



# WHAT'S NEW SOLIDWORKS 2023





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# Welcome to SOLIDWORKS 2023

This chapter includes the following topics:

- Top Enhancements
- Performance

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• For More Information



Model courtesy of Philipp Burgstaller, SOLIDWORKS 2022 Beta Splash Screen Prize Winner.

At SOLIDWORKS<sup>®</sup>, we know that you create great designs, and that your great designs get built. To streamline and accelerate your product development process from concept through

manufactured products, SOLIDWORKS 2023 contains new, user-driven enhancements focused on:

- **Working Smarter**. Reduce your workload when designing large assemblies and detailing drawings, and handle more complex electrical routing scenarios.
- **Working Faster**. Work with bills of materials (BOMs) more efficiently, improve the performance for searching and saving assemblies, and design large assemblies faster.
- **Working Together**. Help teams share and manage data more productively with a streamlined user interface, reduced latency, and improved upgrade performance in SOLIDWORKS PDM. Communicate changes to design data better with the ability to customize PDM notification templates.

## **Top Enhancements**

The top enhancements for SOLIDWORKS<sup>®</sup> 2023 provide improvements to existing products and innovative new functionality.

Parts and Features on page 25	<ul> <li>Coordinate Systems</li> <li>Dimension Support for the Move/Copy Body Command</li> <li>Single-Line Fonts for Wrap Features</li> </ul>
Sheet Metal on page 31	Sensors
Structure System and Weldments on page 35	<ul> <li>Corner Treatment to Similar Corners in Structure Systems</li> <li>Configuration-Specific Size of Weldment Members</li> </ul>
Assemblies on page 39	<ul> <li>Automatically Optimize Resolved Mode</li> <li>Repairing a Missing Mate Reference</li> <li>Additional End Conditions for Cut Features</li> </ul>
Detailing and Drawings on page 53	<ul><li>Overrides for Bills of Materials</li><li>Filters for Bill of Materials</li></ul>
SOLIDWORKS Inspection on page 111	<ul> <li>Autoballoons for PDF Files</li> <li>Balloon Sequence</li> <li>Balloon Drawing Sheets Separately</li> <li>Export Sheets as Separate 2D PDF Files</li> <li>Export FAI Reports to Separate Microsoft<sup>®</sup> Excel<sup>®</sup> Documents</li> <li>User Interface Redesign</li> </ul>
SOLIDWORKS MBD on page 121	<ul><li>Limiting Geometric Tolerances to a Standard</li><li>Dimension Extension Lines</li></ul>

#### Performance

SOLIDWORKS<sup>®</sup> 2023 improves the performance of specific tools and workflows.

Some of the highlights for performance and workflow improvements are:

# SOLIDWORKS Fundamentals

You can use the **Level of detail** system option to improve the performance when manipulating views.

To access this option, click **Tools** > **Options** > **System Options** > **Performance**.

Performance is improved for **Zoom In/Out**, **Pan**, and **Rotate** in draft quality HLR/HLV and wireframe modes of models.

## Section Views of Models

Starting and exiting the **Section View** for command is about five times faster than before. While you are in the **Section View** for command, adding or removing section planes is instantaneous.

## Assemblies

- Improved performance when searching with **Include sub-folders** selected.
- The save efficiency of assemblies is improved to avoid saving unchanged model data. This performance improvement is most noticeable in very large assemblies.

# SOLIDWORKS PDM

• SOLIDWORKS PDM 2023 makes significant performance improvements in the presence of high latency to SOLIDWORKS PDM servers.

The improvements might vary depending on the amount of latency, size of datasets, and operations.

- The following operations perform better when latency to the database server is high:
  - Check-in
  - Check-in with automatic transition
  - Undo Check-out
  - Interactive operations such as menus, shortcut menus, reference dialog boxes, SOLIDWORKS PDM File Explorer tabs
- The following operations perform better when latency to the archive server is high:
  - Add files
  - Change state on file modifications
  - Get files
  - Get latest version of files

• With performance improvements in the File Version Upgrade tool, you can upgrade SOLIDWORKS files in a SOLIDWORKS PDM vault faster.

If you are upgrading assembly files and you select **Overwrite** and **Latest Version** in the Version Settings page, the software skips upgrading and uploading references that are not the latest in the assembly's as-built version. This improves the performance of the upgrade process.

# Structure Systems and Weldments

Performance is improved when you create or edit a corner management feature. A corner gets trimmed when you select it under **Corners** in the Corner Management PropertyManager or in the graphics area.

Previously, the corners were trimmed as soon as you created or edited the corner management feature.

## For More Information

Use the following resources to learn about SOLIDWORKS:

What's New in PDF	This guide is available in PDF and HTML formats. Click:
	• ⑦ > What's New > PDF
	<ul> <li>⑦ &gt; What's New &gt; HTML</li> </ul>
Interactive What's New	In SOLIDWORKS, <sup>(1)</sup> appears next to new menu items and the titles of new or significantly changed PropertyManagers. Click <sup>(2)</sup> to display the topic in this guide that describes the enhancement.
	To enable Interactive What's New, click $\textcircled{O}$ > What's New > Interactive.
Online Help	Contains complete coverage of our products, including details about the user interface and examples.
SOLIDWORKS User Forum	Contains posts from the SOLIDWORKS user community on the ${\bf 3D}{\rm EXPERIENCE}^{\otimes}$ platform (login required).
Release Notes	Provides information about late changes to our products, including changes to the <i>What's New</i> book, online help, and other documentation.
Legal Notices	SOLIDWORKS Legal Notices are available online.

# Installation

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This chapter includes the following topics:

- Adding Serial Numbers for SolidNetWork License Installations
- Renewing and Reactivating Term SNL Licenses
- Load SOLIDWORKS Flow Simulation Modules
- Faster Startup for Borrowed Licenses

Adding Serial Numbers for SolidNetWork License Installations

SOLIDWORKS Product Activation Activate/Deactivate Your SOLIDWORKS Product	×
To activate your SOLIDWORKS product you must request a licen: The Activation Wizard will help you collect the information nece product.	se key from SOLIDWORKS. sssary to activate your
Serial#:	Add Select All Deselect All Remove
Add Serial Numbers	×

Adding serial numbers to licenses for all SOLIDWORKS applications and add-ins is much simpler for CAD administrators to handle.

You can use the Product Activation Wizard from the SOLIDWORKS SolidNetWork License Manager Server to add and activate serial numbers for perpetual and term licenses. Previously, serial numbers were entered when installing the License Manager Server. The new method makes it easier to add serial numbers to existing installations. You can also use the Activation Wizard to remove serial numbers that have not been activated. Renewing and Reactivating Term SNL Licenses

Serial					
All 🗸					
Associated					
Product	#Licenses	Version	Expires	Reactivation Date	
SOLIDWORKS Manage Professional Editor	5	30.0	Never	12-17-2022	
SOLIDWORKS Manage Professional Viewer	5	30.0	Never	12-17-2022	
SOLIDWORKS PDM Professional Processor License	25	30.0	Never	12-17-2022	
SOLIDWORKS Manage Professional Contributor	5	30.0	Never	12-17-2022	
SOLIDWORKS Composer Path Planning	1	30.0	Never	04-01-2022	
SOLIDWORKS Composer Professional	5	30.0	Never	12-17-2022	
SOLIDWORKS Composer Check (*)	1	30.0	04-01-2022	04-01-2022	
SOLIDWORKS Composer Enterprise Sync	1	30.0	Never	12-17-2022	
SOLIDWORKS PDM Standard CAD Editor	1	30.0	Never	12-17-2022	
SOLIDWORKS Standard	5	30.0	Never	12-17-2022	
SOLIDWORKS CAM Standard	5	30.0	12-31-2025	12-31-2025	

For term licenses, renewal and reactivation warnings display in all SOLIDWORKS add-ins, SOLIDWORKS PDM, and SOLIDWORKS Manage.

When users start these applications, if any term licenses are due to expire or require activation within 30 days, warnings notify the users and prompt them to contact their CAD administrator.

When you click **Reactivate** on the SolidNetWork License Manager **Server Administration** tab, the Reactivate Licenses dialog box highlights in red all license expiration and reactivation dates within 30 days. Asterisks mark licenses that require renewal. You can also filter and view licenses associated with a specific serial number.

# Load SOLIDWORKS Flow Simulation Modules



As a SolidNetWork license customer, you can control how to load the HVAC and Electronics Cooling add-in modules for Flow Simulation using two methods.

The first method lets you specify the default behavior for the add-ins when you load Flow Simulation. The option **Automatically load Flow Simulation add-in modules if available** is in the **SolidNetWork License Manager Client** > **License Order** tab. Clear this option if you do not have a need for the Flow Simulation add-ins. The HVAC and Electronics Cooling add-in modules do not load when Flow Simulation is added in with existing methods.

The second method lets you override default behavior and include the HVAC and Electronics

Cooling add-in modules when you load Flow Simulation. Click the Down arrow `` on **SOLIDWORKS Flow Simulation** (SOLIDWORKS<sup>®</sup> Add-ins toolbar) or **SOLIDWORKS Flow Simulation** with one or both of the add-modules to load.

Without these methods, the licenses for the add-in modules are consumed whether or not you need them and may be denied to other users on your network.

Faster Startup for Borrowed Licenses



If you are working remotely and your SOLIDWORKS apps are slow to open, you can make them faster by blocking license server queries and using only borrowed licenses.

License server queries are slow in some environments. Your apps will start faster if you block these queries, but will only use the licenses you have borrowed.

To block license server queries and use only borrowed licenses, click **Start** > **SOLIDWORKS Tools 2023** > **SolidNetWork License Manager Client** > **License Borrowing** > **Only use borrowed licenses for faster startup** and restart your apps.

# 3

# SOLIDWORKS Fundamentals

This chapter includes the following topics:

- Application Programming Interface
- Changes to System Options and Document Properties
- Ellipses
- Performance Improvement When Manipulating Views

#### **Application Programming Interface**

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

#### Support

There is API support for:

- Mate controller features in assemblies
- Simple, two-member, and complex corner treatments in structure systems

### Redesign

Sheet metal sketched bend features have been redesigned.

#### **Other Enhancements**

- Get whether an assembly configuration is defeatured
- Get and set whether to use gauge table values in a sheet metal sketched bend feature
- Get whether to override a preset bend angle in a sheet metal sketched bend feature
- Get and set whether to symmetrically thicken a bidirectional base flange feature
- Get and set whether to symmetrically thicken a bidirectional lofted bends feature
- Convert a solid body to a mesh body
- Create a surface body from a mesh body
- Create a surface body from a graphics body

# Changes to System Options and Document Properties

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

# System Options

Option	Description	Access
Drawings, Modified Cells (BOM)	Changes the default color of overridden cells.	Colors
Sketch, Exploded	Specifies a color for an explode line.	Colors
Automatically optimize resolved mode, hide lightweight mode	Loads component data as resolved when you open an assembly.	Performance
Manually manage resolved and lightweight modes	Controls when a component loads in lightweight or resolved mode.	Performance
Load component lightweight	Renamed from Automatically load components lightweight.	Performance
Export assembly components as separate STEP files (recommended for large assemblies)	Exports assemblies as atomic STEP files.	Export
Export appearances	Exports file's appearances with reduced performance. Clear the option to skip exporting appearances but make the performance faster.	Export

# **Document Properties**

Option	Description	Access
Enable/Disable options per standard	Includes all geometric tolerance symbols. To limit symbols to a standard, select the option.	Annotations > Geometric Tolerances
Connect dimension extension lines to model	Completes connection of dimension extension lines to model.	DimXpert > Display Options
See through transparent components in HLR/HLV	(Available for <b>High quality</b> drawing views only). Shows edges behind transparent components with solid lines.	Drawings > Detailing

Option	Description	Access
Default Bill of Materials part number for new configurations	Specifies a value for the default BOM part number. Options are <b>Document Name</b> or <b>Configuration Name</b> .	Configurations

# Ellipses



When you sketch an ellipse, you can apply construction geometry to the major and minor axes.

In the Ellipse PropertyManager, under **Options**, select **Add construction lines**.

## Performance Improvement When Manipulating Views

You can use the **Level of detail** system option to improve the performance when manipulating views.

To access this option, click **Tools** > **Options** > **System Options** > **Performance**.

Performance is improved for **Zoom In/Out**, **Pan**, and **Rotate** in draft quality HLR/HLV and wireframe modes of models.

# User Interface

This chapter includes the following topics:

- Comments Enhancements
- Enforcing Legal File Names
- Restoring Default Settings
- Component Name Display

### **Comments Enhancements**

Fillet1			
В <u>I</u> <u>U</u>			
4/19/2022 8:54:53 /	Description		
Show at start	Save and Close Cancel	Help	

Comments are enhanced to improve usability.

- You can choose different background colors and add bold  $\,B$  , italics  $\,I$  , and underline  $\,\underline{U}$  to text.
- Click Options in the Background Colors tab to open the Tools > Options > Document Properties > Model Display dialog box where you can specify the Comment background colors. You can customize the names or reset all names back to the default. You cannot customize the colors.



- Select **Show at start** in various places to show the comments in the View All Comments dialog box when you reopen the file.
- In the View All Comments dialog box, in the **Show at start** column, you can select the comments to show when you open the file. In the **Show** menu, you can specify the comments to show at the start or based on a color. Click **Apply** to save changes to the **Show at start** status of comments.



• When you right-click a comment, a context toolbar and shortcut menu appear so you can directly edit features when you read the comments.



 Comment tooltips show the text <image> to indicate that the full comment contains an image. The tooltip name matches the name from the **Comment background** colors section in the Model Display dialog box.

Fillet1 ▶ @ Cut-Extr	Description Comment: 4/19/2022 8:54:53 AM JEU: Is this fillet large enough? 4/19/2022 9:33:21 AM JEU:	
	<image/>	s

- Images in comments retain their current aspect ratio and the display quality is improved.
- When you import images into comments, in the Open dialog box, the All files
   (\*.bmp,\*.jpg,\*.tif,\*.tiff,\*.png) option is the default to display all acceptable image
   types.
- In the FeatureManager<sup>®</sup> design tree, comment indicators appear by default next to features that have comments. The color circle matches the color of the comment.



To control the display of the comment indicators, in the FeatureManager design tree, right-click the part node and click **Tree Display** > **Show Comment Indicator**.



# Enforcing Legal File Names



To ensure compatibility with the **3D**EXPERIENCE platform, you cannot save new SOLIDWORKS documents with preceding or trailing spaces in their names. The software automatically removes the preceding or trailing spaces when you save the file. Spaces in the middle of the file name are supported.

This enforcement applies in these cases:

- Save As dialog box, including the Advanced option
- Renaming parts and assemblies in the FeatureManager design tree
- Mirror Components PropertyManager
- Split PropertyManager and Save Bodies PropertyManager
- Pack and Go

# **Restoring Default Settings**



You can more easily restore the factory default settings in SOLIDWORKS<sup>®</sup>.

SOLIDWORKS Settings Wizard

The Copy Settings Wizard is renamed to the SOLIDWORKS Settings Wizard that has an improved and simplified user interface with three options:

Save Settings	Saves SOLIDWORKS settings to a file. You choose the location for the settings file and the settings to save.
<b>Restore Settings</b>	Restores SOLIDWORKS settings from a file.
Reset Settings	Resets all custom settings and returns SOLIDWORKS to the factory defaults. You have the option to save all custom settings to a file before resetting SOLIDWORKS to the factory defaults.

Customize Dialog Box

- The Reset to Defaults button appears at the bottom of each tab in the Customize dialog box so you can customize the settings to reset.
- The Customization tab is removed.

