



Altium Modeler for SolidWorks Getting Started

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Printed: August 2016 in Australia

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1 Introduction

This product is a powerful tool enabling you to create a SolidWorks Assembly from your PCB document. It uses SOLIDWORKS installed on your PC to create a SOLIDWORKS assembly.

This document starts with a Getting Started Quickly topic that provides a brief description of how to use the most common features. The following topics guide you through the process of creating a SOLIDWORKS Assembly from a PCB and introduces some of the bi-directional features.

Use this guide in conjunction with example files provided in the folder Documents\SwModeler\Demos

We suggest that you get to know how the product works using this guide. Once you are comfortable with the concepts, you can move on to your own PCB's.

More detailed information on the features of this product is provided in the Help. You can access Help via the Help menu in the PCB Editor or via the Help button in the product toolbar.

In the process of working through the topics, when you reach the end of a window, click on the Next link at the top of the window to move on to the next topic. If you want to go back, click on Previous link at the top of the window.

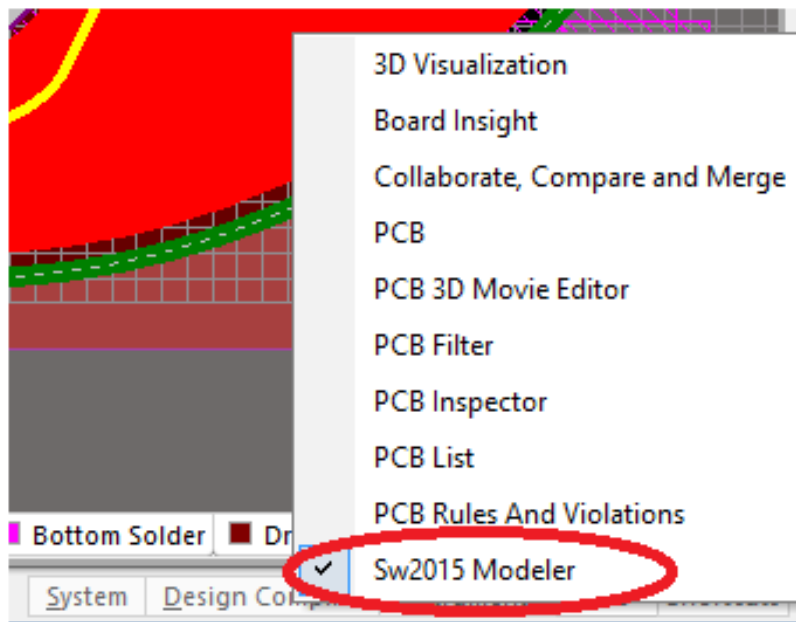
2 Getting Started Quickly

The following topics provide instructions for getting started quickly

2.1 Creating an Assembly

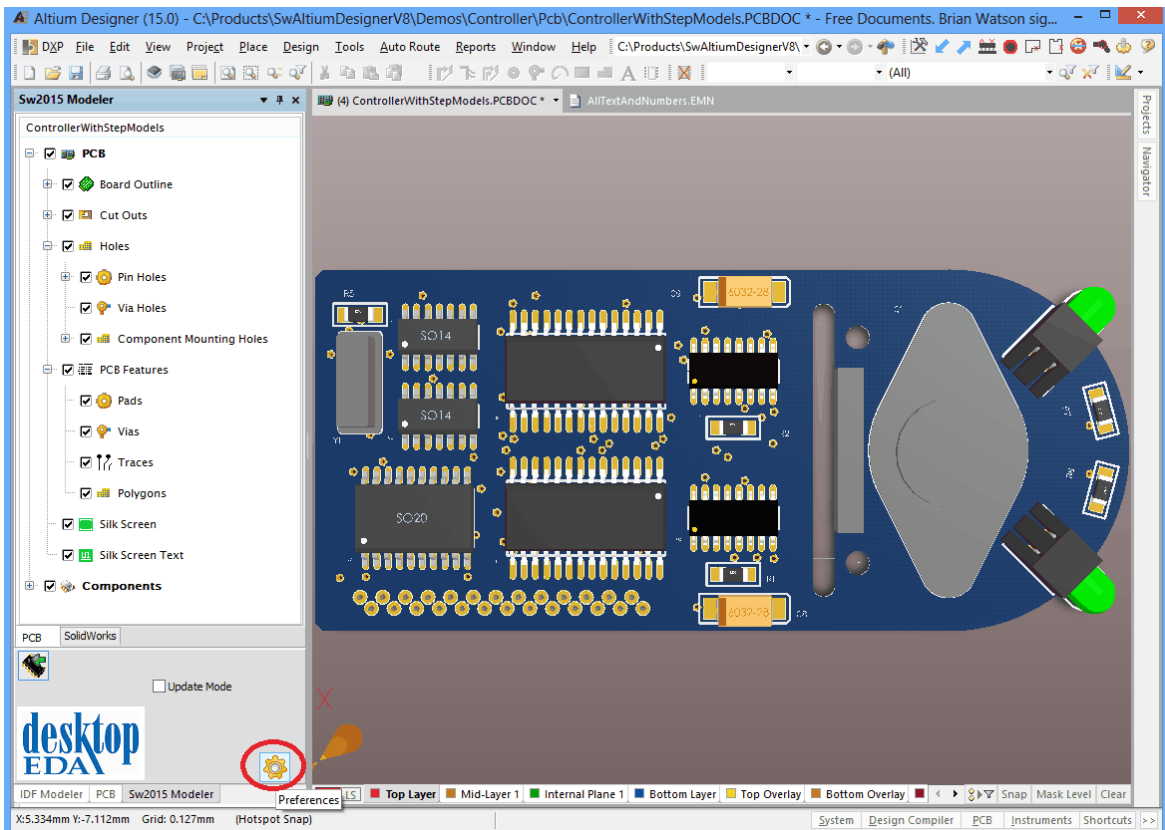
All functions for this product can be accessed from the Workspace Panel.

To display the panel, click on the PCB button and click on the menu item shown below:

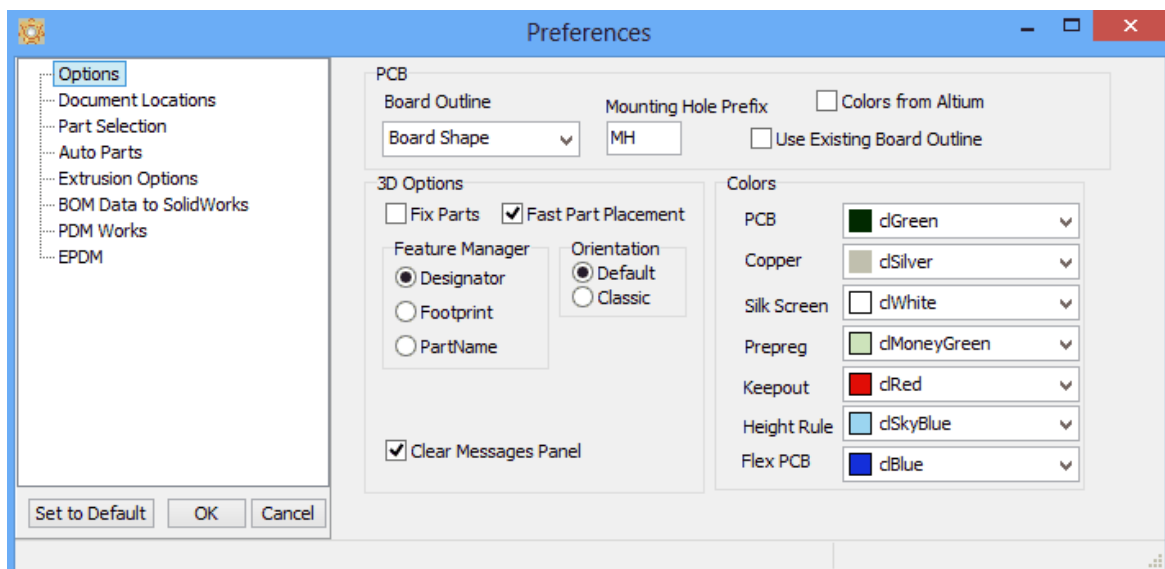


Open C:\Products\SwAltiumDesignerV8\Demos\Controller\Pcb\ControllerWithStepModels.PCBDOC

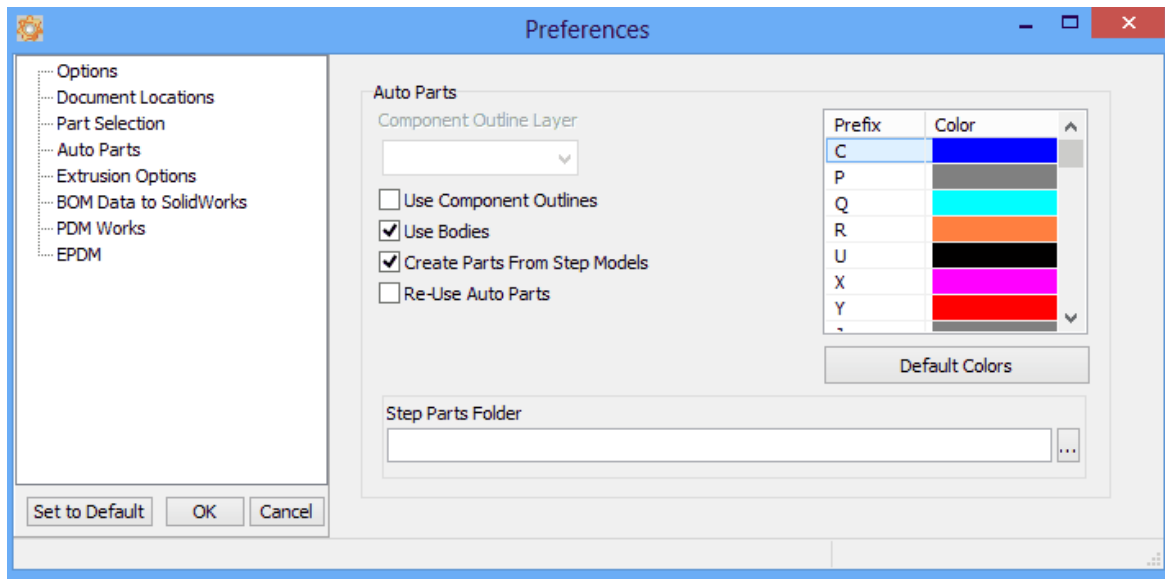
This displays the Workspace panel. The panel includes a tree view of the currently active PCB. Each node on the tree view corresponds to elements in the PCB. The check boxes on the tree view nodes allow you select what elements are included in the SolidWorks assembly.



