SOLIDWORKS PCB Collaboration 2019 & SOLIDWORKS PCB Services
Best Practices

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1. Managed Projects

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.

For general collaboration compatibility, Altium Designer’s users need to check the table below before installing SOLIDWORKS PCB Connector extension, for compatibility between Altium Designer/Nexus, SOLIDWORKS PCB Connector and SOLIDWORKS Version.

<table>
<thead>
<tr>
<th>Altium Designer</th>
<th>SOLIDWORKS PCB Connector</th>
<th>SOLIDWORKS PCB Services</th>
<th>SOLIDWORKS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Altium Designer 17</td>
<td>SOLIDWORKS PCB Connector 2017</td>
<td>SOLIDWORKS PCB Services 2017, SOLIDWORKS PCB Service 2018. Recommended to have all on same version</td>
<td>SOLIDWORKS 2017</td>
</tr>
<tr>
<td>Altium Designer 18</td>
<td>SOLIDWORKS PCB Connector 2018</td>
<td>Only SOLIDWORKS PCB Services 2018</td>
<td>SOLIDWORKS 2018 and SOLIDWORKS 2017 SP5 with the hotfix 1041422 – Solution S-074212</td>
</tr>
<tr>
<td>SOLIDWORKS PCB Connector 2019</td>
<td>SOLIDWORKS PCB Services 2019, SOLIDWORKS PCB Services 2018. Recommended to have all on same version</td>
<td>SOLIDWORKS 2019, SOLIDWORKS 2018 - without copper or decals</td>
<td></td>
</tr>
<tr>
<td>Altium Nexus</td>
<td>SOLIDWORKS PCB Connector 2018 For Nexus</td>
<td>Altium Vault/Nexus Server</td>
<td>SOLIDWORKS 2018</td>
</tr>
<tr>
<td>SOLIDWORKS PCB Connector 2019 For Nexus</td>
<td>Altium Vault/Nexus Server</td>
<td>SOLIDWORKS 2019, SOLIDWORKS 2018 - without copper or decals</td>
<td></td>
</tr>
</tbody>
</table>
A Managed Project is the combination of meta information stored on the server, along with tight integration with the Vault's local SVN-based Version Control service. A user can create a new managed project via the New PCB Project dialog in SOLIDWORKS PCB or from the Projects browser-based interface in SOLIDWORKS PCB Services. During creation, the user defines the project's name, description, and design repository.

**Please Note:** For Altium Designer users only, a standard non-managed project can be converted to managed project for collaboration with SOLIDWORKS mechanical only with the purchased extension of SOLIDWORKS PCB Connector.

Managed Projects are version controlled by default, and can be collaboratively worked upon without having to worry about shared drives.

Prior to creating a new Managed Project, or converting an existing non-managed project, ensure that a design repository exists. Design repositories are centrally managed as part of the SOLIDWORKS PCB Services. A default repository will be created upon the installation of SOLIDWORKS PCB Services. Alternatively, you can connect to external repositories (see Section 1.2 below External Access to a Local SVN Design Repository)

Irrespective of how it is created, centralized management of your organization's design repositories ensures that all of your projects are stored exactly where they are supposed to be stored.
1.1. Creating a Repository through SOLIDWORKS PCB Services

Repositories can be created through the local Version Control service. Also, connection to external repositories can be set. 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 them (or roles in the case of repositories defined through the local Version Control service)
- Repository availability – Add or remove them centrally, rather than individual designers having to create and connect to repositories independently

A default Design Repository is available, provided by the local Version Control service and named “Default Repository”. While you cannot rename this repository, you can add a description for it, manage user access to it, or remove it, as required.

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

To add a Design Repository, click the button located at the top-right of the page. The Add Repository window will appear, use this to define the repository. The properties required depend on whether you are creating a new repository using the local Version Control service, or linking to an existing, external repository:

- **New** – Gives the repository a name and a description
- **Existing** - In addition to a name and description, the user needs to supply the URL to the repository, and credentials (User Name, Password) 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 A-Z, a-z, or 0-9. Underscores, dashes, and spaces are allowed 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 distinguished in the list of repositories by its External property being ticked (✓).

To edit the properties of a Design Repository at a later stage, simply click its associated Edit control (✓). To remove a Design Repository, simply click its associated Remove control (✗).
1.1.2. Sharing a Repository

To share a repository, simply click on the repository's associated Share control.