#### User Interface

Customize	2					
Toolbars	Shortcut Bars	Commands	Menus	Keyboard	Mouse Gestures	Customization
Short	cut customizatio	n –				
	Show All	]				
Re	set to Defaults	╝┥┥			2022	
Menu	customization -					
	Show All					
Re	set to Defaults					

The options from the Customization tab are reorganized as follows:

Categories:	Commands:	
File Edit	(Separator)	Rename
View Insert	Open Open Recent	Remove
Tools Window	SOLIDWORKS ×	
Help Others Search Macros	Reset User Interface Customizations	
	→ Reset SOLIDWORKS menus Resets all customizations of all menus to the factory defaults.	2023
	→ Reset shortcut menus Resets all shortcut (right-click) menus to the factory defaults.	
	→ Reset all customizations Resets all customizations on all tabs to the factory defaults.	

• Shortcut customization:

Show All

Removed because this was rarely used. The preferred method is to keep the menu small and click s to expand the list

**Reset to Default** Moved to **Menus tab** > **Reset shortcut menus**.

• Menu customization:

Show All	Removed because there was very little difference between
	this command and <b>Reset to Default</b> for this customization.

**Reset to Default** Moved to **Menus tab** > **Reset SOLIDWORKS menus**.

Customizations and Upgrades

When you upgrade to a new version of SOLIDWORKS, the software keeps all the customizations and also shows any new tabs, menus, toolbars, etc. that were added to the software. Previously, warning dialogs prompted you to make choices that might have led to lost settings.

## Component Name Display

S Model View	(?)
× ×	۲
Message	^
Select a part or assembly from which to create the view, then click Next.	
Part/Assembly to Insert	^
Open documents:	
wormgear_assembly	

When you specify display options for component names in a drawing, the names in the PropertyManager and View Palette match those in the FeatureManager design tree.

#### To specify display options for component names:

- 1. In the FeatureManager design tree, right-click the drawing name and click **Tree Display** > **Component Name and Description**.
- 2. (Optional.) Under **Primary**, specify an option:
  - Component Name
  - Component Description

Based on the selection under **Primary** display options:

- The Model View and Standard 3 View PropertyManagers update the **Open documents** field.
- The View Palette updates. The View Palette displays the part or assembly icon and the file name.

# 5

# Parts and Features

This chapter includes the following topics:

- Allowance of Library Feature Errors
- Coordinate Systems
- Dimension Support for the Move/Copy Body Command
- Hybrid Mesh BREP Modeling Improvements
- Section View Performance Improvements
- Section View through Axis
- Single-Line Fonts for Wrap Features

Allowance of Library Feature Errors

SOLIDWORKS	×	
Unable to create library feature with selected references.		
c	К	
	SOLIDWORKS The library feature has error Click Yes to keep this result	× ors with these selections. t or No to modify selections.
	Don't show again	Yes No
2022	20	23

When you insert a library feature into a model and get error messages, you can keep the results to view and manually repair the errors.

In the warning message, click **Yes** to keep the results. In the FeatureManager<sup>®</sup> design

tree, you can manually repair the library feature or click **Undo** to remove it. If you click **No** in the warning message, in the Library Feature PropertyManager, you can modify

the selections or click  $\,\,{}^{\bigstar}\,$  to restore the FeatureManager design tree without the library feature.

Previously, the warning message blocked you from inserting library features that generated errors.

# Coordinate Systems

Select Other		x
🕕 Coordinate S	ystem1@[bishop1_black]	
Coordinate S	ystem1\XY Plane@[bishop1_black]	
Coordinate S	ystem1\XZ Plane@[bishop1_black]	
Coordinate S والمحكوم	iystem1\X Axis@[bishop1_black]	
Y Mardinate	System	
	5 System 1	

You can reference coordinate system elements from additional entities, features, and commands, including the **Select Other** command.

Item	Description
3D sketches	Create sketch relations between 3D sketch entities and coordinate system axes, planes, and origin.
2D sketch dimensions	Create 2D sketch dimensions that reference coordinate system axes, planes, and origin.
<ul> <li>Mirror F features and bodies</li> <li>Mirror Components F</li> <li>Insert &gt; Mirror Part</li> </ul>	Select a coordinate system plane as the mirror plane.
Reference <b>Plane</b> 다	Select a coordinate system axis to create a new plane perpendicular to that axis. Use a vertex or point from the model to position the plane.

Item	Description
Select Other 🗇	Select coordinate system axes, planes, and origin.

Dimension Support for the Move/Copy Body Command



When you use the **Move/Copy Body** command, you can control the **Translate** and **Rotate** values with equations. You can add these equations to a design table and control them there. You can double-click the dimensions in the graphics area and specify them using the **Modify** dialog box.

Under **Rotate**, you can select **Euler's angular rotation** and specify values for **Yaw** (side-to-side), **Pitch** (up-down), and **Roll** (twist). This method delivers the same resulting rotation regardless of the order in which you specify these values.

If models have configurations, you can specify the configurations to which the dimensions apply for **Translate** and for **Rotate** when you select **Euler method of angular dimensions**.



## Hybrid Mesh BREP Modeling Improvements

Hybrid mesh BREP modeling is improved to support more SOLIDWORKS<sup>®</sup> BREP features that you can combine with mesh BREP geometry.

Hybrid mesh BREP modeling now supports features such as boss and cut sweeps, lofts, and boundaries. For a complete list of supported features, see *SOLIDWORKS Online Help*: *List of Features Supporting Hybrid BREP Bodies*.

## Section View Performance Improvements

Performance is improved when you work with section views in models.

Starting and exiting the **Section View** for command is about five times faster than before.

While you are in the **Section View** for command, adding or removing section planes is instantaneous.

# Section View through Axis



You can define a section view through an axis without constraining it to a secondary point, such as a vertex. Previously, a secondary selection was mandatory. When you click **Plane** 

about axis  $^{(5)}$ , you can click **Reverse Section Direction**  $^{1}$  to flip the section view across the created plane.

Wrap1 Vrap Type Decision

Single-Line Fonts for Wrap Features

You can create wrap features from sketches that use single-line fonts when you select **Scribe** for **Wrap Type** in the Wrap PropertyManager. Previously, wrap features did not support single-line fonts.

Single-line fonts are also called stick fonts. This functionality is particularly helpful for users who want to engrave stick fonts on parts.

If text or characters do not display properly, you may be using an inauthentic single-line font. To check the font, dissolve the sketch text and review the text entities.

# 6

# **Sheet Metal**

This chapter includes the following topics:

- Gauge Values in Cut List Properties
- Sensors
- Sensor PropertyManager
- Symmetrical Thickness

### Gauge Values in Cut List Properties



If you use gauge tables to define the thickness of sheet metal parts, the gauge number appears in the Cut-List Properties dialog box. In the associated drawing, the gauge value is available for annotations and cut lists.

In the Cut-List Properties dialog box, the **Evaluated Value** for **Sheet Metal Gauge** is **Gauge Number GA**. For example, if you use a 3 Gauge in a sheet metal part, the **Evaluated Value** is **3 GA**.

In drawings, you can link an annotation to the **Sheet Metal Gauge** property and you include the property in a cut list.

#### To link annotations to gauge values:

- 1. In a drawing of a sheet metal part, click an annotation, such as **Note**  $\mathbb{A}$ .
- 2. In the PropertyManager, click Link to Property 🗐.
- 3. In the Link to property dialog box, in **Property name**, select **Sheet Metal Gauge**.

#### To include gauge value properties in cut lists:

- 1. Right-click a flat pattern view and select **Annotations** > **Cut List Properties**.
- 2. Click in the sheet to place the cut list properties.

#### Sensors



You can use sensors to alert you when sheet metal models deviate from the limits specified for a bounding box. In multibody parts, you can create sensors for individual bodies.

During the design process, if the model exceeds the bounding box parameters, an alert appears in the FeatureManager<sup>®</sup> design tree. You can double-click an alert to see the values in the Cut-List Properties dialog box.

## Sensor PropertyManager

Updates to the Sensor PropertyManager include support for sheet metal models.

## Message

Displays a message if you do not select **Create Cut Lists Automatically** and **Update Automatically**. Sheet metal bounding box sensors depend on up-to-date cut list

properties. In the FeatureManager design tree, right-click **Cut list** <sup>1</sup>/<sub>2</sub> and select **Create Cut Lists Automatically** and **Update Automatically** to generate automatic alerts.

# Sensor Type

Sheet Metal Bounding Box Properties	Specifies the smallest rectangle in which the flat pattern can fit.
---	---

# Properties

2	Sheet Metal Bounding Box Length	Specifies the longest side of the bounding box. For multibody parts, this is the sum of the bounding box length for all bodies in the part. For assemblies, this is the sum of the bounding box length for all parts in the assembly.
	Sheet Metal Bounding Box Width	Specifies the shortest side of the bounding box. For multibody parts, this is the sum of the bounding box width for all bodies in the part.
		For assemblies, this is the sum of the bounding box width for all parts in the assembly.
	Sheet Metal Bounding Box Area	Sheet Metal Bounding Box Length $\boldsymbol{x}$ Sheet Metal Bounding Box Width
		For multibody parts, this is the sum of the bounding box area for all bodies in the part.
		For assemblies, this is the sum of the bounding box area for all parts in the assembly.
She	Sheet Metal Bounding Box	Specifies the area of the flat pattern excluding through cut-outs.
	Area Blank	For multibody parts, this is the sum of the bounding box area blank for all bodies in the part.
		For assemblies, this is the sum of the bounding box area blank for all parts in the assembly.
<b>%</b>	Entities to Monitor	Specifies the individual body or bodies to which the sensor applies.

# Symmetrical Thickness



When you create a sheet metal part as a base flange or lofted bend (with the **Bent Manufacturing Method**), you can specify a symmetrical thickness that adds an equal amount of material to both sides of the sketch.

Symmetrical thickness helps you create sheet metal parts from sketches to help achieve equal bend radii for upward and downward bends. In the image above, **Symmetric** is cleared for the example on the left and selected for the example on the right.

In the Base Flange or Lofted Bends PropertyManager, under **Sheet Metal Parameters**, select **Symmetric**.

# Structure System and Weldments

This chapter includes the following topics:

- Corner Treatment to Similar Corners in Structure Systems
- Pattern Connection Element PropertyManager
- Configuration-Specific Size of Weldment Members
- Structure Systems and Weldments Performance Improvement

Corner Treatment to Similar Corners in Structure Systems

F Corner Management 🛛 🕐 🕐	
✓ ×	
Simple Two member Complex	
Corners ^	
Corner Group6	
Similar Corners	
Group Similar Corners	
ក្រៃ Corner2 ^	e e
Corner1 🗸	

You can use **Group Similar Corners** to group similar corners, apply a corner treatment to the group, and edit the groups as required.

This is helpful for large structure system files that have multiple corners.

- 1. Open the structure system file.
- 2. In the FeatureManager<sup>®</sup> design tree, click **Corner Management** and select **Edit Feature**.
- 3. In the Corner Management PropertyManager, under Corners, select a corner.
- 4. Click Group Similar Corners.

The similar corners are grouped together, appear as **Corner Group xx** under **Corners** in the PropertyManager, and highlighted with arrows in the graphics area.

5. Apply corner treatment to the group.

The FeatureManager design tree groups the similar corners as **Corner Group** *XX*. In the Corner Management PropertyManager, you can edit a group of similar corners and remove a corner from the group. You can add this corner back to the group of similar corners, if required.

• Right-click a corner of the group and select one of the following:

Remove from Corner group	Removes the selected corner from the group of similar corners
	Right-click the removed corner and select <b>Include back to Corner group</b> .
Zoom to Selection	Zooms in the selected corner
Clear Selections	Degroups the corners
	To degroup the corners, you can also right-click the group and select <b>Dissolve Corner group</b> .
Customize Menu	Lets you select the context menu

You can use similar corner treatment for simple, two member, and complex corners.
## Pattern Connection Element PropertyManager

Pattern Connection Element	?
Message	~
Selected Connection Element	^
Connection Element Gusset Plate2	
0	
Identify Similar Corners	^
☐ Ignore corners that have connection elements	I
Corner28	^
Corner36	~
Instances To Skip	^

In the Pattern Connection Element PropertyManager, you can insert a connection element that is at a corner to similar corners.

To access this PropertyManager, click **Insert** > **Structure System** > **Pattern Connection**.

## Pattern Connection Element PropertyManager

Select the connection element in the graphics area. Similar corners are highlighted with arrows and previews of connection elements appear at the corners.

Selected Connection Element	Displays the name of the selected connection element.
Identify Similar Corners	Lists similar corners on which you can insert the connection element. You can right-click a corner and select <b>Skip</b> <b>Instance</b> to exclude a connection element.
Ignore corners that have connection elements	Excludes corners with connection elements.
Instances to Skip	Lists the corners that you excluded using <b>Skip Instance</b> .

## Configuration-Specific Size of Weldment Members

	Profile Alignment:
<ul> <li>Structure System1</li> <li>Corner Management1</li> <li>Simple Corner Group1</li> <li>Corner Group6</li> <li>Two member Corner Group7</li> <li>Corner Group7</li> <li>Corner Group7</li> <li>Corner Group7</li> <li>Corner Group7</li> </ul>	Configurations Offiguration Offiguration Offiguration Offigurations Offigurations Offigurations Default <as machined=""> Default<as welded=""> Next As Welded&gt; Next A</as></as>

If you select a set of weldment members of the same size and type, in the Profile tab, you can change the size of the members for specific configurations.

#### Using Zoom to Selection for Corners

If the structural member has multiple corners, you can zoom to a corner from the FeatureManager design tree or the Corner Management PropertyManager using **Zoom to Selection**.

Configuring the Size for Configurations in Structure Systems

#### To configure the size for configurations of structure systems:

- 1. Edit the structure system from the FeatureManager design tree.
- 2. Select the members in the graphics area.
- 3. Optional: To edit a specific member, in the graphics area, right-click the member, and select **Edit Feature**.
- 4. In Profile tab, change the size.
- 5. Under **Configurations**, select specific configuration and click **OK**.

#### Structure Systems and Weldments Performance Improvement

Performance is improved when you create or edit a corner management feature.

A corner gets trimmed when you select it from the **Corners** group in the Corner Management PropertyManager or in the graphics area.

Previously, corners were trimmed when you created or edited the corner management feature.

# Assemblies

This chapter includes the following topics:

- Automatically Optimize Resolved Mode
- Repairing a Missing Mate Reference
- Additional End Conditions for Cut Features
- Previewing a Replacement Component
- Specifying a Color for an Explode Line
- Selecting a Configuration for Skipped Instances
- Specifying a Default BOM Part Number
- Magnetic Mates
- Overriding Calculated Values in a Design Table
- Assembly Visualization
- Configuration Publisher: Updating a Multiple-Configuration Model
- Propagating Assembly Features to Parts Blocked for Toolbox Parts
- Assembly Performance Improvements

Automatically Optimize Resolved Mode

Assembly loading Automatically optimize resolved mode, hide Manually manage resolved and lightweight	e lightweight mode modes	
<ul> <li>Load component lightweight</li> <li>Always resolve subassemblies</li> </ul>		
Check out-of-date lightweight components:	Don't Check	$\sim$
Resolve lightweight components:	Prompt	$\sim$
Rebuild assembly on load:	Prompt	$\sim$

You can optimize resolved mode automatically. Using this system option improves performance by selectively using lightweight technology when you load components in resolved mode.

Options for lightweight and resolved modes are hidden as the components load as resolved. Lightweight states do not appear in the FeatureManager<sup>®</sup> design tree.

You can manually manage when to resolve lightweight components by selecting **Manually manage resolved and lightweight modes**.

Automatically load components lightweight is renamed to Load component lightweight.

To automatically optimize resolved mode:

- 1. Click Tools > Options > System Options > Performance.
- 2. Under Assembly loading, select Automatically optimize resolved mode, hide lightweight mode.

