component** tool.

To insert a third-party native CAD file into a SOLIDWORKS assembly:

1. Open *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\import_export\gear_box_1.sldasm.
2. Click **Insert Component**  (Assembly toolbar).
3. In the dialog box, browse to *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\import_export\Round Cover Plate_2.ipt.
4. Click **Open** and assemble it in the SOLIDWORKS assembly.
5. Click **Insert Component**  (Assembly toolbar) and browse to *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\import_export\wormgear.asm.1.
6. Click **Open** and assemble it in the SOLIDWORKS assembly.

The third-party Autodesk® Inventor and PTC® Creo® files are inserted in the SOLIDWORKS assembly.

Opening a Third-Party Native CAD File into SOLIDWORKS

You can use the Open dialog box to open a third-party native CAD file in the SOLIDWORKS software. When you open a third-party part, a new SOLIDWORKS part is created and the third-party file is inserted as a derived or a base part in it.

To open a third-party native CAD file into SOLIDWORKS:

1. Click **Open**  (Standard toolbar) or **File > Open**.
2. In the dialog box, browse to *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\import_export\Round Cover Plate_2.ipt.
3. Make changes to the part using SOLIDWORKS features.
4. Update the part with the second version of the part.

Any modifications you made to the base part are updated in the SOLIDWORKS part without losing the downstream features.

System Options for Interoperability

The Import/Export options have moved to the System Options dialog box. Previously, you accessed them from the Open and Save As dialog boxes.

Click **Tools > Options > System Options** and click **Import** or **Export**. Under **File Format**, set the type of file.

15

SOLIDWORKS Inspection

This chapter includes the following topics:

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

SOLIDWORKS Inspection Add-in

Usability Improvements

All icons previously located in the PropertyManager are moved to the CommandManager to simplify the user interface.

These icons are modified:

Tool Name	Icons
Edit Inspection Project	
Add/Edit Balloons	
Update Inspection Project	

Manually Ballooning Drawings

Manually ballooning drawings gives you more control over how SOLIDWORKS Inspection adds balloons to drawings.

When you select automatic ballooning, the software balloons all characteristics in the drawing, using the specified sort order (clockwise or counter-clockwise).

If you select manual ballooning, the software:

- Balloons only the specific characteristics you identify.
- Orders the balloons based on the order in which you select them.

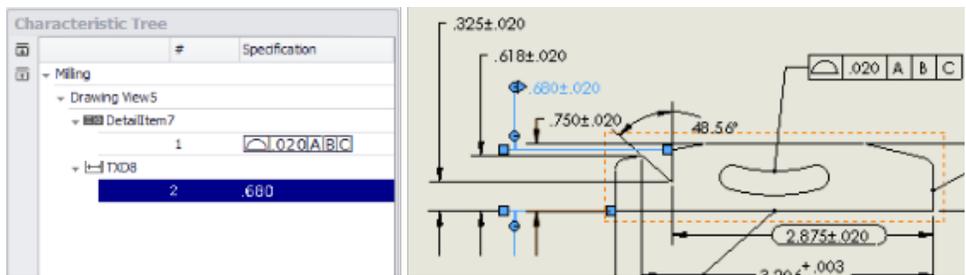
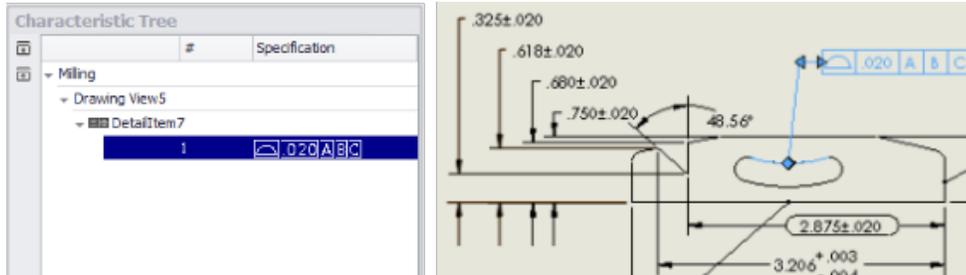
To manually balloon drawings:

1. Click **New Inspection Project**  (SOLIDWORKS Inspection CommandManager).
2. Select a project template.
3. On the General Settings tab of the Create Inspection Project PropertyManager, under **Characteristic Info**, for **Extraction**, select **Manual**.
4. Click  to validate the project settings.
5. Click **Add Characteristic**  (SOLIDWORKS Inspection CommandManager).

6. In the graphics area, use one of the following techniques to select the characteristics to balloon:

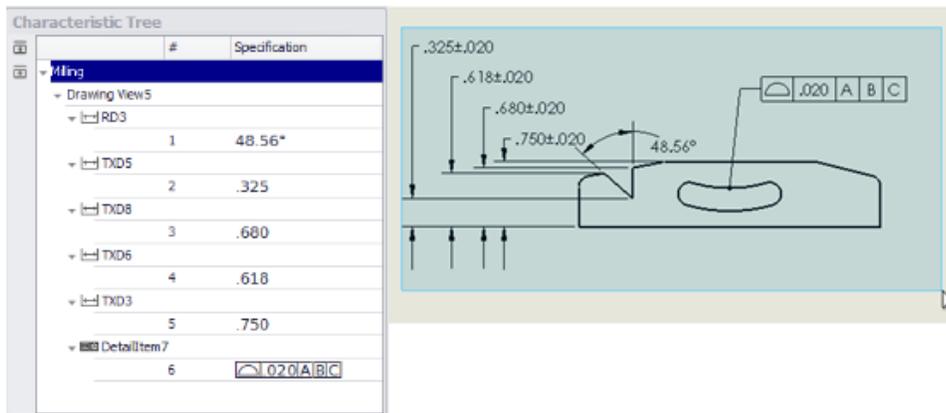
- Select individual characteristics.

The characteristics are added to the Characteristic Tree in the order in which you select them.



- Box select a set of characteristics.

All the characteristics within the box are added to the Characteristic Tree.

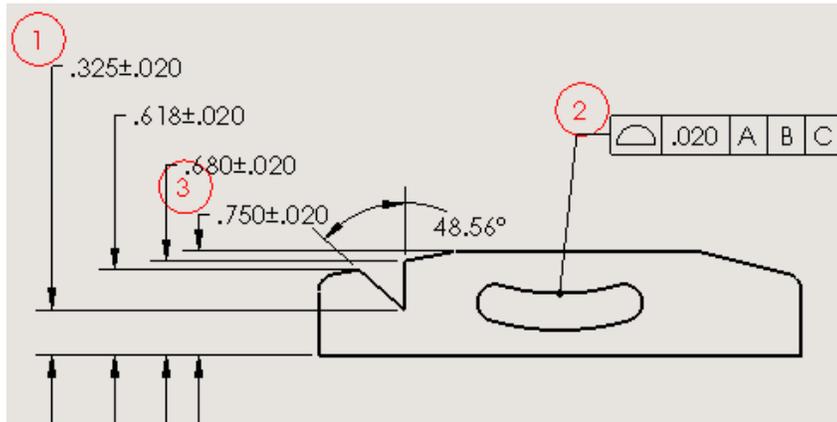


To remove a characteristic, click it in the graphics area.

If you want to balloon most but not all characteristics, you can box select the entire drawing and then remove characteristics you do not want to balloon.

- Click **Add/Edit Balloons**  (SOLIDWORKS Inspection CommandManager).
The software creates balloons for the characteristics you selected, in the order in which you selected them.

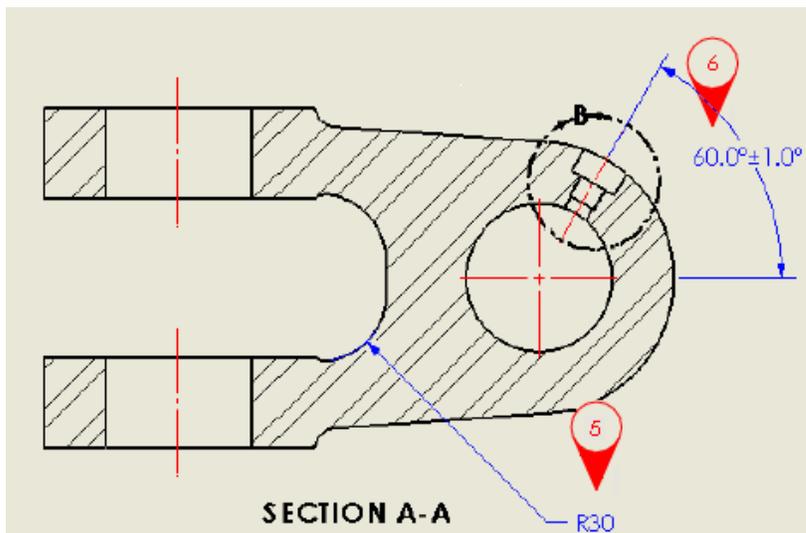
This step is not needed if you selected **Auto Balloon** in the Inspection Project Settings.



Choosing a VDA Balloon Shape

VDA balloons, which were available in standalone SOLIDWORKS Inspection, are now available in the SOLIDWORKS Inspection add-in.

VDA balloons are characterized by a drop shape and are frequently used in the German automotive industry.



To choose a VDA balloon shape:

- Click **Add/Edit Balloons**  (SOLIDWORKS Inspection CommandManager).

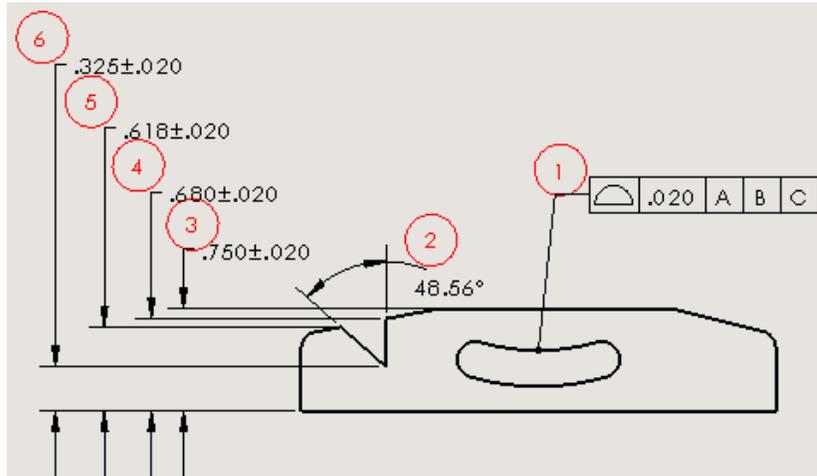
2. In the Balloon Settings PropertyManager, under **Regular Format**, for **Shape**, select **VDA**.
3. Click .

Preventing Balloon Renumbering by Locking Balloons

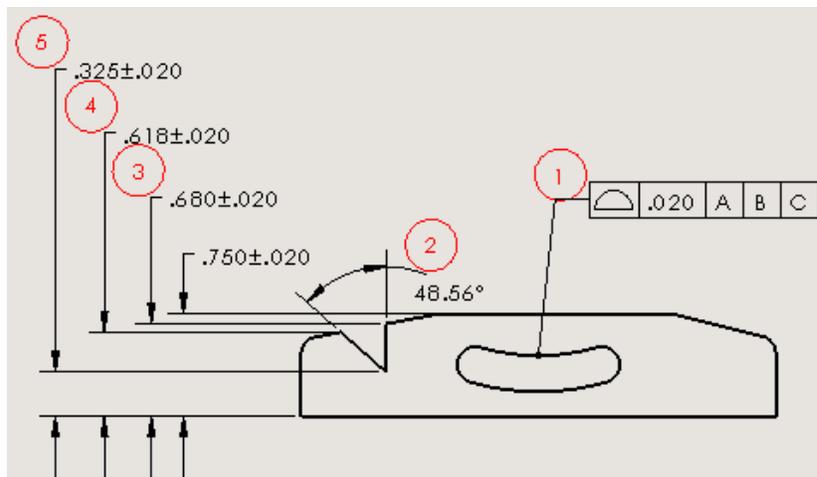
Prevent Automatic Renumbering lets you add or remove balloons without affecting the arrangement and numbering of the balloons that remain.

When you revise a SOLIDWORKS Inspection project by deleting or adding characteristics and then refresh it, by default the software renumbers the balloons in the document. If you specified a clockwise or counterclockwise sort order, when the balloons are renumbered, that order is maintained.

For example, if you remove the characteristic that is represented by balloon 3, the next sequential characteristic is renumbered to balloon 3. Depending on the version of the inspection report, the features to measure are different. This can create confusion and lead to costly mistakes.



Original balloon numbering



Balloons renumbered after removing characteristic 3

You can override this behavior by using **Prevent Automatic Renumbering**. When to use prevent automatic renumbering:

- If you remove a balloon, its number is removed from the sequence without affecting the other balloon numbers.
- If you add new characteristics, they are added at the end of the balloon number sequence.

To prevent balloon renumbering:

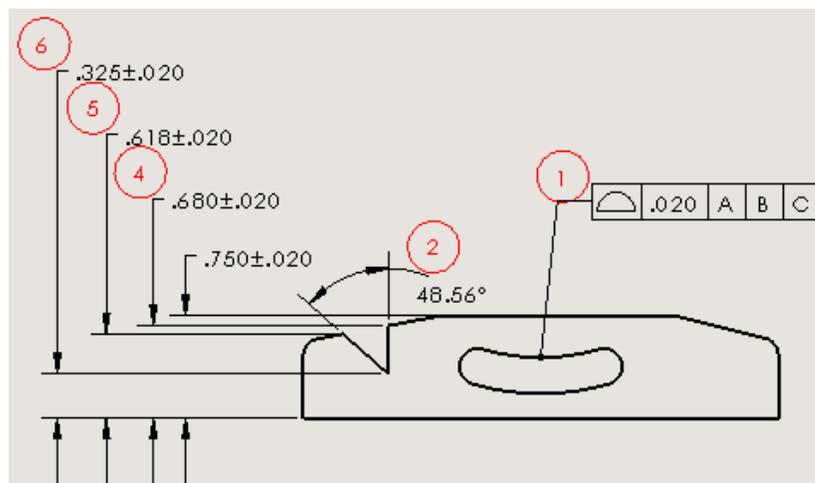
1. Click **New Inspection Project**  (SOLIDWORKS Inspection CommandManager).
2. Select a project template.
3. On the General Settings page, under **Characteristic Info**, select **Prevent Automatic Renumbering** and click .
4. To remove a balloon, do one of the following:
 - In the **Characteristic Tree**, select the balloon. Under **Properties**, clear the **Selected** check box.

The characteristic is no longer selected for inspection; however, the balloon number remains associated with the characteristic.

If you use this method, you can restore the balloon if needed by re-selecting the check box.

- In the graphics area, delete the characteristic that is associated with the balloon, then click **Update Inspection Project** .

The software removes the balloon number from the sequence, while maintaining the original numbering of the remaining balloons.



Original balloon numbering maintained

SOLIDWORKS Inspection Standalone

Mouse Wheel Zoom in SOLIDWORKS Inspection

Mouse Wheel Zoom lets you navigate an inspection project with controls that are similar to those in the SOLIDWORKS software.

Mouse Wheel Zoom is selected by default on the Application Options tab of the Options dialog box.

To use mouse wheel zoom:

- Scroll upward to zoom out.
- Scroll downward to zoom in.
- Press and hold the middle mouse button to pan.

In addition, you can use the following to navigate:

Alt + Left	Go to the previous sheet.
Page Down	Go to the previous sheet.
Alt + Right	Go to the next sheet.
Page Up	Go to the next sheet.
Alt + Up	Scroll up.
Alt + Down	Scroll down.
Home	Go to the first sheet.
End	Go to the last sheet.

16

SOLIDWORKS MBD

This chapter includes the following topics:

- **3D PDF Accuracy**
- **3D PDF Template Editor Improvements**
- **Annotation Display Orientation**
- **Attaching Files to 3D PDFs**
- **Comparing 3D PMI Between Parts**
- **Exporting PMI Annotations to STEP 242**
- **Names Match in 3D PDFs**
- **Reordering 3D Views**
- **Sketches Display in 3D PDFs**

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

3D PDF Accuracy

When you save or publish as 3D PDF, you can set the level of accuracy for a 3D PDF.

To set 3D PDF accuracy:

1. Click **Publish to 3D PDF**  (SOLIDWORKS MBD toolbar).
2. In the PropertyManager, under **Accuracy**, select an option:

Level of Accuracy	Description
Maximum	No lossy compression (may result in very large file size).
High	0.001mm tolerance (low compression, high accuracy) Low lossy compression may result in large file size.
Medium	0.01mm tolerance (medium compression, medium accuracy) Medium lossy compression may result in moderate file size.

Level of Accuracy	Description
Low	0.1mm tolerance (high compression, low accuracy) High lossy compression may result in smaller file size.

- To apply lossy compression to the polygons in the model, select **Use lossy compression on tessellation**.

For more information, see [Lossy Compression](#).

3D PDF Template Editor Improvements

The 3D PDF Template Editor gives you more control of the layout and formatting of the templates.

Aligning Elements

You can align two or more template elements by top, bottom, left, right, center, or middle.

To align elements:

- In the 3D PDF Template Editor, select two or more elements, for example, an image and a text area.
- Click **Align**  and click either **Left** , **Center** , **Right** , **Top** , **Middle** , or **Bottom** .

Copying and Pasting Between Pages

Tabs are now called pages. You can copy an element from one page to another.

To copy and paste between pages:

- Select the item to copy.
- Click **CTRL + C**.
- Select a page.
- Click **CTRL + V**.

Formatting Text

You can format text using the styles bold, italic, and underline, and you can align text left, right, center, and justify.

To format text:

- In the 3D PDF Template Editor, click **Text** .
- Click to place the text area placeholder on the template.
- In the PropertyManager, under **Properties**, set the formatting options.

4. Click outside of the text block.

Using Format Painter

Format Painter copies a text style to another text element.

To use Format Painter:

1. Select the text whose style (font, size, bold, italics, underline, strikethrough, color, and alignment) you want to copy.
2. Click **Format Painter** .
3. Click the text where you want to place the style.

The original text style is applied to the selected text.

Grouping and Ungrouping Elements

You can group and ungroup elements on a template page.

To group elements:

1. Press **Ctrl** and select two or more elements.
2. Click **Group > Group** .

To ungroup elements:

1. Select a set of grouped elements.
2. Click **Group > Ungroup** .

Inserting Elements at Pointer Location

When you insert an element such as text or a rectangle in a 3D PDF template, it is placed at the pointer location.

Inserting Rectangles

You can insert a rectangle in a 3D PDF template.

To insert rectangles:

1. In the 3D PDF Template Editor, click **Rectangle** .
2. Place the rectangle on the page.
You can position and resize the rectangle.
3. Set options such as fill color and border, border thickness, and corner radius.
4. Click .

Locking Elements

You can lock elements on a template page so that they cannot be moved or resized.

- To lock an element, select the element and click **Lock** .
- To unlock an element, select the element and click **Unlock** .

Ordering Elements

You can set the order of elements from front to back.

To order elements:

1. Select an element and click **Order** .
2. Click an option:

Option	Description
Bring to front	Moves element to the first position. 
Bring forward	Moves element to the front by one position. 
Send to back	Moves element to the last position. 
Send backward	Moves element to the back by one position. 

Renaming Pages

You can rename pages in a template.

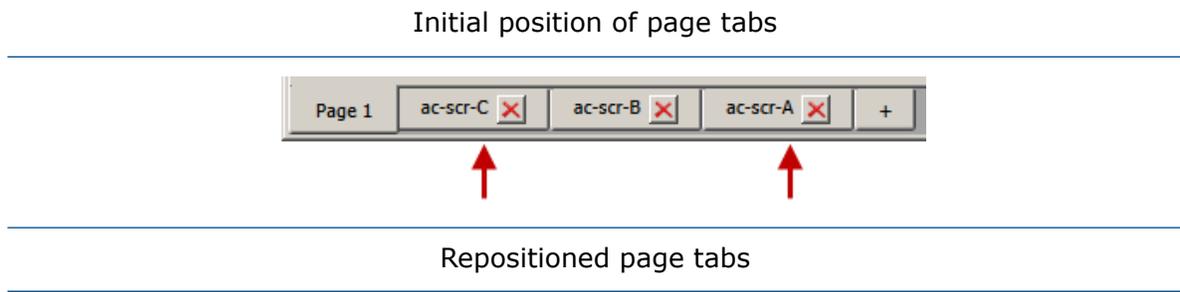
To rename pages:

1. Double-click the page name.
2. Type the new name.
3. Click off of the name to save the new name.

Repositioning Pages

You can change the position of the pages in a template by dragging and dropping the page tab to a new location. This lets you begin a document with a cover page or other introductory information such as a disclaimer or regulation statements.





Resizing Rows and Columns in Generic Tables

To resize rows or columns, position the pointer over the row or column divider and drag the divider.

Annotation Display Orientation

When publishing to 3D PDF, annotations are oriented so that they are readable in the default orientation of each 3D view.

Attaching Files to 3D PDFs

When you publish to 3D PDF, you can attach files.

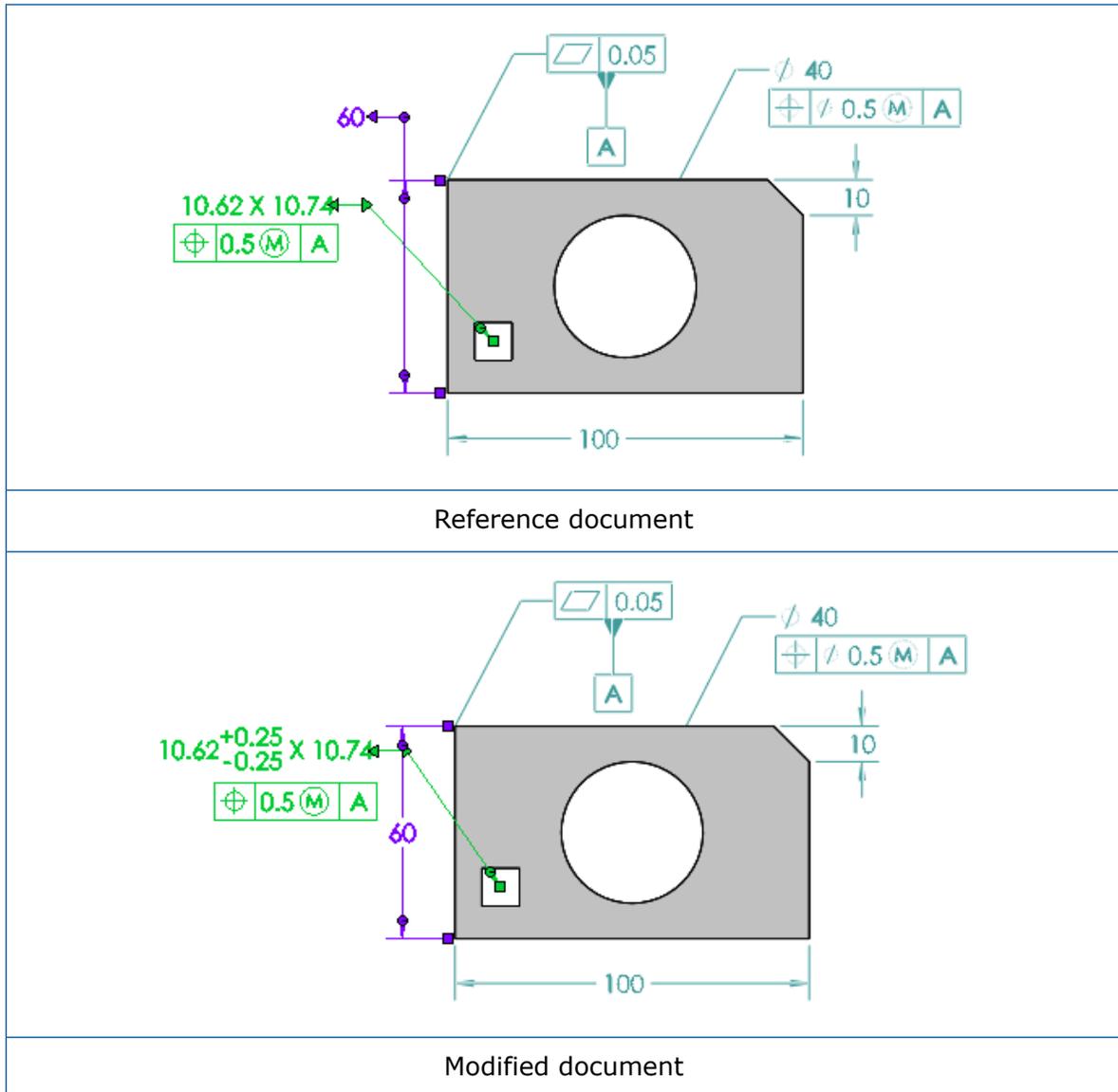
To attach files to 3D PDFs:

1. Click **Publish to 3D PDF** .
2. In the PropertyManager, under **Attachments**, click **Attach Files**.
3. Browse to the file to include and click **Open**.

The file is included as an attachment.

Comparing 3D PMI Between Parts

You can use the **3D PMI Compare** tool to compare DimXpert annotations, reference dimensions, and other annotations between two part documents.



To compare 3D PMI between parts:

1. Click **Tools > Compare > 3D PMI**.
2. In the Task Pane, select the **Reference document** and **Modified document**.
3. Click **Run Comparison**.

Exporting PMI Annotations to STEP 242

You can export SOLIDWORKS parts and assemblies with PMI annotations to STEP 242 format.

To export PMI annotations to STEP 242:

1. Click **Publish STEP 242 File**  (SOLIDWORKS MBD toolbar).
2. In the dialog box, type a file name.
3. Click **Save**.

Names Match in 3D PDFs

Names assigned to DimXpert PMI in 3D PDFs match the original names in the SOLIDWORKS software.

Reordering 3D Views

You can reorder 3D views on the 3D Views tab by using drag and drop or **Sort Order**.

Reordering 3D Views Using Drag and Drop

To reorder 3D views using drag and drop:

1. Select a 3D view.
2. Drag and drop it to a location.

When you reorder views using drag and drop, the reordered view is called **Custom**.

Reordering 3D Views Using Sort Order

To reorder 3D views using Sort Order:

On the 3D Views tab, under **Sort order**, select an option.

Option	Sorting Order
Name	Alphabetical by 3D view name.
Configuration	Alphabetical by configuration, then by 3D view name.
Display State	Alphabetical by display state, then by 3D view name.
Custom	Order saved from drag and drop reorganization.
History	Order created.

Sketches Display in 3D PDFs

Sketches that are visible in 3D Views display in 3D PDFs within the same 3D View.

17

Model Display

This chapter includes the following topics:

- **Controlling Decals and Scenes by Display States**
- **Displaying SOLIDWORKS Simulation Results in the Graphics Area**
- **Evaluating Models for 3D Printing**
- **PhotoView 360 Network Rendering Supports Frame-by-Frame Rendering**
- **Section Views**

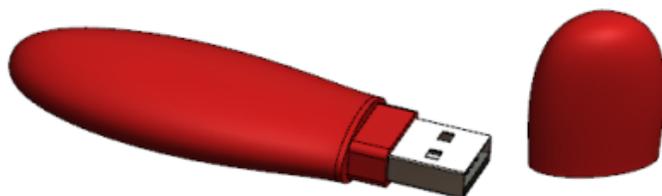
Controlling Decals and Scenes by Display States

You can define display states to use different instances of decals and scenes.

You can also configure light sources for all light types including sunlight. This feature requires PhotoView 360.

To control decals and scenes by display states:

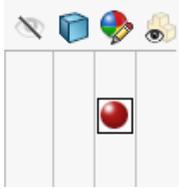
1. Open *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\model_display\usb_drive1\usb_drive1.sldasm.



2. In the FeatureManager design tree, right-click **usb_case1** and click **Open Part** .



3. Add a display state named `Barcode` and customize it:
 - a) Click **>** at the top of the FeatureManager design tree.
The Display Pane appears.



- b) Right-click in the Display Pane and click **Add Display State** .
- c) Type `Barcode` for the name of the field that appears at the top.
- d) Select the DisplayManager  tab.
- e) Click **View Decals**  and click **Open Decal Library**.
- f) In the Task Pane, drag the **Barcode** decal to the **Case_body**.

When you drag the decal and place it on your component, a manipulator appears above the decal. You can change the direction and alignment of the decal using the manipulator.

- g) In the Decals PropertyManager, under **Display States**, select **This display state** and click .
- h) In the graphics area, right-click and click **Edit Scene** .
- i) In the Task Pane, click **Presentation Scenes** and drag the **Kitchen Background** scene to the graphics area.
- j) Click .



