



# WHAT'S NEW

## SOLIDWORKS PDM 2020



**DS SOLIDWORKS** / Visualize



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# SOLIDWORKS PDM

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Topics:

- **Defining Child Reference State Conditions**
- **Search User Interface**
- **SOLIDWORKS PDM Performance Improvements**
- **Web2 Enhancements**
- **Using AND, OR, and NOT Operators in Search**
- **Searching in Multiple Variables**
- **Resizable Dialog Boxes**
- **Logging Out from Windows Explorer**
- **Reordering Columns in Search Results**
- **Scaling the Paper Size in Print Task**
- **Warning Users When Transitioning Files**

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.

## Defining Child Reference State Conditions ★

	<p><b>Video: What's New in SOLIDWORKS PDM 2020 - Workflows</b></p>
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You can define conditions on the state of immediate child references to control the parent file transition.

When you define conditions, you can transition the parent file if:

- Its child references are not selected for transitioning with the parent file and are already in a state that meets the defined condition.
- Its child references are selected for transitioning with the parent file and, after transition, are in a state that meets the defined condition.
- The warning, **Child references do not meet the state conditions**, is not set to blocking.

By default, when this warning is not set to blocking, only those parent files that fulfill the child reference state condition are selected for transition.

To change the state of the parent file, in the Do Transition dialog box, select **Change state**.

The warning is set to blocking by default for new and upgraded users.

This helps avoid situations such as approving an assembly when the referenced parts are not approved or when the assembly uses obsolete parts.

#### To define child reference state conditions:


1. In the Administration tool, click a workflow transition.
2. In the transition's Properties dialog box, in the Conditions tab:
  - a) Select **<Click here to add a condition>**.
  - b) In the conditions list, select **Child Reference State**.
  - c) Expand **Comparison** and select a comparison operator.
  - d) Under **Value**, select the state if you selected the **Text Equal To** or **Text Not Equal To** comparison operator.

For the **Text Compare** operator, enter the value in the *workflow name.state name* format. You can also use wildcard characters to specify the pattern.

- e) Under **Configuration / Path**, enter the wildcard characters to specify the file name or file path pattern.  
Leave **Configuration / Path** blank to apply the condition to all immediate child reference files.

3. Click **OK** and save the workflow.

## Search User Interface ★

	<p><b>Video: What's New in SOLIDWORKS PDM 2020 - Search</b></p>
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You can perform a search with the quick search box in the SOLIDWORKS PDM File Explorer without setting up a search card.

The quick search box is available in the SOLIDWORKS PDM File Explorer as a part of the SOLIDWORKS PDM menu bar.

With the quick search box, you can:

- Define your search string using sample text.
- Select a string from a list of the previous five searches when you click in the quick search box.

- Use AND, OR, and NOT operators in the search string.
- Click in the quick search box, type, and select any string from the list of previously matching searches.
- Use the following keys:

<b>Ctrl + F</b>	Selects the search field
Arrow keys + <b>Enter</b>	Selects the search string
<b>Enter</b>	Starts the search
<b>Esc</b>	Closes the search

## Customizable Variables - New Variable List Dialog Box

You can use this dialog box to specify the variables that you can search from the quick search for specific users or groups.

### To display Customizable Variables dialog box:

1. In the Administration tool, expand **Lists**.
2. Right-click **Quick Search Variable List** and click **New List**.

## Name

Enter the name of the new variable list.

## Variables

<b>Add</b>	Adds the administrator-defined variables. <div>You can add up to five variables.</div>
<b>Delete</b>	Deletes the selected variable.
Up and down arrows	Moves the listed variables up and down.

## Selected Variable

<b>Variable</b>	Displays a selected variable and lets you select a variable to replace the selected variable.
<b>Name</b>	Displays the name of the selected variable.

## Users

Lists users and lets you select the users who can select the variables in quick search.

## Groups

Lists groups and lets you select the groups whose members can select the variables in quick search.

## Using Quick Search

You can search faster using the quick search. The quick search box is available in the SOLIDWORKS PDM menu bar.

### To use quick search:

1. In the SOLIDWORKS PDM File Explorer, navigate to the vault or folder to search.
2. Press **Ctrl + F** to access the quick search box.
3. Click the list arrow in the quick search box to specify the search settings.

