

SOLIDWORKS DRAWINGS TOP ENHANCEMENTS IN 2023

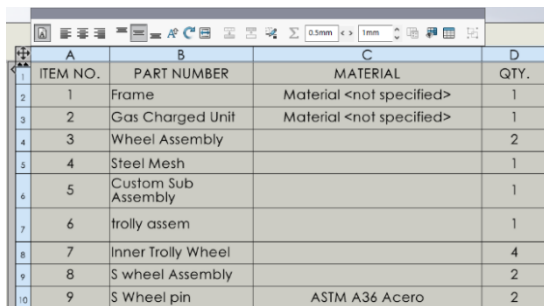
SOLIDWORKS 2023 offers a wide range of enhancements and new additions of convenient tools to improve the user experience. This blog illustrates some of the enhancements and upgrades in drawing front

DETAILING AND DRAWINGS:

Overrides for Bills of Materials

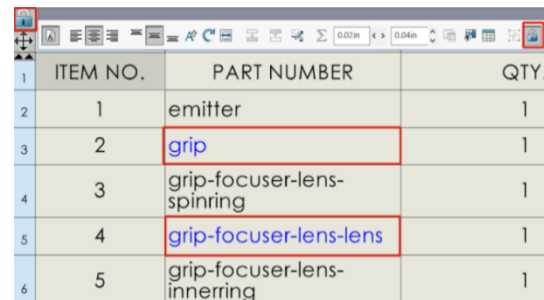
When overriding a value in a bill of materials (BOM) using Break Link, the value turns blue.

- Change the highlight colour when you override values using Break Link. Click Tools > Options > System Options > Colours, and specify the colour 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.



	A	B	C	D
	ITEM NO.	PART NUMBER	MATERIAL	QTY.
2	1	Frame	Material <not specified>	1
3	2	Gas Charged Unit	Material <not specified>	1
4	3	Wheel Assembly		2
5	4	Steel Mesh		1
6	5	Custom Sub Assembly		1
7	6	trolley assem		1
8	7	Inner Trolley Wheel		4
9	8	S wheel Assembly		2
10	9	S Wheel pin	ASTM A36 Acero	2

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	A	B	C	D
	ITEM NO.	PART NUMBER	MATERIAL	QTY.
1				
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

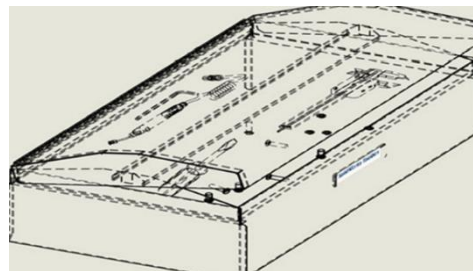
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New Feature: Displaying Transparent Models

Display transparent models in drawings with Hidden Lines Removed (HLR) and Hidden Lines Visible (HLV) modes. See through transparent bodies in HLR and HLV display states including components in assemblies and bodies in multibody parts.

Where to find:

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

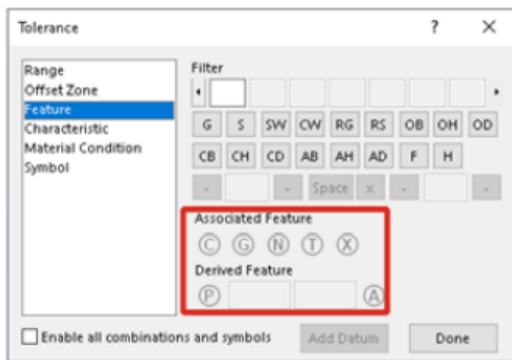


Limiting Geometric Tolerances to a Standard

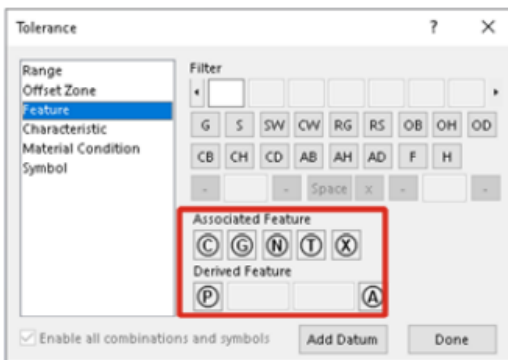
Choose all symbols for geometric tolerances or limit symbols to a standard. The image on the left has Enable/Disable options. For example, selecting the ISO standard and select Enable/Disable options per standard, limiting the symbols and values to ISO standards.

Where to Find:

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



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Filtering Bill of Materials Columns

Filtering a bill of materials (BOM) when filtering a BOM:

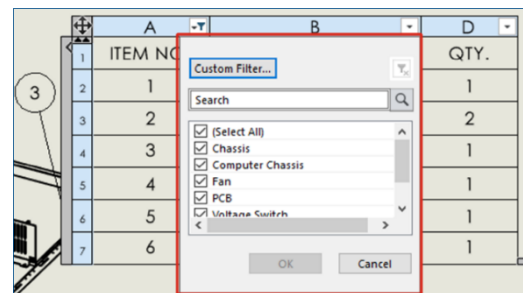
- Filter a range of data based on your criteria. Hover over the BOM column headers, there is an arrow in each column.
- Only the balloons for the filtered items appear in the view.

To filter BOM columns:

- Click in the column to filter.
- In the dialog box, select filter criteria.
- The changes indicating that the filter applies to the column.

	A	B	C	D
1	ITEM NO.	PART NUMBER	MATERIAL	QTY.
2	1	Main Frame	Material <not specified>	1
3	2	Gas Charged Unit	Material <not specified>	1
4	3	Wheel Assembly		2
5	4	Steel Mesh		1
6	5	Custom Sub Assembly		1
7	6	trolley assem		1
8	7	Inner Trolley Wheel		4
9	8	S wheel Assembly		2
10	9	S Wheel pin	ASTM A36 Acero	2

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Evolution thro' Knowledge

Component Name Display

When specifying display options for component names in a drawing, the names in the Property Manager and View Palette match those in the Feature Manager Design tree.

Where to find:

1. In the Feature Manager design tree, right-click the drawing name and click Tree Display > Component Name and Description.
2. Under Primary, specify an option:
 - Component Name
 - Component Description Based on the selection under Primary display options:
 - The Model View and Standard
3. View Property Managers update the Open documents field.
 - The View Palette updates.