4. Add another display state named `Designed_with_SOLIDWORKS` and customize it.
 - a) Right-click in the Display Pane and click **Add Display State** .
 - b) Type `Designed_with_SOLIDWORKS` for the name of the field that appears at the top.
 - c) In the DisplayManager, right-click `barcode1`  and click **Delete**.
 - d) In the Decals PropertyManager, click **Open Decal Library**.
 - e) In the Task Pane, drag the **Designed with SOLIDWORKS** decal to the **Case_body**.
 - f) In the Decals PropertyManager, under **Display States**, select **This display state** and click .
 - g) In the graphics area, right-click and click **Edit Scene** .
 - h) In the Task Pane, click **Presentation Scenes** and drag the **Factory Background** scene to the graphics area.
 - i) Click .



5. In the ConfigurationManager  tab, under **Display States**, double-click **Barcode** and **Designed_with_SOLIDWORKS** to change between display states.
6. In the Render Tools CommandManager, click **Display States Target** . The Display State Target dialog box appears.

When you change between display states, the active display state is selected in the Display State Target dialog box. This is useful for deleting or customizing the decals and scenes in the active display state.

Displaying SOLIDWORKS Simulation Results in the Graphics Area

You can display SOLIDWORKS Simulation results in the SOLIDWORKS graphics area along with other components or bodies in the model that are not part of the simulation.

This way, you do not have to switch between the simulation results and the modeling environment. It is not ideal to display simulation results on bodies and components. This display helps with marketing purposes and predictive design. Previously, if you ran a simulation on a component in an assembly, only the component was shown with the results and all other components were hidden.

An example of predictive design is making model changes based on simulation results.

Available with SOLIDWORKS Simulation licenses.

To display SOLIDWORKS Simulation results in the graphics area:

1. Open *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\model_display\sunglasses.sldasm.

If the Resolve Lightweight Components dialog box appears, click **OK**. The assembly has a simulation study saved with it.

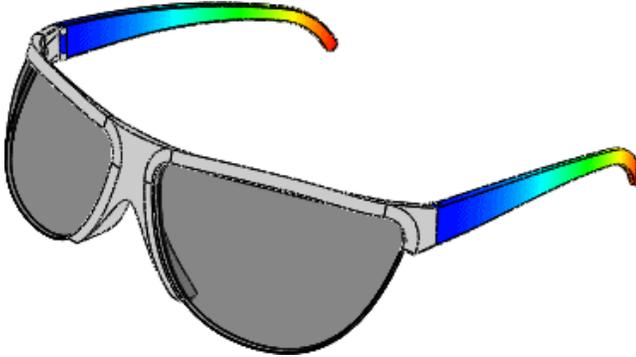


2. Click the Static 1 tab.
3. In the Simulation Study tree, right-click **Static 1**  and click **Run**.
4. Click the Model tab.
5. Click **Simulation Display**  (View toolbar) or **View > Display > Simulation Display**.

If the Simulation Display dialog box appears, click **OK**.

6. In the PropertyManager:
 - a) Under **Selected Result**, select **Displacement1**.
 - b) Click .

The assembly is displayed with the simulation results in the context of the assembly.



The **Simulation Display** command only lists result plots for the studies for which you have loaded all simulation data. To load all simulation data, click **Simulation > Options > System Options > General** and select **Load all Simulation Studies when opening the model (requires more time to open a model)**. You can also load simulation data for a specific study by clicking individual simulation tabs.

Evaluating Models for 3D Printing

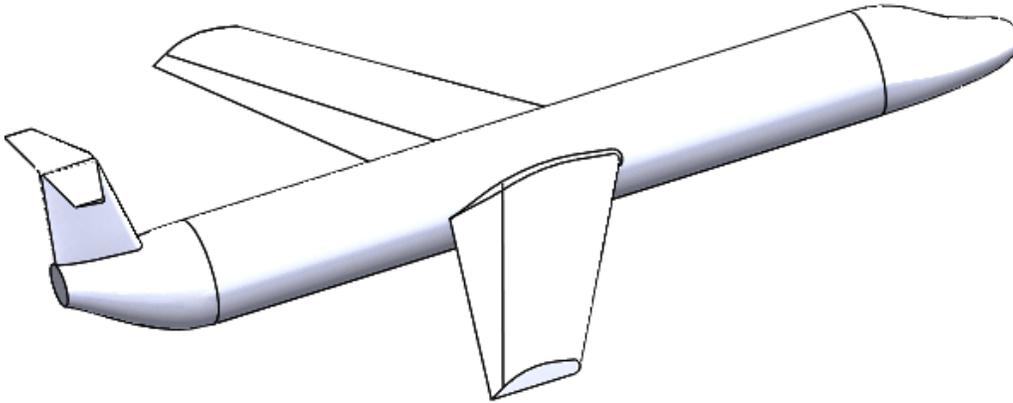
Before printing a model using a 3D printer, you can evaluate the thickness and gaps in a model to see if they can be printed using Fused Deposition Modeling (FDM) 3D printing technology.

Based on the layer height and the selected thermoplastic material, you can identify wall thickness and gap width issues before printing the model.

The **3DPrint** tool identifies where wall thickness or the distance between two faces in a groove are below a certain threshold, then highlights the geometry in the model. This tool is helpful when large models are scaled down or when detailed models are printed on a 3D printer.

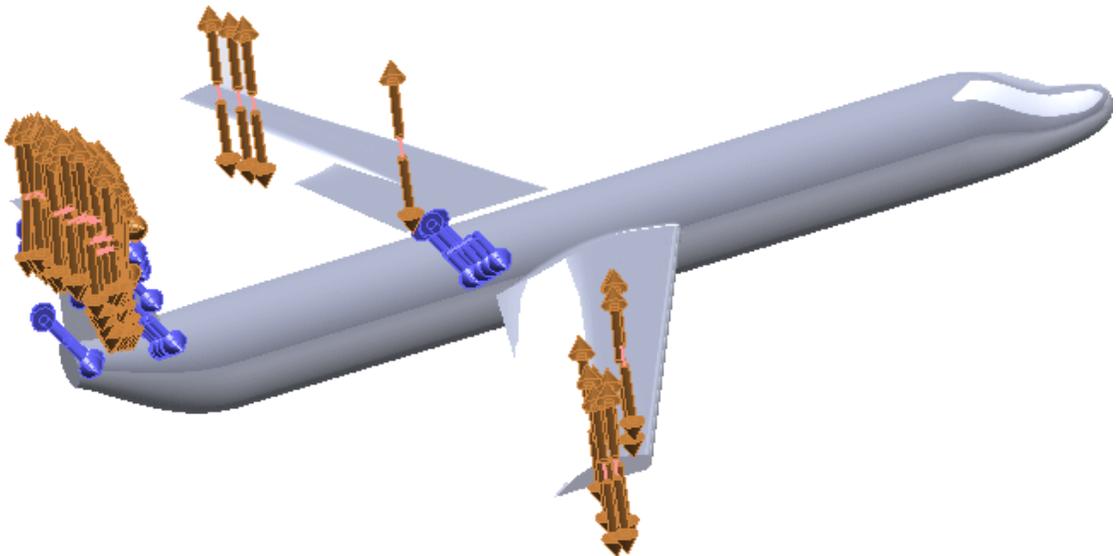
To evaluate models for 3D printing:

1. Open *drive letter:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS version\whatsnew\model_display\airplane.sldprt*.



2. Click **Print3D**  (Standard toolbar) or **File > Print3D**.
3. In the PropertyManager, click the Settings tab and do the following:
 - a) Set **Bottom plane of model**.
 - b) Set the value for **Scale**.
 - c) Click **Scale to Fit**.
4. In the PropertyManager, click the Preview tab and do the following under **Layer height**:
 - a) Set **Layer Height** to 0.5.
 - b) Select **Thickness/Gap Analysis** and **Custom thickness and gap**.
 - c) Set **Thickness/Gap** to 2.5.
 - d) Set **Angle** to 178.
 - e) Click **Calculate**.

Areas of the model that are too thin or with too large of gaps are shown.



PhotoView 360 Network Rendering Supports Frame-by-Frame Rendering

PhotoView 360 supports “frame-by-frame” network rendering when you save an animation, which increases rendering performance.

The PhotoView 360 network rendering feature allows you to distribute rendering for an animation to up to 10 networked client machines running the PhotoView 360 Net Render Client.

In previous releases several client computers would share the work of rendering the same frame, which requires a significant amount of network intercommunication. In SOLIDWORKS 2017, each client computer works independently on its own frames. This reduces the need for communication among the client computers and can dramatically shorten the time needed to complete rendering. No changes are required in the way the clients are configured.

To configure frame-by-frame rendering, you only need to set up a network of computers.

The list of available computers displays at the bottom of the PhotoView Network Render Client interface. You can also view the list by expanding the tab at the bottom of the final render window.

Note that computers on wireless networks do not interact well when it comes to network rendering or frame-by-frame animation rendering. The Coordinator computer or the computer running the PhotoView Network Render Client should not be using a wireless connection to access the network.

Section Views

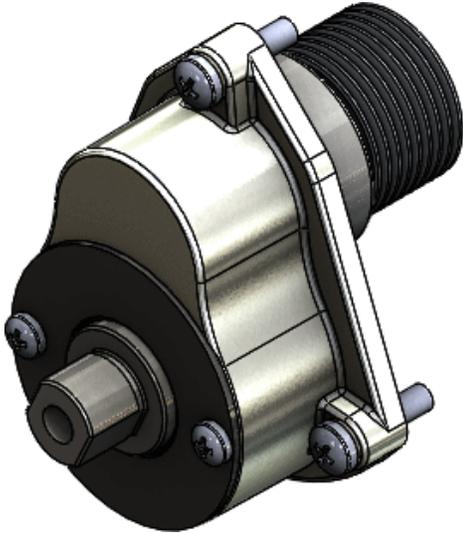
Creating Transparent Section Views

While creating section views, you can set individual bodies and components as transparent.

The transparency helps you see through to the internals of the model. The transparent section view is available in PhotoView 360.

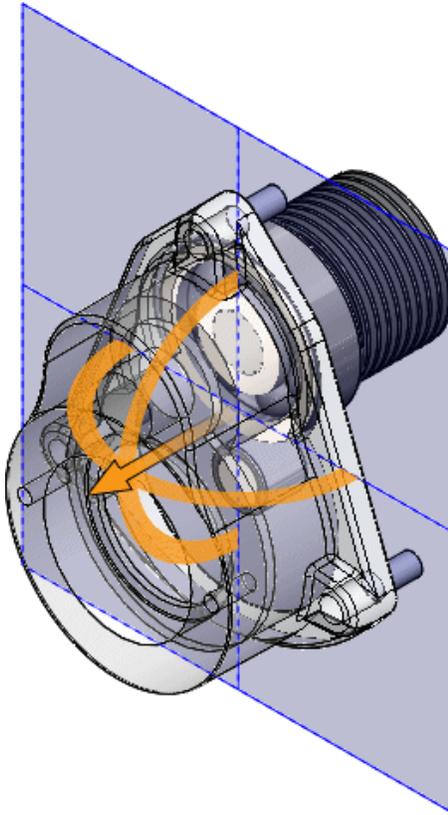
To create transparent section views:

1. Open *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\model_display\gearbox\GEARBOX.sldasm.

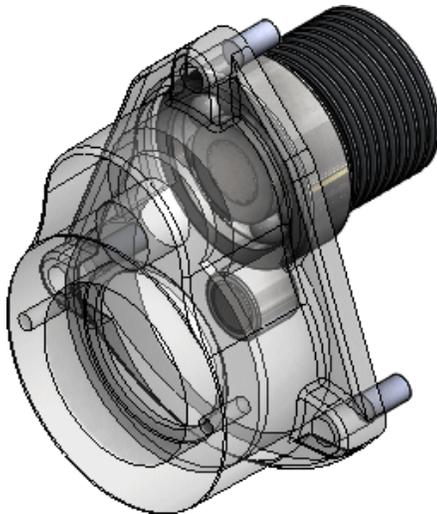


2. Click **Section View**  (View toolbar) or **View > Display > Section View**.
3. In the PropertyManager:
 - a) Under **Section Method**, select **Zonal**.
 - b) Select **Transparently Section Components**.
4. In the flyout FeatureManager design tree, select **HOUSING - GEARS**.

5. In the PropertyManager, under **Transparently Section Components**:
 - a) Select **Include selected** to make the housing transparent.
 - b) Set **Transparent amount** to 0.75.

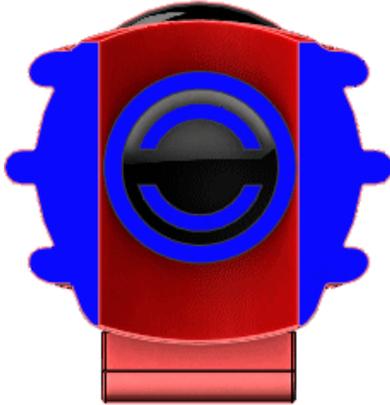


- c) Click .
- The section view appears with a transparent housing.



Keep Cap Color

In the Section View PropertyManager, the **Keep cap color** option is selected by default. Additionally, in previous versions, if you selected **Graphics-only section**, the **Keep cap color** option was automatically selected. This no longer occurs. When selected, **Keep Cap Color** is always reflected in the model display with a solid color cap or a part colored cap.



18

Parts and Features

This chapter includes the following topics:

- **Advanced Holes**
- **Bidirectional Circular Patterns**
- **Chamfer Enhancements**
- **Converting Features to Bodies and Surfaces**
- **Cut List Sorting**
- **Derived Part References are Retained**
- **Disabling and Enabling Equations in All Configurations**
- **Extrude from Any Size Planar Face**
- **Rebuilding All Configurations**
- **Running FeatureWorks After Adding Items to Imported Parts**
- **Select All for Split Features and Save Bodies**
- **Sweep Profile Selection of Faces, Edges, and Curves**
- **Thread Enhancements**
- **What's Wrong Enhancements**
- **Window Selection for Instances to Skip**
- **Wrap Creates Geometry on Any Face**

Advanced Holes

With the **Advanced Hole** tool, you can define advanced holes from the near and far side faces. Hole element flyouts help guide the process. A **Favorites** group box is available in the PropertyManager.

Creating Advanced Hole Elements

1. Create a simple extruded block part.

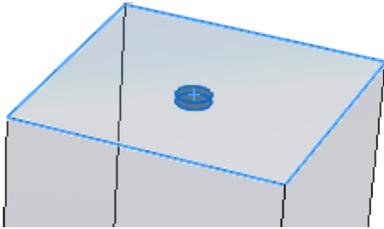
To better visualize the previews, **Ctrl +select** all faces and click **Appearances**. In the Task Pane, under **Glass**, double-click **clear glass** to apply this appearance.

2. Click **Advanced Hole**  (Features toolbar) or **Insert > Features > Advanced Hole**.

The PropertyManager opens with the **Near Side** flyout displayed.

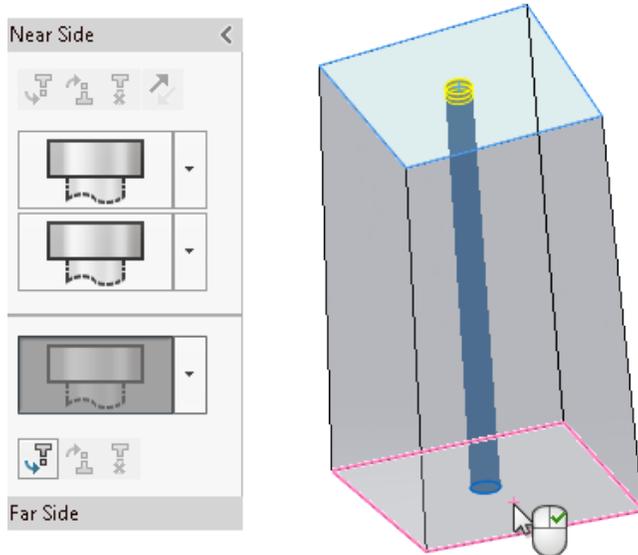
3. In the PropertyManager, on the Type tab, under **Near And Far Side Faces**, select a face.

A temporary preview of the hole appears. The location is temporary, based on your initial selection on the face. You set the position later.



4. In the **Near Side** flyout, click **Insert Element Below Active Element**  to add a **Near Side** element to the advanced hole.
5. In the PropertyManager, under **Near and Far Side Faces**, select **Far Side**, then select the opposite face on the model.

The **Near Side** flyout now displays two **Near Side** elements. The **Far Side** flyout is active and contains one element. A bar separates the flyouts.

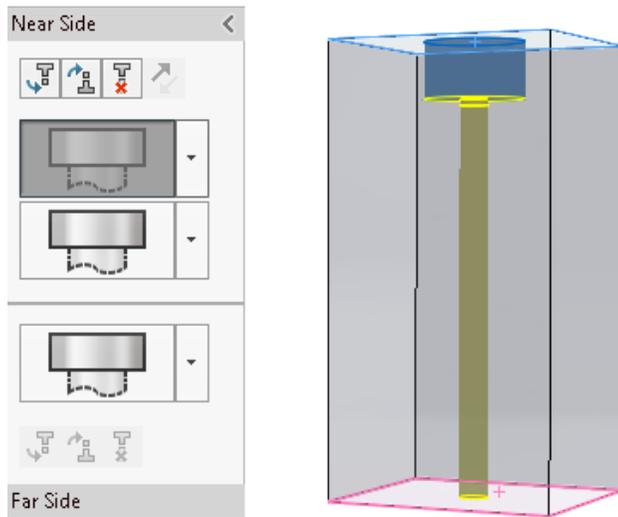


Next, you set the element specifications.

Setting Element Specifications

1. In the **Near Side** flyout, select the top element to activate it. You want to keep **Counterbore** as the hole type but modify some settings.
2. Under **Element Specification**:
 - a. In **Type**, select **Socket Head Cap Screw**.
 - b. In **Size**, select **1/4**.

The temporary preview of the **Near Side** element is updated.



3. In the flyout, click ▾ for the second **Near Side** element, then click **Straight**  .
You want to modify the specifications for this element.

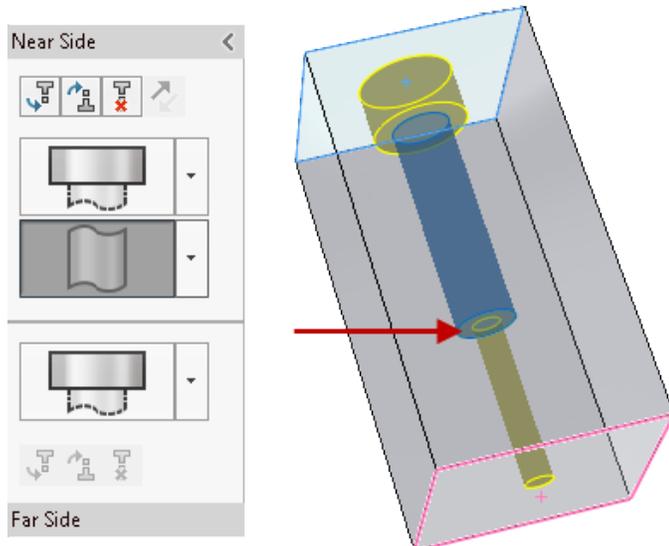
4. Under **Element Specification**:

a. In **Type**, select **Screw Clearances**.

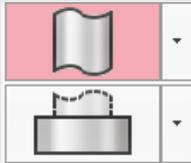
b. In **Size**, select **1/4**.

c. For **Custom sizing**, set **End Condition** to **Blind** and **Depth**  to **1**.

The preview updates to show the modified size and depth for the **Straight** element.



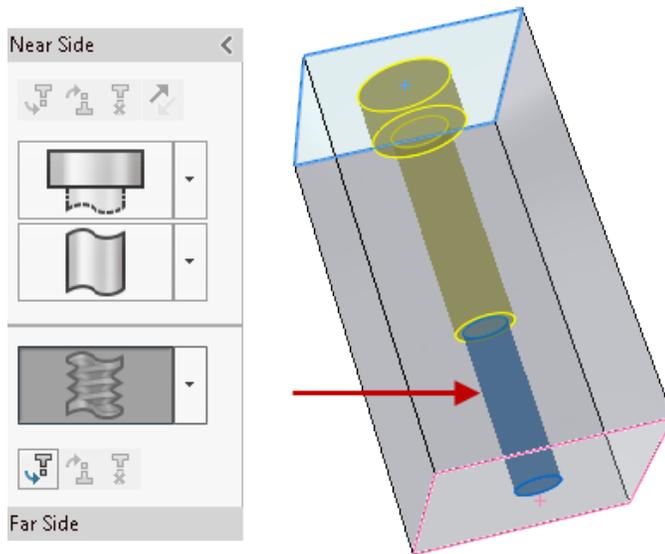
In the flyouts, you can drag elements to reposition them. If the drag operation causes a failed element, that element is highlighted. Hover over the element to display an error message.



The **Far Side** element is set as **Counterbore**. You want to modify it to be a tapped thread element.

5. In the **Far Side** flyout, click ▾ for the **Far Side** element, then click **Thread** .
6. Under **Element Specification**:
 - a. In **Type**, select **Tapped hole**.
 - b. In **Size**, select **1/4-20**.

The **Far Side** element updates up to the middle **Near Side** element because of the **Up to Next Element** end condition.



You can save advanced hole features as **Favorites** to reuse them. In the PropertyManager, on the Type tab, under **Favorite**, you can add, update, delete, save, or load a favorite advanced hole.

Next you create and position instances.

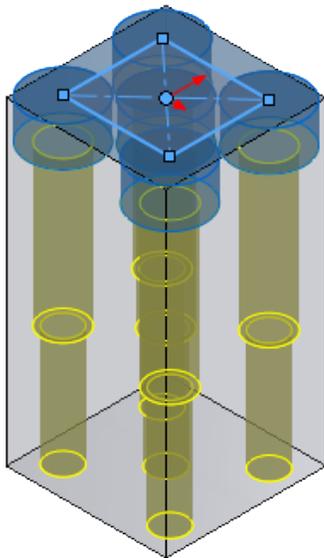
Creating and Positioning Advanced Hole Instances

1. In the PropertyManager, select the Positions tab.
You can select multiple points or use a sketch to create multiple holes.

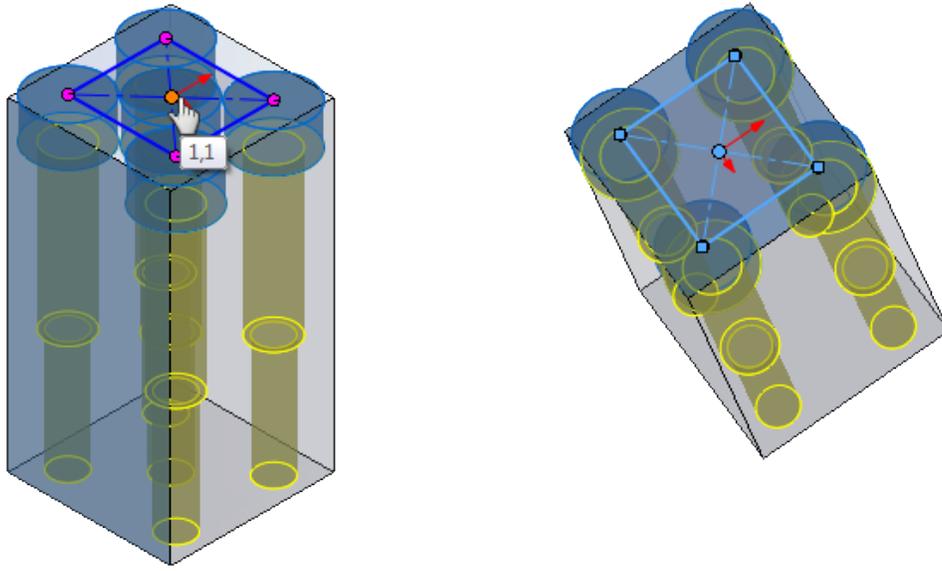
If you forget to select the Positions tab and click ✓, the software prompts you to select a point on the face to locate the hole. Click **OK** in the prompt to open the Positions tab and the Sketch toolbar.

2. Click **Center Rectangle**  (Sketch toolbar) and sketch a rectangle, using the sketch origin  on the face as the center.
3. Click ✓ to close the Center Rectangle PropertyManager and return to the Hole Position PropertyManager.

The software creates five instances of the hole, including a sketch point at the selected origin. You want to remove the center instance.



4. Under **Instances to Skip**, select the center instance orb to remove only this instance from the preview.



Under **Sketch Options**, you can select **Create instances on construction geometry** to create instances using construction geometry.

Bidirectional Circular Patterns

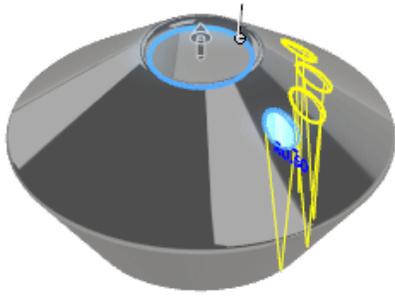
You can create circular patterns symmetrically or asymmetrically in both directions from the seed geometry. This is useful when the seed is not located at the end of the pattern arc.

You can adjust the angle, number of instances, and spacing settings for each direction independently. The **Symmetric** option applies the same settings to both directions.

To create bidirectional circular patterns:

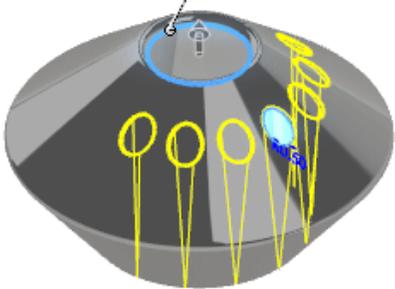
1. In a part model, click **Circular Pattern**  (Features toolbar) or **Insert > Pattern/Mirror > Circular Pattern**.
2. In the PropertyManager, under **Features to Pattern** , select a seed feature to pattern.
3. Under **Direction 1**, select a **Pattern Axis**.

- Set **Angle**  to 25 and **Number of Instances**  to 4.



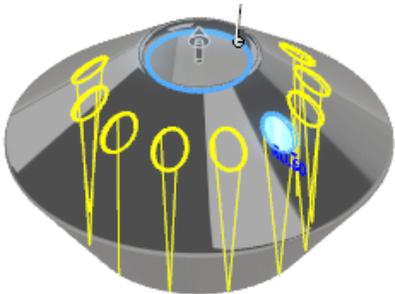
- Select **Direction 2**, then select **Symmetric**.

Three instances of the same seed appear symmetrically in the reverse direction.



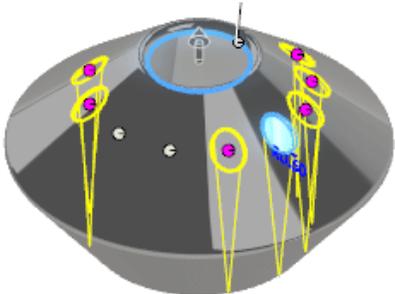
- Clear **Symmetric**, and set **Angle**  to 30 and **Number of Instances**  to 6.

The pattern updates in **Direction 2** only. The **Direction 1** pattern is unchanged.

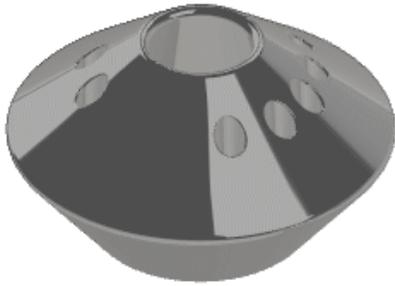


- Under **Instances to Skip**, select instances 2 and 3 in **Direction 2**.

The instance orbs turn white, indicating the skipped instances.



8. Click  to complete the pattern.

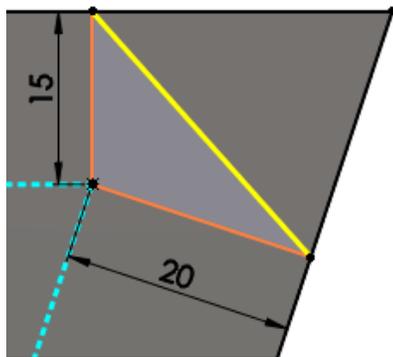


Chamfer Enhancements

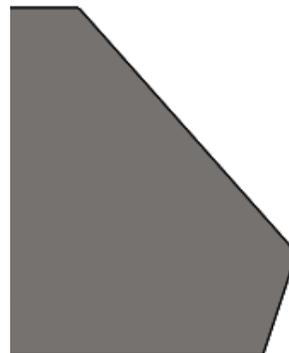
The Chamfer PropertyManager has been updated to align with the **Fillet** tool. There are two new chamfer types, **Offset Face** and **Face Face**, which you can convert from chamfers to fillets and back.

