

Tips and Suggestions for Working from Home Using SOLIDWORKS® PDM

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3DEXPERIENCE®

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Revision History

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1) Preface

During these extraordinary times, many of our valued SOLIDWORKS PDM customers are working from home for an extended duration due to COVID-19 restrictions. You may have worked from home occasionally in the past, or this might be your first work from home experience.

Your use of SOLIDWORKS PDM now relies on the network connection from your home to your company's SOLIDWORKS PDM vault. Your connection is likely substantially different from the speed and stability that you are accustomed to when working from your company's office.

You might also be using a personal home computer or possibly a borrowed company computer that is not as optimally configured for SOLIDWORKS 3D CAD as the professional grade workstation you use when working from your company's office.

These factors may present challenges, but the SOLIDWORKS PDM Technical Support team offers some useful tips and suggestions to help you to maintain efficiency with your data management experience while working from home.

2) Optimal PDM Server and PC Client Connectivity

- For optimum performance, use a remote desktop application (such as GoToMyPC® or Windows® Remote Desktop) to connect remotely to your workstation in the office. Make sure the workstation in the office is powered on, has an Ethernet connection and that the Windows power settings configuration allows prompt connections.
- Upgrade to SOLIDWORKS PDM 2020. In the 2020 version, PDM File Explorer and the SOLIDWORKS PDM Add-in make the day-to-day experience more efficient when accessing files and folders. Data now loads asynchronously, which means that you do not have to wait for the file contents to be read from the database before you can work with files, browse folders, or select the **BOM**, **Contains**, and **Where Used** tabs. This delivers a faster experience overall. See '[What's New](#)' in SOLIDWORKS PDM for more details.
- When working from home over a Virtual Private Network (VPN):
 - Add the IP addresses of SQL and archive servers to the PDM client *HOSTS* file. See Knowledge Base solution [S-037242](#) for details.
 - If experiencing a failed login or connection drop with a PDM vault, use the *Connectivity test tool* to verify that the PDM client can reach PDM servers. The tool is available in Knowledge Base solution [S-069274](#).
 - If you have trouble attaching to the archive server over a VPN when creating a local file vault view, use a scripted view setup file. See Knowledge Base solution [S-045521](#) for details.

3) SOLIDWORKS PDM Add-in

When working in high latency (remote) environments with large assemblies, some vaults may experience overhead while the PDM add-in talks to the server and refreshes the reference tree and column information. The following recommendations reduce SQL traffic and events that automatically refresh the reference tree, leading to a more responsive session in SOLIDWORKS.

- Disable the following options under **Tools > SOLIDWORKS PDM > Options > Automatically update the tree in the extra tree pane.**
 - **After a component's window has been activated.**
 - **After a component has been added, deleted, or renamed.**
 - **After a component has been reloaded.**
 - **After a component has been suppressed or un-suppressed.**
 - **After the get, check out and check in operations.**

- Remove custom display information and columns, including **Description**.

Note: To see the PDM custom columns, you must log in to a PDM vault view.

1. Open the SOLIDWORKS PDM **Options** dialog box via **Tools > SOLIDWORKS PDM > Options**.
 2. Select the **View Setting** tab.
 3. Under **Information to display**: Select **Description** from the list.
 4. Clear the check boxes for the following options:
 - **Display as a node in the tree**
 - **Display in preview**
 - **Display in a column of its own**
 - **Display as part of the file name Column**
 5. Repeat to remove additional information as needed.
- Disable the **Prompt to check out file if opened in read-only mode** option. Disabling this option dismisses checks and prompts when editing read-only files. Therefore, to avoid losing changes in memory, always check out the files you are editing.
 - Disable the **Show File data card when new file is saved** option. Then review and update the file data card in SOLIDWORKS PDM File Explorer when the file is not open in SOLIDWORKS.
 - When handling large assemblies, consider planning operations that require high SQL traffic and data transfer to be done outside working hours or during lunch breaks. For example, the check in or change state for a large file set could take longer when working with high latency.

- In more extreme cases, disable the SOLIDWORKS PDM add-in and use SOLIDWORKS PDM File Explorer to perform file operations such as check in, change state, or check out. Remember to check out the file before opening it in SOLIDWORKS and to close the files before checking in.

Note:

- Recommendations such as disable custom columns can be applied to multiple computers by exporting a registry key that includes the desired settings. This registry key can be applied with a group policy or shared with individual PDM clients. For more information about which SOLIDWORKS PDM settings are controlled by registries, see the Knowledge Base solution [S-057238](#).

4) SOLIDWORKS PDM File Explorer

- Use the **Version** tab selections instead of the **Preview** tab to avoid caching files when browsing the vault view.
- Consider disabling custom columns in SOLIDWORKS PDM File Explorer and Search results by using the default columns.
- Remove extra custom columns in reference dialog boxes such as **Where used**, **Contains**, **Check out**, **Get Version**, **Check in**, and **Change State**.
- Remove **Card Preview** under the **Display** menu. This reduces some database traffic when previewing already cached files.
- Consider removing commands that may create overhead from the action bar and right-click context menus. For example, the change state command may cause additional overhead to calculate permissions when you right-click a file.
- Disable the Use automatic login for this vault option in the **SOLIDWORKS PDM Administrations > Settings > Automatic Login** tab.
- When connecting via VPN, use the **Exit from PDM** tray icon to unload SOLIDWORKS PDM and log out from file vaults before you disconnect from VPN or shut down the client.

5) SOLIDWORKS PDM Administration

- Avoid making changes to vault and user or group settings over a VPN connection. Use instead a remote desktop application via a PC client locally to the SQL Server.
- In the **Cache Options**, disable the **Refresh cache during login** and **Clear cache during logout** options. These settings slow down the login and logout operations.
- Disable the option **Include drawings automatically when checking in the model** option, which makes the check-in command slower. This option is under **SOLIDWORKS PDM Administration** right-click **Users/Groups > Settings > Check in**.

6) SOLIDWORKS PDM Task Host

- Do not use VPN clients as a task host. Configure a dedicated system as a task host on the Local Area Network (LAN). For more recommendations about best practices for task hosts, see the Knowledge Base solution [S-054854](#).

7) SOLIDWORKS PDM Archive Server

- Attach to the closest physical archive server where files are replicated.
- Set the file vault view listings refresh interval to **100** seconds. You can do this from **Archive Server > Tools > Default Settings > Refresh Interval**. This reduces the database traffic for server and client applications.
- Increase the time check messages in the background to **120** seconds. For more details, see the Knowledge Base solution [S-072959](#).

8) Working in Offline Mode

- If your VPN connection is slow or unstable, consider working in *offline mode*. Offline mode allows users to work on files in the local cache when not connected to a SOLIDWORKS PDM vault (PDM Database, Archive and Network License Server). In offline mode, only files in the local cache are accessible and only files currently checked out are writeable.
- The general work offline process is to:
 1. Cache the required files.
 2. Check out files that will be edited.
 3. Set work offline mode.
 4. Close the PDM server connection and exit using the SOLIDWORKS PDM tray icon.
 5. Perform work on the files.
 6. When the files are ready to check in, connect to SOLIDWORKS PDM.
 7. Check in the files.

When working offline, it is recommended to unload the SOLIDWORKS PDM Add-in. For additional information about working offline, see the help documentation in chapter [Working Offline](#) and the Knowledge Base solution [S-016224](#).

- Before switching to offline mode, ensure that the following data is cached: working project files, and the Toolbox and Design Library if managed inside the PDM vault.
- While working in offline mode, files are saved in the local view folder and should never be renamed or moved until returning to on-line mode.

- Ensure that the **Show all files** is used under **SOLIDWORKS PDM Administration** > right-click **Users/Groups** > **Settings** > **Explorer**. This setting allows viewing files in the vault and local files or folders that currently exist only in the local vault view.
- When switching to *online* mode, files saved to the local view during offline mode appear as **<local file>** in the state column, and local folders display an inactive folder icon. Local files and folders must be added to the vault via the right-click menu **Add to File Vault** or **Search Local Files** features.
- Use VPN only when you need online access to the vault.

When working offline, consider creating a registry shortcut that can be used before connecting to a vault to login faster. For more details see the Knowledge Base solution [S-048164](#).