

# SOLIDWORKS PCB

## PDM Implementation for SW PCB



**3DEXPERIENCE®**

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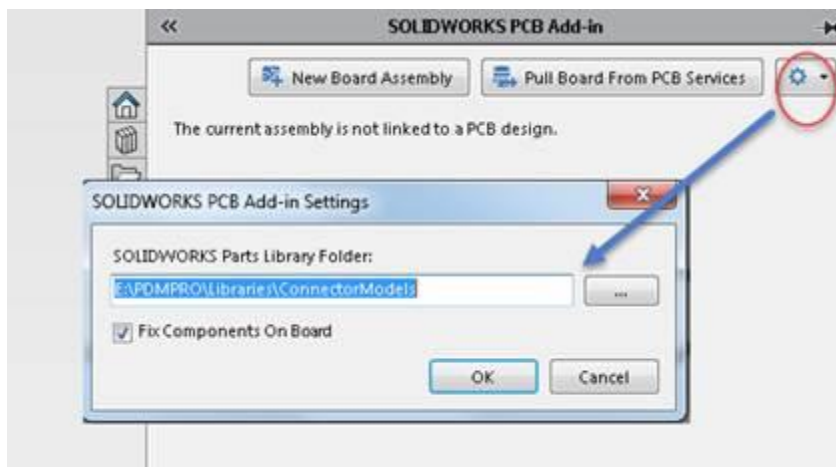
## SOLIDWORKS PDM implementation for SOLIDWORKS PCB

The out of the box solution for collaboration between ECAD (SW PCB or Altium) to SOLIDWORKS works as designed. In the section below, we explain how our solution works between SOLIDWORKS and SW PCB or Altium Designer.

### Out of The Box Solution:

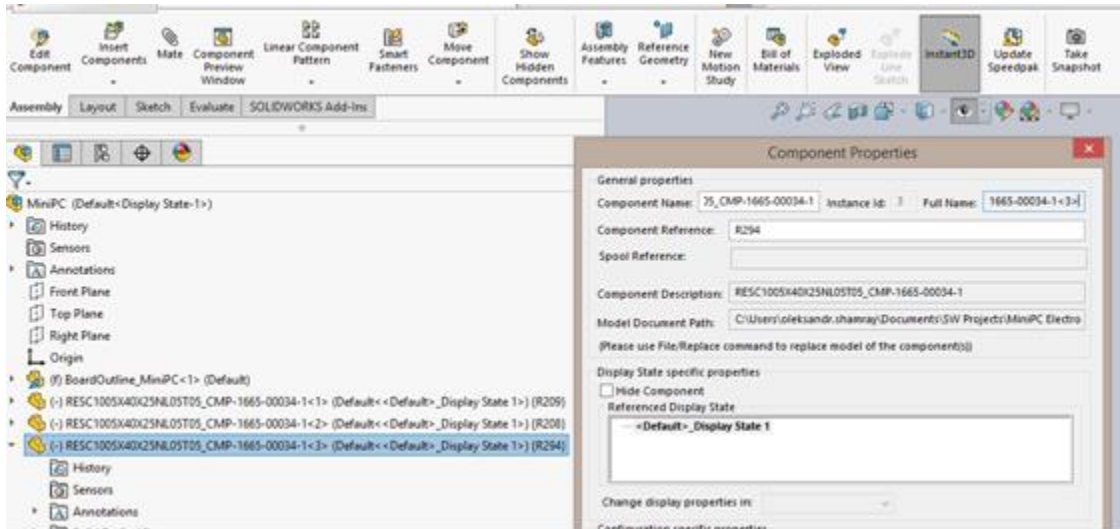
The out of the box solution available today, when a model (Generic or Extruded) embedded to the footprint in the PCB and a push is performed from SW PCB or Altium Designer to SOLIDWORKS, the collaboration engine creates “new instances” in Electronics Folder. By default, this folder is created in the same directory as the assembly.

If needed or required, this location can be changed by pointing to any other folder location in SOLIDWORKS PCB Add-in panel (from SOLIDWORKS) under the settings icon.



The component name is set as the “footprint name” and component reference is set as “designator”. Each component also has “instance id”, assigned by SOLIDWORKS. It is assigned as “1” if there is only one component with a model name.

If there are more than one component with same model name, the instance id will increase for each component where the instance id numbering is controlled by SOLIDWORKS.

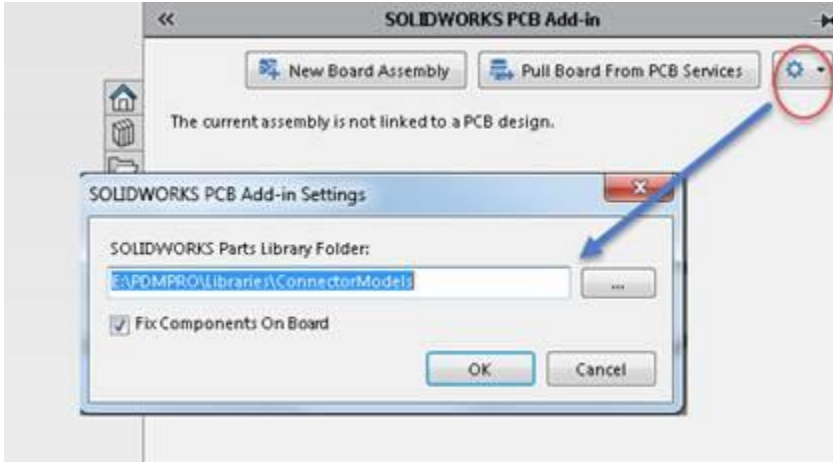


These models are now added to the feature-tree in SOLIDWORKS, and special parameters are created to maintain the link between the model in SOLIDWORKS CAD and the footprint in SOLIDWORKS PCB. If a new part is added to the PCB and has a model that was already generated from a previous pull (using the SW PCB Connector) for another part, the collaboration process will just pull from the generated part using existing models. It will NOT create a new instance. The new part will be listed in the feature-tree with a unique index <> appended at the end, as shown in the above screen.

### Using Existing Mechanical Parts from PDM:

It is possible to leverage and use existing mechanical models from PDM, however this will work under certain conditions and restrictions:

- 1- The footprint name in SW PCB or Altium Designer and mechanical model name must match
- 2- The footprint must be placed directly from the PCB Library in the PCB document and NOT through the ECO (update PCB from schematic)
- 3- The mechanical user must point the SOLIDWORKS parts library folder to existing mechanical models (the location where they keep the mechanical parts)



With this technique, the collaboration engine will redirect SOLIDWORKS to use the existing mechanical models and it will NOT generate to what we refer to as “downloaded models”, which consists of para-solid models and the .sldprt.

Note: The mechanical teams don't end up with duplicated mechanical models.

The currently-available PDM solution is at an intermediate stage. This means, both teams (electrical and mechanical) can start collaborate using a streamline mechanism. AND they can track changes of the printed circuit board outline and critical parts (such as connectors, LEDs, Displays, etc).

Assuming that these component footprints exist and their correspondent SOLIDWORKS model also exist, those components can be placed directly on to the PCB before the schematic development. Once the printed circuit board-outline and parts positions are locked down, then the Electrical Engineering team can add the symbols and establish component links manually between the existing footprints and their correspondent schematic symbols.