Invoke Preferences dialog from the button shown on the Panel. The following Preferences dialog displays:



Invoke Auto Parts of the Preferences to set up following options:

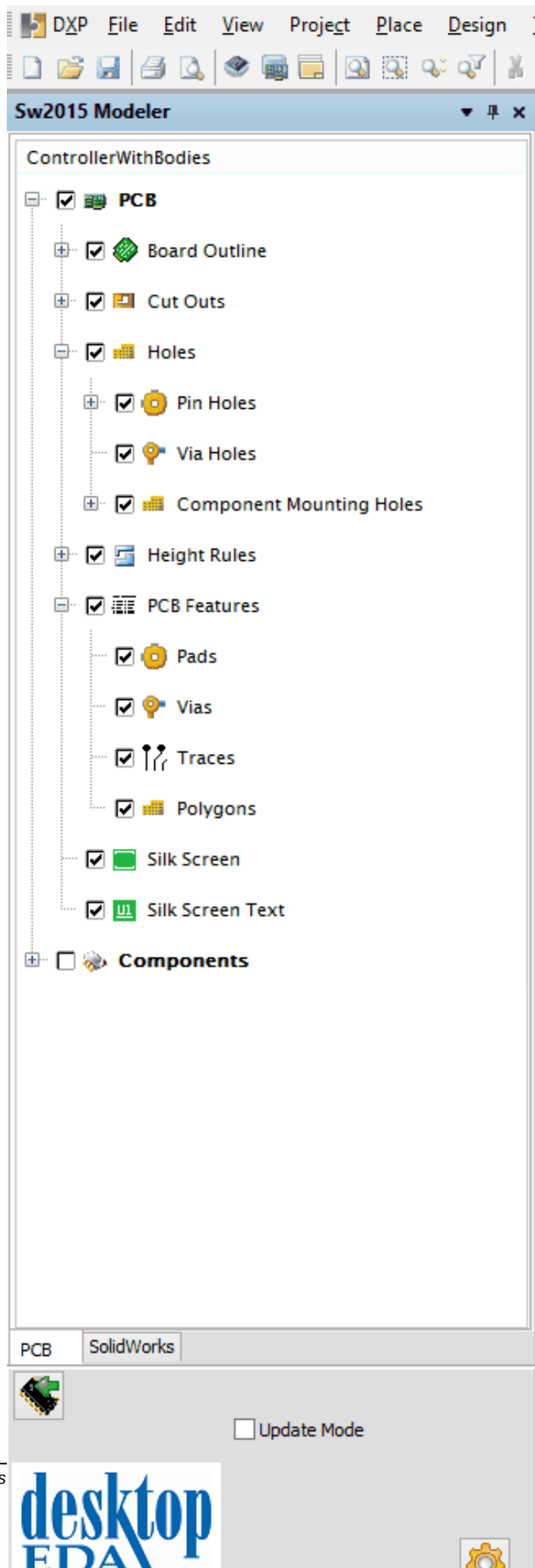


Check Create Parts from Step Models check box to create parts from embedded step models.

To create an assembly, click on the button with the green arrow. The program then connects to SOLIDWORKS and creates an assembly from the PCB.

2.2 Creating the Board Outline

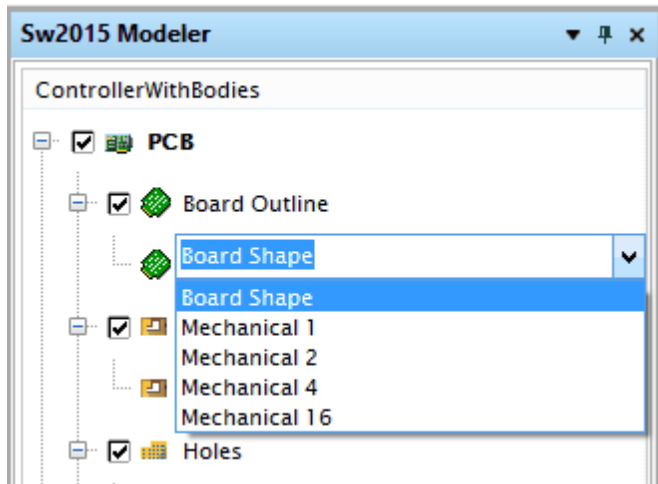
To Create the Board Outline without creating an assembly including all components, you can un-check the Components check box.



2.3 Panel Functions

By default, the board Shape is used to create the board outline in SolidWorks. You have the option of using a mechanical layer.

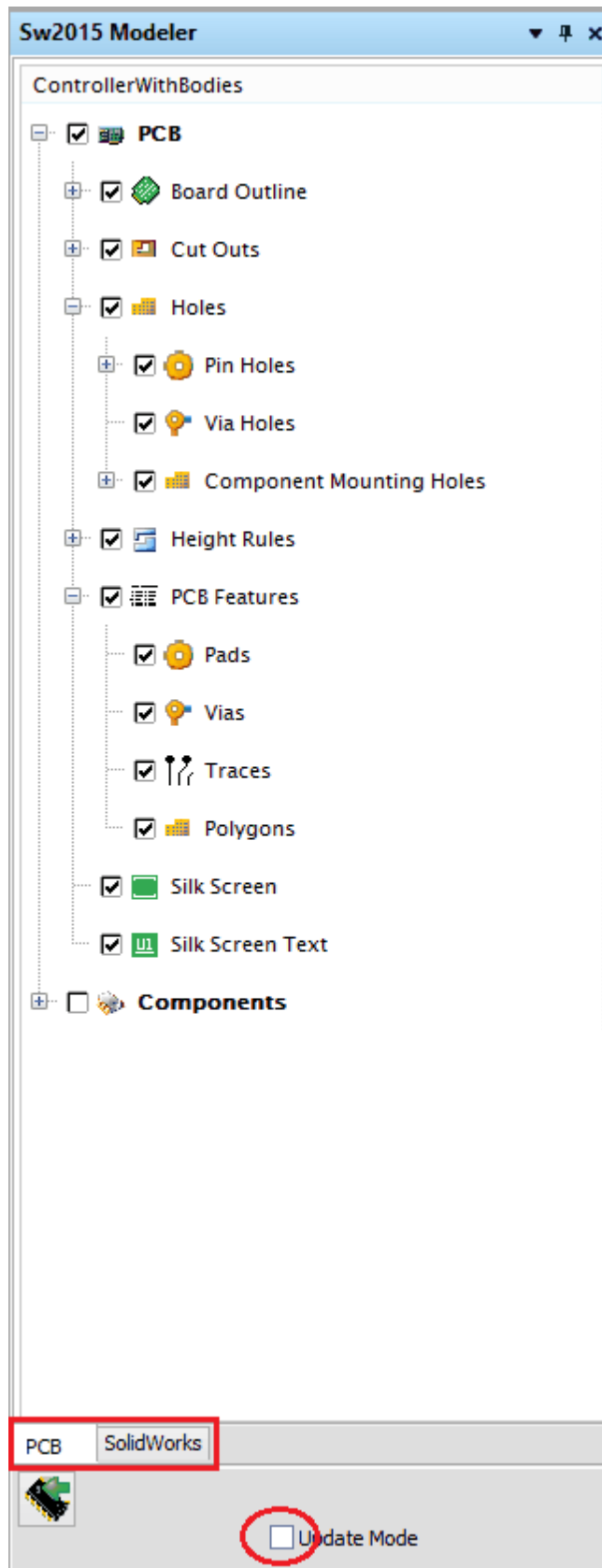
To define what is used, expand the Board Outline node and double click on the child node to display the list of options.



The tree view in the Panel has two tabs, to display the PCB view and the SolidWorks view.

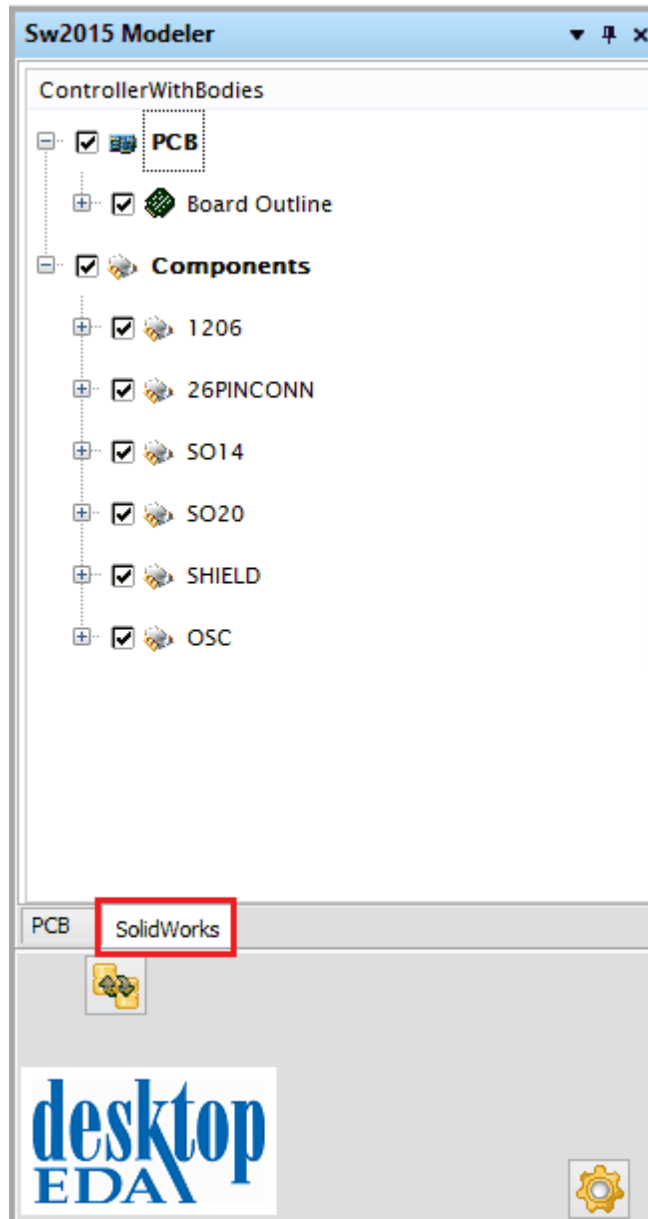
The PCB view allows you to see what elements exist in the current PCB. Double click on any node in the tree and the elements represented by that node are highlighted in the PCB window.