Offset Face Chamfers

Offset face chamfers are solved by offsetting the faces adjacent to selected edges. The software calculates the intersection point of the offset faces, then calculates the normal from that point to each face to create the chamfer.



Calculation of chamfer offset



Chamfer applied

This method yields predictable results when chamfering between non-planar faces. Offset face chamfers can change direction on an edge-by-edge basis, and they support chamfering entire features and surface geometry.

Face Face Chamfers

Face face chamfers can create symmetric, asymmetric, hold line, and chord width chamfers.

There are two ways to convert offset face and face face chamfers to fillets and back.

- Right-click a chamfer feature and click **Edit Feature** . In the PropertyManager, for **Feature Type**, click **Fillet Type** or **Chamfer Type**, and click .
- Right-click a chamfer feature and click **Convert Chamfer to Fillet** or **Convert Fillet to Chamfer**. The PropertyManager opens with the converted **Feature Type** already selected. Click .

There are no changes in functionality to the other chamfer types. Their PropertyManagers have been reorganized.

To access the new chamfer types, click **Chamfer**  (Features toolbar) or **Insert > Features > Chamfer**. In the PropertyManager, under **Chamfer Type**, select **Offset**

Face  or **Face Face** .

Converting Features to Bodies and Surfaces

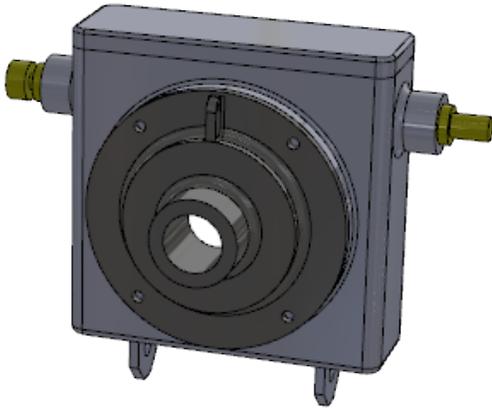
For parts, you can convert features to bodies and surfaces which maintaining geometric references from other parts, assemblies, and drawings. This lets you remove unneeded feature history, while retaining the bodies and surfaces.

To access this functionality, in the FeatureManager design tree, right-click the part name and click **Convert to Bodies**.

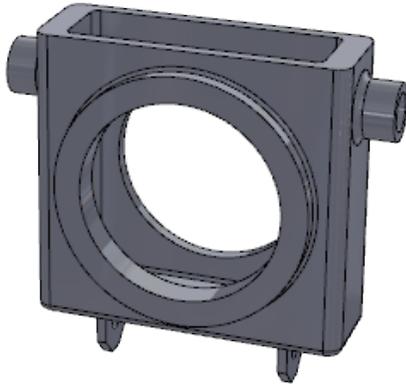
To convert features to bodies and surfaces:

1. Open an assembly file, such as the `HousingAssy.SLDASM` model shown.

The **Housing** component is part of an assembly. Two components are external references to **Housing**. Another component is mated to **Housing**.

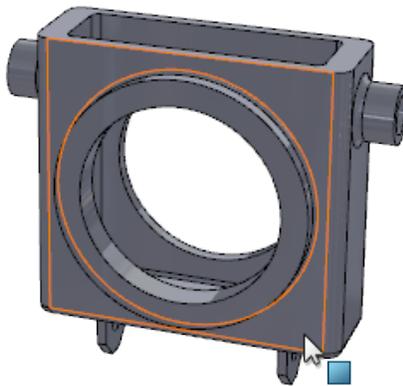


2. In the FeatureManager design tree, right-click **Housing** and click **Open Part** . The **Housing** part opens.



3. At the top of the FeatureManager design tree, right-click **Housing**  and click **Convert to Bodies**.
4. In the dialog box:
 - a) Under **File Name**, change the name to `Housing2.SLDPRT`.
 - b) Select **Save as**.
 - c) Select **Preserve reference geometry and sketches**.
 - d) Click **OK**.

The converted file retains all the sketches and **Plane1** geometry.



5. Click **Window** > **HousingAssy.SLDASM** to return to the assembly. Because the assembly was open in the background, the original **Housing** component was replaced with **Housing2**.
6. In the FeatureManager design tree, expand **Mates** . There were no failures to external references or mates, which demonstrates that the model kept all of its geometric references.

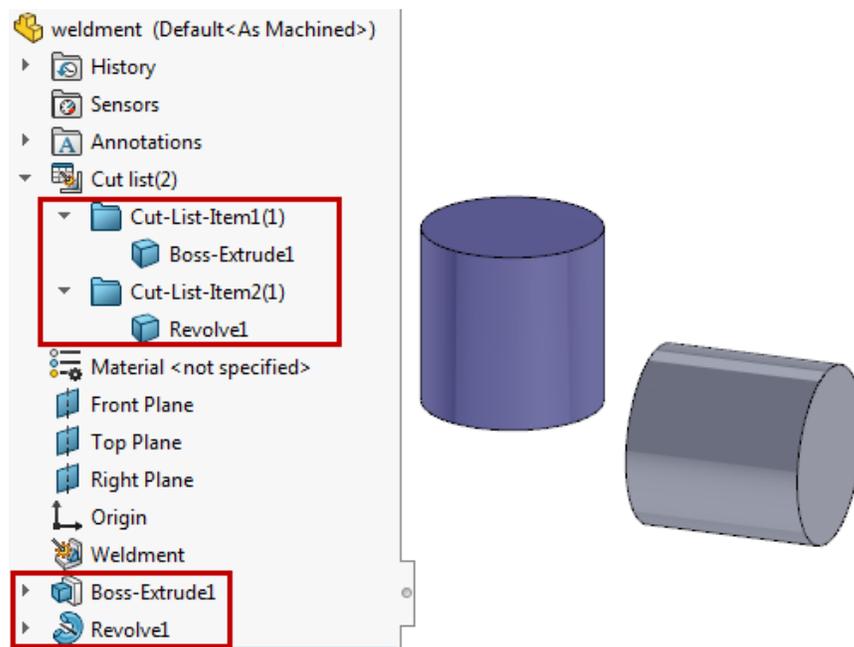
Cut List Sorting

You can choose sorting options for cut lists from a shortcut menu on the top level of the **Cut list** folder and from the Document Properties tab.

You can collect identical bodies and choose faces and features to exclude in sorting. This option collects all bodies in a **Cut-List-Item** folder that are geometrically identical, but are generated by different features. For example, cylinders that are created by extrude or revolve features, but are identical in volume.

To use cut list sorting:

In this model, **Extrude1** and **Revolve1** bodies have the same volume, but appear in separate **Cut-List-Item** folders.

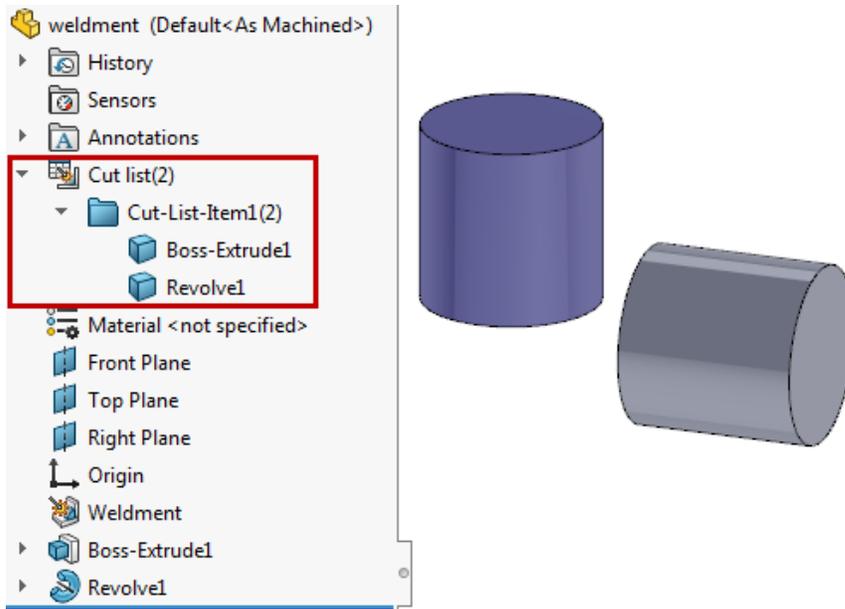


Do one of the following:

- In the FeatureManager design tree, right-click the top level of the **Cut list** folder and click **Cut List Sorting Options**. Under **Sorting Options**, select **Collect Identical Bodies** and click **✓**.
- Click **Tools > Options > Document Properties > Weldments**. Under **Cut list options**, select **Collect identical bodies**.

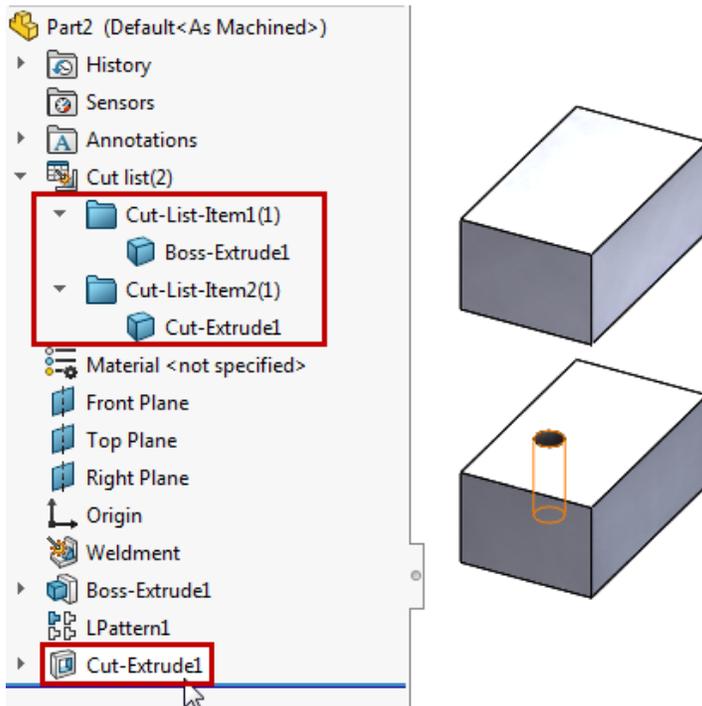
To preserve the **Collect identical bodies** option in the Document Properties dialog box, create a template with this option selected. When you create new weldment parts with this template, the software automatically checks for identical bodies in the **Cut-List-Item** folders.

Both bodies now appear in the same **Cut-List-Item** folder.



To choose faces and features to exclude:

In this model, **LPattern1** body was created from **Boss-Extrude1** body, then **Cut-Extrude1** was added to one pattern instance. You want to exclude the **Cut-Extrude1** feature geometry from the cut list sorting.

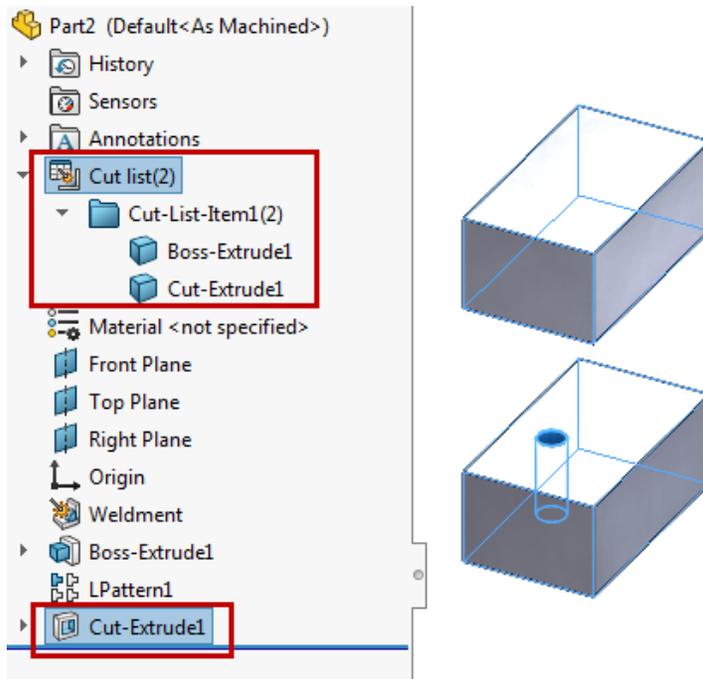


1. In the FeatureManager design tree, right-click the top level of the **Cut list** folder and click **Cut List Sorting Options**.
2. In the PropertyManager, under **Faces/Features to Exclude**, select **Cut-Extrude1** from the FeatureManager design tree.

You can select features from the FeatureManager design tree or faces from bodies in the graphics area. This option is useful for collecting bodies generated from identical stock material by ignoring features such as holes, fillets, and chamfers.

3. Click .

The bodies **Boss-Extrude1** and **Cut-Extrude1** appear in a single **Cut-List-Item** folder. The **Cut-Extrude1** feature geometry was excluded when comparing bodies in the cut list sorting process.



Derived Part References are Retained

Derived parts retain their references when you modify the original body using Boolean operations with features such as combine, cut extrude, split, and core.

Previously you received an error message that the base part had no solid when you performed Boolean operations in the parent part.

Disabling and Enabling Equations in All Configurations

You can disable and enable equations in the **Equation View** , **Sketch Equation View** , or **Ordered View**  in the Equations, Global Variables, and Dimensions dialog box.

The equations are disabled in all configurations. You can enable the equations if no other active equation controls the same parameter.

To disable equations, in Equations, Global Variables, and Dimensions dialog box, in any view, right-click an equation and click **Disable Equation**. The equation disappears from the view. **Ordered View**  retains all equations. The disabled equations are unavailable. To enable a disabled equation, in the Equations, Global Variables, and Dimensions dialog box, click **Ordered View** . Right-click the equation and click **Enable Equation**. The equation reappears in all views where it previously appeared.

In **Dimension View** , you can enable and disable **Global Variables** and **Features**, but not **Dimensions**.

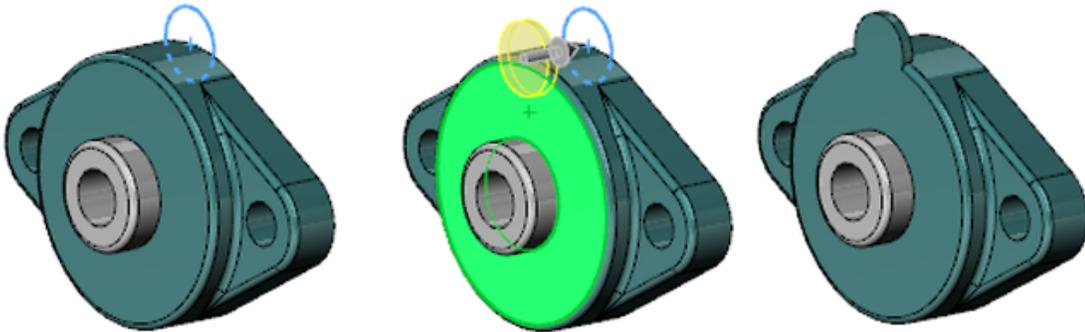
Extrude from Any Size Planar Face

You can create boss, cut, and surface extrudes from any size surface, face, or plane. The surface or face must be planar. Previously the **From > Surface/Face/Plane** selection needed to encapsulate the entire sketch.

To extrude from any size planar face:

- In parts, click **Extruded Boss/Base**  (Features toolbar) or **Insert > Boss-Base > Extrude**. This functionality is also available for **Extruded Cut**  and **Extruded Surface** .
- In assemblies, click **Insert > Assembly Feature > Cut > Extrude**.

Select a sketch. Under **From**, select **Surface/Face/Plane**, and select a planar surface or face, or a plane.



Rebuilding All Configurations

The **Rebuild All Configurations**  tool is available in multiple, more visible locations.

To rebuild all configurations:

Do one of the following:

- In the ConfigurationManager, right-click the file name and click **Rebuild All Configurations**.

- In a file, click **Edit > Rebuild All Configurations**.
- Press **Ctrl + Shift + B**.

Rebuild All Configurations is available only when there are multiple configurations in a file.

Running FeatureWorks After Adding Items to Imported Parts

You can add various features and references to an imported part and not lose them if you subsequently run FeatureWorks on the imported part.

The following features and references are supported:

- Drawing views
- Drawing annotations
- Drawing dimensions
- Assembly mates
- Assembly features

Select All for Split Features and Save Bodies

You can use **Select All** for split features to select all split bodies without saving the resulting bodies to new parts.

Previously you had to select check boxes individually, or select **Auto-assign Names**, which created separate, unnecessary part files for each body.

To select all bodies:

Do one of the following:

- Click **Insert > Features > Split**. Under **Resulting Bodies**, click **Select All** .
- Click **Insert > Features > Save Bodies**. Under **Resulting Parts**, click **Select All** .

Sweep Profile Selection of Faces, Edges, and Curves

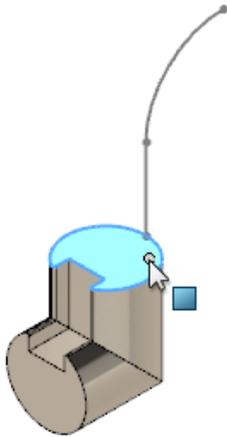


You can select faces, edges, and curves directly from models as sweep profiles. Enhanced selection is supported with Boss, Base, Cut, Surface, and Assembly cut sweep features. Previously you needed to create an extra sketch with converted entities as the profile.

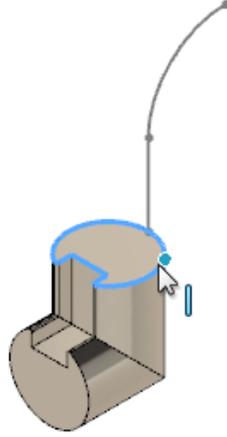
You can select:

- Faces from model geometry.
- A single edge or reference geometry curves that contain a smooth, closed loop.
- A group of edges or curves selected as a loop using SelectionManager.

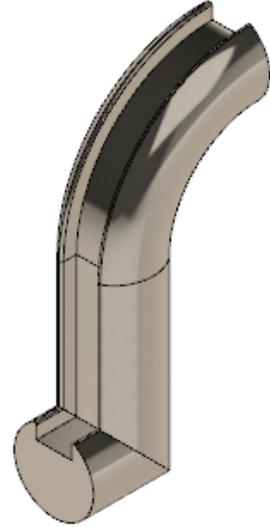
To access this functionality, click **Swept Boss/Base**  (Features toolbar) or **Insert > Boss/Base > Sweep**. In the PropertyManager, under **Profile and Path**, select **Sketch Profile**, then select a face, edge, or curves.



Select face for **Sketch Profile**



Select edges using the SelectionManager for **Sketch Profile**



Sweep feature

Thread Enhancements

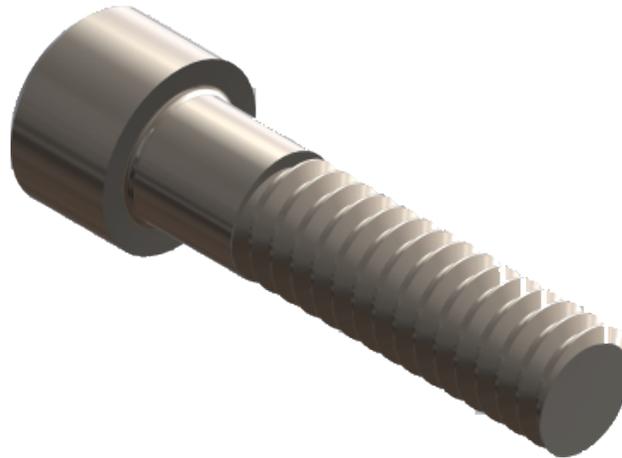
Trimmed Threads

You can align threads to end faces. A **Cut thread** is extended and cut to match the end face. An **Extrude thread** is cut to match the end face. In the PropertyManager, under **Thread Options**, select **Trim with start face** and **Trim with end face**.

Before trim

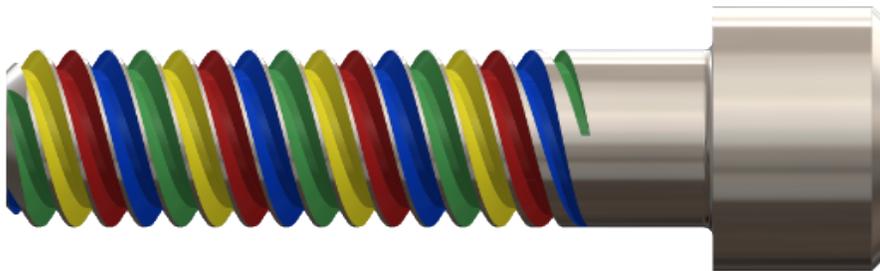


After trim



Multiple Starts

Threads can have multiple starts. In the PropertyManager, under **Thread Options**, select **Multiple Start**. Set the number of starts to define the number of times the thread is created in an evenly-spaced circular pattern around the hole or shaft. The image shows a four-start thread with a different color per thread.



The thread pitch value must permit multiple starts without causing crossing, self-intersecting threads. In the example, the pitch of the yellow thread is wide enough to allow the green, blue, and red threads to be nested inside of it.

Favorites

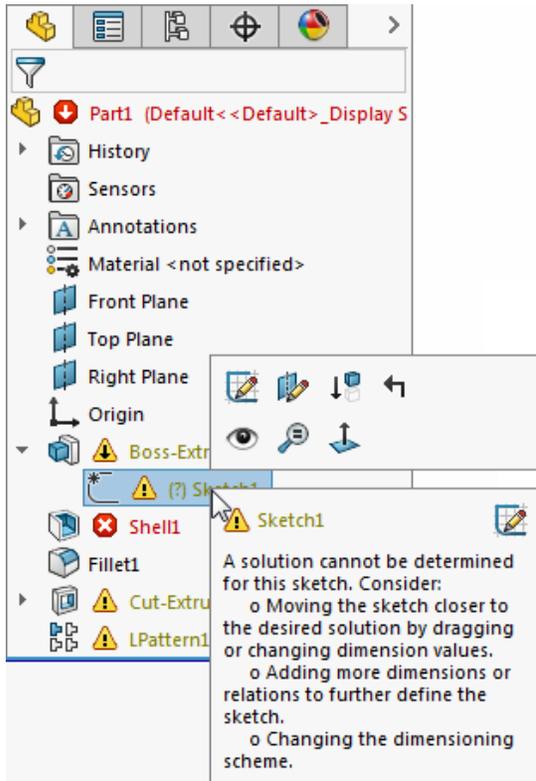
A **Favorites** group box is available in the PropertyManager.

What's Wrong Enhancements

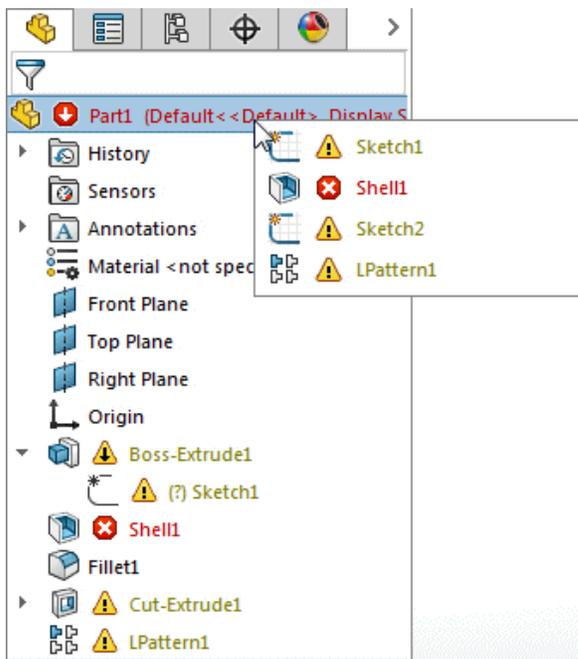
What's Wrong details and calls to action appear within the FeatureManager design tree, breadcrumbs, and graphics area. You no longer need to open the What's Wrong dialog box.

To display a contextual **What's Wrong** box, select an error or warning in the FeatureManager design tree, breadcrumbs, or graphics area. The box describes the issue

and offers suggestions to correct it. Links appear to either **Edit Feature**  or **Edit Sketch** . Depending on the situation, there may also be links to **Preview**  and **Help** .



To display a list of all errors and warnings, left-click the top node in the FeatureManager design tree. Click an item in the list to select the item in the FeatureManager design tree.



You can still display the What's Wrong dialog box. Right-click the feature with an error or warning and click **What's Wrong**.

Window Selection for Instances to Skip

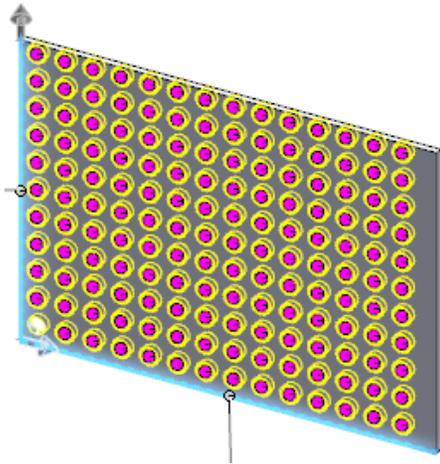
You can use box and lasso selection to add or remove instances to skip in patterns.

To use window selection for instances to skip:

1. Create or edit a pattern feature.

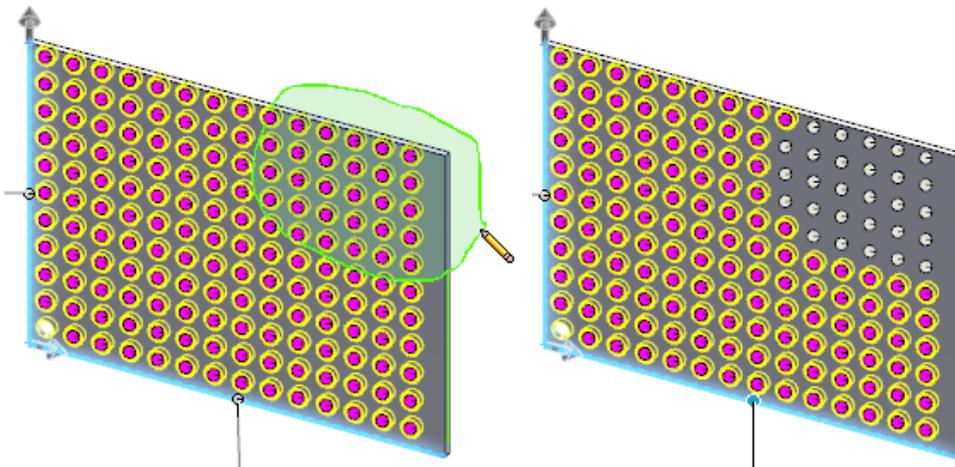
This selection does not work for sketch patterns.

2. In the PropertyManager, expand **Instances to Skip**.
Selection orbs appear for all instances of the pattern.



3. In the graphics area, right-click and click **Lasso Selection**, and select instances to skip.

The orbs change to white, indicating they are skipped from the pattern. The instances are listed under **Instances to Skip**.



4. Click ✓.

- If the selection contains skipped and non-skipped instances, the software toggles the instances from their current state. Skipped instances become non-skipped and vice versa.
- **Shift +** select only adds instances to **Instances to Skip**.
- **Alt +** select only removes instances from **Instances to Skip**.
- Selection orbs for instances must be completely within the selection region to be added to or removed from **Instances to Skip**.

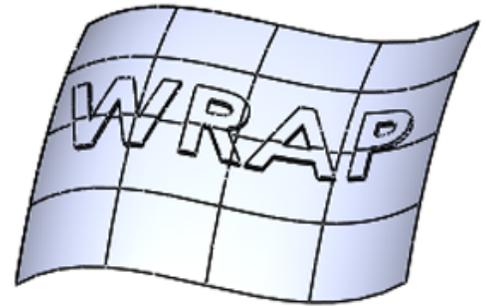
Wrap Creates Geometry on Any Face

The **Wrap** feature lets you create geometry on any face type. Not only can you wrap a sketch on all face types, you can wrap a sketch on multiple faces. Previously, the **Wrap** feature only handled cylindrical and cone shaped faces, and you could only work on a single face at a time.

