



Altium Modeler for SOLIDWORKS

Copyright © Desktop EDA

Altium Modeler for SOLIDWORKS

Copyright © Desktop EDA

All rights reserved. No parts of this work may be reproduced in any form or by any means - graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems - without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

Printed: February 2017 in Australia

Table of Contents

Foreword	I
Part I Altium Modeler for SolidWorks	2
1 Introduction	2
2 Getting the Version	2
3 Licensing	3
Desktop EDA Licensing	3
Altium Licensing	5
Installing Extensions	6
License Management	11
Uninstalling Extensions	13
Updates	15
4 Command Summary	15
5 Using the Workspace Panel	20
6 Preferences Dialog	24
Options	26
Document Locations	28
Part Selection	29
Auto Parts	30
Extrusion Options	31
BOM Data to SolidWorks	32
PDM Works	33
Solder Mask	34
7 Interpreting the Altium Layer Stack Manager	34
8 SolidWorks Functions	35
Setting Up For Export	35
Step Model Conversion to Parts	36
Creating an Assembly or PCB Part	36
Updating an Assembly or Part	37
Updating the PCB From SolidWorks	38
New PCB from SolidWorks	39
Representing Variants in SolidWorks	39
Working With Height Rules	40
Working With Keepouts	41
BOM Data to SolidWorks	41
Setting Colors	43
9 PCB Features in SolidWorks	43
Guidelines on Exporting Tracks Polygons and Silk Screens	44
Trouble Shooting Extrusion Problems	45
Potential Causes of Extrusion Problems	45
Finding and Correcting Extrusion Failures	46
Internal Planes	49
Solder Masks	49
Solder Masks Using Polygons	49
Two Layer or Multi-Layer	50
Adding Klipper Logos	51

10 Managing Parts	52
SolidWorks Parts	53
Parts in Folders	54
Using Cross Reference (Xref) Files	55
PCB Library Commands	58
Step Model Conversion	60
Parts Library	61
IC Naming Convention.....	61
IC Listing	62
Part Origin Convention.....	64
11 Product Tools	64
Check Component Outlines	64
Check Outline	64
Check Outline (PCB Library Editor)	65
Height Editor	65
Set Mounting Hole	67
Duplicate Objects	67
Zero Length Tracks and Arcs	68
Show Part Usage	68
Compare	68
Compare PCB.....	69
Compare Component Locations	69
12 Rigid/Flex Support	69
Setting up Rigid/Flex Board Regions	70
Creating the Rigid Flex Assembly	70
Rigid/Flex Layer Stacks	70
Updating the Rigid Flex Board Outlinefrom SolidWorks	71
13 Starting Designs in SolidWorks	72
Guidlines For Starting Designs In SolidWorks	72
PCB Tools	72

Index

75

1 Altium Modeler for SolidWorks

1.1 Introduction

This product enables you to model your PCB designs in SOLIDWORKS. In addition, it allows you to make changes to the Board Outline and component locations in the SOLIDWORKS assembly and synchronize these changes with the Altium Designer PCB.

The product has Workspace Panels in both the PCB Editor and the PCB Library editor.

The Board outline is modeled by exporting the board outline to a SOLIDWORKS sketch and extruding it. Holes and cut outs are represented as cut extrudes in the Board Outline feature.

PCB Components are represented in the SOLIDWORKS assembly by either:

- Placing a SOLIDWORKS part if one exists, or
- The program automatically creates a part using geometry or a Step Model extracted from the PCB footprint.

The program can use either SOLIDWORKS Parts or Assemblies as Parts.

Options are available to include pads, via pads, traces, polygons, silk screen and solder mask in the PCB part.

The board outline is defined in Altium using either the Board Shape or an enclosed shape comprising lines and arcs on a mechanical layer.

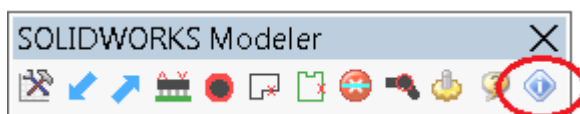
This product requires a SOLIDWORKS license to be available. When the Create Assembly command is invoked, the SOLIDWORKS application is launched and the program interacts with SOLIDWORKS to build the SOLIDWORKS Assembly.

The product also has some commands in the PCB Library editor to allow you to automatically create Component bodies from extrusions in a SOLIDWORKS part and to create a SOLIDWORKS part from Bodies in a component.

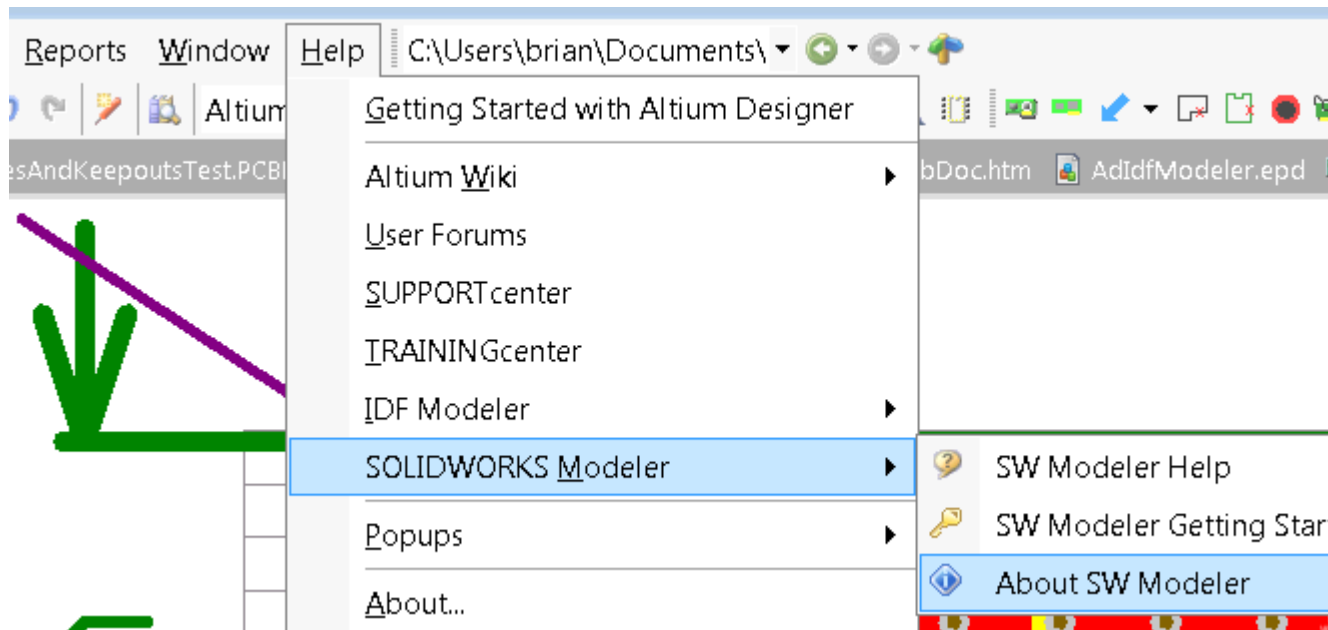
1.2 Getting the Version

You can get version information by invoking the About command.

This can be accessed from the toolbar:



or from the PCB Editor Help menu:



1.3 Licensing

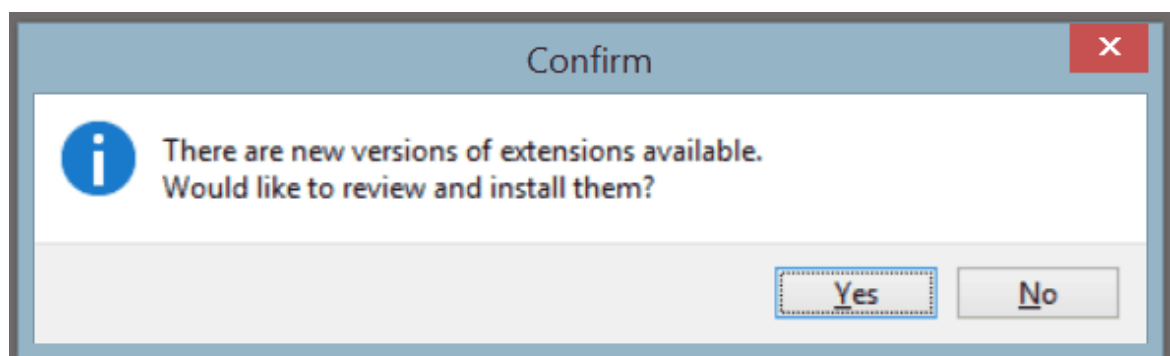
Whether you are running the trial or have purchased a license, you will require a license number and access code to run this product.

If you have received the product via the Altium Extensions and Updates page, the product is licensed in the same manner as the Altium application.

Otherwise, refer to the next topic.

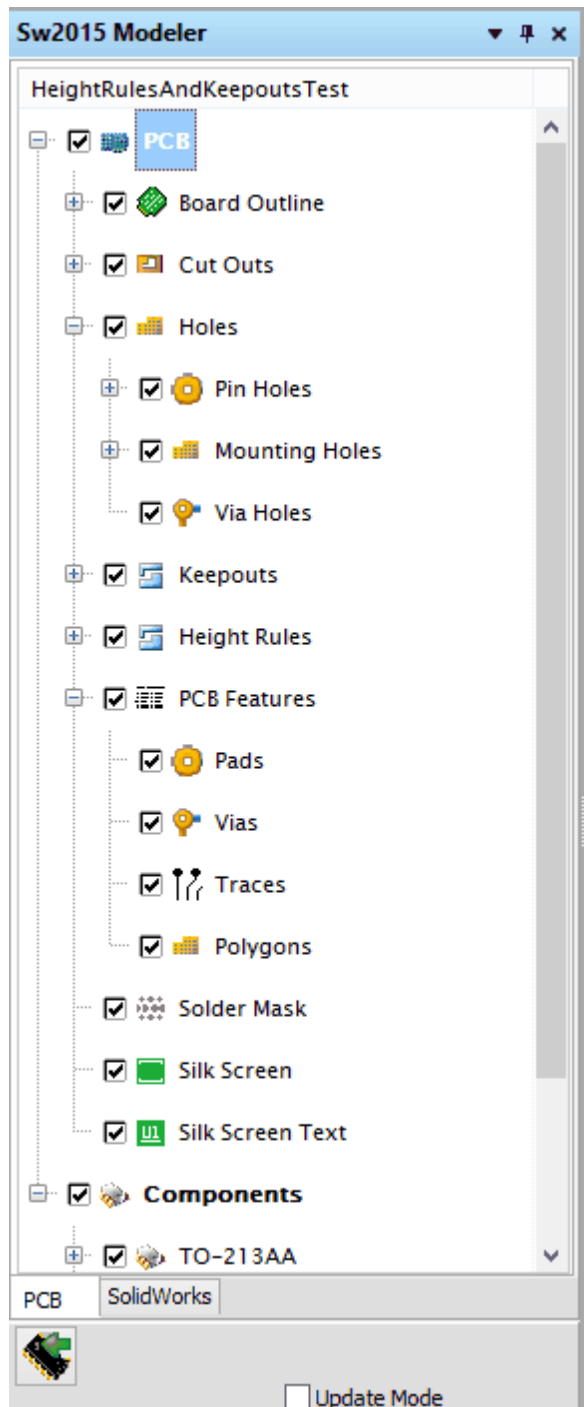
1.3.1 Desktop EDA Licensing

The first time you invoke Altium Designer after running the installation program, you are prompted for a security key.



If you have been provided with License Number and Access Code, this is when you enter it.

Click the OK button and the following dialog displays:



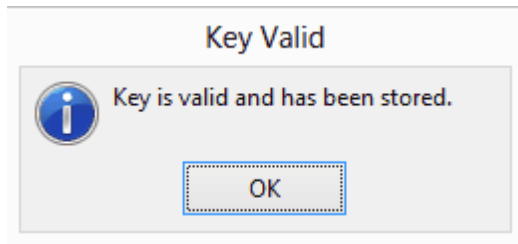
The License Number and Access Code are provided as a line of text in the format show below:

<license number> 111111-222222-333333-444444-555555-666666-777777-888888-999999-BBBBBB-CCCCC

There is a space between the License Number and the Access Code.

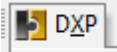
You will most likely receive the licensing information in an email. Select the line of text containing the License Number and Access Code, copy it to the Clipboard, then paste it into the Key: field.

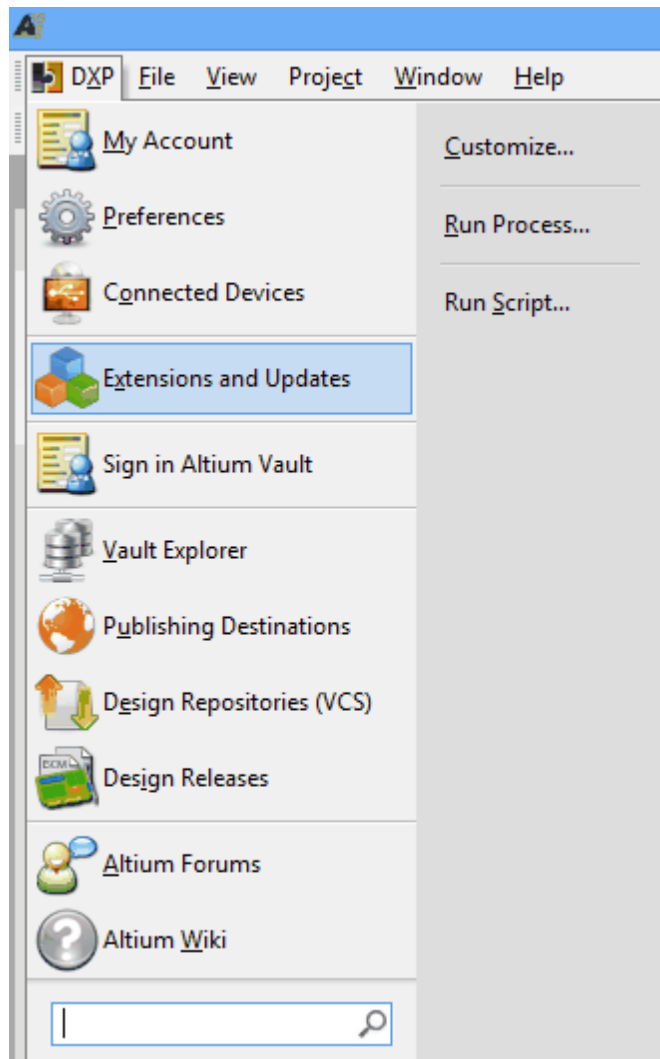
If the license is accepted, this dialog displays:



1.3.2 Altium Licensing

This topic guides you through the process of installing, uninstalling Altium Designer Extensions and License management.

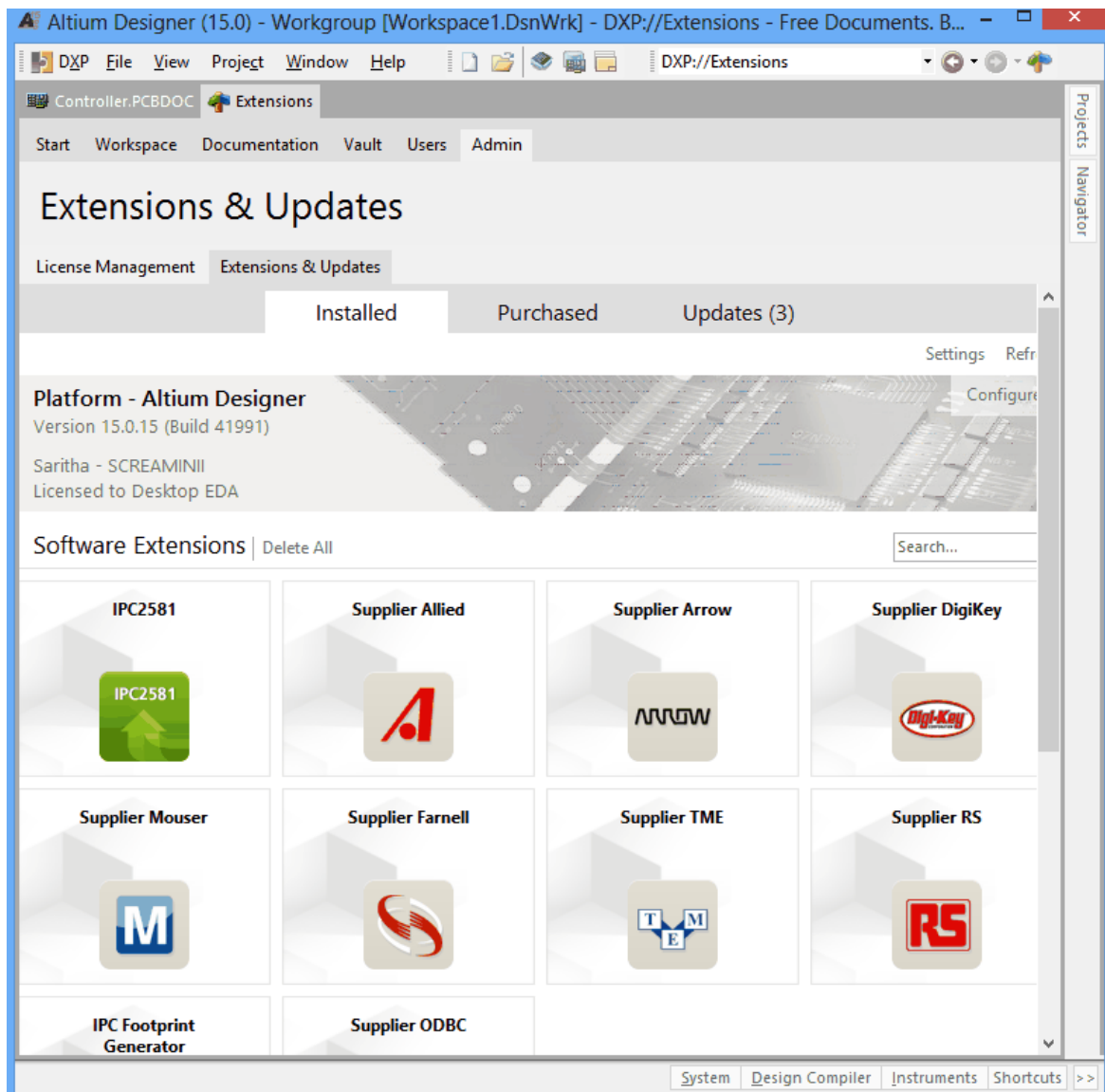
Click on the  button, then the following software's resource pane opens. Choose Extensions and Updates command



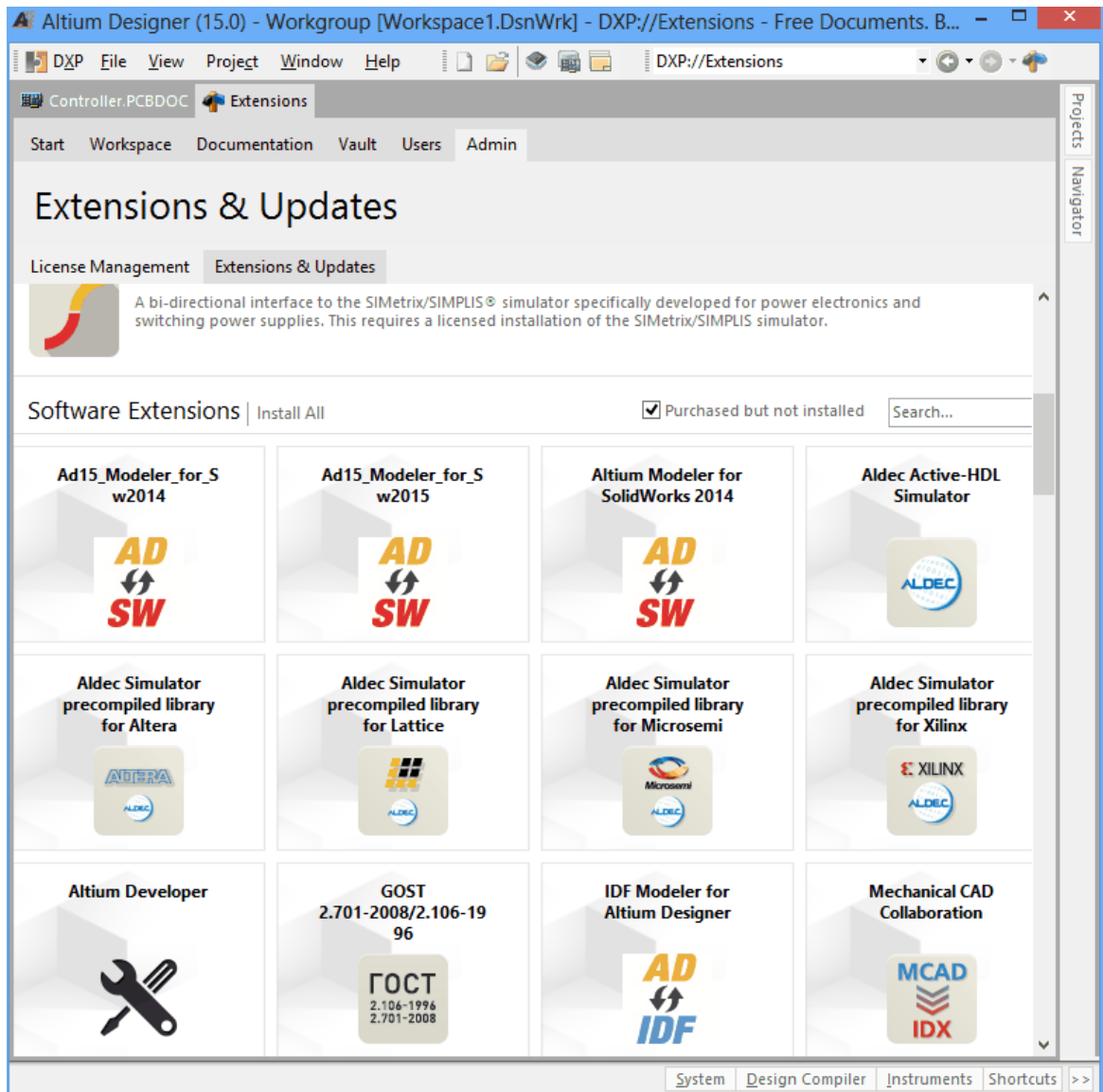
1.3.2.1 Installing Extensions

Click on DXP button> Extensions and Updates

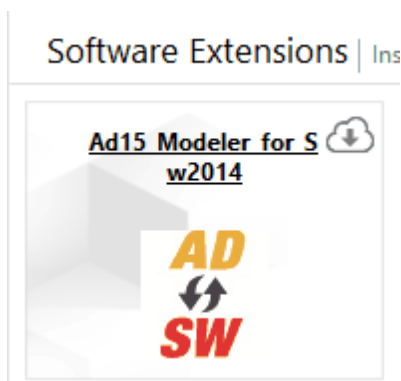
By clicking on Extensions and Updates entry, the following Extensions& Updates view displays

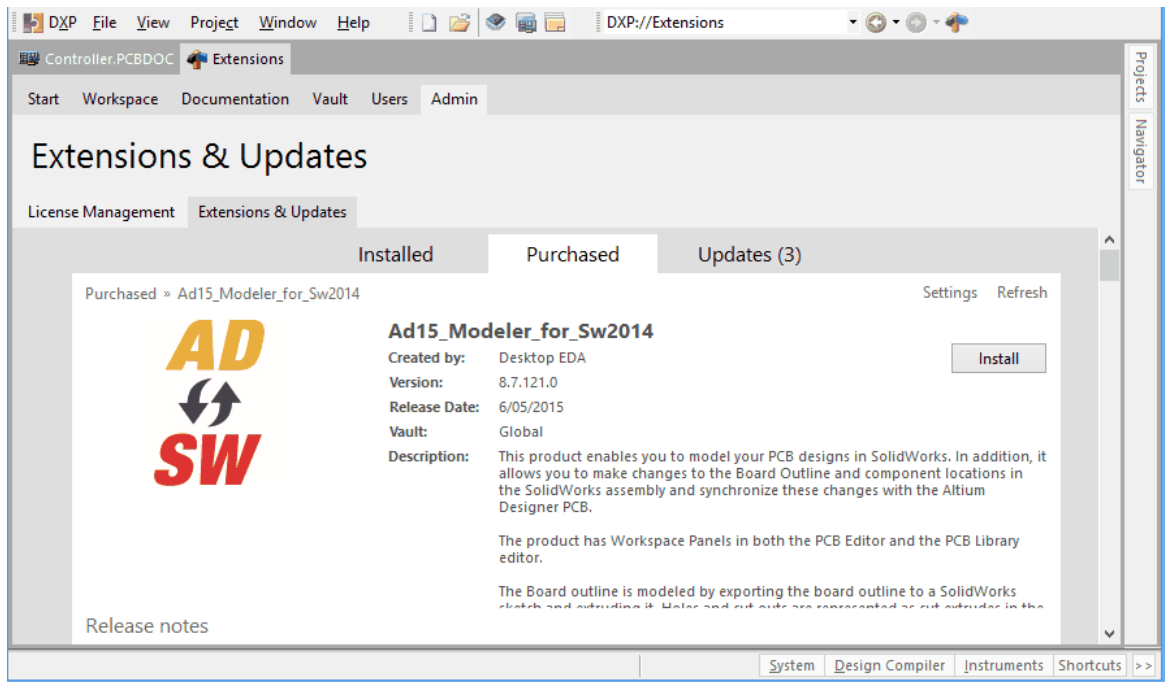


Click on Purchased tab, the Purchased page is displays as shown below - consisting of platform extensions, software extensions and hardware devices.

