

SOLIDWORKS 3D CAD TOP ENHANCEMENTS IN 2023

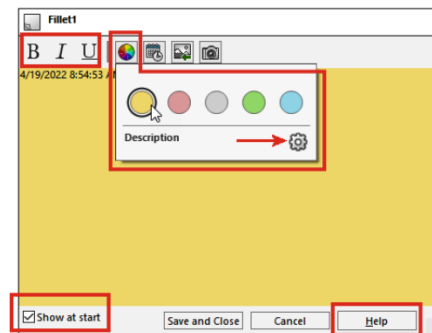
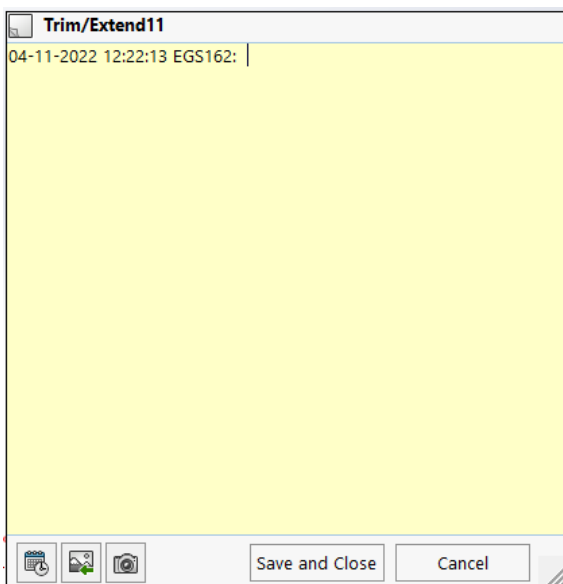
SOLIDWORKS 2023 brings a wide range of enhancements and new additions of convenient tools to improve the user experience. This blog illustrates some of the major upgrades in User Interface, Part modelling and Assembly creation.

USER INTERFACE:

Comments Enhancements

In SOLIDWORKS 2023, User can choose different background colours and add bold, italics, underline to text by clicking the Options in the Background Colours tab to open the Tools > Options > Document Properties > Model Display dialog box where you can specify the Comment background colours.

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



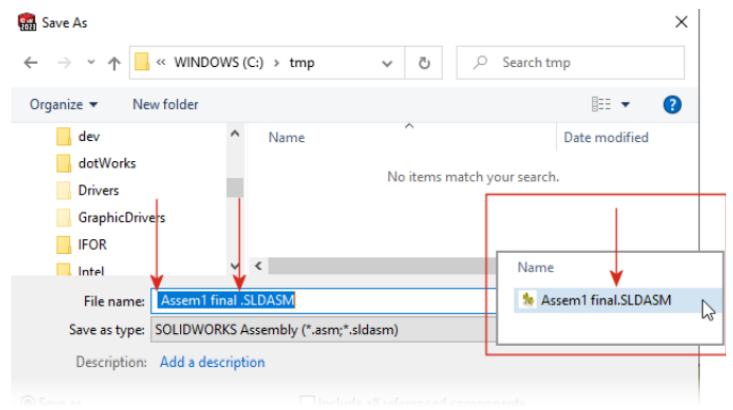
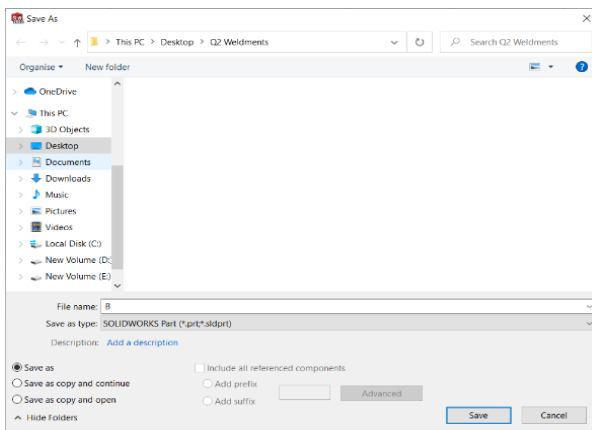
Comment tooltips shows the text to indicate that the full comment contains an image. The name of the tool tip matches the name from the Comment background colours section in the Model Display dialog box.

Enforcing Legal File Names

To ensure the compatibility with 3DEXPERIENCE platform, we cannot save new SOLIDWORKS documents with preceding or trailing spaces in their names. The software automatically removes the preceding and trailing spaces when you save the file.

This enforcement applies in following cases like:

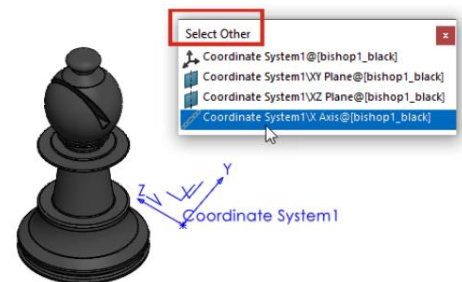
- Save as dialog box, including the advanced option
- Renaming parts and assemblies in the Feature Manager Design tree
- Mirror Components Property Manager
- Split Property Manager and Save Bodies Property Manager
- Pack and Go



PARTS AND FEATURES:

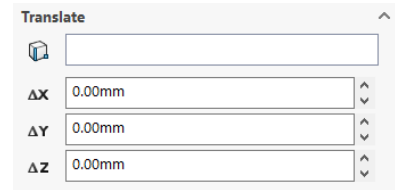
In SOLIDWORKS 2023, coordinate system referenced elements from additional available entities, features, commands including the select other command.

- In 3D sketches, create sketch relations between 3D sketch entities, coordinate system axes, different planes and origin.
- In dimensioning, create 2D sketch dimensions that reference coordinate system axes, different planes and origin.
- In mirroring features and inserting mirrored parts in a part file, select a coordinate system plane as the mirroring plane for the feature or that existing part.



Dimension Support for the Move/Copy Body

In SOLIDWORKS, move/copy body command, can now control the "translate and rotate values" with equations and add these equations to a design table and also control them directly from there.

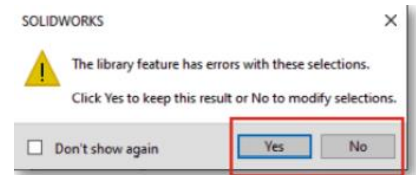


Allowance of Library Feature Errors

Inserting a library feature into a model and get error message. In SOLIDWORKS 2023, this can be viewed and manually repaired by clicking yes to the warning message to keep the results. In the Feature Manager Design tree, this can be edited or click undo to remove it.



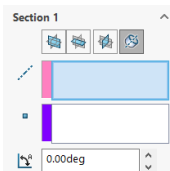
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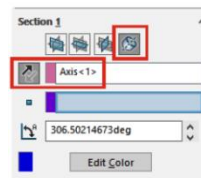
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Section View through Axis

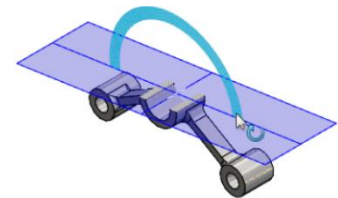
- Section view through an axis without constraining it to a secondary point, such as a vertex.
- Previously, a secondary selection was mandatory.



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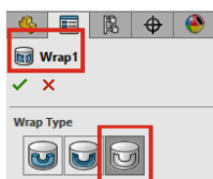


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Single-Line Fonts for Wrap Features

- Wrap features from sketches that use single-line fonts when you select Scribe for Wrap Type.
- In previous versions, wrap features did not support single-line fonts.



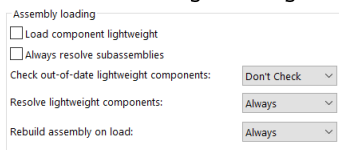
ASSEMBLIES:

Automatically Optimize Resolved Mode

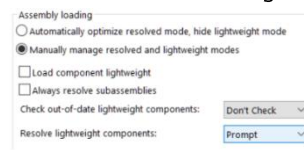
Optimize resolved mode automatically feature in SOLIDWORKS 2023, this system option improves performance by selectively using lightweight technology when tends to load components in resolved mode.

Where to find:

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



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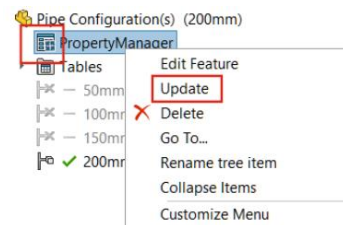
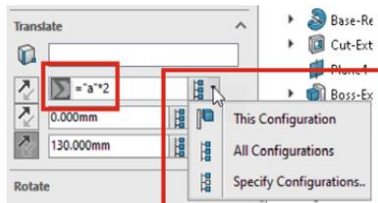
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Configuration Publisher: Updating a Multiple Configuration Model

Models that have multiple configurations created with a custom Property Manager,

The Property Manager Icon changes after manually adding or deleting, or renaming a configuration.

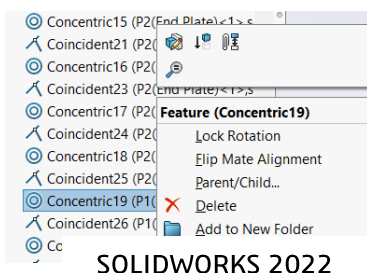
- Open a model with multiple configurations created with a custom Property Manager. Use Configuration Publisher to create a custom Property Manager.
- In the Configuration Manager tab, add a configuration. The Property Manager Icon changes
- Right-click the Property Manager and click Update, the Excel design table opens, updates, and closes. The Property Manager Icon changes



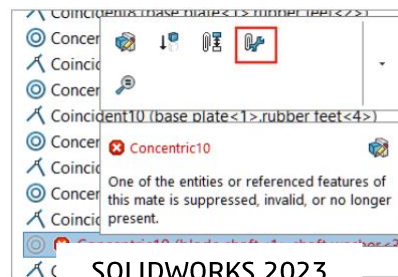
Repairing Missing Mate References

SOLIDWORKS repairs the missing reference for a component by selecting an existing reference of the component that is in the same location and orientation to the missing reference.

- Open a model that contains mate errors.
- Right-click the mate and in the context toolbar for the mate, click Auto Repair



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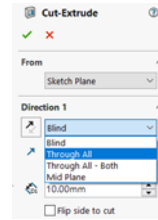


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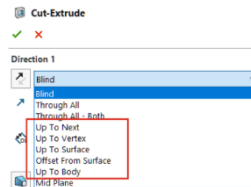
Additional End Conditions for Cut Features

More end conditions are available for cut features in assemblies. Select these options for Direction 1 and Direction 2:

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



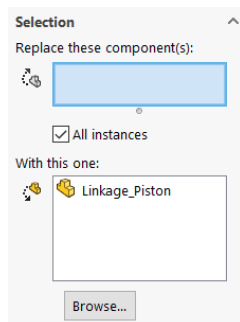
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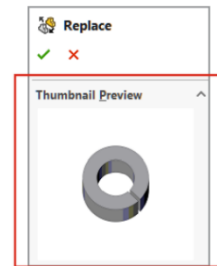
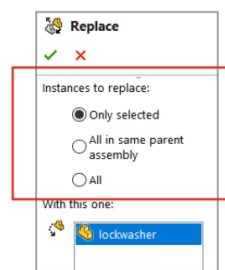
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Previewing a Replacement Component

Preview a replacement component in the Replace Property Manager and specify the scope of the replacement. In the Property Manager, Instances to replaces component based on the three given selection criteria.



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