You can choose between two methods to create a wrap. The **Analytical** method preserves legacy behavior where you can wrap a sketch completely around a cylinder or cone. The **Spline Surface** method wraps a sketch on any face type. A limitation with this method is that it cannot wrap around a model.



Analytical method

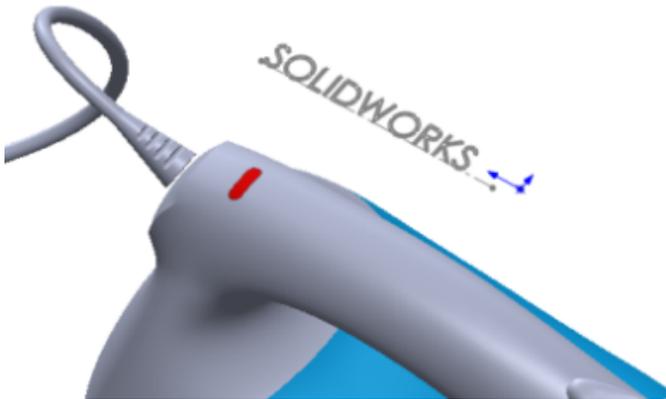


Spline Surface method

The PropertyManager and the preview have also been redesigned. The options in the PropertyManager are listed in a more logical order with large icons to help you distinguish between the various wrap types available. The preview displays a dot guide to show where a wrap starts from on a target face.

To create wrap geometry on any face:

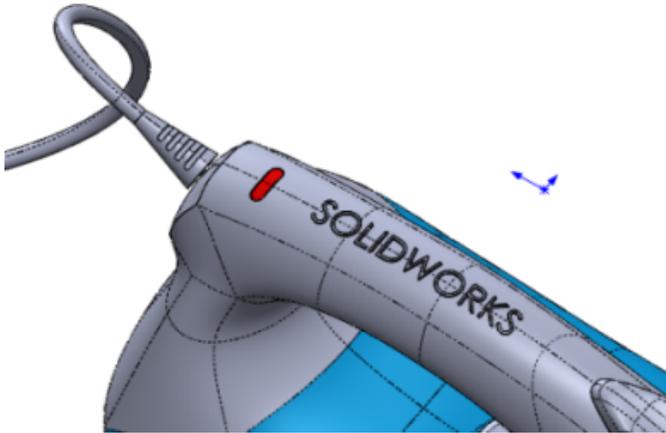
1. Open *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\parts\iron.sldprt.



2. Click **Wrap**  (Features toolbar) or **Insert** > **Features** > **Wrap**.
3. In the graphics area, select the SOLIDWORKS sketch.
4. In the PropertyManager:
 - a) For **Wrap Type**, click **Emboss** .
 - b) For **Wrap Method**, click **Spline Surface** .
5. Select the faces along the top handle of the iron. There are a total of six.



6. In the PropertyManager, under **Wrap Parameters**, set the **Thickness**  to 2mm and click .



If you want to make the wrap feature stand out, you can change its appearance by coloring the decal or turning the edges on or off.

19

SOLIDWORKS PDM

This chapter includes the following topics:

- **Adding Custom Columns**
- **Changing a Category of a Checked in File**
- **Controlling the Download Permissions for Users and Groups**
- **Copying Group Permissions and Group Members**
- **Generating SOLIDWORKS MBD 3D PDFs**
- **Latest Version Overwrite**
- **Preview Enhancements**
- **Quality Enhancements in SOLIDWORKS PDM**
- **Reference Tree Display Enhancements**
- **Rolling Back a File with References**
- **Searching Local Files**
- **SOLIDWORKS PDM Client and Server Service Pack Compatibility**
- **SOLIDWORKS PDF Task Add-in**
- **SOLIDWORKS PDM Web2 Viewer Enhancements**
- **Vault Database Replication**

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

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

Adding Custom Columns

The Custom Column functionality has expanded to the Do Transition (Change State) and Get dialog boxes.

To add custom columns:

1. Do one of the following:
 - Right-click any column heading and click **More**.
 - Right-click anywhere in the dialog box and click **Columns > More**.

2. Select the columns to display.

The added custom columns appear alphabetically on the right side of the default columns. You can sort, resize, and reorder the custom columns. SOLIDWORKS PDM saves these settings per user.

Changing a Category of a Checked in File

Admin login can change a Category of a checked in file doing one of the following:

- Select the file and click **Modify > Change Category**.
- Right-click the file and click **Change Category**.

Controlling the Download Permissions for Users and Groups

In the Web2 client, Administrators can control download permissions for users and groups by adding and configuring the following settings into the `<configuration>` node of the `Web.config` file:

```
<vaultSettings>
<vault name="EPDM" defaultDelimiter="|" denyDownloadGroups="Group1">
<user name="admin" denyDownload="false"/></vault>
<user name="User1" denyDownload="true"/></vault>
</vaultSettings>
```

Vault setting	Description
name	Name of the vault. Create a separate <code><vaultSettings></code> section for each vault that needs download permissions controlled.
defaultDelimiter	Delimiter used to separate group names in the <code>denyDownloadGroups</code> setting. The default delimiter is <code> </code> .
denyDownloadGroups	Groups that have download permissions denied in the specified vault. You must separate group names by the delimiter specified in the <code>defaultDelimiter</code> setting.

User setting	Description
name	User login name. Create a separate <code><user></code> node for each user.
denyDownload	Denies download permissions for users from the specified vault. If set to <code>false</code> , the user can download files. If set to <code>true</code> , the user cannot download files.

A user cannot download files if the user is denied as part of group or as a user.

Copying Group Permissions and Group Members

Administrators can copy group permissions and group members from an existing group when creating a new group and while importing from Active Directory.

To copy group permissions and group members:

1. Right-click **Groups** and click **New Group**.
2. In the dialog box, for **Group Name**, type a name.
3. In **Copy from group**, select the source group.
4. Select **Copy Permissions** and/or **Copy Members**.
5. Click **Next** and **OK**.

Generating SOLIDWORKS MBD 3D PDFs

SOLIDWORKS PDM Professional customers who use SOLIDWORKS MBD functionality, can automatically generate SOLIDWORKS MBD 3D PDF output using Convert Task.

This helps you define, organize, and publish 3D product and manufacturing information (PMI), including 3D model data, in industry standard file formats. This functionality enhances the integration between SOLIDWORKS PDM and SOLIDWORKS MBD.

To generate SOLIDWORKS MBD 3D PDFs:

1. In the Administration tool, on the left pane of the Convert - Properties dialog box, click **Conversion Settings**.
2. Set **Output file format** to **3D PDF - MBD (*.pdf)**.

Assembly or Part

The Assembly and Part tabs let you define the themes and views to publish in the output.

Theme for SW Assembly or Theme for SW Part

3D PDF Theme Path

Path of the theme file. If the theme file is in the vault, it must be cached before the task execution. If the theme file is outside the vault, it must be accessible to the task host.

Allow the user to change the theme path

Lets users change the path of the theme file during the task execution.

Select Views to include

3D Views

(Default.) Includes all captured 3D views except the current model view in the output file.

Primary Views

Lets you select the primary views like front, top, and back to include in the output file.

Allow the user to change this setting

Lets users change the setting for the views during the task execution.

Latest Version Overwrite

SOLIDWORKS PDM lets users overwrite the latest file version during check in and change state operations. This reduces the number of archives significantly.

Previously, SOLIDWORKS PDM created a new file version for any change to a file or its metadata.

Typical use cases include:

- Day end check in.
- Fixing a typo.
- Updating child references of the referencing file to the latest version.
- Upgrading the latest file version to match the current SOLIDWORKS version.
- Updating metadata through the set variable transition action.
- Any minor changes retaining the revision tag on the latest version.

Administrators can allow users to overwrite the file version by granting the overwrite folder permission and state permission.

The version overwrite functionality is supported for documents or files. It also supports transitions that are triggered via Web2 client. It is not supported for:

- Items.
- Named BOMs.
- File versions that have associated items set to **Auto Update**.
- Files checked in via Web2 client.

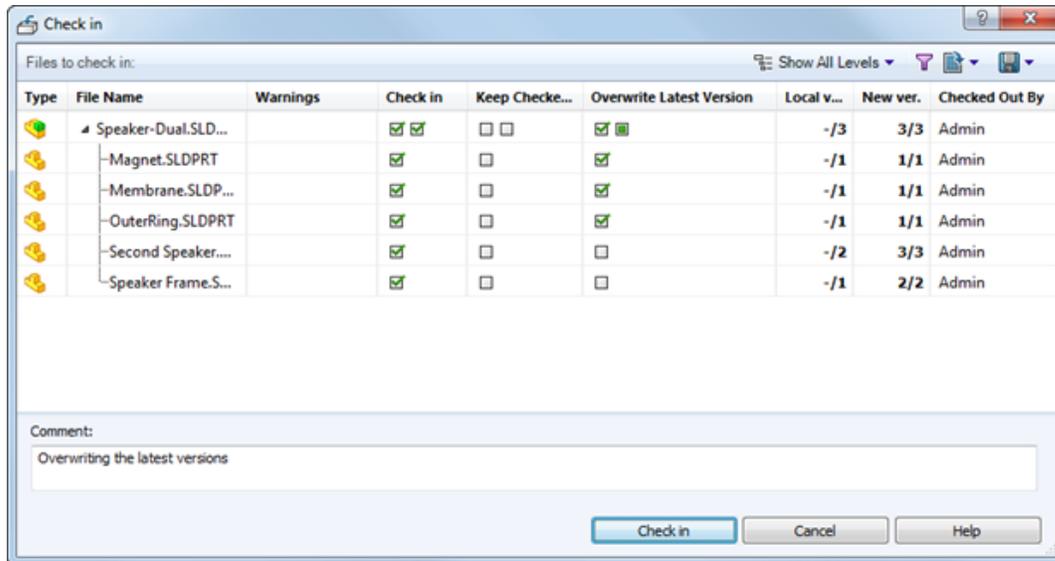
User Interface Changes with Version Overwrite

Many of the user interfaces have been changed with version overwrite functionality.

The Check in dialog box displays an **Overwrite Latest Version** column. This option is turned off by default. You can select it only if the **Check in** option is selected. To overwrite

all files, right-click in the dialog box and click **Overwrite Latest Version on All Files** or press **Ctrl+R**.

When you select **Overwrite Latest Version**, the **New ver.** column is updated with the latest version value.



The transition's Properties dialog box displays **Can overwrite latest version during check in** check box. Administrators can set the version overwrite action while running the transition.

The Check in file dialog box in Dispatch action displays an **Overwrite Latest Version** check box. The version overwrite is not supported in the Dispatch action if:

- The logged in user lacks overwrite folder permission or state permission.
- File versions have associated items set to **Auto Update**.

The History dialog box displays  icon followed by **Checked in with version overwrite** under **Event** for the latest version. For the overwritten version,  icon is updated to  icon indicating that the file archive associated with its history event is overwritten by the latest event.

When you have a cached file version and another user overwrites that file version on another machine the:

- **Version Number** column in File Explorer displays as  -/latest version. When you select this file, the icon is updated in the Version tab and the **Local version** status is updated to **Local version has been overwritten**.
- Child Quick Info icon in the SOLIDWORKS PDM add-in displays as . The **Version Number** column displays -/latest version, with the file name and version number in bold.

If a named BOM is associated with the overwritten version of the source file, the icon on the Bill of Materials tab appears as .

In the Administration tool, in the user's Properties dialog box, the Warnings page for **Check In** operation displays a warning, **Local version has been overwritten**.

Administrators can set this warning to stop check in of a parent file that has overwritten references.

Preview Enhancements

When previewing SOLIDWORKS files in Windows Explorer (Preview tab), SOLIDWORKS PDM offers more functionality and controls.

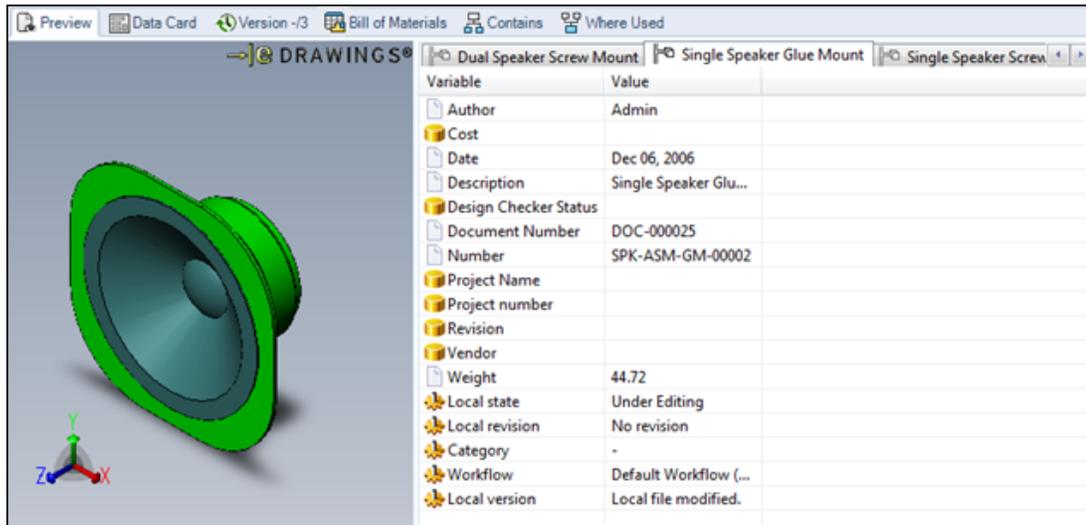
The viewer enhancements let you:

- Toggle the exploded view by clicking **Explode**.
- Turn off the card details display in the Preview tab of Windows Explorer, Item Explorer, and File Viewer. This lets you view the eDrawings preview in the entire preview area.

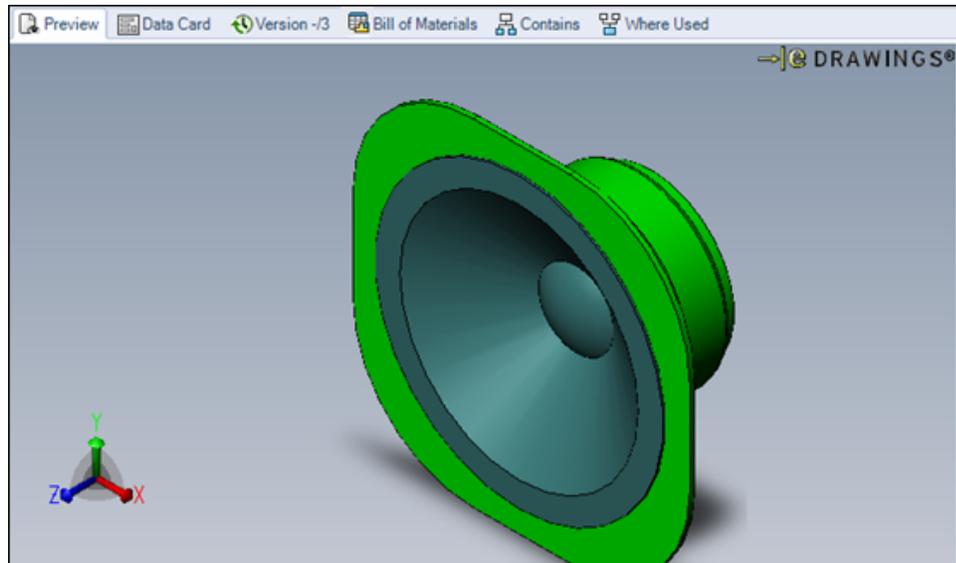
The **Card Preview** is turned on by default. To turn it off:

- In Windows Explorer, click **Display > Card Preview**.
- In File Viewer, click **View > Show File > Card Preview**.
- In Item Explorer, click **View > Card Preview**.

SOLIDWORKS PDM defaults to the Background color and Highlight color settings saved in the eDrawings full viewer.



Card Preview on



Card Preview off

Quality Enhancements in SOLIDWORKS PDM

You can move a file using the **Move/Rename file** command in the Dispatch action. Previously, you could only copy, delete, and rename the file.

The Convert task supports using the file data card variables in the output file name.

Input formulas for search card **Edit** boxes update when you enter new values in the connected fields in searches in Windows Explorer.

For example: In a Search Card, you add an edit box, connect it to the *Project* variable, and add an **Input formula** as *PRJ<Number>*. In searches in Windows Explorer, when you enter a value for the *Number* variable in the Search Card, the input formula triggers, and displays PRJ followed by the value of the Number variable.

The **Reset All** button on the Copy Tree and Move Tree dialog boxes resets all settings to the defaults. Previously, **Reset All** used to reset only the **Destination Folder Path** and **Target File Name** to the defaults.

You can turn off the user information pop-up in Windows Explorer when hovering over the user name in the **Checked Out By** column. To turn off the display, in the Administration tool, on the Explorer page of the User's Settings, clear **Show user information pop-up**.

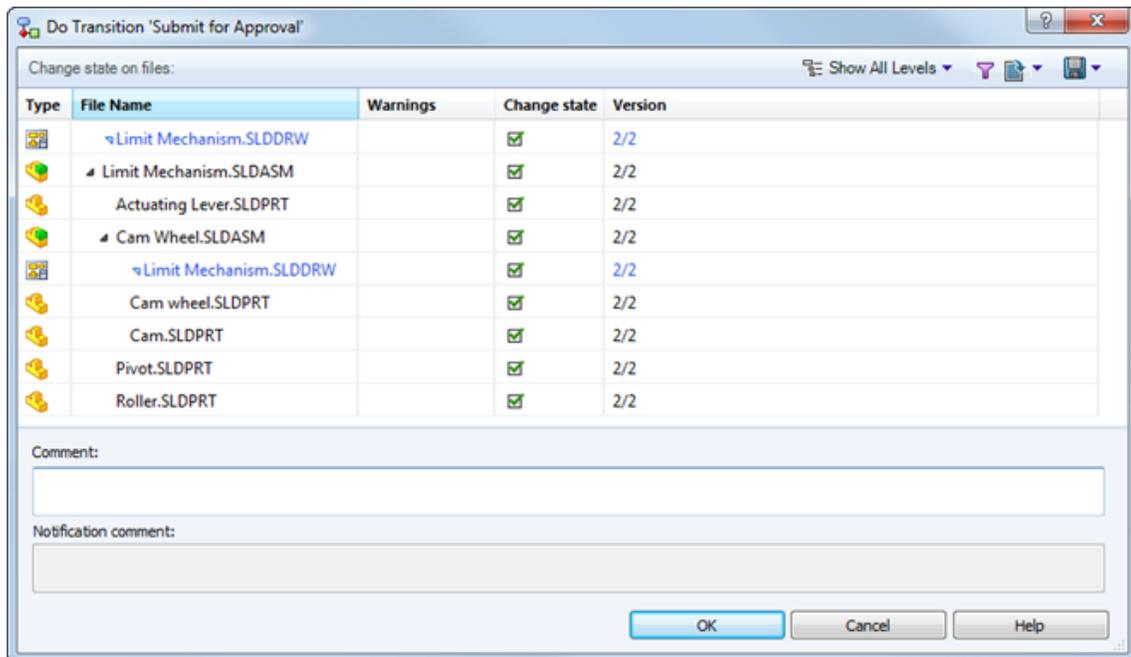
Reference Tree Display Enhancements

SOLIDWORKS PDM displays an enhanced reference tree with sub-parent relations.

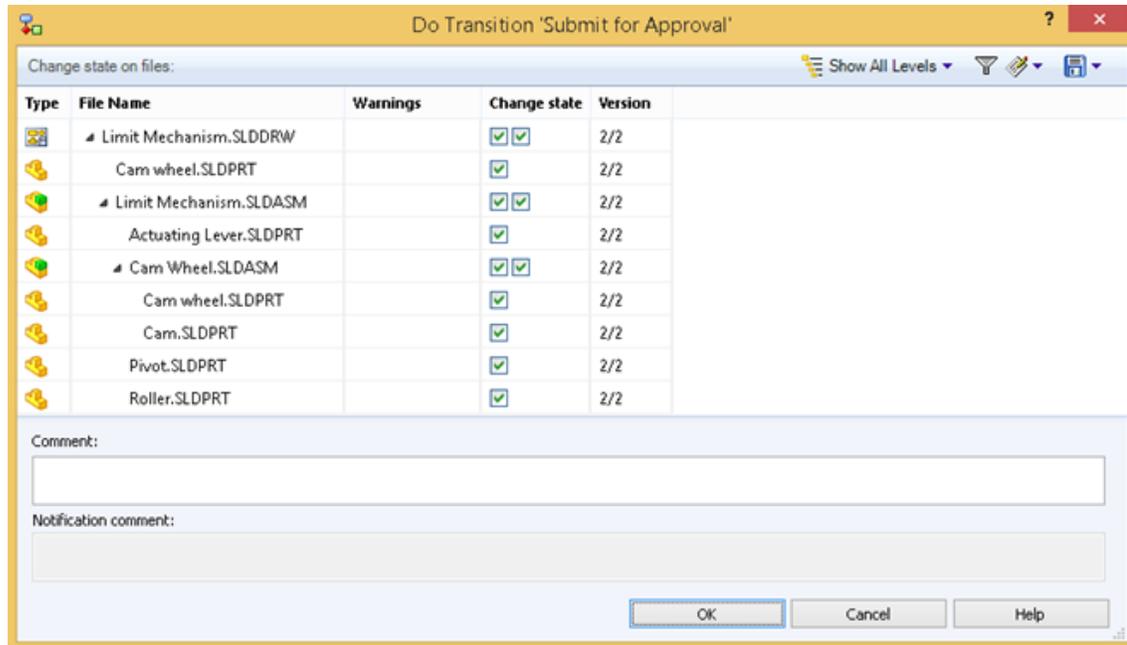
The Get, Set Revision, Do Transition (Change State), Copy Tree, and Move Tree dialog boxes, and the Contains tab display the reference tree with only one node for the selected file that is displayed at the top of the tree.

The sub-parent relation is displayed with the file name in blue and an upward arrow.

If the selected drawing file has multiple references, the sub-parent relation is displayed multiple times in the reference tree.



SOLIDWORKS PDM 2017



SOLIDWORKS PDM 2016

Rolling Back a File with References ★

SOLIDWORKS PDM enhances Rollback to handle reference redirection and rollback more efficiently.

To roll back a file with its references:

- Users must have the **Rollback** folder permission.
- The selected file and all its parent references must be in checked in state.
- The rollback target version must not be cold stored.

You can roll back a parent file with its child references and files indicated as sub-parents (such as drawing files) to a previous as built state.

When you roll back a child reference (such as a part file), you can redirect the parent references of the rolled back versions to the target version.

Previously, you could not roll back a file with all of its references and the parent references were removed.

After rollback, SOLIDWORKS PDM resets the referenced configurations of the files. If the same configuration is:

- Available in the target version, the references are retained for that configuration.
- Not available in the target version, the references are redirected to the active (last saved) configuration of the target version.

Reference redirection and roll back a file with references functionality is not supported for items.

Rolling Back a File Structure

To roll back a file structure:

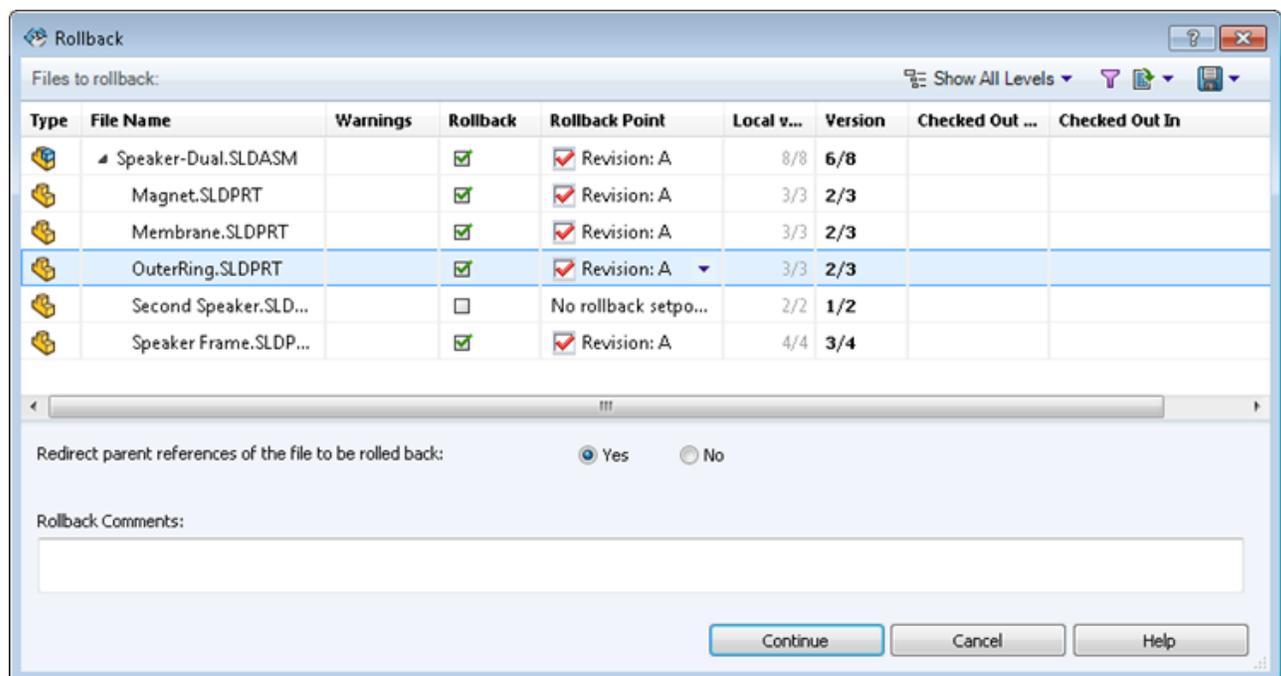
1. In the History dialog box, select the event to roll back to and click **Rollback**.
SOLIDWORKS PDM selects the parent file and the references that have same events as the referenced version to roll back.
In the Rollback dialog box, you can select other references to roll back and the **Rollback Point**.
SOLIDWORKS PDM selects **Yes** for **Redirect parent references of the file to be rolled back** to redirect the references.
2. For **Rollback Comments**, type a comment.
3. Click **Continue**.
The list of the referencing files for which the references are redirected or deleted appears.
4. Click **Rollback** to confirm the operation.

Rollback Dialog Box

The Rollback dialog box lets you roll back a file with or without references. You can either redirect the parent references or delete them.

To open the Rollback dialog box:

1. In the History dialog box, select the event to roll back to and click **Rollback**.



Settings

	Next Warning / Previous Warning	<p>Only shown when warnings are present in the Warnings column.</p> <p>In the file list, changes the focus to the next file with a warning or the previous file with a warning.</p>
	Show All Levels	<p>Displays file references for the entire file hierarchy (Show All Levels) or just the top level of the file hierarchy (Top Level Only).</p>
	Select Files	<p>Opens the Select Files dialog box, where you can specify a wildcard pattern to select files.</p>
	Open File List	<p>Open All opens a list of files in Microsoft Excel as a comma-delimited file.</p> <p>Open Visible opens a list of visible files in Excel as a comma-delimited file.</p>
	Save File List	<p>Export All exports the list of all files as a comma-delimited <code>.txt</code> file.</p> <p>Export Visible exports the list of visible files as a comma-delimited <code>.txt</code> file.</p>

Files to rollback

Type	Displays a thumbnail preview of the file when you hover over the file type icon.
File Name	Displays the file to roll back and its referenced files (if any).
Warnings	Displays warnings, if any of the rollback conditions are not met.
Rollback	Displays the selected parent file and its references that have the same Revision or Transition or Checked in event. You can select or clear the files to roll back.

Rollback Point

Click any Rollback Point to display the list of **Event** in the file history for the target version. The default selected **Rollback Point** is highlighted in blue. When you change this setting, the rows are highlighted in yellow.

The **Rollback Point** is set to:

Setting	Description
No rollback setpoint	<p>If the files are not selected for rollback.</p> <p>When you select a file for rollback, SOLIDWORKS PDM sets the Rollback Point to the highest valid event in the target version.</p>
Internal Component	<p>If a parent file has an internal component, it gets selected or cleared for rollback with its parent file. You cannot select or clear the internal component explicitly.</p>

SOLIDWORKS PDM attempts to match child files to Revisions, Transitions, or Checked in events from the rollback triggered parent file. Any change to the Rollback Point of the parent file recomputes the default rollback points for child files in the tree.

Local ver. Displays the local version and the latest version as local version/latest version of the file.

Version Displays the referenced version/latest version of the file.

