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Note

All SolidPractices are written as guidelines. You are recommended to use these documents only after properly evaluating your requirements. Distribution of this document is limited to Dassault Systèmes SolidWorks employees, VARs and customers that are on active subscription. This document may not be posted on blogs or any internal or external forums without prior written authorization from Dassault Systèmes SolidWorks Corporation.

This document was updated using version SOLIDWORKS 2019 SP01. If you have questions or need assistance in understanding the content, please get in touch with your designated reseller.
1) Preface

SOLIDWORKS PCB enables the productivity you need to design printed circuit boards (PCBs) quickly along with unique, collaboration between electrical and 3D mechanical design teams. It offers a clear advantage where ECAD-MCAD collaboration is critical for overall success of electronic product design.

SOLIDWORKS PCB in conjunction with the “Vault” technology in SOLIDWORKS PCB Services brings support for Managed Projects. SOLIDWORKS PCB Services also provides localized and centralized version control.

The intention of this SolidPractice document is to help customers and VARs become familiar with the SOLIDWORKS PCB functionality and user experience.

Your Feedback Requested

We would like to hear your feedback and also suggestions for new topics. After reviewing this document, please take a few minutes to fill out a brief survey. Your feedback will help us create the content that directly addresses your challenges.
2) Managed Projects

A managed project is the combination of Meta information stored on the server, along with tight integration with the version control service (SVN). Users can create a new managed project via the New PCB Project dialog box in SOLIDWORKS PCB, or from the Projects browser-based interface in SOLIDWORKS PCB Services. During creation, the user defines the project name, description, and design repository.

Note: For Altium Designer users only, you can convert a standard non-managed project to a managed project for collaboration with SOLIDWORKS mechanical only with the purchased extension of SOLIDWORKS PCB Connect.

Managed projects are version controlled by default. You can work on these projects collaboratively without the need to worry about shared drives.

Prior to creating a new managed project or converting an existing non-managed project, make sure that a design repository exists. Design repositories are centrally managed as part of the SOLIDWORKS PCB Services. Installing SOLIDWORKS PCB Services automatically creates a default repository. Alternatively, you can connect to external repositories as discussed later in section 2b “External Access to a Local SVN Design Repository”.

Irrespective of how you create a repository, centralized management of your organization's design repositories ensures that all of your projects are stored exactly where they are supposed to be.

For general collaboration compatibility, users of Altium Designer need to refer to the following table before installing the SOLIDWORKS PCB Connector extension. The table lists compatibility between Altium Designer/Nexus, SOLIDWORKS PCB Connector, and the SOLIDWORKS version.
Table of Compatibility between SOLIDWORKS and Altium Designer products

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a) Creating a Repository through SOLIDWORKS PCB Services

You can create repositories through the local version control service (VCS). It is also possible to set the connection to external repositories. All repositories are centrally managed through the VCS browser-based interface in SOLIDWORKS PCB Services, in terms of:

- Repository name – **Description** and **Repository Path**
- Repository accessibility – Which specific users can access the repository (or roles in the case of repositories defined through the local VCS)
• Repository availability – Users can add or remove repositories centrally, rather than create and connect to repositories independently.

The local VCS provides a repository with the name **Default Repository**. While you cannot rename this repository, you can add a description for it, manage user access to it, or remove it, as necessary.

When a user signs in to SOLIDWORKS PCB Services in SOLIDWORKS PCB, the design repositories available to them will be added to the **Data Management - Design Repositories** page of the **Preferences** dialog box in SOLIDWORKS PCB automatically.

**i) Adding a Repository**

To add a design repository, click the **Add Repository** button at the top-right of the page. The **Repository Properties** dialog box appears from which you will define the repository. The properties required depend on whether you are creating a new repository using the local VCS, or linking to an existing, external repository:

• **New** – This option requires that you enter the repository a name and a description.
- **Existing** - In addition to a name and description, this option requires that you supply the URL to the repository, and the credentials (**User Name** and **Password**) necessary to access that repository (if required).

When creating a repository that is internal to the PCB Services, the name should start with, and can contain the characters A-Z, a-z, or 0-9. Underscores, dashes, and spaces are allowable within the middle of the name. You cannot use the following words: AUX, COM1-COM9, LPT1-LPT9, CON, NUL, and PRN. There are no naming restrictions for an external repository.

