



3DEXPERIENCE®

# WHAT'S NEW

## SOLIDWORKS PDM 2021



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# SOLIDWORKS PDM

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Topics:

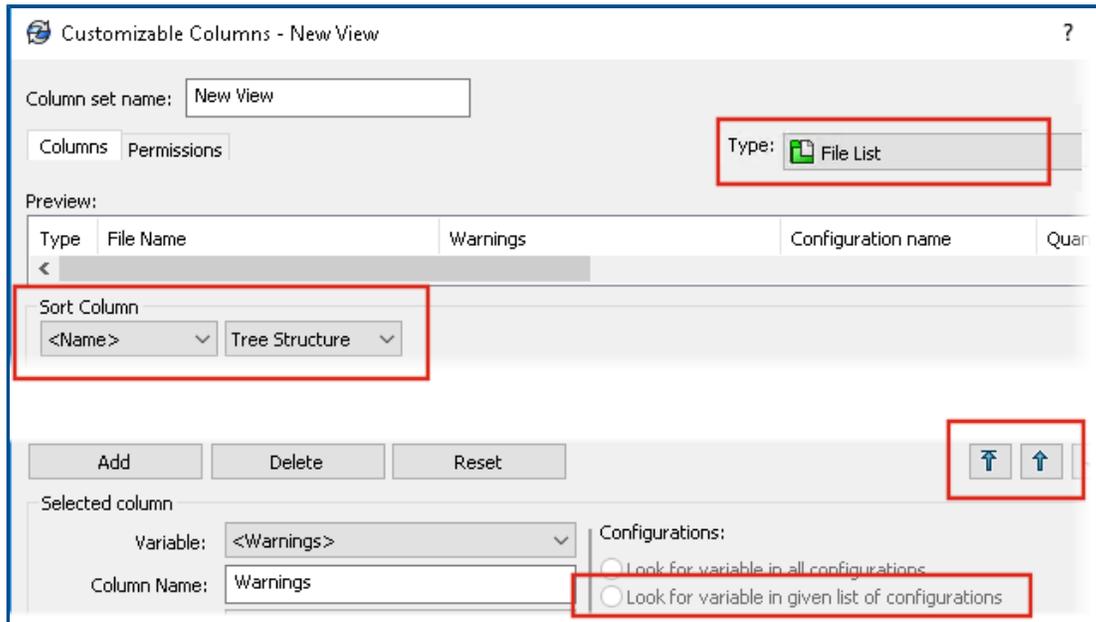
- **Customizing Columns**
- **Supporting Cut List References in Computed Bill of Materials**
- **SOLIDWORKS PDM File Explorer Enhancements**
- **Icon Changes for Workflow States and Transitions**
- **Treehouse View in the Where Used Tab**
- **SOLIDWORKS PDM Performance Improvements**
- **Displaying Derived Part References**
- **Using Bill of Materials Options Defined in SOLIDWORKS**

	<p><b>Video: What's New in SOLIDWORKS 2021 - SOLIDWORKS PDM</b></p>
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SOLIDWORKS® PDM is offered in two versions. SOLIDWORKS PDM Standard is included with SOLIDWORKS Professional and SOLIDWORKS Premium, and is available as a separately purchased license for non-SOLIDWORKS users. It offers standard data management capabilities for a small number of users.

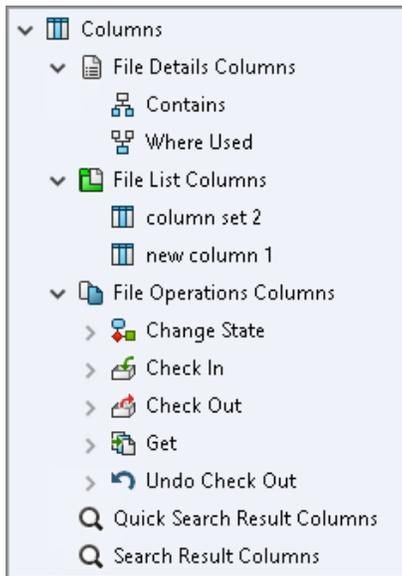
SOLIDWORKS PDM Professional is a full-featured data management solution for a small and large number of users, and is available as a separately purchased product.