The file is rolled back from the latest version to the referenced version.

Checked Out By Displays the user who checked out the file. This is blank if the file is checked in.

Checked Out In Displays the computer and local folder where the file is checked out, or blank if not checked out.

Found In Displays the vault folder containing the file.

State Displays the state of the file.

Redirect parent references of the file to be rolled back **Yes** Default. Redirects the parent references of the rolled back versions.

Redirect parent references of the file to be rolled back **No** Deletes the parent references of the rolled back versions.

Rollback Comments	Lets you type the rollback comment.
Continue	Lets you see the redirected or deleted referencing files of the rolled back version.

Parent Reference Handling

Depending on the selection of **Redirect parent references of the file to be rolled back** in the Rollback dialog box, the parent references of the rolled back child versions are redirected or deleted.

Open File List	Exports the details in Microsoft Excel as a comma-delimited file.
Save File List	Saves the details in a text file.
File to Rollback	Lists the file to roll back.
Parent File	Lists the parent reference file names of the file to be rolled back.
Referencing Version or Versions	Displays the affected versions of the parent file separated by semi-colons.
Relative File Path	Displays the file path to the root vault.
Rollback	Rolls back the selected files.

Searching Local Files

You can search local files that are added to the vault in offline mode.

To search local files in File Explorer, right-click inside the folder and click **Search Local Files**.

In the Administration tool on the Menus page, administrators can modify the **Search Local Files** menu option to appear to a user when they are viewing the files in Windows Explorer.

SOLIDWORKS PDM Client and Server Service Pack Compatibility

SOLIDWORKS PDM allows a service pack mismatch between client and server components within the same major version.

Previously, the client and server components had to have the same major version and service pack.

SOLIDWORKS PDF Task Add-in

SOLIDWORKS PDM Professional provides a task add-in that lets users convert Microsoft Office files and DWG/DXF format files to PDF files.

The SOLIDWORKS PDF Task Add-in provides the following tasks:

Task	Description
Office to PDF	Converts Microsoft Word, Excel, and PowerPoint files.
DraftSight to PDF	Converts DWG/DXF format files.

To convert files to PDF, users must have the following installed on the task host:

- Microsoft Office 2010 or later versions for Microsoft Office files.
- SOLIDWORKS DraftSight Professional/Premium for DWG/DXF format files.

For the DraftSight to PDF task to complete, you must disable **SOLIDWORKS PDM Add-in** in DraftSight on the dedicated task host.

Accessing the SOLIDWORKS PDF Task Add-in

To access the SOLIDWORKS PDF Task Add-in:

1. In the Administration tool, load the **SWPDFTaskAddIn**.
 - a) Right-click the vault and click **Import**.
 - b) In the dialog box, navigate to *drive letter:\Default Data*.
 - c) Select the `Draftsight_to_PDF_gb.cex` and `Office_To_PDF_gb.cex` files, and click **Open**.
2. Verify that execution of **SOLIDWORKS PDF Task Add-in** is permitted on the client machine.
3. Configure the tasks.

Configuring the Office to PDF Task

Administrators can configure the **Office to PDF** task to convert Microsoft Office files to PDF files.

To configure the Office to PDF task:

1. In the Administration tool, expand **Tasks** and double-click **Office to PDF**.
2. In the dialog box, for **Add-in**, select **SOLIDWORKS PDF Task Add-in**.
3. In the left pane, set options:

Option	Description
Execution Method	Selects the computers that can execute the task.
Menu Command	Specifies whether and how the command appears to users in the File Explorer.
File Card	Maps variables from the source file's data card to the output file's data card or adds Free Text to the output file's data card.
Output PDF Details	Specifies the name and location of the output PDF file.
Office Options	Defines settings for Microsoft Word, Excel, and PowerPoint files.
Permissions	Lets you select users and groups that are allowed to initiate the task.
Success Notification and Error Notification	Lets you select users and groups to notify.

Office to PDF Properties - File Card

Use the File Card page of the Office to PDF Properties dialog box to specify the mapping of variables from the source file data card to the output file data card.

To display the Office to PDF Properties - File Card page:

1. Expand the **Tasks** node and double-click **Office to PDF**.
2. In the left pane, click **File Card**.

Option	Description
Variable list	<p>Source variable</p> <p>Variable used in source file data card.</p> <p>Destination variable</p> <p>Variable in output file data card that maps to Source variable.</p>
Add variable	Activates the Selected variable section so you can specify variable mapping.
Remove variable	Removes the selected variable mapping.

Selected variable
Sets variable mapping.

Option	Description
Source variable	Variable used in source file data card.
Free text	Text to display in the destination variable if you select Free text as the source.
Destination variable	Variable in output file data card.

Office to PDF Properties - Output PDF Details

Use the Output PDF Details page of the Office to PDF Properties dialog box to set up basic properties for the PDF file such as the default path and vault log path.

To display the Office to PDF Properties - Output PDF Details page:

1. Expand the **Tasks** node and double-click **Office to PDF**.
2. In the left pane, click **Output PDF Details**.

Option	Description
Primary output path	<p>Default path</p> <p>Location where converted PDF files are saved.</p> <p>Build the output file path by typing static text and clicking  to select from the following dynamic variables:</p> <ul style="list-style-type: none"> • Source File Name • Source File Extension • Source Folder Path • Vault Root Folder Path • Source File Revision • Source File Version • Source File Workflow State • Today's Date • Variables <p>The example in the dialog box shows the currently selected file format and path.</p> <hr/> <p>Allow the user to change the output path</p> <p>Lets users change the output path during the task execution.</p>
Secondary output path	<p>Second location where converted PDF files are saved.</p> <p>Build the output file path by typing static text and clicking  to select from dynamic variables.</p> <p>The example in the dialog box shows the currently selected file format and path.</p>
Create reference from the destination file to the source file	<p>When selected, the destination file contains a reference to the source file.</p> <p>To see the reference, select the output file in File Explorer and view the Contains tab.</p>
Duplicate file name handling	<hr/> <p>Replace existing files</p> <p>Overwrites a file if a file of the same name already exists.</p> <hr/> <p>Create new version of existing files</p> <p>Creates a new version of the file if a file of the same name already exists.</p>

Option	Description
Logging settings	<p>Vault log path</p> <p>Click Browse to build the vault log path.</p>
Advanced Scripting Options	<p>Displays the Advanced Scripting Options dialog box, where you can:</p> <ul style="list-style-type: none"> • Modify the VB script that controls the task. • Change the user interface type used by the task.

Office to PDF Properties - Office Options

Use the Office Options page of the Office to PDF Properties dialog box to set options for Microsoft Word, Excel, and PowerPoint files.

To display the Office to PDF Properties - Office Options page:

1. Expand the **Tasks** node and double-click **Office to PDF**.
2. In the left pane, click **Office Options**.

Option	Description
Word	<p>Page Selection</p> <ul style="list-style-type: none"> • All Pages Selects all pages for the conversion. • Page(s) From Selects pages in the range.
	<p>Document Properties</p> <p>Includes the document properties.</p>
	<p>Document showing markup</p> <p>Includes the document showing markup.</p>
	<p>Allow user to change export settings</p> <p>Lets users change the export settings.</p>

Option	Description
Excel	<hr/> <p data-bbox="618 298 852 327">Sheet Selection</p> <ul data-bbox="618 346 1149 527" style="list-style-type: none"><li data-bbox="618 346 1149 426">• All Sheets/Entire Workbook Selects all sheets for the conversion.<li data-bbox="618 445 1149 527">• Sheet(s) From Selects sheets in the range. <hr/> <p data-bbox="618 592 938 621">Document Properties</p> <p data-bbox="618 640 1081 669">Includes the document properties.</p> <hr/> <p data-bbox="618 722 1154 751">Ignore Print Areas in sheets if exist</p> <p data-bbox="618 770 1382 800">Ignores print areas in sheets to convert the entire sheet.</p> <hr/> <p data-bbox="618 852 1170 882">Allow user to change export settings</p> <p data-bbox="618 900 1133 930">Lets users change the export settings.</p> <hr/>

Option	Description
PowerPoint	<hr/> <p>Slide Selection</p> <ul style="list-style-type: none"> • All Slides Selects all slides for the conversion. • Slide(s) From Selects slides in the range. • Allow user to change the slide settings Lets users change the slide settings. <hr/> <p>Document Properties Includes the document properties.</p> <hr/> <p>Frame Slides Includes the frame slides.</p> <hr/> <p>Include hidden slides Includes the hidden slides.</p> <hr/> <p>Allow user to change export settings Lets users change the export settings.</p> <hr/>

Configuring the DraftSight to PDF Task

Administrators can configure the **DraftSight to PDF** task to convert DWG/DXF format files to PDF files.

To configure the DraftSight to PDF task:

1. Expand **Tasks** and double-click **DraftSight to PDF**.
2. In the dialog box, for **Add-in**, select **SOLIDWORKS PDF Task Add-in**.

- In the left pane, set options:

Option	Description
Execution Method	Selects the computers that can execute the task.
Menu Command	Specifies whether and how the command appears to users in the File Explorer.
File Card	Maps variables from the source file's data card to the output file's data card or adds Free Text to the output file's data card.
Output PDF Details	Specifies the name and location of the output PDF file.
Conversion Settings	Defines the settings for DWG/DXF format files.
Permissions	Lets you select users and groups that are allowed to initiate the task.
Success Notification and Error Notification	Lets you select users and groups to notify.

DraftSight to PDF Properties - File Card

Use the File Card page of the DraftSight to PDF Properties dialog box to specify the mapping of variables from the source file data card to the output file data card.

To display the DraftSight to PDF Properties - File Card page:

- Expand the **Tasks** node and double-click **DraftSight To PDF**.
- In the left pane, click **File Card**.

Option	Description
Variable list	<p>Source variable Variable used in source file data card.</p> <p>Destination variable Variable in output file data card that maps to Source variable.</p>
Add variable	Activates the Selected variable section so you can specify variable mapping.
Remove variable	Removes the selected variable mapping.

Selected variable

Sets variable mapping.

Option	Description
Source variable	Variable used in source file data card.
Free text	Text to display in the destination variable if you select Free text as the source.
Destination variable	Variable in output file data card.

DraftSight to PDF Properties - Output PDF Details

Use the Output PDF Details page of the DraftSight to PDF Properties dialog box to set up properties for the PDF file such as the default path and vault log path.

To display the DraftSight to PDF Properties - Output PDF Details page:

1. Expand the **Tasks** node and double-click **DraftSight to PDF**.
2. In the left pane, click **Output PDF Details**.

Option	Description
Primary output path	<p>Default path</p> <p>Location where converted PDF files are saved.</p> <p>Build the output file path by typing static text and clicking  to select from the following dynamic variables:</p> <ul style="list-style-type: none"> • Source File Name • Source File Extension • Source Folder Path • Vault Root Folder Path • Source File Revision • Source File Version • Source File Workflow State • Today's Date • Variables <p>The example in the dialog box shows the currently selected file format and path.</p> <hr/> <p>Allow the user to change the output path</p> <p>Lets users change the output path during the task execution.</p>

Option	Description
Secondary output path	<p>Second location where converted PDF files are saved.</p> <p>Build the output file path by typing static text and clicking  to select from dynamic variables.</p> <p>The example in the dialog box shows the currently selected file format and path.</p>
Create reference from the destination file to the source file	<p>When selected, the destination file contains a reference to the source file.</p> <p>To see the reference, select the output file in File Explorer and view the Contains tab.</p>
Duplicate file name handling	<p>Replace existing files</p> <p>Overwrites a file if a file of the same name already exists.</p> <p>Create new version of existing files</p> <p>Creates a new version of the file if a file of the same name already exists.</p>
Logging settings	<p>Vault log path</p> <p>Click Browse to build the vault log path.</p>
Advanced Scripting Options	<p>Displays the Advanced Scripting Options dialog box, where you can:</p> <ul style="list-style-type: none"> • Modify the VB script that controls the task. • Change the user interface type used by the task.

DraftSight to PDF Properties - Conversion Settings

Use the Conversion Settings page of the DraftSight to PDF Properties dialog box to set options for DWG/DXF files.

To display the DraftSight to PDF Properties - Conversion Settings page:

1. Expand the **Tasks** node and double-click **DraftSight to PDF**.
2. In the left pane, click **Conversion Settings**.

Option	Description
Override current DraftSight settings	Overrides the current DraftSight settings and lets users change settings.
Sheets	<p>All Sheets Selects all sheets for the conversion.</p> <p>Named Sheets Selects named sheets.</p> <p>Allow user to change settings Lets users change the sheet settings.</p>
Paper size	<p>Standard Lets you select the sheet size from the list.</p> <p>Custom Lets you set the Width, Height, and Units.</p>
Document Properties	Includes the document properties.
Source File references	
Use the referenced version of the referenced files (as built)	Uses the referenced version of the referenced files when the task is triggered.
Use the latest version of the referenced files	Uses the latest version of the referenced files when the task is triggered.

SOLIDWORKS PDM Web2 Viewer Enhancements

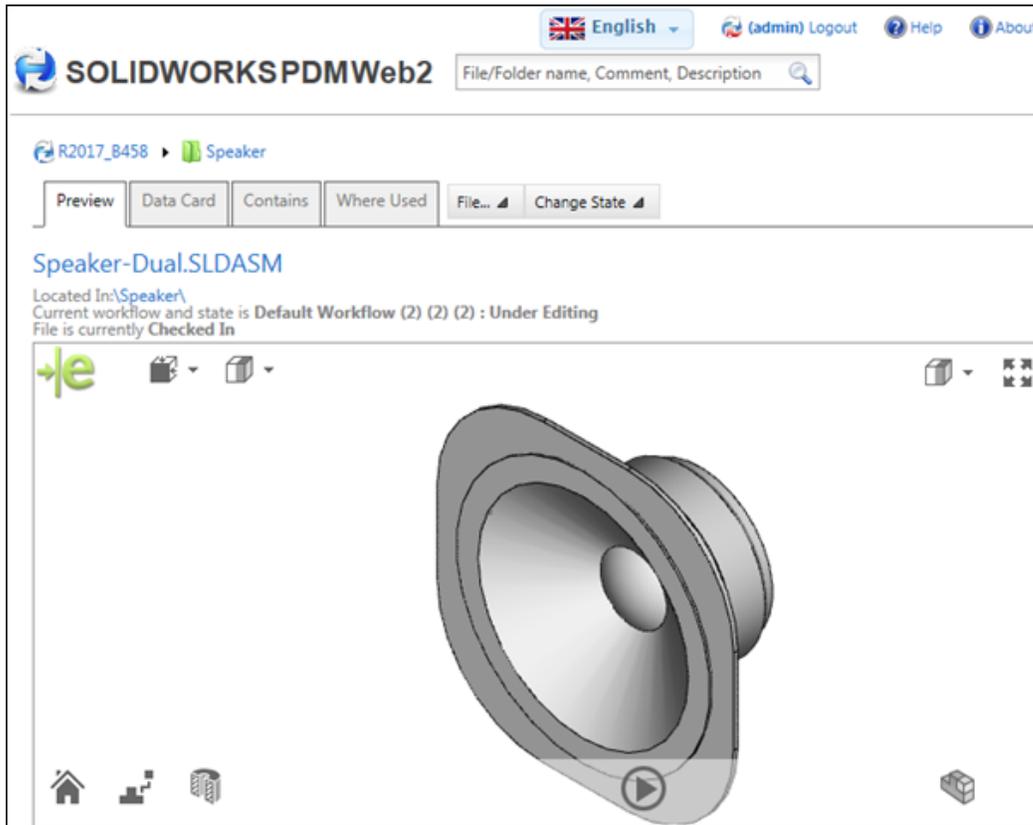
SOLIDWORKS PDM Professional enhances the viewing technology in the Web2 client for SOLIDWORKS files and other common file types.

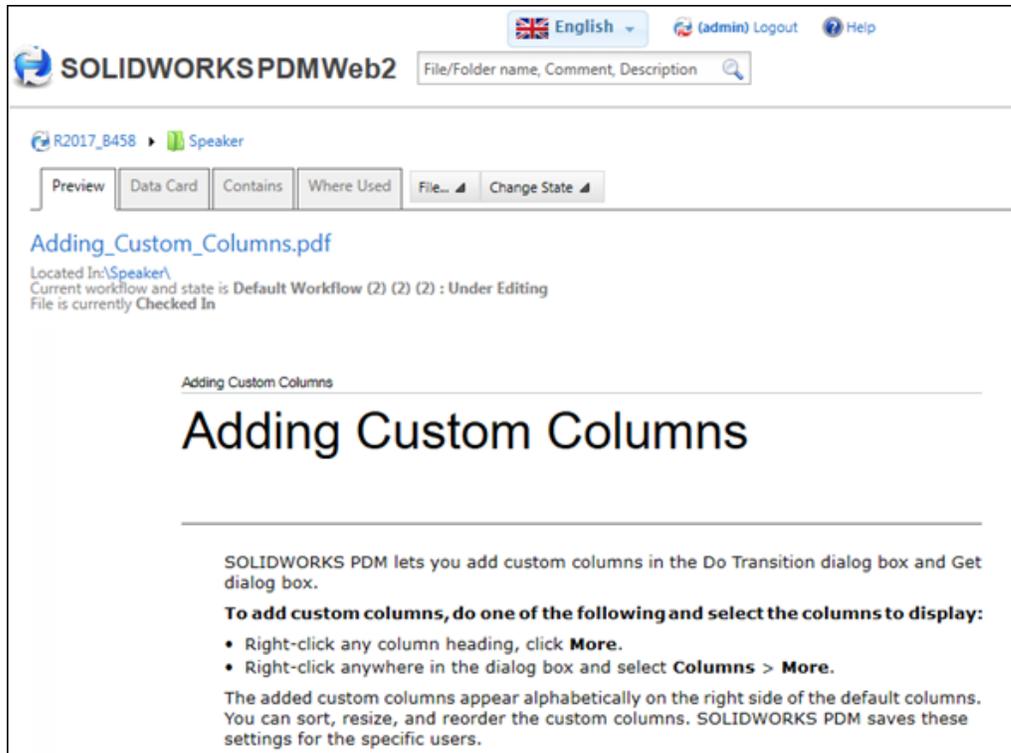
- SOLIDWORKS files. You can dynamically preview files in supported browsers such as Microsoft Internet Explorer, Google® Chrome™, Mozilla® Firefox®, and Apple® Safari® without loading plug-ins.

Previously, you could dynamically preview files in Microsoft Internet Explorer with an eDrawings ActiveX® plug-in only.

- Other common file types. The preview quality is more like viewing the file in its native application.

You can no longer convert various file formats to PDF in the Web2 client.





Vault Database Replication ★

SOLIDWORKS PDM Professional supports replicating a vault database that can improve performance in high latency (typically more than 50 milliseconds) and high usage environments.

Administrators can use Always On Availability Groups (AOAG) technology from Microsoft SQL Enterprise edition 2012 or higher. For details, see <https://msdn.microsoft.com/en-us/library/ff878265.aspx>.

For each availability group, AOAG maintains a single read-write primary database and multiple read-only secondary databases. The primary SQL server updates the secondary SQL servers periodically.

SOLIDWORKS PDM Web2 client and Web client cannot take advantage of the secondary database.

You can create the secondary SQL servers over LAN and WAN to the primary SQL server. Using AOAG, SOLIDWORKS PDM Professional can direct the read operations to the secondary SQL server. This improves the performance in the following ways:

- Read operations are directed to the secondary SQL server, significantly reducing the load on the primary SQL server so it can respond faster, and handle more load.
- For read operations, users access the secondary SQL database server instead of accessing the primary SQL database. This speeds up read operations such as browsing through vault folders, and accessing Data Cards, Contains, and Where Used information.
- In case of high latency between the client and primary SQL Server, having a secondary SQL server locally to the client improves the performance for read operations.

It is recommended to use synchronous-commit mode on the secondary SQL servers that have very little latency to the primary SQL server.

All write operations are directed to the primary SQL server. During login, the user is connected to the:

- Primary and secondary SQL servers if the secondary SQL server is reachable.
- Primary SQL server if the secondary SQL server is not reachable. In this case, the primary SQL server handles all database operations for that session.

Configuring the Secondary SQL Servers

Use the Server Configuration dialog box to configure the secondary SQL servers.

To configure the secondary SQL servers:

1. In the Administration tool, expand **Replication** and double-click **Database**.
2. In the Database dialog box, click **Add Server**.
3. In the Server Configuration dialog box, in **Server Name**, select the secondary SQL server name.
4. For **Max. Lag from Primary**, enter a value.
5. Click **Add Member** to assign users and groups.
6. Click **Test Settings** and **OK**.

Server Configuration Dialog Box

The Server Configuration dialog box lets you configure the secondary SQL server.

Option	Description
Server Name	Lists secondary SQL server. Select from the list. This name must be unique. Use the IP address if SOLIDWORKS PDM is configured to communicate using IP address.
Max. Lag from Primary	The maximum permissible lag between the primary and the secondary SQL servers. This value can vary from 1 minute to 60 minutes. If the actual time lag exceeds the maximum value, all read and write operations are directed to the primary SQL server only. Decimals, zero, and negative values are not supported.

Option	Description
Add Member	<p>Assigns users and groups to the secondary SQL server.</p> <p>If you assign a user to a database through a group and the user is explicitly assigned to another database, SOLIDWORKS PDM directs the user to the database to which the user is explicitly assigned.</p> <p>If you assign a user to multiple secondary SQL servers through group assignments or explicitly, SOLIDWORKS PDM directs the user to the database to which the user is added most recently.</p>
Remove Member	Removes the selected users and groups.
Test Settings	<p>Verifies the following:</p> <ul style="list-style-type: none">• The secondary SQL server name is unique.• The vault database on the primary SQL server is replicated on the secondary SQL servers.• The username (sa) and the password for the primary and secondary SQL servers are the same.

20

SOLIDWORKS Plastics

This chapter includes the following topics:

- **Automatic Valve Gates**
- **Configuration Improvements for PlasticsManager**
- **Customize Min and Max Values of Plots**
- **Faster Loading of Parts**
- **High Accuracy Cooling**
- **Plot of Part Mass Versus Time**
- **Solid Mesh Edit**
- **User Interface Improvements**

SOLIDWORKS Plastics Standard, SOLIDWORKS Plastics Professional, and SOLIDWORKS Plastics Premium are available as separately purchased products that can be used with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium.

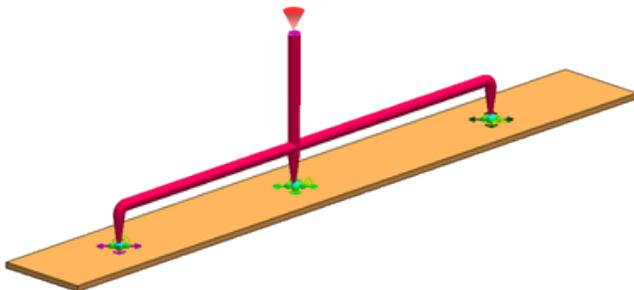
Automatic Valve Gates

An option automatically defines control valves so you can predict the control valve opening times based on the melt front arrival.

Valve gates are used with hot runner systems to sequentially inject plastic at several locations in the cavity. Plastics simulation can help determine the timing sequence for opening each valve. Two options make these simulations easier.

In the Injection Location PropertyManager, click **Automatically Add Valves** to let the program define control valves at the common interfaces between runners and cavities. You no longer need to select mesh elements manually, when you activate this option.

By selecting **Automatic** under **Valve Open Range**, the control valve opens automatically when the melt front passes the gate. You can then view the predicted opening times of each control valve in the log file.



Configuration Improvements for PlasticsManager

The associativity of configurations between the SOLIDWORKS software and SOLIDWORKS Plastics is improved.

Configuration improvements include:

- Any updates you make in the ConfigurationManager, for example adding a new configuration or switching between configurations, propagate to the PlasticsManager tree.
- SOLIDWORKS Plastics supports derived configurations.
- **Duplicate Study** copies the plastics study settings to a new configuration.

Customize Min and Max Values of Plots

Improvements in plots include editing the Min and Max values outside the range of the current data, and resetting the Min and Max back to the default values.

When comparing result plots from different simulations, you can set an appropriate plot range (which could be outside the min and max values of the individual plot) to facilitate the comparison and interpretation of results.

The default minimum and maximum limits are shown above the plot legend.

In the Results PropertyManager, click  to reset the min and max values to their default values.

Faster Loading of Parts

An option lets you control the loading of plastics analysis data when you open part documents.

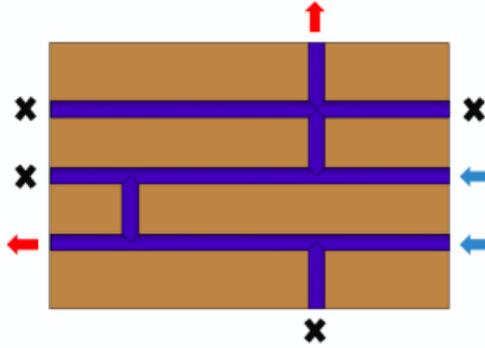
In the Settings PropertyManager, **Load plastics data when .sldprt is opened** is cleared by default. Saved plastics simulation data is loaded with the activation of the PlasticsManager tree.

Selecting **Load plastics data when .sldprt is opened** results in longer loading time when opening `sldprt` files with saved plastic simulation data.

High Accuracy Cooling

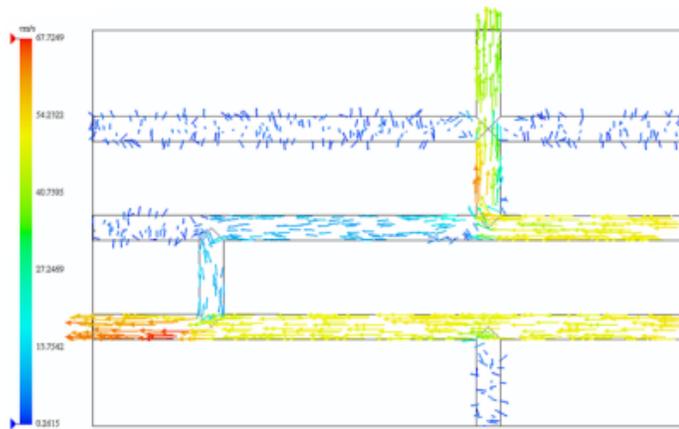
The Cool analysis module is enhanced.

A solution option is available for the Cool analysis module which allows simulation of cooling circuits with multiple inlets and outlets. All side passages without inlets or outlets are considered dead ends. Select **Coolant Flow Field** under **Boundary Conditions**.

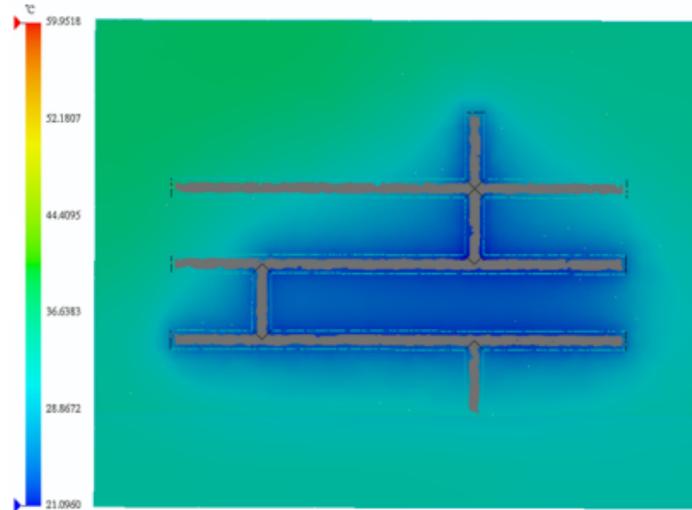


Red arrow: Inlet; Blue arrow: Outlet; X: Dead end

After completion of the **Coolant Flow Field** analysis, new result plot options are available for coolant velocity and coolant pressure.



Velocity of coolant system



Mold Temperature at end of cooling

The mold open time is included in the Cool solver calculations. This accounts for the effect of coolant circulating while the mold is open. The mold open time is listed under the **Cycle Time** summary in the report.

Plot of Part Mass Versus Time

The **Part Mass** plot shows the part mass changes during the filling and packing processes.

The **Part Mass** plot includes only the cavity domains. Runner domains are excluded. The part mass plot helps you determine when the cavity is fully packed so you can shorten packing time to eliminate unnecessary additional packing of the sprue and runners.

You can also use the **Part Mass** plot along with the gate temperature plot to determine the gate freeze and pressure holding time.