Repairing a Missing Mate Reference

Concentricio
<ul> <li>One of the entities of referenced features of this mate is suppressed, invalid, or no longer</li> <li>✓ Coincic present.</li> <li>✓ Coincic Feature (Concentric10)</li> </ul>

You can repair missing mate references for references to faces, edges, planes, axes, and points.

SOLIDWORKS<sup>®</sup> repairs the missing reference for a component by selecting a reference on the component that is in the same location and orientation to the missing reference.

For models that contain multiple missing mate references, right-click the **Mates**  $\mathbb{P}$  folder and click **Auto Repair**  $\mathbb{P}$ .

#### To repair a missing mate reference:

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

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

## Additional End Conditions for Cut Features

© ~	Cut-Extrude	
Dire	tion 1	
$\mathbf{Z}$	Blind	$\sim$
_	Blind	
1	Through All	
<u>ا</u>	Up To Next	
$\diamond$	Up To Vertex	
- 1	Up To Surface	
- 1	Offset From Surface	
	Up To Body	
	Mid Plane	

More end conditions are available for cut features in assemblies.

You can select these options for **Direction 1** and **Direction 2**:

- Up To Next
- Up To Vertex
- Up To Surface
- Offset From Surface
- Up To Body

#### To select an end condition for a cut feature:

- 1. Open a model and create a sketch.
- 2. Click Insert > Assembly Feature > Cut > Extrude.
- 3. In **Direction 1**, select an end condition.

## Previewing a Replacement Component



You can preview a replacement component in the Replace PropertyManager. You can specify the scope of the replacement.

In the PropertyManager, **Instances to replace** replaces **All instances**. To specify the scope of the replacement, select an option for **Instances to replace**:

Only selected	Replaces only the selected instance.
All in same parent assembly	Replaces all instances of the selected component that are in the same assembly level. If a selected component is in the top-level assembly, only instances in the top level are replaced. If a selected component is in a subassembly, only instances in the subassembly are replaced.
All	All instances of the component are replaced in the assembly and all subassemblies.

#### To preview a replacement component:

- 1. Open a model and click **File** > **Replace**.
- 2. In **Replace these component(s)** (4), select a component.
- 3. For **Instances to replace**, select an option.
- 4. In the PropertyManager, select a component for **With this one**

The selected component appears under **Thumbnail Preview**.

## Specifying a Color for an Explode Line



You can use a system option to specify an explode color. You can change the color of explode lines in an exploded view.

#### To specify a color for an explode line:

- 1. Click **Options** > **System Options** > **Colors**.
- 2. Under Color scheme settings, select Sketch, Exploded.

#### To change the sketch color in an exploded view:

- 1. Open a model that has an exploded view.
- 2. In the ConfigurationManager 🛱 tab, click 📩 to expand **Exploded View** 💞.
- 3. Right-click **3DExplode** <sup>(1)</sup> and click **Sketch Color** <sup>(2)</sup>.
- 4. In the Sketch/Curve Color PropertyManager, select a color.

## Selecting a Configuration for Skipped Instances

	Linear Pattern	
	Instances to Override	
	Skipped instances	
ſ	(1, 3)	
	This Configuration	
	All Configurations	
	B Specify Configurations	

## You can select This Configuration P, All Configurations 🖁 , or Specify

**Configurations** <sup>‡</sup> for skipped instances in assembly component patterns.

The configuration option is available for these assembly component patterns:

- Linear Component Pattern
- Circular Component Pattern
- Pattern Driven Component Pattern
- Sketch Driven Component Pattern
- Curve Driven Component Pattern

In a design table, you can use <code>\$skip@<type of pattern><number></code> to specify skipped instances. For example, <code>\$skip@Lpattern1</code>. If there is more than one skipped instance, use a semicolon to separate values. For example: 10,1;10,2;.

#### To select a configuration for skipped instances:

- 1. Open a model with multiple configurations and edit a pattern feature.
- 2. In the PropertyManager, select an instance to skip and select **This Configuration**

🏴 , All Configurations 🛱 , or Specify Configurations 🛱 .

## Specifying a Default BOM Part Number



In a model, you can select **Document Name** or **Configuration Name** as the default Bill of Materials (BOM) part number for new configurations.

This option does not apply to derived configurations.

The selected default value shows in the Configuration Properties PropertyManager under **Bill of Materials Options** for **Part number displayed when used in a bill of materials**.

#### To specify a default BOM part number:

- 1. Open a model and click **Tools** > **Options** > **Document Properties**.
- 2. Click Configurations.
- 3. Under **Default Bill of Materials part number for new configurations**, select **Document Name** or **Configuration Name**.

## **Magnetic Mates**

▼ 🕅 Mates	
🥵 Magneti 🛛 💵	
C Magneti	
C Magneti 🎤	
C Magnetic4	
C Magneti Feature (Magnetic4)	
<ul> <li>References</li> </ul>	
🚑 Ground Pla	
G Connector1 ↓	
Connector2	
Connectora Invert Selection	

You can suppress magnetic mates and connection points.

When you suppress a component, magnetic mates associated with the component are suppressed. Suppressed connection points are not available when you create a magnetic mate.

When you have unused connection points and move a component chain, magnetic mates are no longer created between components within the chain. When you move a component, the color for all connection points is magenta.

#### To suppress a magnetic mate:

- 1. Open a model that contains magnetic mates.
- 2. In the FeatureManager design tree, expand the **Mates 1** folder.
- 3. Right-click **Magnetic Mate**  $\subseteq$  and click **Suppress**  $\downarrow$

#### To suppress a connection point:

- 1. Open a model that contains magnetic mates.
- 2. In the FeatureManager design tree, expand a component and then expand **Published References Processor**.
- 3. Right-click **Connector**  $\blacksquare$  and click **Suppress**  $\downarrow$ .

## Overriding Calculated Values in a Design Table



When you close a design table that contains manually entered values for SW-MASS and SW-COG, SOLIDWORKS prompts you to use either the manually entered values or the values calculated from the model geometry.

The prompt provides these options:

Use the mass value specified in the Design Table	Enables <b>Override mass</b> or <b>Override center of mass</b> and assigns the manually entered values to the mass properties.
Use the mass value calculated from the model geometry	Deletes the \$SW-MASS column or the \$SW-COG column that has the manually entered values. The model uses values calculated from the model geometry.

#### To override calculated values in a design table:

- 1. Open a model that has a design table with a *\$SW-MASS* column or a *\$SW-COG* column.
- 2. Clear the override properties:
  - a. Click **Tools** > **Evaluate** > **Mass Properties**.
  - b. Click Override Mass Properties.



- c. In the Override Mass Properties dialog box:
  - For \$SW-MASS, clear Override mass.
  - For \$SW-COG, clear Override center of mass.

Override Mass Properties		
Properties for: rubber feet.SLDPRT		
Override mass: 0.51grams		
Override center of mass:		

If these override options are enabled, the prompt does not show and the model is saved with the manually entered values.

- 3. In the Configuration Manager  $\mathbb{B}$  tab, expand **Tables**  $\blacksquare$ .
- 4. Right-click Excel Design Table 🔀 and click Edit Table.

	🔌 🖹 🖪
	Configurations
<ul> <li>W rubber feet Confi</li> <li>Tables</li> </ul>	iguration(s)
Excel Desi	🗑 Edit Feature
	Edit Table
	Edit Table in New Window

- 5. In the table, manually enter values for \$SW-MASS or \$SW-COG.
- 6. Close the design table.
- 7. When prompted, select **Use the mass value specified in the Design Table** to override the calculated values with the manually entered values.

## Assembly Visualization

File Name	Quan	Mass	•
base plate	1	83.39	
🍕 drive sh	1	0.13	
drive sh-	1	1.96	

In lightweight mode, SOLIDWORKS calculates more Assembly Visualization columns without setting the lightweight components to resolved.

If SOLIDWORKS cannot calculate a column value for a component and you select **Manually manage resolved and lightweight modes** and **Prompt** for **Resolve lightweight components**, SOLIDWORKS prompts you to resolve the component.

To modify these options, click **Options** > **System Options** > **Performance**.

Assembly loading					
O Automatically optimize resolved mode, hide lightweight mode					
Manually manage resolved and lightweight modes					
Load component lightweight					
Check out-of-date lightweight components:	Don't Check $\sim$				
Resolve lightweight components:	Prompt 🗸 🗸				

In the Custom Column dialog box, the columns display in alphabetical order. The Assembly Visualization rollback bar increases in size when hovering over the bar. The font color is black for all column values. You can add more than seven columns.

#### To view the Assembly Visualization improvements:

- 1. Open a model in lightweight mode.
- 2. Click Tools > Evaluate > Assembly Visualization.

All column values display in a black font color and the rollback bar is wider when you hover over the bar. For calculated properties like **Mass**, the calculated values display instead of 0.00.

- 3. To open the Custom Column dialog box, in the Assembly Visualization panel, click the arrow ) to the right of the column headers and click **More**.
- 4. In the dialog box, click **Properties**.

The columns display in alphabetical order.

Configuration Publisher: Updating a Multiple-Configuration Model



For models that have multiple configurations created with a custom PropertyManager,

the **PropertyManager** icon changes to **III** after you manually add, delete, or rename a configuration. **Update** appears when you right-click **PropertyManager**.

#### To update a multiple-configuration model:

1. Open a model with multiple configurations created with a custom PropertyManager.

Use Configuration Publisher to create a custom PropertyManager.

2. In the ConfigurationManager  $\mathbb{R}$  tab, add a configuration.

The PropertyManager icon changes to **III**.

3. Right-click **PropertyManager** is and click **Update**.

After clicking **Update**, the **Excel design table** vopens, updates, and closes. The PropertyManager icon changes to **E**.

Propagating Assembly Features to Parts Blocked for Toolbox Parts



Assembly features cannot be propagated to Toolbox parts.

In an assembly feature PropertyManager, SOLIDWORKS displays a warning and removes

any Toolbox parts from **Components to Affect** <sup>4</sup> when you have **Selected components** and **Propagate features to parts** selected.

When you select **All components** and **Propagate features to parts** under **Feature Scope** and the model contains Toolbox parts, the warning does not appear. The assembly features are not propagated to the Toolbox parts.

When **Propagate features to parts** is selected, you cannot select a Toolbox part.

For assembly features like fillets and chamfers that do not have the **All components** and **Selected components** options, the warning does not appear and the assembly features are not propagated to Toolbox parts. You cannot select a Toolbox part for a **Hole Series** as this assembly feature is always propagated to parts.

#### To view the warning:

- 1. Open a model that contains a Toolbox part.
- 2. Select a sketch and click **Insert** > **Assembly Feature** > **Cut** > **Extrude**.
- 3. In the PropertyManager, under Feature Scope, clear Auto-select.
- 4. For **Components to Affect**  $\clubsuit$ , select a Toolbox part.
- 5. Select **Propagate features to parts**.

SOLIDWORKS removes the Toolbox part from **Components to Affect**  $^{4}$  and the warning appears in the graphics area.

## Assembly Performance Improvements

Assembly performance is improved when:

- Searching with **Include sub-folders** selected.
  - To select this option, click **Tools** > **Options** > **External References** > **Reference Documents in specified in File Locations** > **Include sub-folders**.
- Saving assemblies as only changed model data is saved. This improvement is noticeable in very large assemblies.

# 9

# **Detailing and Drawings**

This chapter includes the following topics:

- Overrides for Bills of Materials
- Displaying Transparent Models
- Limiting Geometric Tolerances to a Standard
- Filtering Bill of Materials Columns
- Component Name Display
- Detached Drawings

Overrides for Bills of Materials

<b>□</b> ⊕		= A <sup>0</sup> C 🖻 🗄 📲 🐙 ∑ 0.02in <> 0	.04in 🗘 🖷 🐺 🏛 🗜 🙆
1	ITEM NO.	PART NUMBER	QTY.
2	1	emitter	1
3	2	grip	1
4	3	grip-focuser-lens- spinring	1
5	4	grip-focuser-lens-lens	1
6	5	grip-focuser-lens- innerring	1

When you override a value in a bill of materials (BOM) using **Break Link**, the value turns blue.

You can:

- Change the highlight color (default is blue) when you override values using Break Link. Click Tools > Options > System Options > Colors, and specify the color for Drawings, Modified Cells (BOM).
- Restore overridden values to their original values. Right-click a row, column, or the entire BOM and click **Restore Original Value**.

Prevent unwanted changes to BOMs. Click a table cell, row, or column, and click the lock in the context toolbar. To unlock, click the lock in the context toolbar.

## **Displaying Transparent Models**



You can display transparent models in drawings with Hidden Lines Removed (HLR)  $\square$ 

and **Hidden Lines Visible** (HLV) <sup>©</sup> modes.

You can see through transparent bodies in HLR and HLV display states including components in assemblies and bodies in multibody parts.

Available for **High quality** drawing views only.

#### To display transparent models:

- 1. Click Tools > Options > Document Properties > Detailing .
- 2. Select See through transparent components in HLR/HLV.

olerance	? ×	Tolerance	? >
Range Offset Zone Fosture Characteristic Material Condition Symbol	Filter • G S SW CW RG RS OB OH OD CB CH CD AB AH AD F H - Space x Associated Feature C G N T X Derived Feature	Range Offset Zone Feature Characteristic Material Condition Symbol	Filter • G S SW CW RG RS OB OH OD CB CH CD AB AH AD F H - Space × Associated Feature © © ® ® © ® Derived Feature
Enable all combinat	ions and symbols Add Datum Done	Enable all combinat	cions and symbols Add Datum Done

## Limiting Geometric Tolerances to a Standard

You can choose all symbols for geometric tolerances or limit symbols to a standard.

The image on the left has **Enable/Disable options per standard** selected. For example, if you select the ISO standard and select **Enable/Disable options per standard**, you limit the symbols and values to ISO standards.

The image on the right has **Enable/Disable options per standard** cleared. All symbols and values are available regardless of the tolerancing standard.

#### To limit geometric tolerances to a standard:

- 1. In a drawing, click Tools > Options > Document Properties > Annotations > Geometric Tolerances.
- 2. Under Base geometric tolerance standard, select:
  - a. A standard.
  - b. Enable/Disable options per standard.

	4	2	A	-T	B 🔹	D -
	1		ITEM NC	Cu	stom Filter	QTY.
3	2	2	1	Se	arch	1
$ \prec $	3	3	2		(Select All)	2
	4		3		Chassis Computer Chassis	1
	5	;	4		Fan PCB	1
	6		5		Voltage Switch	1
	7	,	6		OK Cancel	1
j/					Cancer	

Filtering Bill of Materials Columns

Filtering a bill of materials (BOM) lets you focus on what you need to see.

When you filter a BOM:

- You can filter a range of data based on your criteria. When you hover over the BOM column headers, there is an arrow in each column.
- Only the balloons for the filtered items appear in the view.
- To filter BOM columns:
  - 1. Click in the column to filter.
  - 2. In the dialog box, select filter criteria.

The  $\frown$  changes to  $\boxdot$  indicating that the filter applies to the column.

## **Component Name Display**

🚳 Model View	(?)
× ×	۲
Message	^
Select a part or assembly from which to create the view, then click N	lext.
Part/Assembly to Insert	^
Open documents:	
wormgear_assembly	

When you specify display options for component names in a drawing, the names in the PropertyManager and View Palette match those in the FeatureManager design tree.

#### To specify display options for component names:

- 1. In the FeatureManager design tree, right-click the drawing name and click **Tree Display > Component Name and Description**.
- 2. (Optional.) Under **Primary**, specify an option:
  - Component Name
  - Component Description

Based on the selection under **Primary** display options:

- The Model View and Standard 3 View PropertyManagers update the **Open documents** field.
- The View Palette updates. The View Palette displays the part or assembly icon and the file name.

#### **Detached Drawings**

Detached drawings are no longer supported.

Detached drawings are no longer supported as of SOLIDWORKS 2023. Detailing mode is available. Existing detached drawings continue to work as they do today. Once you save a detached drawing back to a regular drawing, you cannot save it as a detached drawing in SOLIDWORKS 2023.