Search in All Folders		Q	▼
<b>Search For</b>			
✓	File/Folder Name		
	Keywords		
	Description		
	FileName		
✓	Comment		
<b>Search In</b>			
	Current Folder		
	Current Folder & Subfolders		
●	All Folders		
<b>Search Variables In</b>			
	Latest Version		
	All Versions		

4. From **Search for**, **Search in**, and **Search variables in**, select the appropriate search categories.

You can select multiple options to search for.


<b>Search for</b>	<b>File/Folder name</b>	Searches for the names of files or folders that match the search string.
	Administrator-defined variables	A list of administrator-defined variables. Searches for the value of selected variables in the data card fields.
<b>Search in</b>	<b>Current Folder</b>	Searches for the string in the current folder.
	<b>Current Folder &amp; Subfolders</b>	Searches for the string in the current folder and its subfolders.
	<b>All Folders</b>	Searches for the string in all folders.
<b>Search variables in</b>	<b>Latest version</b>	Limits the search to the latest version.
	<b>All versions</b>	Searches for the string in all versions.

The **Latest version** and **All versions** options are available based on the following settings in the Administration tool:

- Whether variables are selected in the Quick Search Variable Lists or not.
- Selection of **Always work with latest version of files**.

5. In the quick search box, enter the search strings and press **Enter**.  
The search results appear in the same window.
6. Click **Close Search** to clear the search results and to return the file list.

## SOLIDWORKS PDM Performance Improvements

	<p><b>Video: What's New in SOLIDWORKS PDM 2020 - Performance</b></p>
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You can perform the following actions faster:

- Logging into the vault when you set a large number of files or folders for automatic caching.
- Viewing the System History of a vault that includes a large number of items.
- Opening a file from a folder that includes a large number of files.
- Creating a new file or subfolder in a folder that contains large number of files.

## Performance Improvements in Browsing

SOLIDWORKS PDM loads the data in the background, which improves the browsing responsiveness.

Browsing folders that have a large number of files is faster due to:

- Faster database queries for custom columns.
- Background loading and incremental loading of the data.

When you double-click a folder, SOLIDWORKS PDM loads data in this order:

1. All subfolders synchronously. You can browse subfolders while the remaining data loads in the folder.
2. Files with standard columns in the file list.
3. Information in custom columns. You can add more custom columns and maintain the same browsing speed.

When you select a file and switch between the Bill of Materials, Contains, and Where Used tabs, SOLIDWORKS PDM loads the data in the tabs in the background. When you switch to a different file or folder, the background loading stops and loading of the new file or folder starts.

Performance improves significantly for folders with a large number of files or for database servers with high latency.

## Performance Improvements in the SOLIDWORKS PDM Add-in

The SOLIDWORKS PDM task pane tree refreshes faster and the commands in the SOLIDWORKS PDM toolbar are correctly enabled in the task pane instantaneously when you select a file.

- Data in the task pane tree loads in the background.
- When you select a file in the task pane tree, calls to the database are eliminated.

In the FeatureManager design tree or in the graphics area, when you right-click a file and select **SOLIDWORKS PDM**, the menus are correctly enabled.

When you select a file in the graphics area or in the FeatureManager design tree, the file listed in task pane tree is selected and the commands in the SOLIDWORKS PDM toolbar are correctly enabled.

## Web2 Enhancements

	<b>Video: What's New in SOLIDWORKS PDM 2020 - Web2</b>
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When you drag a folder to the file list, the Check In File Structure dialog box opens automatically.

In the **Details** section of the data card, in **Comment**, you can see version comments.

### Bill of Materials Tab in Web2 ★

SOLIDWORKS PDM Web2 lets you see a read-only view of the Bill of Materials.