Automatic Packing Time The first stage of packing will automatically end if the polymer flow rate becomes nearly zero, and any remaining **Pressure Holding Time** will be added to the **Pure Cooling Time** to determine the new duration of the second stage of packing. Polymer flow rate is monitored at the interface between the runner and cavity domains, or at the injection locations if no runner domain has been defined. The criteria for nearly zero flow rate is satisfied when the rate of change of part mass falls below 0.01% per second.

Settings

Pressure Holding Time	8 sec
-----------------------	-------

Results

Cycle Time (= Filling Time + Total time in Pack Stage)	16.64 sec
Filling Time	0.62 sec
Total time in Pack Stage (= Pressure Holding Time + Pure Cooling Time)	16.02 sec
Pressure Holding Time	4.52 sec
Pure Cooling Time	11.50 sec

Solid Mesh Edit

An option in the Solid Mesh - Manual PropertyManager maintains the previously defined domain definitions when you create a solid mesh with the **Manual** method.

In the Solid Mesh - Manual PropertyManager, select **Use Previous Design**.

The program maintains the previous definitions for Cooling Channels, Runners, and Virtual Molds, and proceeds to the step of surface mesh creation. This option saves you time as you can modify the mesh for the cavity only, without redefining the domain assignments in a model.

User Interface Improvements

Several improvements in the user interface enhance productivity.

- Icons next to the parameters' numeric fields are active, and restore the parameters' default input values.
- The Polymer dialog box retains the custom size and location settings of the last display.
- Long names of polymer materials are fully displayed in the Polymer dialog box.
- Animation speed control is improved. The speed range is increased by a factor of 2.
- In the Runner Design Wizard PropertyManager, click **Use draft angle** to define a sprue geometry with a given diameter dimension (D1) at the beginning of the sprue and a draft angle to automatically compute the diameter at the end of the sprue geometry. To reverse the dimensions for a selected sprue, click **Flip Dimension**.
- A more intuitive **Weld Line** contour plot reflects the severity of the plastic melt fronts intersection. On the **Weld Lines** contour plot, the locations on the cavity where the plastic melt fronts meet at small (acute) angles are colored red. The locations where the angles are large (obtuse) are colored blue.

21

Routing

This chapter includes the following topics:

- **Auto Route Enhancements**
- **Automatic Zoom to Fit Option on Adding Route Components**
- **Exploded View Tool**
- **Flatten Enhancements**
- **Pipe Nipples**
- **Quality Enhancements and Fixes for Routing**
- **Reusing Routes for Tubes and Hoses**
- **Securing Distinct Routes Through Clips**

Available in SOLIDWORKS Premium.

Auto Route Enhancements

For orthogonal and flexible route types, **Auto Route** failures are automatically corrected by repositioning the route.

Previously, you received an error message while using **Auto Route**, when the endpoints were very close or the radius was too small for the geometry.

The **Alternate paths** solutions are listed in the Auto Route PropertyManager. A maximum of seven solutions are provided at a time. You can access the alternate path solutions by clicking in the **Alternate paths** box and then clicking in the graphics area to view the manipulator. In the manipulator, you can right-click to change to the next possible solution and left-click to accept the current solution offered.

Automatic Zoom to Fit Option on Adding Route Components

You can set the behavior of the automatic **Zoom to Fit** tool after adding route components in Electrical, Piping, and Flexible Tubing.

Previously, the **Zoom to Fit**  tool was automatically enabled while adding route components in the graphics area.

To set this option, click **Tools > Options > Routing** and select **Automatically zoom to fit after addition of route components**.

Exploded View Tool

You can create exploded views of Routing assemblies similar to SOLIDWORKS assemblies using the **Exploded View** tool.

Previously, while using **Exploded View** in Routing, the fittings returned to their original positions, changing the position of other pipe segments in the assembly.

The enhanced functionality covers the following:

- You can create exploded views at the subassembly level.
- You can create exploded views with the following types:
 - **Regular step**
 - **Radial step**
- You no longer receive an error message when you try to select a route part and drag it while using **Exploded View** in Electrical, Piping, and Flexible Tubing.
- You can drag or select route parts in subassemblies or components similar to the way that you can drag them in the top level assembly.
- You cannot use the **Exploded View** tool with flattened routes.

Flatten Enhancements

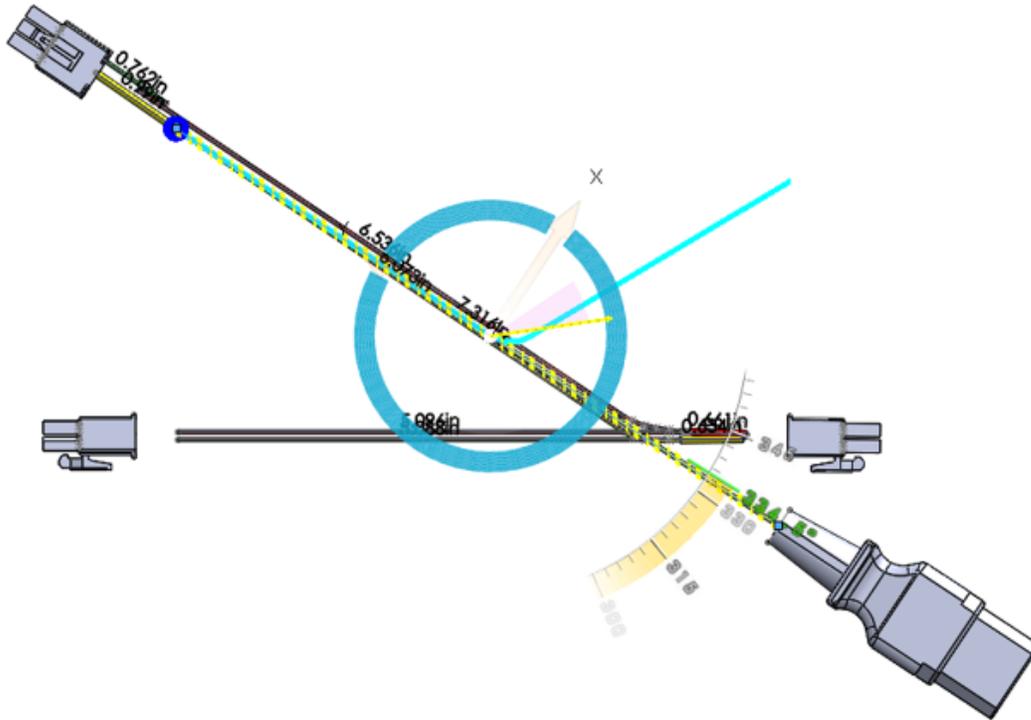
Enhancements in Edit Flattened Route

You can edit complex manufacturing-style flattened routes with the enhanced edit tools in the **Edit Flattened Route** tool.

You can also access **Edit Flattened Route** from the shortcut menu of the flattened configuration in the FeatureManager design tree.

The following aspects are enhanced:

- Temporary graphics.
- Manipulator to change bend angle and bend radius.
- Fixed point selection to determine which side of the route segment is to be edited.



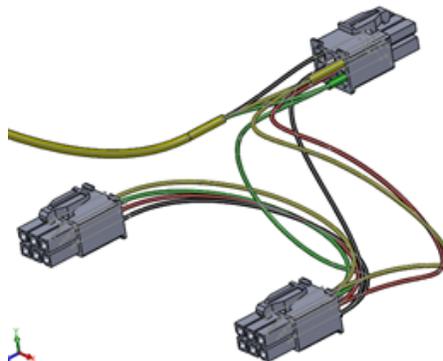
Flattened Disjointed Routes

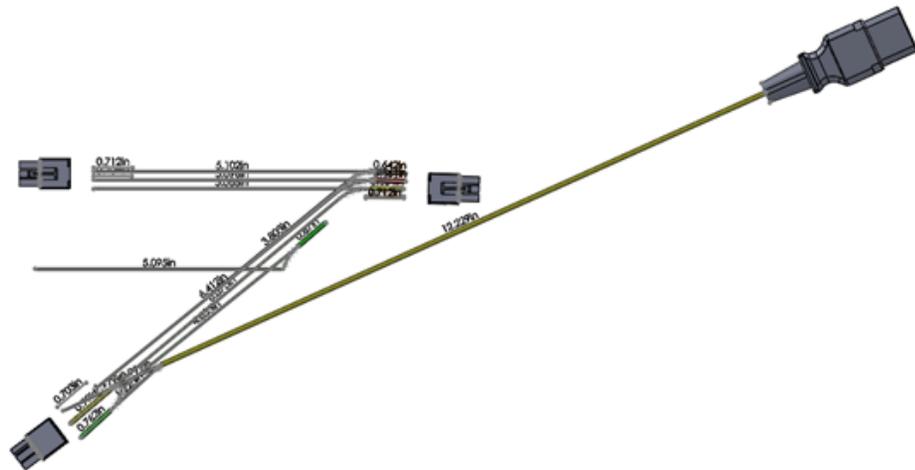
The electrical flattening functionality allows disjointed routes in Manufacturing and Annotation styles.

Previously, you received an error message while trying to flatten disjointed routes.

Display of disjointed routes is supported in flattened drawings.

A route is a disjointed route segment when a route assembly having multiple individual routes that are not connected to each other, form a circuit-like system.



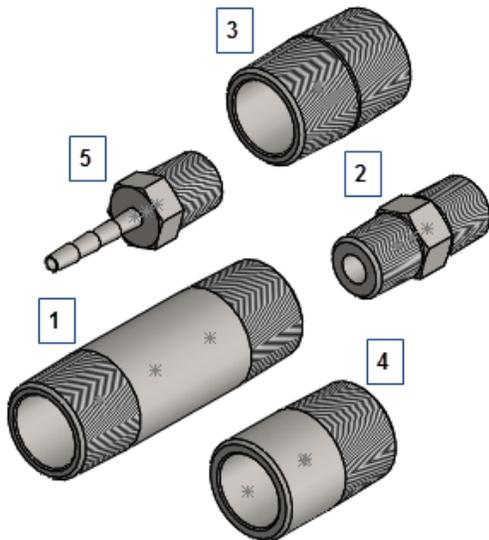


Pipe Nipples

You can use Nipples for connecting two fittings in a route assembly.

A Pipe nipple is a short piece of pipe with a male pipe thread at both ends or either end. Support for five types of pipe nipples is provided in the Routing Library Manager.

Five new types of Nipples are provided in the Design Library.



1. Barrel nipple
2. Hex nipple
3. Close nipple
4. Weld nipple
5. Hose nipple

Quality Enhancements and Fixes for Routing

General Routing

- You can override the automatic creation of stubs when you **Alt** + drag-drop connectors or fittings in the graphics area. You can:
 - Insert parts without automatically starting a route.
 - Mate or position the part before starting a route.
- The **Sketcher Triad** tool is available in Piping and Flexible Tubing. Previously, it was only available in Electrical routes.
- Multiple instances of the same component are numerically referenced in the Wire From-to List. For example, when a plug is in the top assembly and also in a subassembly, both instances are uniquely referenced in the Wire From-to List.

Piping

You are notified with an error message that points out the junction, if a penetration is missing at the junction of pipes.

Reusing Routes for Tubes and Hoses

You can reuse an existing tube route in the same assembly or in other assemblies. The reused route uses the same part numbers, components, and lengths of hoses as the original route, but different sketch paths as per the design intent.

To reuse routes for tubes:

1. In the FeatureManager design tree, right-click a flexible tube route assembly and click **Reuse Route**  or click **Tools > Routing > Flexible Tubing > Reuse Route**.

2. In the PropertyManager:
 - a) Select **Search all subassemblies**.
 - b) Click **Browse** to search for an existing flexible tube route assembly to open the route document.
 - c) Select **Maintain route segment length**.

The lengths of each route segment are assigned to each route segment of the reused tube route assembly and the fixed length parameter is applied. All configurations, if more than one exists, are applied to the reused route.

The reused tube route assembly inherits properties from the parent tube route assembly, including custom properties and part number from configuration properties. If the part number is blank in the parent, then the reused tube route assembly uses the assembly name from the parent assembly.

If a reused route is in the same assembly as the original route, they are grouped together in the BOM as if they are two instances of the same assembly.

Edits do not propagate from the reused route to the parent tube route assembly. The reused route is a copy. To preserve the design intent and keep the reused route from deviating from the parent, several features are disabled. For example, you cannot add fittings or assembly features to a reused route.

- d) Click .
3. To change the design of the reused route from the parent, you can remove the link between the reused and the parent routes by right-clicking the reused route in the FeatureManager design tree and clicking **Delink Route**.
All Routing features are restored, the icon is removed, and the route is not grouped with its parent in a BOM.

To edit the route paths of the routes, right-click the route and click **Edit Route**.

Securing Distinct Routes Through Clips

When you pass multiple routes through a clip, the individual wires are not bound together and are offset from the clip axis.

To secure distinct routes through clips:

1. Open *drive Letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\Routing\ClipRoutes\ClipRoutes.sldasm.
2. Right-click the clip and select the configuration **4-01 - 6.4mm Dia** from the top section of the shortcut menu.
3. Click .
4. In the FeatureManager design tree, right-click **Route1** and click **Edit Route**.
5. Click **Tools > Routing > Routing Tools > Route/Edit through Clip** .

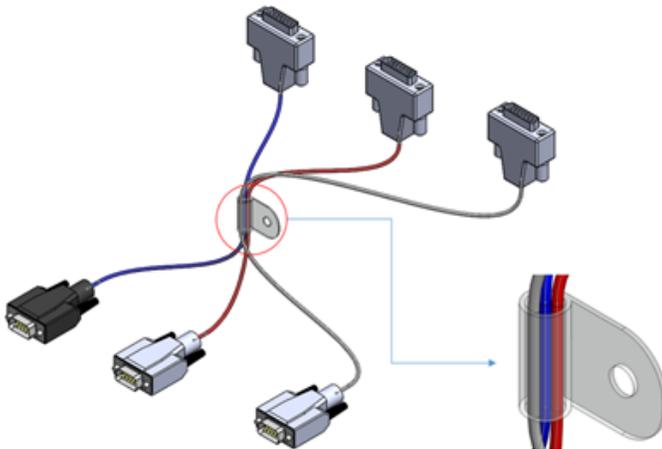
6. Click any of the route segments and the Clip Axis.
The selected segment and Clip Axis are displayed in **Current Selections**, in the PropertyManager.
7. In the PropertyManager:
 - a) Select **Offset from Clip Axis**.
 - b) Set **Offset Distance** to 10.
8. Click ✓.
9. Repeat steps 6 through 7 for the second route.
10. In the PropertyManager:
 - a) Select **Offset from Clip Axis**.
 - b) Set **Offset Distance** to 5.
11. Click ✓.
12. Repeat steps 6 through 7 for the third route.
13. In the PropertyManager:
 - a) Select **Offset from Clip Axis**.
 - b) Set **Offset Distance** to 2.

14. Click Auto Align .

All the routes are placed around the third route.

Auto Align bundles all routes close to the last modified route.

15. Repeat step 6 and click the Clip Axis.
Three routes appear in the PropertyManager, under **Routes passing through clip**.



22

Sheet Metal

This chapter includes the following topics:

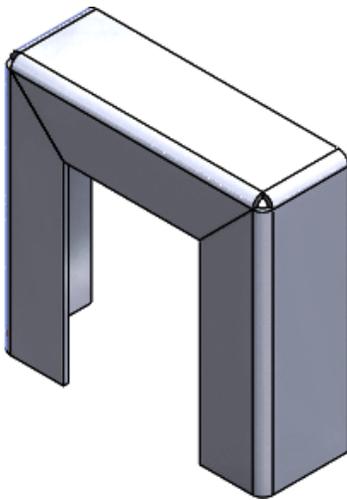
- **Creating Three Bend Corner Reliefs**
- **Normal Cuts**
- **Punch Table Support for Mirrored and Derived Parts**
- **Sheet Metal Options**

Creating Three Bend Corner Reliefs

You can create corner reliefs where three bends meet at one common point. The corner relief is applied at the common intersection point of the bend lines.

To create three bend corner reliefs:

1. Open *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\sheet metal\bracket.sldprt.



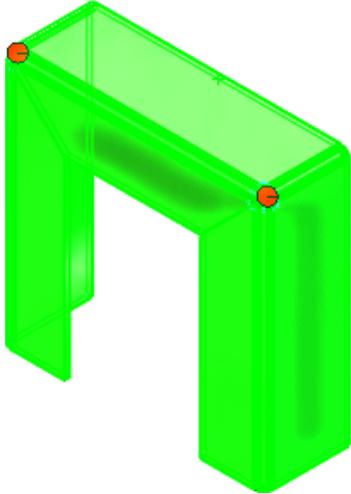
2. Click **Corner Relief**  (Sheet Metal toolbar) or **Insert** > **Sheet Metal** > **Corner Relief**.

3. In the PropertyManager:

a) Under **Corner Type**, select **3 Bend Corner**.

To create a three bend corner, the model must have three bends where the bend lines meet exactly at one point.

b) Under **Corners**, click **Collect all corners**.

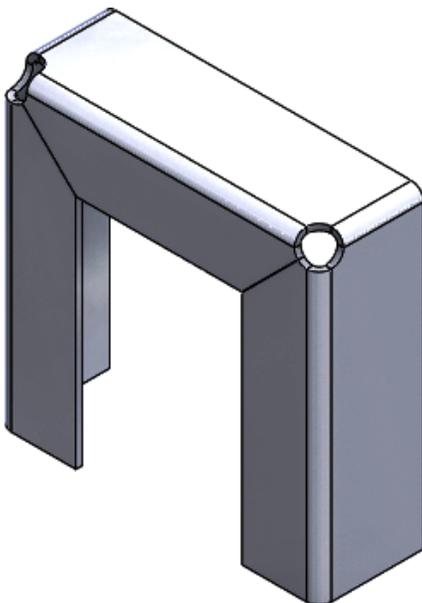


c) Under **Relief Options**, click **Circular**  and set **Diameter** to 4.

Three bend corner reliefs can have rectangular, circular, tear, or full round reliefs.

d) Click .

Reliefs are added to the corners.

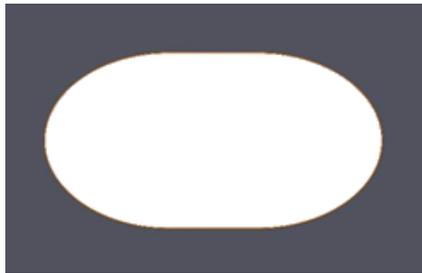


Normal Cuts

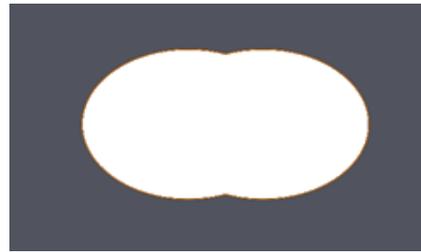
The **Normal cut** option in the Cut-Extrude PropertyManager is improved and delivers better results.

For sheet metal parts with extruded cut features created prior to SOLIDWORKS 2017, select **Normal cut** and **Optimized geometry** in the Cut-Extrude PropertyManager to take advantage of the enhanced functionality.

The geometry now produces smoother results. The following example shows an extruded cut based on a circular sketch.



SOLIDWORKS 2017



SOLIDWORKS 2016

Punch Table Support for Mirrored and Derived Parts

For drawings of mirrored and derived sheet metal parts, punch information is supported in punch tables.

For example, if you create a drawing view of a mirrored sheet metal part, any forming tools or library features in the mirrored part are available in punch tables.

When creating a derived part, you must select **Sheet metal information** in the Insert Part PropertyManager.

See [Retrieval of Sheet Metal Components on Drawings](#) on page 91.

Sheet Metal Options

You can define the default behavior that controls whether sheet metal bodies follow the parameters defined in the **Sheet Metal** folder in the FeatureManager design tree.

The **Sheet Metal**  folder in the FeatureManager design tree contains sheet metal definitions for individual sheet metal bodies. In this folder, you can define common sheet metal parameters for all bodies or have individual settings for each body. The following options let you control the default behavior for newly created sheet metal bodies - whether sheet metal definitions follow the **Sheet Metal** folder settings or not.

You can save these options in a template because they are document specific.

In **Tools > Options > Document Properties > Sheet Metal**, under **New Sheet Metal Bodies**, select:

- **Override default parameters**
- **Override bend allowance parameters**
- **Override auto relief parameters**

When selected, these options override the **Sheet Metal** folder parameters. In the individual sheet metal body PropertyManagers, the corresponding **Override default parameters** check boxes are selected. When all options are cleared, all individual body sheet metal parameters are driven by the settings in the **Sheet Metal** folder.

23

SOLIDWORKS Simulation

This chapter includes the following topics:

- **Automatic Update of Beam Joints**
- **Converting a Static Study to a New Study**
- **Detecting Stress Hot Spots**
- **Displaying SOLIDWORKS Simulation Results in the Graphics Area**
- **Editing Multiple Contact Sets**
- **Improved Arc Length Control Method for Nonlinear Contact Analysis**
- **Intelligent Solver Switch**
- **Managing Dynamic Analysis Results**
- **Macro Recording in Simulation**
- **Mass Properties in Simulation**
- **Offloaded Simulation**
- **Performance Improvements for SOLIDWORKS Simulation**
- **Post Processing Improvements for SOLIDWORKS Simulation**
- **Probing Results at Selected Nodes**
- **Remote Loads and Mass for Beams**
- **Self-dismissed Solver Messages**

SOLIDWORKS Simulation Standard, SOLIDWORKS Simulation Professional, and SOLIDWORKS Simulation Premium are available as separately purchased products that can be used with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium.

Automatic Update of Beam Joints

When you make design changes to structural members that mesh with beam elements, you do not need to recalculate the existing definitions of beam joints in a Simulation study. The program automatically updates the beam joints when you switch to the Simulation study tab.

In the System Options - General dialog box, select **Automatically update beam joints when study is activated**.

During the design process, you can change weldment features in the Model tab such as profile, length, beam rotation angle, suppress or unsuppress status, and conversion to solid body.

The program recalculates the beam joint definitions of the updated weldments when you switch to the Simulation study tab.

Converting a Static Study to a New Study

You can copy a static study to a new linear dynamic study or nonlinear study. Only supported boundary conditions are transferred to the new study.

To convert a Static study to a new study:

1. In a static study tree, right-click the top icon, and click **Copy Study**.
2. Under **Source Study**, select the source static study to copy to a new target study.
3. Under **Study name**, type the name of the new study.
4. Under **Configuration to use**, select the configuration linked to the target study.
5. Under **Target Study**, select one of the available study types **Static**, **Nonlinear**, or **Linear Dynamic**.
6. Click .

All admissible loads and boundary conditions are copied to the new target study. The program issues warning messages when features of the source static study, such as connectors and loads, are not transferred to the target study due to limitations.

You may still need to define time or frequency dependent loads for the target study to run the analysis.

Detecting Stress Hot Spots

The **Stress Hot Spot** diagnostics tool detects regions of the model that exhibit the highest stresses.

Available for static studies.

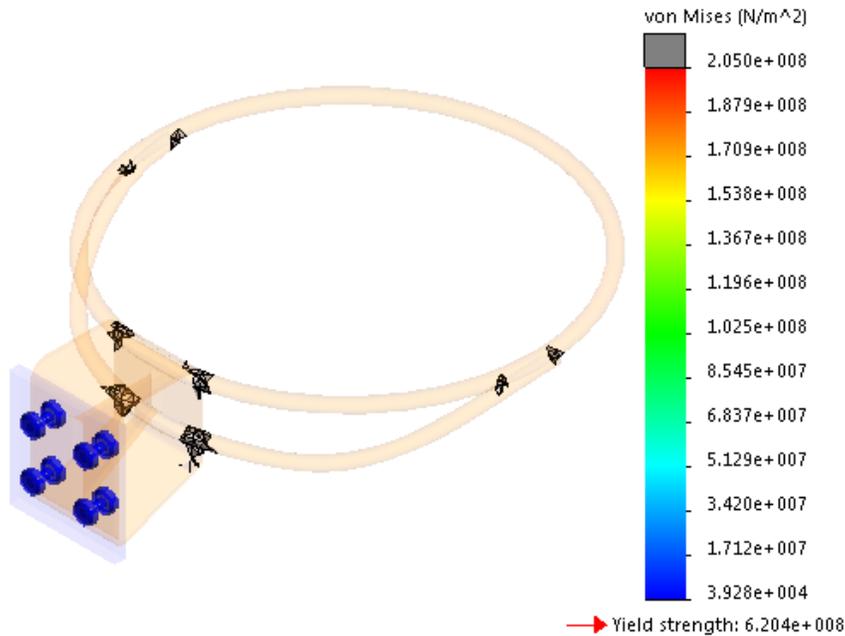
Right-click the **Results**  folder and click **Stress Hot Spot Diagnostics**.

The highest stresses in the areas of concern can be attributed to stress singularities due to geometry features such as sharp corners, point loads, or fixed boundary conditions.

Stress singularities are divergent in nature, meaning that by reducing the mesh size, you expect the value of stress singularities to increase to the theoretical value of infinity. Legitimate stress concentration values are convergent in nature, and they reach a steady value as you refine the mesh.

When stress hot spots are detected, you can view these areas of the model with a stress hot spot plot which shows the variation of the von Mises stresses. You can specify a custom color (default color is gray) for stress values larger than the threshold value that triggers the hot spot detection.

Click **Isolate Stress Hot Spots** in the Stress Hot Spot PropertyManager to show only the areas identified as stress hot spots.



Displaying SOLIDWORKS Simulation Results in the Graphics Area

You can display SOLIDWORKS Simulation results in the SOLIDWORKS graphics area along with other components or bodies in the model that are not part of the simulation.

This way, you do not have to switch between the simulation results and the modeling environment. It is not ideal to display simulation results on bodies and components. This display helps with marketing purposes and predictive design. Previously, if you ran a simulation on a component in an assembly, only the component was shown with the results and all other components were hidden.

An example of predictive design is making model changes based on simulation results.

Available with SOLIDWORKS Simulation licenses.

To display SOLIDWORKS Simulation results in the graphics area:

1. Open *drive letter*:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\model_display\sunglasses.sldasm.

If the Resolve Lightweight Components dialog box appears, click **OK**. The assembly has a simulation study saved with it.

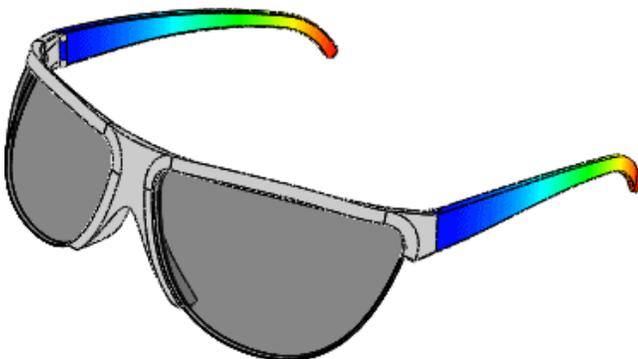


2. Click the Static 1 tab.
3. In the Simulation Study tree, right-click **Static 1**  and click **Run**.
4. Click the Model tab.
5. Click **Simulation Display**  (View toolbar) or **View > Display > Simulation Display**.

If the Simulation Display dialog box appears, click **OK**.

6. In the PropertyManager:
 - a) Under **Selected Result**, select **Displacement1**.
 - b) Click .

The assembly is displayed with the simulation results in the context of the assembly.



The **Simulation Display** command only lists result plots for the studies for which you have loaded all simulation data. To load all simulation data, click **Simulation > Options > System Options > General** and select **Load all Simulation Studies when opening the model (requires more time to open a model)**. You can also load simulation data for a specific study by clicking individual simulation tabs.

Editing Multiple Contact Sets

You can select multiple contact sets in a single step to modify their common properties. You can convert multiple contact sets to a common contact type that is different from their original settings.

The time it takes to edit multiple contact sets is reduced by allowing the selection and modification of multiple contact sets in one step.

Select more than two contact sets and click **Edit Definition**. When you select multiple contact sets of the same type, the properties of the first contact set propagate to the rest.

When you select contact sets with mixed types, the type of contact you select in the Edit Multiple Contact Sets PropertyManager applies to all selected contact sets. All contact sets are converted to a common contact type. The admissible pairs of contact sets for conversion are referenced below.