A linked, external design repository is distinguishable in the list of repositories by the appearance of a check (✓) in the **External** property check box.

To edit the properties of a design repository at a later stage, simply click the associated (Edit) control. To remove a design repository, click the associated (Remove).

**ii) Sharing a Repository**

To share a repository, click on the (Share) control that is associated with the repository.

After centralizing your design repositories, you need to make sure that users who require access to a given repository - including the designers who will be working on board designs - have that access. You can achieve this by sharing that repository, or rather managing its access permissions.
The **Manage permissions** dialog box will appear, with all the controls necessary to share the repository with other users.

### Considerations for sharing a repository

For a local repository created through the SOLIDWORKS PCB Services VCS (i.e. not external), that repository is shared, by default, with the user who creates the repository, and with users who have the Administrators role. Both of these have full Read and Write access.

By default, an external repository is shared with no one, not even with the user who added a connection to it through the interface.

Only local repositories can be shared with defined roles for the vault, or publically. Roles are listed after individual users.

In terms of permissions, a user or role has Read and Write access when the **Can Write** option is active. If this option is not active, the user has only **Read** access.

To remove an existing user or role from having shared access to a repository, click the associated × **(Remove)** control.
iii) Sharing a Local Repository with a User

To share a local (not external) repository with another user:

1. In the Manage permissions dialog box, click the Add button.

2. In the User Properties dialog box that appears, start typing the full name, username, or email address of a user in the Users field. This reveals a list of matching users. Select a user from this list. You can select multiple users.
   - To remove a user, click the ✗ (Remove) control at the far right of their name.

3. Set the permission for the users by using the Rights options to choose between Read access, or Write access, or both.

4. To save your changes and return to the Manage permissions dialog box, click OK.

5. Click Close.
To edit the permissions for an existing user, simply use the **Edit** option in the **Manage permissions** window, to toggle between **Read** and **Write** access.

**iv) Sharing a Local Repository with a Defined Role**

**To share a local (not external) repository with another role:**

1. Switch to the **Roles** tab in the **Manage permissions** dialog box, and then click **Add**.

2. In the **Role Properties** dialog box that appears, start typing the name of a role in the **Role** field, to reveal a list of matching roles. Select a role from this list. You can select multiple roles.
   - To remove a role, click the ✗ (**Remove**) control, to the far right of its name.
3. Set the permission for the roles by using the Rights options to choose between Read access, or Write access, or both.

4. To save your changes and return to the Manage permissions dialog box, click OK.

5. Click Close.

To edit the permissions for an existing role, simply use the Edit option in the Manage permissions dialog box, to toggle between Read and Write access.

If the repository was created through the local VCS, rather than an external repository, then you also have the ability to share with roles defined for SOLIDWORKS PCB Services vault.
v) Sharing a Local Repository Publically

To share a local (not external) repository publically:

1. In the Manage permissions dialog box, click the Add Public button.

The public entry will be added directly to the list of shared entities, from which you can choose between Read and Write access.

2. In the User Properties dialog box, click the OK button.

In the Manage permissions dialog box, click Close.
To edit the permissions for the public entity, simply use the **Edit** option in the **Manage permissions** dialog box, to toggle between **Read** and **Write** access.

If the repository was created through the local VCS, rather than an external repository, then you also have the ability to share that repository publically, which allows all vault users to access and use it.

**vi) Sharing an External Repository with a Vault User**

To share an external repository with a vault user:

1. In the **Manage permissions** dialog box, click the **Add** button.
2. In the **User Properties** dialog box that appears, start typing the full name, username, or email address of a vault user in the **User** field, to reveal a list of matching users. Select a user from this list.

- To remove the user, click the × (**Remove**) control, at the far right of their name.

Notice that the **SVN User** field will populate with the same user name. While you can only add a single user at a time, you can define as many users for access to the external SVN repository as you need.

3. To save your changes and return to the **Manage permissions** dialog box, click the **OK** button.
Typically, you might use the same (company domain) user name for authentication when connecting to your vault, and your company external SVN repository. This is why the SVN User field populates with the same user name that you select for the vault user. However, if this is not the case, you have the ability to change the SVN User name to suit your particular SVN access. That is, you can map the vault user to the SVN user.