## Customizing Columns



SOLIDWORKS PDM offers flexibility and control for column sets. You can assign multiple column sets to users or groups for all column set types. In SOLIDWORKS PDM File Explorer, you can view the file list based on assigned column sets.

- In the Administration tool, the **Columns** node contains column set types that list column sets.



- The following table describes the column set types that you can define:

Type	Available In
File Details	<ul style="list-style-type: none"> <li>• Contains tab</li> <li>• Where used tab</li> </ul>
File Operations	<ul style="list-style-type: none"> <li>• Change state dialog box</li> <li>• Check in dialog box</li> <li>• Check out dialog box</li> <li>• Get dialog box</li> <li>• Undo check out dialog box</li> </ul>

For Search Result columns, you can assign multiple column sets through a search card.

- In the SOLIDWORKS PDM File Explorer, to view and switch between column sets, right-click any column header or empty space and select **Column Sets**. You can also change the order, adjust the width, and sort by column for each column set.

User customizations for column width and positions that were made on SOLIDWORKS PDM clients in a prior release are not available in the SOLIDWORKS PDM 2021 client.

- If you have the required administrative permissions, you can add and remove columns directly in the file details and file operations user interface. Right-click any column header, select **Column Sets**, and then select **My Columns** to specify it as the active column set. You can then use the **Columns** menu to add or remove columns.

This column set is specific to the user's login and the client machine.

You must have the following administrative permissions to access **My Columns**:

- **Can view and modify My Columns in File Details**
- **Can view and modify My Columns in File Operations**

## Configuring Column Sets

The Customizable Columns dialog box has new features that let you configure column sets.

### Columns Tab

Option	Description
 and 	The move to top and move to bottom controls let you arrange the columns.
<b>Sort Column</b>	Defines a default sort column and sort direction for each column set.

Option	Description
<b>Reset</b>	Restores columns to the default for the column set type.
<b>Look for variable in given list of configurations</b>	Lets you enter a configuration name and add it to the list of configurations. <div style="border: 1px solid gray; padding: 5px; margin-top: 5px;">Available only for <b>File List</b> column sets.</div>

## Permissions Tab

Option	Description
<b>View</b>	Assigns a column set to a user or group.
<b>Preferred</b>	Assigns a column set as the preferred one for users or groups.

You can also assign user and group permissions for column sets through the user properties and group properties dialog boxes.

1. In the Properties dialog box, click **Columns**.
2. In the Columns page, select **Type**.  
A list of available column sets appears.
3. Select **View** to assign permission and select **Preferred** if you want to specify it as the preferred column set.

## Supporting Cut List References in Computed Bill of Materials

Bill of Materials - BOM

Bill of materials name: BOM Type: Bill of Materials

Include derived part references

Include cut list references

Weldment Cut list

Weldment BOM

Preview:

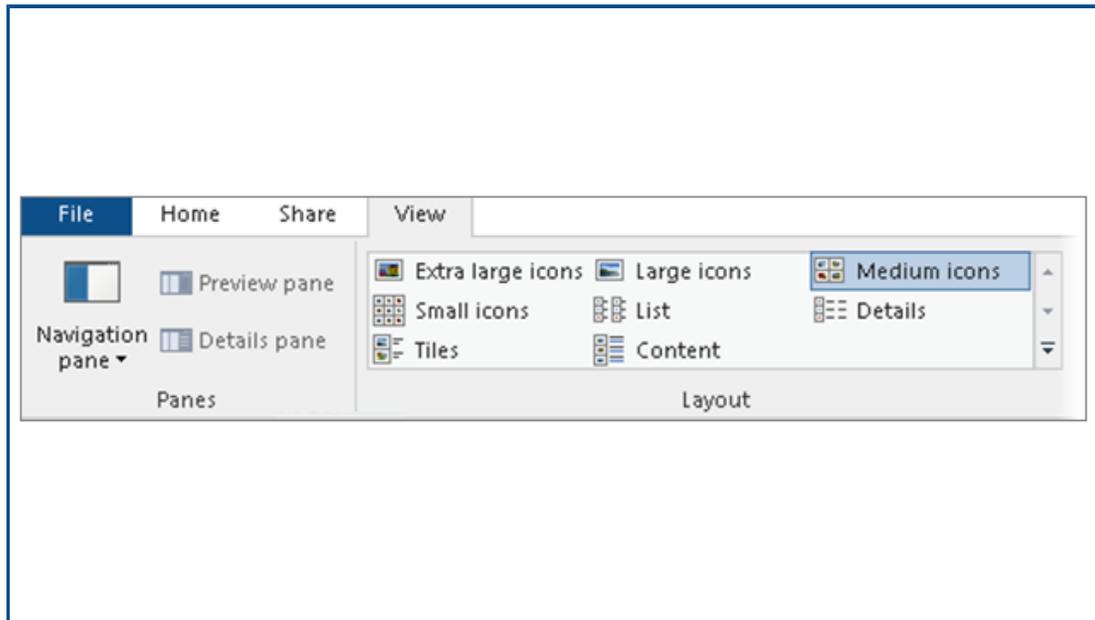
File Name	Configuration	Part Number	Qty	State	Description
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You can configure a computed BOM to include cut list references and then export it to an XML file.

In the Administration tool, in the Bill of Materials dialog box, select **Include cut list references**. You can base the cut list references on the **Weldment cut list** or the **Weldment BOM**.

If a cut list item is defined as **Exclude from cut list** in SOLIDWORKS, it is not displayed in the computed BOM.

## SOLIDWORKS PDM File Explorer Enhancements



The ribbon control of Microsoft® Windows® File Explorer is also available in SOLIDWORKS PDM File Explorer.

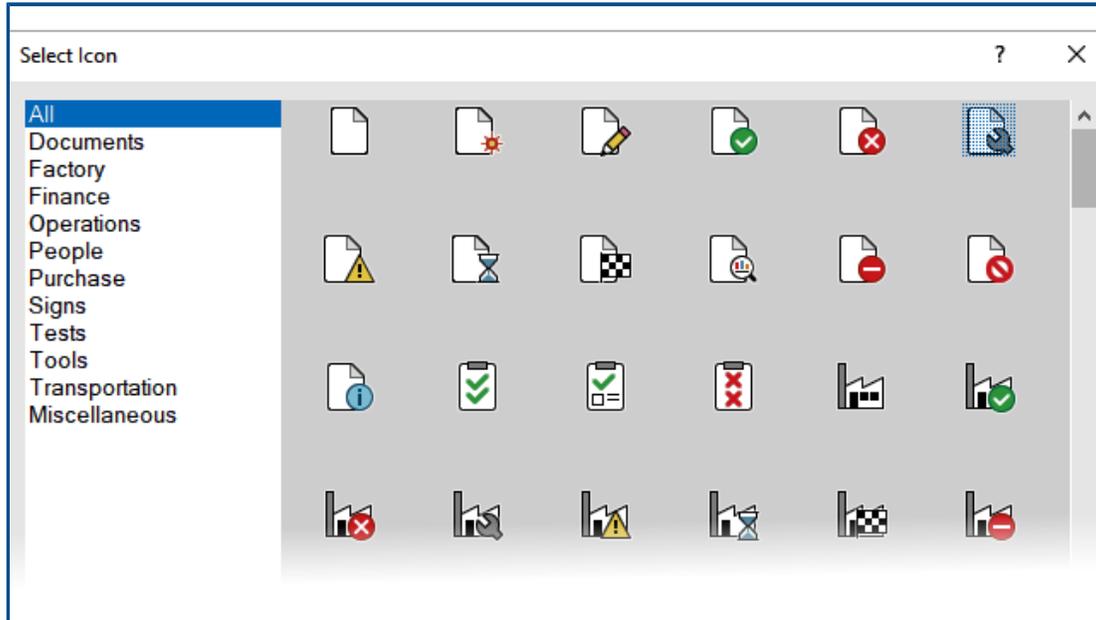
Currently, in SOLIDWORKS PDM File Explorer, only View commands and few other commands of the ribbon control are supported.

- In SOLIDWORKS PDM File Explorer, you can use the **Quick Access Toolbar** to easily access commonly used commands. In the Administration tool, right-click a user or group and click **Settings**. In the left pane of the Settings dialog box, click **Quick Access Toolbar** and specify the commands.
- The file list and the search result lists support all Windows File Explorer view types except for Content.
- Workflow state icons beside the state name in the file list view make it easier to identify the state of the file.
- Click  to change the **Language** and view the **About** information of the installed SOLIDWORKS PDM product.
- Use **Back** and **Forward** from the address bar and return to the previously selected file.
- Use keyboard shortcuts for commonly used commands:

Action	Keyboard Shortcut
Check In	<b>Ctrl + I</b>
Check Out	<b>Ctrl + O</b>
Undo Check Out	<b>Ctrl + U</b>

Action	Keyboard Shortcut
Change State	<b>Ctrl + T</b>

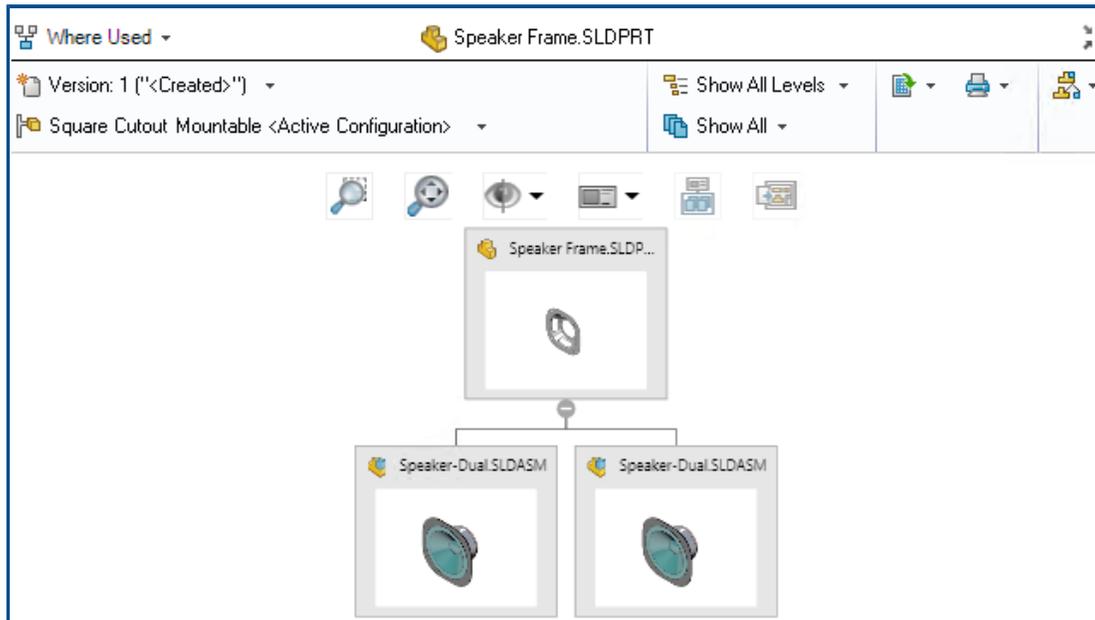
### Icon Changes for Workflow States and Transitions



SOLIDWORKS PDM has new icons for workflow states and transitions. The colors and appearance of existing icons are improved.

In the Administration tool, open a workflow, and in the Properties dialog box of a state, click **Change**. In the Select Icon dialog box, icons are grouped under categories that make the selection process easier.

## Treehouse View in the Where Used Tab



SOLIDWORKS PDM 2021 extends Treehouse view in the Contains tab to the Where Used tab. This helps you to view all the parent relationships of the selected child in a hierarchical and graphical format.

Available in SOLIDWORKS PDM Professional only.

## SOLIDWORKS PDM Performance Improvements

SOLIDWORKS PDM 2021 has improved performance of file-based operations and related workflows.

- **Add Files, Check-in, and Change State**

Improvements in system performance help you to quickly open, add, check in, and change the state of files with large reference structure. The Add Files operation is between 1.5 and 3 times faster. The Check-in and Change State operations are approximately 25% faster.

The level of improvement can vary depending on the number of files, network bandwidth, and CPU cores.

The Check-in dialog box itself displays much more quickly (in seconds rather than minutes) for very large assemblies or parts with a high number of configurations.

- **Destroy**

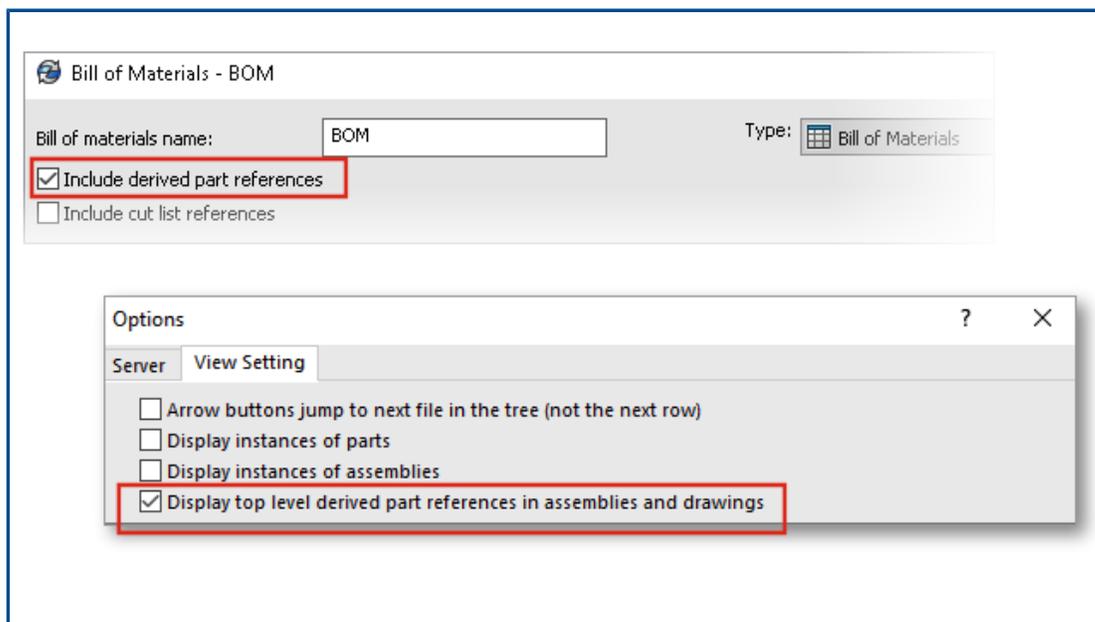
You can more quickly destroy files or folders. The operation is many times faster when you are destroying a large number of files.

- **Other Operations and Workflows**

Performance is significantly improved for the following operations:

- For folders containing a very large number of files:
  - Check-in and Check-out of a file
  - Adding a new file by dragging or by copying and pasting
  - Creating a new file from a template
- For assemblies with a large number of components at a single level, when working in SOLIDWORKS:
  - Opening files
  - Saving files
  - Switching windows
- Editing a target file name in the Copy Tree dialog box in a high latency environment
- Using Copy Tree when a user setting is added to clear variable values

## Displaying Derived Part References



The support for the derived part references is improved and you can view them in the file view tabs and SOLIDWORKS PDM Add-in.

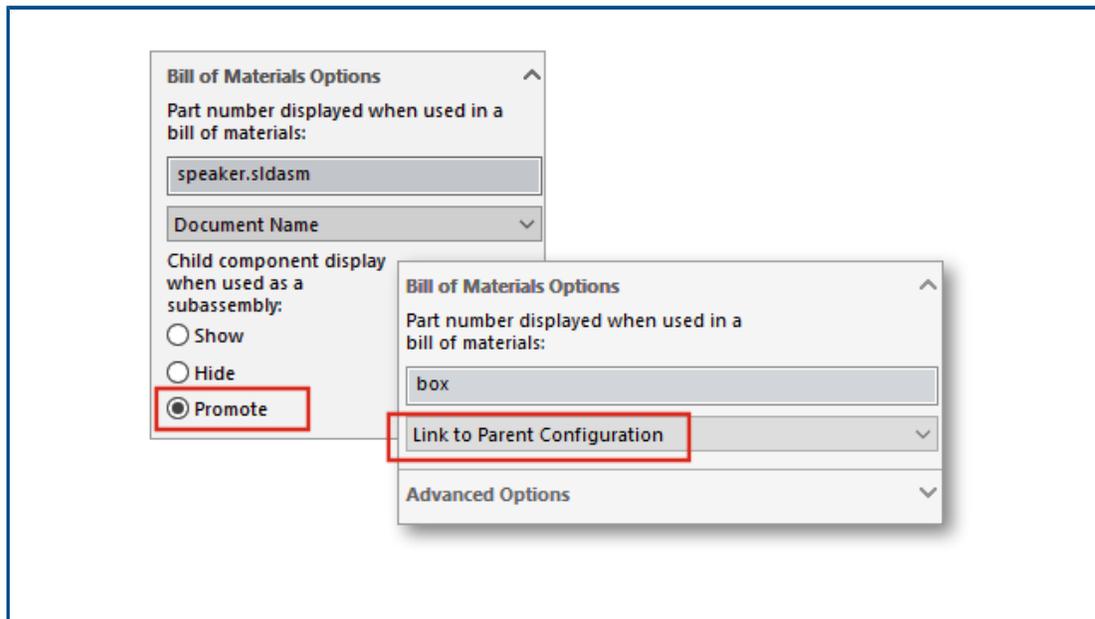
- Computed BOM
  - You can configure a computed BOM to include derived part references. In the Administration tool, open the Bill of Materials dialog box, and select **Include derived part references**.
- Contains and Where Used tabs
  - You can view derived part references in the Contains and Where Used tabs regardless of the selected configuration. Previously, derived part references were displayed only when you selected the **Do not show configurations** option for the configuration.

- SOLIDWORKS PDM Add-in

In SOLIDWORKS, click **Tools > SOLIDWORKS PDM > Options**. In the View Setting tab, select **Display top level derived part references in assemblies and drawings**. You can view the top-level derived part references in the reference structure for assemblies and drawings.

You can also view 3D Interconnect derived part references in the reference structure of the parent part file.

## Using Bill of Materials Options Defined in SOLIDWORKS



SOLIDWORKS PDM uses the Bill of Materials component display settings that are specified in SOLIDWORKS.

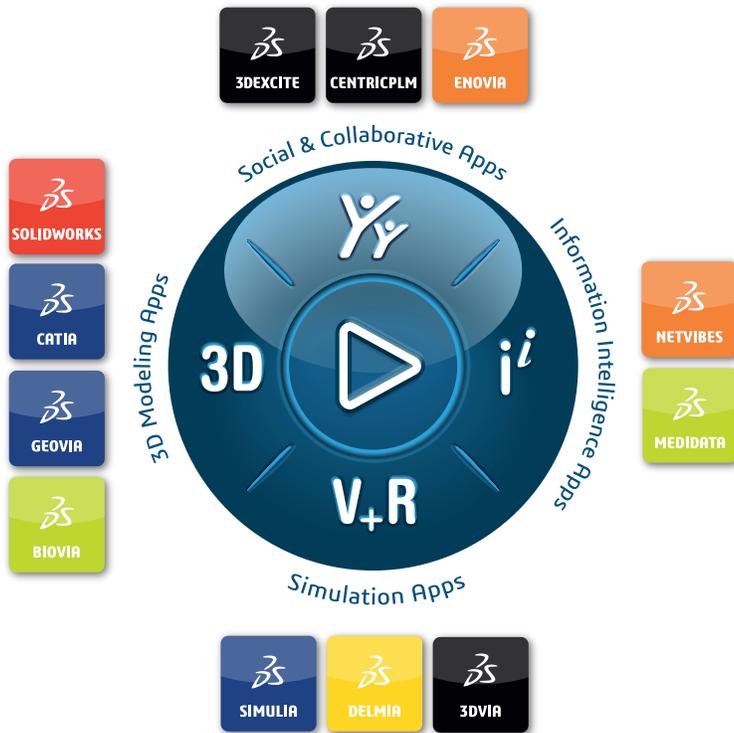
In SOLIDWORKS, in the Configuration Properties PropertyManager, the parameters that you specify in the Bill of Materials Options are also supported in computed BOMs of SOLIDWORKS PDM.

- In SOLIDWORKS, the **Promote** option for **Child component display when used as a subassembly** dissolves the subassemblies in the BOM and displays the child components. The computed BOMs follow this behavior for subassembly references that are defined to **Promote**.

This option applies only to computed BOMs. The assembly is included as a reference in the Contains and Where Used tabs.

- In SOLIDWORKS, the **Link to Parent Configuration** option for **Part number displayed when used in a bill of materials** sets the configuration to use the same part number as the parent configuration. If you define this option, the computed BOM uses the part number of the parent configuration.

This option is available only for derived configurations.



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