In the large screen layout, the Bill of Materials tab is available in the file detail view. The supported BOM types are:

- Computed BOMs
- Weldment BOMs
- Weldment cut lists
- SOLIDWORKS BOMs

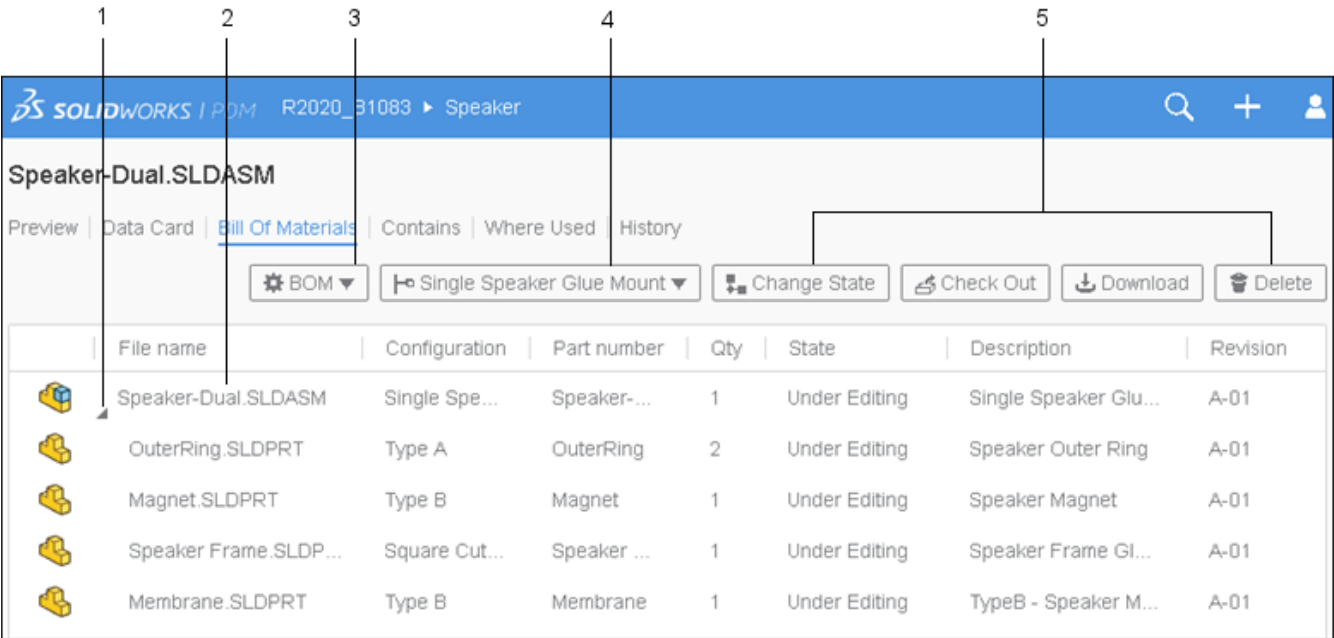
In the small screen layout, the Bill of Materials tab is available from the ellipsis menu. The tab contains two customizable columns.

#### Bill of Materials Tab in Web2

The Bill Of Materials tab displays the bill of materials of SOLIDWORKS PDM items or of a SOLIDWORKS assembly, drawing, part, or weldment part.

#### To display the Bill of Materials tab:

In the large screen layout, click the file name and click the Bill Of Materials tab.



### Bill of Materials Tab

1	Expand or collapse	Expands or collapses the indented BOMs.		
2	Table view	<b>BOM</b>	Displays the selected item for <b>Indented</b> or <b>Top Level Only</b> .	
3	BOM type and options	Displays the name of the selected BOM at the top with the list of other BOMs and options based on the selected BOM.		
		<b>Selected BOM</b>	<b>List of BOMs</b>	<b>View option and reference version</b>
		BOM	Computed BOM, Weldment BOM, Weldment cut list, SOLIDWORKS BOM	Indented, Parts Only, Top Level Only, As Built, Latest
		Weldment BOM or Weldment cut list	Computed BOM, Weldment BOM, Weldment cut list	-
		SOLIDWORKS BOM	Computed BOM, SOLIDWORKS BOM	-
4	Configurations or sheets	Displays the name of the selected configuration or sheet for Computed BOM, Weldment cut list, and Weldment BOM.		