When the Update Mode check box is checked, the Update from PCB button is displayed and SolidWorks is updated from PCB elements whose check boxes are checked in the tree view.



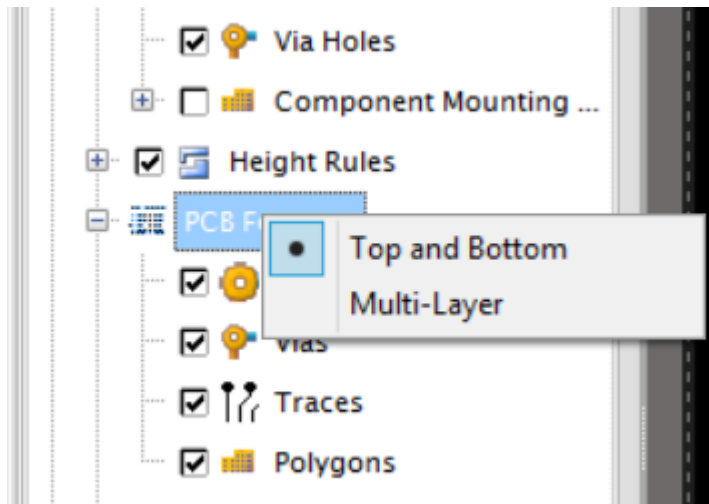
The SolidWorks view displays nodes for each feature in SolidWorks. When the SolidWorks view is active, the Update from SolidWorks button displays.

When this button is pressed, the PCB is updated from SolidWorks features whose check boxes are checked in the tree view.

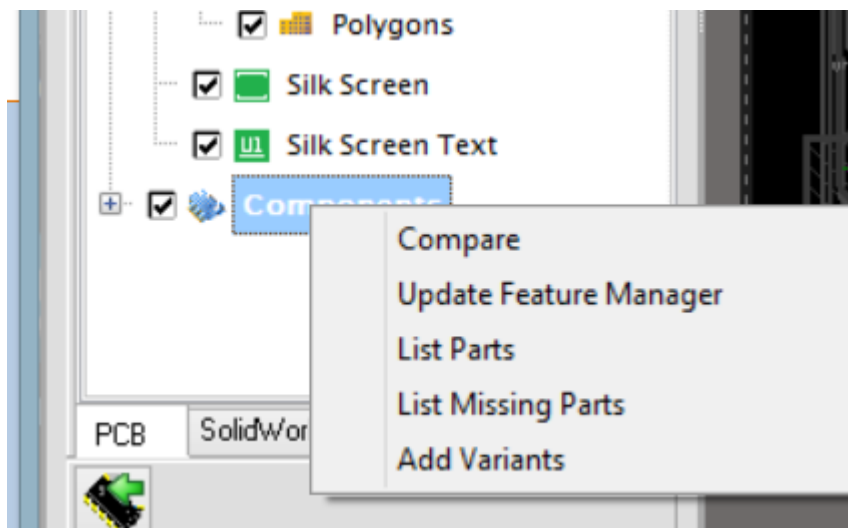


Right Click on the PCB Features node allows you to toggle between Top and Bottom mode and Multi-Layer mode.

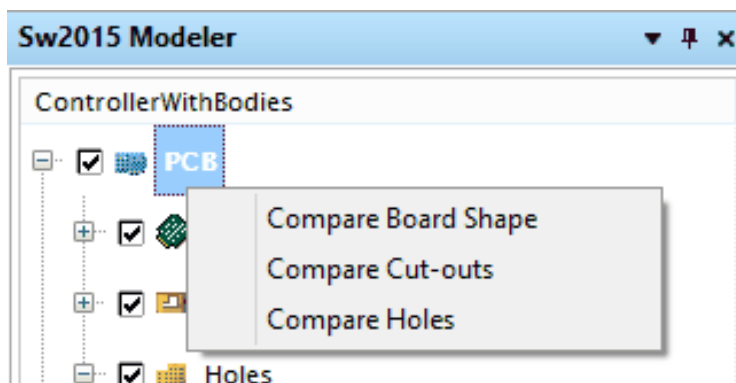
In Top and Bottom mode, copper modeling is performed for the top and bottom layers only. In Multi-layer mode, copper modeling is performed on all layers.



Right click on the Components node provides access to functions related to part placements. See the Help document for details.



Right click on the PCB node provides access to the compare functions.



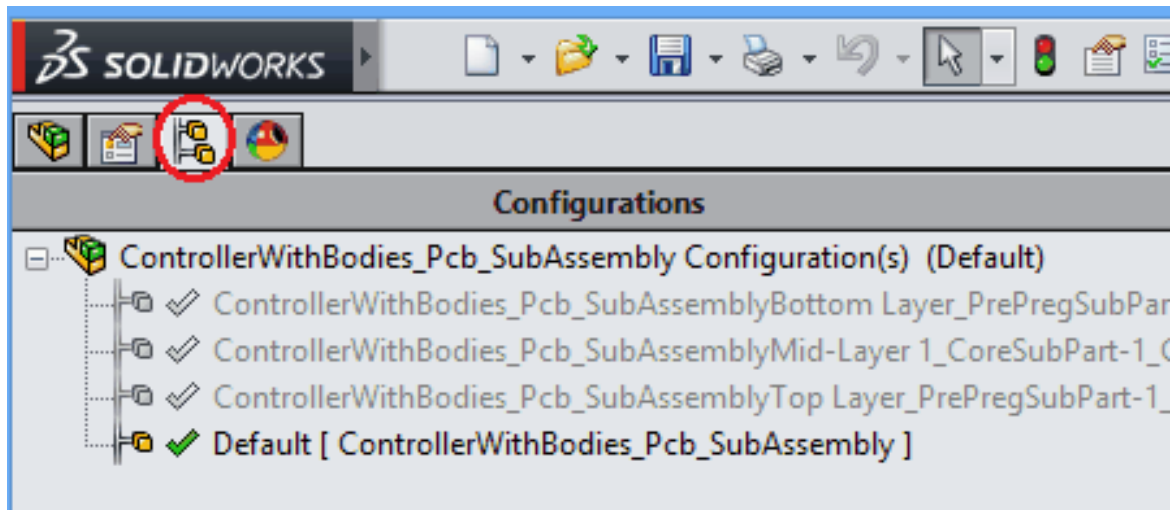
2.4 SolidWorks Configurations for Multi-Layer

When a multi-layer board outline part is created, a configuration for each layer pair of the PCB is added to the Board Outline assembly.

Each configuration allows you to view the layer in 3D from any view without the other layers.

To view a configuration, click in the Configurations tab shown below, then double click on any on the configurations listed.

The Default configuration displays all layers.



2.5 Modeling Copper

The following topics cover the modeling of copper features.

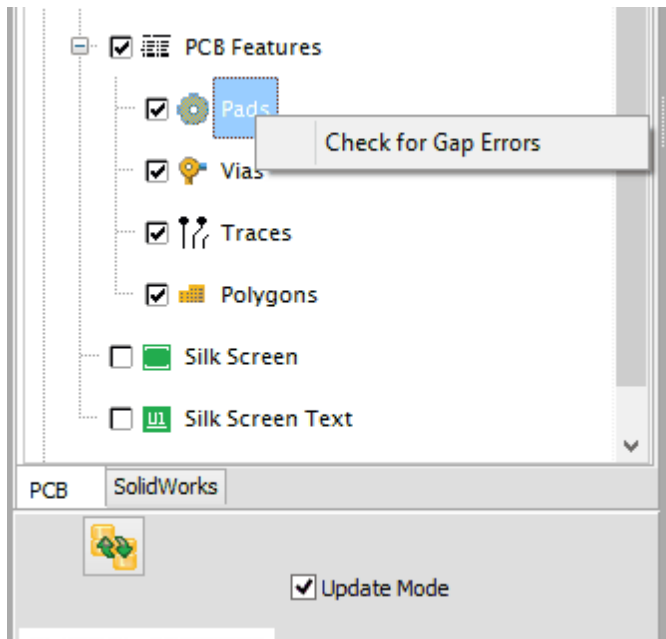
2.5.1 Top and Bottom Mode

When the PCB Features node is visible, the Panel is in Top and Bottom Mode.

When you check any of the check boxes below this node, the feature in SolidWorks will be created for the top layer and the bottom layer only. ie if you check Pads, copper pads will be created on the top and bottom of the PCB outline.

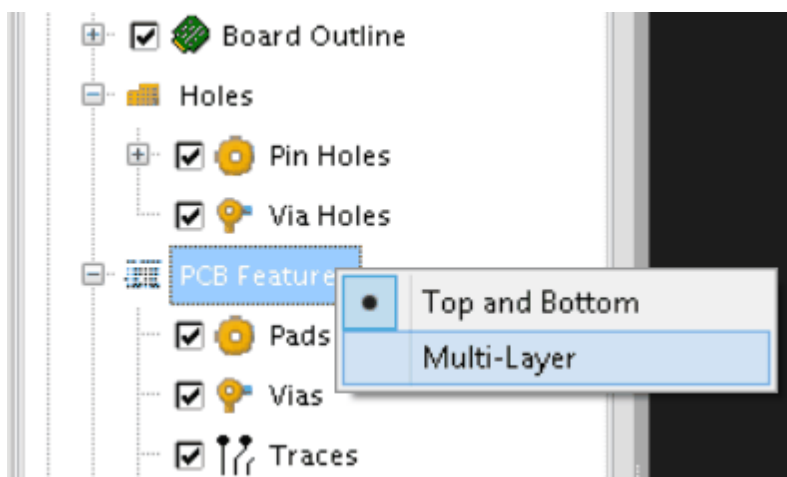
Right click on any of the nodes, allows you to invoke the Check for Gaps command - see the topic on resolving extrusion issues.

When the Update Mode check box is checked, when the Update from PCB button is pressed, the features corresponding to the node are updated or created if they do not exist.



2.5.2 Multi-Layer Mode

To invoke Multi-layer mode, right Click on the PCB Features node and choose Multi-Layer.



The tree then re-populates with a node for each layer. This allows you to see the layers of the PCB. Check boxes to select elements to export are provided on the Top Layer node. Checking a check box selects the elements for all layers.

2.5.3 How Copper Features are Created

Copper features are created by a process of creating perfectly enclosed shapes around all elements of the type selected. These shapes are initially created using lines. A Lines to arcs routine is then applied to reduce the number of elements that define the shapes.

These shapes are then written to a sketch and extruded.

SOLIDWORKS requires that sketches contain perfectly enclosed shapes before extrusion succeeds. The program checks all shapes for gaps before sending them to the sketch. Most gaps can be closed, but if any cannot be closed, a separate sketch is created containing elements that connect be extruded.