To map a vault user to the SVN user:

1. In the Manage permissions dialog box, click the (Edit) control (•) to the right of the SVN User entry.

2. In the User Properties dialog box, type the user name in the SVN User field. Remember to click the OK button to save your changes.
3. Click Close.

The actual access (Read-only, or Read/Write) is determined externally as part of the SVN repository installation. The vault has no access to the “authz” file for the external SVN repository. Therefore, it is not possible to manage these permissions from the vault side.

b) External Access to a Local SVN Design Repository

You can access a design repository that is created through the local VCS by using an SVN client. Access is made by using the regular network protocol (currently, only svn:// is supported). You can copy the correct repository address from two places:

- From the VCS page of the vault's browser interface. The URL for the target repository appears in the Repository Path field.
From within SOLIDWORKS PCB, from the **Data Management - Design Repositories** page of the **Preferences** dialog box. The URL for the target repository appears in the **Repository** field for the target repository.

The address will depend on whether you are accessing the repository from the same machine on which the vault is installed (such as `svn://127.0.0.1/[RepositoryName]`, or `svn://localhost/[RepositoryName]`), or from a remote computer (such as `svn://[ComputerName]/[RepositoryName]`, where **ComputerName** is the name of the computer on which the vault resides).

c) **Deleting a Repository**

To delete a repository from the vault's VCS, simply click the ✗ (**Remove**) control associated with that repository, on the **VCS** page of the vault's browser-based interface.
Note that removing a repository from the **Data Management - Design Repositories** page of the **Preferences** dialog box only removes that repository from use by that instance of SOLIDWORKS PCB. It does not delete the repository from the vault's VCS. If you sign out of the vault and then sign back in again, the repository will appear once more in the list of available repositories to that instance of SOLIDWORKS PCB.

You can delete external repositories, irrespective of whether or not they contain any projects. On the other hand, it is not possible to delete local repositories (those created internally through the vault's VCS) if they contain one or more projects. You will receive an alert to this, and should remove the projects first before proceeding with deletion. Even then, if a local repository has no projects, the repository will not actually be deleted, but will instead be moved to an archive directory (`\ProgramData\SOLIDWORKS\SOLIDWORKS PCB Services\RepositoryDumps`) for a default vault installation. The repository will be contained within a “dump” file, with a Unique ID prefix (such as `775f6c22-b9a1-468c-9f1f-4f217bb3be6b_central design repository.dump`).
3) **Vault Permissions**

a) **Mechatronics Folder Sharing Level**

All users must be added to the SOLIDWORKS PCB Services and given read/write permissions to the desired repository. See the following topics in section 2a: “Sharing a Local Repository with a User”, “Sharing a Local Repository with a Defined Role”, “Sharing a Local Repository Publically”, and “Sharing an External Repository with a Vault User”.

For collaboration between SOLIDWORKS PCB or Altium Designer (collaboration is only applicable to Altium Designer users who purchase the SOLIDWORKS PCB Connector extension) and SOLIDWORKS mechanical teams, all added users in SOLIDWORKS PCB Services are automatically given permissions to the access the Mechatronics folder where the 3D models are exchanged between the electrical and mechanical teams.

It is a recommendation to maintain the “Anyone” in my organization can change permission, which is the default setting.

If necessary, you can change permission levels by logging in to SOLIDWORKS PCB Services as an admin. Under the vault page, right-click on the Mechatronics 3D Models folder and choose **Share**. To edit the Sharing Level, click **Change**.
Note: If only using any SOLIDWORKS PCB Services releases prior to SOLIDWORKS PCB Services 2017 SPx3, vault permissions must be shared with Anyone in my organization.

b) Altium "Paid" Vault

This section is for Altium Designer users who have an Altium paid vault and want to use it to work in place of SOLIDWORKS PCB Services. Users can configure Altium Vault servers’ version 2.5.9 to 3.0.14 for use in place of SOLIDWORKS PCB Services. If you are using the paid Altium Vault to manage design data, you can continue to use Altium Vault instead of SOLIDWORKS PCB Services for ECAD/MCAD collaboration.

There are two considerations for this implementation:

i) You must create and share the 'Mechatronics 3D Models' folder in the Altium Vault

To create this folder:

1. Run SOLIDWORKS PCB/Altium Designer.
2. Sign in to the Altium Vault as a vault admin user.
3. Open or create a project containing a PCB with 3D components.
4. In the SOLIDWORKS PCB Connector, click Push > Post.