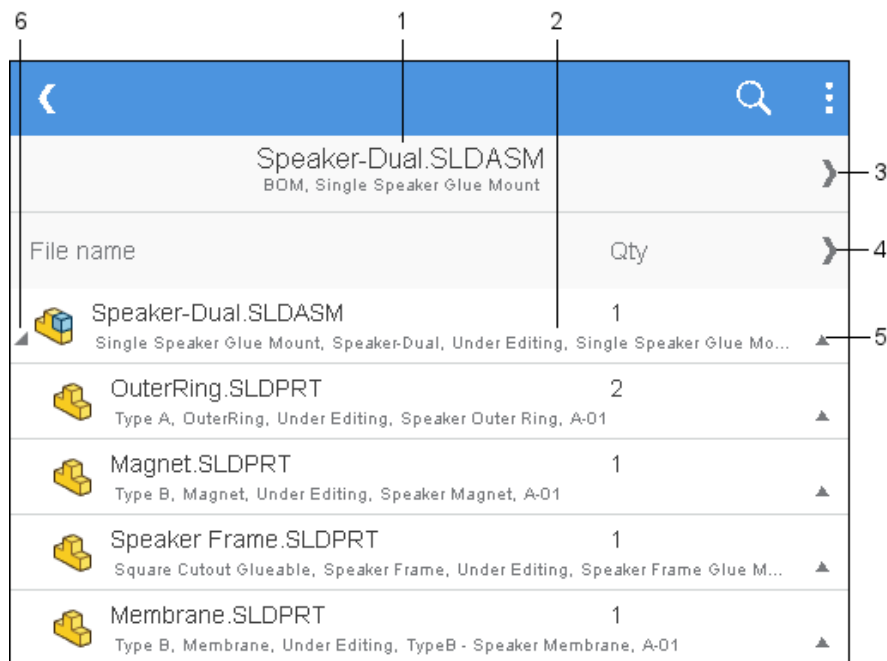
5	Actions	Lets you select options and perform actions based on user permissions: <ul style="list-style-type: none"> <li>• <b>Change State</b></li> <li>• <b>Check Out</b></li> <li>• <b>Download</b></li> <li>• <b>Delete</b></li> </ul>
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#### Bill of Materials Tab - Small Screen Layout

The Bill of Materials tab contains two columns.

#### To display the Bill of Materials tab:

In the file detail view, click the ellipsis, and click **Bill Of Materials**.



1	File name header	Displays the file name, selected BOM name, and configuration or sheet name.
2	Row details	Displays the values of two columns and comma-separated values of remaining columns.
3	Settings control	Displays the selected BOM, view option, reference version, and configuration or sheet.
4	Columns control	Lets you select the left and right columns to display.
5	Expand/collapse control for row	Expands or collapses the row to display values of other columns.

- |   |                         |  |
|---|-------------------------|--|
| 6 | Expand/collapse control | Expands or collapses the indented, multilevel computed BOMs. |
|---|-------------------------|--|
- 

## History Tab

SOLIDWORKS PDM Web2 lets you see a read-only view of the history.

In the large screen layout, the History tab is available in the file detail view. It displays the table view that includes the **Event** column and customizable columns:

- **Version**
- **User**
- **Date**
- **Comment**

Click  and select or clear the columns to display.

