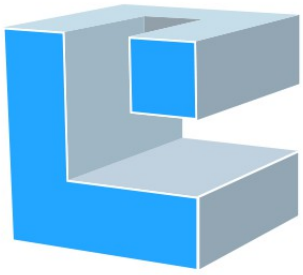

BACKWORKS

A SOLIDWORKS ADD-IN

USER MANUAL

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Copyright and Legal information

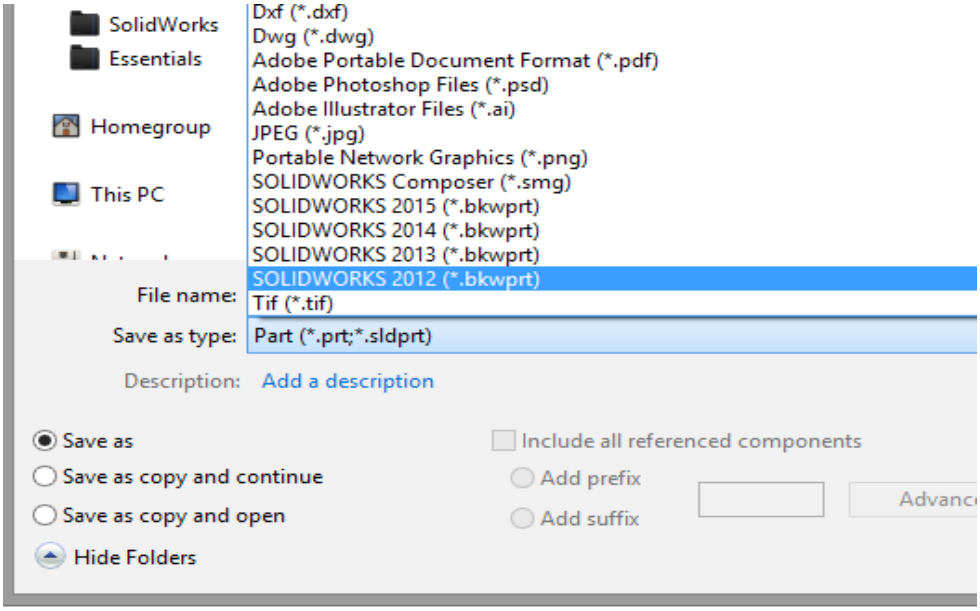
Introduction

BACKWORKS is a fully integrated SOLIDWORKS add-in that enables SOLIDWORKS to save files back into an older version, making previous release interoperability possible. For the time being, BACKWORKS enables saving and opening only Part documents into up to 7 versions back (up to SOLIDWORKS 2012) only to 64-bit Windows Vista, 7 and 10 operating systems.

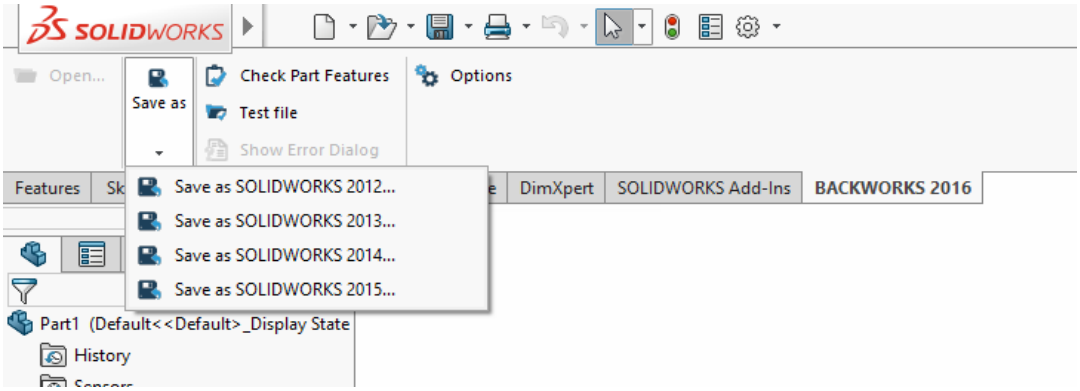
Quick Start

To save a SOLIDWORKS Part document into a previous version:

- Click **File** > **Save As...**
- Navigate to the file destination folder.
- Open the **Save as type:** pull down menu and scroll down.
- Select the SOLIDWORKS previous version (e.g. SOLIDWORKS 2013 (.bkwprt)) file type.



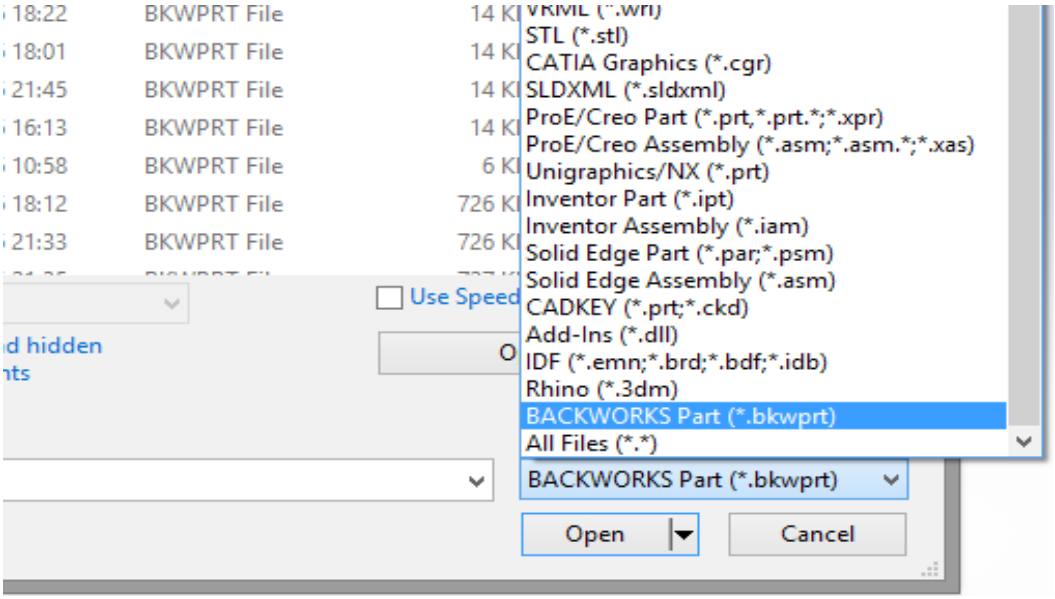
- Click Save.
  OR
- Click the Save As icon on the BACKWORKS 2017 toolbar tab.
- Select the SOLIDWORKS previous version (e.g. SOLIDWORKS 2013 (.bkwprt)) file type.
- Navigate to the file destination folder.



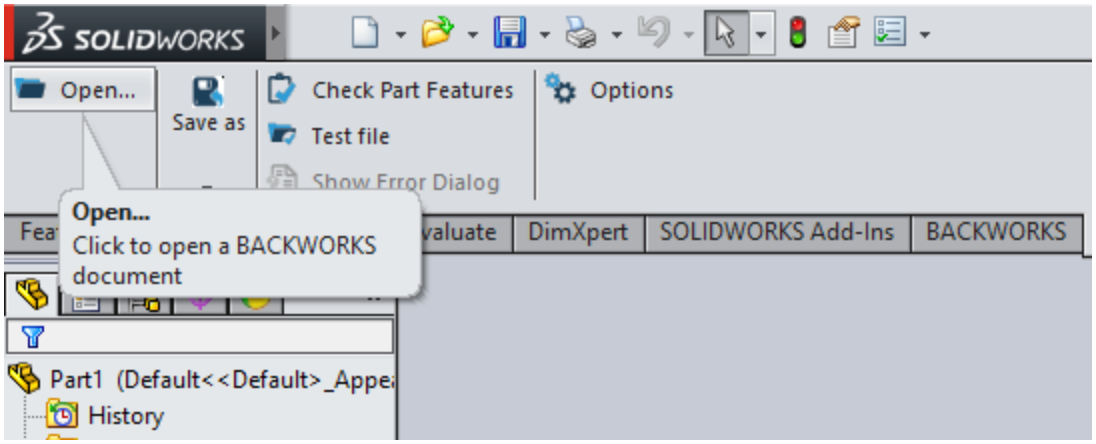
- Click Save.

To open a BACKWORKS Part document:

- Click **File** > Open**...** or click the Open icon at SOLIDWORKS Standard Toolbar.
- Open the File **type** pull down menu and scroll down.
- Select **BACKWORKS Part (*.bkwprt)** file type.



- Select the BACKWORKS Part and click Open.
  OR
- Click the Open icon on the BACKWORKS 2016 toolbar tab.
- Navigate to the file destination folder.



- Select the BACKWORKS Part and click Open.

BACKWORKS Toolbar

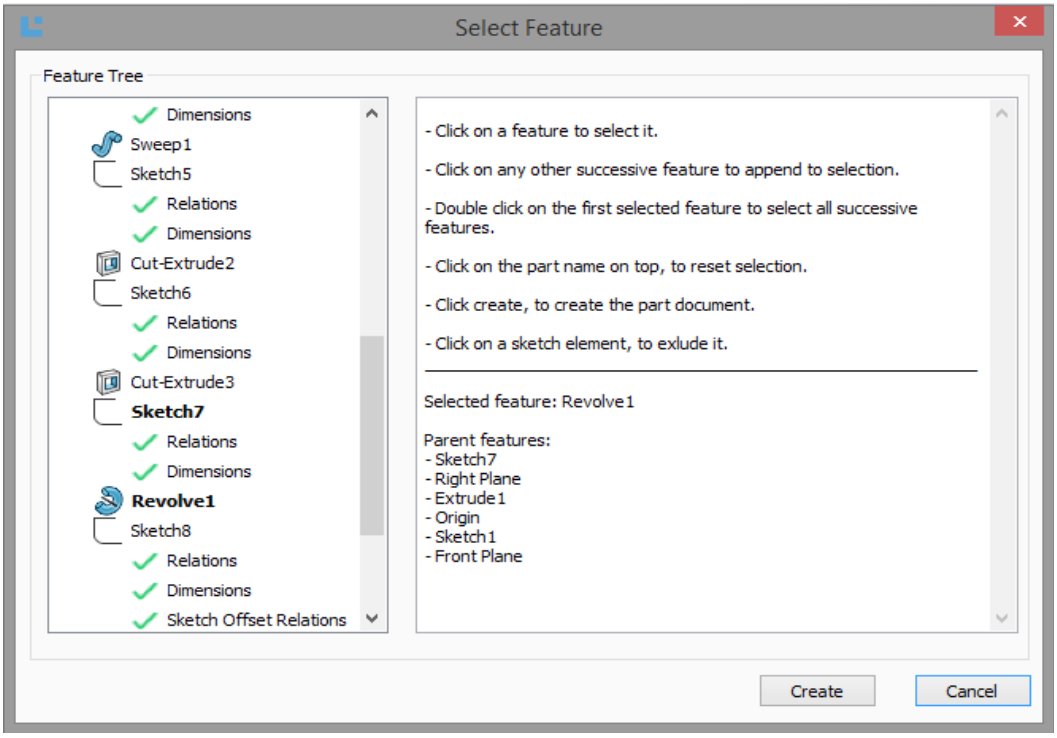
Open
Open a BACKWORKS Part Document.

Save as
Save SOLIDWORKS Part into a previous version.

Check Part features
Check Part features checks if the Part document contains BACKWORKS unsupported features or SOLIDWORKS features unsupported in previous versions. The check applies only to top level features (e.g. the features the user can see at the SOLIDWORKS Feature Manager Tree).

Test File
Test File opens a BACKWORKS file like the previous version would. It is meant to test the file to be sent to the other user.

Select feature
"Select feature" reads all features that a BACKWORKS file contains. It allows the user to select which features he wants to add into the model.
This function is useful in the case of a failed feature or a feature unsupported. It allows the user to manually edit all failed or unsupported features and still continue using BACKWORKS.
It is also useful in order to create library features that can be used in older versions of SOLIDWORKS. If a feature has references only to standard planes (Front, Top and Right Plane), it can be recreated into any Part document.



"Select feature" recognizes the features that already exist in the model and greys them up. Available features are shown normally. Sketch features Relation, Dimensions and Sketch Offset Relations elements are also shown and can be excluded. The user can perform the following actions:

Click on a feature – The feature is selected. If the user clicks on a feature at the middle of the model, all parents features are also selected.

Click on any other successive feature – The feature is appended to selection.

Double click a feature – All successive features are selected.

Click on a sketch element to exclude it. This function is useful in case a sketch feature fails, due to failure of a sketch element (Relation, Dimension or Sketch Offset Relation). The user can exclude an element from creation in order to trouble-shoot the problem.

Click on the part name on top – Selection is reset.

Click Create – to create all selected features.

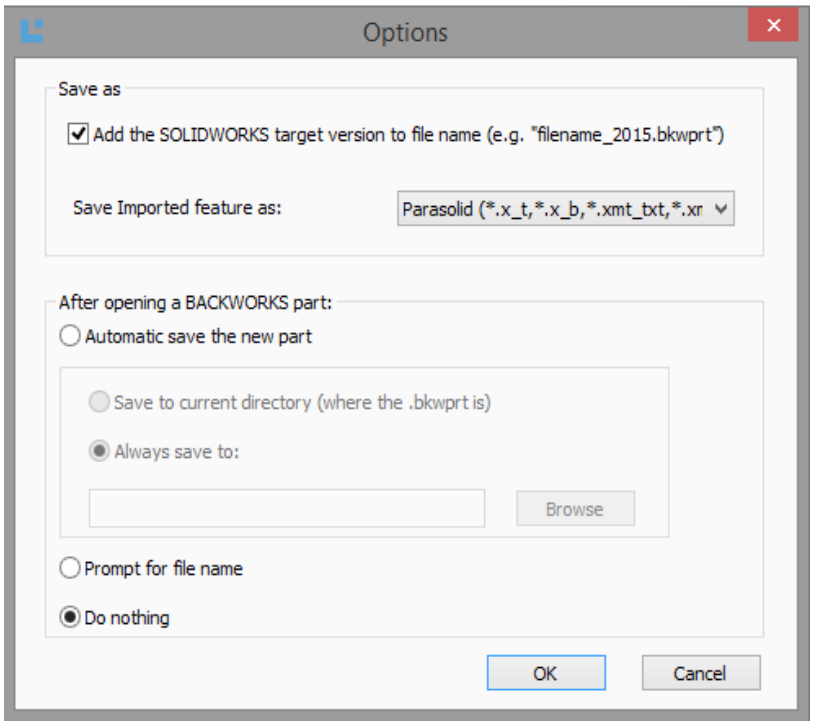
Options

Change BACKWORKS options.

Show Error Dialog

Show error dialog is available after BACKWORKS has finished saving or opening a file. It makes the dialog, that shows all error and warnings that have occurred while saving or opening a file, reappear.

Options



Add the SOLIDWORKS target version to file name

This option adds a SOLIDWORKS target version extension to all files that BACKWORKS is saving, like filename_2015.bkwprt instead of filename.bkwprt.

When the SOLIDWORKS menu command is used (or icon for saving), the target extension does not appear in the file name field of the SOLIDWORKS Save as dialog. However, the extension *is* added to the file name. Any SOLIDWORKS warning that the file already exists and must be replaced can be ignored.

Save Imported feature as

This option determines which neutral format BACKWORKS will use when saving an Imported feature. Supported formats are Parasolid, Iges, Step, ACIS and STL.

Automatic save the new Part

Automatic saves the new Part opened by BACKWORKS to a user defined directory or to the current SOLIDWORKS directory (current SOLIDWORKS directory is the directory that user has last use to open or save a SOLIDWORKS document).

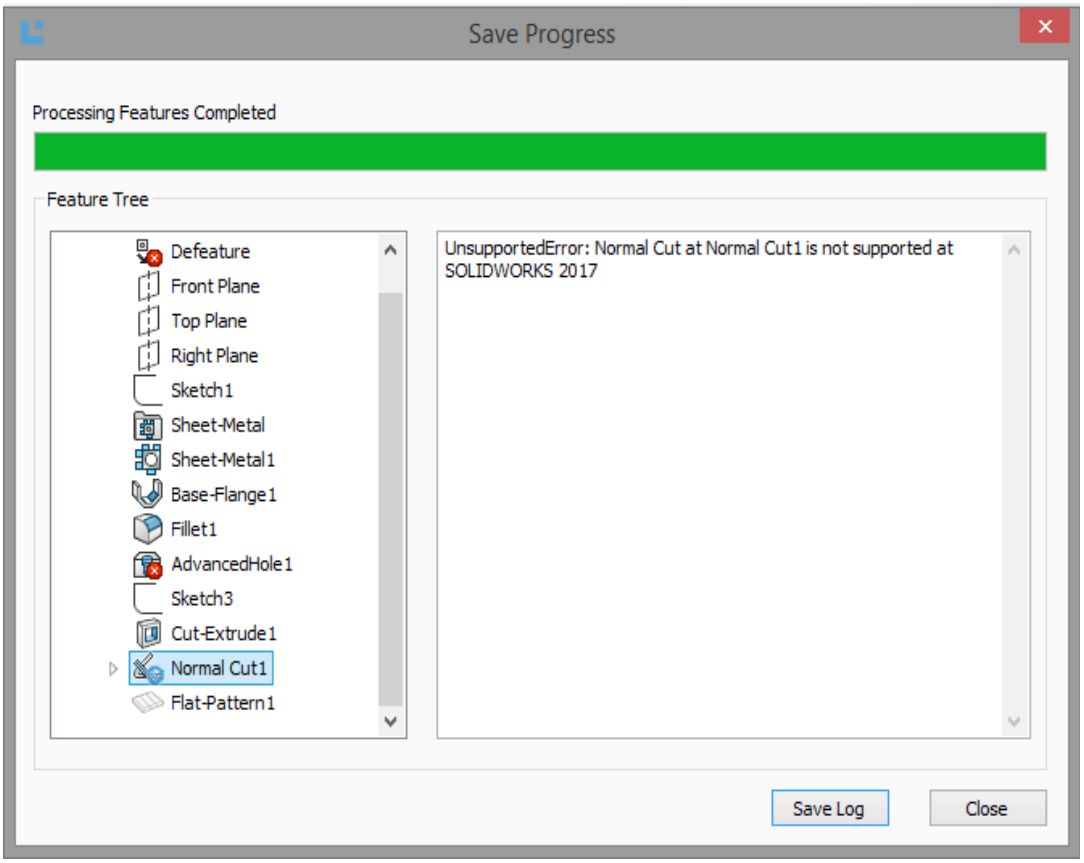
Prompt for file name

BACKWORKS opens the file save dialog after a file has opened.

Do nothing

BACKWORKS finishes, leaving a default SOLIDWORKS new Part name(usually Part1, Part2).

BACKWORKS Progress Dialog



A progress dialog appears while BACKWORKS is opening or saving a file. It demonstrates how the Part is being processed feature by feature. BACKWORKS has three types of warnings:

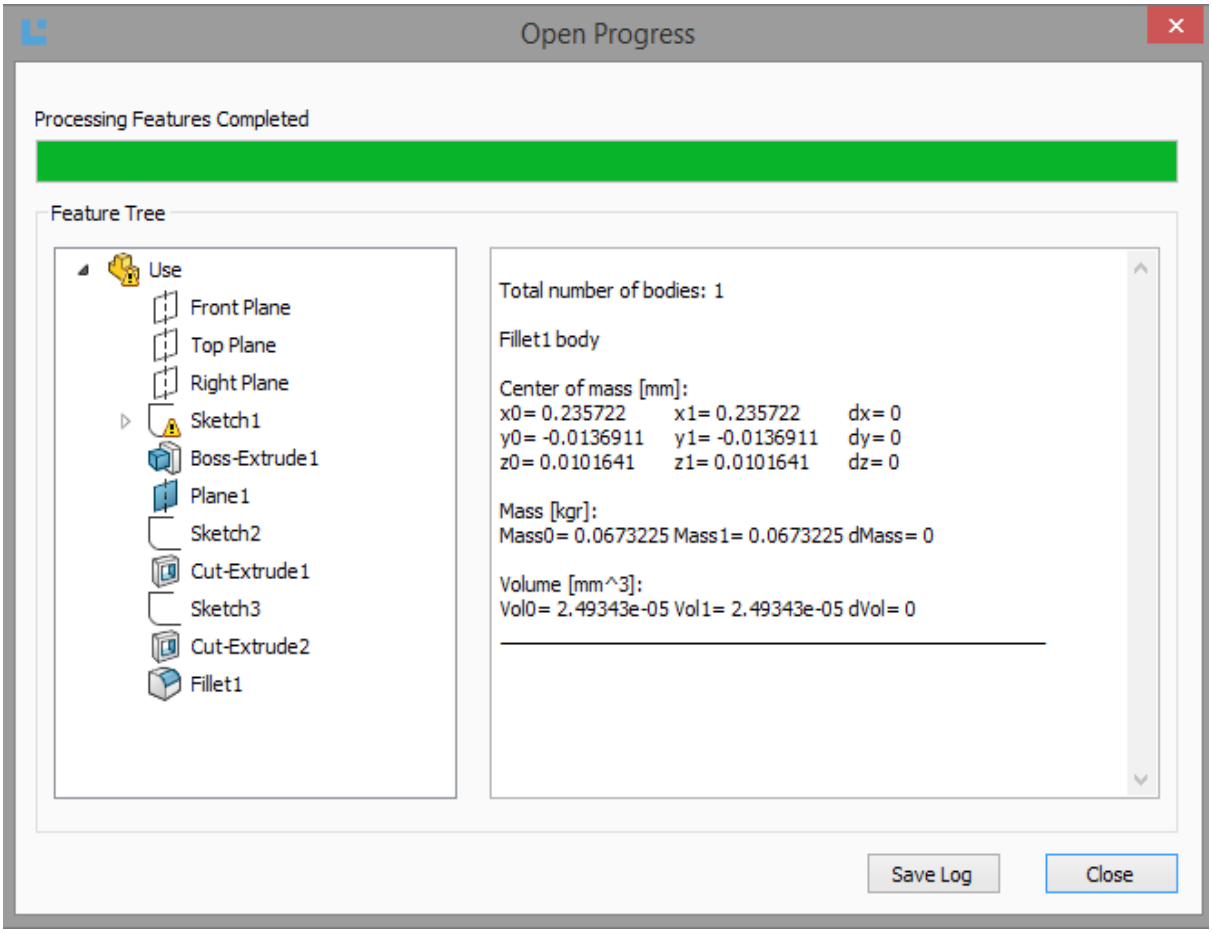
SOLIDWORKS icon
Feature processed ok.

Error
Feature process failed. When an error occurs while BACKWORKS is saving a file, the procedure can be continued by excluding the failed feature and all the children of that feature. Part will be saved, but only features processed correct will be sent to the other user. If BACKWORKS opens a file, and an error occurs, it stops at that point.

Unsupported error
Feature is not supported in the target SOLIDWORKS version. That error occurs when BACKWORKS tries to save a file containing a feature that does not exist in the target SOLIDWORKS version. BACKWORKS handles the unsupported error as Error.

Warning
Feature process succeeded but there are warnings. Warnings are small errors that do not affect the geometry. A warning, for example, is the failure to create a sketch dimension or a sketch relation. After a warning, BACKWORKS continues to function normally.

After a Part has been successfully opened, the progress dialog shows the mass properties (center of mass coordinates, mass, volume) of each body compared with the mass properties of the original Part. The comparison is made by subtraction. BACKWORKS performs a variety of checks to the final geometry. In some cases there are small differences between the opened Part and the original Part that are not actual errors. The progress dialog demonstrates the differences, so the user can decide if they are acceptable or not. For example if the difference between the y coordinates of the opened Part and the original is 0.00000005, it can be ignored.



The zero marker applies to the original Part and the 1 marker applies to the opened Part. At the example above, all differences are equal to zero and the Part has been properly saved.