To share the folder:

1. Open a web browser and access the SOLIDWORKS PCB Services by typing [Machine Name]:9780
2. Sign in to SOLIDWORKS PCB Services as a vault admin user.
3. Click the **Vault** tab.

4. In the left pane, right-click on **Mechatronic 3D Models > Share Folder**.

5. In the Sharing Settings dialog box, add MCAD and ECAD users to the list and then activate the **Can Edit** attribute to ensure proper sharing.

Alternatively, you can add **Public** to share the folder with public users.

MCAD users require a client access license (CAL). If there is no such license, Altium will create this special license upon request.

To activate a CAL from the dashboard and restrict the license usage, follow these steps:

1. Access the Altium Vault web interface.
2. In the ** USERS > ROLES** section, add a new role (such as *MCAD* or *Mechanical Engineers* etc.)

3. Assign the MCAD users to this role.

4. Go to **ADMIN > LICENSES > ROLES** and add a new license assignment (using the new Collaboration CAL) to this new role.
4) ECAD- MCAD Collaboration

The collaboration technology available with SOLIDWORKS PCB Connector creates new instances of any models (Generic or Extruded) embedded in the footprints in the PCB. Attaching a model to the footprint for the first time creates a mesh body. When performing a push from SOLIDWORKS PCB or Altium Designer (only applicable to Altium Designer users with a purchased extension of SOLIDWORKS PCB Connector) to SOLIDWORKS mechanical, the connector will convert the mesh data to a Parasolid and store it in the mechatronics folder in SOLIDWORKS PCB Services. When the mechanical user in SOLIDWORKS pulls the design, the SOLIDWORKS PCB Add-in converts the Parasolid to a solid part, which creates new instances in the Electronics folder. The software creates this folder by default in the same directory as the board assembly.

Note: A folder for downloaded models is created, which consists of parasolid models.

You can change the path to the Electronics folder from the SOLIDWORKS PCB Add-in by clicking the settings icon and the pointing to any other location.

Upon generation of these new instances, the instances follow a certain naming convention. The part consists of three different entities from three different fields. The first part is the “Footprint name” from PCB Library (a space will be replaced by an underscore “_”). The second part is the “Symbol reference’ from the Schematics library (a space will be replaced by an underscore “_”). The third entity is the revision of the part in the PCB Services (shown in blue square). Each of these are separated by an underscore “_” to create a full name.

If there is more than one component with same model name, the instance ID will increase for each component, where SOLIDWORKS controls the instance ID numbering.
These models become available in the FeatureManager® design tree in SOLIDWORKS mechanical, and special parameters are created to maintain the link between the model in SOLIDWORKS mechanical and the footprint in SOLIDWORKS PCB or Altium Designer (when using SOLIDWORKS PCB Connector extension). If a new part is added to the PCB with a model that was generated from a previous pull (using SOLIDWORKS PCB Connector) for another part, the collaboration process will just pull from the “previously generated” part using the existing model. The process will NOT create a new instance. The new part will appear in the FeatureManager tree.

a) Using Existing Mechanical Parts

Typically, solid part models reside in a PDM vault or in a specific shared network location. It is possible to leverage and use existing mechanical models without creating new instances. However, this will work under certain conditions and restrictions:

- The footprint name and mechanical model name must match
- The footprint must be placed directly from the PCB Library in the PCB document and NOT through the standard ECO process (update PCB from schematic)
- The mechanical user must point the SOLIDWORKS parts library folder to existing mechanical models (the location where they keep the mechanical parts)
**Note:** Electrical users must embed a 3D model in the footprint.

With this technique, the collaboration engine will redirect SOLIDWORKS to use the existing mechanical models and it will NOT generate duplicate part files. These components can be placed directly onto the PCB even before the schematic development. After locking down the printed circuit board-outline and parts positions, the electrical team can then add the symbols and establish component links manually between the existing footprints and their corresponding schematic symbols.
5) ECAD- MCAD Collaboration Consideration

a) PCB Assembly in SOLIDWORKS

The mechanical user should only pull a board from the SOLIDWORKS PCB Add-in to create the assembly for the first time in SOLIDWORKS mechanical. In turn, the board becomes linked to its corresponding PCB document in SOLIDWORKS PCB or Altium Designer (only applicable to Altium Designer users with a SOLIDWORKS PCB Connector purchased extension). After creating, saving, and closing the assembly, you should open the assembly through the standard File > Open (assembly) action and not through a pull action from the SOLIDWORKS PCB Add-in.