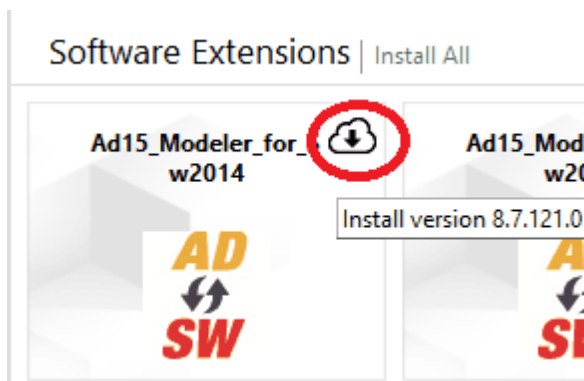


Clicking on the name of an extension accesses a **detailed** page for that extension as shown below:

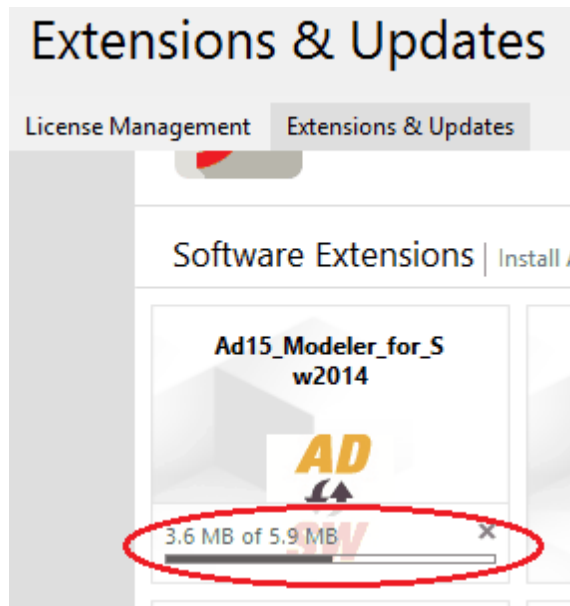




Install the extension either from the summary-level **purchased** page, by clicking on the  icon at the top-right of its entry. Hover the mouse over the download icon to view details of version available.

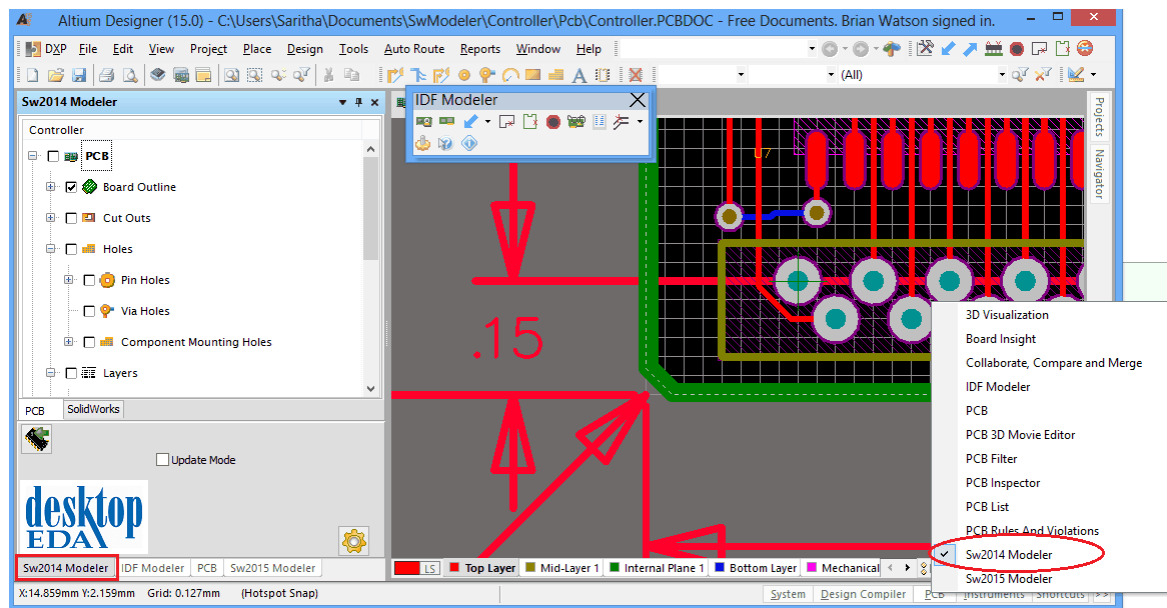


OR extensions can be installed from its **detailed** page, by clicking on the Install button. A progress bar will appear showing progress of the download as shown below:



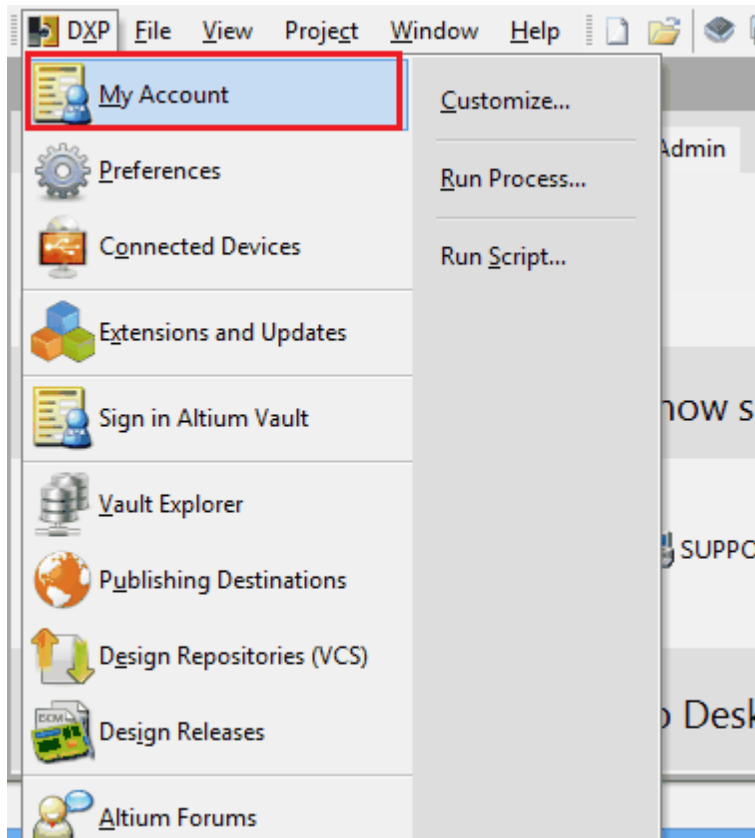
A restart of Altium Designer is required after an extension finishes installing, in order for the functionality it provides to take effect.

Open the Altium Designer(AD), you will see the currently installed extension is added to the AD on the status bar as shown below

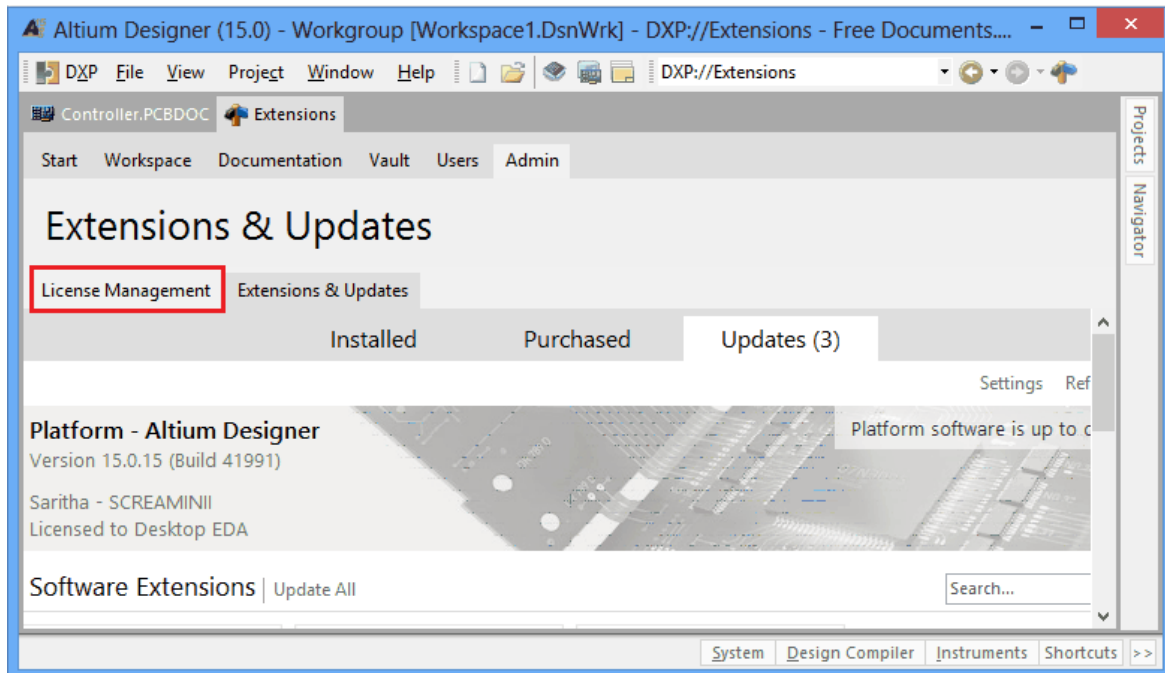


1.3.2.2 License Management

Go to DXP>>My Account command.

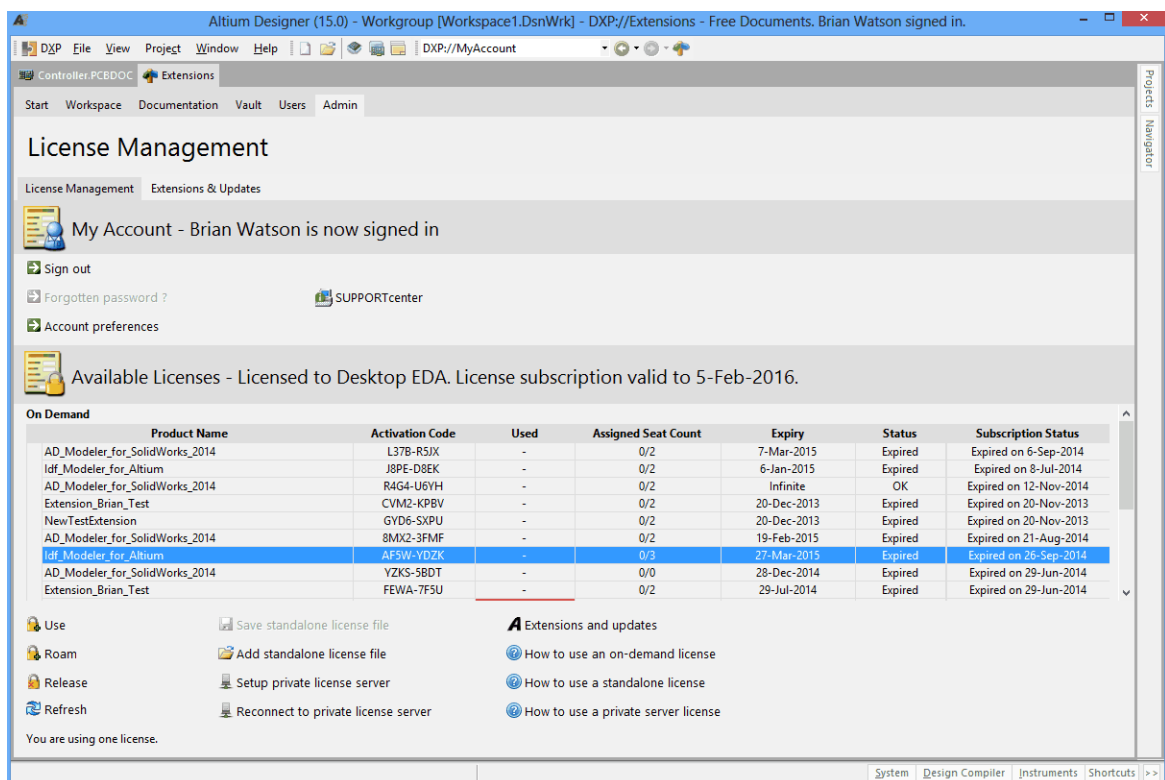


OR Click on License management on the Extensions&Updates page to access the License management page.



The following License Management page displays:

Sign-in to your Altium account and view and select On-Demand licenses available for you. Once signed in, your Standalone license will be presented in a grid, within the Available Licenses region of the page. If any other licenses (or license types) are available to you, these will also be presented



Select the product and click on Use button to use the license then the Used field will change to 'Used by me' as shown below

License Management

License Management Extensions & Updates

My Account - Brian Watson is now signed in

Sign out
Forgotten password?
Account preferences

SUPPORTcenter

Available Licenses - Licensed to Desktop EDA. License subscription valid to 5-Feb-2016.

Product Name	Activation Code	Activated	Expiry	Status	Subscription Status
AD_Modeler_for_SolidWorks_2014	YZKS-5BDT	-	0/0	28-Dec-2014	Expired
Extension_Brian_Test	FEWA-7F5U	-	0/2	29-Jul-2014	Expired
Altium Designer	4YVW-ATWX	Used by me	1/1	Infinite	No seats left
Altium Designer	547V-F7W7	-	0/1	Infinite	OK
Altium Designer	UGV2-BX9D	-	0/1	Infinite	OK
Ad15_Modeler_for_Sw2014	3GBJ-V3WG	Used by me	1/1	Infinite	No seats left
Ad15_Modeler_for_Sw2015	GJ5G-5QWY	-	0/1	Infinite	OK

Standalone

Product Name	Activation Code	Activated	Expiry	Status	Subscription Status
Altium Designer	SHON-TCWN	-	6-May-2013	Expired	Expired on 6-May-2013

Use
Roam
Release
Refresh

Save standalone license file
Add standalone license file
Setup private license server
Reconnect to private license server

Extensions and updates
How to use an on-demand license
How to use a standalone license
How to use a private server license

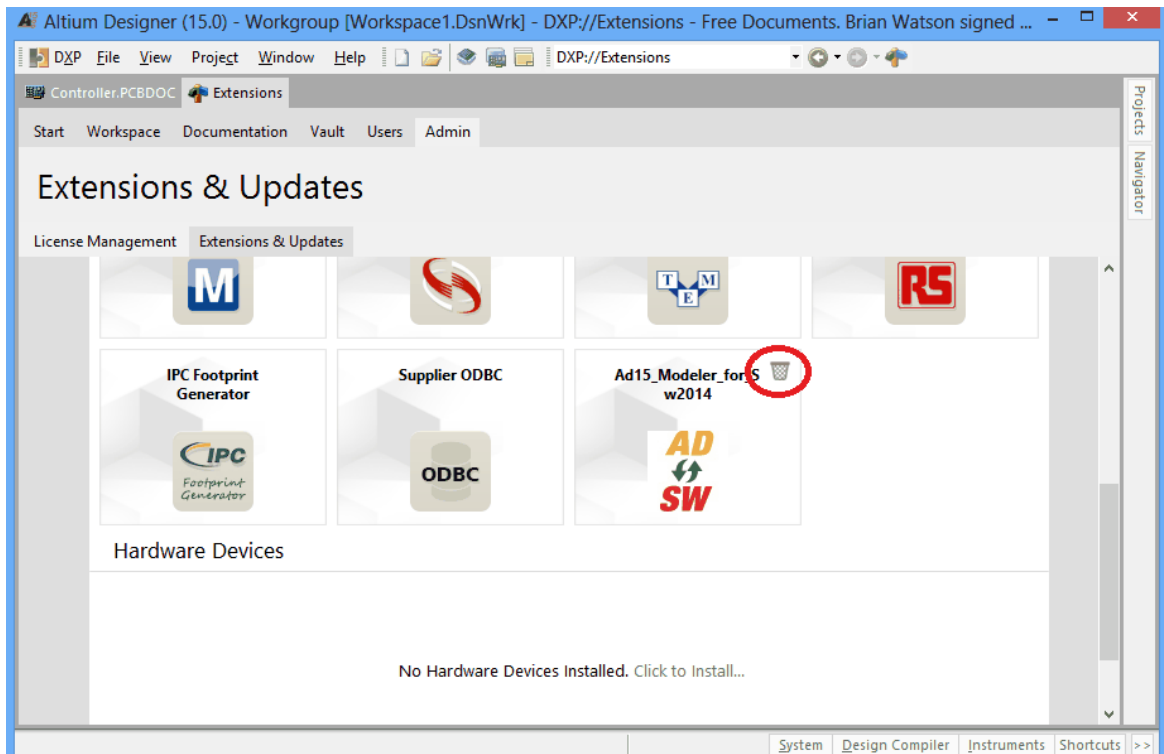
You are using 2 licenses.

System | Design Compiler | Instruments | Shortcuts >>

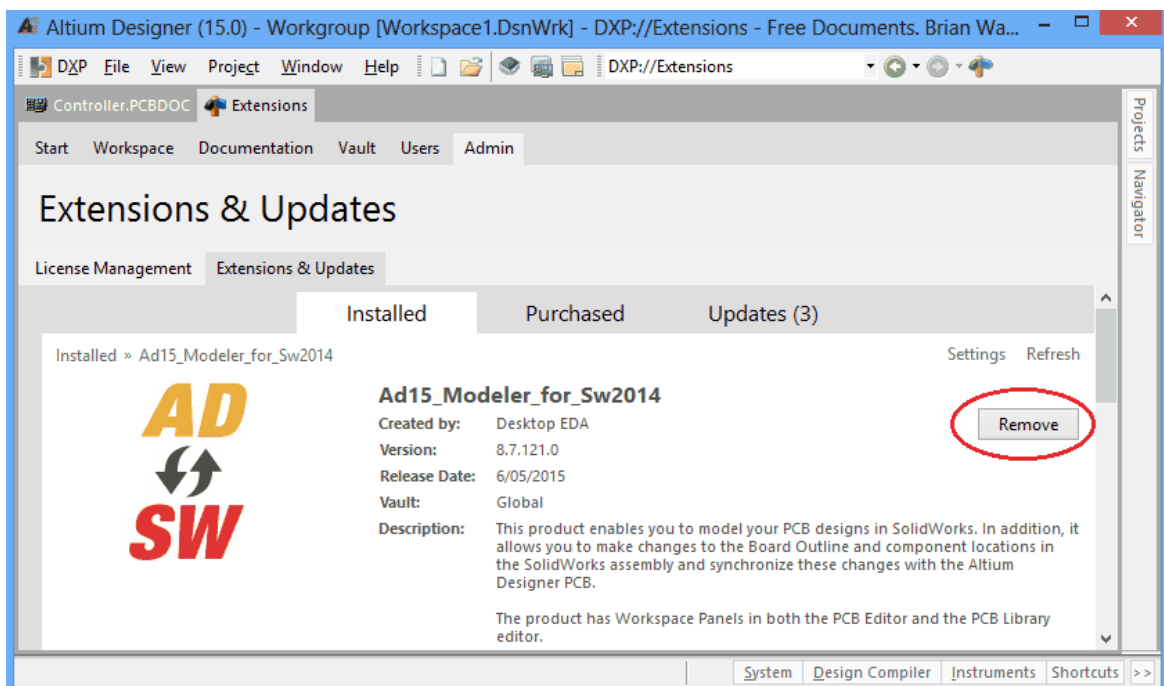
1.3.2.3 Uninstalling Extensions

A currently installed extension can be uninstalled in one of two ways:

From the summary-level **Installed** page, by clicking on the  icon at the top-right of its entry

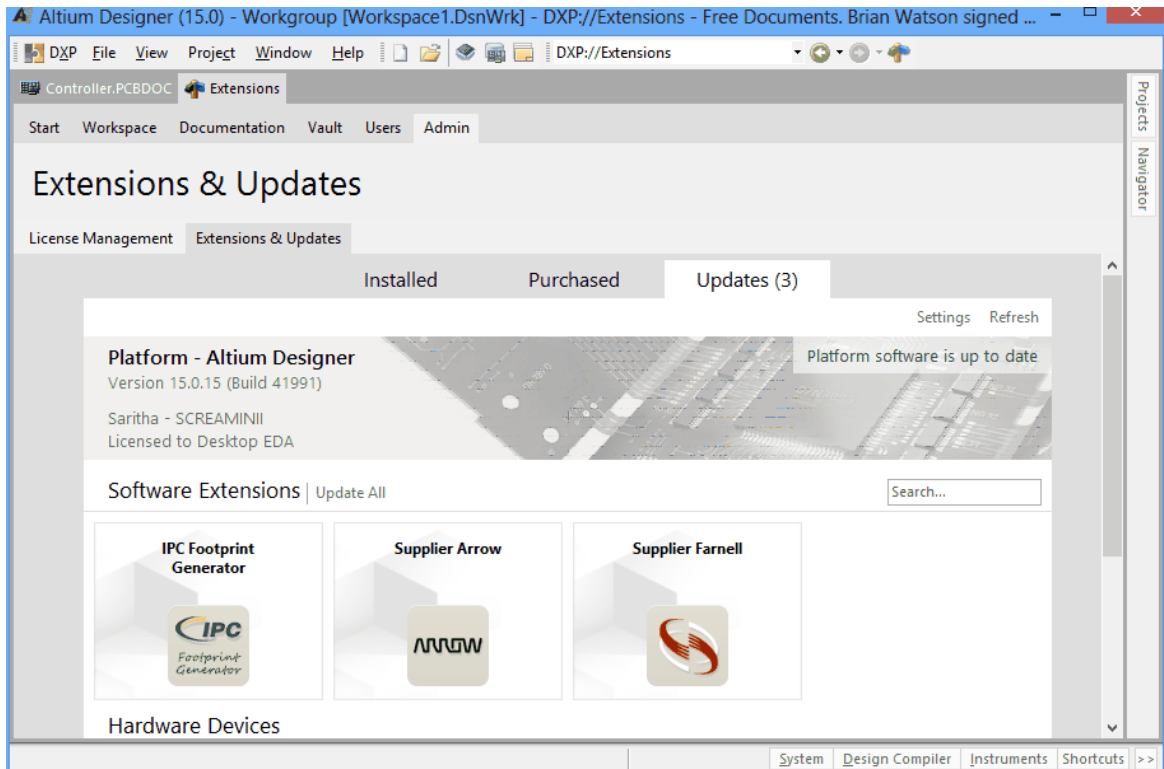


From its detailed page, by clicking on the **Remove** button as shown



1.3.2.4 Updates

Click on Updates tab to access the **Updates** page to view list all extensions for which there is an update available.

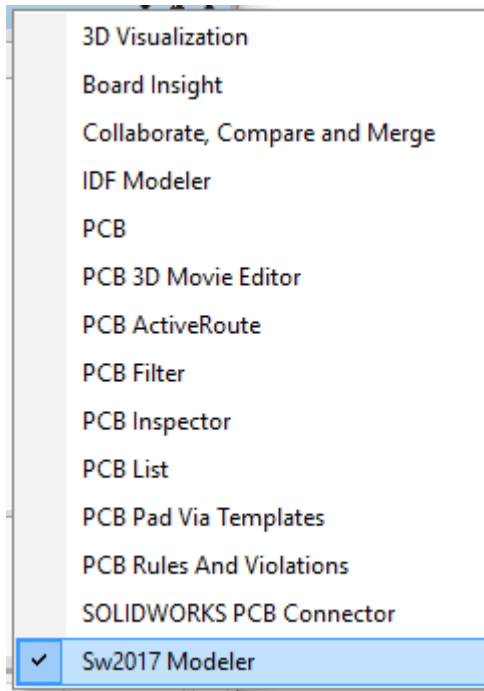


1.4 Command Summary

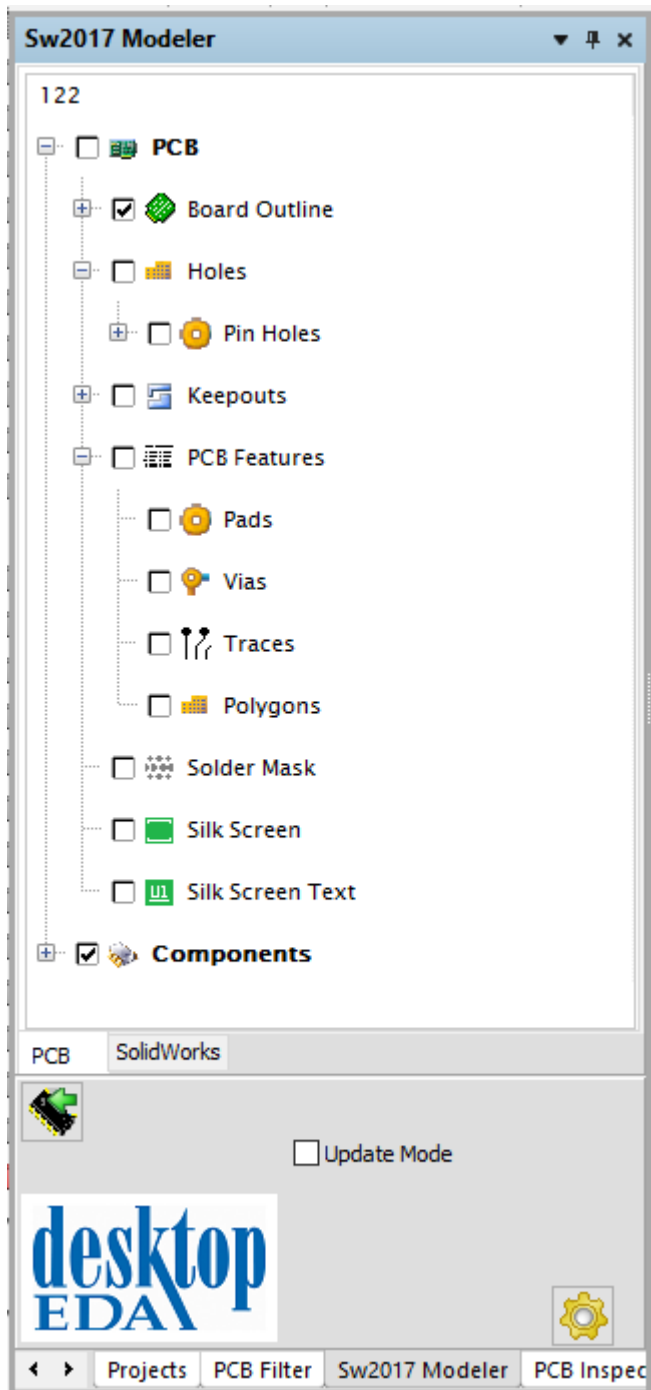
Below is an overview of the product commands and where to access them:

PCB Editor Workspace Panel

While a PCB document is open, this panel can be displayed by pressing the PCB button in the Status bar to display the list of panels available on the PCB editor. Choose Sw201* Modeler as shown:

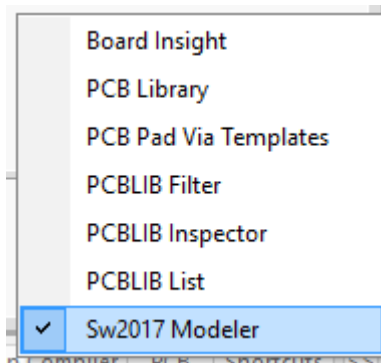


This displays the Panel shown below:

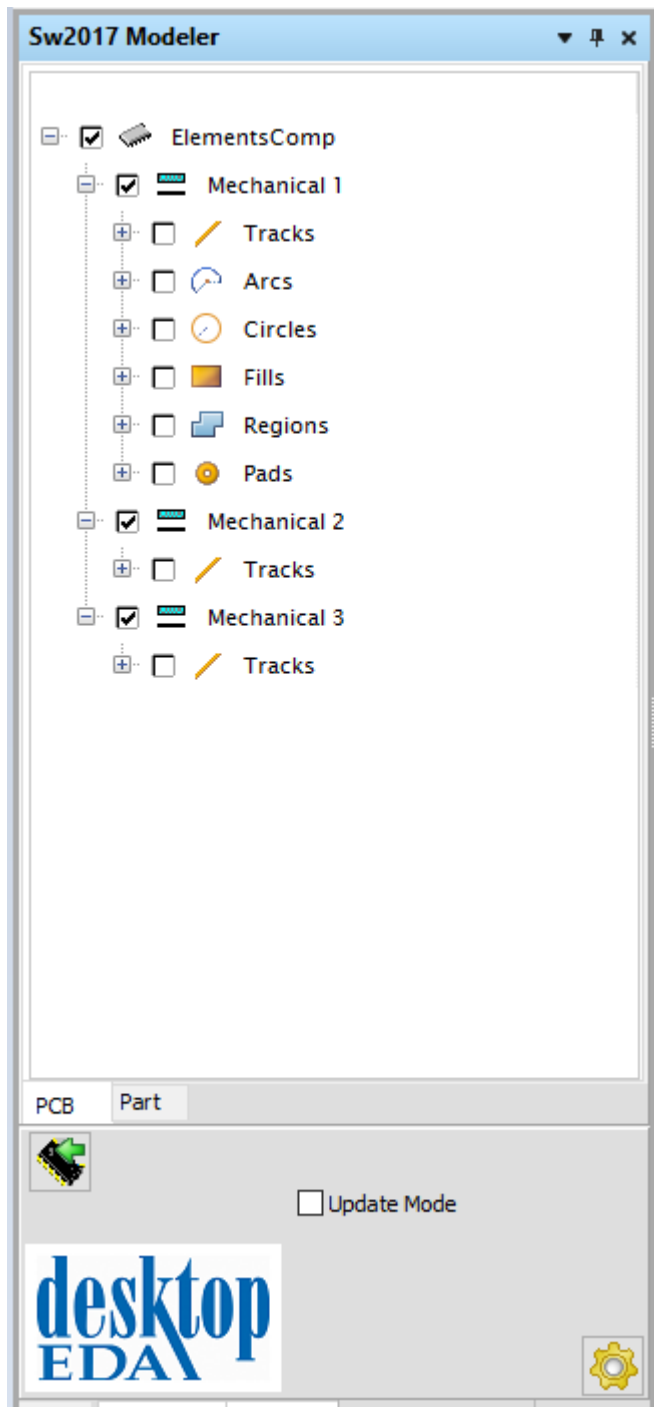


PCB Library Editor Panel

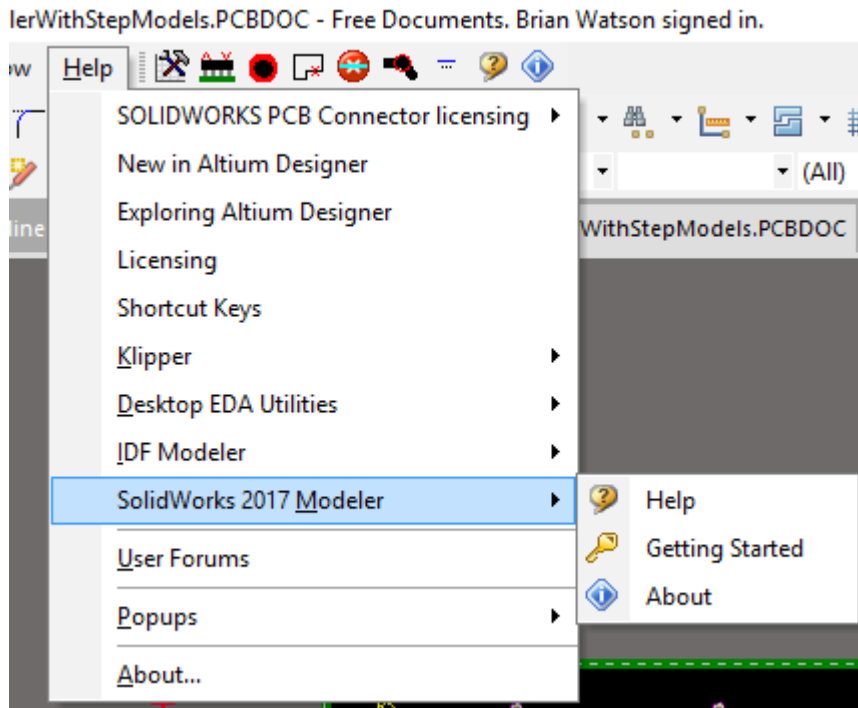
While a PCB Library document is open, this panel can be displayed by pressing the PCB button in the Status bar to display the list of panels available on the PCB editor. Choose Sw201* .



This displays the Panel as shown below:



Help Menu



File>New menu

The New PCB from SOLIDWORKS command is added to the File>New menu.

1.5 Using the Workspace Panel

The Workspace Panel provides access to most of the product functions.

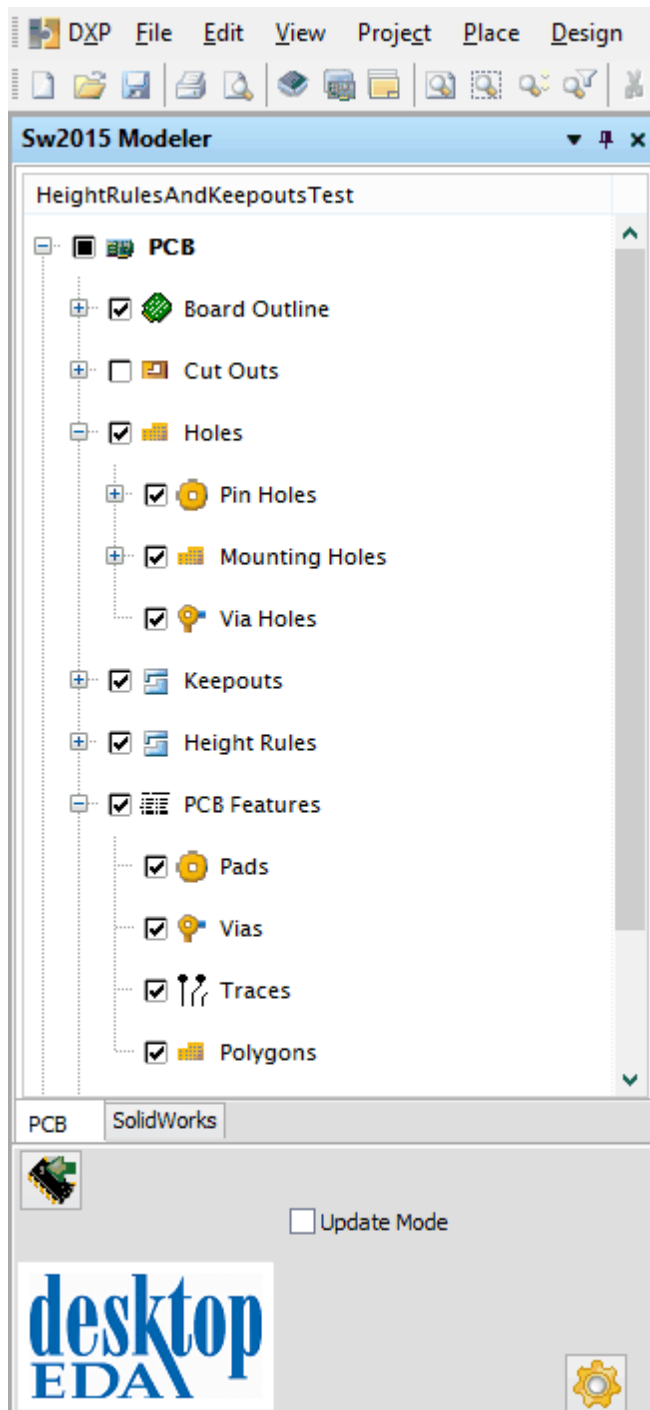
The various functions that can be performed on the panel are described below:

Tree View of the PCB

When you click on the PCB tab, a tree view of the elements contained in the current PCB is displayed in the tree view. Each node in the tree view has a check box. When a check box is checked, the elements for that node are sent to SOLIDWORKS when Export to SOLIDWORKS is invoked.

Double clicking on any node invokes display filtering for the elements associated with that node. For example, if you double click on the Plated Pin Holes node, display filtering is applied Plated Pin Holes are hi-lighted.

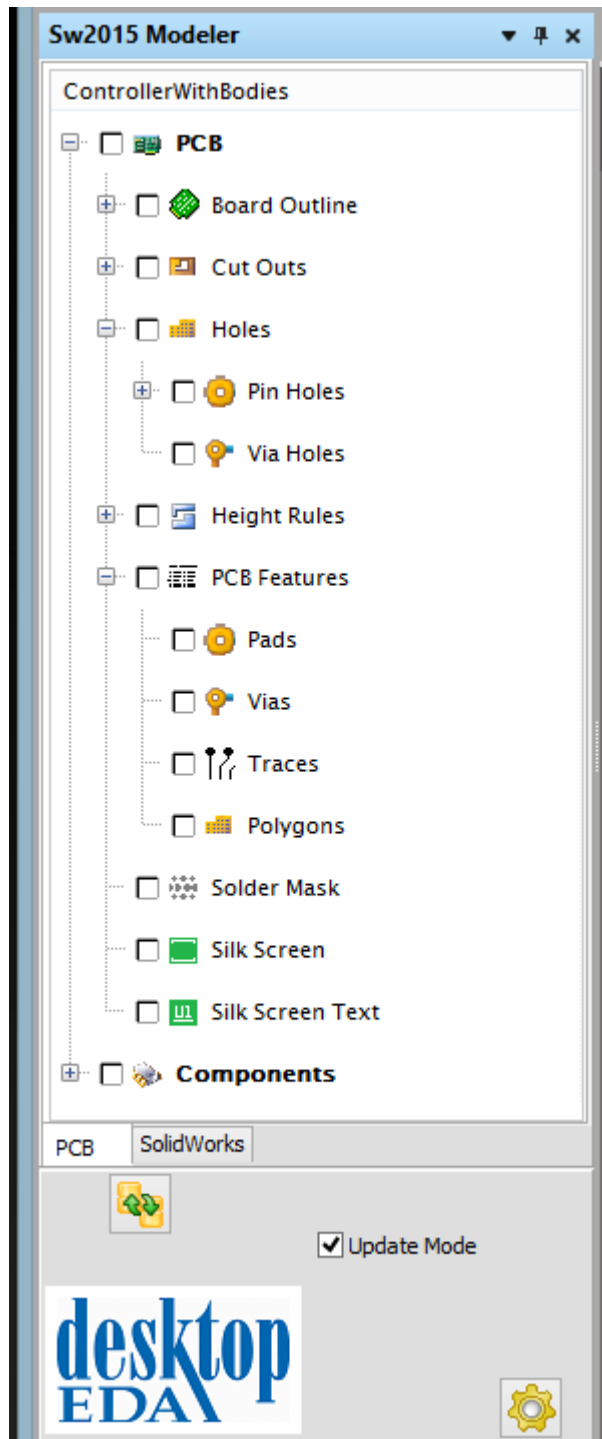
When a node in the tree has child nodes, clicking on the node will toggle the state of the child nodes. This is useful to check or un-check all nodes in a particular branch of the tree view.



Updating SOLIDWORKS

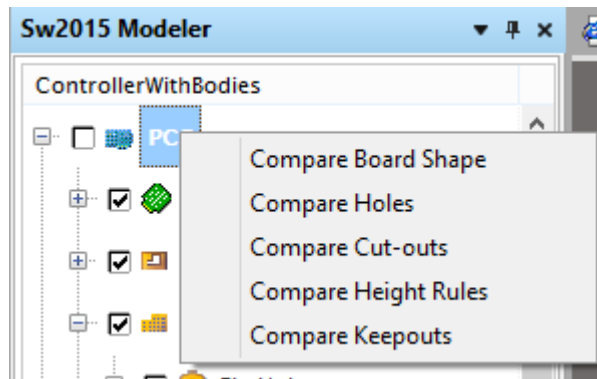
When the Update Mode check box is checked, the Export to SOLIDWORKS button is made invisible and the Update button is displayed. In this mode, you can update the document in SOLIDWORKS with changes made in the PCB.

Check any check boxes for elements you want to update in SOLIDWORKS, then press the Update button.



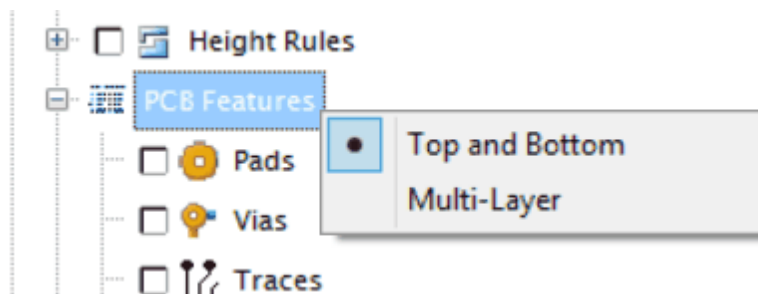
Right Click menus

Right click on the PCB node to invoke the PCB Compare commands. These compare the SOLIDWORKS PCB features and the Altium elements.

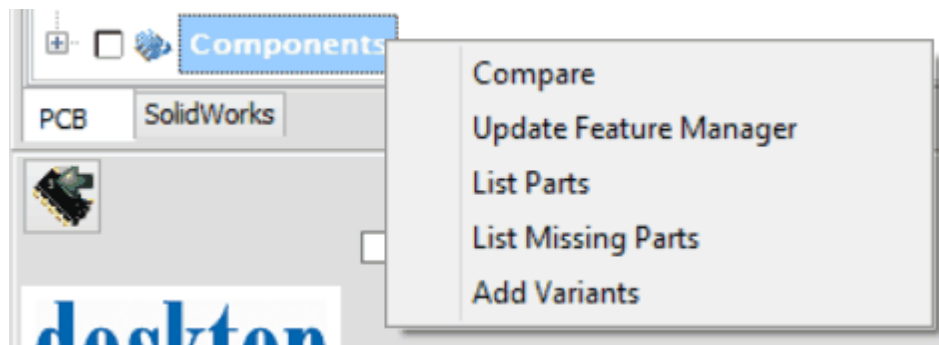


Right clicking on the PCB Features node allows you to switch between Top and Bottom mode and Multi-Layer mode.

In Top and Bottom mode, if any of the PCB Features check boxes are checked, the features for the top layer and the bottom layer are sent to SOLIDWORKS. In Multi-layer mode, features for all layers are sent to SOLIDWORKS.



Right clicking on the Components node allows you to invoke commands related to Components. The table below describes these commands.



Components node right click menu commands

Command	Description
Compare	Reports on the differences in components between the SOLIDWORKS assembly and the PCB
Update Feature Manager	Updates the Feature Manager to display to the current setting on the Options tab of the Preferences dialog
List Parts	Reports the file names of the parts that will be used in the SOLIDWORKS assembly. Use this to check that the program is

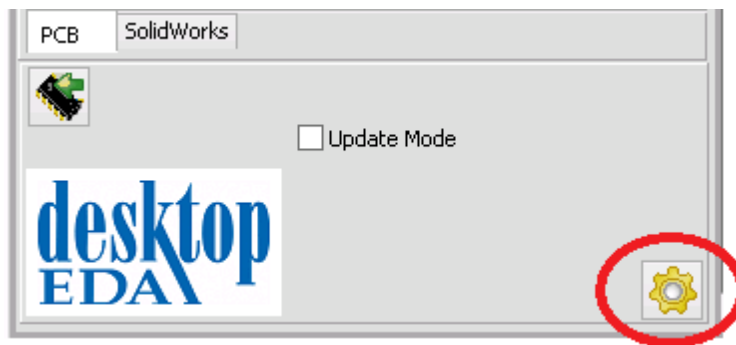
List Missing Parts
Add Variants

finding the correct SOLIDWORKS parts.
Lists any Footprints that do not have an associated Part
Creates a configuration for each Variant defined in the PCB

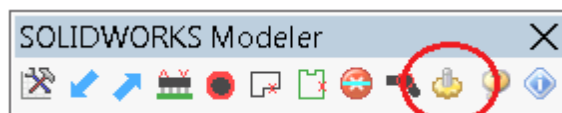
1.6 Preferences Dialog

Any settings or options for this product are accessible via the Preferences dialog.

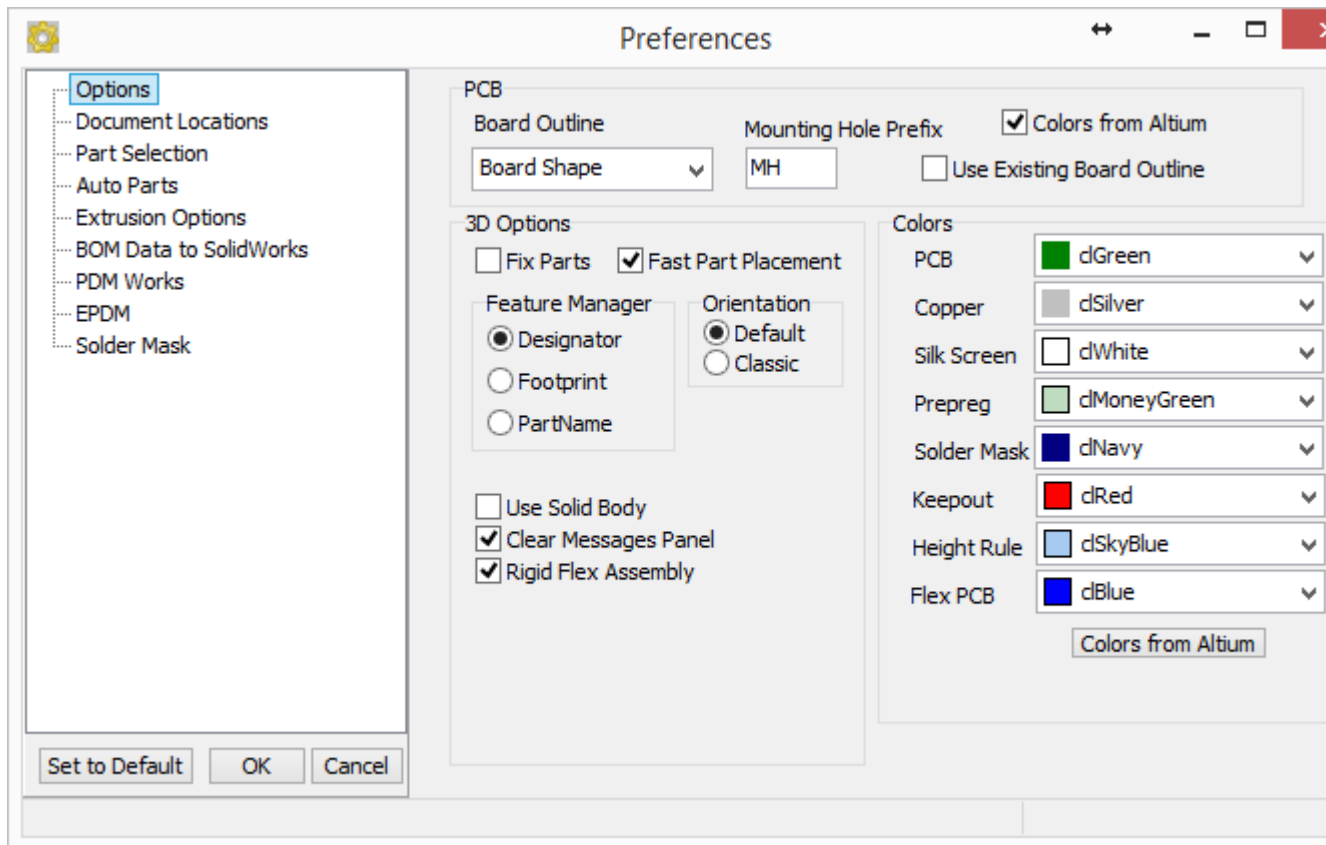
You can invoke the Preferences dialog from a button on the Panel



or from the toolbar



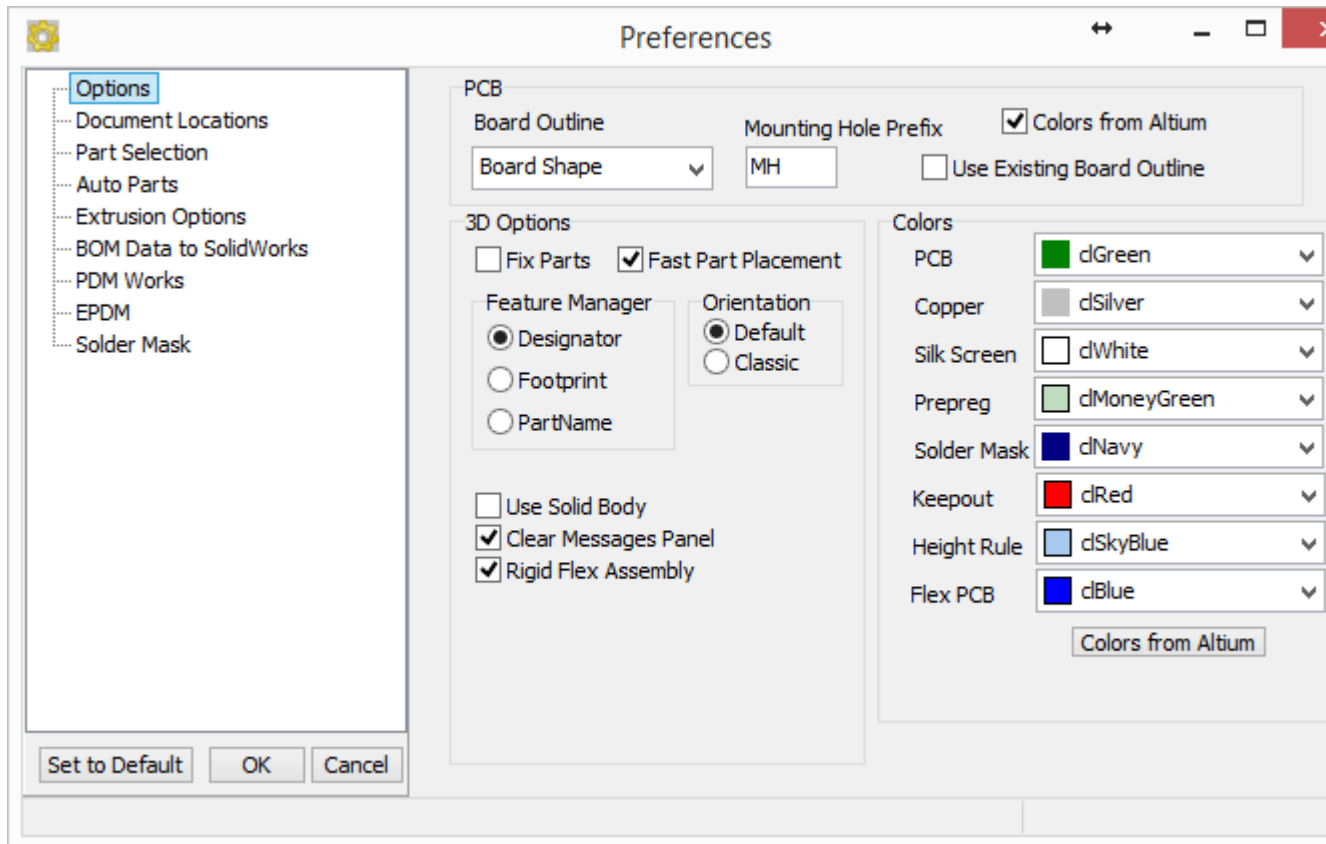
When the preferences command is invoked, this dialog displays:



The settings on the various pages of this dialog are described in the following topics.

1.6.1 Options

The settings on this page are described below.



PCB Section

Option	Description
Board Outline	Determines where the program gets the board Outline geometry from. The default is the Board Shape, if you click on the down arrow and you can choose from the Board Shape or an enabled Mechanical Layer
Mounting Hole Prefix	A Pad is assigned as a mounting hole setting the Pad designator to use the prefix assigned here for example MH1, MH1
Colors from Altium Check box	When checked, the SOLIDWORKS features use colors assigned in the selected PCB View Configuration in the View Configurations dialog (L key)
Use Existing Board Outline	If checked, if the Board Outline part exists, it is used as the Board Outline part when creating an assembly.