	Bonded	No Penetration	Allow Penetration	Shrink Fit	Virtual Wall
Bonded	Yes	Yes	Yes	Yes	N/A
No Penetration	Yes	Yes	Yes	Yes	N/A
Allow Penetration	Yes	Yes	Yes	Yes	N/A
Shrink Fit	Yes	Yes	Yes	Yes	N/A
Virtual Wall	N/A	N/A	N/A	N/A	Yes

	Thermal Resistance (Thermal)	Insulated (Thermal)	Bonded (Thermal)
Thermal Resistance (Thermal)	Yes	Yes	Yes
Insulated (Thermal)	Yes	Yes	Yes
Bonded (Thermal)	Yes	Yes	Yes

If you select mixed contact sets (for example, virtual wall and no penetration contacts) that are not eligible for a group conversion to another contact type, the software prompts you for a corrective action.

Improved Arc Length Control Method for Nonlinear Contact Analysis

You can analyze the buckling behavior of any structure more accurately with the inclusion of no penetration contacts between components that may come into contact during the analysis.

The Arc Length Control method supports nonlinear static studies with no penetration contact definitions. 2D Simplification Nonlinear studies also support the improved Arc Length Control Method.

The table lists the features supported by the Direct Sparse and Intel Direct Sparse solvers for each Control method.

	Force Control	Displacement Control	Arc Length Control
Force	Yes	Yes	Yes
Prescribed Displacement	Yes	No	No
No Penetration Contact	Yes	No	Yes
Force + Prescribed Displacement	Yes	No	No
Prescribed Displacement + No Penetration Contact	Yes	No	No
Force + No Penetration Contact	Yes	No	Yes
Force + Prescribed Displacement + No Penetration Contact	Yes	No	No

Neither the Arc Length nor the Displacement Control method can resolve rigid body motions that can occur during a nonlinear solution. If you encounter a rigid body motion during a nonlinear analysis, use the Force Control method with a Prescribed displacement to improve chances of reaching convergence.

Intelligent Solver Switch

The software can automatically select the most robust solver for Nonlinear studies.

In the Nonlinear - Static dialog box, under **Solver**, select **Automatic Solver Selection**.

The program selects the most robust between two solvers depending on the size of the model and available RAM:

- Intel Direct Sparse solver. For small and medium size models with slim geometry. The Intel Direct Sparse solver requires more RAM than the FFEPlus Iterative Solver.
- FFEPlus Iterative Solver. For medium size models with bulky geometry, and large models.

Managing Dynamic Analysis Results

The management of large result files from Linear Dynamic Studies is improved.

To reduce the file size of analysis results for linear dynamic studies, in the Result Options PropertyManager, click **Nodal von Mises stress only (Faster)**.

After the analysis completes, only the von Mises stress component is available for plotting.

Improved algorithms allow for a faster retrieval of sensor-tracked results.

Macro Recording in Simulation

You can record operations performed with the SOLIDWORKS Simulation user interface in a macro script.

With macro recording you can capture these tasks for static and nonlinear studies:

- Create studies
- Define study properties (static studies)
- Apply material
- Apply loads and boundary conditions
- Create Mesh and Mesh Control
- Run the study
- Create Plots

Following is a list of loads and boundary conditions supported for macro recording:

Pressure, Gravity, Centrifugal, Bearing load, Temperature, Convection, Heat flux, Heat power, Radiation, Distributed mass, Uniform and selected base excitation, Fixtures, Drop test setup, Manual contact definitions (only with entity selection not automatically detected contacts), and Component contacts.

Macro operations are accessed from the Macro toolbar or the **Tools > Macro** menu. Recorded macros are saved as `.swp` Visual Basic for Applications (VBA) project files.

Mass Properties in Simulation

You can calculate mass properties of all or selected bodies included in a Simulation study.

Right-click the top icon in the Simulation study tree, and click Mass properties.

The Mass Properties tool considers:

- Solid, beam, and sheet metal bodies with their material definitions assigned in Simulation.

- Thicknesses assigned to shell and surface bodies.
- Composite shells with assigned materials.
- Remote masses.
- Masses added with bolt and pin connectors.

These properties are calculated: mass, volume, surface area, center of mass, and principal moments of inertia (taken at the center of mass).

Offloaded Simulation

You can offload running a simulation to another computer connected on your local network domain. By offloading a simulation, you can continue working with SOLIDWORKS in the coordinator computer executing parallel tasks.

Available for SOLIDWORKS Simulation Premium license and for static and nonlinear studies.

Offloaded simulation distributes the equation solving part of the simulation process to a worker agent computer that is connected on the same network domain.

To offload a simulation you need:

- A computer running SOLIDWORKS Simulation Premium to act as the coordinator.
- A computer with the **SOLIDWORKS Simulation Worker Agent** installed and active to act as the worker agent.
- A network connection between the coordinator and worker agent.

You create a static or nonlinear simulation study on the coordinator. The task of solving the simulation equations is performed on the worker agent. The contact detection during the analysis is performed on both worker agent and coordinator computers.

While the worker agent processes the Simulation, you can continue working with SOLIDWORKS on the coordinator computer. You view and post-process the simulation results on the coordinator computer.

Only the coordinator computer needs a license of SOLIDWORKS and SOLIDWORKS Simulation Premium.

Running an Offloaded Simulation

To run an offloaded Simulation the coordinator and the worker agent machines must be on the same network domain. The SOLIDWORKS Simulation Worker Agent is installed and activated on the worker agent computer.

To run offloaded simulation:

1. For the worker agent computer on the network:
 - a) From the SOLIDWORKS Installation Manager, install the **SOLIDWORKS Simulation Worker Agent** client.

The Intel[®] MPI Library Hydra Process Manager (includes Intel's Message Passing Interface MPI executables) is installed which establishes communication protocols between the coordinator and worker agent machines to execute successfully an offloaded simulation.
 - b) Run **SOLIDWORKS Simulation Worker Agent** from the Windows Start menu. (**Simulation Worker Agent** default installation folder: *drive letter:*/Program Files/Common Files/SOLIDWORKS Shared/Simulation Worker Agent). Click **Activate Worker Agent**. The worker agent computer is now ready to run an offloaded simulation.

To change the temporary folder location, click **Deactivate Worker Agent**, and edit the folder location where temporary files are stored.

2. For the coordinator computer on the network:
 - a) Create a static or nonlinear Simulation study.
 - b) Click **Manage Network** (Simulation CommandManager). In the User Account Control dialog box, click **Yes** to allow **SOLIDWORKS Simulation Network Share** to make changes to your computer. Type your Windows log-in password for the first time you run an offloaded simulation on the coordinator computer.

In the **SOLIDWORKS Simulation Network Manager**, all worker agent machines with the SOLIDWORKS Simulation Worker Agent installed and activated are listed. The coordinator computer is highlighted in green at the top of the list. Click **Refresh** to update the list of available worker agents detected in the network.

- c) Click **Offloaded Simulation** (Simulation CommandManager).

The solver switches to **Intel Network Sparse**.

- d) From the list of available worker agents on the network, select one to process the offloaded simulation, and click **OK**.

If the **SOLIDWORKS Simulation Network Manager** does not detect any worker agents, make sure at least one worker agent computer is on and the worker agent is active; the coordinator and worker agent machines must be on the same subnetwork.

Computers highlighted in red are not usable because they are either worker agents that are running offloaded Simulation jobs, or act as coordinators and are sharing the Simulation folder for other offloaded simulations that are in process.

- e) Run the study.

Once the offloaded simulation is running, the status of the selected worker agent computer changes from idle to active. Information about the status of the offloaded simulation is shown in the **SOLIDWORKS Simulation Worker Agent** window.

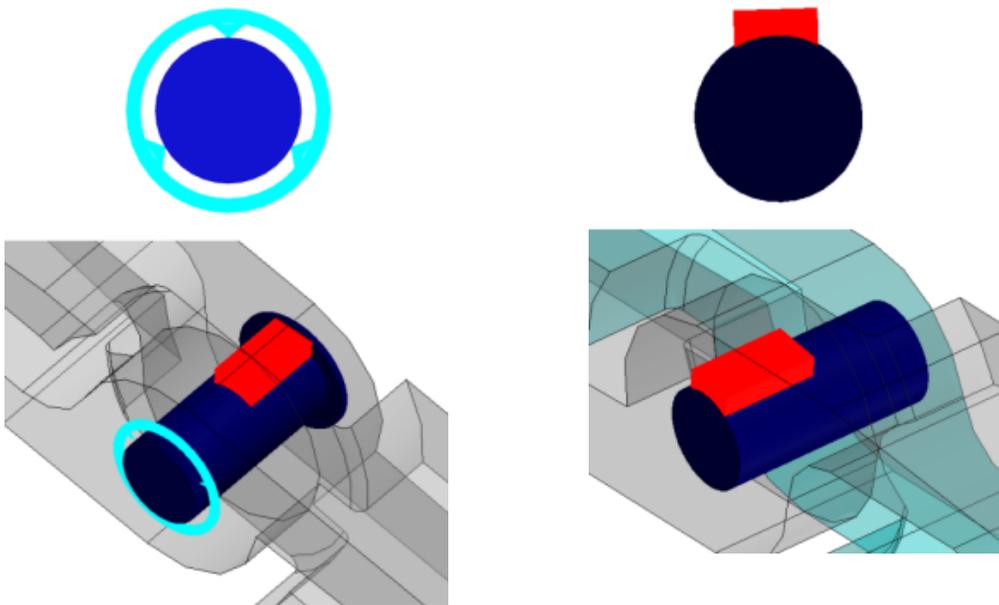
When the offloaded simulation completes, the **Simulation Worker Agent** client copies the result files back to the coordinator computer's hard drive.

- You do not need to install SOLIDWORKS or SOLIDWORKS Simulation on the worker agent computer to run offloaded simulation.
- You do not need to install the SOLIDWORKS Simulation Worker Agent client on the coordinator computer, unless you plan to use the coordinator computer as a worker agent.
- To check that both the coordinator and worker agent computers are on the same network domain, open a Windows Command Processor window, and run the command `net view`. This command returns a list of computers on the same subnetwork with the one being used to run the command. Both computers, coordinator and worker agent, should appear on the list to run an offloaded simulation.
- Make sure that the Windows Firewall does not block the executables of offloaded Simulation: `mpiexec.hydra.exe`, `hydra_service.exe`, and `pmi_proxy.exe` located in `drive letter:\Program Files\Common Files\SOLIDWORKS Shared\Simulation Worker Agent`. If you are still unable to run an offloaded simulation, consider turning off the Windows Firewall for the network domain.

Performance Improvements for SOLIDWORKS Simulation

Several improvements in the areas of graphics, meshing, and contact formulation enhance the overall performance of the Simulation product.

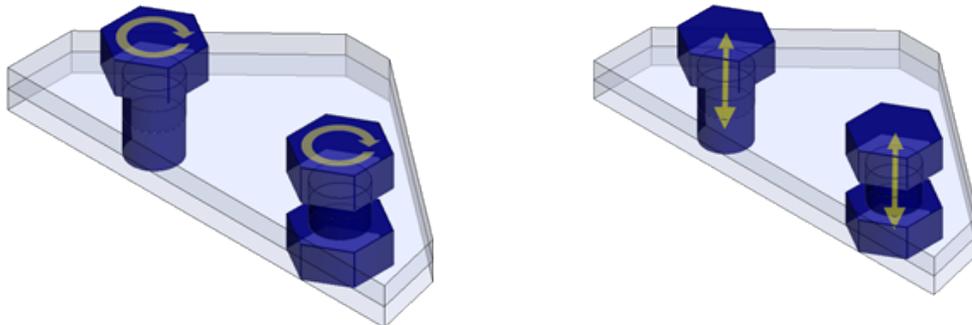
- The symbols for Pin connectors are redesigned. With the redesigned symbols, you can identify in the graphics area the type of pin connector (with retaining ring or key) without opening the pin connector's PropertyManager.



Symbol for pin with retaining ring
(no translation)

Symbol for pin with key (no rotation)

- Pre-load symbols for bolts identify the axial or torque applied pre-load.



Symbols for bolts with torque
preload

Symbols for bolts with axial preload

- Failure indices for the composite shells failure criteria (Tsai-Hill, Tsai-Wu, and Maximum Stress) are calculated from nodal stresses. Previously, the failure indices were calculated from elemental stress values.
- The blended curvature based mesher supports h-adaptive and p-adaptive methods.
- Optimized memory allocation and multi-threaded support for the Intel compilers improve the performance of studies with surface to surface bonding and no penetration contact definitions. The time spent by the program on the formulation phase for bonding and no penetration contacts is reduced.
- Notable performance improvement on reducing the FFEPlus iterative solver solution time for static studies.

Post Processing Improvements for SOLIDWORKS Simulation

Several improvements in post processing of results and plot viewing enhance productivity.

- With the addition of the **Synchronize Views** tool  in the **Compare Results** mode, the synchronization of view orientation and size of the model (pan, zoom, and rotate) is automatic among all the view panes.
- In the **Compare Results** dialog box, click **Save image as** to save the active multi-pane result plots to a *.png, *.jpg, or *.bmp file format.
- You can set the maximum value of a stress plot to the material's yield stress value. Click the down arrow  of the maximum value box, and select **Yield Strength: xxx**.
- In a results plot's chart legend, you can: (a) click **Reset color above maximum to default**  to reset the custom color applied to the results above a given maximum back to the default color, (b) click **Reset color below minimum to default** , or (c) click the top or bottom color swatch outlined by a box to set a custom color.
- You can easily toggle the display of mode shapes in the graphics area. At the bottom of the mode shape legend, click << to display the previous mode shape, and >> to display the next mode shape.
- Right-click **Results** and click **Compare Modes**  to compare selected modes up to a maximum of four (available for frequency, linear dynamic, and buckling studies).
Click **Mode Selection Filter**  to select the number of modes to compare.
- You can save the deformed geometry of a model as a tessellated body, surfaces only, or mesh formats. In the **Body from Deformed Shape** PropertyManager, expand **Advanced Export**, and select one of the available options.

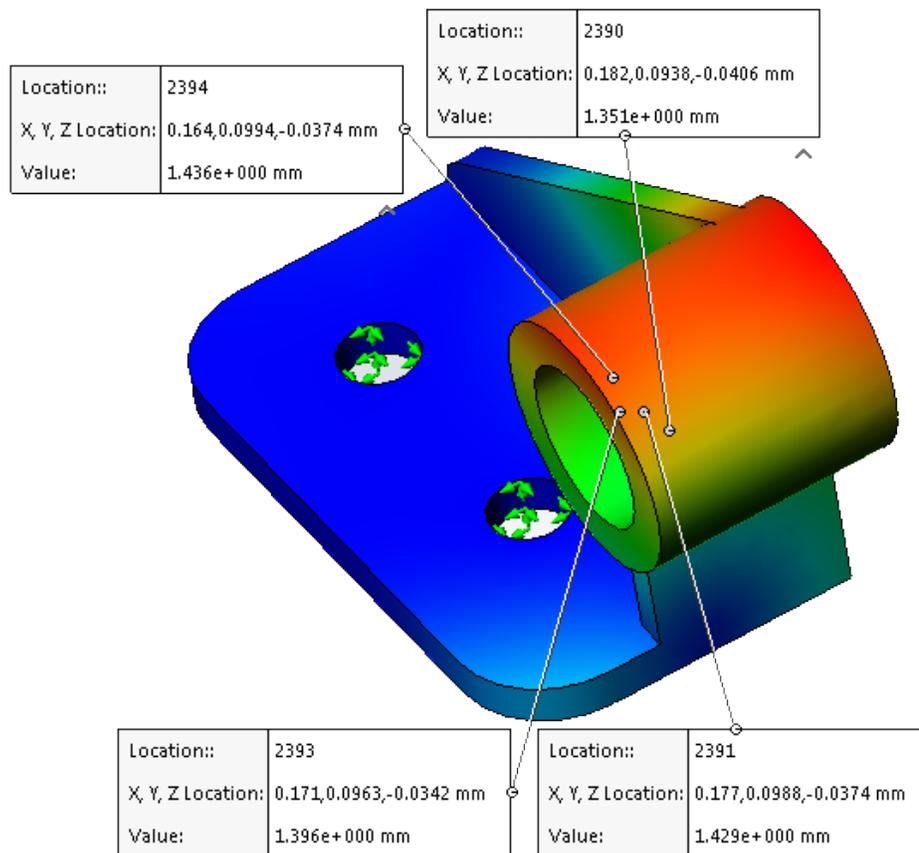
Probing Results at Selected Nodes

After you run a Simulation, you can probe results at nodes identified by their node number. The nodes you select from List Result and Probe Result PropertyManagers are highlighted on the model with their annotations.

To probe results at selected nodes:

1. In the Probe Results PropertyManager, under **Options**, select **At Node number**.

2. Type the number of a single node or a range of nodes, for example 1000-1020. The locations of the selected nodes with their annotations are visible in the graphics area.



3. Under **Report Options**, click **Capture Image** .
4. Save the active view of the model in *.png , *.jpg or *.bmp file format.

Remote Loads and Mass for Beams

You can define remote loads, remote mass, and distributed mass over beams.

All available options related to remote load, remote mass, and distributed mass in the Remote Loads/Mass PropertyManager extend to studies with beam elements.

You can convert a solid body to a remote mass and transfer the remote mass to a beam element. In the Treat as Remote Mass PropertyManager, select **Joints**  or **Beams** .

Available for Static, Frequency, Buckling, and Linear Dynamics studies.

Self-dismissed Solver Messages

During a Simulation run, solver messages that prompt you with a choice of action automatically time out.

Click **Simulation > Options > System Options > General** and select **Automatically timed out decision solver messages after n seconds**.

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Sketching

This chapter includes the following topics:

- **Creating Sketch Offsets on 3D Geometry Surfaces**
- **Enhancements to the Segment Tool**
- **Prevention of Accidental Micro Lines**
- **Shaded Sketch Contours**
- **Extruding Shaded Sketch Contours**

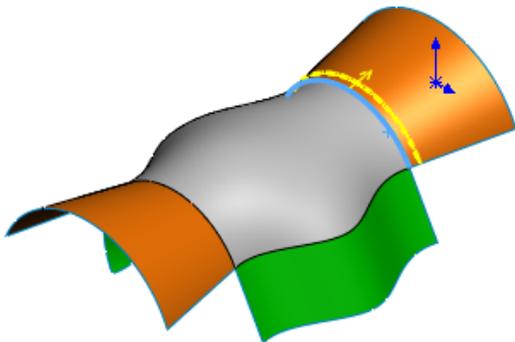
Creating Sketch Offsets on 3D Geometry Surfaces

You can use the **Offset on Surface** tool to offset 3D model edges and model faces in a 3D sketch.

Previously, you had to create extra features for offsetting an edge.

To create sketch offsets on 3D geometry surfaces:

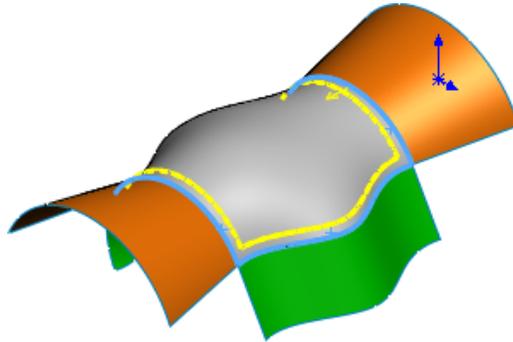
1. Open *drive Letter:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS 2017\whatsnew\Sketching\CurvedSurface.sldprt*.
2. Click **Offset on Surface**  (Sketch toolbar) or **Tools > Sketch Tools > Offset on Surface**.
3. In the graphics area, select the edge of **Surface-Loft4** as shown.



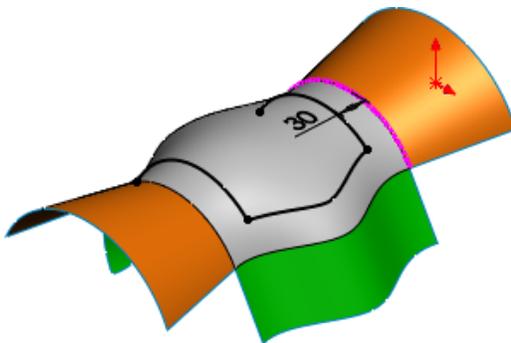
4. In the PropertyManager:
 - a) Set **Offset Distance** to 10.
 - b) Select **Reverse**.

The entity is projected on the opposite face.

You can only use **Reverse** when the selected edge is connected to faces that belong to the same body.



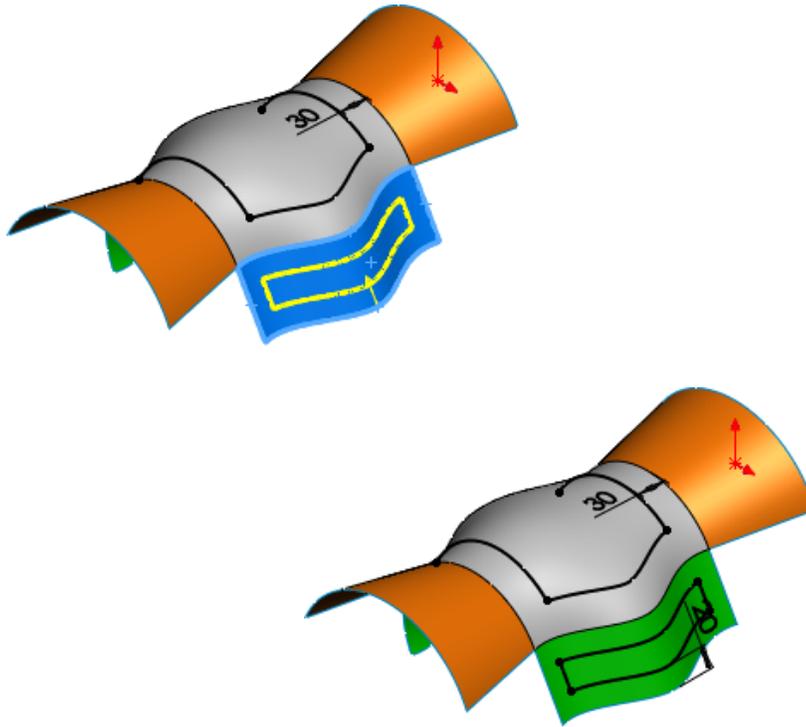
5. Select the interior edges of **Surface-Loft5** and **Surface-Loft3**.
6. Click .
7. Double-click the dimension value of the three edges and enter 30 in the dimension input value box.



8. Click **Offset on Surface** .

9. In the PropertyManager:
 - a) Click **Surface-Loft5** in the graphics area.
 - b) Set **Offset Distance** to 20.
 - c) Click **✓**.

All edges of **Surface-Loft5** are offset.



Enhancements to the Segment Tool

You can use the **Segment** tool to create equal length segments in splines, parabolas, ellipses, partial ellipses, and conics.

Previously, the **Segment**  tool was available for circles and arcs only.

When you use the **Segment** tool for splines, parabolas, ellipses, partial ellipses, and conics:

- Equidistant relations are created in a single **Segment** operation.
- The equidistant relation is replaced by a coincident relation after the equidistant relation is deleted from a single **Segment** operation.

Prevention of Accidental Micro Lines

A dynamic buffer zone helps to maintain a consistent zone in which the click-click sketch mode results in the expected geometry with fewer cases of accidental micro lines.

Previously, the creation of accidental micro lines caused open contours, so you could not use the sketch for creating a feature.

Shaded Sketch Contours

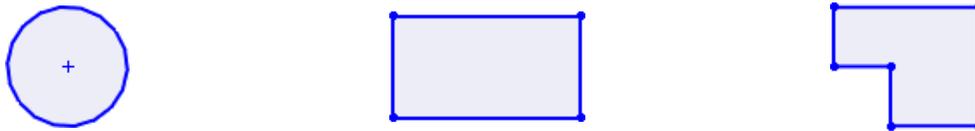
You can use the **Shaded Sketch Contours** setting to view closed sketch contours and sub-contours as shaded entities. You can drag, resize, and apply relations to shaded sketch contours.

Previously, you had to use the **Move** tool to drag sketch contours.

When you select the **Shaded Sketch Contours** setting, only the closed sketch shapes are shaded. This makes it easier to tell if a shape is fully closed or not.

Contours and sub-contours

Contour. A contour is a closed shape.



Sub-contour. A sub-contour is the contour that is formed when you combine contours together.



The **Shaded Sketch Contours** setting lets you:

- Drag the shaded area.
- Apply relations to entities.
- Make blocks of the shaded sketch selections with the **Make Block** tool on the shortcut menu.
- Use the **Extruded Boss/Base** tool on the context toolbar by right-clicking the shaded sketch.
- Use shaded contours as input for the **Mirror Entities** and **Offset Entities** tools.

The **Shaded Sketch Contours** tool is available only for new sketches in SOLIDWORKS 2017.

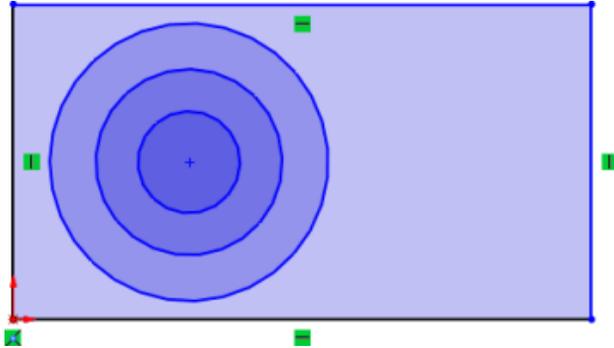
Extruding Shaded Sketch Contours

To extrude shaded sketch contours:

1. In sketch mode, click **Shaded Sketch Contours**  (Sketch Toolbar) or **Tools** > **Sketch Settings** > **Shaded Sketch Contours**.

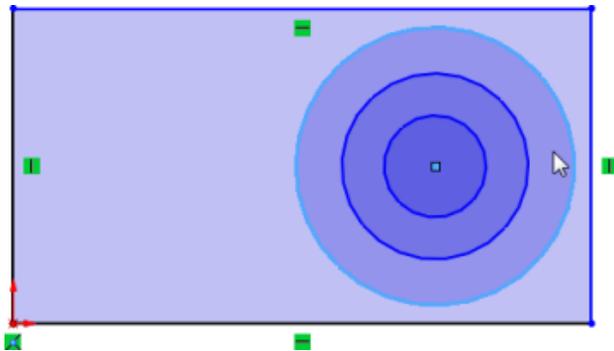
Use **Shaded Sketch Contours** to turn-off the tool.

2. Sketch three concentric circles inside a rectangle.



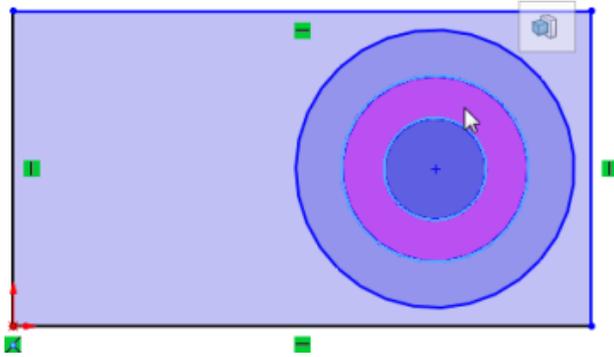
The outermost circle is the lightest and the innermost circle is the darkest shade of blue.

3. Click any sub-contour of the sketch entity and drag it.



The whole sketch entity moves.

4. Press **Alt** and click a shaded area.



The **Extruded Boss/Base**  tool appears on the context toolbar.

5. Click **Extruded Boss/Base**.
6. Set options in the PropertyManager and click  .
The shaded area is extruded.

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SOLIDWORKS Toolbox

This chapter includes the following topics:

- **Purging Inactive Toolbox Configuration Data**
- **Resizing PEM Fasteners**
- **Updating Toolbox Components in Assemblies**

SOLIDWORKS Toolbox is available in SOLIDWORKS Professional and SOLIDWORKS Premium.

Purging Inactive Toolbox Configuration Data ★

The Toolbox Settings **Purge Inactive Data** tool lets you purge inactive configuration data from Toolbox part files.

By removing unused data that increases file size, **Purge Inactive Data** improves performance for actions like opening the Insert Component PropertyManager.

Inactive configurations are configurations that are not listed in Toolbox Settings and are not in the part file. If a configuration existed in the past and was used in an assembly, it is still considered active even if it is not listed in the configuration list in Toolbox Settings.

Purge Inactive Data purges the following data from inactive configurations:

- Custom properties
- Configuration names
- Designations
- Part numbers
- Descriptions
- Comments

The command does not:

- Purge geometry data or active configuration data.
- Affect custom configurations you create using the Insert Component PropertyManager when you insert a Toolbox part into an assembly.

It is highly recommended that you back up your data before purging inactive data.

To purge inactive Toolbox configuration data:

1. From Windows, click **Start > All Programs > SOLIDWORKS version > SOLIDWORKS Tools > Toolbox Settings** .
2. In the Toolbox dialog box, click **2 Customize your hardware**.

3. Expand the hardware display to the level where you want to purge inactive data. You can purge at the folder level or on a part-by-part basis.
4. Click **Purge Inactive Data**.
5. In the Purge Inactive Data dialog box, expand **Options** and select from the following:
 - **Purge inactive configuration data from disabled hardware.** Available if you purge at the folder level. Purges data from enabled and disabled configurations.
 - **Save Purge Report.** Browse to the location to save the report. Generates a purge report summarizing the results of the data operation. The report is a tab delimited text file that you can open in Microsoft Excel.

The file name is in the following format:

*hierarchical address of the selected component data purge results
date and time stamp*

For example:

Ansi Inch_bolts and screws_hex head data purge results
-2016-03-25_01-55-02-PM.txt

6. Click one of the following:
 - **Yes, purge all inactive configuration data.** A progress bar indicates that the purge is taking place.

Folders or parts containing large numbers of configurations may take a long time to purge.

- **No, do nothing and cancel the operation.** The dialog box closes without purging any data.

Resizing PEM Fasteners

When you insert PEM fasteners, the outer diameter of the collar matches the inner diameter of the hole.

To resize PEM fasteners:

1. In the Configure Component PropertyManager for a mated fastener, select **Autosize to mated geometry**.
2. Rebuild the assembly containing the fastener.

The software updates the fastener size.

If the fastener was correctly sized, the software does not make any changes.

Updating Toolbox Components in Assemblies

You can update all Toolbox components in a single action.

When an assembly contains Toolbox components at the top level or in a subassembly, the assembly shortcut menu includes an **Update Toolbox Components** menu item.

Depending on the assembly structure, a flyout submenu give you options to update:

- **Top Level Assembly Only**
- **Sub-level Assemblies Only**
- **All Assembly Levels**

To see the **Update Toolbox Components** item, Toolbox components must be present and you must select the SOLIDWORKS Toolbox Library Add-in.

You can update:

- Toolbox parts that were created using the Toolbox editor.
- User created parts that were added to Toolbox and are driven by a design table, configured feature, or configured dimension.
- Copied Toolbox standards.

You cannot update:

- Suppressed Toolbox components.
- Parts that were Toolbox components but have had their Toolbox flags removed.
- Toolbox components in a Smart Fasteners folder.

Newly added custom properties are not applied to active configurations. Each component with a newly added custom property is flagged with a warning. You must manually edit the component to correct the warning.

To update Toolbox components in assemblies:

1. In the FeatureManager design tree, right-click the top level assembly, click **Update Toolbox Components**, and choose a submenu option.

A message warns you that the operation cannot be undone and that the process may be lengthy.

2. Click **OK**.

The software copies data and dimensions updates from the Toolbox Settings master part file and applies them to the active configurations in the appropriate level of the active assembly.

3. To save the updates, save the assembly.

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SOLIDWORKS Visualize

This chapter includes the following topics:

- **Installing SOLIDWORKS Visualize**
- **SOLIDWORKS Visualize Add-in**
- **SOLIDWORKS Visualize Boost**
- **Geometry Opacity and Animation Support**
- **Context Sensitive Help**
- **Creating Section Views**
- **Home Screen**
- **Pivot Point Snapping**
- **Using SOLIDWORKS Camera Navigation and Shortcuts**
- **Additional Enhancements**

Installing SOLIDWORKS Visualize

SOLIDWORKS Visualize is available from the SOLIDWORKS Installation Manager.

SOLIDWORKS Visualize allows you to render high-quality images from files created in SOLIDWORKS and other design tools. You can install, update, and remove SOLIDWORKS Visualize using the Installation Manager. It requires an additional serial number, and is usually installed on a computer that is running the SOLIDWORKS software.

SOLIDWORKS Visualize now supports SolidNetWork Licensing (SNL) license types (floating), as well as offline activations.

Previously, you could only install SOLIDWORKS Visualize from a separate installation package.

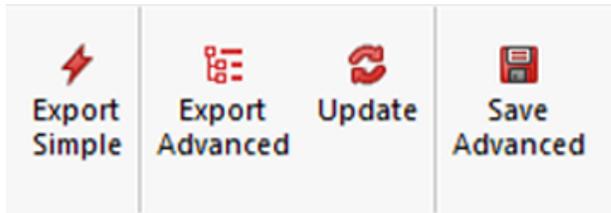
SOLIDWORKS Visualize Add-in

The SOLIDWORKS Visualize add-in, which is available in all versions of SOLIDWORKS, lets you export SOLIDWORKS features directly to SOLIDWORKS Visualize.

You can export the following SOLIDWORKS features:

- Geometry
- Appearances
- HDR lighting
- Decals

With SOLIDWORKS Professional, you can also export motion studies. Animation export is limited to rigid body, camera, and appearance animations. The following commands are available in the add-in:



- **Export Simple** saves a copy of the model and loads it into SOLIDWORKS Visualize using **Appearance** mode part grouping.
- **Export Advanced** saves a copy of the model and loads it into SOLIDWORKS Visualize with **Automatic** mode part grouping and **Monitor File** enabled. You can also upload motion studies if you are using Visualize Professional.
- **Update** exports the latest version of the model and triggers an update in SOLIDWORKS Visualize. This button is grayed out until you click **Export Advanced** for the first time.
- **Save Advanced** saves a copy of the model and motion studies to a specific location, so you can later import the model into SOLIDWORKS Visualize.

SOLIDWORKS Visualize Boost

SOLIDWORKS Visualize Boost is a service that allows you to distribute rendering from a computer running SOLIDWORKS Visualize Professional to one or more computers on a network.

You can perform other tasks on your machine while the rendering is being carried out, and the final rendered images are available sooner. Images can be rendered on a single machine, or on a cluster of several machines running Visualize Boost.

One complimentary license of SOLIDWORKS Visualize Boost is included with each SOLIDWORKS Visualize Professional on active subscription. Additional seats of Visualize Boost are available for purchase.

Installing Visualize Boost with SLDIM

You install Visualize Boost using the SOLIDWORKS Installation Manager. It requires a separate serial number.

Because the purpose of Visualize Boost is to offload rendering, you should install Visualize Boost on computers different from the ones running SOLIDWORKS Visualize Professional. Rendering performance can be maximized by installing Visualize Boost on computers with multiple CPU cores and graphics processing units (GPUs).

To install SOLIDWORKS Visualize Boost:

1. Start the SOLIDWORKS Installation Manager.
2. On the Welcome page, select **Individual**.

3. Do the following:
 - a) On the Serial Number page, expand **Visualization**.
 - b) Select **SOLIDWORKS Visualize Boost** and type the serial number.
 - c) Click **Next**.
4. Follow instructions for system warnings and click **Next**.
5. On the Summary page, accept the terms of the SOLIDWORKS License Agreement and click **Install Now**.

Double-click the SOLIDWORKS Visualize Boost shortcut icon on the desktop to open the interface.

Activating the Visualize Boost License

You must activate and configure SOLIDWORKS Visualize Boost through a web interface before you can use the product for rendering. Chrome, Firefox, or Internet Explorer browsers are supported.

After you activate and configure Visualize Boost, it launches automatically at system startup and runs in the background. No further action is required.

To activate the SOLIDWORKS Visualize Boost license:

1. Do one of the following:
 - a) From your desktop, double click the **Visualize Boost** shortcut icon.
 - b) Open a browser and enter the URL: `http://localhost:8993/config.html`.
2. Enter an email address and click **Enable License**.

Configuring Visualize Boost

For most networks, the default settings are sufficient to configure a computer for Visualize Boost, but if your network environment requires other settings, follow these steps to manually configure a computer.

To configure a computer to be part of a rendering cluster in Visualize Boost:

1. Do one of the following:
 - a) If your system is running SOLIDWORKS Visualize Boost, double click the Visualize Boost shortcut icon on your desktop or open a browser and enter the URL:
`http://localhost:8993/config.html`.
 - b) If you are on a different system, open a browser and enter the URL: `http://<IP Address>:8993/config.html` or `http://<DNS name>:8993/config.html`.
2. Select your **Language** from the drop-down list in the upper right corner of the screen.
3. Click the **Setup** tab to configure the Visualize Boost node.

4. In the **Cluster Name** field, specify a computer name, such as *Superfast Render*, *Presentation Cluster*, or *4th Floor Computers*.

The cluster name displays in SOLIDWORKS Visualize Professional as a cluster profile. To render an image, users running SOLIDWORKS Visualize Professional select a cluster profile from a list of available Boost clusters.

It is recommended that you specify the cluster name on the Visualize Boost computer, rather than locally on the Visualize Professional computer, so that all Visualize users have access to the same cluster names in their Visualize Boost Profiles list.

5. Enter the **Cluster Client Access Port**.

This is the port that computers running SOLIDWORKS Visualize use to connect to the Boost cluster. The default cluster client access port is 8998.

6. In the **Networking Mode** area, select an option based on your network environment:

- Select **No Networking** to prevent this computer from belonging to any other cluster. This mode is useful for troubleshooting issues with Visualize Boost.
- Select **UDP Multicast Mode** if your network supports UDP Multicast. Visualize Boost will search the network and form a cluster of all computers using UDP Multicast Mode on the same port. This mode is recommended, if your network supports it.
- Select **TCP Mode with Discovery (Default)** if you cannot use UDP Multicast mode. Visualize Boost will search the network and form a cluster of all computers using TCP Mode with Discovery on the same port.
- Select **TCP Mode** if you need to create an explicit list of the computers in the cluster, because the other modes are not successful in finding the computers you want to include. This mode requires additional work to configure and is not recommended if either UDP Multicast or TCP Mode with Discovery are available.

7. If you select **UDP Multicast Mode** or **TCP Mode with Discovery**, enter the **Cluster Discovery Address**.

This setting must be an IP address and a port above 1024. The default port varies among the networking modes:

- 8994 for UDP Multicast
- 8995 for TCP Mode with Discovery
- 8996 for TCP

These ports are used for Visualize Boost computers to connect with each other. All computers configured with the same **Cluster Discovery Address** and that are on the same local network will form a cluster.

8. If you select **TCP Mode**:

- Enter a **Cluster Access Port**. This is the port that Visualize Boost nodes in the cluster use to communicate with each other. The default port for TCP Mode is 8996.
- If this machine is going to be the "head node" of the cluster (the machine that coordinates the work of the cluster), use the **Cluster Address List** field to list the IP addresses of other nodes in the cluster. Each IP address (such as 192.168.0.100:8996) must include the **Cluster Access Port** configured for all nodes in the cluster. Leave the field blank if this machine is not going to be the head node of the cluster.

9. Click **Save Configuration**. Visualize Boost saves the new configuration, then stops and restarts the service.
10. Click the **Status** tab. If the status is **Ready**, the screen displays the computers connected in the cluster and the **Sessions** (jobs) assigned to the cluster.

Selecting a Visualize Boost Cluster for Rendering

Before you can use a Boost cluster for rendering, you must add it to a Profiles list on your SOLIDWORKS Visualize Professional computer and then, select it.

To add and select a Boost cluster in the Profiles list:

1. In SOLIDWORKS Visualize, click **Tools** > **Options** and select the **Boost** tab.
2. Click **Scan** to search the network for Boost clusters.
3. Click **Add** to add a new cluster to the list of cluster profiles. Only select NVIDIA VCA if you have installed an **NVIDIA Quadro VCA** on your network.
4. In the Profiles list, highlight the cluster name. The **Boost Settings** area displays the status, cluster name, and the cluster discovery address.
5. If needed, adjust PowerBoost settings for real-time ray tracing and click **Connect**.

Adjusting PowerBoost Settings for Ray Tracing

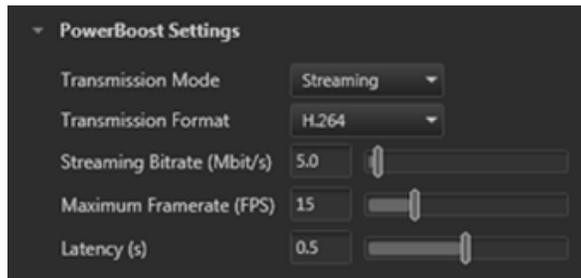
Ray tracing is a technique for rendering 3D scenes. Ray tracing traces the path of every ray of light from its source until it either leaves the scene or becomes too weak to have an effect. The term also applies to the reverse method: tracing the path of every ray of light from the camera back to the light source.

PowerBoost (available only with SOLIDWORKS Visualize Professional) provides a render mode that streams ray tracing directly to your Visualize Viewport. An Internet connection is required, and a multi-GPU machine for a Visualize Boost computer or an NVIDIA Quadro VCA is recommended for the best performance.

To adjust PowerBoost settings for ray tracing:

1. In SOLIDWORKS Visualize, click **Tools** > **Options** and select the **Boost** tab.
2. In the **Profiles** list, select a cluster profile with a high performance machine or an NVIDIA Quadro® VCA.

- The default settings for PowerBoost are acceptable for most networks. You can adjust the following settings as needed:



- **Transmission Mode: Streaming** is the default and applies to network connections with low bandwidth and high latency. Select **Synchronous** if the network connection has very high bandwidth and low latency, which is common in local area networks.
- **Transmission Format: H.264** is the default and uses relatively little bandwidth, which increases performance. **Lossless** provides slightly higher image quality, but consumes much more bandwidth and reduces performance.
- **Streaming Bitrate:** Higher values increase image quality but consume more network bandwidth. Lower values save bandwidth but reduce image quality.
- **Maximum Framerate:** Higher values increase image quality but make the application less responsive. Lower values increase responsiveness but reduce image quality.
- **Latency:** Higher values increase image quality but make the software less responsive to user input (such as camera movement). Lower values increase responsiveness but reduce image quality.

Show Log Files

In SOLIDWORKS Visualize, from **Tools > Options > Boost** tab, click **Show Log Files** to review log files for the Boost service, Queue, Visualize Queue Server, Offline Renderer, and others.

Share Cluster Profiles

You can save a cluster profile and share it with other SOLIDWORKS Visualize Professional users on your network.

To share a cluster profile:

- In SOLIDWORKS Visualize, click **Tools > Options** and select the **Boost** tab.
- Click **Save** to save the profile on your hard drive and send the file to a colleague.
- Your colleague can then start SOLIDWORKS Visualize Professional, go to **Tools > Options > Boost** tab, and click **Load** to load the file to the **Profiles** list.

Rendering Images

To render output to a Visualize Boost cluster:

1. In SOLIDWORKS Visualize Professional, open a model and click **Tools** > **Render**, or in the main toolbar, click **Output Tools** .
2. In the Output Tools dialog box, under the **Render Settings** area, click **PowerBoost** .

The Cluster name displays.

3. Set the **Render Mode**. You can set a time limit or select **Quality** and enter a number of **Passes**. The default of 500 passes is adequate for most render jobs.
4. To start rendering at a later time, select **Send to Queue**.

The job is sent to the SOLIDWORKS Visualize Queue and does not complete until you click **Start Queue** in the Queue Client.

5. If you are ready to render, select **Show Progress** to monitor the rendering process. Then click **Start Render**.

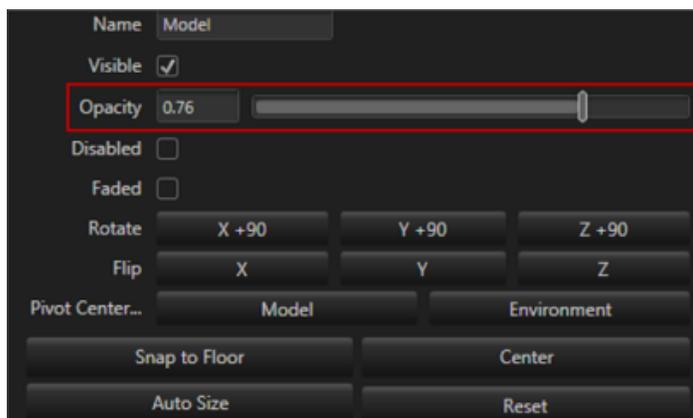
You can click **Pause**, **Save Now**, or **Cancel** to stop the rendering.

When using Visualize Boost for rendering, the Visualize Professional computer must stay online and connected to the network. The computer cannot be turned off or unplugged from the Internet until Visualize Boost has completed the job.

Geometry Opacity and Animation Support

You can adjust the opacity of parts, groups, and models to help show the inner details of complex projects.

With the part selected, adjust the **Opacity** slider in the Models tab to fade the transparency.



Results are updated in real-time in the Viewport.



For animations, you can also adjust keyframes using the **Opacity** slider.

The fading of parts is only supported in **Preview** and **Accurate** modes.

Context Sensitive Help

In SOLIDWORKS Visualize dialog boxes, you can display help that is specific to the dialog boxes by pressing **F1**.

In other places in the user interface such as on a Palette tab, pressing **F1** opens the help at the *Welcome to SOLIDWORKS Visualize* topic. Use the Search tab to locate specific help topics.

Creating Section Views

To display the internal parts of a model, you can create section views by adding cut planes.

You can manipulate cut planes using the standard move controls. You can use multiple cut planes to further dissect your model. There is no limit.

In SOLIDWORKS Visualize Professional, you can also animate the effect of cut planes the same way that you animate a model.

To create section views:

1. On the Models tab, click **New Cutting Plane**.

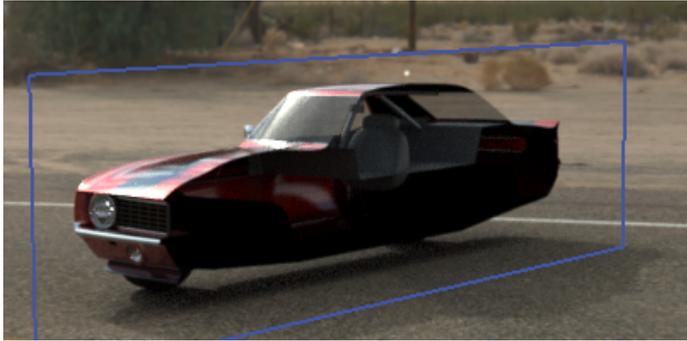
You can also create a cutting plane by clicking **Project > Models > New Cutting Plane**.

The cutting plane appears in the Viewport as a faint, mostly transparent plane that automatically stretches across your entire model.

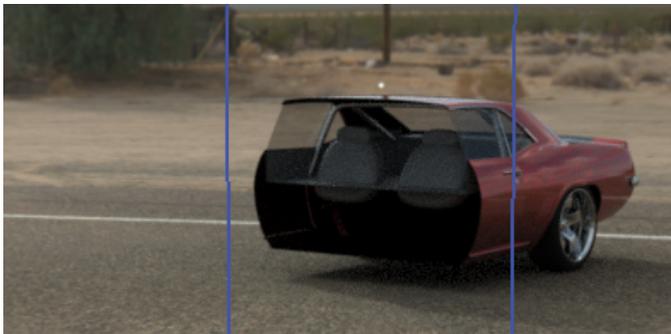


2. In the model tree, select the cut plane.
The cut plane parameters are listed below the model tree.
3. Type a **Name** for the plane.

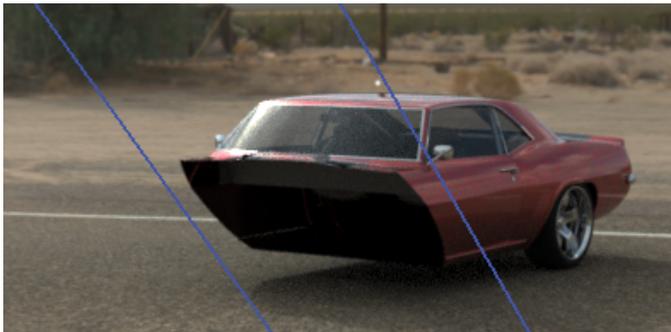
4. Use the **Rotate** and **Transform** options to modify the plane's orientation.
 - To reverse the plane, click **Flip**.



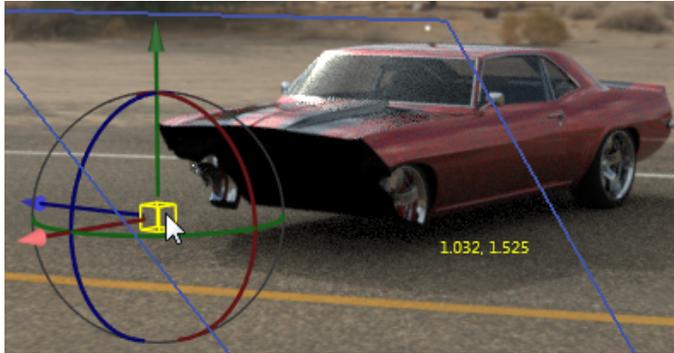
- To rotate the plane 90°, click one of the **Rotate** options.



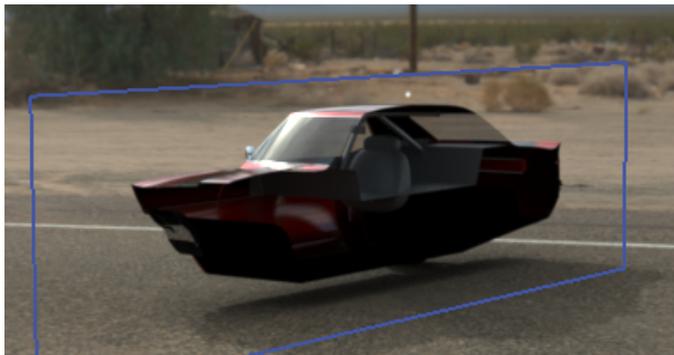
- To tip the plane, under **Transform**, change one or more of the **Rotation XYZ** values.



You can also use the transform manipulator to modify the position of the plane. To enable the manipulator, right-click the cut plane in the **Model Set** and click **Manipulate > Move**.



5. To add an additional cutting plane, click **New Cutting Plane**.



6. Select the second plane and use the same controls to manipulate it.
7. To hide a cut plane, do one of the following:
- In the **Model Set**, right-click the cut plane and click **Hide Cutting Plane**.
 - In the cut plane parameters, click **Hidden**.
8. To remove a cut plane, in the **Model Set**, right-click the cut plane and click **Delete**.

You can also animate cut planes as well.

Cut planes affect all geometry in a scene. They cannot be applied to only specific parts, groups or models.

Home Screen

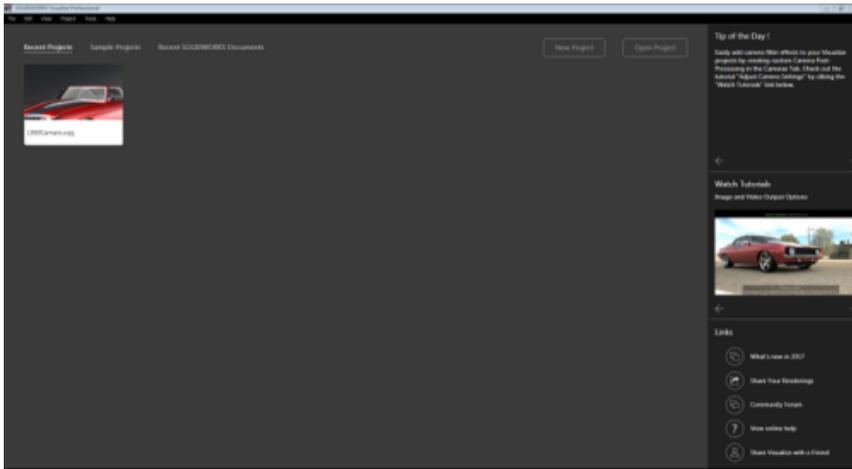
In Visualize's home screen, you can easily find projects that you opened recently, and perform other tasks.

You can view thumbnails of recently opened SOLIDWORKS Visualize (*.SVPJ) and Bunkspeed (*.BIF) projects, open the projects with a click, "pin" projects so they always appear on the home screen ahead of unpinned projects, and show the folder in which each project file is stored in Windows Explorer.

In a separate tab, you can view thumbnails of recently opened SOLIDWORKS assemblies and parts (*.SLDASM, *.SDPRT), open them by clicking on a thumbnail, and show the folder in which each file is stored in Windows Explorer.

You can find and download sample projects stored in the Visualize cloud library.

Links on the home screen also make it easy to read tips, watch tutorials, view online help, and access community forums, where you can share and discuss renderings.



Pivot Point Snapping

Pivot point snapping lets you accurately locate a model's pivot point on the model surface.

To place the pivot point:

1. Click **Object Manipulation Tools** > **Pivot** .
2. Hold the **Ctrl** key.

A small blue dot that appears in the Viewport, overlaid on top of the model, shows where you can snap the pivot location.

Using SOLIDWORKS Camera Navigation and Shortcuts

Users familiar with SOLIDWORKS CAD camera navigation can use it instead of the standard SOLIDWORKS Visualize camera navigation.

To use SOLIDWORKS CAD camera navigation, do the following:

- While running SOLIDWORKS Visualize, click **Tools** > **Options**. In the Options dialog box, on the User Interface tab, for **Navigation**, select **SOLIDWORKS CAD** mode.

When you manipulate the camera, the pointer corresponds to the selected tool.

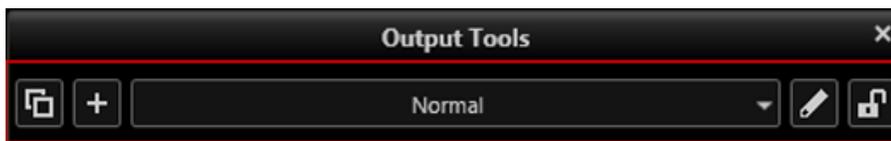
The following navigation behavior is available:

Action	Cursor	Visualize keyboard shortcuts	SOLIDWORKS keyboard shortcuts
Rotate view		Alt + click	Drag with the middle mouse button. Press the arrow keys. Hold down Shift to rotate in 90° increments.
Pan camera		Alt + middle mouse button	Hold down Ctrl and drag with the middle mouse button. Hold down Ctrl and press the arrow keys.
Roll view		Ctrl + Alt + Scroll	Hold down Alt and drag with the middle mouse button. Hold down Alt and press the left-right arrow keys.
Zoom in and out		Alt + right-click	Hold down Shift and drag with the middle mouse button. Press Z to zoom out or Shift + Z to zoom in.
Zoom to area		Home	Scroll wheel when Zoom about screen center is cleared.

Additional Enhancements

- Smart Control Bar for rendering profiles

A newly designed **Smart Control Bar** allows you to add frequently used render profiles to a convenient drop-down menu.



The controls allow you to:

Duplicate the current item	Copies the current render profile.
Add a new item	Lets you name the profile.
Render Profiles	Displays settings for a profile.
Rename the current item	Lets you rename a profile.
Lock the current item	Locks profile to prevent changes.

- Duration field to set the time of animations, turntables and sun studies
A new **Duration** field makes it easy to set the length of animations, turntables and sun studies. In previous releases, you had to divide the number of total frames in the animation by FPS (Frames Per Second) to calculate the duration of the movie.
- Improvements for setting size and resolution
Interface improvements make it easier to set the size and resolution of outputs. You can select from supported units of measure for the size, input a size and resolution, and see the resulting image and print size in the selected units.
- Improvements in the Import Settings dialog box
 - The layout of the **Part Grouping** mode and settings is clearer.
 - A new **Snap To Floor** check box automatically snaps the lowest point of your model to the floor of your Visualize scene.
 - When you import a project into Visualize from any supported file format, the scene automatically scales to fit your model. The **Auto-Size** check box has been removed, since this setting is no longer needed.
- Rearrangement of commands in menus
Commands, especially in the **File** and **Project** menus, have been rearranged and logically grouped. The redesign reduces menu complexity while still providing access to all commands that were previously available.
- Support for .jtc and CREO 3.0 files was added

www.solidworks.com

Dassault Systèmes SolidWorks Corp.
175 Wyman Street
Waltham, MA 02451
Phone: 1 800 693 9000
Outside the US: +1 781 810 5011
Email: generalinfo@solidworks.com

Europe Headquarters
Phone: +33 4 13 10 80 20
Email: infoeurope@solidworks.com

Asia/Pacific Headquarters
Phone: +65 6511 9188
Email: infoap@solidworks.com

Japan Headquarters
Phone: +81 3 6270 8700
Email: infojapan@solidworks.com

Latin America Headquarters
Phone: +55 11 3186 4150
Email: infoa@solidworks.com

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