If the mechanical user pulls the board from SOLIDWORKS PCB Services while the original assembly already exists:

- This action will replace the existing assembly and will bring the assembly to the last pushed state from the electrical side.
- This action will break all of the predefined mates that mechanical users have defined in the original assembly.

Note: You should use the SOLIDWORKS PCB Add-in panel to view the changes, and to accept or reject changes after creating the assembly in SOLIDWORKS mechanical.

After creating an assembly from a pull action in SOLIDWORKS mechanical, the solid part file representing the PCB and all generated solid part models corresponding to footprints in the PCB are saved in the automatically created folder with the suffix *Electronic Parts. By default, this path is blank in SOLIDWORKS mechanical, which means that this *Electronic Parts folder is generated in the same directory as the assembly. You can access this path under the Settings options in the SOLIDWORKS PCB Connector panel. It is also possible to change this path to any folder on a network or PDM vault to redirect only all generated solid parts for the 3D models to that folder. All mechanical users can point to the same path to use existing generated 3D models inside that folder. Mechanical users can use existing models (see Section 4a – “Using Existing Mechanical Parts”).
b) Create Managed Projects

Users can create new managed projects in SOLIDWORKS PCB or SOLIDWORKS mechanical. If the user decides to create a managed project in SOLIDWORKS mechanical, then the electrical project pulled from SOLIDWORKS PCB becomes the PCB project (.PrjPcb). While, the assembly in SOLIDWORKS mechanical will be given its own name and part number defined by the organization.

c) Unique Pad Designators

All pad designators in the PCB must be unique. The SOLIDWORKS PCB Connector does not support duplicate pad designators. It is not possible to perform a push unless each pad has a unique designator, otherwise an error message will appear and display a listing all duplicate pads.
d) Electronics Parts with 3D Model

Any electronic parts placed in a PCB without a 3D model (extruded, or any supported files such as STEP, Parasolid, or .SLDPR) will not transfer to SOLIDWORKS mechanical through the connector. Only the holes of “thru holes” parts and the Decal for copper (for thru hole or surface mount components) will be collaborated to SOLIDWORKS mechanical. However, the will not include any part references in the FeatureManager tree.
e) **Footprint 3D Body Height**

If “extruded” 3D bodies are associated to a PCB footprint, each 3D body must have a height. Collaboration does not accept null or “0” (zero) heights. SOLIDWORKS mechanical will not interpret such heights correction, which will result in the appearance of an error message.

f) **Valid 3D Models for the Footprint**

The SOLIDWORKS mechanical (MCAD) team must verify any mechanical models that are associated to PCB footprints. Any 3D models should be verified in SOLIDWORKS mechanical to ensure that they are valid 3D solid models.

g) **Directory Name Character Limitation**

The Directory name must be fewer than 248 characters. If the file name or file path is too long, the following error message will appear.
h) Non-Electrical 3D Body Added in SOLIDWORKS

A non-electrical 3D body (like envelopes for keep-out areas) added in SOLIDWORKS cannot be moved or edited from SOLIDWORKS PCB. If the 3D body is moved or deleted, this change cannot be accepted in SOLIDWORKS.
i) Copper Collaboration Considerations

i) Plated through holes

Plated holes defined in SOLIDWORKS PCB or in Altium Designer will be collaborated through the connector to SOLIDWORKS mechanical as non-plated holes. Any plated or non-plated holes instantiate in SOLIDWORKS mechanical as non-plated holes because the software does not support copper barrels. (see section Error! Reference source not found. – “Error! Reference source not found.”).

ii) Top and Bottom, Solder, Overlay and Paste layers

By default, Top and Bottom, Solder, Overlay, and Paste layers are available as Decals in SOLIDWORKS mechanical when the mechanical user performs a pull on a managed project.
iii) Top and Bottom signal layers

By default, Top and Bottom signal layers will be collaborated through the connector to SOLIDWORKS mechanical as decals. This first requires performing a push from the PCB side. If the mechanical user decides to switch to Modeled Copper (from the Settings in the SOLIDWORKS PCB Add-In panel), they must first hide the decals for Top and Bottom layers in SOLIDWORKS before pulling the design. After accepting the changes, each signal layer from the PCB will be created in a separate subassembly and will then appear in the SOLIDWORKS FeatureManager tree.