# 10

# Import/Export

This chapter includes the following topics:

- Exporting Large Assemblies as STEP Files
- OBJ Mesh Import Improvements
- Installing the 3DEXPERIENCE Exchange Add-In

Exporting Large Assemblies as STEP Files

System Options - STEP	
System Options - STEP  System Options  General  MBD  Drawings  Display Style  Area Hatch/Fill  Performance  Colors  Sketch Relations/Snaps Display	File Format: STEP   Output as  Solid/Surface geometry  Wireframe  Export sketch entities  Set STEP configuration data
Selection Performance Assemblies External References Default Templates File Locations FeatureManager Spin Box Increments View	Export appearances     Export 3D Curve features     Split periodic faces     Export assembly components as separate STEP files     (recommended for large assemblies)

You can export large SOLIDWORKS<sup>®</sup> assemblies as STEP files.

In **Tools > Options > System Options > Export > STEP**, select **Export assembly components as separate STEP files (recommended for large assemblies)**. This option exports assemblies as atomic STEP files. Separate STEP files get created for each component in the assembly.

## **OBJ Mesh Import Improvements**

Textured .OBJ import by ScanTo3D is improved.

.OBJ import to graphics body supports textures, excluding bump maps.

Installing the 3DEXPERIENCE Exchange Add-In



The **3D**EXPERIENCE Exchange add-in installs by default with your installation.

The **3D**EXPERIENCE Exchange add-in lets SOLIDWORKS users safely work with a package of content from a **3D**EXPERIENCE source, such as SOLIDWORKS Connected or the Design with SOLIDWORKS app. SOLIDWORKS users can seamlessly open the package, modify files, and return the **3D**EXPERIENCE content package to the **3D**EXPERIENCE source.

To enable the add-in, in SOLIDWORKS, click Tools > Add-ins and under Other Add-ins,

select **3D**EXPERIENCE Exchange. The **3D**EXPERIENCE Exchange s appears in the Task Pane.

See SOLIDWORKS Help: Import and Export > 3DEXPERIENCE Exchange.

# 11

# SOLIDWORKS PDM

This chapter includes the following topics:

- Managing Deleted Users
- Configuring Transition Actions
- Adding Tooltips to the Data Card Controls
- Automatic login to the Administration Tool
- Data Card Controls
- Performance Improvements for File Version Upgrade Tool
- Logging File Retrieval Operations
- Support for Microsoft Edge WebView2-Based Controls
- Removing Computers from the Task Execution List
- Display Groups for Inherited Permissions
- Notification Templates
- SOLIDWORKS PDM Performance Improvements
- Other SOLIDWORKS PDM Enhancements

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

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

## Managing Deleted Users

8 Doc_2023a1	_B220 - Deleted	Users				
7						
Name	Full Name	Initials	ID	Restore login	Warning	
8 Carla 8 John 8 Mary 8 Richard	Carla Coe John Doe Mary Major Richard Miles	CC JD MM RM	7 8 9 10			em Open
				> 🔜 < > 🗊 > 🕅 = 1 > 8 + 2 > 8 + 2 + 2 + 2 + 2 + 2 + 2 + 2 + 2 + 2 + 2	Serial Nu SOLIDWi Fasks Femplate Jsers an Grou Busers	New User Settings Permission Assignments Export History Validate Logins Deleted users
					8 Admi	n (System Administr

You can use the Administration tool to restore deleted users and to refuse login to users.

You must have **Can administrate users** administrative permission.

To restore a deleted user, expand **Users and Groups Management**, right-click **Users**, and select **Deleted users**. In the Deleted Users dialog box, select the user in the **Restore login** column.

To refuse login to a user, expand **Users and Groups Management**, right-click **Users**, and select **Open**. In the Users dialog box, select a user in the **Refuse login** column.

#### Deleted Users Dialog Box

Use the Deleted Users dialog box to restore the login for deleted users.

#### To display this dialog box:

Expand Users and Groups Management, right-click Users, and select Deleted users.

Name	Name the user enters in the login dialog box.
Full Name	The user's full name. The default is the user's login name.
Initials	The user's initials. The default is the first letter of the user's login name.
ID	ID that is unique for each user.

Restore login	When selected, restores the login of the selected user.
Warning	Displays a warning message if the login name you are attempting to restore is already active.

To save changes, click **Save** or close the dialog box and click **Yes** when prompted to save changes.

## **Configuring Transition Actions**



The variable list in the Transition Action dialog box includes the user properties fields that were introduced in SOLIDWORKS PDM 2022.

These fields are:

- First name
- Last name
- Middle name
- Prefix
- Suffix
- E-mail
- E-mail2
- E-mail3
- User Data
- User Data2
- User Data3
- Work phone
- Home phone
- Cellphone
- Company

- Department
- Job title
- Office
- Address
- Zipcode
- City
- State
- Country

These are available under the **User** list and administrators can use these fields in the following transition actions:

- Import data from XML
- Inc. Revision
- Send Mail
- Set Variable

#### Adding Tooltips to the Data Card Controls

E	dit-box properties		
Value			
Variable name:			
🐼 BOM Quantity		$\sim$	
	Variables		
Tooltip			
Title			
Body			
		$\sim$	
Validation			
(Decimal value range	a)		
(a contar reaco tange			

In the Card Editor, you can add tooltips to the data card controls.

In the **Tooltip** pane, enter the **Title** and **Body** of the tooltip.

You cannot specify tooltips for tab and frame controls.

In SOLIDWORKS PDM File Explorer, when you hover over the corresponding data card control, the tooltip displays.

Automatic login to the Administration Tool

Administration Login	
Vault:	
Use automatic login for this vault	
Login as the following user.	
Username:	
Password:	
O Login as the currently logged in user in Windows.	
OK Cancel Help	

You can configure SOLIDWORKS PDM to log you in to the Administration tool using a specific name or the name of the current Microsoft<sup>®</sup> Windows<sup>®</sup> profile.

In the Administration tool, under **Local Settings**, double-click **Settings** or right-click **Settings** and click **Open**.

In the dialog box, under **Administration Login**, select a **Vault** and **Use automatic login for this vault**. You can specify:

- Login as the following user. Logs you in using the specified user name and password.
- Log in as the currently logged in user in Windows. Logs you in using the user name and password of the current Microsoft Windows profile. (Available only for vaults that are configured to use a Microsoft Windows login.)

## **Data Card Controls**



You can apply control logic in data cards more easily.

In addition to the **Control** menu, you can also access the **Control Logic** command when you:

- Click ⅔ in the Controls toolbar.
- Click **Control Logic** in the properties pane.

Not available for card search control and variable search control.

In the Controls toolbar, if you select **Show Controls with Control Logic**, an indicator appears in the top-left corner of the controls that have control logic applied.

You can also configure the **Find User** command type for a button control to return more user properties. Previously, only **Login name** was available as the user value to populate the **Destination variable** for the **Find User** list.

Command type:		
Find User	~	
Dialog box caption:		
Destination variable:		
SenderName	~	
User Value:		
Full name		
Permit multiple selection		

## Performance Improvements for File Version Upgrade Tool

You can upgrade SOLIDWORKS files in a SOLIDWORKS PDM vault faster.

If you are upgrading assembly files and you select **Overwrite** and **Latest Version** in the Version Settings page, the software skips upgrading and uploading references that are not the latest in the assembly's as-built version. This improves the performance of the upgrade process.

### Logging File Retrieval Operations

	× 🔒 🔍	Search te	xt:		
уре	Date	Description	Destination Machine	Destination IP Address	User
Information	2022-03-04 10:30:10	Get via PDM Client	CS/W3064PUP	10.45.48.87	Admin
Information	2022-03-04 10:37:30	Get via PDM Client	CSIW3364PUP	333-895-648-877	Admin
Information	2022-03-04 11:00:17	Get via PDM Client	CSIW3364PUP	333-895-683-877	Admin
Information	2022-03-04 11:24:12	Get via PDM Client	CSIN/384P/P	333-895-683-877	Admin
Information	2022-03-04 11:25:27	Get via PDM Client	CSIN/3064PUP	333-855-683-877	Admin
Information	2022-03-04 11:28:55	Get via PDM Client	CSIW3364PUP	333-855-683-877	Admin
Information	2022-03-04 11:29:36	Get via COM API	CSIW3364PUP	Local	Admin

You can maintain a server-side log of files retrieved for a vault. (SOLIDWORKS PDM Professional only.)

To enable logging, in the Administration tool, right-click a vault and select **Properties**. In the File Vault Properties dialog box, under **Logging operations**, select **Get**. This enables logging of the **Get** operation for the vault on all archive servers.

You must have **File vault management** administrative permission.

To view the file retrieval log, in the Administration tool, right-click a vault, and select **Show the Get Log**. The Log File dialog box displays log information for each file in the vault that is downloaded from the archive server to the local view of a client machine, Web2 server, or Web API server.

#### Log File - Get Operations Dialog Box

The Get Operations dialog box lets you view informational and error events for the files retrieved for a vault on all archive servers.

#### To display the log file:

In the Administration tool, right-click a vault, and select **Show the Get Log**.

Туре	Event type: error, informational, or warning.
Date	Date and time of the event.
Description	<ul> <li>Method used to invoke the Get operation:</li> <li>Desktop client</li> <li>COM API</li> <li>Web API</li> <li>Web2 server</li> </ul>
Destination Machine	Name of the destination machine that receives data from the archive server.
Client IP Address	IP address of the client machine For file retrieval operations through Web2 or Web API servers, the IP address of the server machine is logged.
User	Login name of the user who performed the Get operation.
Document ID	ID of the document
Document Name	Name of the document
Version	Version of the document
Document Path	Path of the document in the vault
Archive Server	Name of the archive server on which the event occurred.

## Support for Microsoft Edge WebView2-Based Controls

In SOLIDWORKS PDM, the Internet Explorer-based controls are replaced with WebView2-based controls.

This applies for:

- Preview pane in SOLIDWORKS PDM File Explorer. You can preview the HTML documents that are present in the vault.
- EXALEAD OnePart Search view
- Web Card Configuration Editor dialog box

You must have the WebView2 runtime installed on your machine.

Removing Computers from the Task Execution List



You can remove computers from the list of hosts that support task execution.

In the Administration tool, open the task's Properties dialog box. In the left pane, click **Execution Method**. In the **Computers supporting the task** list, right-click a computer, and select one of the following:

- **Remove from** *Add-in Name*. This removes the computer as a supported host for all tasks that use the same add-in.
- **Remove from All Add-ins**. This removes the computer as a supported host for all task add-ins.

A confirmation message lists all the affected tasks.

The Task History dialog box displays a list of computers that are removed for that task.

## **Display Groups for Inherited Permissions**

)e - Properties		?
Carla Coe	Administrative permissions:  Can accept tasks to execute on a host Can administrate add-ins Can administrate templates  Can delete labels Can purge history Can administrate users Can administrate users	× 8
roperties	Can undo check o Purchase Management Can update cold s Quality Assurance Can update colum Can update colum Can update docur	
<u>ative Permissions</u> missions missions Permissions ns per File	Can update ERP I Can update histor ✓ Can update index Can update item s Can update licens	
	Can update lists	

In the Administration tool, in the user Properties pages, you can hover over the group icon of a permission to view the groups from which the permission is granted to the user.

A pop-up dialog box appears that lists these groups.

## Notification Templates

Customize Notification	×	
Name: Document added	Preview	
Template:		
O Default Custom		
Subject: Document '%filename%' has been added	>	
Body:		
<html> <style></style></html>		

SOLIDWORKS PDM lets administrators customize the templates for automatic notifications for a vault. This helps to display information that is more relevant to the user.

- You must have **Can update mail configuration** administrative permission.
- You can customize notifications only for files and not for items and Bills of Material (BOMs).

You can customize notification templates for the following file operations:

- Added
- Changed State
- Checked In
- Checked Out
- Delayed in State



You can edit the subject and body of the template in HTML format and also use variables in the notification message.

You can also export notification templates. To export:

- All notification templates. Right-click the **Notification Templates** node and select **Export**.
- File notification templates. Expand Notification Templates, right-click File Notifications, and select Export.
- The notification template for a file operation. Expand **Notification Templates** > **File Notifications**, right-click a file operation, and select **Export**.

**Customize Notification Dialog Box** 

This dialog box lets you customize the templates for automatic notifications for a vault.

To access this dialog box, do one of the following:

- Expand Notification Templates > File Notifications, right-click a subnode, and select Open.
- Expand **Notification Templates** > **File Notifications**, and double-click a subnode.

Name	Displays the name of the notification template. This field is read-only.
Preview	Displays how the template appears to users.
Template	Lets you select either of the notification templates:
	<ul><li>Default</li><li>Custom</li></ul>
	If you select <b>Custom</b> , the fields listed below in the table are enabled. This lets you customize the notification template.
Subject	Displays the notification subject. You can edit the notification
	subject for custom notifications. To include variables, click $\geq$ and select a variable.
Body	Displays the notification body. You can edit the notification
	body for custom notifications. To include variables, click $\ge$ and select a variable.
Load from File	Imports HTML text into the body of the notification.
Reset	Resets the notification template to the default template.

#### SOLIDWORKS PDM Performance Improvements

SOLIDWORKS PDM 2023 makes significant performance improvements in the presence of high latency to SOLIDWORKS PDM servers.

The improvements might vary depending on the amount of latency, size of datasets, and operations.

- The following operations perform better when latency to the database server is high:
  - Check-in
  - Check-in with automatic transition
  - Undo Check-out
  - Interactive operations such as menus, shortcut menus, reference dialog boxes, SOLIDWORKS PDM File Explorer tabs
- The following operations perform better when latency to the archive server is high:
  - Add files
  - Change state on file modifications
  - Get files
  - Get latest version of files

## Other SOLIDWORKS PDM Enhancements

SOLIDWORKS PDM 2023 offers enhanced data security, support for the latest Windows  $^{\rm @}$  operating system, and other improvements.

- SOLIDWORKS PDM encrypts all the data traffic to and from the archive server.
- SOLIDWORKS PDM 2023 supports the Windows<sup>®</sup> 11 operating system.
- In SOLIDWORKS PDM File Explorer, you can copy static text from data cards.
- The language for the user interface of eDrawings<sup>®</sup> Web Viewer follows the language specified in Web2.
# 12

## SOLIDWORKS Manage

This chapter includes the following topics:

- Automatic Update of Mapped Linked Record Properties
- Using the New From Option
- Task Administration Access
- Linked Record BOM Compare Field Selection
- Support for Variable Driven BOM Quantities
- Timesheets to Approve Node
- Cancelled Task Line in Capacity Planning
- Capacity Planning User Efficiency
- Hide Labels
- Marking Tasks as Complete
- Task Snooze Reminder Time
- Tasks Board Time Display

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

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

#### Automatic Update of Mapped Linked Record Properties

🗞 Linked Record Options				
Fields Mapping Document fields on the	LEFT - Record fields on the	RIGHT		
Document Object Column	Record Object Column	Update method	Update Automatically	Info
Constructed by				
ContextCategory				
Copyright				
Cost				
CreationTime				
Customer Name				
Date				
Derived From				
Description	Description	Always	V	
Description				
Description 2				
Dest. doc. ID				

You can update mapped properties automatically to connect primary references and the linked record they are associated with.

The **Linked Record** updates when the **Primary Reference** changes and the record is checked in. Previously, you must check out the linked record and click **Get values from Primary Reference**. You can define each mapped field to update automatically.

Updating Mapped Fields Automatically

- 1. In the SOLIDWORKS Manage Administration Settings tool, edit the **Primary Reference** object.
- 2. On the Linked Record Object wizard page (for PDM objects) or on the Linked Record tab for document objects, click **Configure**.
- 3. On the Fields Mapping wizard page:
  - a) Select the check box in the **Update Automatically** column for the field to update automatically.
  - b) In the right pane, for **Linked Record** with the status, select a status to update when the primary reference is updated.