3D Options Section

Option	Description
Fix Parts	When checked, SOLIDWORKS parts are constrained by setting the Fixed property
Fast Placement	When checked, all parts are placed in one action. Otherwise,

	parts are placed sequentially. Fast Place is the default.
Feature Manager	Determines the Feature Manager display name for each part.
Orientation	Determines the reference plane for the assembly. Default uses the Front (X-Y) plane, Classic uses the Top (X-Y) plane. The option is provided for compatibility with earlier versions of the product. Use Default unless you have a parts library build in the X-Z plane.
Use Solid Body	When checked, the top face geometry of the Board Outline feature is used for Update from SOLIDWORKS commands
Clear Messages Panel	When checked, the Altium Messages Panel is clear for each run of the program
Rigid Flex Assembly	When checked, if the current PCB is a Rigid Flex PCB, the SOLIDWORKS assembly will be created as a Rigid Flex assembly containing a PCB for each PCB making up the PCB design, otherwise the SOLIDWORKS PCB is created as a single PCB.
Solder Mask Opacity	Sets the degree of transparency of the solder mask feature

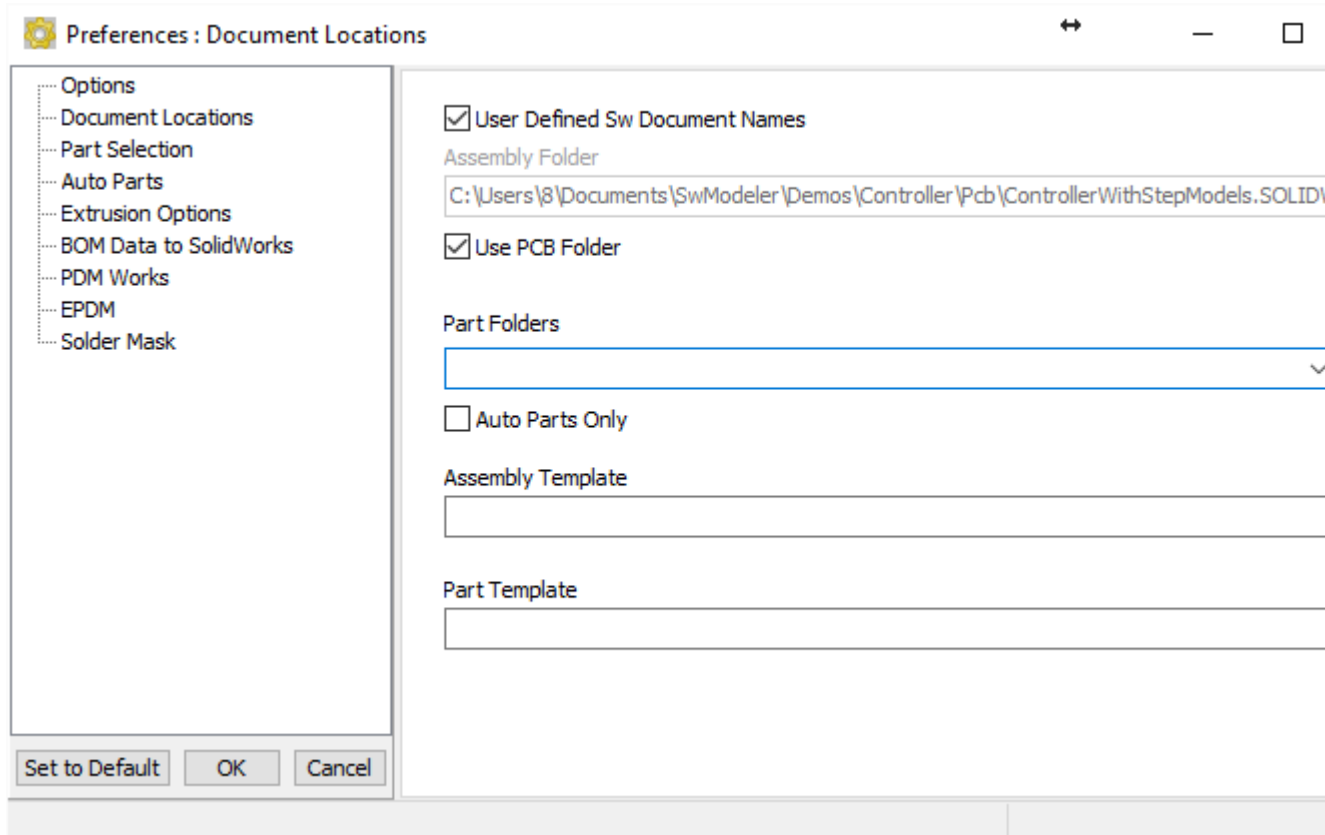
Colors Section

The color settings here are used to set the colors of features in SOLIDWORKS.

Pressing the Colors for Altium loads the color settings from the Altium 3D view - this only works when the PCB displays the 3D View.

1.6.2 Document Locations

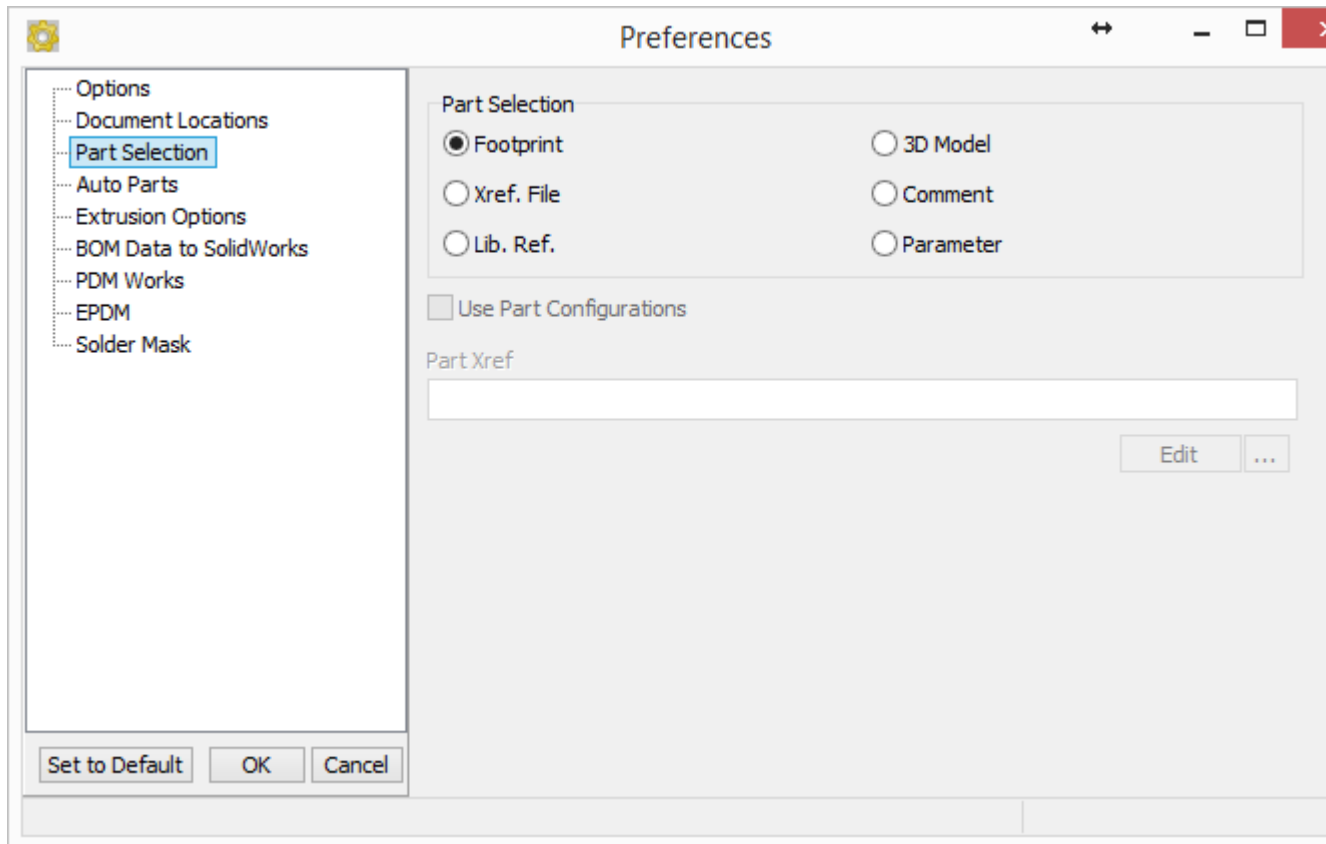
The settings on this page are described below.



Option	Description
User Defined Sw Document Names check box	When checked, you are prompted the the name of the SOLIDWORKS Assembly and Board Outline part. When they are created, their names are added to the current Project. (Note: this setting is ignored if the PCB is added to a project or the project is not active.
Assembly Folder	Determines the folder where the Assembly is created. Click on the browse button to the right of this field to browse to the assembly folder.
Use PCB Folder check box	When check, the assembly is created in the same folder as the PCB document, overriding the Assembly folder setting.
Part Folders	This drop down list holds a list of folder names that the program searches looking for part names to match Footprint names. Click on the browse button to the right of this field to add folders to this list (and set the order of the search)
Auto Parts Only check box	When checked, the program does not search for existing SOLIDWORKS parts, it simply creates a part for each footprint.
Assembly Template	Specifies the SOLIDWORKS template that is used when the program creates a new Assembly
Part Template	Specifies the SOLIDWORKS template that is used when the program creates a new Part

1.6.3 Part Selection

The settings on this page are described below.



Options	Description
Part Selection	Determines how Parts are matched to Footprints as described in the table below this table.
Use Part Configurations	This option is enabled when the Xref File part selection is active. This allows you to relate a configuration of a part to a Footprint
Part Xref	This field displays the name of the Xref. File. Click on the browse button to the lower right of this field to browse to an existing Xref file. Use the Edit button to invoke the Xref Editor to create or edit an Xref. file

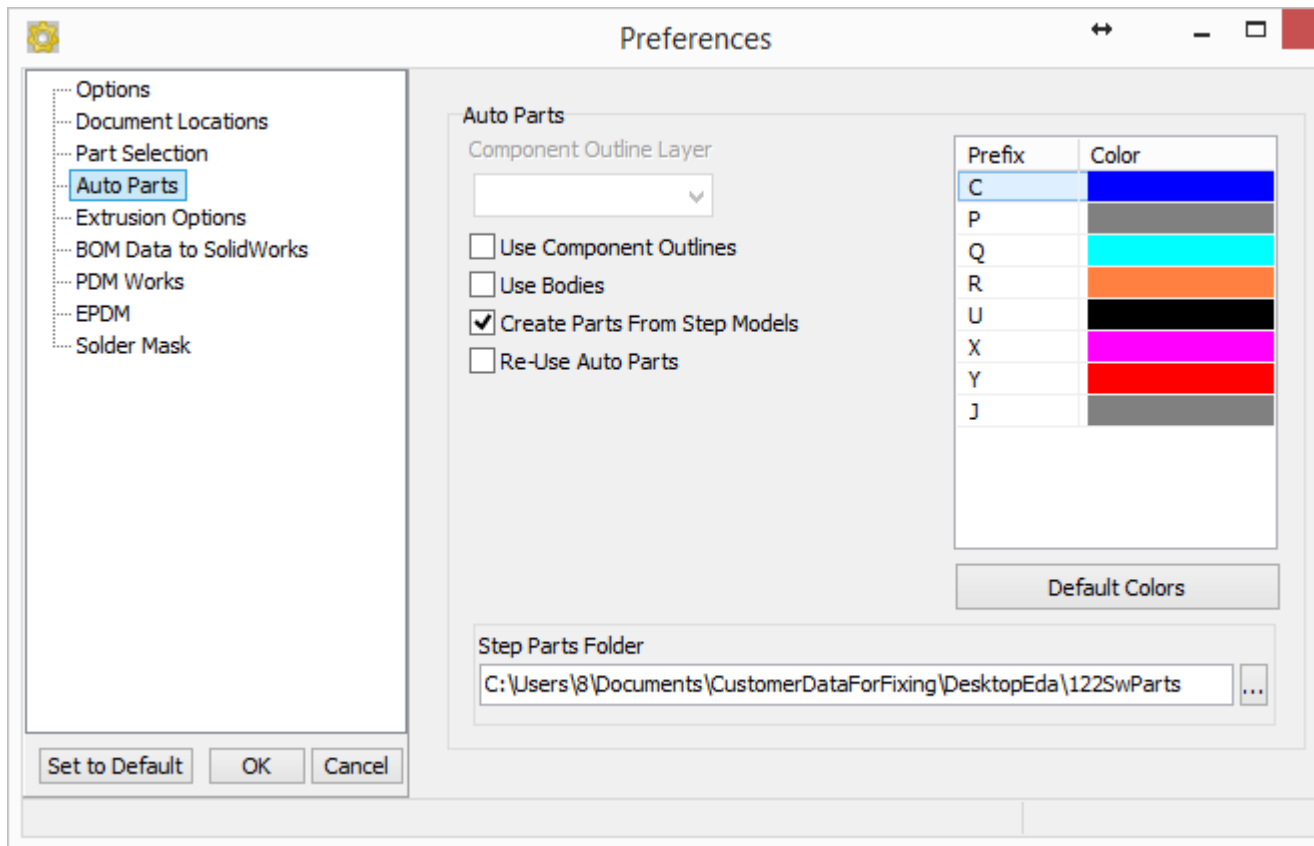
Part Selection Items

Part Selection Item	Description
Footprint	Part with the same name as the Footprint is used
Xref. File	The part name is obtained from a text file that relates Footprint names to Part Names. When this option is selected, the Part Xref field is enabled to allow you to create/edit the Xref file.
Lib. Ref.	The part name is read from the Lib. Ref. field of the component
3D Model	The part name is read from the 3D Model field of the component
Comment	The part name is read from the Comment field of the component
Parameter	The part name is read from a Parameter assigned to the Components schematic component. The name of this Parameter must be :

Solid Model.

1.6.4 Auto Parts

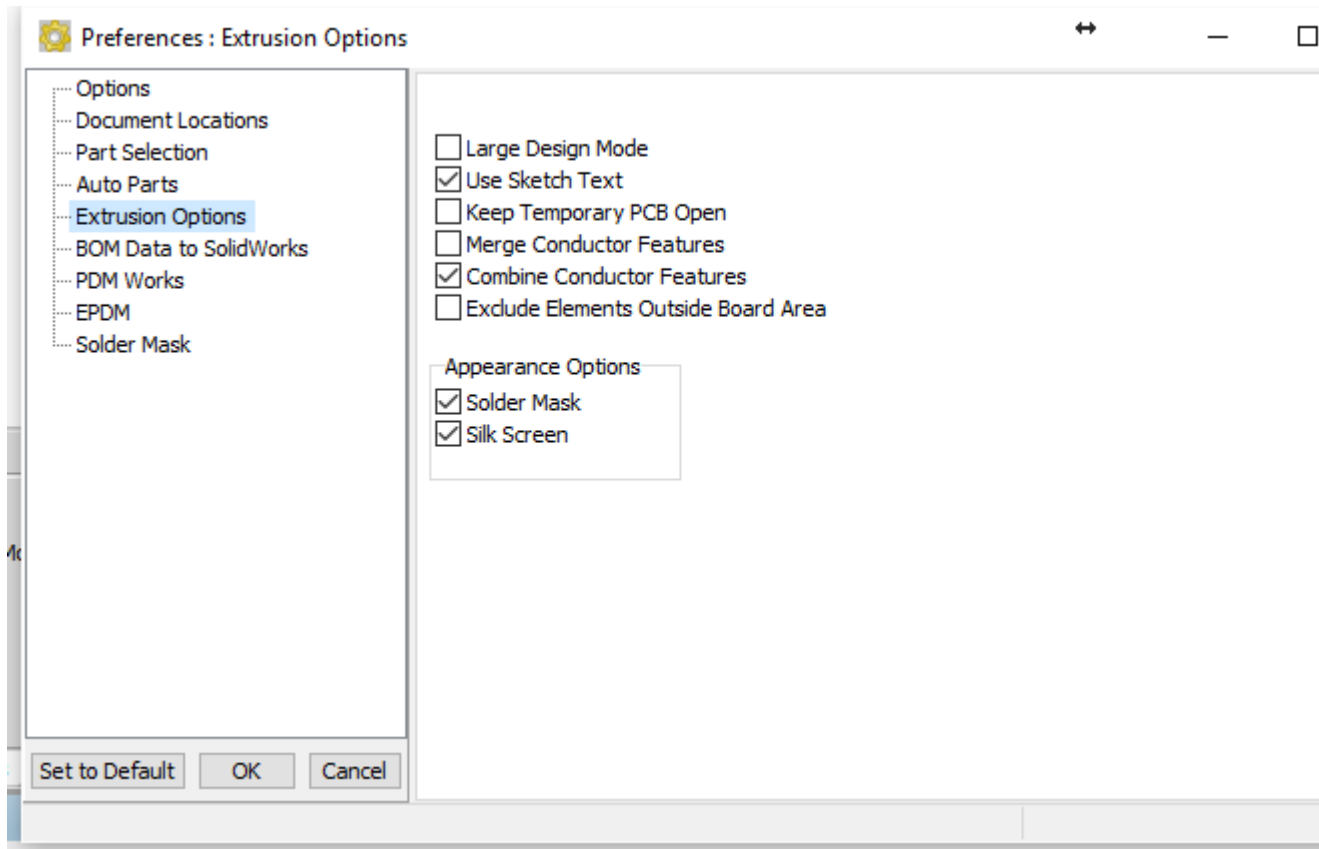
The settings on this page are described below.



Option	Description
Component Outline Layer	Set the Mechanical layer the program searches to get the shape to use to create the extrusion for for the part
Use Bodies	Uses bodies in the component to create the part
Re-use Auto Parts	When checked, if auto parts exist, they use used to create the assembly.
Create Parts from Step Models	The parts are created from Embedded Step Models if they exist
Colors table	This table sets the color used when creating a simple extrusion from geometry in the component. The color is chosen by the first letter if the Designator.
Step Parts Folder	If a folder is entered into this field, the Parts resulting from Step Model conversion are stored here. Otherwise, they are stored in a folder named <PcbName>.STEP_PARTS in the folder when the PCB is located.

1.6.5 Extrusion Options

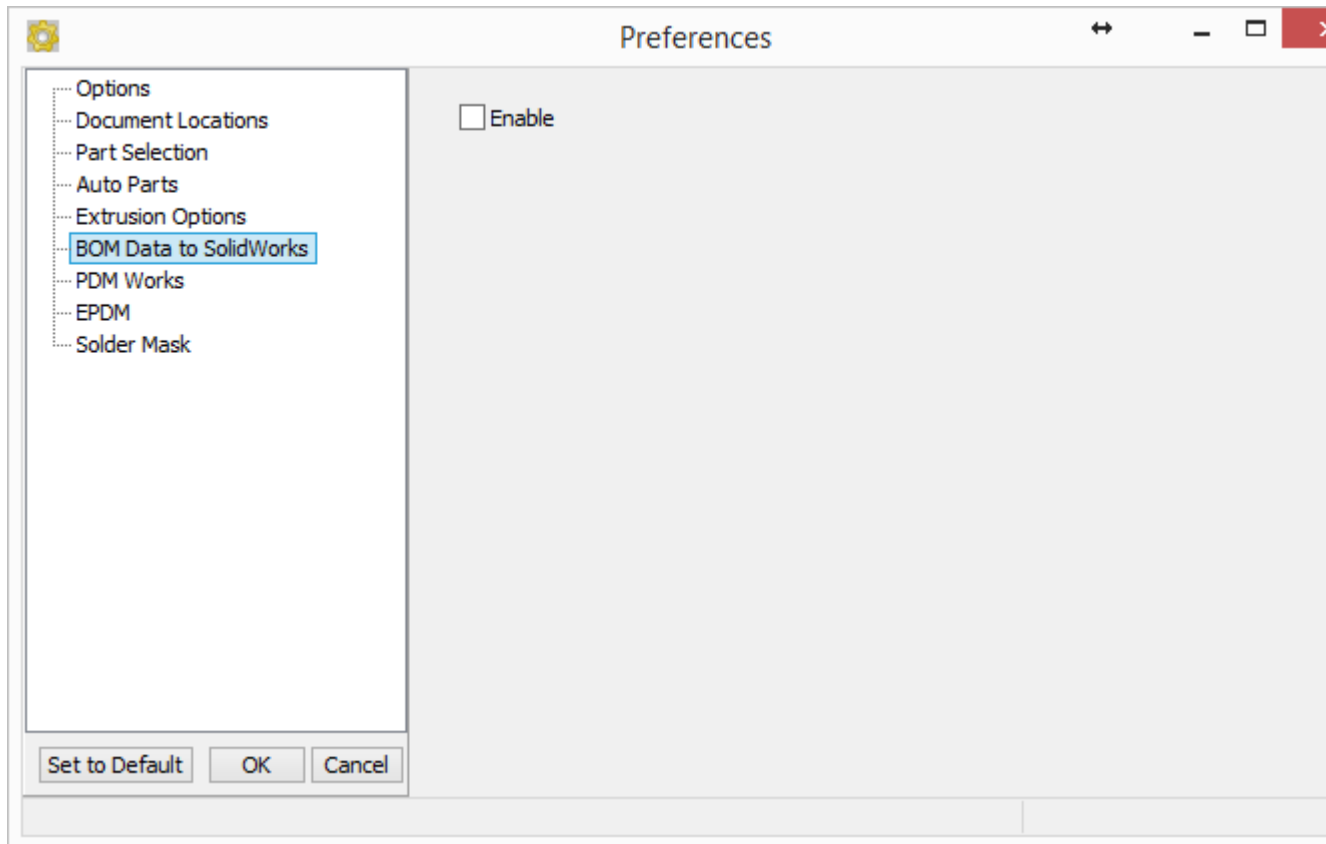
The settings on this page are described below.



Option	Description
Large Design Mode	When checked, each feature is separated up into multiple features. The Shapes Per Feature field sets the number of shapes for each feature
Use Sketch Text	When checked, text in the PCB is matched to a windows font and placed into the Sketch as test elements. When un-checked, stroke font text is added to the sketch as shapes. True Type fonts are always handled as sketch text
Keep Temporary PCB open	When checked, a PCB document is created that contains the shapes that we sent to the SOLIDWORKS sketch. This can use useful in trouble shooting for extrusion failures
Merge Conductor Features	When checked, the Bodies of conductor features that are connected are combined into a single body.
Combine Conductor Features	When checked, pads, vias and traces on each layer are combined into a single feature
Exclude Elements Outside Board Area	When checked, elements outside the board area are trimmed. This applies to conductor, silk screen, solder mask layers and holes
Solder Mask	When checked, the Solder Mask features have the Blue Thick Glass appearance applied. Use this when the Altium blue colored Solder Mask is used.
Silk Screen	When checked, the Silk Screen features have luminosity applied. Use this when the Silk Screen color is white.

1.6.6 BOM Data to SolidWorks

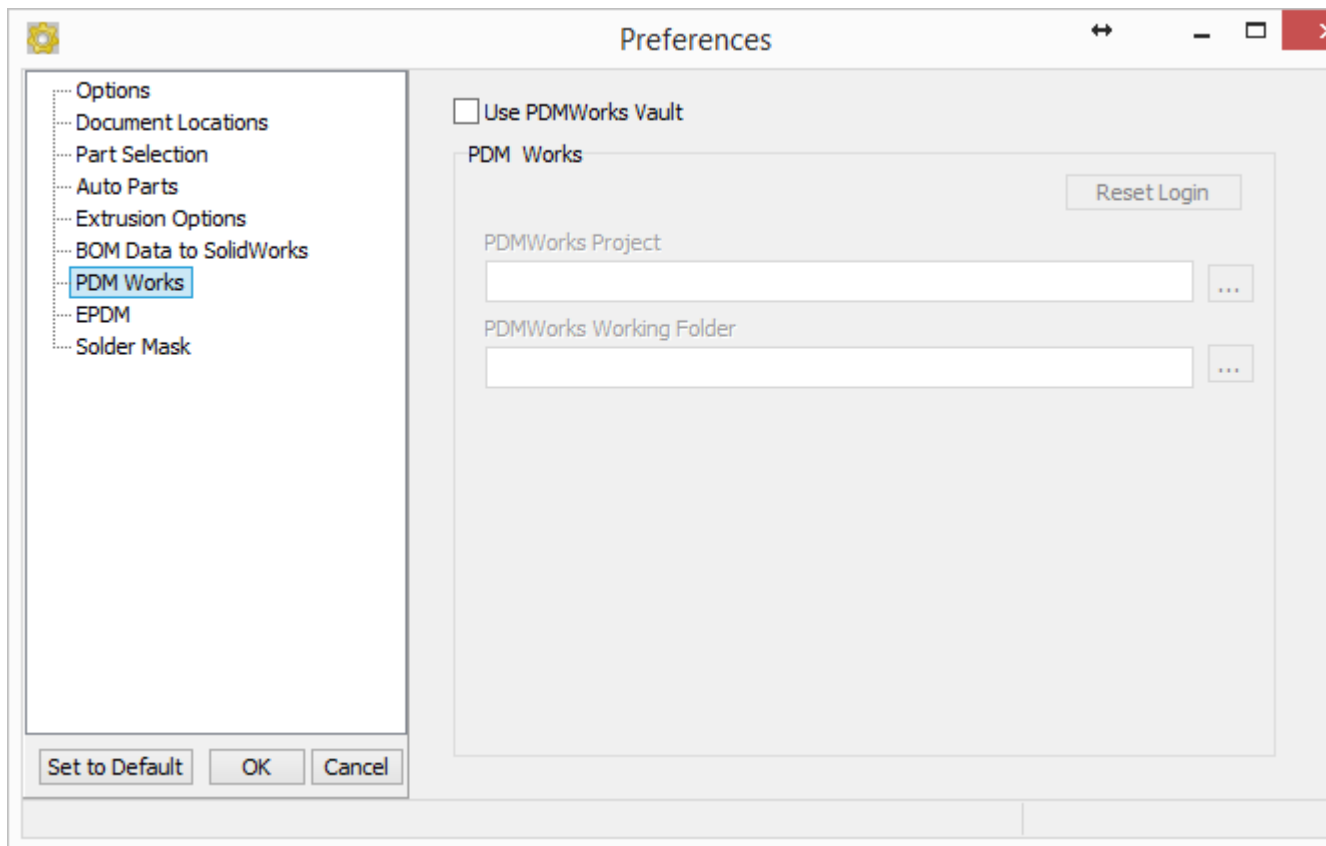
The settings on this page are described below.



The BOM Data to SOLIDWORKS function uses values assigned to schematic parameters to be added to Parts as Custom Properties.