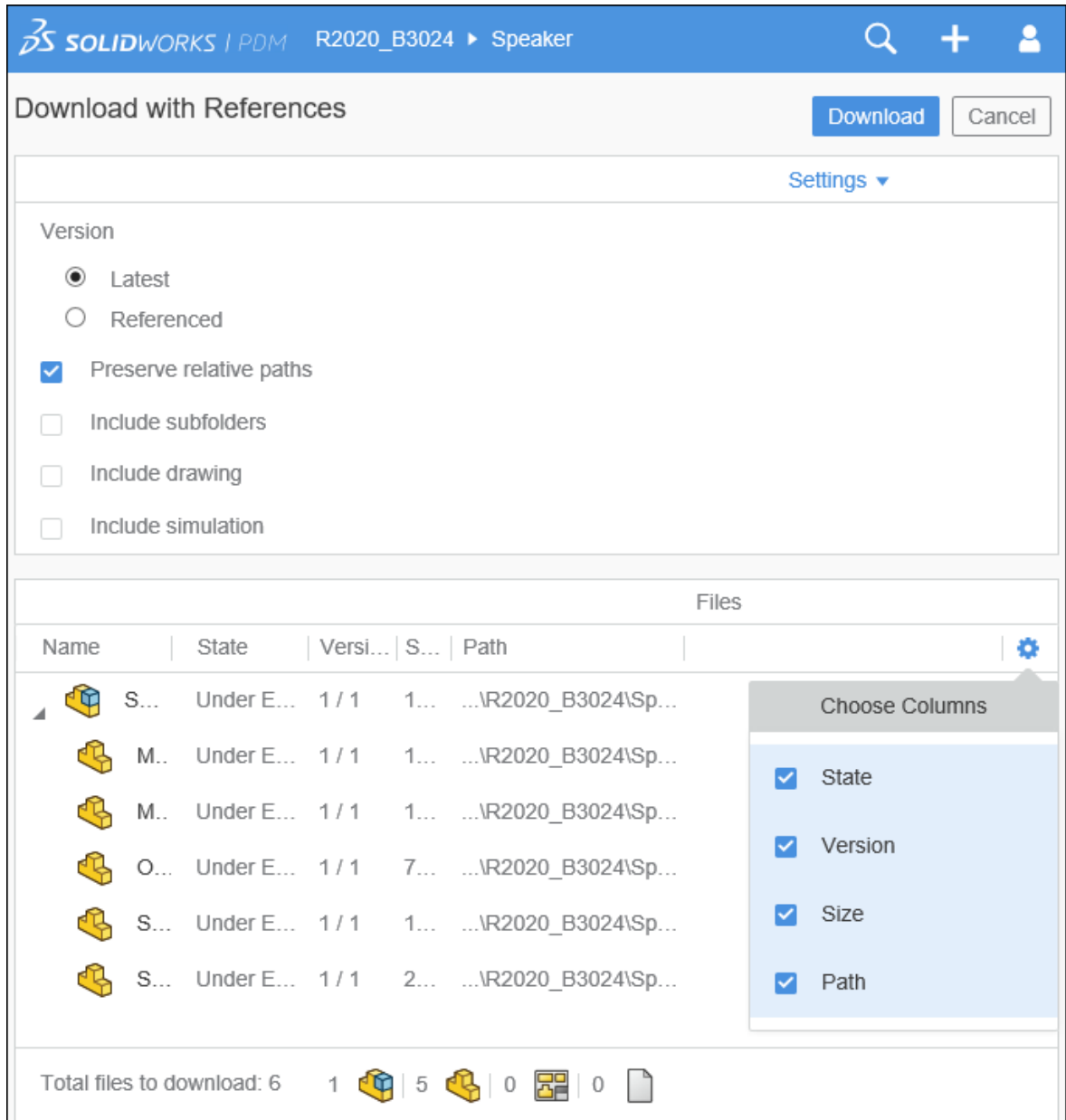
In the small screen layout, to access the History tab, in the file detail view, click the ellipsis. The History tab has:

- A list of events in descending order of date and time.
- A separate row for each event.
- Information such as **Version**, **User**, **Date**, and **Comment** in each row for each event.
- A common control to expand or collapse all rows.
- Individual controls for each row.

## List of Files to Download in Web2

SOLIDWORKS PDM Web2 displays the list of files to download.

In the large screen layout, the Download with References dialog box displays the collapsible **Settings** option and the list of files to download.