#### Using the New From Option

Name	e		Visible On Grid	Unique Key	Required	Туре О
<b>4</b>	Properties					
	ab Part Number		$\checkmark$	$\checkmark$	Yes	System (Numbering Scheme) <
	ab Description		$\checkmark$		Yes	System (Text)
	ab Current Stag	je Name	$\checkmark$			System (Text)
	ab Derived From	n	$\checkmark$			System (Text)
	🐻 Web Link					System (Hyperlink)
	ab Decision	10 Edit				List Text 🔹
		Eur     New From     Copy selected value     Refresh     Expand All     Collapse All      Delete     Export     Export				

**New From** lets you copy a field within an object to create new fields with the same settings.

It saves the time required for administrators to configure objects.

**New From** is not available when you configure BOM fields or process item fields.

To use the **New From** option:

- 1. In the Administration tool, edit an object or right-click an object in the main user interface.
- 2. Navigate to the Fields tab or wizard page.
- 3. Right-click a field and select **New From**.
- 4. In the New From dialog box, enter a display name for the new field and do one of the following:
  - Click **Save** to create the new field and close the dialog box.
  - Save and create a new field based on the originally selected field.

#### Task Administration Access

All Tacher All Cr Cr Cr Cr Cr Cr Cr Cr Cr Cr	Add filter Edit filter Delete Filter Refresh Expand All Collapse All		New Ope	Subject	Tasks Board	Charts E
	Administration	<b>≗</b> ≎	User (Admin User All Users Replace User Tasks	)		

You can open the Task Administration dialog box using the **Tasks** option.

To open the dialog box, in the Administration tool, click **All Tasks** > **Administration** > **Tasks**.

Linked Record BOM Compare Field Selection

Include
OK Cancel .::

When you compare a linked record BOM to the BOM of its primary references, you can select fields to compare.

For example, when you assign a new part number to the linked records, the field is always different from the primary reference part number. So you can exclude this field from comparison because it always shows a difference.

Support for Variable Driven BOM Quantities

Edit Variable				— <b>—</b>
Variable name: BOM Quantity Variable type: Decimal number Attributes:			~	<ul> <li>Version free</li> <li>Mandatory values</li> <li>Only look for mandatory</li> <li>values in @ tab for SLDD files</li> <li>Unique values</li> </ul>
Block	Attribute	Extensions		
CustomProperty	UNIT_OF_MEASURE	sldprt		

SOLIDWORKS Manage reads the BOM quantity from a SOLIDWORKS PDM variable set and uses it for the *UNIT\_OF\_MEASURE* attribute.

Previously, SOLIDWORKS Manage used the instance count for the *UNIT\_OF\_MEASURE* attribute.

## Timesheets to Approve Node

Home My Dashboard My Work Tasks Tasks Favorite Reports	<	My Timesheets Timesheets to approve Other Users	Performed in the second s	Properties Number Week Number 1 2 3
1 Timesheets				

**Timesheets to approve** lets you see the timesheets that a user needs to work on, to complete the timesheet approval process.

Cancelled Task Line in Capacity Planning

User Details User: Admin User		Year:	20	22 🔻	3			
Activity	Jan	Feb	Mar	Apr	May	Jun	Jul	Aug
$\Sigma$ Total Time	31	28	31	30	31	30	31	31
Non-Working Time	10	8	8	9	9	8	10	8
🔺 💓 Tasks	7.2	0	0	0	0	0	0	0
Massigned Tasks	22.2	0.2	0	0	0	0	0	0
Completed	0	0.2	0	0	0	0	0	0
⊘ Cancelled	15	0	0	0	0	0	0	0
Available capacity	13.8	20	23	21	22	22	21	23
📢 Task Charts								

You can see the number of hours of cancelled tasks in the **Cancelled** row of the User Details dialog box.

The **Assigned Tasks** row displays total assigned task hours including the cancelled task hours. The **Available capacity** row displays assigned task hours minus cancelled task hours.

#### Capacity Planning User Efficiency

When you view user details, the **Efficiency** (%) value that appears in the Capacity Planning dialog box is applied to the user's time.

Previously, the **Efficiency** in user details was 100% regardless of the setting in the Capacity Planning dialog box.

#### Hide Labels

4			\$	SW-201	1781		GEAR	HOUSING	
Ø			<b>%</b>	SW-201	1826		WASH	HER	
<									~
E Proper	rties R	Show Tabs a	the Top	rations	Related Files	10 F	iistory 🖧 Wherel	Jsed 🔠	Audit Trail t Data
		Hide Labels	•			* * ©	Number Description Revision Created By: Checked Date		SW-20 SPINDI 2 System

**Hide Labels** lets you hide captions. This creates more space in the software so you can see the displayed tabs on the user interface.

To hide captions for the following items, right-click tabs, and select Hide Labels:

- Bottom tabs in the main user interface for all objects
- Property card tabs for processes
- Administration for document and record objects (not SOLIDWORKS PDM)
- Administration for relationships and resource objects

To hide labels for property cards for document and record objects, click  $\equiv$ .

### Marking Tasks as Complete

Ale Relationships	Other options
A Resources	Al Order tasks by
💀 Web Options	Always disable "Complete (%)"
(2) Dashboards	If disabled, the "Progress" section will be used. This field will be calculated automatically.
Timesheets	Note: If a task is assigned to multiple users, this field will be disabled for the specific task.
📝 Tasks	
	Allow more time than allocated
1	O Do not allow more time than allocated
	Create new tasks as "private" or "public"
	Allow users to decide (My Options)  This setting does not overwrite task templa
	Default time for new tasks (hours) 8
	Email task owner when task is completed This setting can be changed in individual tasks
	Enable "Mark as completed" for any assignee when multiple users are assigned

In the System Administration dialog box, you can select a check box that enables **Mark as Completed** for a task that is assigned to multiple users.

Previously, the assigned user must enter the required time in the progress area to mark the task as complete.

Task Snooze Reminder Time

M TSK	(000792 Create CAD Models i	n SOLIDWORKS	
Due 8/9	9/2021 15:00		
Subject		Due In	Туре
TSK000792 Cre	eate CAD Models in SOLIDWO	OVERAL OVERALE 179 Days	Normal

The SOLIDWORKS Manage Reminder saves the last task snooze time.

If you want to keep the same reminder time, only click **Snooze**. Previously, the task snooze time was 5 minutes by default so every time user must change the time.

#### Tasks Board Time Display



The Tasks Board displays the time required for completion, total assigned time, and number of due days until the next task.

The text is color coded depending on the state of the task. Green indicates complete tasks, red indicates overdue tasks, and gray indicates tasks that are not overdue.

# 13

# **SOLIDWORKS Simulation**

This chapter includes the following topics:

- Bonding Interactions
- Diagnostic Tools
- Linkage Rod Connector
- Penalty Stiffness Control for Contact
- Simulation Solvers
- Underconstrained Bodies

SOLIDWORKS<sup>®</sup> Simulation Standard, SOLIDWORKS Simulation Professional, and SOLIDWORKS Simulation Premium are separately purchased products that you can use with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS Premium.

#### **Bonding Interactions**



An improved surface-to-surface bonding algorithm enforces bonding interactions for faces that are not facing each other and have no projection area between them.

A pair of faces qualifies for bonding if these conditions apply:

• The angle between the faces is less than a threshold value.

If the angle between the faces exceeds the threshold value, the pair of faces does not qualify for bonding.

• At least one face of the pair is a flat face (selected as **Set 1** or **Set 2** in the Local Interactions PropertyManager).

The solver considers the flat face as the target face to enforce bonding. The source face can be either flat or curved.

The enhanced bonding algorithm applies to the surface-to-surface formulation and is available for bonded interaction sets that you define using the Local Interactions PropertyManager. It applies to these studies: linear static, frequency, buckling, linear dynamic studies, and also fatigue and design studies associated with linear static studies.

#### **Diagnostic Tools**



You can access the diagnostic tools available for simulations from the CommandManager.

#### To access the diagnostic tools:

From the **Simulation** toolbar, click **Diagnostic Tools**.

#### Linkage Rod Connector



The application of **Linkage Rod** connectors is extended to nonlinear static and nonlinear dynamic studies.

You can specify a **Linkage Rod** connector between cylindrical faces, circular edges (for shells), or vertices to model the behavior of connecting rods.

#### To open the Linkage Rod PropertyManager:

In the simulation study tree, right-click **Connections**  $\mathfrak{P}$  and click **Linkage Rod**  $\mathscr{P}$ .

You can list a linkage rod connector's forces such as shear force, axial force, bending

moments, and torque after you run a simulation. Right-click **Results** is and click **List Connector Force**.

Penalty Stiffness Control for Contact

Static						×	1
Options	Adaptive F	Flow/Ther	mal Effects	Notification	Remark		
Solva							
50176							
Sei	Automatic						
	Manual						
0	Manual						
	FFEPlus		~	1			
	lse inplane ef	fect					
	lse soft sprin	g to stabil	ize model				
	lse inertial rel	lief					
Cont	tact penalty s	tiffness so	ale factor:				
		0.01	0.03	0.1	0.3	1.0	
		1	1	I	1	I I	
	(Appr	oximate)			(	(Precise)	

You can specify a scale factor for the penalty stiffness for contact that is used in linear static studies.

SOLIDWORKS Simulation uses a default scale factor of 1.0 for the penalty stiffness to reach an accurate solution for linear static studies with contact interactions. However, you can select a lower scale factor to obtain an approximate solution faster to assess design iterations and the overall behavior of a model.

You can set the study-level scale factor for the penalty stiffness in the Study Properties dialog box. For global-level settings, select **Simulation** > **Options** > **Default Options** > **Interactions**.

#### **Simulation Solvers**



Function-based processing for the FFEPlus iterative solver is extended to frequency and buckling studies, and to the linear static studies that include node-to-surface interactions and virtual wall definitions.

#### • FFEPlus Iterative Solver

The transfer of stiffness data to solve the systems of equations is optimized for linear static, frequency, and buckling studies because file-based processing is replaced with function-based processing. Solution performance is improved for frequency and buckling studies. The image illustrates the solution performance for a typical frequency study.

In addition, solution performance is improved for linear static studies that contain:

- Node-to-surface bonding and contact interactions
- Virtual wall definitions
- The **FEEPlus** and **Large Problem Direct Sparse** solvers can now handle multiple translational and rotational degrees of freedom that are applied remotely with the distributed connection type.

#### **Underconstrained Bodies**



In SOLIDWORKS Simulation Professional and SOLIDWORKS Simulation Premium, for static studies only, the **Underconstrained Bodies** algorithm includes contact interactions and bolt connectors when detecting rigid (or free) body modes of components.

In SOLIDWORKS Simulation Standard and SOLIDWORKS Premium, the **Underconstrained Bodies** algorithm does not include these improvements and remains unchanged from previous releases.

#### To open the Underconstrained Bodies PropertyManager:

In the Simulation study tree, right-click the **Connections** node, and click **Find Underconstrained Bodies**.

The improved algorithm offers the following advantages compared to the legacy **Underconstrained Bodies** functionality:

- Faster detection of free body modes because the solver performs Singular Value Decomposition (SVD) over the reduced stiffness matrix.
- Inclusion of contact interactions and bolt connectors when analyzing free body modes.
- Realistic visualization of free body modes in planes that are inclined to the orthogonal planes that are defined by the global X-, Y-, and Z-axes.
- Improved visualization of unconstrained displacements for the whole assembly.

# 14

# SOLIDWORKS Visualize

This chapter includes the following topics:

- Color Picker
- DSPBR Material Support
- Importing Options
- Stellar Physically Correct

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

#### **Color Picker**



In the Color Picker, you have more ways to define colors, you can display recently used colors, and you can import and export color swatch palettes.

The Color Picker includes two tabs: Color and Swatches.

On the Color tab you can:

- Define color codes including:
  - **CMYK**, which lets you modify color through CMYK components.
  - **HEX**, which lets you view and edit the hexadecimal code.
- Access the most recently used colors in **Recent Colors**.

On the Swatches tab you can:

- Click **Import Swatch Palette** to import existing color swatch palettes from:
  - SOLIDWORKS Palettes (.sldclr)
  - ACO files (.aco)

- SVG files (.svg)
- HTML files
- Click **Export Swatch Palette** to export color swatch palettes from SOLIDWORKS Visualize and use them in SOLIDWORKS.
- Select preset swatch palette libraries such as Bright, Pastel, and Grayscale.
- Manage existing or new swatch palettes by adding, naming, duplicating, deleting, and locking them.



#### **DSPBR Material Support**

In the **Preview** render mode, display has improved for Physically Based Rendering (PBR) materials, as well as NVIDIA<sup>®</sup> Material Definition Language (MDL) appearances, and  $AxF^{TM}$  appearances.

Improvements include the display of:

- Physically based rendering appearances for better render consistency.
- Materials that use texture distilling in the background, such as MDL and AxF appearances.

#### **Importing Options**



When you import SOLIDWORKS parts and assemblies in SOLIDWORKS Visualize, the **Part Grouping** options include only two choices to streamline the import process.

#### To access the importing options:

- 1. Open a project.
- 2. Click File > Import.
- 3. Browse to a .SLDPRT or .SLDASM and click Open.
- 4. In the dialog box, on the **Geometry** tab, specify **Part Grouping** options.

#### Part Grouping includes:

- The **Part Grouping** menu, which provides two choices when importing SOLIDWORKS .SLDPRT or .SLDASM files.
  - Component/Part/Body
    - Creates nested SOLIDWORKS Visualize groups for SOLIDWORKS assembly components down to the SOLIDWORKS parts and bodies.
    - Creates nested groups for subassembly components, mirror components, pattern components, and SOLIDWORKS folders.
    - Refines SOLIDWORKS Visualize parts based on appearance assignments in each SOLIDWORKS body.
    - Imports saved SOLIDWORKS configurations, animations, and **Monitor File** functionality.

#### • Appearance

- Creates a SOLIDWORKS Visualize part based on every unique SOLIDWORKS appearance.
- Combines SOLIDWORKS geometry having the same appearance into a single SOLIDWORKS Visualize part.

- Imports saved SOLIDWORKS configurations.
- Does not import saved SOLIDWORKS animations or **Monitor File** functionality.

Imported part names are based on their ancestry. Previously, the name was based on the imported appearance.

#### **Stellar Physically Correct**

Stellar Physically Correct yields photo-realistic rendering results.

SOLIDWORKS Visualize supports Stellar Physically Correct, Dassault Systèmes Global Illumination renderer, using the Deep Learning AI Denoiser for optimal rendering performance. It is designed to run on high-performance devices like GPUs or multicore CPUs.

#### To use Stellar Physically Correct:

- 1. Click Tools > Options > 3D Viewport.
- 2. Under Render Engine, select Stellar Physically Correct.

# 15

# SOLIDWORKS CAM

This chapter includes the following topics:

- Barrel Tool Support
- Geometry Tab in Operations Parameters Dialog Box
- Improved Leadin and Leadout Calculations for Contour Mill Toolpaths
- Lines Only Option in VoluMill Toolpath
- Update to Machine to Mean Calculation

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

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

Edit Tool Parameters		
Barrel Tool Mill Holder S	tation	
Tool type :	Conical 🗸	Preview
Tool Dimensions Max diameter (D1) :	Standard Conical	ש ר−ק
Comer radius (R1) :	Tapered	
Flute length (L2) :	Lens	3 15in
Overall length (L1) :	3.10m	
Profile radius (R2) :	1.97in 🗘	5.79in 1.97in
Taper angle (A2) :	3deg	
Upper radius (R4) :	0.02in	0.39in 🥥 🦉 Usin
No. of flutes :	4	
Center cutting :		

Barrel Tool Support

You can define barrel tools to generate toolpaths, mainly for multiaxis finishing operations in SOLIDWORKS CAM and the Technology Database. An advantage to using barrel tools is that they provide a larger area of contact, which results in a better surface finish.