Having centralized your design repositories, you need to ensure that those requiring access to a given repository - including the designers who will be working on board designs - have that access. This is achieved by sharing that repository, or rather managing its access permissions.

The Manage Permissions window will appear, with all the controls necessary to share the repository with other users.

**Things to be aware of:**

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

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

Only local repositories (not external) can be shared with defined roles for the Vault, or publically. Roles are listed after individual users.

In terms of permissions, a user/role has Read/Write access when the Can Write option is enabled. If this option is disabled, they have Read access only.
To remove an existing user/role from having shared access to a repository, click the associated Remove control (x).

1.1.3. Sharing a Local Repository with a User
To share a local (not external) repository with another user:

Click the Add User control in the Manage Permissions window.

In the Add Users window that appears, start typing the full name, username, or email address of a user in the Select Users to be added field, to pop-up a list of matching users. Select the required user from this list. Multiple users can be chosen. To remove a user, simply click the delete cross, to the far right of their name. Set the permission for the user(s) using the Rights field to choose between Read access, or Write access or both.

Click the Ok button once ready, to effect addition of the user(s) and return to the Manage Permissions window.

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

### 1.1.4. 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 window and click Add.

2. In the Role Properties window that appears, start typing the name of a role in the Select Roles to be added field, to pop-up a list of matching roles. Select the required role from this list. Multiple roles can be chosen. To remove a role, simply click the delete cross, to the far right of its name.
Set the permission for the role(s) using the Rights field to choose between Read access, or Write access or both.

Click the Ok button once ready, to effect addition of the role(s) and return to the Manage Permissions window.

Click Close button in the Manage Permissions window.

To edit the permissions for an existing role, simply use the Edit option in the Manage Permissions window, to toggle between Read/Write access.

If the repository has been created through the local Version Control service, rather than an external repository, then you also have the ability to share with roles defined for SOLIDWORKS PCB Services’ Vault.
1.1.5. **Sharing a Local Repository Publically**

To share a local (not external) repository publically:

Click the Add Public control in the Manage Permissions window.

The Public entry will be added directly to the list of shared entities, and you can choose between Read/Write access. Click the Ok button in the Manage Permissions window.

Click Close button in the Manage Permissions window.
To edit the permissions for the Public entity, simply use the Edit option in the Manage Permissions window, to toggle between Read/Write access.

If the repository has been created through the local Version Control service, rather than an external repository, then you also have the ability to share that repository publicly - allowing all Vault users to access and use it.

1.1.6. Sharing an External Repository with a Vault User
To share an external repository with a Vault user:

Click the Add User control in the Manage Permissions window.

In the Add User window that appears, start typing the full name, username, or email address of a Vault user in the Select User to be added field, to pop-up a list of matching users. Select the
required user from this list. To remove the user, simply click the delete cross, to 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.

Click the OK button once ready, to effect addition of the user and return to the Manage Permissions window.

Typically, you might have the same user name (company domain user name) used for authentication when connecting to your Vault, and your company external SVN repository. This is why the SVN User field is filled with the same user name selected 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 - mapping the Vault user to the SVN user, as it were.
To do so, simply click the Edit control (📍) to the right of the SVN User entry - in the Manage Permissions window - and enter the required user name in the SVN User field, in the Edit User dialog. Don't forget to click the OK button to effect the change.

Click the Close button in the Manage Permissions window.

Access itself (Read-only, or Read/Write) is determined externally as part of the SVN repository installation. The Vault has no access to the external SVN repository's authz file, and so cannot manage these permissions from the Vault side.
1.2. **External Access to a Local SVN Design Repository**

A design repository that is created through the local Version Control Service can be accessed using an SVN client. Access is made using the regular network protocol (only svn:// is currently supported). The correct repository address can be copied from two places:

The VCS page of the Vault’s browser interface - within the Repository Path field for the target repository.

From within SOLIDWORKS PCB, from the Data Management - Design Repositories page of the Preferences dialog - within 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 (e.g. svn://127.0.0.1/<RepositoryName>, or svn://localhost/<RepositoryName>), or from a remote PC (e.g. svn://<ComputerName>/<RepositoryName>), where ComputerName is the name of the PC on which the Vault resides.
Use the acquired address in your Subversion client’s repository browsing facility. To access the repository, you will need to sign-in using your Vault login credentials (when a user is created in the Vault, the credentials are used by both the Vault itself, and the Version Control service - see Version Control Service Passwords for more information).
1.3. **Vault & VCS User Synchronization**

When a new user for the Vault is created, the defined credentials (User Name and Password) for that user are stored in both the Vault’s database and the Version Control service, since the latter cannot access the password from the former directly. The password is stored with the Version Control service in plain text format (in the \Repository\passwd file). If this presents a security issue, you can simply remove these password entries manually. Additionally, the ‘VcsSyncUserPasswords’ entry in the ‘localhost.ini’ file should be set to ‘0’ to prevent further password synchronization.

The repository has a folder for security containing files that need to be configured using a plain text editor.

authz:

[groups]
administrators = admin

[CentralRepo:]
admin = rw
@admin = rw
* = rw

passwd:

[users]
admin = admin

svnserv.conf:

[general]
anon-access = none
password-db = passwd
force-username-case=lower

Be sure to back them up first, then edit them. Add your other vault passwords in there like admin = admin.

* = rw in the authz file is quite important because it allows read/write access.

The svnserv.conf file in the \ProgramData\SOLIDWORKS\SOLIDWORKS PCB Services\Repository\conf folder has some entries with ../../ and this instructs the default svn server to read the authz and passwd files from the folder two higher, so you'll find the files in \ProgramData\SOLIDWORKS\SOLIDWORKS PCB Services\. You could look at those two files because they may have the correct settings for your test server. But the two files within the conf folder are not used, due to the ../../
1.4. Deleting a Repository

To delete a repository from the Vault's Version Control service, simply click the Remote 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 only removes that repository from being used by that instance of SOLIDWORKS PCB. It does not delete the repository from the Vault's Version Control service. If you sign out of the Vault and back in again, the repository will appear once more in the list of available repositories to that instance of SOLIDWORKS PCB.

External repositories can be deleted, irrespective of whether or not they contain any projects. Local repositories on the other hand (those created internally through the Vault's Version Control service) cannot be deleted if they contain one or more projects. You will be alerted to this, and should remove the projects first before being able to proceed with deletion.

Even then, if a local repository has no projects, it will not actually be deleted, but rather 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 (e.g. 775f6c22-b9a1-468c-9f1f-4f217bb3be6b_central design repository.dump).
1.5. HTTPS Protocol Support for “SOLIDWORKS PCB Services”

For those that require an additional layer of security for exchanging data with the SOLIDWORKS PCB Services, the server provides for secure communication over networks using the HTTPS protocol to protect the privacy and integrity of the data. This was introduced in the SOLIDWORKS PCB Services 2018 version.

The Port number used for secure connections is defined during the “SOLIDWORKS PCB Services” installation process, which uses default port (9785), alternatively you can use other port.

The configuration and server bindings for the “SOLIDWORKS PCB Services” can be accessed from the Windows Internet Information Services (IIS) Manager panel - available from the Administration Tools window (Control Panel\System and Security\Administrative Tools). Select the SOLIDWORKS PCB Services' entry in the panel's Connections list, and choose the Features View tab.
The available ports and server names for the “SOLIDWORKS PCB Services” are shown as selectable links in the Actions pane. Use the links to note and test the vault connection protocols, or access the current name/port bindings by selecting the Bindings link in the Edit Site section of the pane. Binding, including its protocol port number, can be edited from the Site Bindings dialog via the Edit button.

**Please Note:** Server name localhost only applies to the local machine, so PCs other than the one that is hosting the “SOLIDWORKS PCB Services” need to use the actual (server) name of that host machine to connect.

To confirm the name of the host PC on which the “SOLIDWORKS PCB Services” is installed and running, on that machine go to Control Panel\System and Security\System and note both the Computer name and Full computer name (its qualified domain name) - the latter will be that used by the “SOLIDWORKS PCB Services” for secure connections over the network (as also indicated in the Site Bindings dialog, above).
• From “SOLIDWORKS PCB” - simply use a HTTPS prefixed server address and the configured HTTPS port. By default, this is `https://localhost:9785` when using the PC that hosts the Vault or `https://<computername>:9785` for a remote connection (where `[computername]` is the host machine’s Computer Name or qualified domain name).

• From a Web Browser - enter the address of the local service (`https://localhost:9785`) or the host computer name equivalent (`https://<computername>:9785`).
2. **Vault Permissions**

All users must be added to the SOLIDWORKS PCB Services and given permissions (read/write access) to the desired Repository (See Section 1.1 Sharing a Local Repository with a User, Sharing a Local Repository with a Defined Role, Sharing a Local Repository Publically, Sharing an External Repository with a Vault User)

For Collaboration between SOLIDWORKS PCB or Altium Designer (Collaboration is only applicable to Altium Designer users with a purchased extension of SOLIDWORKS PCB Connector) 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.

We recommend to leave the permission to Anyone in my organization can change which is the default setting.

If needed, permission levels can be changed by logging to SOLIDWORKS PCB Services as an admin, under Vault page → Right click on the Mechatronics 3D Models folder and choose Share. Click on Change to access the Sharing Settings dialog.

```
Sharing Settings

Sharing Level - Private

Add Users | Add Roles

Sharing with Specific Users and Groups

<table>
<thead>
<tr>
<th>User Name</th>
<th>Can Edit</th>
<th>Description</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
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</tr>
</tbody>
</table>

Apply To Children

Ok  Cancel
```
**Please note:** Vault Permissions must be shared with Anyone in my organization if only using any SOLIDWORKS PCB Services releases prior to SOLIDWORKS PCB Services 2017 SPx3.

2.1. **Altium “Paid” Vault**

Altium Designer users who have an Altium paid Vault and want to use it to work in place of SOLIDWORKS PCB Services. Users of Altium Vault servers from 2.5.9 till 3.0.14 versions can be configured to work 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:

1. **The ‘Mechatronics 3D Models’ folder needs to be created and shared in Altium Vault**

   To create this folder:
   a. Launch SOLIDWORKS PCB/Altium Designer
   b. Sign in to the Altium Vault as a Vault admin user
   c. Open or create a project containing a PCB with 3D components
   d. In the ‘SOLIDWORKS PCB Connector’, click ‘Push’ > 'Post'

   The software will create the "Mechatronics 3D Models" folder in Altium Vault

   To share the folder:
   a. Open a web browser and access the SOLIDWORKS PCB Services by typing “[Machine Name]:9780”
   b. Sign in to the SOLIDWORKS PCB services as a Vault admin user
   c. Click the ‘Vault’ tab
   d. In the left pane, right-click on ‘Mechatronic 3D Models’ > ‘Share Folder’
e. Add the MCAD and ECAD users to the list and tick “Can Edit” to ensure it is shared properly
   Alternatively, you can add “Public” to share the folder with Public

2. **Client Access License (CAL) is required for MCAD person. If there is no such a license, Altium will create this special license upon request**
   Follow the instructions below to activate the CAL license from dashboard and set restrict the license usage once you receive the CAL license.

   In the Altium Vault web interface:
   a. In the ‘USERS’ > ‘ROLES’ section, add a new role (e.g. MCAD or Mechanical Engineers etc.)
   b. Assign the MCAD users to this Role
c. Go to 'ADMIN' > 'LICENSES' > 'ROLES' section and add a new license assignment (using the new Collaboration CAL) to this new Role

<table>
<thead>
<tr>
<th>Add Assignment</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Product Name</strong></td>
</tr>
<tr>
<td><strong>License</strong></td>
</tr>
<tr>
<td><strong>Assign to role</strong></td>
</tr>
<tr>
<td><strong>Options</strong></td>
</tr>
<tr>
<td>[ ] Automatic leasing</td>
</tr>
<tr>
<td>[x] All available seats</td>
</tr>
<tr>
<td><strong>Seats count</strong></td>
</tr>
<tr>
<td>[ ] Allow Roaming</td>
</tr>
<tr>
<td><strong>Notifications</strong></td>
</tr>
<tr>
<td>[ ] Notify when 90 % of available seats have been leased</td>
</tr>
<tr>
<td>[ ] Send to users with role</td>
</tr>
</tbody>
</table>

[Save] [Cancel]
3. ECAD-MCAD Collaboration

The Collaboration technology offered by SOLIDWORKS PCB Connector creates “new instances” of any models (Generic or Extruded) embedded in the footprint(s) in the PCB. When a model is first attached to the footprint a mesh body gets created, when a push is performed 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 take the mesh data and will convert it to a Parasolid and stores it in the mechatronics folder up 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 and “new instances” are created in Electronics Folder. By default, this folder is created in the same directory as the board assembly.

**Please Note: Downloaded models folder is created which consists of para-solid models.**

The path to the Electronics Folder can be changed by pointing to any other location in SOLIDWORKS PCB Add-in panel from SOLIDWORKS mechanical under the settings icon.

When these “new instances” get generated, they follow a certain naming convention. The part is made up of 3 different entities from 3 different fields, the first part is Footprint name from PCB Library (space will be replaced by “_”), the second part is “Symbol reference’ from Schematics library (space will be replaced by “_”), and the third one is the revision of the part in the PCB Services (shown in blue square). Each of these will be separated by “_” to make a full 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 become available in the feature 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 and has a model that was already 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. It will NOT create a new instance. The new part will be listed in the feature-tree.

3.1 Using Existing Mechanical Parts

Typically, solid part models reside in a PDM Vault or a specific location on a shared network. 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)
**Please Note: Electrical user(s) 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 parts files. Now, these components can be placed directly onto the PCB even before the schematic development. Once the printed circuit board-outline and parts positions are locked down, then the electrical team can add the symbols and establish component links manually between the existing footprints and their corresponding schematic symbols.
4. ECAD-MCAD Collaboration Considerations

4.1 The mechanical user should only pull board from SOLIDWORKS PCB Add-in to create the assembly for **the first time** in SOLIDWORKS mechanical which will get linked to its correspondent PCB document in SOLIDWORKS PCB or Altium Designer (only applicable to Altium Designer users with a SOLIDWORKS PCB Connector purchased extension). Once the assembly is created and if it is saved and closed, it should be **opened** through the standard File → Open (assembly) action 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 existed:
- 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 the predefined mates that the mechanical user(s) has defined in the original assembly.

**Please Note:** The SOLIDWORKS PCB Add-in panel should be used to view the changes (accept/reject) changes once the assembly is created in SOLIDWORKS mechanical

4.2 Once the assembly is created 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 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. This path can be accessed under the Settings options in the SOLIDWORKS PCB Connector panel. This path can also be changed to any folder on a network or PDM Vault to redirect only all generated solid part 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 3.1 – Using Existing Mechanical Parts)
4.3 Users can create new managed projects in SOLIDWORKS PCB or SOLIDWORKS mechanical. If the user decided 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.

4.4 All Pad designators must be unique in the PCB. SOLIDWORKS PCB Connector doesn’t support duplicate pad designators. A push can’t be performed unless each pad had a unique designator, otherwise an error message will display listing all duplicated pads.

4.5 Any electronic parts placed in the PCB without a 3D model (extruded, or any supported files: i.e. STEP, Parasolid, .SLDPRT) will not transfer over to SOLIDWORKS mechanical through the connector. Only the holes of thru holes’ parts and the Decal for copper (for thru hole or the surface mount components) will be collaborated to SOLIDWORKS mechanical but still without any part’s references in the feature tree.
4.6 If “Extruded” 3D body are associated to a PCB footprint, each 3D body must have a height. 0 and no heights are not acceptable through collaboration and these will not be interpreted correctly in SOLIDWORKS mechanical and will cause an error message.

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

4.8 Directory name needed to be under 248 characters, otherwise the user will receive this error message if specified Path, File name, or both too long.
5. **Copper Collaboration Considerations**

5.1 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 are instantiated in SOLIDWORKS mechanical as Non-Plated Holes since copper barrels are not supported. (See Section 6 – ECAD-MCAD Collaboration Limitations)

5.2 By default, Top & Bottom, Solder, Overlay and Paste layers are available as Decals in SOLIDWORKS mechanical when the mechanical user performs a pull on a managed project.
5.3 By default, Top & Bottom signal layers will be collaborated through the connector to SOLIDWORKS mechanical as decals (a push from the PCB side must first be performed). If the mechanical user decides to switch to Modeled Copper (from Settings in the SOLIDWORKS PCB Add-In panel), they must first hide the decals for Top & Bottom layers in SOLIDWORKS before they pull the design. Once they accept the changes, each signal layer from the PCB will be created in a separate subassembly and listed in the SOLIDWORKS feature tree.

*Hide the Top & Bottom Copper Decals before enabling Modeled Copper*

Enable Modeled Copper then pull the changes from the SOLIDWORKS PCB Add-In panel
Each copper layer from the PCB will be listed as a subassembly in the feature tree.

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

5.5 When a component is repositioned or flipped in SOLIDWORKS mechanical, none of the decals (e.g. silkscreen, paste, solder and copper) will update dynamically to reflect the change. For the mechanical user to see the updates of the decals a full round trip of push and pull must be completed. For example, the mechanical user flipped a component, he 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.

5.6 Holes can be created as a footprint, or a free pad in SOLIDWORKS PCB or Altium Designer and collaborated over to SOLIDWORKS mechanical through the connector. Also holes can be created in SOLIDWORKS mechanical from 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 create patterns within the sketch, using the following tools: Linear Sketch Pattern, Circular Sketch Pattern, Mirror Entities. However, SOLIDWORKS PCB Connector does “NOT” support Patterning/Mirroring features, etc. These features will “NOT” survive round trip collaboration.
However, special considerations must be taken when creating Holes in SOLIDWORKS mechanical.

- If holes are created using the Hole Wizard, mirror or pattern and the electrical user decides to change the size of a hole, a new sketch will be created in SOLIDWORKS mechanical when the mechanical user accepts the change
- If the mechanical user creates holes’ 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

Holes created in SOLIDWORKS PCB or Altium Designer as free pads will get created in SOLIDWORKS’ mechanical feature tree as a feature. Same diameter holes will be combined in the same feature’s sketch. But different hole diameter will get created in a separate feature.
6. **ECAD-MCAD Collaboration Limitations**

6.1 When creating the board profile in SOLIDWORKS mechanical, currently the Front plane is the only plane supported through SOLIDWORKS PCB Connector. By default, when the board is pushed from SOLIDWORKS PCB or Altium Designer to SOLIDWORKS mechanical, the board profile will be created referencing the Front plane. There is also the option to create a New board assembly from SOLIDWORKS PCB connector Add-In panel in SOLIDWORKS mechanical which will automatically create a default board outline referencing the Front plane.

6.2 SOLIDWORKS PCB Connector does “NOT” support Copper Barrels for Pad and Vias. Top & 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.

6.3 SOLIDWORKS PCB Connector does “NOT” support Flex circuits.

**Please Note:** It is “NOT” recommended to create or convert projects with flex circuits.
6.4 SOLIDWORKS PCB Connector does “NOT” collaborate Embedded components (this functionality is only supported in Altium Designer).

6.5 SOLIDWORKS PCB Connector does “NOT” collaborate Multi-board (this functionality is only supported in Altium Designer).

6.6 Electronic parts can only be embedded in the footprint and transferred over from the electrical domain to SOLIDWORKS mechanical. Electronic parts must be placed first in the PCB then transferred (push action from the PCB) over to SOLIDWORKS mechanical.

**Please 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 be representing heat sinks, enclosure but not electronic parts. Electronic parts must be first placed in the electrical domain.**
7. **ECAD-MCAD Known Problems & Solutions**

These problems are related to SOLIDWORKS PCB Connector 2019 sp0

7.1 The Solder layers’ Decals are empty therefore the Solder will “NOT” show up in SOLIDWORKS mechanical.

7.1.1 Since the Solder Decal is empty, tented vias and tented pads are “NOT” displayed in SOLIDWORKS mechanical.

![Tented Via vs Non-Tented Via](image)

7.2 Users must be connected to the internet “on-line” to connect to SOLIDWORKS PCB Services to be able to perform a push from SOLIDWORKS PCB or from Altium Designer.

**Solution:** Use a hot spot.

7.3 Mounting holes disappear if the mechanical user aligns them to the center of a standoff, a mate is automatically added but when you go back to SOLIDWORKS PCB or to Altium Designer, the holes do not move to the position established in SOLIDWORKS mechanical.

**Solution:** Delete the mate in SOLIDWORKS mechanical.
7.4 Graphics Card limitation for SOLIDWORKS mechanical 2019 SP0. If the mechanical user has an Nvidia graphic card “Maxwell or Pascal series”, SOLIDWORKS mechanical CAD will crash when pulling a PCB with components and/or copper and after performing a zoom or rotation actions. Since Decals are always on by default, unless the PCB has no components and/or copper, SOLIDWORKS mechanical CAD will crash.

Solution: To avoid the crash when pulling the design in SOLIDWORKS mechanical, just collaborate the PCB without components and/or copper. This issue is resolved in SOLIDWORKS 2019 SP1.