2.5.4 Procedure for Preparing a PCB for Extrusion

Below is the procedure to follow to prepare for Extrusion a PCB your are unfamiliar with:

1. Extrude each feature separately to determine if there are any extrusion issues.
2. If the PCB is not huge, extrude the complete PCB, otherwise, invoke Large Design Mode
3. Pads and Vias Pads will rarely have issues.
4. If there are any detectable gaps in the shapes sent to SolidWorks, there will be a sketch containing the name Gap Errors - if you see this sketch, open it and see where the gaps are. Go back to the source PCB and determine the cause of the gaps and correct if possible.
5. 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.
6. To find the cause of the Extrusion failure, edit the sketch and invoke the SolidWorks command Tools>Sketch Tools>Check Sketch for Feature. This command will tell why the extrusion failed and guide you to the elements causing the issue.
7. 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.

2.6 Rigid/Flex Notes

This topic provides an overview on the Rigid/Flex support.

Below is the sequence of events that take place when Export to SolidWorks is invoked when a PCB document contains a Flex PCB:

1. All Board Outline Regions are added to Lists and their type (ie Flex or PCB) is stored.
2. The Layer Stack for each board region is added to the the List for each region.
3. For each Board Region, the following occurs:
 - a. A SolidWorks part for board shape is created
 - b. for each PCB feature item that is checked, the spatial filter separates the PCB features for this Board Region and adds them to the part
4. A List for all components in the PCB document is created, then, using the Spatial Filter, each component is assigned to a Board Region
5. An Assembly document for the Rigid.Flex PCB is created
6. For each Board Region, the following occurs:
 - a. An assembly is created
 - b. The Board Shape Part is Placed
 - c. The parts associated with this Board Region is placed

The end result is a SolidWorks assembly containing an assembly for each PCB

3 How the Product Works

When you run the 'Create Assembly' command, if SolidWorks is already running, the program connects to it, if not, SolidWorks is launched and it runs in background mode.

The product then performs the following actions:

1. A SolidWorks part is created containing the PCB outline extruded to the thickness of the PCB. Holes are also added as Cuts.
2. A SolidWorks part is created for each PCB component that does not have a corresponding part in the SolidWorks part folders.
3. A SolidWorks assembly with the same name as the PCB is created
4. The PCB Outline part is added followed by SolidWorks parts for each PCB component.

When you choose the 'Import from SolidWorks' command, the following actions are performed:

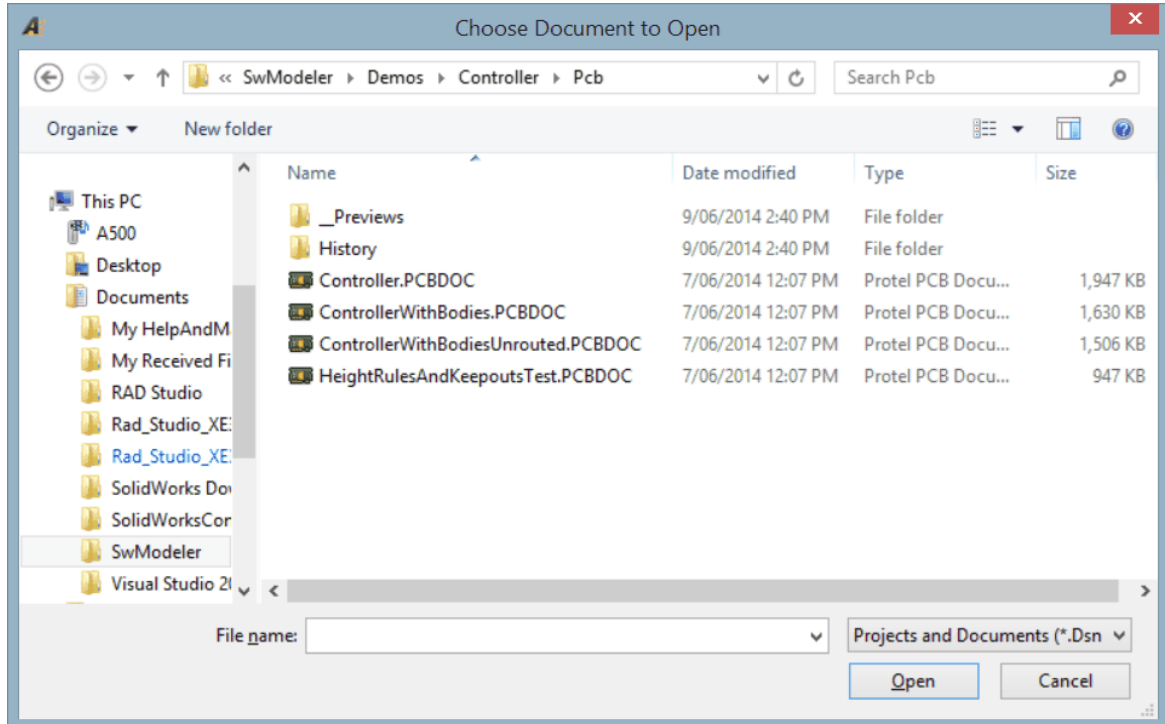
1. The program connects to SolidWorks and opens an Assembly with the same name as the PCB
2. The PCB Board Shape is updated
3. Mounting Moles in the PCB are moved to line up with Mounting Holes defined in the Assembly
4. Components are moved to line up with corresponding Parts in the Assembly.

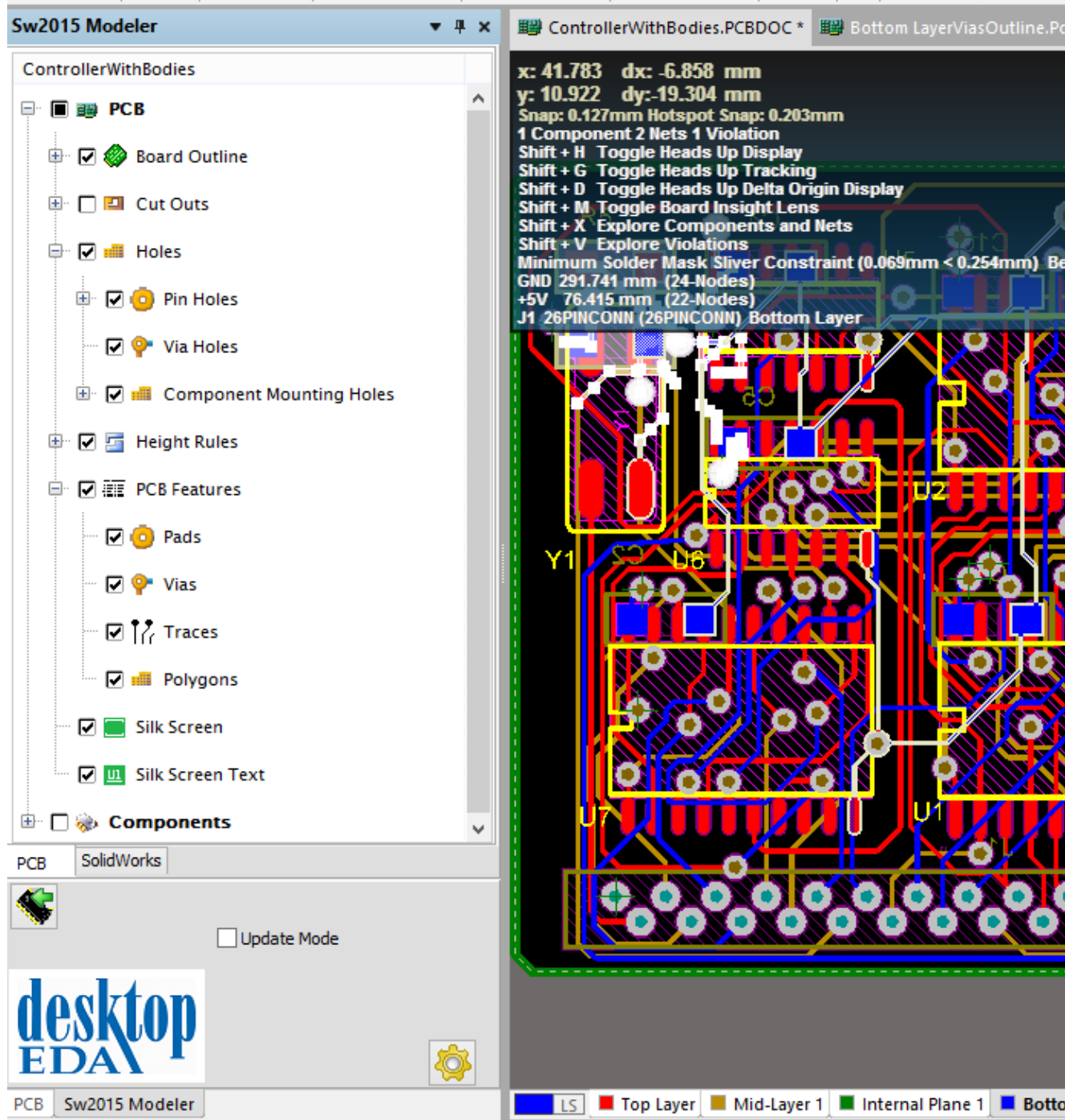
This product requires that SolidWorks is installed on the same PC as Altium Designer. If this is inconvenient for your company, consider the alternative approach of using IDF files. Contact Desktop EDA for more information.

4 Setting Up Your PCB Document

The following sections describe the set up requirements in your PCB document.

Open the PCB named Demos\Contoller\Pcb\ControllerWithBodies.pcbdoc and use this in the following topics.