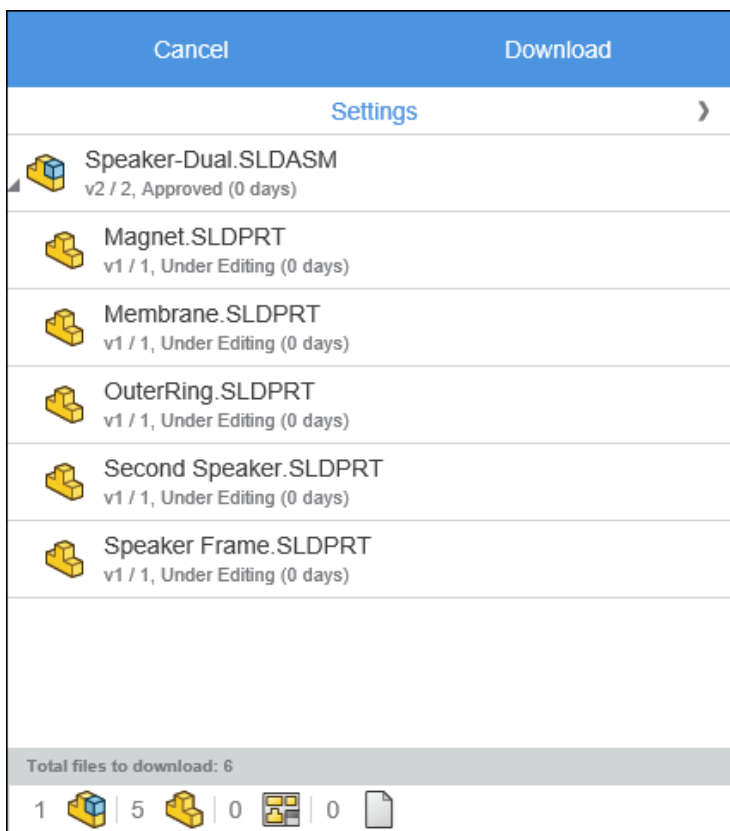


The file list displays the files based on these settings:

- **Version**
  - Latest
  - Referenced
- **Preserve relative paths**
- **Include subfolders**
- **Include drawing**
- **Include simulation**

The file list includes customizable columns such as **State**, **Version**, **Size**, and **Path**. Click and select or clear the columns to display.

In the small screen layout, the Download with References dialog box displays the list of files to download in one column.



Click **Settings** to select the appropriate options. Click **Cancel** to access the file list.

## Using AND, OR, and NOT Operators in Search ★

SOLIDWORKS PDM supports AND, OR, and NOT operators in input search criteria in edit-box type fields on search cards.

You can use these search operators in any search card, Web2 search, the SOLIDWORKS PDM Search tool, and quick search.

Rules:

- Search strings are case-insensitive.
- Operators are case-sensitive and must be capitalized.
- For variable values, the valid operators are: =, !=, <, <=, >, >=, and <>.
- Rules for using escape symbol:

Search for String with	Input String
Double quotation marks	String with an escape symbol \. For example, to search for "speaker", enter the string as \"speaker\"
Double quotation marks that are a part of an explicit search	String with an escape symbol \. For example, to search for 3" pipe, enter the string as "3\" pipe". <div style="border: 1px solid black; padding: 5px; margin-top: 10px;">The use of the escape symbol \ is optional in cases where you use a double quotation mark " as an inch constant. For example, to search for 3", you may enter the string as 3".</div>

- In case of multiple operators used in a search, SOLIDWORKS PDM evaluates the search operators in this order: (), NOT, AND, and OR.
- You can use AND, OR, and NOT search operators or their equivalent symbols &, |, ! respectively.
- You can search for TEXT, INT, FLOAT, and DATE type data.

If you do not specify the data type explicitly and the multi-variable field has:

- TEXT and non-TEXT variables, the applied data type is TEXT.
- DATE and non-DATE variables, the applied data type is TEXT.
- FLOAT and INT variables, the applied data type is FLOAT.
- You can use the AND and OR operators with or without the :. For example, for AND operator, use AND, AND:.
- Use the single colon specifier, :, when you want to search for values across multiple configurations or variables.

For example, when you enter the input string as:

- :part1 part2 part3 part4, SOLIDWORKS PDM returns the matches that may be present in different configurations.
- :!part1, SOLIDWORKS PDM returns the matches that do not have part1 in any of the configurations.
- Along with :, you can nest search strings within {}.  
For example, for the input string : {part1 part2} part3, SOLIDWORKS PDM returns only those matches where part1 part2 are found in the same configuration.
- The search field supports \* and ? as wildcard characters.
- You can use = to search for an exact string. For example, to search for a Tools string, enter the search string as =Tools.

- If a string contains more than one word, you can use "" to search for the exact string. For example, to search for `pipe tools`, enter `"pipe tools"`.

When you upgrade to SOLIDWORKS PDM 2020, the previously saved searches are upgraded to follow the new search rules while maintaining the previous behavior and search results.