Hide the Top and Bottom Copper Decals before activating Modeled Copper Collaboration.
Activate the **Modeled Copper Collaboration** option and then pull the changes from the SOLIDWORKS PCB Add-In panel.
Each copper layer from the PCB will appear as a subassembly in the FeatureManager tree.

iv) Plane layers

Plane layers will NOT be collaborated to SOLIDWORKS mechanical because plane layers in a PCB are defined as negative, which means that objects that can be seen on a plane layer are voids in the copper. The collaboration only supports positive layers.

v) Flipped components

When a component is repositioned or flipped in SOLIDWORKS mechanical, none of the decals (silkscreen, paste, solder, and copper) will update dynamically to reflect the change. For the mechanical user to see the updates of the decals, they must complete a full round trip of push and pull. For example, the mechanical user flipped a component and then pushes back the change to electrical domain. Now, after the electrical user accepts the change and pushes the design back, the mechanical user must accept the change to see the decals update.

vi) Holes as footprints

It is possible to create holes as a footprint, or as a free pad in SOLIDWORKS PCB or Altium Designer, and collaborated over to SOLIDWORKS mechanical through the connector. It is also possible to create holes in SOLIDWORKS mechanical by using the Hole Wizard. SOLIDWORKS PCB Connector 2019 SP0 collaborates holes created from the Hole Wizard.

SOLIDWORKS PCB Connector also collaborates sketches with multiple contours for holes to SOLIDWORKS PCB 2019 or Altium Designer 18.

You can also use the following tools to create patterns within the sketch by using the Linear Sketch Pattern, Circular Sketch Pattern, and Mirror Entities tools. However, SOLIDWORKS PCB Connector does NOT support Patterning and Mirroring features, etc. These features will not survive round trip collaboration.

However, special considerations must given when creating holes in SOLIDWORKS mechanical.

- When using the Hole Wizard to create a holes, mirror or pattern, and the electrical user decides to change the size of a hole, SOLIDWORKS mechanical will create a new sketch when the mechanical user accepts the change.

- If the mechanical user creates hole relations, and the electrical user decides to reposition the hole in the PCB, this action will break the relation in SOLIDWORKS mechanical when the mechanical user accepts the change.
6) ECAD- MCAD Collaboration Limitation

a) SOLIDWORKS PCB Assembly in Front Plane

Currently, when creating the board profile in SOLIDWORKS mechanical, the SOLIDWORKS PCB Connector supports only the Front plane. By default, when the board is pushed from SOLIDWORKS PCB or Altium Designer to SOLIDWORKS mechanical, the board profile created will reference the Front plane. There is also the option to create a New board assembly from SOLIDWORKS PCB Connector Add-In panel in SOLIDWORKS mechanical. This option will automatically create a default board outline that references the Front plane.

b) Plated Holes and Vias

SOLIDWORKS PCB Connector does NOT support Copper Barrels for Pad and Vias. Top and Bottom copper pads and pad holes will be collaborated to SOLIDWORKS mechanical but NOT the copper barrels for the pads. As for vias, ONLY the vias copper will be collaborated to SOLIDWORKS mechanical but NOT the via barrel neither the via hole.
c) **Rigid -Flex Circuits**

SOLIDWORKS PCB Connector does “NOT” support Flex circuits.

*Note:* It is NOT recommended to create or convert projects with flex circuits.

d) **Embedded Components**

SOLIDWORKS PCB Connector does NOT collaborate embedded components (this functionality is only supported in Altium Designer).

e) **Electronics Parts on PCB**

It is only possible to embed electronic parts in the footprint and transfer over from the electrical domain to SOLIDWORKS mechanical. Electronic parts must first be placed in the PCB, and then transferred (by push action from the PCB) to SOLIDWORKS mechanical.

*Note:* SOLIDWORKS PCB 2018 & SOLIDWORKS PCB 2019 Collaboration can transfer non-electrical models placed first in SOLIDWORKS mechanical to SOLIDWORKS PCB or Altium Designer (for Altium Designer users they must have a purchased SOLIDWORKS PCB Connector extension). These models can only represent heat sinks and the enclosure, but not electronic parts. You must first place electronic parts in the electrical domain.