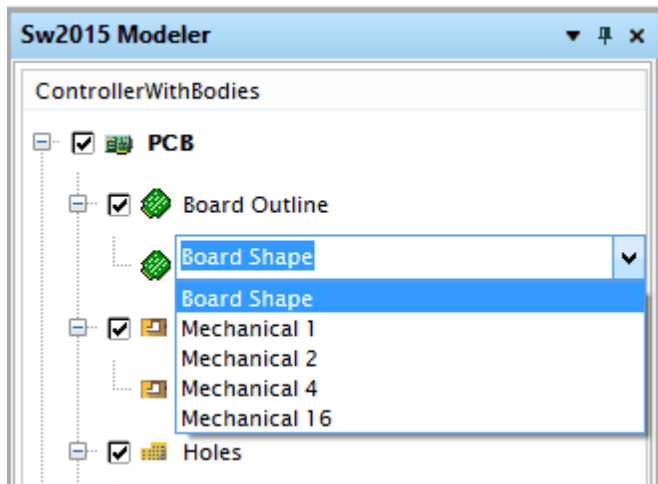




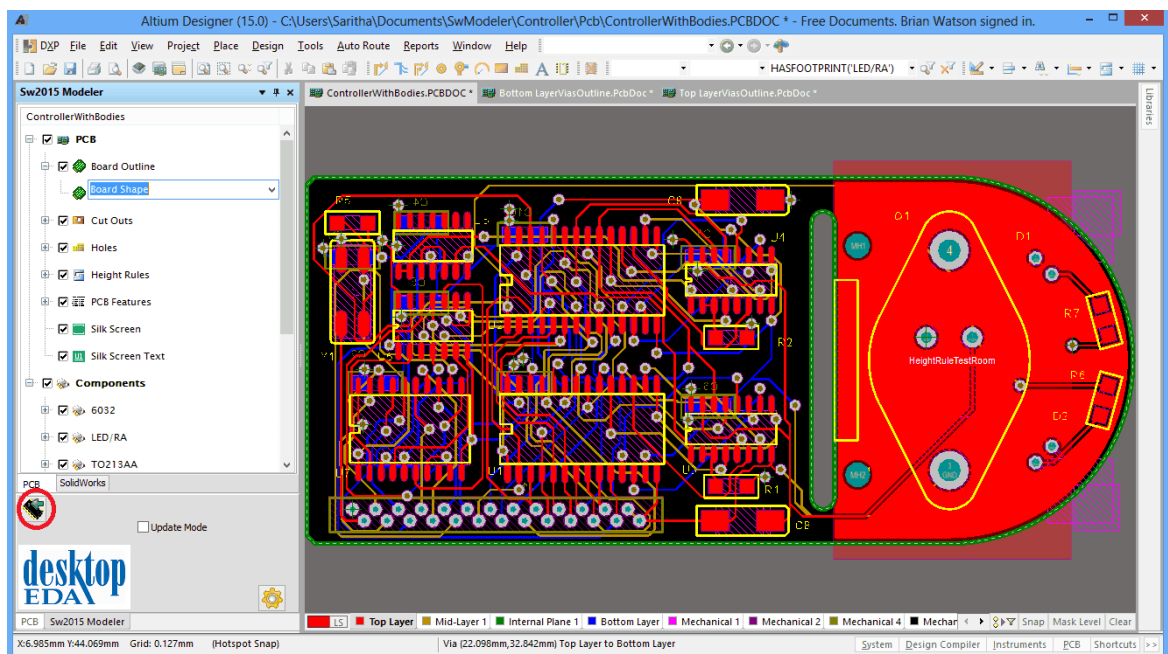
4.1 Defining the Board Outline

You have the choice of either using the Board Shape or a shape made up of Tracks and Arcs drawn on a mechanical layer to define the shape of the PCB that is used in the 3D environment.

You tell the program which method you are using by double clicking on the child node of the Board Outline node to display a drop down list.



If you choose Board Shape, then that is all you need to do.



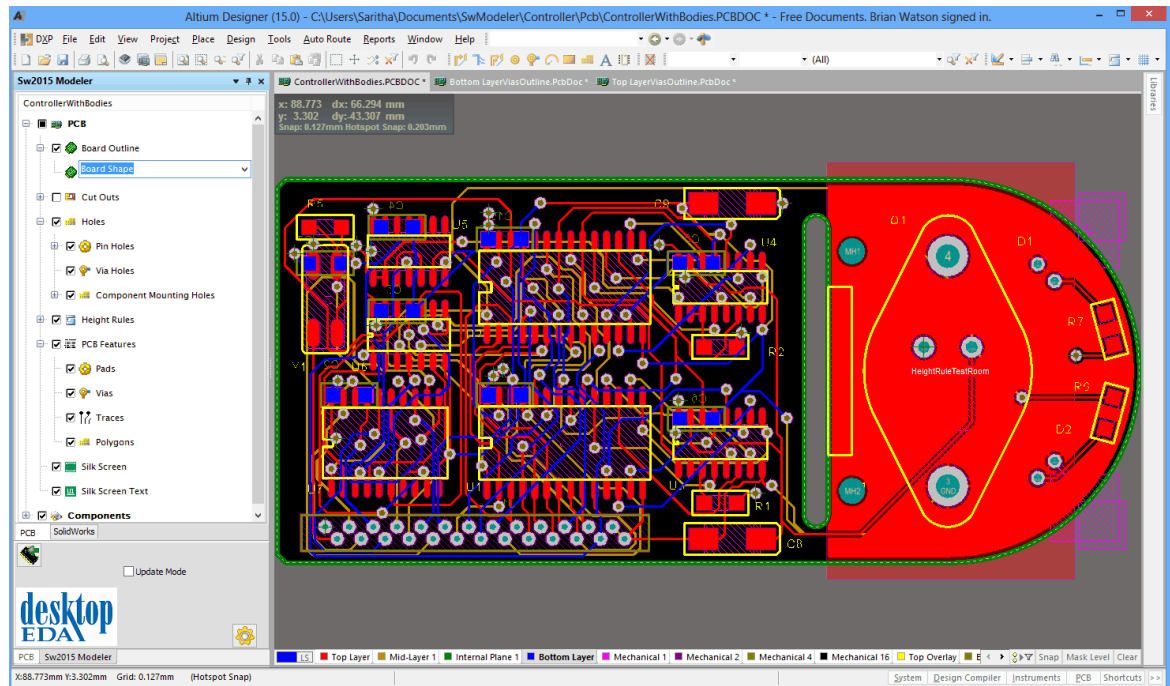
If the PCB has "cut outs", you can either use Altium Regions defined as Board Cutouts (if you choose the Board Shape option) or you can define shapes on the mechanical layer you use as the Board Outline layer.

We will choose Board Shape for this example.

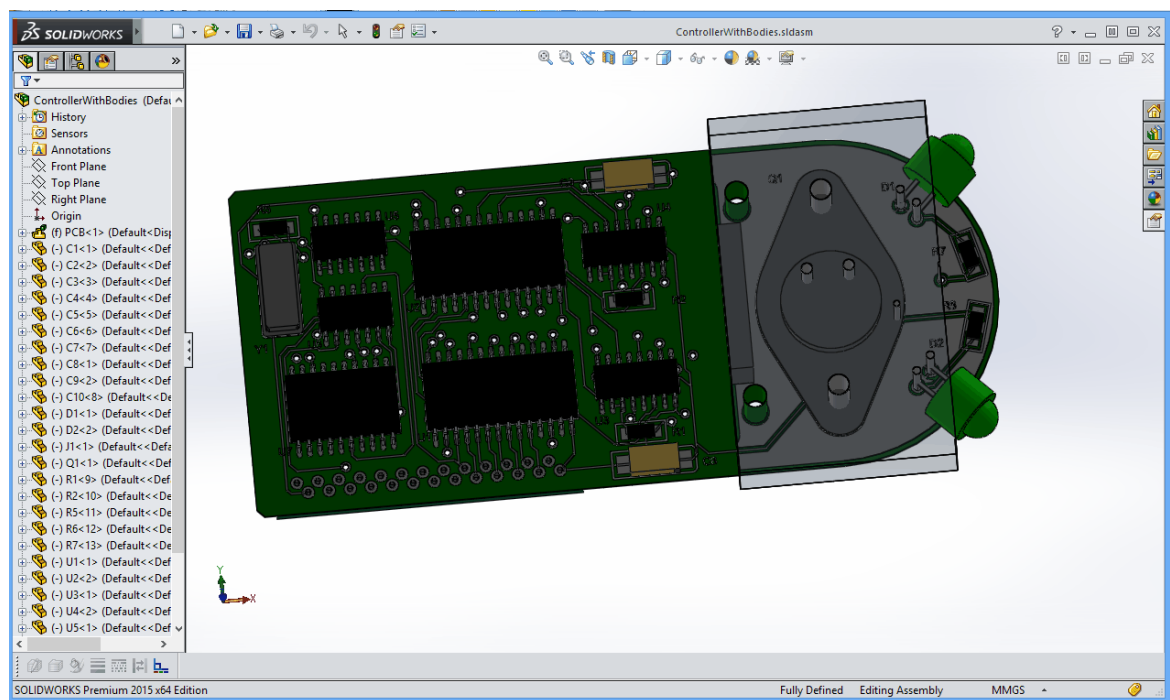
You can define Mounting Holes by placing multi-layer Pads and assigning their designators with the prefix "MH" and setting a hole size. (this prefix can be changed by setting the MH Prefix field in the Preferences dialog).

This PCB has two mounting holes as shown. Mounting Holes are always added exported whereas other holes are exported if the appropriate check boxes are checked. Only Mounting Holes can be

imported from SolidWorks.



This above PCB produces a part in SOLIDWORKS like this:

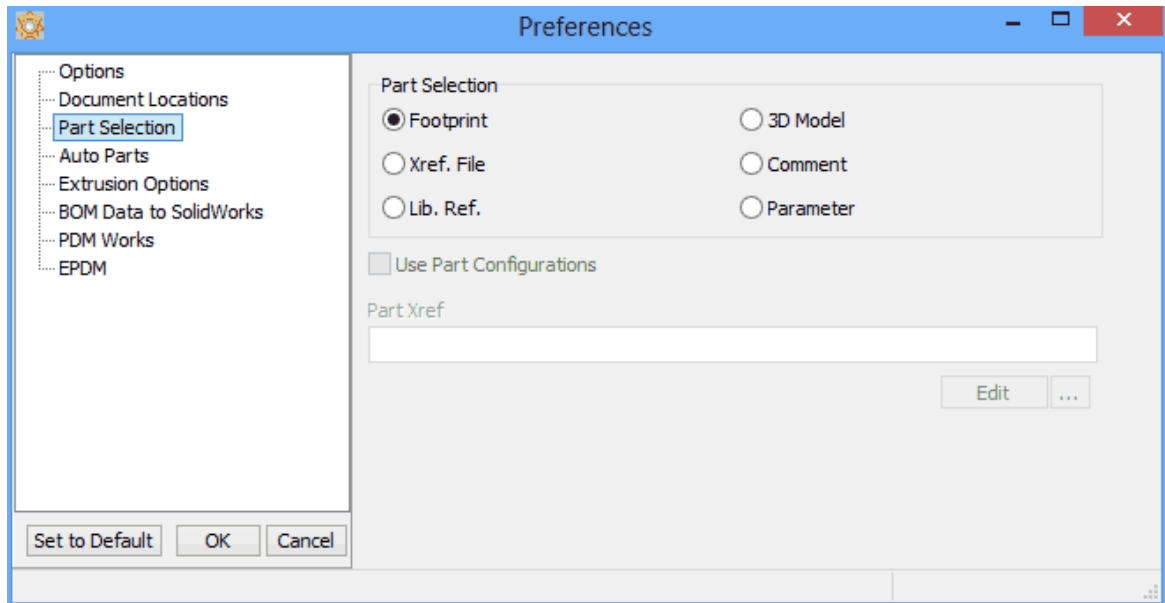


4.2 Using SolidWorks Parts

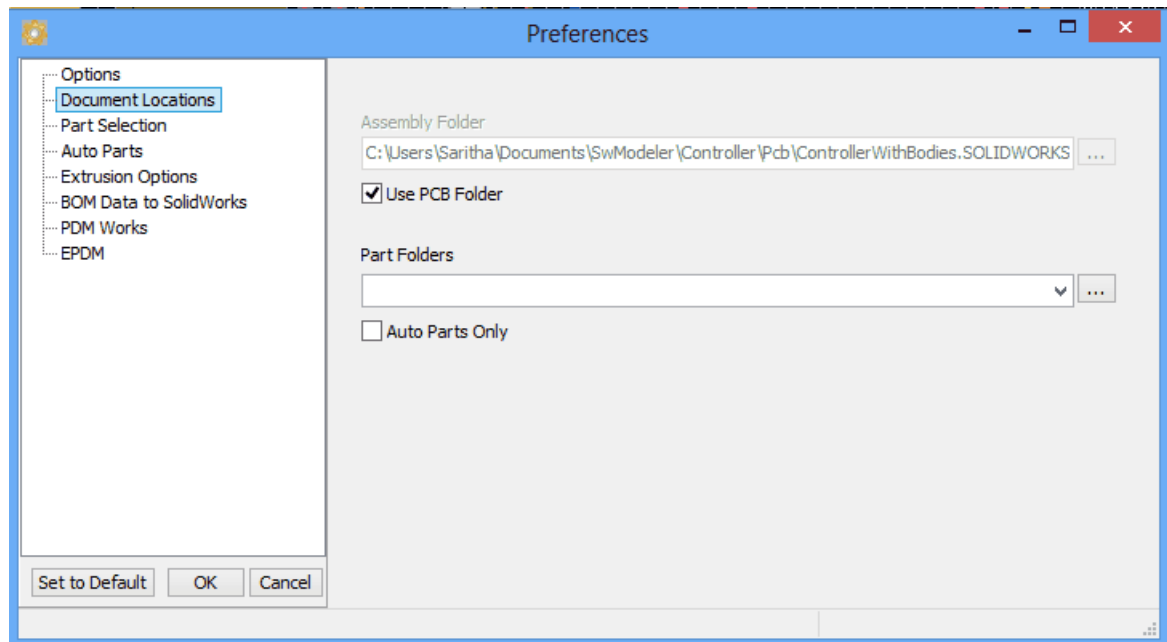
If you have SolidWorks parts that model your PCB parts, the program uses them. Otherwise, it automatically creates parts using Bodies or geometry extracted from the PCB component.

There are a number of methods available to associate a SolidWorks part with a Footprint. You determine the method settings on the Part Selection page of the Preferences dialog.

For this exercise, choose Footprint.



This tell the program to search for a Part with the file name that is the same as the footprint name. You assign the folder that is searched for parts in the Part Folders field on the Document Location page of the Preferences dialog.



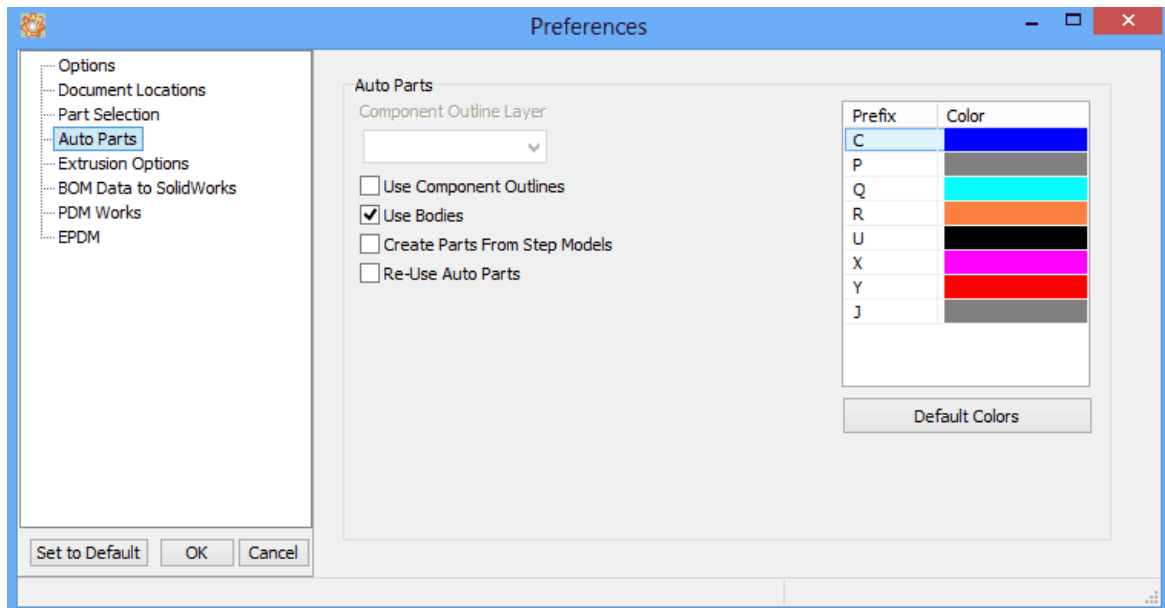
If a Footprint name contains characters that are not allowable in a file name, that character is replaced by an underscore. See the table below for examples.

Footprint Name	Part name that is searched for
Dip14	Dip14.sldprt
DB37RA/F	DB37RA_F.sldprt
RAD0.2	RAD0.2.sldprt

4.3 Setting Up For Auto Parts

If there is no SOLIDWORKS part in the Part Folders for any component on your PCB, the program creates one.

The Auto Parts page of the Preferences dialog allows you to control how Auto Parts are created.



If footprints contain embedded step models, checking the Create Parts From Step Models check box will enable the Step Model to SOLIDWORKS part conversion.

If the Use Bodies check box is checked and the Component contains Bodies, then these are used to create the Auto Part.

If Use Component Outlines is checked, the layer selected in the Component Outline Layer drop down list is searched for an enclosed shape. If one is found, this is used to define the shape of the Auto Part.

If a shape is found, or if Use Component Outlines is not checked, the Silk Screen layer is checked for an enclosed shape. If one is found, this is used.

If all of the above fail to provide data to create an Auto Part, a shape is created from the bounding rectangle of the component, ignoring Designator and Comment text.

The enclosed shape is extruded by the Height assigned to the component to create the Auto Part.

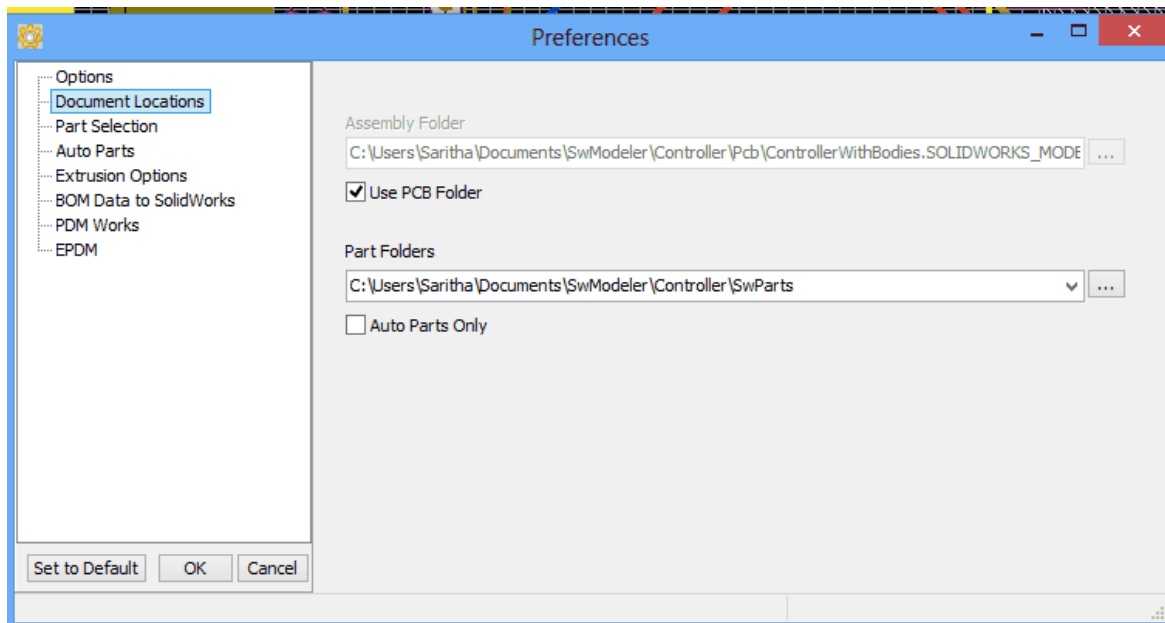
You can edit the values of Components using the Height Editor. You can display the Height Editor by pressing the Height Editor button on the product toolbar

5 Creating An Assembly

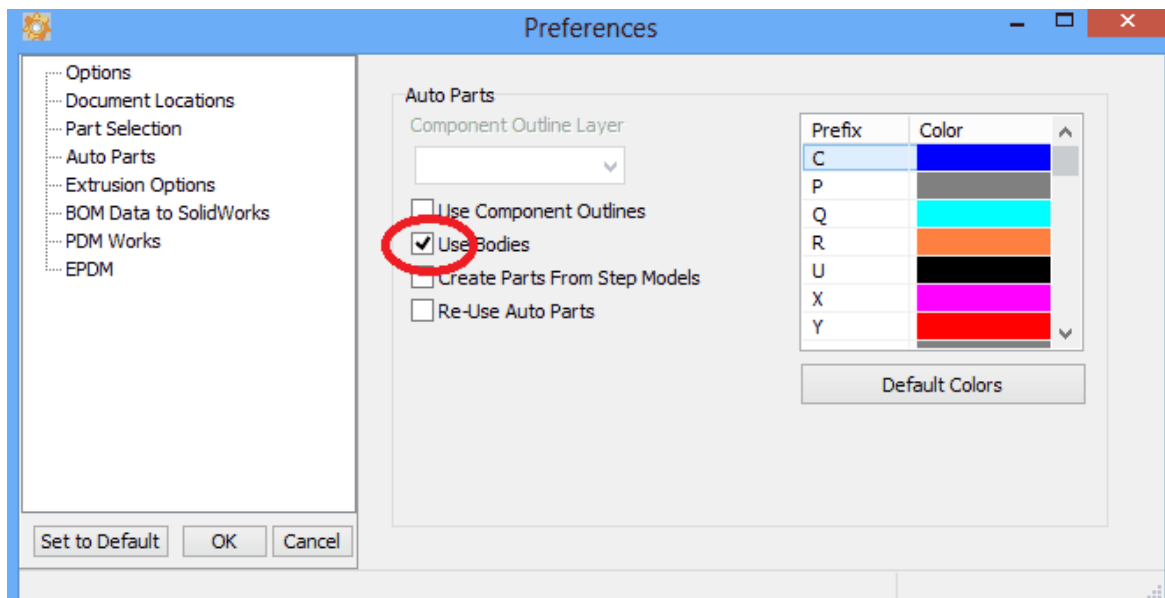
This topic guides you through the process of creating the Assembly.