## AND (&) Operator

Searches for results that include both the term before and the term after the operator.

Search String Examples	Description
<code>term1 AND term2</code> <code>TERM1 &amp; TERM2</code> <code>AND: term1 term2</code> <code>&amp;: term1 term2</code>	<p>Returns the match that includes <code>term1</code> and <code>term2</code>. Terms may appear in any order in the results.</p> <p>Examples of valid search results:</p> <ul style="list-style-type: none"> <li>• <code>term1 term2</code></li> <li>• <code>term2 term1</code></li> <li>• <code>term3 term1 term2</code></li> </ul>
<code>"(*" term1 term2 "*")"</code> OR <code>"(*" term1 term2</code>	<p>AND is the implicit operator for the SPACE character.</p> <p>Returns the match that is enclosed in the parentheses and that contains <code>term1</code> and <code>term2</code>.</p> <p>Examples of valid search results:</p> <ul style="list-style-type: none"> <li>• <code>(term1 term2)</code></li> <li>• <code>(term2 term1 term3)</code></li> </ul>
<code>"(*" "term1 term2" "*")"</code> OR <code>"(*" "term1 term2"</code>	<p>AND is the implicit operator for the SPACE character.</p> <p>Returns the match that is enclosed in the parentheses and that contains the string <code>term1 term2</code>.</p> <p>Examples of valid search results:</p> <ul style="list-style-type: none"> <li>• <code>(term1 term2)</code></li> <li>• <code>(term3 term1 term2 term4)</code> Searches for results that include the term before or after the operator or both.</li> </ul>

## OR (|) Operator

Search String Examples	Description
<code>term1 OR term2</code> Searches for results that include the term before or after the <code>TERM1   TERM2</code> <code>OR: term1 term2</code> <code> : term1 term2</code>	Returns the match that includes either or both <code>term1</code> and <code>term2</code> . Examples of valid search results: <ul style="list-style-type: none"> <li>• <code>term1</code></li> <li>• <code>term2</code></li> <li>• <code>term2 term1</code></li> </ul>
<code>"(*" term1 OR term2 "*")"</code> or <code>"(*)" (term1 OR term2)</code>	Returns the match that is enclosed in the parentheses and that includes either <code>term1</code> or <code>term2</code> or both. Examples of valid search results: <ul style="list-style-type: none"> <li>• <code>(term1)</code></li> <li>• <code>(term3 term2)</code></li> <li>• <code>(term2 term1 term4)</code></li> </ul>

## NOT (!) Operator

Searches for results that do not include the term after the operator.

Search String Examples	Description
<code>NOT TERM1</code>	Returns values other than <code>term1</code> . Examples of valid search results: <ul style="list-style-type: none"> <li>• <code>term2</code></li> <li>• <code>term3 term4</code></li> </ul>
<code>!="term1 term2"</code>	Returns values other than string <code>term1 term2</code> . Examples of valid search results: <ul style="list-style-type: none"> <li>• <code>term3</code></li> <li>• <code>term2 term1</code></li> </ul>
<code>!(2019-01-11)</code>	Returns dates other than <code>2019-01-11</code> . Examples of valid search results: <ul style="list-style-type: none"> <li>• <code>2019-02-12</code></li> <li>• <code>2017-03-11</code></li> </ul>

## Searching in Multiple Variables

You can search for a value in multiple selected variables using the Edit-box control in a search card.

In the File Explorer, when you search for a value, SOLIDWORKS PDM considers OR as the implicit operator, and returns a match if it exists in any variable value.

### To specify a multiple-variable edit-box control on the search card:

In the Card Editor, click **Edit** (Controls toolbar), and click in the card to place the edit box.

**Edit-box properties**

☒ Multi-variable

Value

Album | Approved by | Approved On | Description

Included variable

- ☐ \_SW\_Last\_Saved\_With\_
- ☒ Album
- ☒ Approved by
- ☒ Approved On
- ☐ Artist
- ☐ Assembly No.
- ☐ Attachments

Variables...

Flags

- ☐ Read-only
- ☐ Show in Explorer
- ☐ Multiline
- ☐ Updates all configurations

Default Values

Input formula

> ...

### Multi-variable

Lets you select multiple variables from the existing variables defined in the vault.

When you select the variables, the edit box displays the variables separated by | in alphanumeric order.

---

**Default Values** Lets you define the variable values.  
By default, SOLIDWORKS PDM considers the OR condition for the default values. The values can be the same or unique.

**At run time, when you edit default values with Multi-variable selected, the default values in the single variable edit box (for example, in the Cards tab) are not updated automatically.**

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## Creating a Search Card to Search Within Multiple Variables

You can create a search card that lets you search for values in multiple variables when you open it in the File Explorer.

### To create a search card to search within multiple variables:

1. Create a search card.
2. Add controls to the card.
3. In the Card Editor, click **Edit** (Controls toolbar).
4. Click in the card to place the edit box.
5. Select **Multi-variable**.
6. From the list, select the variables to include.
7. Select the appropriate flags.
8. Click **Default Values** and select the variables you selected in step 6.
9. Assign values to the variables.  
The default values appear in the card controls.
10. Save the card.

## Resizable Dialog Boxes

The following dialog boxes are now resizable:

- Edit Variable
- Serial Number - New Serial Number
- Indexing Settings
- Cold Storage Schema
- Dialog boxes in **Data Import/Export** node such as XML Variable Alias Sets, Export Rule, and Import Rule
- File Type Properties
- Customizable Columns
- Dialog boxes in **Bill of Materials** node such as BOM, Item BOM, Weldment BOM, and Weldment Cut List
- Edit Template
- Dialog boxes in **Dispatch** such as Administrate Actions and Properties

## Logging Out from Windows Explorer

You can see the name of the logged in user in the File Explorer and log out from Windows® Explorer.

To see the name of the logged in user, hover over .

**To log out from Windows Explorer:**

1. Click .
2. Click **Log Out**.

## Reordering Columns in Search Results

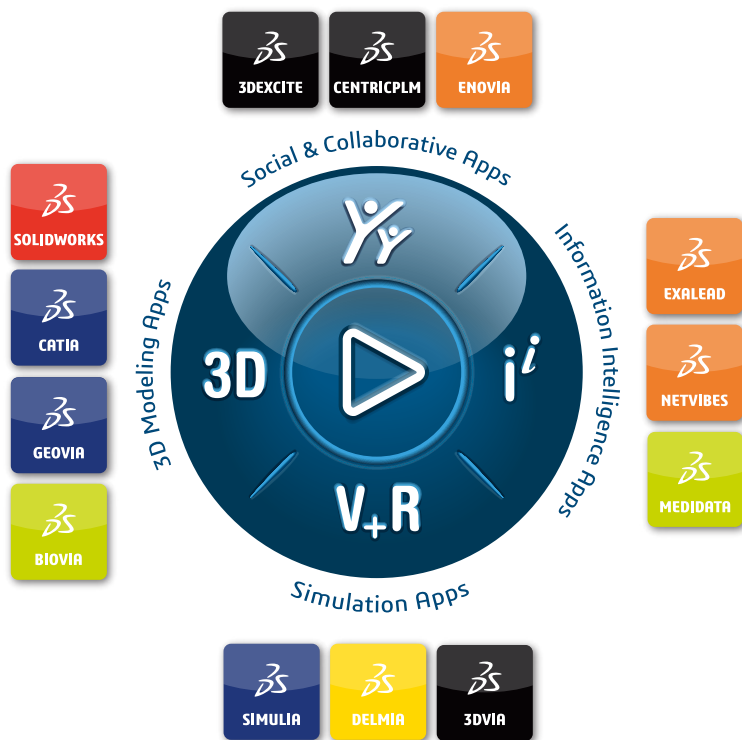
You can reorder the columns in the search results of the embedded search and the Search tool. SOLIDWORKS PDM saves the changes to the column ordering.

## Scaling the Paper Size in Print Task

When configuring the **Print** task, you can select **Scale to fit** or **Scale**. When you select **Scale**, you can select **Allow the user to change the scale** to let users change the scale at run time.

## Warning Users When Transitioning Files

For transitions that include dynamic notifications, you get a warning if you do not select any user or group to receive a notification.



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