In the Operations Parameters dialog box, under the **Tool** tab, you can select **Standard**, **Conical Tangent**, **Tapered**, **Lens**, or **Advanced** as the barrel tool type. You can also define profiles to create custom barrel tools.

Geometry Tab in Operations Parameters Dialog Box

The **Geometry** tab in the Operations Parameters dialog box lets you define and modify **Multisurface features**, **Avoid features**, and **Contain** and **Avoid areas** for 3 axis milling.

The tab also includes group boxes for **Automatic contain area** and **Avoid small profiles** formerly in the **Advanced** tab.

Improved Leadin and Leadout Calculations for Contour Mill Toolpaths

Operation Pa	rameters
Tool F/S	Contour NC Feature Options Leadin Advanced Posting Optimize
	CNC finish parameters
	CNC compensation
	○ None
	Yes (For example G41/G42)
	Toolpath center
	<ul> <li>With compensation</li> <li>(Toolpath is offset by tool radius)</li> </ul>
	Without compensation
	(Tool center is on feature geometry)
	- Look ahead
	Limited look ahead (Gouge check)
	Internal sharp corners (Corner radius <= Tool radius)

The calculation of **Leadin** and **Leadout** moves is improved so that the **XY start** and **end** points are accurately represented in a compensated toolpath.

In addition to maintaining the same **XY start** and **end** points, SOLIDWORKS CAM automatically adds the tool radius to any arc moves you specify for **Leadins** and **Leadouts**.

For example, if you use a 10mm diameter tool with an arc **Leadin** of 3mm, the calculated **Leadin** arc value is 8mm. When the toolpath is compensated on the machine, the resulting motion produces an arc move of 3mm. For other types of **Leadin** and **Leadout** moves, the length and angle of the **Leadin** and **Leadout** line segments automatically adjust.

For legacy parts, you need to regenerate contour mill toolpaths for these changes to take effect.

Lines Only Option in VoluMill Toolpath

VoluMill Settings		?	×
Entry / Retract			
Entry method :	Spiral		$\sim$
<u>R</u> amp angle :	3deg		-
Entry length :	2.5mm		-
Eloor clearance :	2.5mm		•
Full entry spiral :	$\checkmark$		
Options			
Smoothing <u>r</u> adius : (% tool dia)	45		-
Smoothing radius :	9mm		
Enforce min <u>f</u> eedrate :	50.000mr	n/min	* *
Line moves only :	0.05mm		*
Side <u>m</u> ill only			_
Avoid sharp corners fo	r repositio	ning	

You can split arcs that are larger than 90 degrees into smaller lines using **Lines only**. This option is available in the VoluMill Settings dialog box in SOLIDWORKS CAM and Technology Database, and applies to 2.5 axis and 3 axis VoluMill toolpaths.

You can specify a deviation for this option. This value splits the arcs into linear moves.

Also when you use a pattern with the VoluMill module, you can select an option to **Convert rapid moves to high feed moves** in the **F/S** tab of the Operations Parameters dialog box.

#### Update to Machine to Mean Calculation

SOLIDWORKS CAM tolerance-based machining only uses dimensional tolerances for the Machine to Mean value of a feature. It no longer uses positional tolerances to calculate the Machine to Mean value.

When a feature includes asymmetrical tolerances, the tolerance allowance automatically calculates to the Maximum Material Condition. Positional tolerances are not used because they can overcut or undercut a feature.

# 16

# SOLIDWORKS Composer

This chapter includes the following topics:

- Accessing Online Help for SOLIDWORKS Composer Products
- Creating View Collections
- Player Documentation No Longer Bundled in Composer Packages
- Data Management with ENOVIA 3DLive
- Composer Player 64-Bit Version
- Supported Import Formats
- Support of Meta-Properties from Pro/E Creo and STEP Files
- Other SOLIDWORKS Composer Enhancements

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

#### Accessing Online Help for SOLIDWORKS Composer Products

2

You can access the online help for all SOLIDWORKS Composer products.

Click **Help** > **Use Web Help** to access the online help, clear it to access the local help. Previously, only local help was available. See *Composer Help*: *Working with the Ribbon* 

- > Help Menu, Sync Help: Menus and Commands, and Player Help: Interface Description
- > Menu Bar > Help Menu.

#### **Creating View Collections**



You can manage and organize views into collections in the Views pane.

You can also:

- Drag views into collections
- Expand or collapse collections
- Rename collections
- Activate or deactivate collections
- Delete collection views

You can create a collection of views in the Views pane by doing the following:

- In the Views pane, click **Create View Collection** 10.
- Right-click a view and select Add Collection.

See Composer Help: Working with Views > View Collections and Working with the Left Panes > Views Pane.

#### Player Documentation No Longer Bundled in Composer Packages

When you save the active Composer document as an auto-executable package (.exe file), the Player documentation is no longer bundled in the package.

You can access the Player documentation online from the Player. See *Composer Help*: *Importing and Opening Files* > *About Composer File Types*, *Interface Description* > *Dialog Boxes* > *Save Package As*, and *Interface Description* > *Dialog Boxes* > *Send by E-mail*.

#### Data Management with ENOVIA 3DLive

You can no longer import and update models from an ENOVIA<sup>®</sup> 3DLive session.

The following commands are removed from the user interface:

- Import from 3DLive and Update from 3DLive from the File menu.
- ENOVIA 3DLive > Import and ENOVIA 3DLive > Open in ENOVIA 3DLive right-click commands from the Assembly pane.

#### Composer Player 64-Bit Version

The Use Composer Player 64-bit version option is removed from the Package options in (Default) Document Properties > Output.

See Composer Help: Managing (Default) Document Properties > Output.

#### Supported Import Formats

SOLIDWORKS Composer and SOLIDWORKS Composer Sync support higher versions of several import formats.

See Composer Help: Importing and Opening Files > About Supported Import Formats and Sync Help: About Import Formats and File Types > About Supported Import Formats.

#### Support of Meta-Properties from Pro/E Creo and STEP Files

You can import meta-properties from Pro/E Creo (\*.prt, \*.asm), and STEP (\*.stp, \*.step) files in SOLIDWORKS Composer and SOLIDWORKS Composer Sync.

Meta-properties are user-defined attributes such as part number (P/N), designation, and revision number. They appear as user properties of the string type in the Properties pane. See *Composer Help: Managing (Default) Document Properties > Input* and *Sync Help: Managing Default Document Properties > Input*.

#### Other SOLIDWORKS Composer Enhancements

SOLIDWORKS Composer products have additional enhancements.

- In the Technical Illustration workshop, you can no longer select SVGZ as the format for saving vector images. The SVG format is still available. If the SVG file is too large, you can convert it into the SVGZ compressed format using a standard file compression tool. See *Composer Help: Working in Workshops > Technical Illustration Workshop*.
- On the **Publish** page, you can no longer select SVGZ as the output format for vector images. The SVG format is still available. If the SVG file is too large, you can convert it into the SVGZ compressed format using a standard file compression tool. See *Sync Help: Configuring Conversion Properties > Publish*.
- In SOLIDWORKS Composer Player, you can see views grouped into collections in the Views pane.

# 17

## SOLIDWORKS Electrical

This chapter includes the following topics:

- Defining Open-Ended Wires for Routes
- Propagating Electrical Properties from 2D to 3D
- Associate Feature Improvements
- Electrical Manager Tree Improvements
- Dynamic Connection Labels
- Origin-Destination Symbol
- Displaying Column Header Text in All Electrical Project Languages
- Manufacturer Part Weight
- Automatic Orientation in PDF Export
- Deleting Components
- Additional Variables in Origin-Destination Formula
- Resetting Dismissed Messages
- Multilingual Attributes
- Microsoft Excel Automation Improvements
- Embedding Report Tables
- Additional File Formats for Images

SOLIDWORKS<sup>®</sup> Electrical is a separately purchased product.

# Defining Open-Ended Wires for Routes

Open-ended wires have one extremity that is disconnected from a component or terminal. The open-ended wire property automatically propagates to the From-To list for routes.

Propagating Electrical Properties from 2D to 3D

📰 Properties 🍾 Circuits, terminals							
Depth:	Sumr	nary Informatio	n				
4 Use	- Cum	nani Custan	Configur	ation Specific			
Voltage:	Sum	mary custom	comiga	adon spearre			
Frequency:		A	pply to:				
<ul> <li>Control</li> </ul>		Delete	21113 Sch	neider Electric	45::89	k(75 V	
Voltage:		Descente	blama	Tring		Unline LTend Europeaning	
Frequency:		Property	Name	Туре	_	Value / Text Expression	
<ul> <li>Manufacturer data</li> </ul>	1	Reference		Text	~	21113	-
Current rating:	2	Manufacturer		Text	$\sim$	Schneider Electric	1
Curve:	3	Description		Text	$\sim$	20-25A P25M motor protection mcb	1
Current of thermal setting:	4	Class name		Text	~	Circuit-breakers\Magneto-thermal	(
Current of magnetic setting:	5	Туре		Text	~	Base	l
Residual current threshold:	6	Article numbe	r	Text	~		
Time:	7	External ID		Text	~	-	
Description	8	Series		Text	~		
<ul> <li>國 Description (English):</li> </ul>	9	Data sheet		Text	~	-	
國家 Description (French):	10	Supplier name		Text	~	1	-
<ul> <li>▲ 函型 Commercial reference (English):</li> </ul>	11	Stock number		Text	~		-
國 Commercial reference (French):	12	Number of cir	cuits	Text	~	3	1
A User data	12	Number of br	and the set of	T	-	2	-

For each component, you can propagate the electrical properties to the 3D part or assembly.

If you modify the electrical properties in the 2D application, you must click **Update BOM Properties** in the SOLIDWORKS Electrical menu, to update the 3D part properties.

You can use these properties to complete tables like the Bill of Materials.

#### Associate Feature Improvements

		Associate	e component	3				
<ul> <li>×</li> </ul>	×.							
Messag	e			^				
Select a	component	in 3D assembly						
Associa	te compone	nt		^				
Mark	Reference	Manufacturer	Description 3D Part/Assembly	1				
Q1	06557	Legrand	Magneto-the 06557-2@481					
Q2	06468	Legrand	Magneto-the					
Q3	06468	Legrand	Magneto-the					
Part or a	Assembly to : 17-2@481	Associate:	_	1	o FN o			
			0					
Propert	ies			^				
Mark	Cir   Co.	CPoint	2D Sch					

In SOLIDWORKS Electrical 3D, the **Associate** command lets you associate electrical components with 3D parts or assemblies.

The **Associate Component** PropertyManager lets you:

- Select several components to associate them with several 3D parts or assemblies.
- Display the properties of the selected component to check the consistency with the 3D part or the assembly.



**Electrical Manager Tree Improvements** 

In SOLIDWORKS Electrical 3D, the **Electrical Manager** tree displays the list of components in the electrical project.

In the **Electrical Manager** tree, an icon before each component indicates whether the component is associated or not associated with an assembly or a 3D part. Selecting a component in the tree highlights the associated assembly or 3D part in the graphical area.

Selecting the location from the **Electrical Manager** highlights the assembly in the **Feature Manager** tree if the corresponding assembly is inserted in the main assembly.

The Electrical Manager lets you:

- Show or hide inserted components
- Expand or collapse all tree nodes

#### **Dynamic Connection Labels**



Dynamic connection labels are specific symbols that display connection information about a component. You can dynamically insert the symbols into schemes.

The dynamic connection labels use configuration files where you enter the parameters to draw them. Click **Electrical project** > **Configurations** > **Connection label** to manage the configuration files.

To insert dynamic connection labels, click **Schematic** > **Connection label** > **Insert a dynamic connection label for a component**.

#### **Origin-Destination Symbol**

Ar	row symbol insertion	
<b>X</b> -H	SOLIDWORKS Electrical	×
Message	Origin - destination	group
Specify insertion p		SOLIDWORKS Electrical ×
Specify the symb	ightarrow Create new gro	Origin - destination group
Speeny the symb	$\rightarrow$ Select existing	Name:
	$\rightarrow$ Cancel	
L.		→ ок
	<b>V</b>	→ Cancel
	Other symbol	

You can manually insert origin-destination arrow symbols at the end of the wires. You associate a group number with each inserted symbol. All the wires connected on the same group number have the same equipotential number.

To insert an origin-destination symbol, click **Schematic** > **Origin** – **destination arrows** > **Insert a single origin** – **destination arrow symbol**.

1-My book						
Drawing Folio Výkres	Location Localisation Umístění/loka	ice	Revision Révision Verze			
ral 🛕 Themes 🎹 General	Columns 🝙 Layout 🚦	ž↓ S( • •	Header	ext height:	2	
ral A Themes Themes	Columns 🝙 Layout 🚦	₽ S( • •	Header	ext height:	2 Tahoma	
General Text height: Font:	Columns 📾 Layout 💈 2 Tahoma	<u>د</u>	Header F B	ext height: ont: old:	2 Tahoma	
General Text height: Font: Bold:	Columns 📾 Layout 💈 2 Tahoma	- -	Header T F B	ext height: ont: old: :alic:	2 Tahoma	
General Text height: Font: Bold: Italic:	Columns 📾 Layout 💈 2 Tahoma	- -	Header T F B 10	ext height: ont: old: :alic: olor:	2 Tahoma	
General Text height: Font: Bold: Italic: Color:	Columns 📾 Layout 💈 2 Tahoma	- -	Header T F B IN C R	ext height: ont: old: calic: iolor: ow style:	2 Tahoma	

Displaying Column Header Text in All Electrical Project Languages

In the report, you can display column header text in all three languages specified in the electrical project configuration.

To activate this feature, in the report configuration, select **Show all project languages**.

#### Manufacturer Part Weight

📰 Properties 🛛	🖺 User data 💊 Circu	its, terminals	<b>6</b> 06468-4@481
Description	(English):	20-25A P25M motor pro	
All Commercia	I reference (English):		Querride Mass Properties
Supplier			Overnue Mass Properces
Information		🗹 Include hidden bodies/component	
Illustration		Create Center of Mass feature	
4 Size			Show weld bead mass
Width:		45 mm	Depart coordinate unlines relative ter
Height:		89 mm	Report coordinate values relative to:
Depth:		75 mm	Mass properties of 06468
Weight (kg):		0.08548	Configuration: Defaut Coordinate system: default
⊿ Use			
Voltage:		690	Density = 0.00 grams per cubic millim
Frequencia		50-60	e child and group per exercision

You can manage the weight of a manufacturer part.

The weight propagates to the **Mass properties** of the 3D assembly associated with the component on which you assign the manufacturer part. For components that are cut, like rails or ducts, the weight updates according to the **Depth** value that you use in the cabinet layout.

Automatic Orientation in PDF Export

Export PDF riles			
Paper format:	ISO A3 (297.00 × 4	20.00 MM)	~
File name	E:\MyProject.pdf		
🗹 Create bookma	rks and hyperlinks		
Export one PDF	file by book		
Export data file	95		
Orientation		Drawings	
Automatic		⊖ All	
🔘 Portrait		Selection	
OLandscape	A		
Plot with symm	netry		
		Export fonts as geometry	
Scale			

When you export the electrical project drawings to a PDF file, you can keep the same page orientation.

Drawings in **Landscape** orientation convert to PDF pages in **Landscape** orientation, and drawings in **Portrait** convert to PDF pages in **Portrait** orientation.

Select the **Automatic** option, available in **Import/Export > Export PDF files**.

**Deleting Components** 

Components	및 ≍ 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	
Training     Training     Training     The second sec	Delete component     Are you sure that you want to delete the selected component and all the symbols representing i     This operation cannot be undone.     Symbols from Component "F3" will be removed from following drawings:         -1 - My book:             *03 - Electrical scheme:             +03+8 - One pole + neutral fuse disconnector (1,2)     *21 - TGET:	× t.
	Yes No	
€	Copy         Ctrl+C           Paste         Ctrl+V	_
€ =F1-5	X Delete component Del	

In the **Components** dockable panel, you can delete a component and all the symbols representing it in the drawings.

A component without inserted symbols is automatically deleted. If you want to delete a component with inserted symbols in the drawings, a dialog box opens, lets you select the option to delete all the symbols associated with this component. If the component is associated with a 3D part or assembly, it is automatically dissociated.

#### Additional Variables in Origin-Destination Formula

Electrical Project Configuratio	n: Training Symbol 💿 Attribute	A Text 🔎	Mark 🝙 Title blo	ck 👔 Library a	and palette			
Multilingual texts:								
Description	Font	Height	Bold	Italic	Underline	Color		
🚰 Main language (English) 🚰 Second language (French)	Tahoma Tahoma	2	-			<b>I</b> Black ■Black	/ White / White	
Ither texts:								
Description Font	Height Bol	d Italic Co	lor ×	Y Formula	1			Fx
Equipotential Taho	ma 💌 1.5 - 🗌		Black / White 5	1 ~				
Wire number Taho	ma 💌 1.5		By Laver 0	1 ~				
Origin destination Taho	ma 💌 1.5 🗌		By Layer 1	0 - IIF(LEN	(BOOK_TAG) =	= 0, '''', BOC	DK_TAG + "''')	
🖋 Scheme cable core 🛛 Taho	ma 💌 1.5		Red	-2 - CABLE_	TAG + "\\" + D	ABLE_CORE	E_DESCRIPTI	
ELine diagram cable Taho	ma 💌 3 — 🗌		By Layer	0.5 - CABLE_	TAG + '' ('' + CA	BLE_REFE	RENCE + ")"	
Function outli	management: Origin - d	estination			- 0	$\times$	, LOCATION , FUNCTIO	
Black box ter 🔊 Predefin	ed formulas 🕤 Rece	nt formulas	Variables and :	simple formulas	🕼 Functio	ns		
Use symbol for Simple formula	5	D	escription					
- FOLDER	ORDERNO	0	rder number			^	have been in all as	entire al
- STRZ	(VAL(FOLDER_ORDERN)	), 2, 0) F	older order number	on 2 characters			box terminal - W	enical
- STRZ	(VAL(FOLDER_ORDERN)	D), 3, 0) F	older order number	on 3 characters	k.			
- FILE_TAG	;	Fi	ile mark					

New variables for components and wires are available in the Formula Management dialog box.

To use these variables, open the electrical project configuration, and in the **Text** tab, click the **FX** cell for **Origin - destination**.

**Resetting Dismissed Messages** 

Do you want to open the la	st opened
Training	Interface configuration
$\rightarrow$ Yes	Graphic options B Preferences Application language Rights manager
→ No	Selection mode: C Use "Contro C Click on ent
Do not show again	
	Dismissed messages Checked messages that will be shown again
	Dpen recent drawing. Do you want to open the last opened drawing for this project?

A new option lets you manage dismissed messages.

Some dialog boxes have the **Do not show again** option to permanently dismiss them. To reset this status, in the **Interface configuration** dialog box, in the **Preferences** tab, select the dialog boxes to show again.

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_			0 10/13/2016 J. Smith REV. DATE NAME		REVISION 0 SO-EME

## **Multilingual Attributes**

The multilingual attributes are specific attributes that propagate a multilingual text. This multilingual text displays the desired language when you change the main language of the electrical project.

You can use this type of attribute in title blocks or specific symbols like wiring cable labels. You manage multilingual attributes in the title block editor or the symbol.

#### **Microsoft Excel Automation Improvements**



**Excel automation** allows you to automatically generate schematic drawings from data contained in a  $Microsoft^{\circ}$  Excel<sup> $\circ$ </sup> file.

With the Excel automation, you can:

- Manage the origin and destination arrows
- Replace a symbol while inserting a macro
- Select the folder for the generated drawings
## **Embedding Report Tables**



You can insert report tables anywhere into electrical project drawings.

According to configuration parameters, you can automatically insert specific report tables into graphical drawings like schematic, wiring line diagram, or 2D cabinet layout schemes. After changes of the electrical project data, you can update the report content.

Additional File Formats for Images



You can use several file formats to insert or attach images to drawings.

The supported file formats are:

- Bitmap images (\*.bmp)
- DIB images (\*.dib)
- JPEG File Interchange Format (\*.jpg)
- Portable Network Graphics (\*.png)
- Tagged Image File Format (\*.tiff)
- Graphics Interchange Format (\*.gif)

# 18

## **SOLIDWORKS** Inspection

This chapter includes the following topics:

- SOLIDWORKS Inspection Add-in
- SOLIDWORKS Inspection Standalone

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

#### SOLIDWORKS Inspection Add-in

#### **Balloon Sequence**

	alloon Sequence	Û
Sequence Nan	e Starting Value	
Default	1	
Hundreds	100	
Custom	2	

You can use the **Balloon Sequence** tool to define the balloon numbering sequence in multisheet drawings.

In the SOLIDWORKS Inspection add-in, click **Balloon Sequence** (SOLIDWORKS Inspection toolbar) and specify options in the PropertyManager.

Balloon Sequence PropertyManager

To open this PropertyManager, in the SOLIDWORKS Inspection add-in, click **Balloon Sequence** (SOLIDWORKS Inspection toolbar).

### Sequence Name

Specifies the different balloon sequences to define. You can change the name of the sequences.

## **Starting Value**

Specifies the starting numeric value (whole numbers only) of each balloon sequence.

#### Add

Adds a row to the balloon sequence table.

#### Delete

Removes a selected row from the balloon sequence table.

**Balloon Drawing Sheets Separately** 



In multisheet drawings, you can specify sheets to balloon instead of ballooning all sheets in a drawing.

In the Create Inspection Project PropertyManager, under **Sheets**, select **Include All** or you can specify sheets. With **Include All**, the entire drawing is ballooned. When you select sheets, you can specify each sheet in the drawing to balloon in inspection reports. You can also specify the balloon sequence. You define the balloon sequence with the

**Balloon Sequence** by tool or by entering it in **Sequence/Starting Value**.

If you have two or more sheets to balloon continuously, in **Sequence/Starting Value**, select **Continue from previous sheet**.

Export Sheets as Separate 2D PDF Files

2D PDF E	port		×
Export Export Selected S	drawing as single PDF sheets as separate PDFs heets		
	Save	Sheets	
▶		Sheet1	
	$\checkmark$	Sheet2	
	$\checkmark$	Sheet3	
	$\checkmark$	Sheet4	
		Ok Cancel	

If you balloon drawing sheets separately, you can export the  ${\tt Microsoft}^{\circledast} \: {\tt Excel}^{\circledast} \: {\tt and} \: .{\tt pdf}$  reports separately. Previously, the reports exported as a continuous report with all sheets included.

In the SOLIDWORKS Inspection add-in, click **Export to 2D PDF** (SOLIDWORKS Inspection toolbar). In the 2D PDF Export dialog box, specify options:

- **Export drawing as single PDF**. Exports the entire drawing as a single .pdf file.
- **Export sheets as separate PDFs**. Exports each drawing sheet as a separate .pdf file.
- Selected Sheets. Specifies the drawing sheets to save in the .pdf file.

Export FAI Reports to Separate Microsoft Excel Documents

When you export a First Article Inspection (FAI) report to Microsoft<sup>®</sup> Excel<sup>®</sup>, you can export characteristics to multiple spreadsheets.

In the SOLIDWORKS Inspection add-in, click **Export to Excel** . In the Exporting PropertyManager, under **Excel Export Settings**, specify options:

- **Export drawing to single excel**. Exports all characteristics from the specified sheets to one Microsoft Excel spreadsheet.
- **Export sheets to separate excel**. Exports all characteristics from the specified sheets to separate Microsoft Excel spreadsheets.

#### SOLIDWORKS Inspection Manager



The SOLIDWORKS Inspection Manager contains a Characteristics section with tabs for Characteristic Table and Characteristic Tree. If you select an item in the characteristic table or characteristic tree, the corresponding entity highlights in the graphics area.

The characteristic table is existing functionality that contains the Bill of Characteristics, which lists each ballooned entity. You can select multiple balloons from the same view and reorder them in the characteristic table. When you drag a balloon to reorder it in the table, scrolling speed and usability is improved. The **View** column in the table lists the view to which each characteristic belongs.

The characteristic tree contains information for each sheet and view in the drawing. It only displays sheets and views that have ballooned items. In the characteristic tree, you can reorder:

- Balloons by dragging views.
- Views that belong to the same sheet.
- Sheets. The sheet and its views are reordered and renumbered according to the new position in the characteristic tree.

#### SOLIDWORKS Inspection Standalone

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

#### Autoballoons for PDF Files



The **Auto Extract** tool automatically applies balloons to characteristics in PDF files, which reduces the time and effort needed to balloon PDF files.

**Auto Extract** uses a machine learning technique to locate the positions and types of annotations, then attaches balloons to them. You can also use this tool to automatically apply balloons to CAD files. The **Auto Extract** tool replaces the **Smart Extract** tool.

In a PDF file, on the Document ribbon, click **Auto Extract** <sup>3</sup>. **Auto Extract** does not support notes or weld symbols.

You can also manually extract characteristics with the **Manual Extract**  $\stackrel{}{\sim}$  tool (Document ribbon).

Autoballooning PDF Files

#### To autoballoon PDF files:

- 1. In SOLIDWORKS Inspection Standalone, click **New** .
- 2. In the Project Template dialog box, select a template and click  $\checkmark$ .
- 3. In the Open CAD/PDF File dialog box, select a PDF file and click **Open**.
- 4. Click **Auto Extract**  $\stackrel{W}{\longrightarrow}$  (Document ribbon).
- 5. Specify settings in the Auto Extract PropertyManager and click  $\checkmark$ .

Auto Extract PropertyManager

You can use the **Auto Extract** PropertyManager to automatically apply balloons to characteristics in CAD models, drawings, and PDF files.

To open the Auto Extract PropertyManager, in a model, drawing, or PDF file, click Auto

Extract 🥍

## **Extraction Settings**

(PDF files only.) Specifies the characteristics to balloon. Notes and weld symbols are not supported.

Include All	Balloons all dimensions, geometric tolerance symbols, surface finish symbols, and datums in the file.
Dimensions	Balloons all dimensions.
GD&Ts	Balloons all geometric tolerance symbols.
Surface Finishes	Balloons all surface finish symbols.
Datum	Balloons all datums.

#### Dimensions

(CAD models and drawings only.)

Include	Balloons all basic, feature, and reference dimensions. Clear <b>Include</b> to select individual <b>Basic</b> , <b>Feature</b> , or <b>Reference</b> dimensions.				
Inspection Only	Balloons all inspection dimensions.				
Auto-explode hole callouts	(SOLIDWORKS files only.) Balloons and splits compound hole callouts into subballoons.				

#### **Notes**

(CAD models and drawings only.)

Include	Balloons all datum feature, datum target, and flag note symbols. Clear Include to select individual Datums, Datum Targets, Flag Notes, Correlate Flag Notes, or Auto-explode multi-line notes.
GD&Ts	
(CAD models and drawings only.)	
Include	Balloons all geometric dimensioning and tolerancing feature control frames.
Welds	
(CAD models and drawings only.)	
Include	Balloons all weld symbols.
Surface Finishes	

(CAD models and drawings only.)

Balloons all surface finish symbols.

## **Geometry Options**

(CAD models and drawings only.)

Load Invisi	ble Components	5
-------------	----------------	---

Loads hidden components into SOLIDWORKS Inspection.

## Page 2 of PropertyManager

Click **Next** ▶ in the PropertyManager to access these options.

Sheets, Include All, sheetn
(Drawings and PDF files only.) Balloons all characteristics in the document.

### **Entire Document**

(CAD models only.)

Entire Document	Balloons all characteristics in the document.
Include All	Includes all annotation views, otherwise you can select individual annotation views.
Captures	
Include All	Balloons all model based definition screen captures, or you can specify screen captures to include.
Views	
Include All	Balloons all annotation views, or you can specify views to include.

Manually Ballooning PDF Files



The **Manual Extract** tool replaces the optical character recognition (OCR) editor tool. It is streamlined to make manual extraction of annotations quicker to balloon.

#### To manually balloon PDF files:

1. In SOLIDWORKS Inspection Standalone, click **New** .

- 2. In the Project Template dialog box, select a template and click  $\checkmark$ .
- 3. In the Open CAD/PDF File dialog box, select a PDF file and click **Open**.
- 4. Click **Manual Extract** (Document ribbon) and select the type of annotation to balloon:
  - Dimension
  - Geometric Tolerances
  - Note
  - Surface Finish
  - Weld
  - **Other**. Includes annotations such as temperature, torque, and electrical resistance.
- 5. In the PDF file, box-select an annotation of the type you selected in the previous step.
- 6. In the Properties/Preferences PropertyManager, under **General**, **Inspection**, **Ballooning**, and **Custom**, enter values for the remaining data in the annotation.

User Interface Redesign

	SOLID	WORKS In	spection	ı					
View Getting Started									
First Previous /12 Next Last	<ul> <li>□ Fit Width</li> <li>↓□ Fit Height</li> <li>↓↓ Fit Page</li> </ul>	<b>Q</b> Zoom	<u>]</u> 100%	Zoom	Zoom Out	Pan (	Zoom to	Top	
Sheet			Zoo	m				S	horr
	2022 SOLIDWORKS Inspection								
View Getting Started									
<b>2 ≥</b>	☆ Fit Width I Fit Height	$\mathbf{P}$		Ð	Ð	¢			
First Previous /12 Next Last	🛃 Fit Page	Zoom	100%	Zoom In	Zoom Out	Pan	Zoom to Area	Top Left	Ti Ri
Sheet			Zoo	m				S	10V
2023									

The user interface for SOLIDWORKS Inspection Standalone is redesigned to provide better support for high-resolution, high-pixel-density displays. Icons are consistent with those in SOLIDWORKS.

Icons are more crisp and readable, and they scale properly when used with high-resolution displays.

# 19

## SOLIDWORKS MBD

This chapter includes the following topics:

- Annotations and Dimensions in 3D PDF Files
- Editing Compound Features
- Dimension Extension Lines
- Wedges
- ISO Tolerance-Modifying Symbols

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



Annotations and Dimensions in 3D PDF Files

When you create a 3D PDF for an assembly, you can view all dimensions in the assembly and 3D views, even in the viewport, with no data loss.

You can view component dimensions in 3D PDF files, including feature dimensions, reference dimensions, and DimXpert annotations.

## **Editing Compound Features**



You can add or remove faces on an existing compound feature while maintaining the annotations.

You can edit compound planes, holes, and cylinders.

#### To edit compound features:

- 1. In the DimXpertManager, right-click the compound feature and click **Edit Feature**.
- 2. Select features to add or remove from the compound feature.
  - To add a face, select a feature and click **OK**.
  - To remove a face, right-click a feature and select **Delete**.

## **Dimension Extension Lines**



You can use dimension extension lines to connect the dimensions to the model.

#### To show dimension extension lines:

- 1. Click Tools > Options > Document Properties > DimXpert > Display Options.
- 2. Under Linear Dimension, select Connect dimension extension lines to model.



## Wedges

MBD supports wedge features. A wedge has two nonparallel planes with opposing vectors.

#### To create a wedge:

- 1. Click **Size Dimension** (MBD Dimension toolbar).
- 2. Select the first plane.



- 3. From the Feature Selector, click Create Width/Wedge Feature **N**.
- 4. Select the second plane.



- 5. In the **Feature Selector**, click **\***.
- 6. Click to place the dimension.



ISO Tolerance-Modifying Symbols

Symbol Library	X
Insert	
Categories:	
ISO Tolerancing 📃 🔺	Symbols
ANSI Lay Symbols	LP LS GG GX GN
ANSI Weld Contour Symbols ANSI Weld Spacer Symbols ANSI Weld Symbols	
Draft Flag Cir 1-99 Flag Cir 1-99	(SN) (SA) (SD) (SR)
Flag Cir A-Z Flag Cir A-Z	$\overline{\text{SQ}} \oplus \overline{\text{C}} \oplus \overline{\text{F}} \oplus \overline{\text{A}}$
Flag Squ 1-99 Flag Squ 199. Flag Squ A-Z	$\widehat{\mathbf{G}}$ $\widehat{\mathbf{N}}$ $\widehat{\mathbf{T}}$ $\widehat{\mathbf{X}}$ acs
Flag Squ A,-Z, Flag Squ-Cir 1-99 Flag Squ-Cir 1-99	SCS ALS CT UF 🔫 🕨

You can apply ISO 14405-1:2016 standards-based symbols to dimensions and tolerances in DimXpert.

You can add symbols and other text directly to ISO dimensions and their tolerances from the **Tolerance Modifier** section in the DimXpert PropertyManager.

# 20

## eDrawings

This chapter includes the following topics:

- Assembly Graphics
- Configuration Pane
- Configuration Support on Mobile Devices
- Display States
- Display Styles
- Supported Versions of File Types
- Using Localized Versions of eDrawings Web HTML
- Using Fractional Measurements

eDrawings<sup>®</sup> Professional is available in SOLIDWORKS<sup>®</sup> Professional and SOLIDWORKS Premium.

#### **Assembly Graphics**

Graphics in eDrawings assemblies have improved. When you publish SOLIDWORKS assemblies as eDrawings assemblies, the graphics more closely resemble the original SOLIDWORKS files.

If the eDrawings assembly contains linked display states, the Configuration pane shows **Display States (Linked)**.

## **Configuration Pane**



When you open files in eDrawings that have only one configuration, the Configuration pane is available. Previously, there was no Configuration pane.

Configuration Support on Mobile Devices



You can specify configurations of SOLIDWORKS parts and assemblies to be available when you open the models in eDrawings for iOS<sup>M</sup> and eDrawings for Android<sup>M</sup>.

In the SOLIDWORKS software, set configurations with **Add Display Data Mark** to view the configurations in eDrawings. For information, see *SOLIDWORKS Help: Generating Display Lists*.

## **Display States**



When you publish SOLIDWORKS assemblies as eDrawings assemblies, display state data (for the selected configurations in the Save Configurations to eDrawings file dialog box) is stored in the eDrawings assembly files. This gives you important information to communicate and collaborate in eDrawings.

On the Configuration pane, the option, **Link display states to configurations**, is removed. Display states are linked or unlinked based on the setting in the SOLIDWORKS assembly at the time of publishing the eDrawings assembly file from SOLIDWORKS. When the display states are linked to the configuration, only the display states that are associated with the selected configurations are published.

Textures are not supported in the display state data.

## **Display Styles**



You can view models in hidden lines visible (HLV) and wireframe modes. The tooltip for **Wireframe** has changed to **Hidden Lines Removed** (HLR).



## Supported Versions of File Types

eDrawings has updated the supported versions for several file types.

Format	Version
ACIS(.sat, .sab)	Up to 2020
Autodesk <sup>®</sup> Inventor <sup>®</sup> (.ipt., .iam)	Up to 2022
CATIA V4 (.MODEL, .SESSION, .DLV, .EXP)	Up to 4.2.5
CATIA V5 (.CATDrawing, .CATPart, .CATProduct, .CATShape, .CGR)	Up to V5_6R2021
CATIA V6 / 3DEXPERIENCE (.3DXML)	Up to V5-6 R2019 (R29)
Creo - Pro/Engineer (.ASM, .NEU, .PRT, .XAS, .XPR)	Pro/Engineer 19.0 to Creo 8.0

#### eDrawings

Format	Version
IGES (.igs, .iges)	5.1, 5.2, 5.3
JT(.jt)	Up to v10.5
NX (Unigraphics) (.prt)	V11 to v18, NX to NX12, NX1847 Series to NX1980 Series
Parasolid (.x_b, .x_t, .xmt, .xmt_txt)	Up to 33.1
Solid Edge (.asm, .par, .pwd, .psm)	V19-20, ST-ST10, 2021
STEP(.stp,.step,.stpz)	AP 203 E1/E2, AP 214, AP 242
Wavefront OBJ (.obj)	Any

Using Localized Versions of eDrawings Web HTML



(Available in eDrawings Professional.) When you save a file as eDrawings Web HTML, then open the .html file, you can select a language to display the user interface in the localized language. The file itself retains its native language.

This is helpful if you share the file with a user who speaks a different language.

#### To use a localized version of eDrawings Web HTML:

- 1. In eDrawings, click **File** > **Save As**.
- 2. In the dialog box, in Save as type, select eDrawings Web HTML Files (\*.html).
- 3. Double-click the file that you saved.

4. In eDrawings Web HTML, click **Languages**  $\overset{*}{\rightarrow}$  and select a language.



## **Using Fractional Measurements**

If you use the English unit system to measure eDrawings models, you can display measurements as fractions with the **Measure** tool.

#### To use fractional measurements:

1. Click Measure



- 2. On the Measure pane, under **Units**:
  - a. In Length, select Inches or Feet and Inches.
  - b. Select **Display In Fractions**.
  - c. For **Denominator**, enter the maximum digits for the fraction denominator.
  - d. Optional: Select **Round to nearest fraction** to round to the next highest fraction if the measurement does not have a fractional value for the denominator.

# 21

## **SOLIDWORKS Plastics**

This chapter includes the following topics:

- Plastics Materials Database Material Grades
- Plastics Materials Database Manager
- Summary and Report

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



Plastics Materials Database - Material Grades

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

Materials	Description
New Materials	Added 14 new material grades from material manufacturer EMS-GRIVORY.

Materials	Description
Modified Materials	Updated 54 material grades with the latest material property values from material manufacturer EMS-GRIVORY.
Removed Materials	<ul><li>Removed obsolete material grades from the database:</li><li>EMS-GRIVORY: 1</li><li>CHI MEI Corporation: 37</li></ul>

1

## Plastics Materials Database Manager

E	PC : Covestro / Makrolon 2556		Ξ	Makrolon 2556	
	Melt Temperature	300 °C		Polymer Family	PC
	Max. Melt Temperature	320 °C		Manufacturer	Covestro
	Min. Melt Temperature	280 °C		Recommended Melt Temperature	300 °C
	Mold Temperature	100 °C		Maximum Melt Temperature	320 °C
	Max. Mold Temperature	120 °C		Minimum Melt Temperature	280 °C
	Min. Mold Temperature	80 °C		Recommended Mold Temperature	100 °C
	Ejection Temperature	130 °C		Maximum Mold Temperature	120 °C
	Transition Temperature	150 °C		Minimum Mold Temperature	80 °C
Ð	Viscosity : 7-Parameters Modified	5.86e+11		Ejection Temperature	130 °C
Ð	PVT : Modified Tait Equation	0.0008722		Thermoset Conversions	Not Available
	Solid Density	1198 Kg/m3		Transition Temperature	150 °C
	Specific Heat : Constant	2100 J/(Kg-K)	÷	Viscosity : 7-Parameters Modified	5.86e+11
	Thermal Conductivity : Constant	0.214 W/(m-K	Ŧ	PVT : Modified Tait Equation	0.0008722
Ð	Elastic Modulus : Constant	2400 2400		Density	1198 Kg/m3
	2022			2023	

Several user interface enhancements to the Plastics Materials Database Manager improve its usability.

Display enhancements:

- Gray color border and gray gridlines replace the yellow color border and gridlines to improve the readability of the material properties.
- The Elastic Modulus, Poisson's Ratio, and Thermal Expansion Coefficient plots are removed.
- The material's polymer family and manufacturer name are removed from the title bar and are listed in separate material property fields.

Material properties enhancements:

• The database stores detailed fiber and filler properties for filled materials. You can view these properties under the **Filler Properties** section.

☐ Filler Properties	
Filler 1	
Description	Glass Fiber
Filler Percentage	55 %
Aspect Ratio	20
Initial Length	Not Available
Average Diameter	Not Available
Density	2500 Kg/m3
Specific Heat	700 J/kg.K
Thermal Conductivity	1 W/m.K
Elastic Modulus (E1)	70000 MPa
Elastic Modulus (E2)	70000 MPa
Tensile Strength (σ1)	3500 MPa
Tensile Strength (σ2)	3500 MPa
Poisson's Ratio (v12)	0.29
Poisson's Ratio (v23)	0.29

• In addition to the material's solid density, the database stores its melt density. You can view the material densities under the **Density** section.

1	Viscosity PVT	Specific Heat Thermal Conductivity
7	Shear Relaxation Modulus Curin	g Model Polymer-Material Parameters
⊡	Density	990 780
	Solid Density	990 Kg/m3
	Melt Density	780 Kg/m3

- The database stores the following Thermoset material properties, if they are available from the manufacturers:
  - PVT data for the cured and uncured states.
  - Heat of Reaction and Induction Time Constants.
  - Ejection Conversion and Initial Conversion.

The solver is also enhanced to account for these additional material properties (when available) to improve the accuracy of Fill, Pack, and Warp simulations.

The classifications of the material's **Data Source and Information** field are revised.

Classification	Description
Manufacturer Name	Name of the manufacturer who provided the grade-specific material data.
Manufacturer Name; Generic PVT Data	The Pressure Volume Temperature (PVT) data are generic, while the rest of the data are grade-specific and provided by the material manufacturer.
Manufacturer Name;*to be phased out	The material manufacturer is planning to phase out the specific material grade.

Classification	Description
SIMPOE	The material data are from a legacy source and are not updated.
SIMPOE; Generic PVT Data	The material data are from a legacy source and have been updated from constant density to generic PVT data to improve the accuracy of Fill, Pack, and Warp simulations.

### Summary and Report



You can access the **Summary** and **Report** features from the PlasticsManager independently. In addition, there are several enhancements for the **Summary** and **Report**.

• To access the Summary:

In the PlasticsManager, right-click **Results** and click **Summary**.

• To access the Report:

In the PlasticsManager, right-click **Results** and click **Report**.

Enhancements to the **Summary** feature include:

- Quick access to key results.
- Improved content organization into three sections for the Model tab:

**Study Characteristics** Lists the following features: study name, injection process, number of injection units, analysis mesh procedure, and the simulation sequence.

•

Injection Unit Setting	<b>s</b> Lists the following features for each injection unit: polymer material and the material's melt, mold, transition, and ejection temperatures.
Physical Attributes	Lists the model volume, mass, and overall dimensions.
Addition of the plastic pa	rt's mass at the end of packing in the Pack tab.
Summary✓ ×	
Model Fill Pack	
X-dir. Clamping Force =	467.2219 Tonne
Y-dir. Clamping Force =	= 1504.9500 Tonne
Z-dir. Clamping Force =	2194.3810 Tonne
Max. real temperature	236.6219 °C
Max. bulk temperature =	≥ 237.5223 °C
Max. shear stress =	0.7938 Mpa
Max. shear rate =	75.3348 1/sec
Max. residual stress =	- 194.3600 Mpa
Mass at the end of packing =	2958.98 (g)

Enhancements to the **Report** feature include:
Updated Microsoft Word report templates. The **Classic** and **Light** report templates of previous versions are replaced with a single template that has a more modern look

and feel.
Additional formats are available for saving a report. In addition to Microsoft Word, you can save a report as a Microsoft PowerPoint or HTML document. The default option All (HTML, MS Word, MS PowerPoint) saves a report to all three formats.

#### SOLIDWORKS Plastics

Report Genera	ator						×
Header Info	Description	Generate Image File	Save Options				
Save as	s:	All (HTML, MS Word	MS PowerPoint	)	~		
Save to	:	All (HTML, MS Word, MS Word MS PowerPoint	MS PowerPoint	)			
						ОК	Cancel

# 22

## Routing

This chapter includes the following topics:

- Routing Library Manager Enhancements
- Complex Splices
- Displaying the Cross Section of a Harness Segment
- Flatten Route Improvements
- Reorienting Connectors in Flattened Routes

Routing is available in SOLIDWORKS<sup>®</sup> Premium.

## **Routing Library Manager Enhancements**

Add	1	or more Optional 🕜 🛢		
Route Point(RPoint) is only a fitting i.e. drop the flange in	🕅 Close Wizard	to use the flacos as an inline	×	
Point Configuration Add all CPoints	Close Routing Closes routing	Component Wizard component wizard & returns to routin	g library manager.	
) Do not connect CPoint ) Select CPoints	Close Routing I Closes routing	Library Manager component wizard & closes routing lit	orary manager.	
	Restart Wizard Restart the wi	CPoint Type		
		<ul> <li>Default CPoint</li> </ul>	Status	Modify
	Do not show a	CPoint with Circuit information     CPoint from Manufacturer part	1 Required	0 ×

The Routing Library Manager has improved ergonomics and usability, several enhancements have been done:

- Improved discoverability and guidance in the user interface by tooltips.
- Better management of the connection points and Mate References (create, edit, and delete workflows).
- Improved saving and canceling of modifications.

## **Complex Splices**



A complex splice has multiple circuits to which you connect wires or cable cores.

To connect wires or cable cores to a complex splice, you must insert the CPoints, and the RPoints if the wire crosses the splice, corresponding to the different circuits.

You can use complex splices in a manual routing or using the From-To list.

The complex splices are not supported in the flattening.

## Displaying the Cross Section of a Harness Segment



You can access the cross section of a harness segment.

In the Electrical Attibutes PropertyManager, select **Show Cross Section** to view a graphical cross section of the harness segment. The formula to calculate the diameter of this segment takes into consideration if it contains a cable.

#### Flatten Route Improvements

* * 🔁 🖹 🖗 🔶	
Flatten Route ③	Image: Flatten Route Items     ⑦       ✓     ×
Flattened Route Drawing.	Message ^
Flatten Type	Select the items to be displayed in the drawing view
Annotation	Tables
O. at akiy     Connector Table     Connector-table     Display Connector Balloons     Display Wire Balloons     Display Wire Balloons	Cable Length Note Leaders Component Reference Manufacturer(Component Name)
V	Accessory Components

Improvements to the flatten route design deliver higher-quality flatten route drawings.

Improvements relate to the following items:

- The Auto-Balloon option applies to balloons for connectors and wires
- When you modify a connector table, you can apply this change to all existing connector tables
- The Show/Hide Flatten Route Items command is available in the Electrical toolbar
- In the **Flatten Route Items** PropertyManager, you can select **Manufacturer** (component name) to split the text displayed for the component in the flattened route drawing

**Reorienting Connectors in Flattened Routes** 

👋 🔲 🕺 🗢		• 🖾 History
Fdit Flattened Route	۲	Sensors     Annotations     Front Plane
Message	^	Top Plane
Route segment: Select an editing option to reshape the flattened layout, remove overlaps, and fit the design within the formboard.		L Origin
Connector: Select faces to reorient relative to planes.		Components     Components     Components
Item to Edit	^	• II Mates
O Route Segment		• "% Route1{->7}
Connector		• The ManufactureFlatt.
322C048H01_RLib-2@Harness_5-FanOut9July		
Maintain current Z-Offset after flattening		
Face Selection	^	
Face<1>@3220048H01_RUB-2		
Front Plane		
<b>@</b>		
Apply		

You can reorient the connectors to be parallel to a selected plane.

In the **Edit Flattened Route** PropertyManager, select a plane in **Faces and Planes**. This does not affect the **Maintain 3D Orientation** process.



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