5.1 Panel Settings

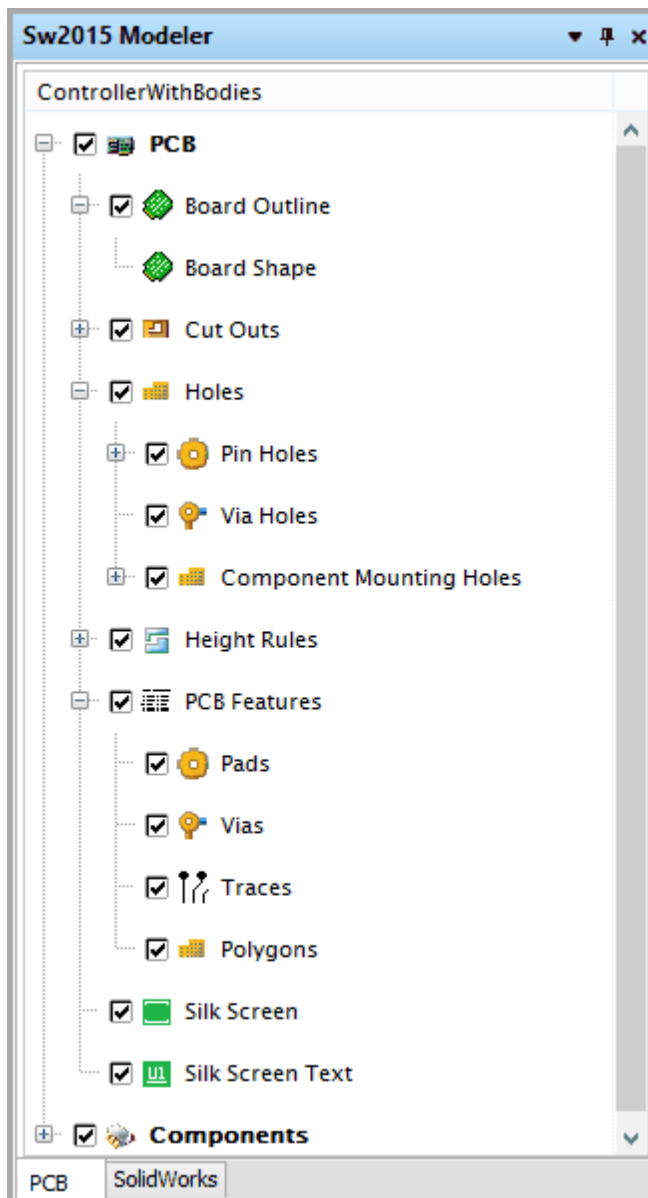
Set up Preferences as shown below.



If you want to work with Use bodies option shown below on the Preferences dialog, Use C:\Products\SwAltiumDesignerV8\Demos\Controller\Pcb\ControllerWithBodies.PCBDOC file



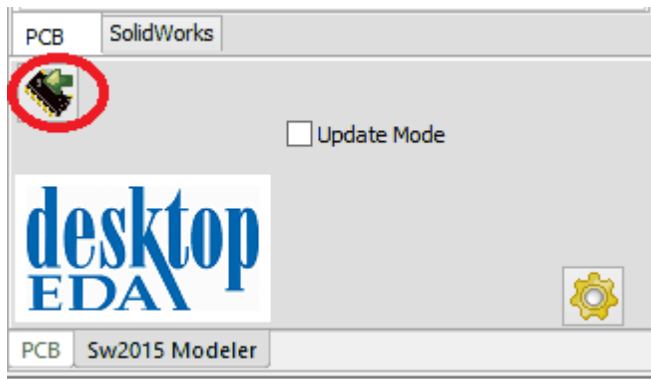
Set the Panel check boxes as shown:



5.2 Export to SolidWorks

Launch SolidWorks and arrange the windows such that you can see both the Altium Designer and SolidWorks windows.

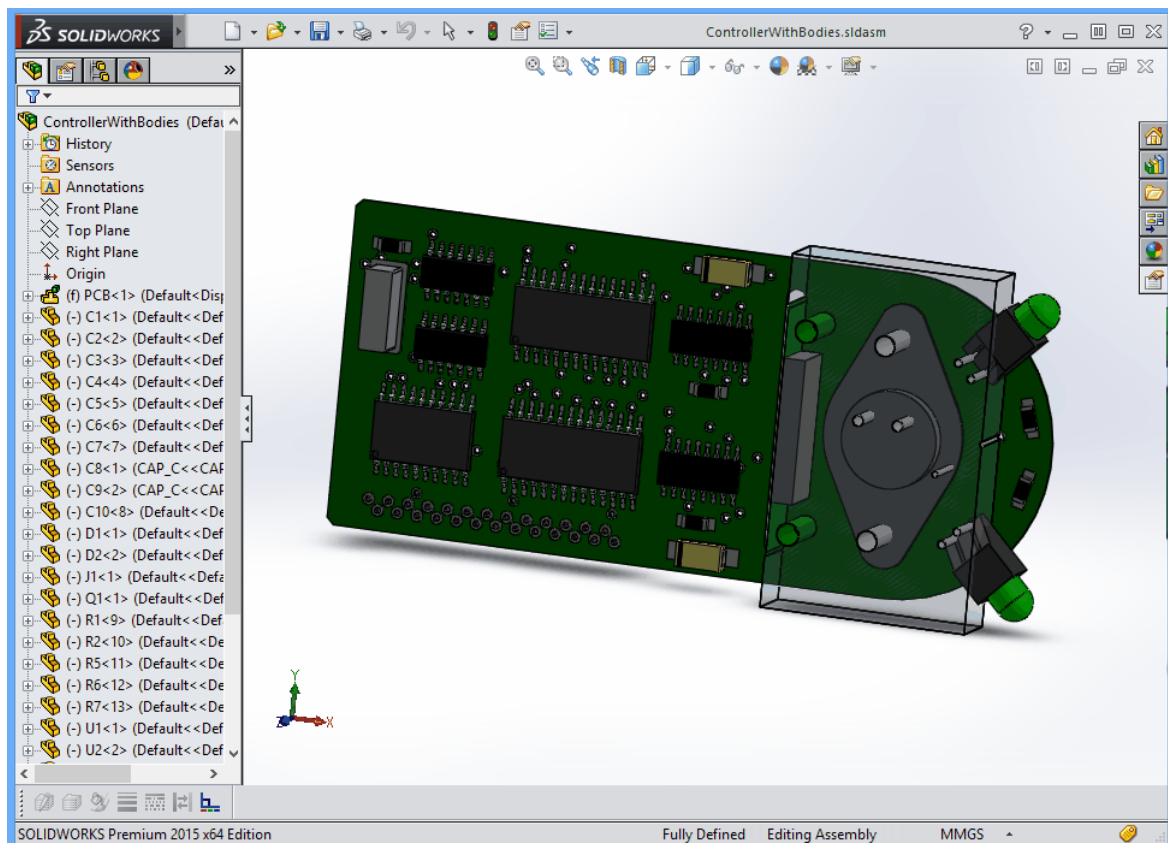
Click on the button shown.



Watch the Altium Design status bar for progress messages and also watch the SolidWorks window. You will see the Assembly being built.

You should see a PCB similar to that shown in the screen shot below.

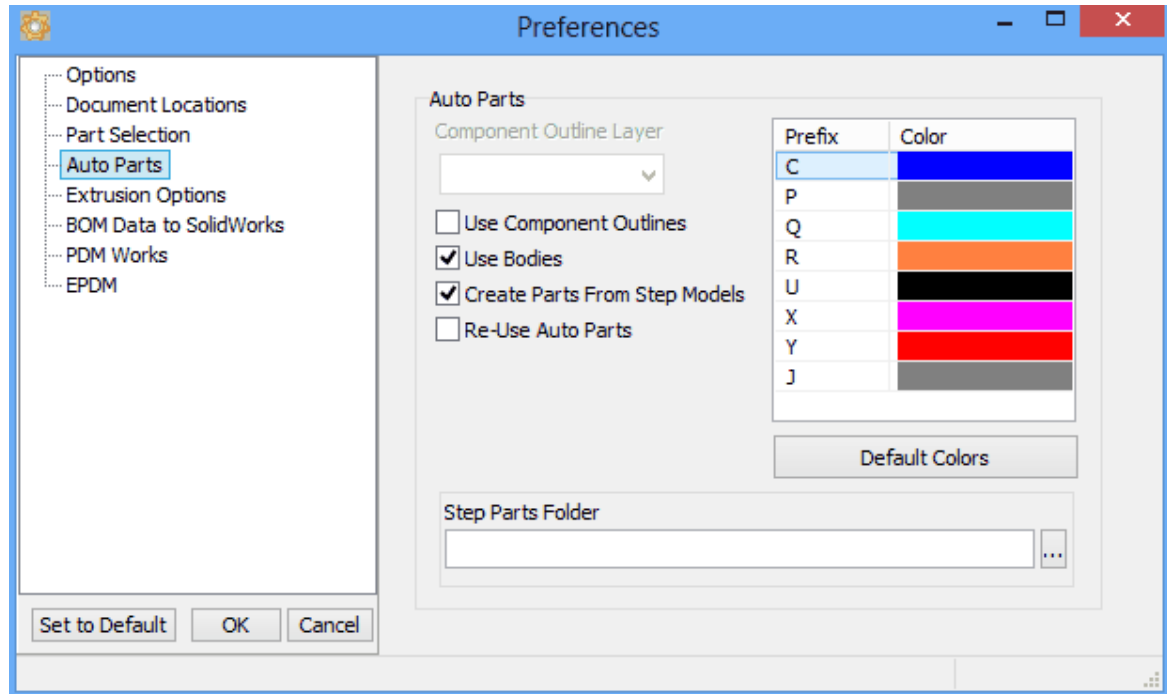
Review the Messages Panel in Altium Designer - this displays a log of the assembly creation process.