1.6.7 PDM Works

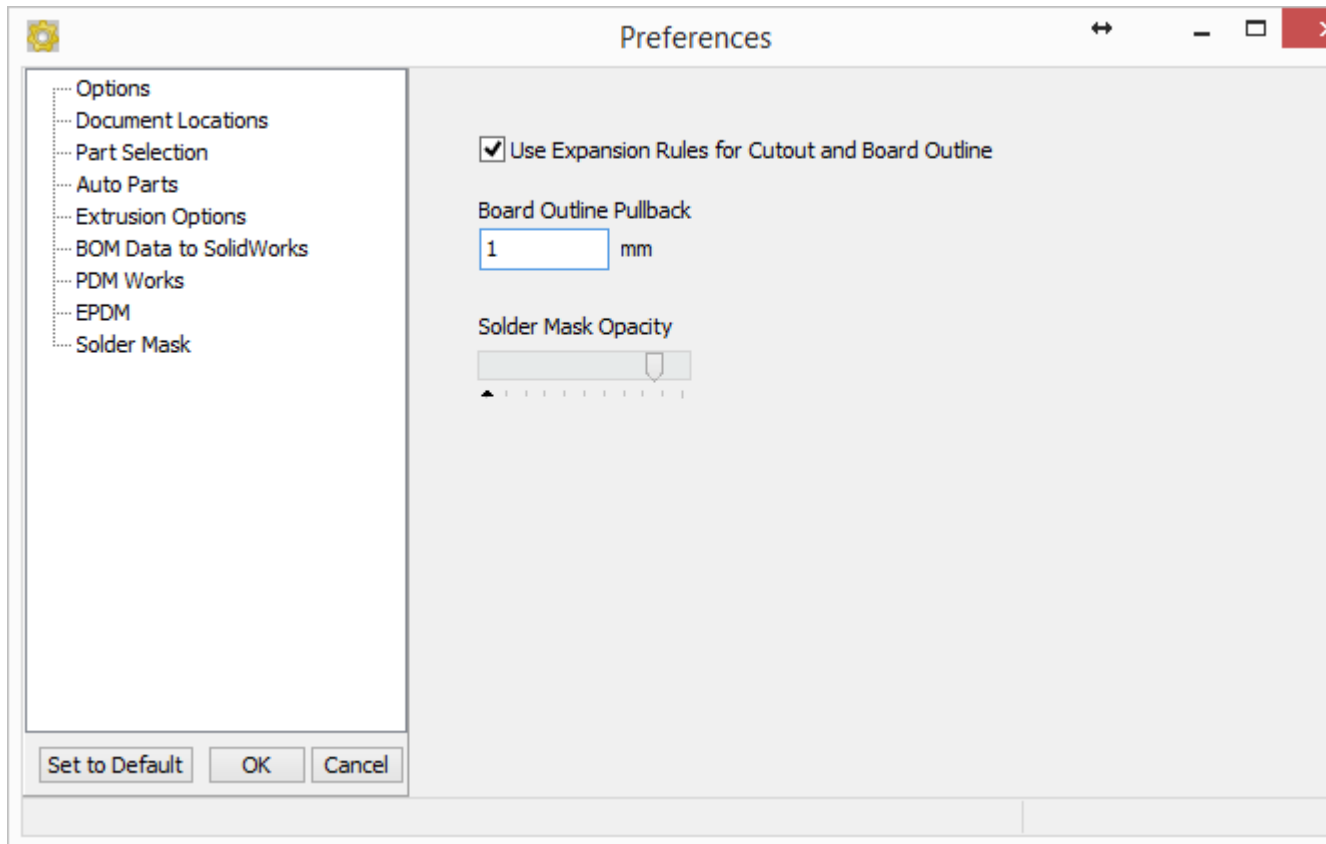
The settings on this page are described below.



When this option is enabled, parts in the PDM Works vault are checked out and used in the assembly.

1.6.8 Solder Mask

The settings on this page are described below.



Option	Description
Use Expansion Rules for Cutout and Board Outline	When checked, the expansion rules are used for set the Solder Mask gap for Cutouts. If this is used, traces and arcs on the Solder mask layers are ignored
Board Outline Pullback	If the above check box is checked, if a solder mask gap from the board edge is required, set this value.
Solder Mask opacity	Allows you to set the level of transparency for the Solder Mask extrusion

1.7 Interpreting the Altium Layer Stack Manager

When layer information is extracted from the Altium Layer Stack Manager, each Altium layer has conductor information such as its material and its type (IE signal or plane). In addition, Altium layers holds information about the dielectric immediately below it, IE whether it is a Core or a Prepreg. The bottom layer does not have a dielectric.

The Altium Modeler for SOLIDWORKS interprets this information and arranges the Altium layers into layer pairs based on the layer stack style assigned in the Layer Stack Manager.

If the PCB is assigned as Internal Layer Pairs stack up, the top layer is assigned as Prepreg with

conductors on the top, the next layer is a core with conductors and the top and the bottom. This continues until the bottom layer, where a Prepreg is added with the bottom layer conductors placed on the bottom side if the Prepreg.

If the PCB is assigned as Layer Pairs stack up, the top layer is assigned as a Core with copper on the top and bottom, the next layer is a Prepreg with no conductors followed a core with copper on the top and bottom conductors. This continues until the bottom layer - this is a Core with conductors on the top and the bottom.

For Custom stack up, if the dielectric is not assigned for the layers, the program will attempt to create a suitable stack up based on the number of layers and other factors. If the dielectrics are assigned, the program will adhere to these provided it can create an appropriate set of layer pairs.

Build Up style layer stack up is not yet supported.

The board thickness is calculated from the stack up.

1.8 SolidWorks Functions

The following topics describe the process of sending data to SOLIDWORKS and getting data back to the PCB.

1.8.1 Setting Up For Export

You need to do the following in the PCB document to before creating an assembly or PCB part:

To define the board outline that will be created in SOLIDWORKS, you can use either the Board Shape or an enclosed shape of tracks and arcs on a Mechanical layer. You can specify this by double clicking on the node below Board Outline node to display a list of options.

To allow the program to differentiate between the Board Shape and Board Cut-outs, if you rename (in Board Planning Mode) the Board Name from the default setting, the Board Name must contain the words 'Default' or 'Stack'.

To define free pads as mounting hole pads, assign the pad designator with a prefix. (e.g. MH). The Designator Prefix is specified on the in the Options page of the Preferences dialog.

The thickness of the PCB in SOLIDWORKS is read from the Layer Stack Manager

If there is no SOLIDWORKS part corresponding to components on the PCB, the program automatically creates a Part. These are referred to as Auto-Parts. These parts are created by extracting geometry from the PCB component in the following order:

1. If the Component contains an Embedded Step Model and the Create Parts From Step Models check box on the Auto Parts page of the Preferences dialog is checked, then the Step Models are converted to Parts.
2. If the Component contains Bodies and the Use Bodies check box on the Auto Parts page of the Preferences dialog is checked, the Bodies are used.
3. If there is an enclosed shape on the Mechanical Layer nominated by the Component Outline Layer on the Auto Parts page of the Preferences dialog, (and you have checked the Use Component Outlines check box) then this shape is used to create the part.
4. If there is an enclosed shape on the Silk-screen Layer, this is used to create the Part.
5. If none of the above exist, a shape is created from the bounding rectangle of the Component

(ignoring the Designator and Comment).

If a shape is used, the program also looks for height information in the Component. You set component heights using the Height Editor which is invoked from the product toolbar. This height is used to determine the height of the extrusion that forms the Auto-Part. (This height value is stored in the Height property of the Component).

You can set the color of the various elements of the PCB Part in the Colors section on the Options page of the Preferences dialog.

The color of the extrusion of an Auto Part created from items 2,3, or 4 above is determined by the Designator. You can set the color associated with a Designator in the table on the Auto Parts page of the Preferences dialog.

1.8.2 Step Model Conversion to Parts

Step Model conversion is enabled using the Create Parts From Step Models check box on the Auto Parts page in the Preferences dialog.

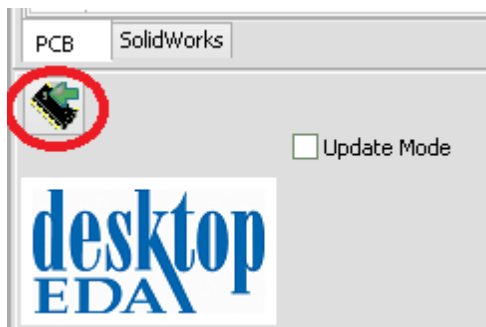
This feature should be viewed as a tool for adding parts to your parts library rather than being the prime method of creating parts.

Step Model conversion is time consuming and cannot be guaranteed to create 100% of the parts. This feature uses the SOLIDWORKS step model conversion command. This function is very reliable but will not handle every step model for every source.

It is recommended that, as you convert parts, you check them for accuracy and if they are acceptable, add them to your SOLIDWORKS parts library.

1.8.3 Creating an Assembly or PCB Part

Pressing the button shown invokes the creation of a SOLIDWORKS document to model your PCB.

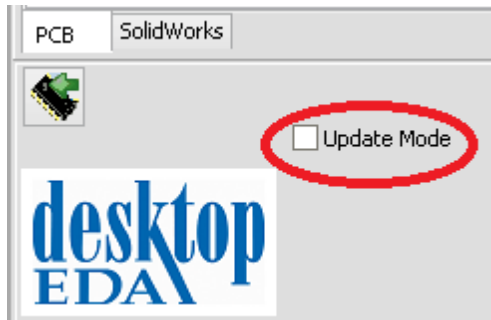


If the Components check box is checked or and component nodes are checked, an assembly is created.

If no component nodes are checked, a part modeling the PCB without components is created.

1.8.4 Updating an Assembly or Part

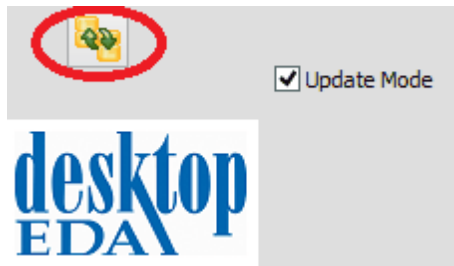
To update an Assembly or Board Outline part with changes made in Altium, check the Update Mode check box.



When this is checked, the Synchronize button displays (and the Export button is not displayed)

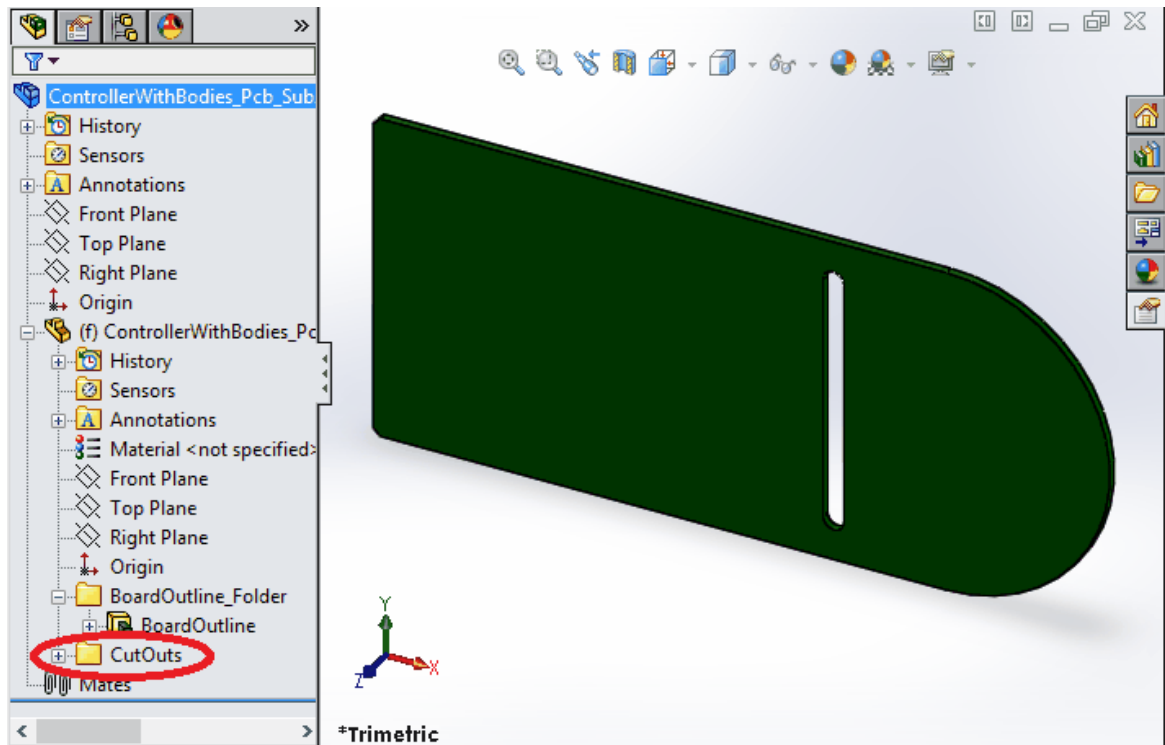
In Update mode, when the Synchronize button is pressed, features associated with checked check boxes are updated.

For example: Create a Board Outline assembly using 'ControllerWithBodies.Pcbdoc' file as shown below:



In Update Mode, Check Cut outs check box and then press 'Synchronize' button

The selected cut feature is sent to the SOLIDWORKS as shown below: Cut outs node is also added to the Feature tree

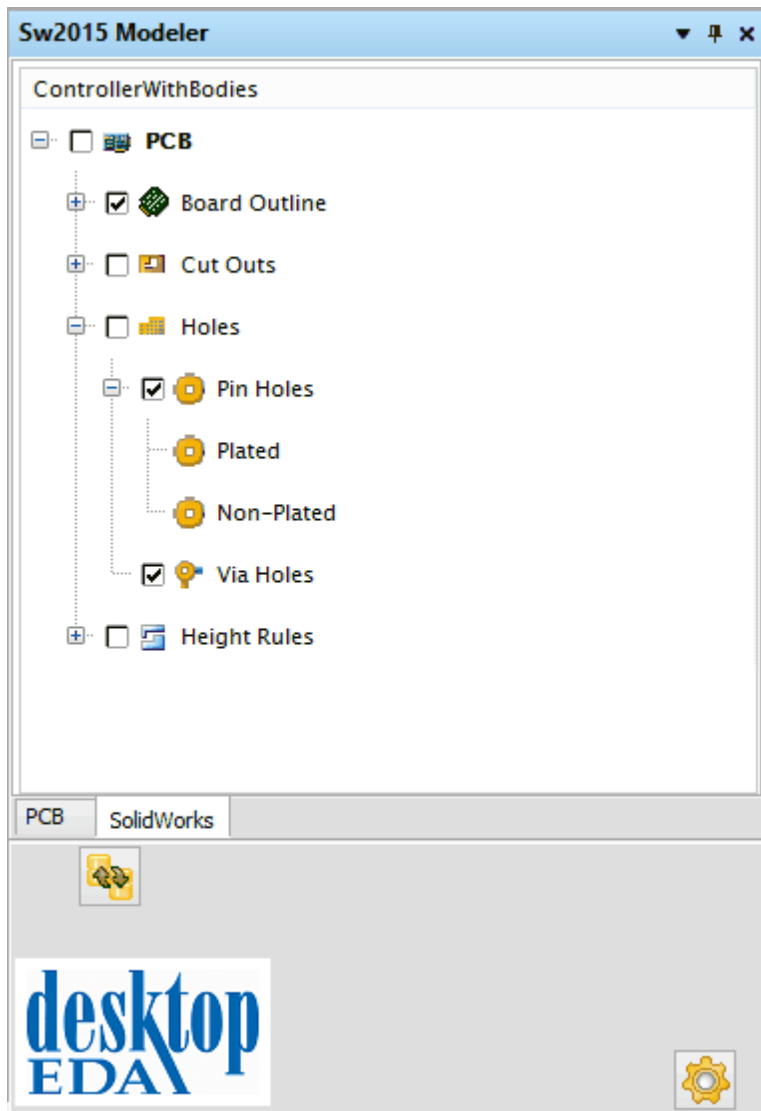


1.8.5 Updating the PCB From SolidWorks

If a SOLIDWORKS document exists for the current PCB, clicking on the SOLIDWORKS tab on the Panel, switches the panel into the SOLIDWORKS view.

In this view, the tree view displays nodes for each SOLIDWORKS feature.

If you edit features in SOLIDWORKS, you can update the PCB with the changes by checking the check box then pressing the Update button.



1.8.6 New PCB from SolidWorks

This command creates a New PCB document from an Assembly or Part that is currently open. This command can be accessed from the File>New>PCB from SOLIDWORKS menu item when no documents are open or the PCB editor File>New>PCB from SOLIDWORKS menu item.

1.8.7 Representing Variants in SolidWorks

If you have Variants defined in your PCB, an option is provided that creates a SOLIDWORKS Assembly configuration for each Variant defined in the PCB Project. Any PCB Components that are assigned as not fitted, are suppressed in the configuration for that Variant.

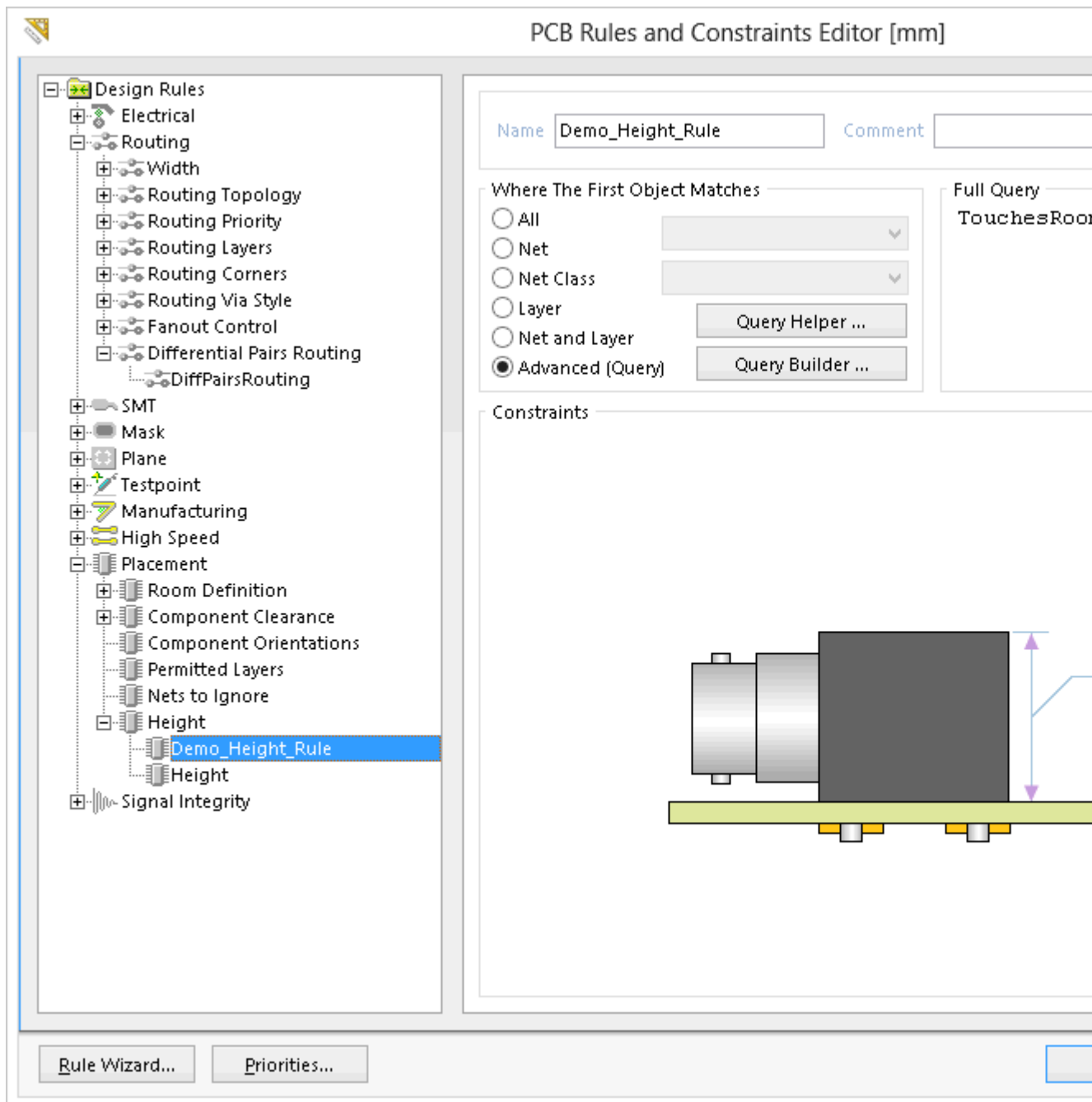
To invoke this command, right click on the Components node on the Panel, and select the Add

Variants menu item. The Altium project that the PCB is part of, must be open for this function to work.

1.8.8 Working With Height Rules

Altium Designer provides the capability of defining an area of the PCB using a Room Definition rule and associating a maximum component height with that area. This is achieved using Height rules that are defined in the Placement section of the PCB Rules and Constraints dialog box.

The definition of a Height Rule is shown below.



This rule is created after the Room Definition rule called 'HeightRuleTestRoom' has been created.

To create a feature in the Assembly that represents Height rules, check the Height Rules check box in the Panel.

You can also add Height rules to an existing assembly using the Update Outline Part command.

See [Starting Designs in SOLIDWORKS](#) topic from details on how to create Height rule features in the board outline part.

1.8.9 Working With Keepouts

Altium Designer provides the capability to define areas on routing layers as Keep-out areas. These are defined using the commands in Place>Keep-out. This Keep-out Fills and Regions are supported.

This product provides bi-directional support for these Keep-out shapes.

To create a Features in the Assembly that represent the Keep-outs, check the Keepouts check box in the Panel.

You can also add Keep-outs to an existing assembly using the Update Outline Part command.

See the PCB Tools topic for details on how to create Keepout features in the Assembly.

1.8.10 BOM Data to SolidWorks

This function allows data assigned to parameters in the PCB Project schematic to be loaded into SOLIDWORKS when the Assembly is created.

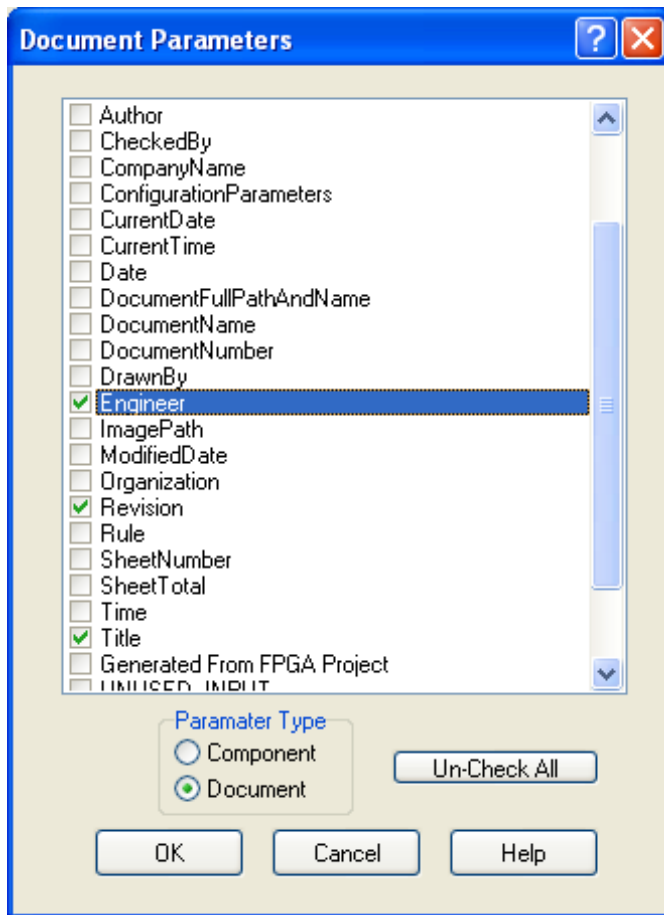
It is enabled by checking the Enable check box on the BOM Data to SOLIDWORKS page of the Preference dialog.

On this page you can perform the following:

Choose the Set Params button to display the dialog box below.

Choosing the Component radio button results in the names of all Schematic components in the project parameters being listed. Choosing the Document Radio button results in all Document parameters in all schematics in the project being listed.

The Un-Check All button, un-checks any checked checked boxes.



For Document Parameters:

Any document parameters whose check boxes are checked are added to the SOLIDWORKS assembly as Custom Properties assigned to the Default configuration.

For Schematic Parameters:

A configuration is automatically added to each SOLIDWORKS part. The name of the configuration is determined by the Library Ref property of the PCB component. For each parameter checked in the Parameters dialog box (see Set Params button) a Custom Property is added to this configuration of the SOLIDWORKS part. The name of the Custom Property is the Parameter name and the Value is the parameter Value.

Once the Assembly is created with the PDM Support option enabled, you can use the BOM functions in the SOLIDWORKS Drawing editor. If you set up a BOM template with each custom property added, these custom property values are automatically added to the BOM.

1.8.11 Setting Colors

You have the option of using the colors set in the Altium View Configurations dialog (Press the L key while in the PCB editor) or defining them in this product.

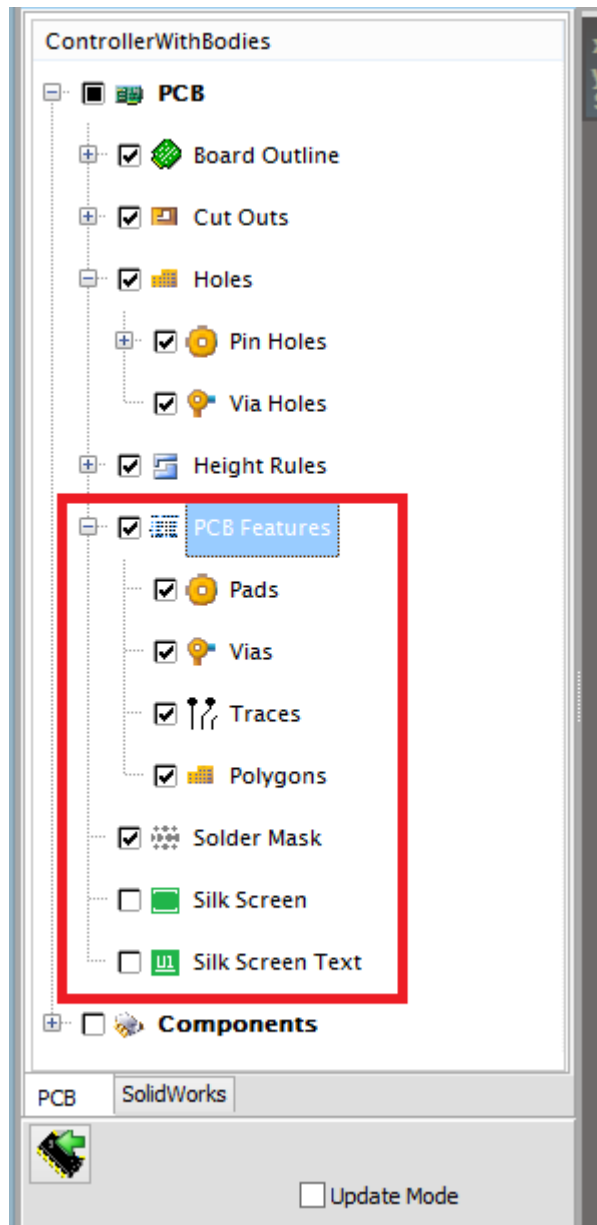
To use colors set in Altium, check the Colors from Altium check box on the Options page of the Preferences dialog.

To define your own colors uncheck the Colors from Altium check box and choose colors in the Colors section on the Options page of the Preferences dialog.

1.9 PCB Features in SolidWorks

The following topics describe how to model conductors, solder mask and silk screens in SOLIDWORKS.

PCB Features are selected by checking the appropriate check box in the Workspace Panel.



1.9.1 Guidelines on Exporting Tracks Polygons and Silk Screens

To model pads, via pads, traces, polygons, internal planes, solder mask and silk screen elements in SOLIDWORKS, the program creates shapes made up of lines and arcs and writes these to a SOLIDWORKS sketch. When all elements are written to the sketch, it is extruded. This process can involve transferring a large number of lines and arcs to SOLIDWORKS so it can be time consuming.

If you check the Use Sketch Text check box, Altium text is matched to a windows font and placed into the sketch as text elements - otherwise text is represented in SOLIDWORKS as extruded shapes. Using Sketch Text creates a much simpler feature in SOLIDWORKS.

The Sketch Text option is mainly provided for modeling True Type fonts as otherwise, the shapes defining true type fonts are made up of many short lines and can result in very complex features

which may fail to extrude.

Selection is supported, so if not all objects are required in SOLIDWORKS, select the elements that are required and only these are converted to shapes and transferred to SOLIDWORKS.

The Combine Conductor Features option is available on the Extrusion Options page of Preferences. When this is enabled, Pads, Via Pads and Traces are combined into a single feature on each layer and plating barrels are added to plated holes. On smaller designs, this provides a simpler feature. This option is also useful when the features is to be used in simulation.

1.9.2 Trouble Shooting Extrusion Problems

For SOLIDWORKS to successfully extrude a sketch, all shapes in the sketch must be perfectly enclosed.

When this program creates the shapes for extrusion it applies extensive checking and correction routines to create the required perfectly enclosed shapes.

Below is the process by which shapes on a layer are prepared for extrusion:

- Each element is converted into a shape (for example a trace is changed from a line with width to a shape made up of 2 lines with arcs at either end.
- Arcs are converted to interpolated lines so that the shapes are formed by lines only
- These shapes are added together. If they overlap, they are merged together
- The shapes are checked for zero length elements
- The merged shapes are checked for gaps, where possible gaps are closed.
- An Arcs to lines routine is applied to the shapes to reduce the number of elements that will be sent to the sketch.
- The shapes are scaled from Altium units to SOLIDWORKS units
- The shapes are checked for zero length elements
- A further gap check is applied

In spite of this, there will be occasions when a layer on a PCB will fail to extrude. The following topics provide some guidelines on how to firstly avoid creating extrusion problems and secondly, how to locate and correct them.

1.9.2.1 Potential Causes of Extrusion Problems

Below are some common causes of extrusion problems that should be avoided if possible:

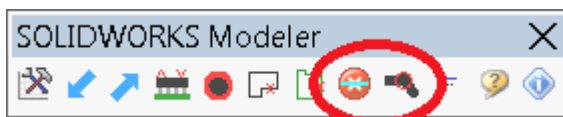
- Duplicate or zero length traces

It is not uncommon to find duplicate or zero length traces.

These sometimes effect extrusion and should be removed.

These commands, accessible on the toolbar are provided:

- o Check for Duplicates
- o Check for Zero length elements



- Traces not finishing on Pad centers
- Traces that are not orthogonal or at 45 degree angles
- Text placed at angles other than 0, 90, 180 or 270 degrees
- Overlapping text
- Touching Pads or Vias
- Touching regions
- Nested Shapes (ie SOLIDWORKS sees a shape within a shape as a void, but simply does not extrude a shape within a shape within a shape)
- Untidy Region vertices

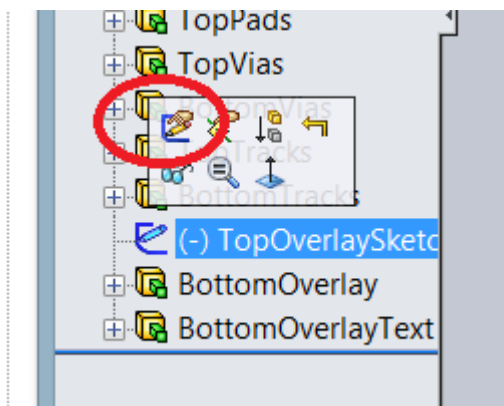
If you have place an arc as a circle on a conductor layer or on the silk screen layer, it will end up in SOLIDWORKS as a filled circle. To avoid this, place two arcs that form a circle.

1.9.2.2 Finding and Correcting Extrusion Failures

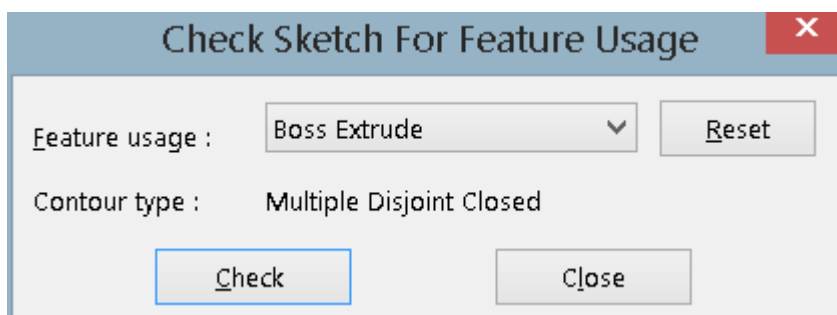
This topic describes the techniques for finding the cause extrusion failures.

For most failures, finding the issue in the sketch will allow you to find the cause. Below is the procedure to find the sketch element causing the failure:

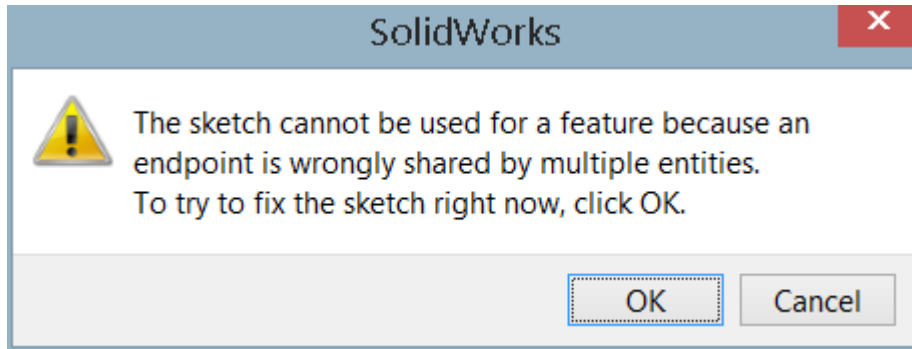
- 1) Edit the Sketch by clicking on the sketch entry in the feature tree and then click on the Edit Sketch icon shown:



- 2) Invoke this command: Tools>Sketch Tools>Check Sketch for Feature
- 3) This dialog displays - choose Boss Extrude in the Feature usage drop down list, then click OK



- 4) This message box displays, explaining the reason for the Extrusion failure:

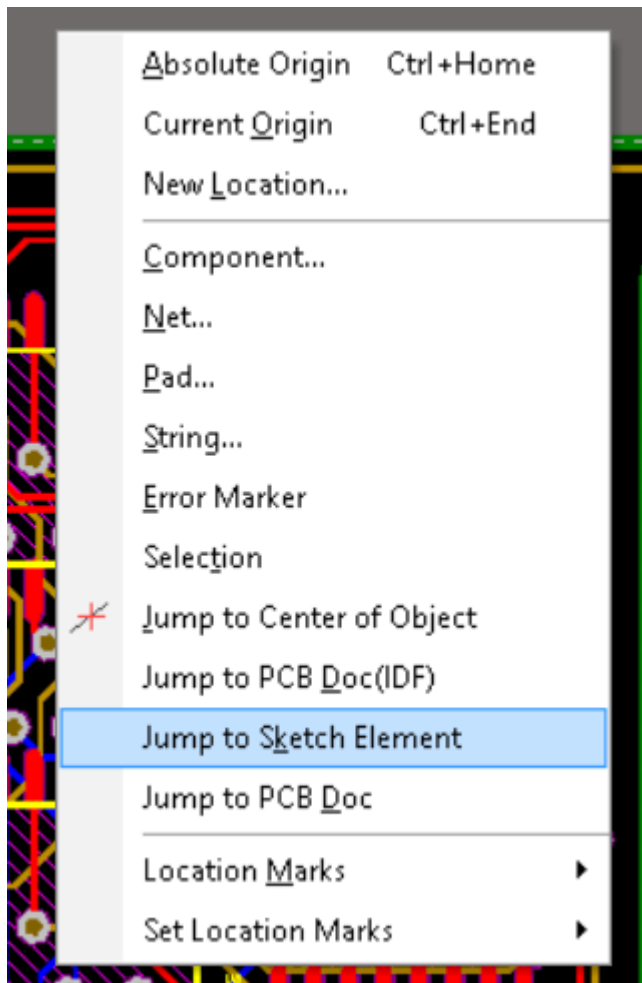


5) Click OK and the sketch element causing the failure will typically be selected.

If the above procedure returns with a message such as "There are no problems", invoke extrusion. An Error message will display telling you what the issue is.

To find the corresponding location in the PCB Document of the extrusion failure in SOLIDWORKS sketch:

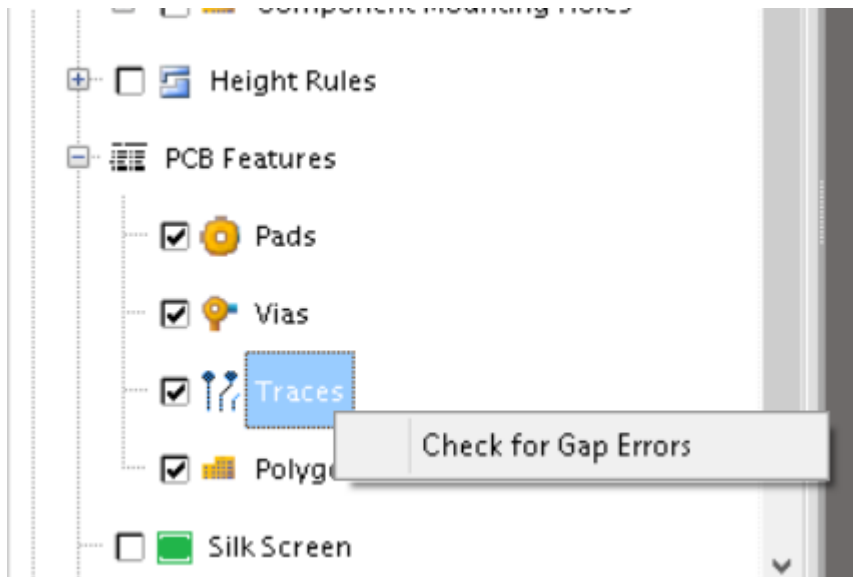
- Select the sketch arc or sketch line at the location hi-lighted by Check Sketch for Feature (only one sketch element should be selected)
- Make the PCB document active.
- Press the J key on the keyboard and choose the Jump to Sketch Element command shown below.
- The cursor will then be located at the location in the PCB document where the selected sketch element is.



If the above procedure does not allow you to find the cause of the failure, follow the procedure below:

1. Extrude each feature separately to determine if there are any extrusion issues (eg if the feature that fails is a trace feature, un-check all other PCB features check boxes so that only traces are extruded)
2. Any elements on the layer that do not belong to an enclosed shape are added to a sketch named GapErrors.
3. If a layer fails to extrude, go to the Preferences dialog and choose Large design Mode, and set the Shapes Per Feature setting to a value - a suggested starting value is 20. When you send PCB features to SOLIDWORKS with Large Design Mode, the shapes are separated into groups, the number determined by the Shapes Per Feature setting. Any of the groups of shapes that has an extrusion issue will be displayed as a sketch rather than an extruded feature.
4. To find the cause of the Extrusion failure, edit the sketch and invoke the SOLIDWORKS command Tools>Sketch Tools>Check Sketch for Feature as described above.. This command will tell why the extrusion failed and guide you to the elements causing the issue.
5. Go to the corresponding location on the PCB. You will most likely find something unusual or not quite right. If you do, correct it.
6. If you have time, (SOLIDWORKS will take longer to create all the features) set the Shapes Per Feature value to 1, This will create a feature for each shape and the failures will be very obvious as you will see a single shape as a sketch. Again, use Tools>Sketch Tools>Check Sketch for Feature will tell you what is wrong if you cannot immediately identify the cause.

Another command that can be used in locating extrusion issues is to right click on a PCB tools node and click on the Check for Gap Errors menu. This writes the shapes for the elements for that node to a temporary PCB document and selects any elements that have gap errors.



1.9.2.3 Internal Planes

Internal Planes are fully supported. When split planes are added to an internal plane layer, if the split plane divider line runs through an area with pads or vias, the resulting shapes may not be perfect and extrusion will fail. To resolve this, if possible, re-route the divider line to make the shapes simpler.

If this is not possible, you will need to correct the internal plane sketch. The errors will be around the edge of the split plane shapes. See the previous topic [Finding and Correcting Extrusion Failures](#) for how to find and fix sketches with gap errors. Once the errors are fixed, manually invoke extrusion.

1.9.3 Solder Masks

Checking the Solder Mask check box on the panel results in an extra Solder Mask extrusion feature being added to the Board Outline part.

The resulting solder mask feature will obey all Solder Mask Expansion design rules.

If you use polygons on the Solder Mask layer to define the Solder Mask, this is supported but you need to follow the guidelines in the next topic.

1.9.3.1 Solder Masks Using Polygons

Solder Mask polygons are supported provided you follow these guidelines:

- The polygon on the Solder Mask layer must cover the whole PCB.
- You define the areas where ink is to be deposited on the PCB using Polygon Cut-Out regions.

If the Polygon Cut-out overlaps other elements, the shape clipping may not extrude in SOLIDWORKS.

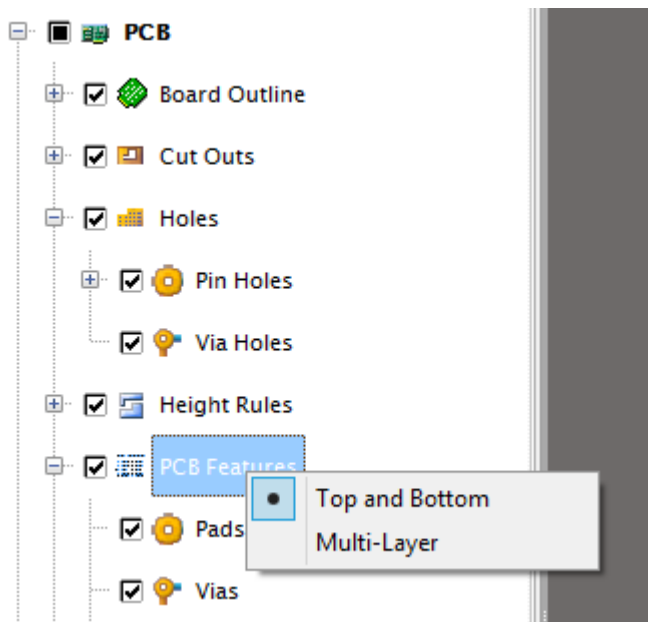
You can either correct the shape in SOLIDWORKS and manually extrude it, or you can edit the polygon cut-out to minimize the overlap with other elements. Doing this will most likely result in successful extrusion.

1.9.4 Two Layer or Multi-Layer

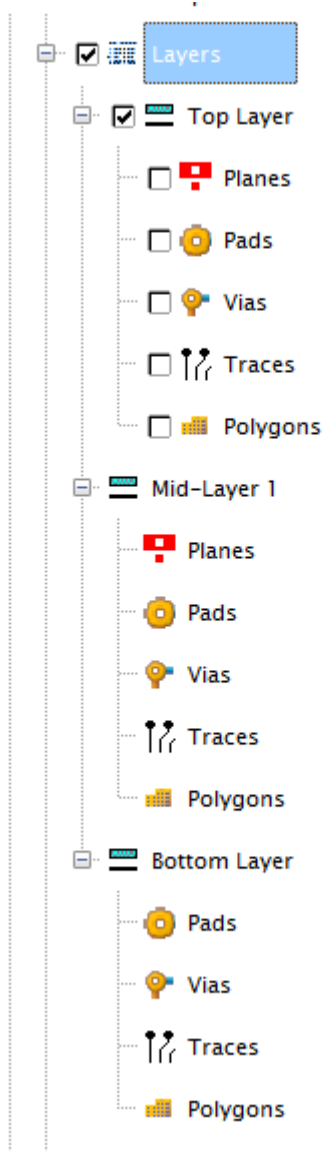
The SOLIDWORKS PCB can be created as a complete Multi-layer PCB or as a simple PCB with just the top and bottom layer elements modeled in SOLIDWORKS.

The simple PCB is known as Top and Bottom Mode.

The mode is set by right clicking on the top node in the PCB Features section of the Panel as shown below and select the mode required:



When Multi-layer mode is active, a node for each layer pair is added to the tree view on the panel. check box to enable or disable sending PCB features to SOLIDWORKS are only added to the Top Layer node. When these are checked, they apply to all layer pairs.



1.9.5 Adding Klipper Logos

Desktop EDA has a free product called Klipper that provides graphics tools for PCB users.

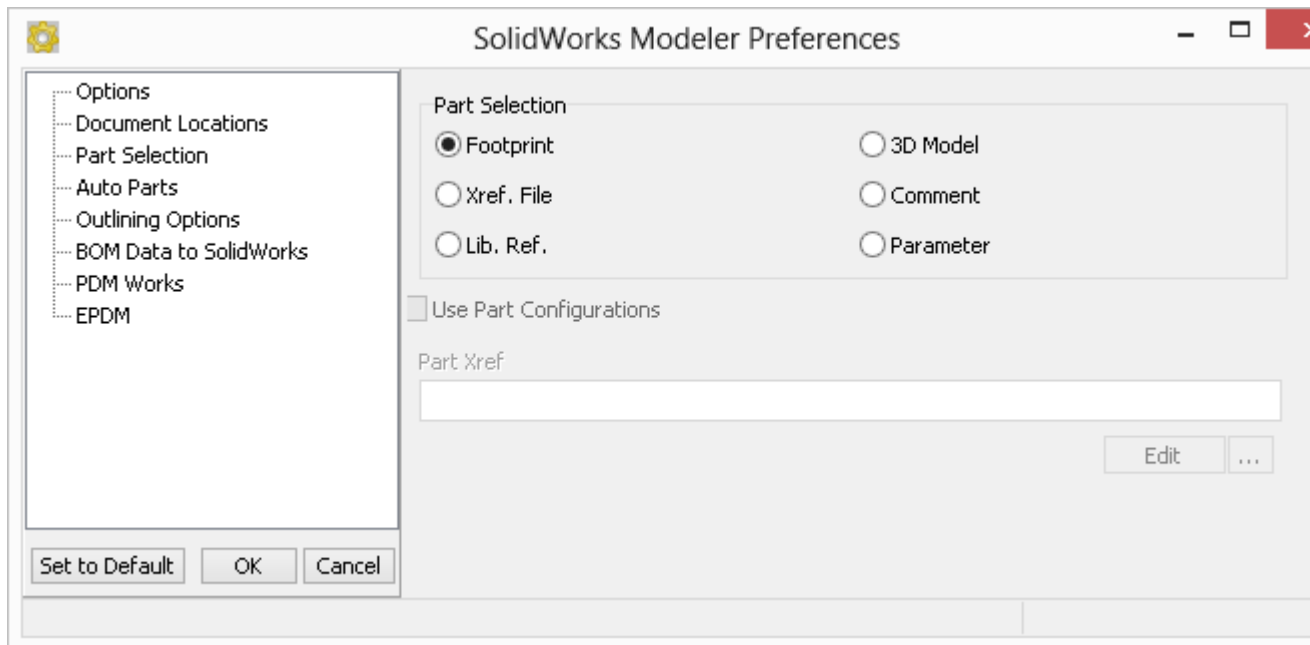
The Place Graphic command has an option to create a graphic using Regions with Voids. This creates a Regions with Holes if the graphic contains voids.

If your PCB contains Regions with Holes on a signal layer or a Silk Screen layer, these are handled separately so the holes are preserved. Regions extrude more reliably than Klipper graphics created using fills or tracks.

Contact Desktop EDA to get the version of Klipper that supports graphics with voids.

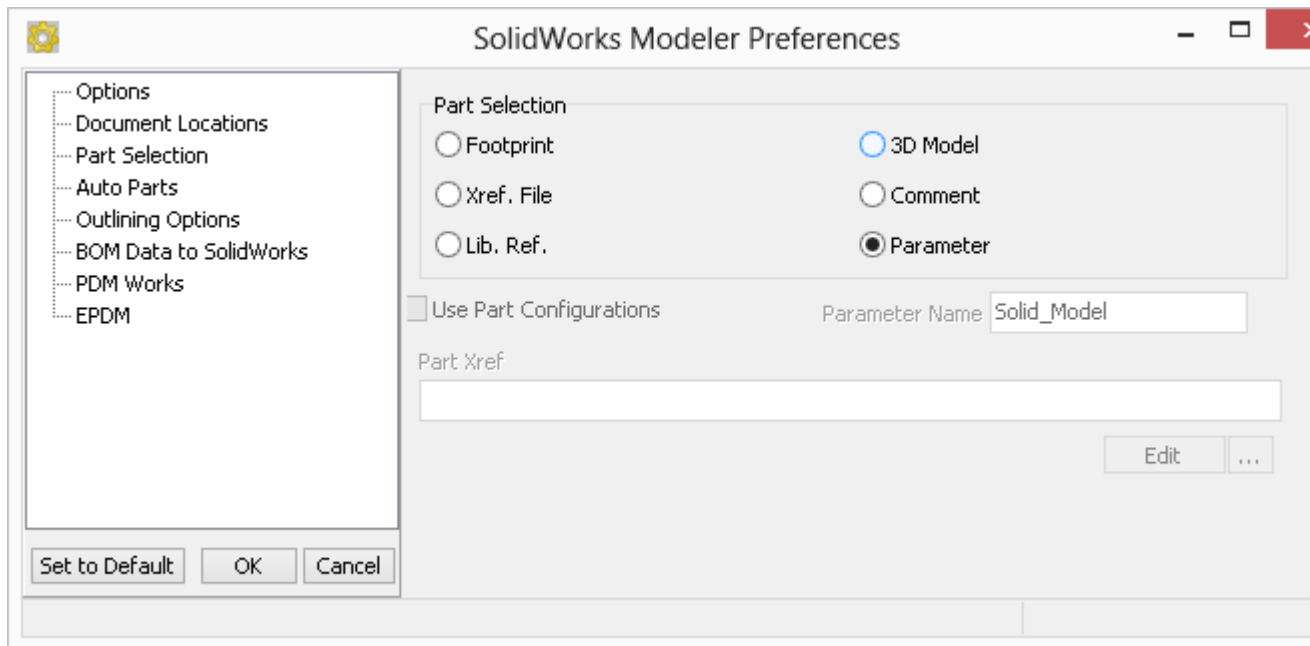
1.10 Managing Parts

The Part Selection radio buttons on the Part Selection page of the Preferences dialog allow you to determine how parts are selected.



The table below describes the Part Selection options.

Setting	Detail
Footprint	The program searches for a part with the same name as the Footprint
Xref. File	The program uses a mapping file to relate the Footprints to Parts. The set the name of the Xref file in the Part Xref field below the radio button group. (see Using Cross Reference (Xref) Files)
Lib. Ref	The part name is read from the Library Ref field of the PCB component
3D Model Field	The part name is read from the 3D Model field of the PCB component (do use this - it will be phased out)
Comment	The Xref function using the Comment field instead of the Footprint field of the PCB component.
Parameter	The program uses a parameter assigned to the corresponding Schematic component to obtain the part name. When this option is selected, the Parameter field is displayed to allow you to set the name of the parameter that holds the part name.



1.10.1 SolidWorks Parts

The program uses the coordinates of the origin of each PCB component as the placement point for the SOLIDWORKS part. The origin of the SOLIDWORKS part must be the same as the origin of the PCB component. Also, the program assumes parts are created in the orientation defined in the Orientation radio buttons on the Options page of the Preferences dialog

Orientation	Detail
Default	When the SOLIDWORKS Front view is selected, you see the Part in plan view
Classic	When the SOLIDWORKS Top view is selected, you see the Part in plan view

Note: If you are a new user, it is recommended that you use the Default Orientation. The Classic option is provided to allow users of earlier versions to use their existing Parts libraries.

If a PCB Footprint name contains any characters that are not allowable in a file name, they are replaced with an underscore in the name of the SOLIDWORKS part that is searched for or created

These characters are: \ / "*"

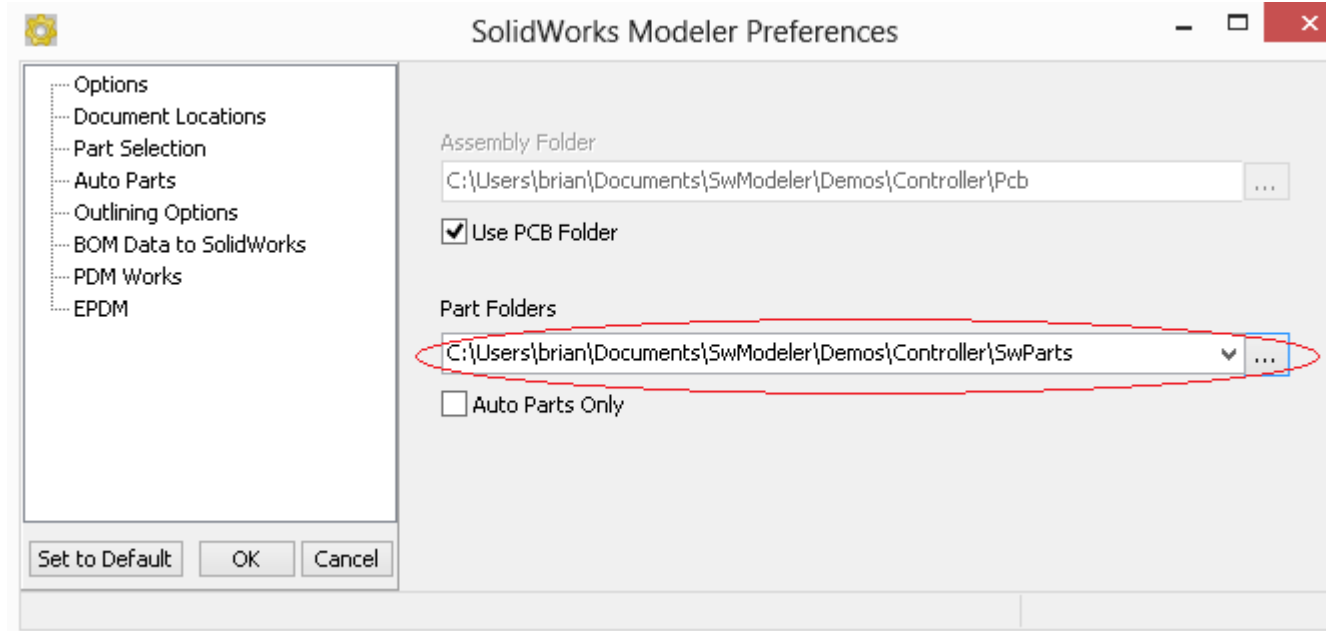
If possible, avoid using characters in Footprint names that are not allowable in a Windows file name.

Hint1: When you are looking down on the SOLIDWORKS Part, it should look the same (ie same origin, same orientation) as the corresponding Altium Component when it is viewed in the Altium PCB Library editor.

Hint2: If you have a SOLIDWORKS Part that does not have the same orientation or origin as the PCB Component, the SOLIDWORKS Part editor command : Insert>Features>Move/Copy is very useful in that it allows you add a default origin and orientation to the Part.

1.10.2 Parts in Folders

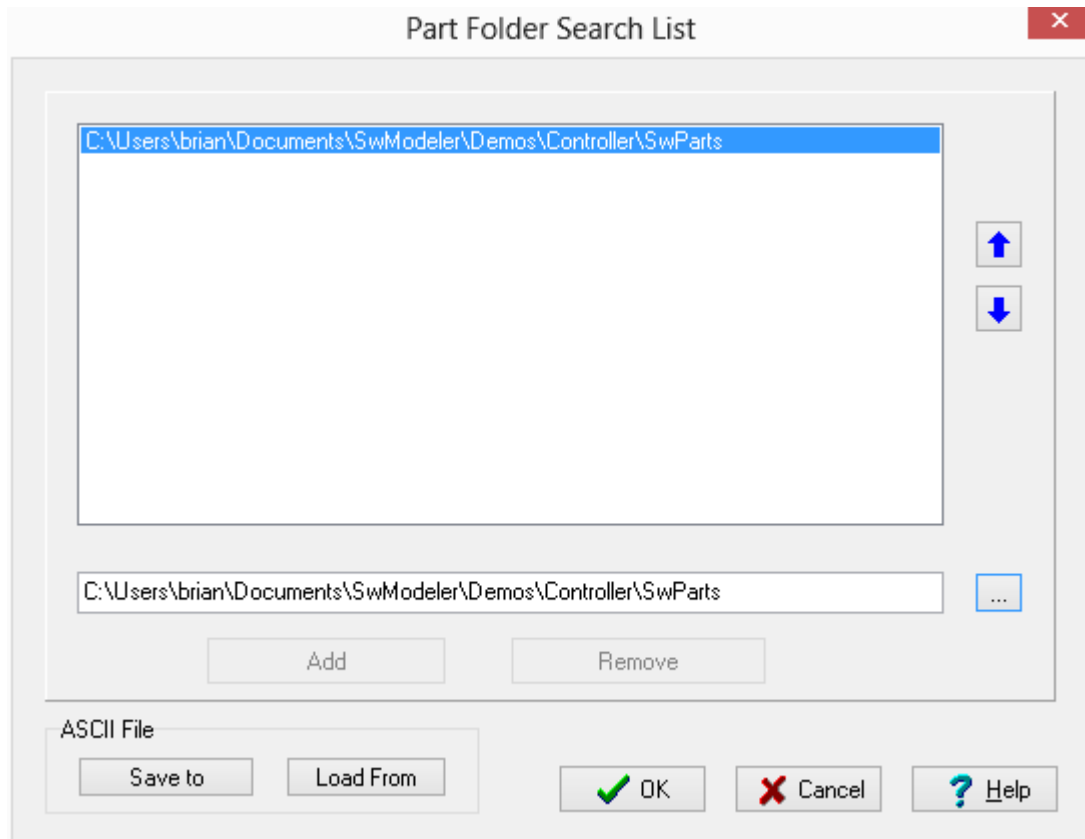
When the Footprint part selection option is active, the program searches the Folders listed in the Part Folders field on the Document Location page of the Preferences dialog.



If you have arranged your parts into sub-folders within one folder, then you only need enter the top level folder. The program searches all sub-folders until it finds a Part.

Otherwise, you should enter each folder.

To enter Folders into the Parts Folders field, click on the Browse button next to the Part Folders field. This displays the Part Folder Search List dialog.



Click on the Browse button towards the bottom right of this dialog, browse to the required Folder, then press the Add button. repeat this for each required folder.

Once you have folders listed, you change change the order in which they are searched by selecting a folder, then press the Up or Down buttons.

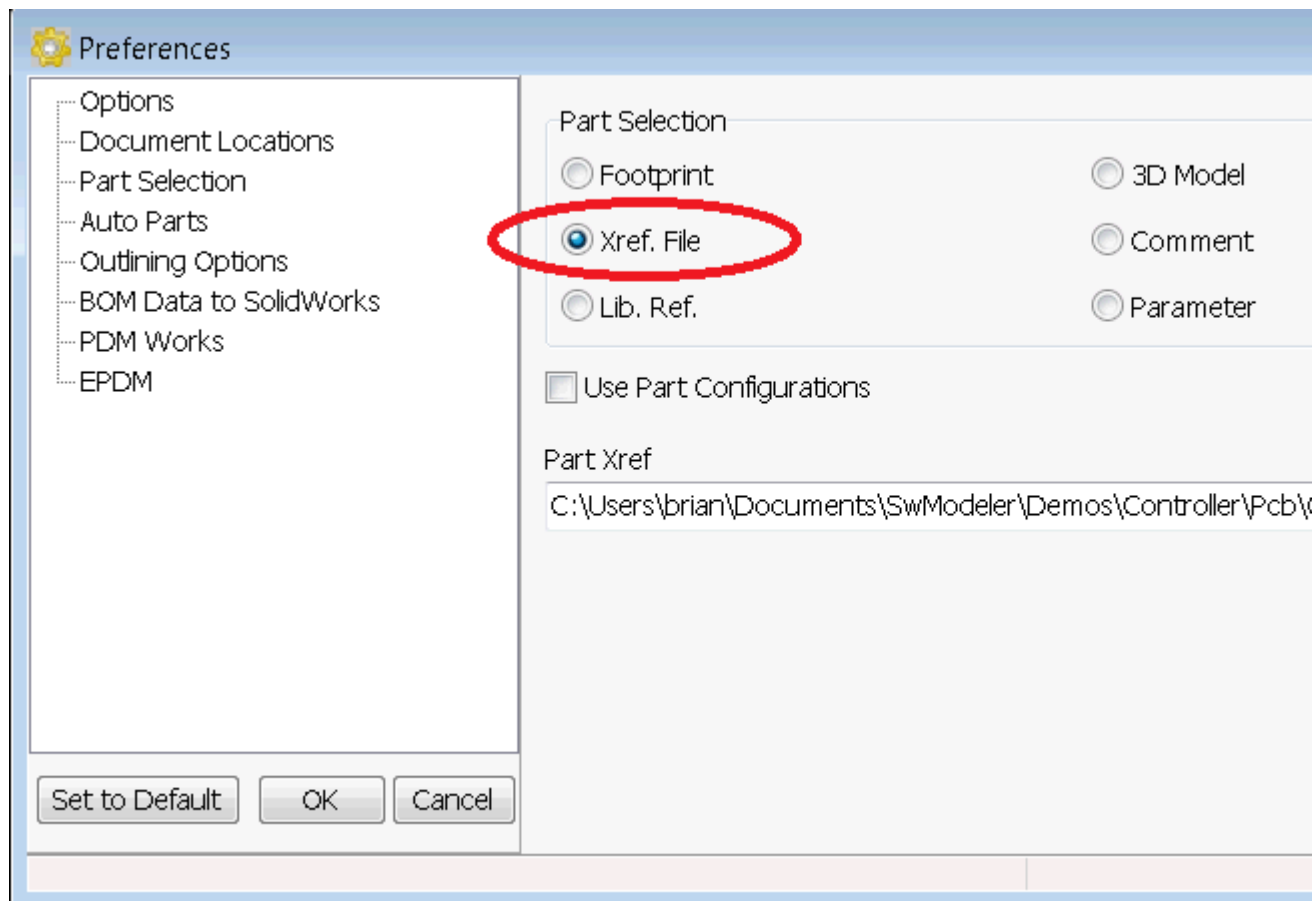
To remove a folder from the list, select it then press the Remove button.

You can save and load this list of folders to/from an ASCII file using the Save To and Load From buttons.

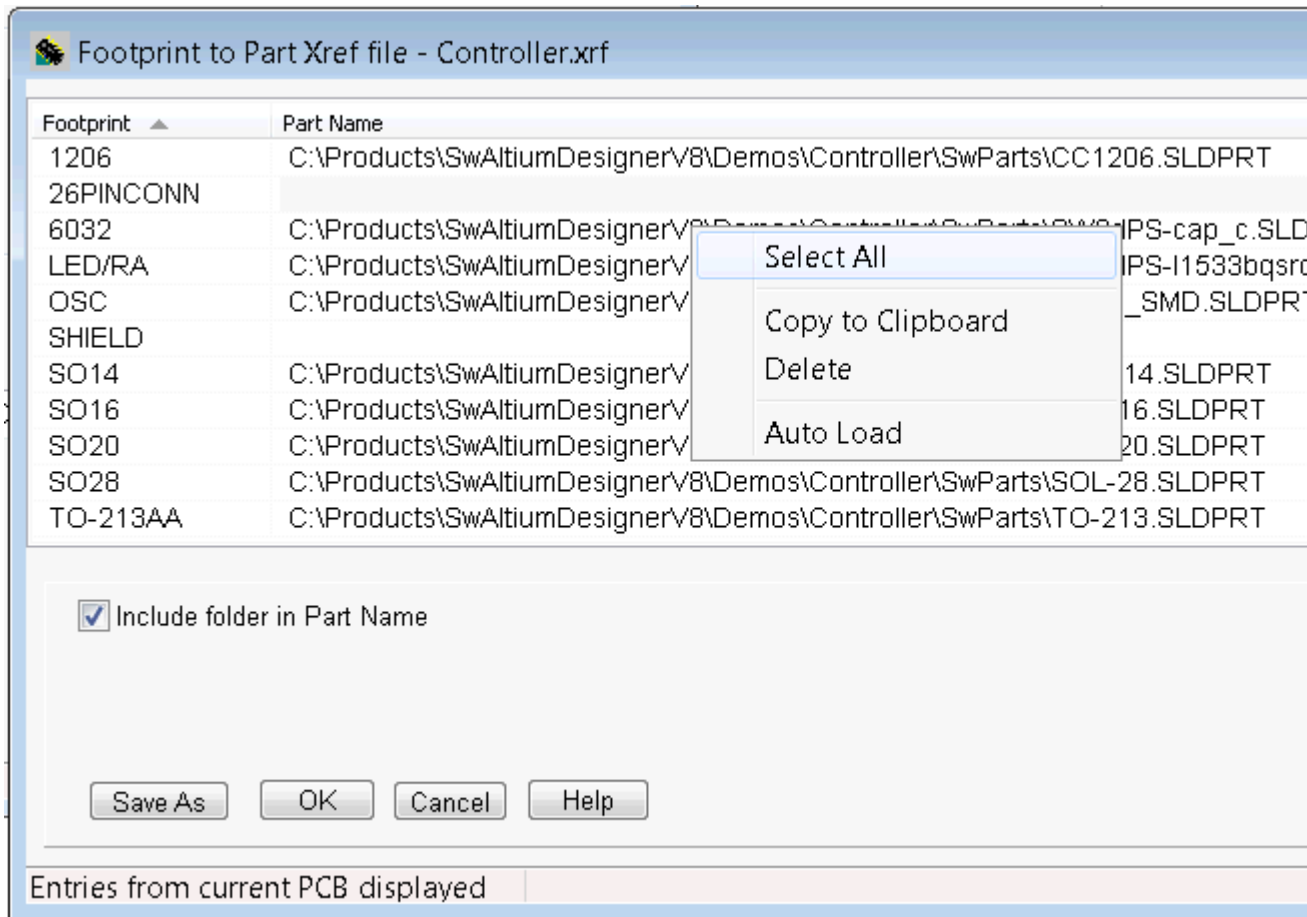
1.10.3 Using Cross Reference (Xref) Files

Cross Referencing enables you to create a "mapping file" to relate a Footprint Name to a SOLIDWORKS Part filename and Configuration. This can simplify the management of parts as it alleviates the need to name SOLIDWORKS Parts with the same name as the Footprint name.

To activate using Xref files, choose the Xref File radio button on the Part Selection page of Preference as shown below.



You set up the Xref file using the Edit button below the Part Xref field. Clicking on the Edit button displays the Xref Editor shown below.



The Footprint column lists the Components from the PCB document. The Part column is provided to list the corresponding Part for the Footprint. To fill in the entry in this column, double click on the required line to display a file browse dialog box. This allows you to browse to the part.

If the Include folder in Part Name check box is checked, the Folder is included in the file name listed in the Part column. When the part name includes the folder, the ordered folder searching is overridden during part placement

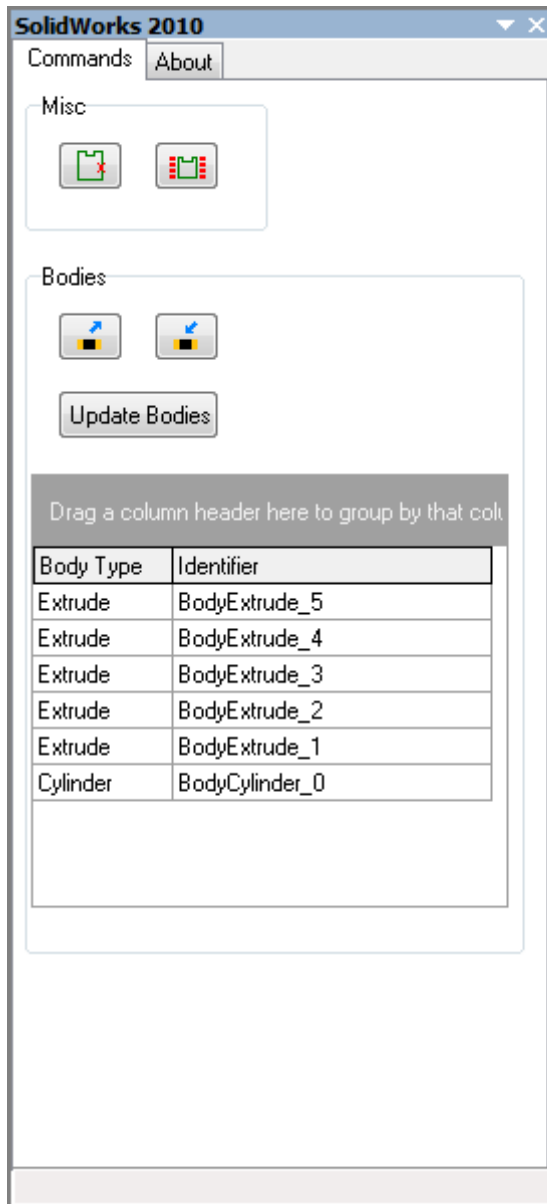
Right click on a row displays the right click menu shown.

If the Use Part Configurations check box in Preferences is checked, a third column is added to the Xref data to allow you to specify a configuration of the Part to use.

If Use Part Configurations is checked and you browse to a Part with configurations, a drop-down list of configurations appears to allow you to choose which configuration to use.

1.10.4 PCB Library Commands

This topic describes the commands available on the PCB Library editor Panel.



Misc Section



Check Outline

This command checks the objects on a layer for gaps between them. This can be used to check that the enclosed shape on that layer will extrude in SOLIDWORKS. When invoked, if the Mechanical Layer defined as the Component Outline Layer is enabled, this is checked. Otherwise, the Silks Screen layer is checked.

**View SOLIDWORKS Part**

This command opens the SOLIDWORKS part for the current Footprint. To find the SOLIDWORKS part, the current Part Selection is used. This is, if Footprint is the Part Selection setting, the Part Folders are searched for a Part with the same name as the Footprint. If Xref is the current setting, this Cross Reference file is used.

Bodies Section**Bodies To Part**

This command creates a SOLIDWORKS part from Bodies in the current Footprint.

**Part To Bodies**

This command creates a new Component and adds Bodies created from the extrusions in the active Part in SOLIDWORKS. You can then select these and copy them to the appropriate component. Base Extrudes, Boss Extrudes and Revolve Extrudes are supported

Update Bodies

This command Updates existing bodies in the current Component from Extrusions in the active SOLIDWORKS part. This command uses names to relate Extrusions to Bodies. That is, the program reads the Feature name of the Extrusion in SOLIDWORKS then searches for a Body whose Identifier field is the same.

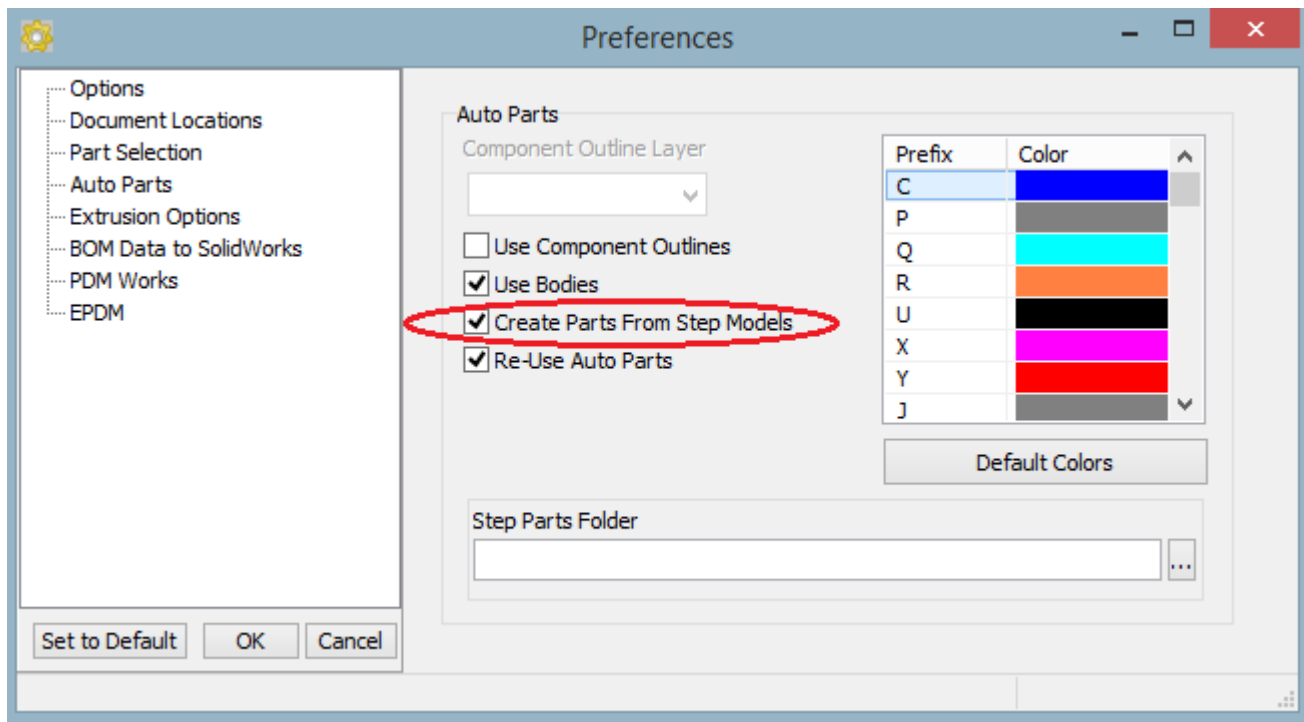
Currently the command supports Body Height and Color.

Body Grid

The grid on the Panel lists the Bodies in the current Component. To populate the grid, double click on the grey area at the top of the Grid. Clicking on a row in the Grid highlights the corresponding body using Object Filtering.

1.10.5 Step Model Conversion

Step Model conversion is invoked by checking the Create Parts From Step Models check box on the Auto Parts page of Preferences.



When this option is checked, if a component has an embedded Step model, this is converted to a SOLIDWORKS part and used in the assembly.

If the Step Parts Folder field is empty, the parts are created in a folder in the PCB folder, otherwise the folder in this field is used.

As the step model conversion is a complex process, it is recommended that once you have converted step models to parts and checked that the conversion is correct, you add these parts to your parts library, rather than converting Step models each time you create an assembly.

When Export to SOLIDWORKS is invoked, if the Create Parts From Step Model check box in Preferences is checked, the following occurs for each Footprint:

If there is no existing SOLIDWORKS part for that footprint and the Footprint contains an embedded Step model, the Step model and its rotations and offsets are extracted from the footprint. The step model is opened in SOLIDWORKS and the Rotations and offsets are applied such that its orientation matches the footprint the it was extracted from.

If a Step model for a Footprint is not converted correctly, you can use the Altium command Export>Step 3D with only that footprint selected.

When the Export Options dialog displays, in the Components With 3D Bodies section, select Export Selected.

Open the resulting step file in SOLIDWORKS and you will have an Assembly containing one part. By using appropriate reference planes you can save this as a Part that will have the correct orientation.

1.10.6 Parts Library

There are a number of SOLIDWORKS parts libraries that are available for download when you purchase this product. These are detailed in the table below.

Library	Detail
Front Orientation IC JEDEC Parts	Contains parts for common ICs based on the JEDEC standard. Each category of IC is modeled in a part with configurations for each IC with different numbers of pins. See the next section for the naming conventions. These are constructed using the Front Plane as the reference plane.
Front Orientation General Parts	Contains various common parts such as resistors, capacitors, connectors etc. Some parts contain multiple configurations, other categories are arranged in directories. These are constructed using the Front Plane as the reference plane.
Top Orientation Parts	Contains various common parts constructed using the Top Plane as the reference plane. These parts are arranged in appropriate directories.

1.10.6.1 IC Naming Convention

The table below describes the naming convention used for the various types of IC packages.

Package Type	Detail
SOJ, LSOJ, SOT, SOP, SSOP, TSOP,	These all follow the same convention. For this typical part name: SOP127P600.SLDPRT SOP identifies the type of package. 127P denotes the center to center distance between adjacent pins in mm (ie 1.27 mm) 600 denotes the tip to tip distance between the two rows of pins in mm (ie 6 mm) This part contains 3 configurations named 08, 14 and 16 for the 8 pin, 14 pin and 16 pin variations for this package.
DIP	For this typical part name: PDIP254P762.SLDPRT: 254P denotes the center to center distance between adjacent pins in mm (ie 2.54 mm) 762 denotes the center to center distance between the two rows of pins when constrained in mm (7.62mm) This part contains 4 configurations named 08, 14, 16, 20 for the 8 pin, 14 pin, 16 pin and 20 pin variations for this package.
BGA	For this typical part name: TBGA127P.SLDPRT: TBGA denotes the JEDEC package designator 127P denotes pitch between row in mm (ie 127mm) The part contains configurations - for this typical configuration name 12X12X160-144:

12X12 denotes the array of solder balls vertically and horizontally
X160 denotes the height of the device measured from the top of the ball to the top of the substrate
-144 denotes the number of pins (balls)

PLCC For this typical part name: PLCC127P-SQ.SLDPRT:
127P denotes the pitch in mm (ie 1.27 mm)
SQ denotes a square package (REC for rectangular)

QFP For this typical part name : PQFP080P.SLDPRT
PQFP denotes the JEDEC package designator
080P denotes the center to center distance between adjacent pins in mm (ie 0.8 mm)

The part contains configurations - for this typical configuration name
1790X1790X230-64:
1790X1790 denotes the tip to tip dimension horizontally and vertically
X230 denotes height
-64 denotes the number of pins

1.10.6.2 IC Listing

The table below lists the parts in the IC Library

IC Family	Part Name	Configurations
BGA	LF-XBGA080P.SLDPRT R-LBGA-B100P.SLDPRT T-XBGA100P.SLDPRT TBGA100P.SLDPRT TBGA127P.SLDPRT TBGA150P.SLDPRT TFR-XBGA065P.SLDPRT TX-XBGA050P.SLDPRT	
DIP	PDIP178P1524.SLDPRT PDIP178P1905.SLDPRT PDIP178P762.SLDPRT PDIP254P1016.SLDPRT PDIP254P1270.SLDPRT PDIP254P1524.SLDPRT PDIP254P2286.SLDPRT PDIP254P762.SLDPRT	
LSOJ	LSOJ065P1120.SLDPRT LSOJ080P1120.SLDPRT LSOJ127P1120.SLDPRT	
PLCC	PLCC127P-REC.SLDPRT PLCC127P-SQ.SLDPRT	

QFP	LQFP063P.SLDPRT
	PQFP065P.SLDPRT
	PQFP080P.SLDPRT
	PQFP100P.SLDPRT
	QFP063P.SLDPRT
	QFP127P.SLDPRT
	T-PQFP065P.SLDPRT
SOJ	SOJ080P1120.SLDPRT
	SOJ080P1300.SLDPRT
	SOJ127P1372.SLDPRT
	SOJ127P850.SLDPRT
	SOJ127P864.SLDPRT
	SOJ127P866.SLDPRT
	SOJ127P935.SLDPRT
SOP	SOJ127P978.SLDPRT
	SOP050P1700.SLDPRT
	SOP080P1425.SLDPRT
	SOP127P1030.SLDPRT
	SOP127P1046.SLDPRT
	SOP127P1200.SLDPRT
	SOP127P1210.SLDPRT
	SOP127P1410.SLDPRT
	SOP127P1605.SLDPRT
	SOP127P600.SLDPRT
	SOP127P700.SLDPRT
SOT	SOP127P780.SLDPRT
	SOT065210.SLDPRT
SSOP	SOT095P280.SLDPRT
	SSOP040P600.SLDPRT
	SSOP050P1190.SLDPRT
	SSOP050P600.SLDPRT
	SSOP063P1036.SLDPRT
	SSOP063P600.SLDPRT
	SSOP065P780.SLDPRT
	SSOP080P1590.SLDPRT
TSOP	SSOP100P1190.SLDPRT
	TSOP040P640.SLDPRT
	TSOP040P810.SLDPRT
	TSOP050P1800.SLDPRT
	TSOP055P1340.SLDPRT
	TSOP065P1430.SLDPRT
	TSOP065P1800.SLDPRT
	TSOP080P1430.SLDPRT
	TSOP080P1800.SLDPRT
	TSOP125P1800.SLDPRT
	TSOP127P1430.SLDPRT
	TSOP127P922.SLDPRT

1.10.6.3 Part Origin Convention

The section details the conventions used for setting the origin of parts.

The table below lists how origins are set.

Type of Part	Location of Origin
Through Hole	Pin 1
Surface Mount	Centroid

1.11 Product Tools

The various tools for checking and set up are described in the following topics.

1.11.1 Check Component Outlines

This command checks the component outline layer of each component for an enclosed shape of tracks and arcs. If one is not found, the component overlay layer is checked. If no enclosed shape is located, the component is selected.

Use this command to check that the program can create Auto-parts from the Component Outline layer or from elements on the the Silk-screen layer .

1.11.2 Check Outline

For SOLIDWORKS to successfully extrude the board outline, all shapes defining the board outline and cut-outs must be perfectly enclosed. (i.e. the end points of adjacent objects making up the shapes must be co-incident).

If you are using the Board Shape:

The Check Outline command on the toolbar will check the elements in the board shape for gap errors. If no errors are found it will report this in the Messages Panel. If errors are found, it will attempt to correct them and the result is displayed in the Messages Panel.

If this command does not correct errors in the Board Shape, re-define the board outline on a mechanical layer, use Check Outline to check it, then re-create the Board Shape from the elements on the Mechanical Layer. (The Desktop EDA PCB Utilities provides commands to assist in this task).

If you are using a Mechanical Layer to define the Board Outline:

The Check Outline command checks the board outline layer for gaps between the elements. If any gaps are found, an attempt is made to fix them. The types of breaks that can be fixed are described below. Check Outline also deletes duplicate objects and zero length objects.

After Check Outline has run, any two objects that are not perfectly joined are selected. You can use Jump to Selection command (JT) in the Jump sub menu in the Edit menu. Also, an entry is added to the Messages Panel. Double-click on this entry to zoom to the object.

Use this command before running the Export To SOLIDWORKS command to ensure that SOLIDWORKS can extrude the Board Outline.

The following outline errors are corrected during Check Outline:

1. Tracks that have a gap of 0.5 mil or less between other Tracks or Arcs are modified so that their end point joins with the other element.
2. Arcs with a 0.5 mil or less gap to other arcs have their radius and start and end angles modified such that they join at their intersection.
3. Zero length tracks and arcs are deleted.

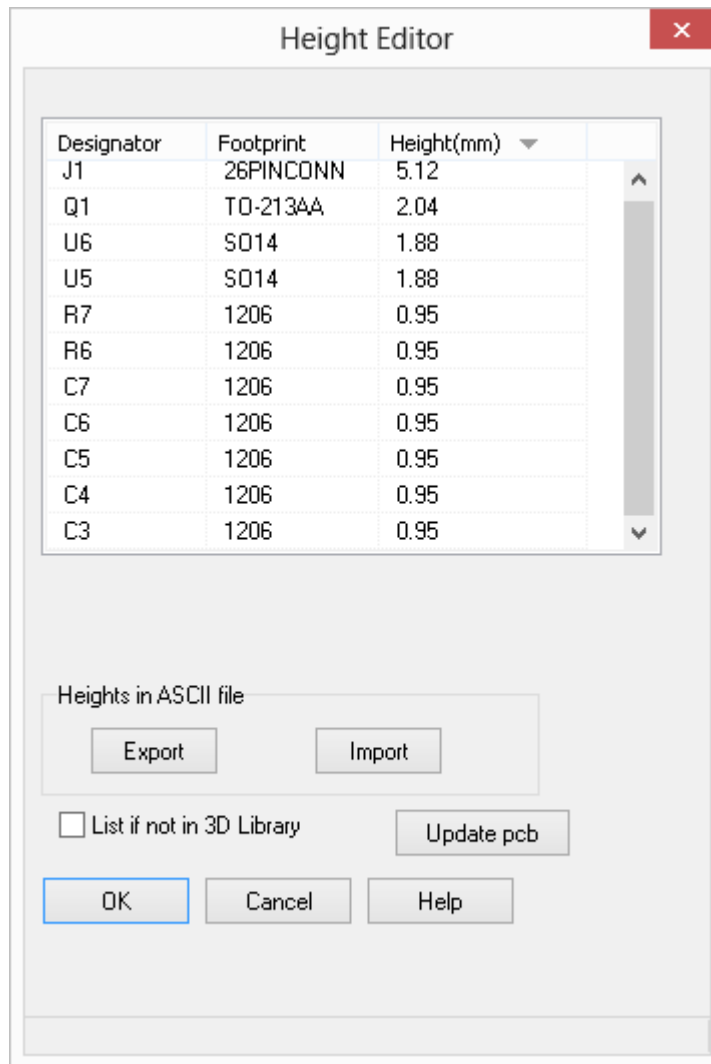
1.11.3 Check Outline (PCB Library Editor)

This command is located on the Library editor panel. It checks the Component Outline Layer of the active component for an enclosed shape for gaps. Gaps will prevent extrusion when creating Auto Parts.

Any elements that have gaps between them are selected.

1.11.4 Height Editor

This command displays the Height Editor dialog box that lists the components in the current PCB with their heights. The Height value is obtained from the Height property of each component.



This list can be sorted by Designator or Footprint. There is also an option to list only components that do not have corresponding SOLIDWORKS parts.

To edit the height of components, click on the height text and edit it. The column widths can be changed.

Component Heights can be exported to an ASCII file using the Export button and imported using the Import button.

The format of this ASCII file is as follows:

```
units=mm
AXIAL0.4=3.81
DB37RA/F=11.43
DIODE0.4=3.81
DIP14=5.08
DIPSW16=6.35
```

If the first line is omitted or the unit string is not mm, the units are assumed to be mils.

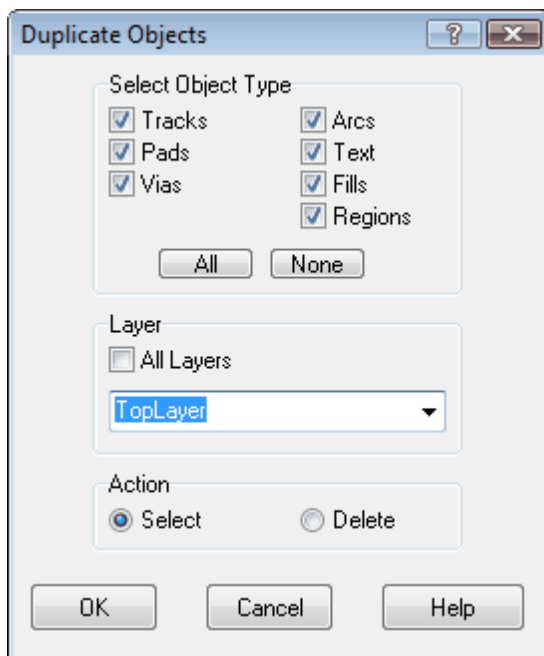
1.11.5 Set Mounting Hole

This command sets the Designator of any selected multi layer pads to the Mounting Hole Prefix defined in the Options tab of the Panel plus a unique number. The designators of Mounting Holes already assigned are preserved.

1.11.6 Duplicate Objects

This command allows you to either Select or Delete duplicate elements in the current PCB document.

A duplicate element is flagged if there are two or more elements that are on the same layer and all co-ordinate properties are the same.



When this command is selected, a dialog box displays. From here you can select which object types you would like checked. You can also choose the Layer to search. You can either select a single layer or all layers.

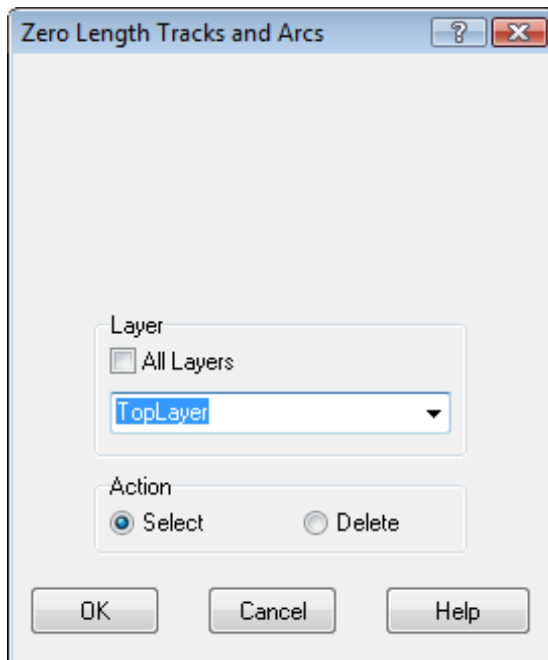
You can choose either to select or delete duplicate objects by clicking on the appropriate radio button.

When the options have been selected, choose OK.

1.11.7 Zero Length Tracks and Arcs

This command allows you to either Select or Delete zero length tracks and arcs in the current PCB document.

A zero length element is flagged if the distance between the start point and the end point is less than 0.1 of a mil.



When this command is selected, a dialog box displays. From here you can choose the Layer to search. You can either select a single layer or all layers.

You can choose either to Select or Delete zero length objects by clicking on the appropriate radio button.

When the options have been chosen, click OK.

1.11.8 Show Part Usage

This command shows the Part that will be used for each Footprint. This displays in the Message Panel.

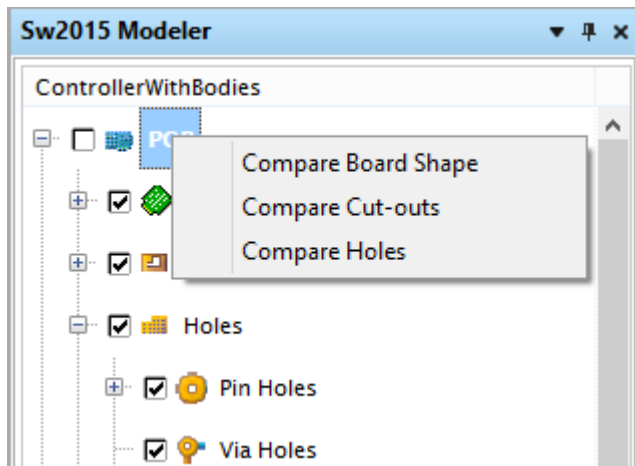
This command is invoked by right clicking on the Components node in the Panel and choosing List Parts

1.11.9 Compare

The topics below describe the compare functions.

1.11.9.1 Compare PCB

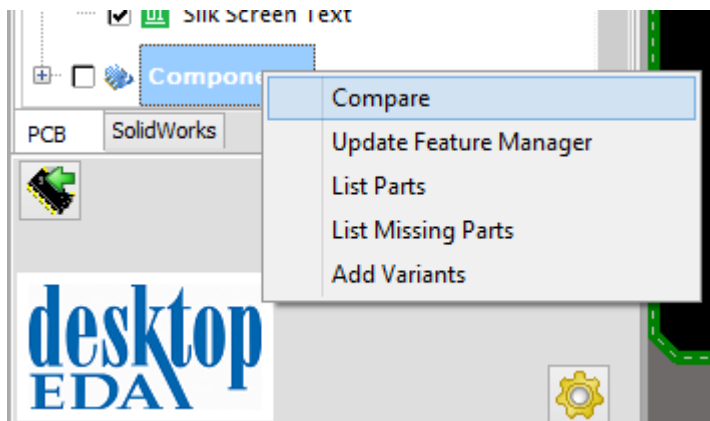
These commands allow you to compare the Board Outline, Cut-Outs and Mounting Hole between the PCB and the Board Outline part.



Right click on the PCB node in the Panel. The result of the compare is displayed in the Messages Panel.

1.11.9.2 Compare Component Locations

The command allows you to compare the locations of components in the PCB and the SOLIDWORKS assembly.



Right click on the Components node in the Panel and choose Compare. The result of the compare is displayed in the Messages Panel.

1.12 Rigid/Flex Support

Rigid/Flex circuit boards can be exported to SOLIDWORKS.

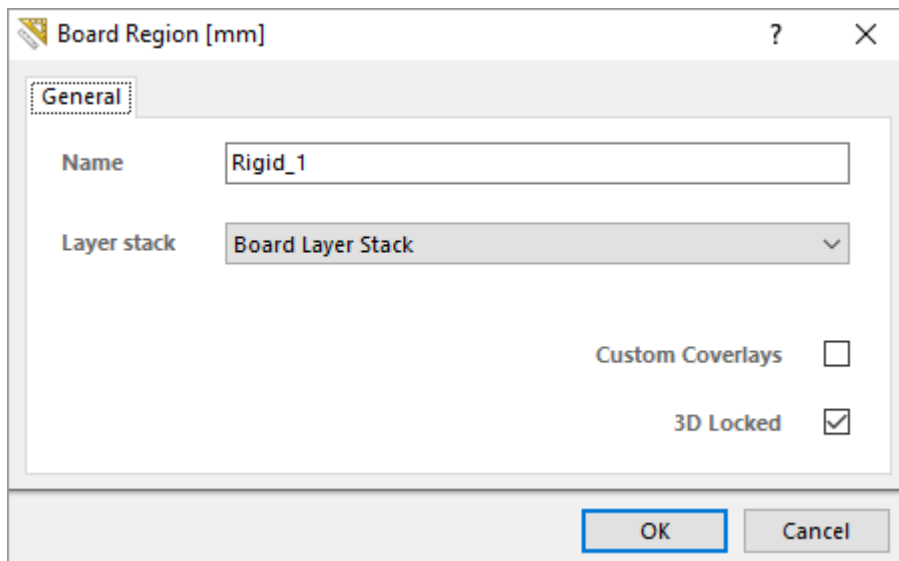
If the Rigid/Flex Assembly check box on the Options page of Preferences is checked and the PCB is a Rigid/Flex PCB, a Rigid/Flex assembly is created in SOLIDWORKS, otherwise, the PCB is handled as a normal PCB.

The following topics describe how Rigid/Flex PCB's are handled.

1.12.1 Setting up Rigid/Flex Board Regions

Each region defining a PCB shape in the Rigid/Flex assembly must have a unique name. Also, the Rigid PCB's should contain the word "Rigid" and flex PCB's should contain the word 'Flex'. The Region name is used to identify the Layer Stack that is used to manage layers and calculate thickness.

You can assign names to each region by invoking the command View>Board Planning Mode (or enter 1 on the keyboard). Edit the name of the Region by double clicking on the Rigid or Flex PCB and editing the Name field.



1.12.2 Creating the Rigid Flex Assembly

If the PCB is a Rigid/Flex PCB, the following occurs:

1. Each Rigid PCB or Flex PCB is created as a separate Assembly in the Board Outline assembly
2. Components for each Rigid PCB or Flex PCB are added to Sub-assemblies which then placed in a master assembly.

PCB Features and multi-layer are supported for Rigid/Flex PCB's.

In the top level assembly, a configuration is created for each PCB so that when the configuration is activated, you can view that PCB in isolation. Also, for each individual PCB assembly, a configuration is created for each layer pair to allow you to view the layer pair in isolation.

1.12.3 Rigid/Flex Layer Stacks

Rigid/Flex PCB's typically use the Custom stack up option in the Altium Layer Stack Manager.

Currently, the program supports one Flex stack in a design.

When the program builds layer stacks for each PCB in the Rigid/Flex PCB, it starts with the Flex

stack. When creating the Rigid stacks, it adds layer pairs to the top and bottom of the common flex layers as appropriate.

When creating a multi-layer PCB in SOLIDWORKS, the program needs to know which dielectric a conductor layer is associated with, so that it can select the correct face of the base extrudes to create the boss extrudes

With the Custom stack up, there is no means of assigning which dielectric a conductor layer is associated with, so the program deduces this.

Below are some guidelines for assigning the Material for a Layer:

- Prepreg at the top of the stack have conductors assigned to the upper surface and at the bottom of the stack conductors can be assigned to the bottom of the stack
- Cores can have conductors on both the upper and lower surfaces
- When the Material is assigned as None, the program will attempt to work out if conductors above or below are assigned and any that are not will be assign to this dielectric
- In Flex stacks, if a Prepreg has unassigned conductors above, that will be assigned to the Prepreg.

Taking into account the above guidelines, assigning layers as either a core or a prepreg to achieve the required stack up will ensure the conductors get assigned to the required dielectrics in the 3D assembly.

1.12.4 Updating the Rigid Flex Board Outline from SolidWorks

The Rigid and Flex PCB's in a Rigid/Flex PCB design are defined in Board Planning Mode by starting with a Board Shape and then adding Split Lines to define the Flex PCB's. When the Rigid and Flex PCB's are exported to SOLIDWORKS, separate Parts with board Outline features are created.

In Rigid/Flex mode, board outlines cannot be updated from SOLIDWORKS as they can for normal PCB's.

If the geometry of a Rigid/Flex PCB design is very complex, you will may choose to design the geometry in SOLIDWORKS as SOLIDWORKS has a rich set of sketch creation commands for this purpose.

Below is the recommended work flow to design a Rigid/Flex PCB geometry in SOLIDWORKS:

1. Create the geometry on a mechanical layer - do not be too concerned about precision
2. Un-check the Rigid Flex Assembly check box on the Options page of Preferences (so the program treats the PCB as a normal PCB)
3. Set the Board Outline Source to the Mechanical Layer. Export the Board Outline to SOLIDWORKS
4. Edit the sketch defining the board outline just created - use the SOLIDWORKS sketch editing features to create the precise board Outline required. Exit the sketch and check that the extrusion is the required geometry.
5. Alternatively, simply create a Part and use the PCB tools to define the Board Outline geometry
6. Update the Altium PCB from the updated Board Outline part
7. Update the Board Shape from the Updated Mechanical Layer

8. In Board Planning mode, Place the Split Lines and Bend lines to define the Flex PCB's
9. Rename the Board Regions such that Rigid regions contain the word "Rigid" and flex regions contain the word "Flex" (so the program knows what they are - Altium does not provide this information.) Also ensure that each Region name is unique as these are used to create the Feature names in SOLIDWORKS
10. Check the Rigid Flex Assembly check box on the Options page of Preferences and change the Board Outline source to be the Board Shape
11. Send the Board Outline to SOLIDWORKS - confirm that each PCB is created correctly

1.13 Starting Designs in SolidWorks

The topic introduces the PCB Tools. These are provided to allow you to assign features in SOLIDWORKS such that they appear as if they have been created by the Export to SOLIDWORKS command. These are typically used to start a PCB design in SOLIDWORKS.

Once the PCB tools have been applied, you can update your PCB from SOLIDWORKS.

The following topics describe the PCB Tools.

1.13.1 Guidelines For Starting Designs In SolidWorks

The PCB must first be created as a SOLIDWORKS Part. The profile of the PCB should be drawn in a sketch and extruded by the thickness of the PCB. The Base-Extrude created by the extrusion must have appropriate attributes assigned to it. This can be achieved either by using the Assign PCB Features in the PCB Tools or creating an assembly using the Export to SOLIDWORKS command. Mounting Holes can be defined using the Set Hole Features command in the PCB Tools.

This part should then be placed into the assembly aligned with the Front plane, at the co-ordinate of 0,0,0. This part should have appropriate attributes assigned to it using the PCB Tools.

Each electronic component on the PCB must have a SOLIDWORKS Part. The profile of the component must be drawn in a Sketch and extruded by the height of the component. These parts can then be placed in the assembly. These parts can then be assigned as PCB parts using the Assign Part Names command in the PCB Tools.
E.g. C1, C2, R1, U1 etc.

To see an example of how the assembly should be set up, export one of the example PCB's installed with the product.

1.13.2 PCB Tools

If you are starting the PCB design in SOLIDWORKS, use the PCB Tools commands.

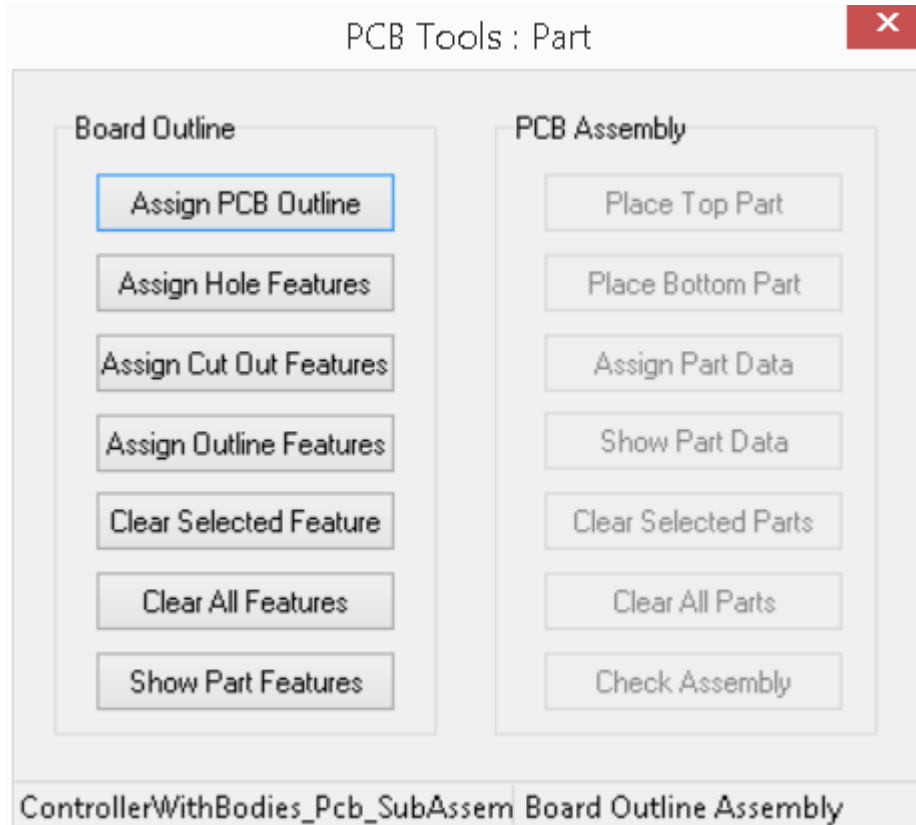
The PCB Tools functions are provided to allow you to assign attributes to features in SOLIDWORKS so that the Update from SOLIDWORKS commands can recognize the features.

You can access the PCB Tools commands from the PCB Tools button on the product toolbar.



Before invoking this command, the Assembly or Part you plan to work in should be open. When invoked the PCB Tools command display.

If a SOLIDWORKS Board Outline is active, the PCB Tools Board Outline commands are enabled. If a PCB Assembly is active, the PCB Assembly commands are enabled.



When you create an assembly in SOLIDWORKS, any feature that is to be imported into Altium must have an attribute assigned to it so that the Update commands can recognize the feature.

The aim of the PCB Tools commands is to make the assembly you create in SOLIDWORKS appear as if it was created by the Altium Modeler.

Use the Board Outline commands to assign attributes to the features in the Board Outline part. You then use the Update from SOLIDWORKS commands to update the PCB document with the geometry from the features assigned in SOLIDWORKS Board Outline part.

If you need to place parts in SOLIDWORKS (ie to accurately locate using other geometry in SOLIDWORKS), you can create the invoke the Export to SOLIDWORKS command with the Components check box in the Panel checked.

Once the PCB assembly is created, you can add parts in SOLIDWORKS using Place Top Part or Place Bottom Part, then assign Designator and Comment using the Assign Part Data command.

(alternatively, if you have the components already placed in the PCB document, you could simply create the assembly using the components, then move them to their required locations in SOLIDWORKS).

Index

- C -

Check 3D Outline (Library Editor) 65
Check Component Outlines 64
Check Outline 64

- D -

Defining the Board Outline in SolidWorks 72

- E -

Export To SolidWorks 36

- G -

Guidelines on Exporting Tracks and Overlays 44
Guidlines For Starting Designs In SolidWorks 72

- M -

Managing Library Search Paths 52

- P -

PDM Support 41

- R -

Review Component Heights 65

- S -

Set Mounting Hole 67
Setting Colors 43
Setting Up For Export 35
Show Part Usage 68
SolidWorks 3D Modeler Commands 15
SolidWorks Parts 53

- U -

Using Cross Reference Files 55

- W -

Welcome 2