5.3 Using converted Step Models

This topic guides you through the process of creating an Assembly using part created by converting the Embedded step models to parts.

Go to Auto Parts page of Preferences dialog and Set up Settings as shown below.

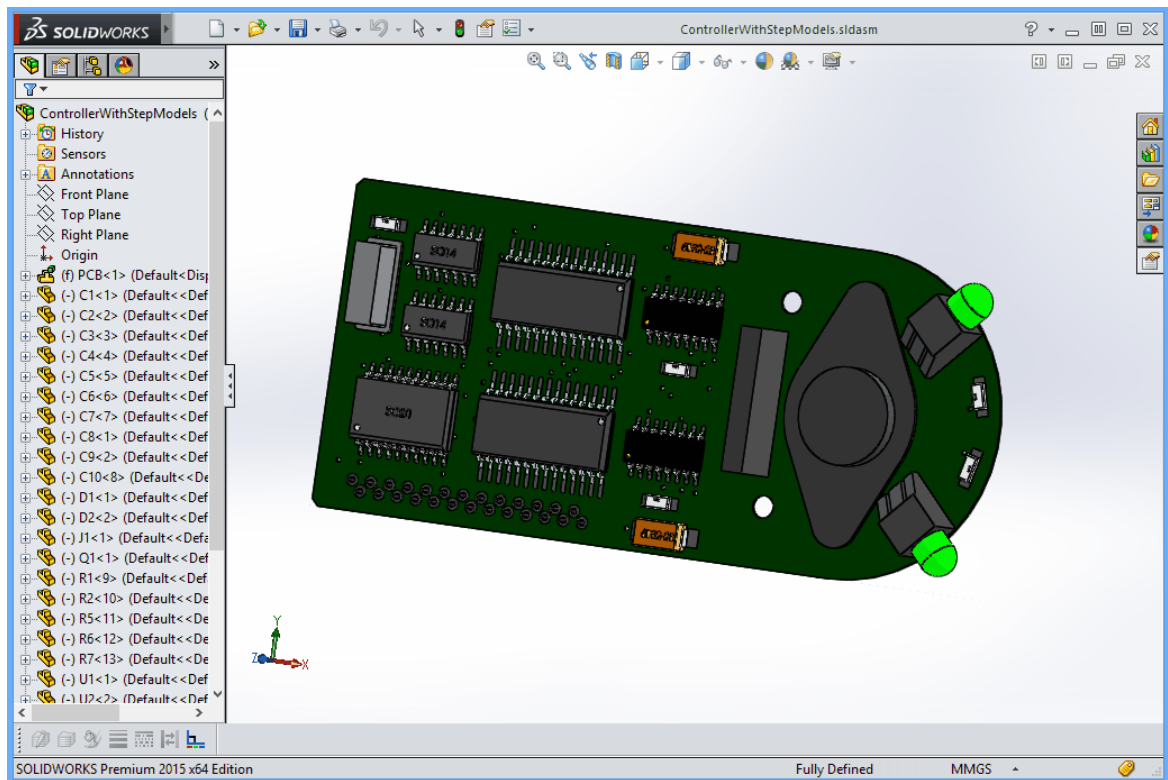


When Create Parts From Step Models check box is checked, the Step Parts Folder field is displayed. Click on the browse button to the right of the field to choose the folder in which Step parts are to be placed.

If Step Parts Folder path is left blank, the Step model parts will be created in a folder within the PCB folder.

For this example we are using ControllerWithStepModels.PCBDOC (It has embedded step models in its footprints) to create a SolidWorks assembly.

You should see a PCB similar to that shown in the screen shot below.



To create the assembly, following the instructions in the Export To SolidWorks topic.

6 Updating a PCB from SolidWorks

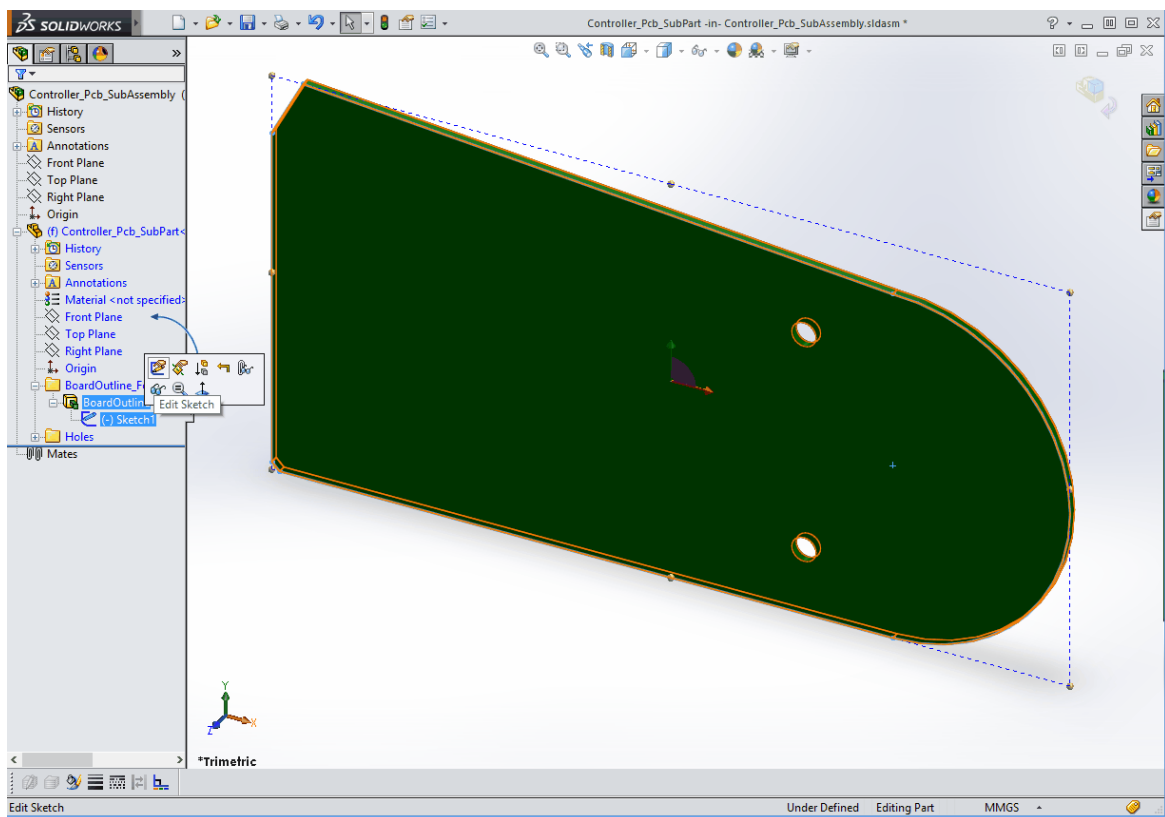
This topic guides you through the process of Updating your PCB from SolidWorks.

6.1 Making Changes in SolidWorks

In the assembly ControllerWithBodies.sldasm', perform the following:

1. In the Feature Manger, expand the entry for the part named Controller_Pcb_SubPart by clicking on the plus sign.
2. Edit the sketch defining the feature named BoardOutline by clicking on the entry in the Feature Manager and choose Edit Sketch
3. "Stretch" a line defining on the the left side of the PCB
4. Exit this sketch (right click in the Sketch and choose Exit).

Now the Part should look like this:



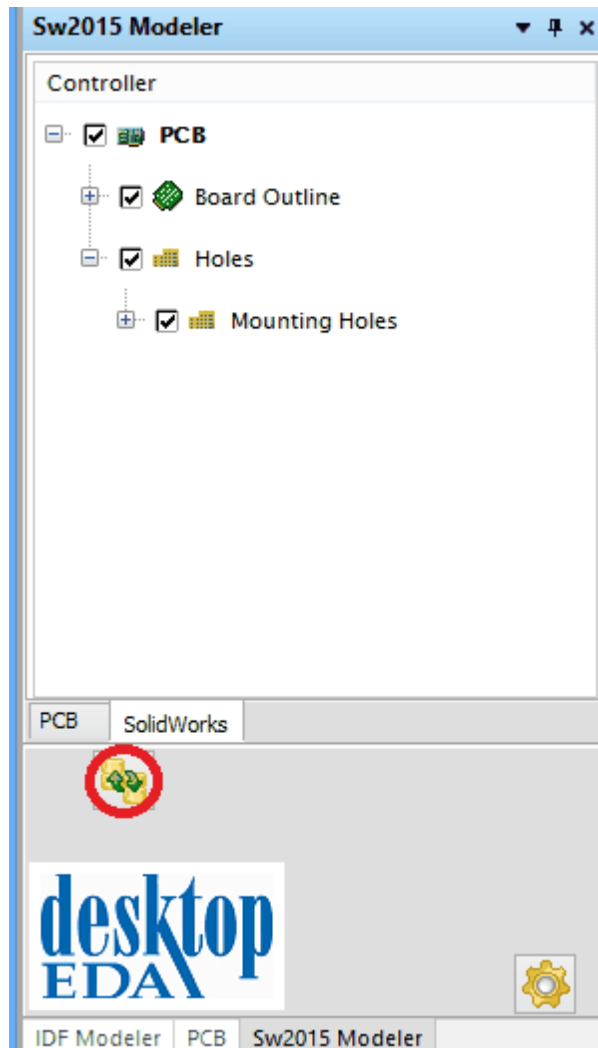
6.2 Updating the PCB

In the Altium Designer window ensure that the ControllerWithBodies.pcbdoc is open.

Invoke the SolidWorks view in the Panel by clicking on the SolidWorks tab as shown.

This displays a tree view of the assembly - with a node representing each feature.

To update the changes made in SolidWorks, check the Board Outline check boxes and press the Synchronise button show below.



The PCB should now look like this:

