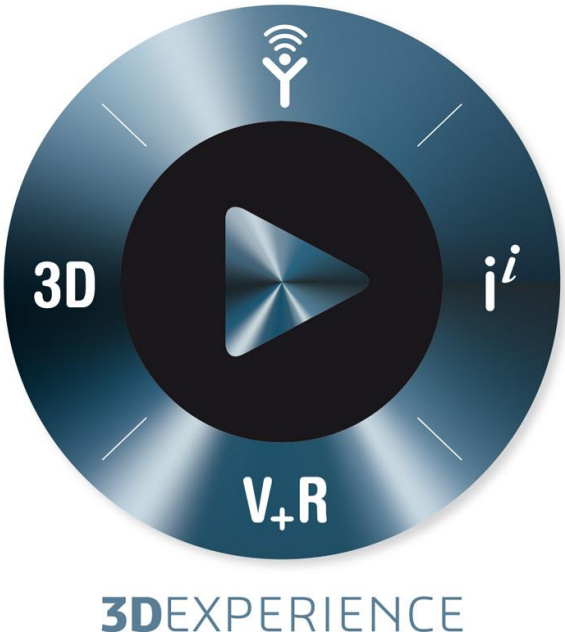
Tips and Suggestions for Working from Home Using SOLIDWORKS® 3D CAD

Last Modified: April 2020

Revision 1.0



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Revision History

Rev #	Date	Description
1.0	*April 2020*	*Document created (SOLIDWORKS 2020)*

1) Preface

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

You may be using a personal home computer or a borrowed company computer that is not as optimally configured as the professional grade workstation you use when working in your company's office. The network connection from your home to your company's data storage locations is likely substantially different from the speed and stability that you are accustomed to when working from your company's office.

These factors may present challenges when working with "large" assemblies and their associated drawings, but the SOLIDWORKS Technical Support team offers some useful tips and suggestions to help you to maintain efficiency with your design process while you are working from home.

2) Licensing

SOLIDWORKS offers current SOLIDWORKS 3D CAD users a host of solutions that enable you to work from outside of your office. For more information, see this post in the SOLIDWORKS blog.

3) Work Locally

Generally, it takes longer to open and save files over a Local Area Network (LAN) than opening or saving files locally to your computer. Additionally, the file open and save time over a Virtual Private Network (VPN) solution may be slower than opening or saving your files over your office LAN.

Refer to the SOLIDWORKS Knowledge Base solution S-051256 for more detailed information about the limitations of working with SOLIDWORKS files over the network.

Even if you maintain SOLIDWORKS files on a central server, copy the files locally, make your changes, and then copy the files back to the server. This is what many PDM systems do, including the SOLIDWORKS PDM software. For tips and suggestions when working from home using SOLIDWORKS PDM, see the corresponding document in the attachments of SOLIDWORKS Knowledge Base solution S-077757.

4) Be Strategic in your Workday Routine

For many, this is a new way of working. Streamline your daily tasks to get the best from your workday.

Consider the following questions:

1. What files do I really need to work with now or later?
    a. Do I need the entire set of files, or do I just need to download a subset of them?
    b. Can I anticipate downloading the files locally for a future job (especially during downtime or break times) without affecting or delaying someone else?
2. Can I reduce my (local) download and upload of files?
    a. How many times do I really need to download my files and upload them again to a server?
3. Is there a benefit to move (anticipate or postpone) a design task?
    a. Does separating or combining my daily design tasks benefit my productivity?
    b. How does this affect my team?

Reassessing your needs and making changes to your daily tasks to align with your environment can make a difference in your productivity.

5) Be Strategic in your Design Process

a) Reduce the need to update your models multiple times to increase efficiency

This is particularly valid when working with large assemblies and large drawings. Whether you make one change or 30 changes to the model, the assemblies and drawings that reference your model will automatically update to include your changes.

Think about opportunities to make multiple changes to part documents and then update the referenced higher-level assemblies or drawing views manually to improve your design time.

Consider enabling **Tools** > **Options** > **System Options** > **Assemblies** > **Suspend Automatic Rebuild** option with the **Tools** > **Large Assembly Settings** function enabled to defer full assembly level updates when making changes within the context of the assembly. This is particularly useful with assemblies that contain a large number of mates, assembly features,

or many in-context relationships. This allows you make multiple changes and then perform a complete assembly update when you are ready for the changes.

Also consider enabling the **Tools** > **Option**s > **System Options** > **Assemblies** > **Do not rebuild when switching to assembly window** option. This is useful when working with multiple parts and assemblies in separate windows, and you spend a lot of time toggling between the document windows while making changes. This option bypasses the automatic full assembly rebuild when switching between referenced documents after making design changes.

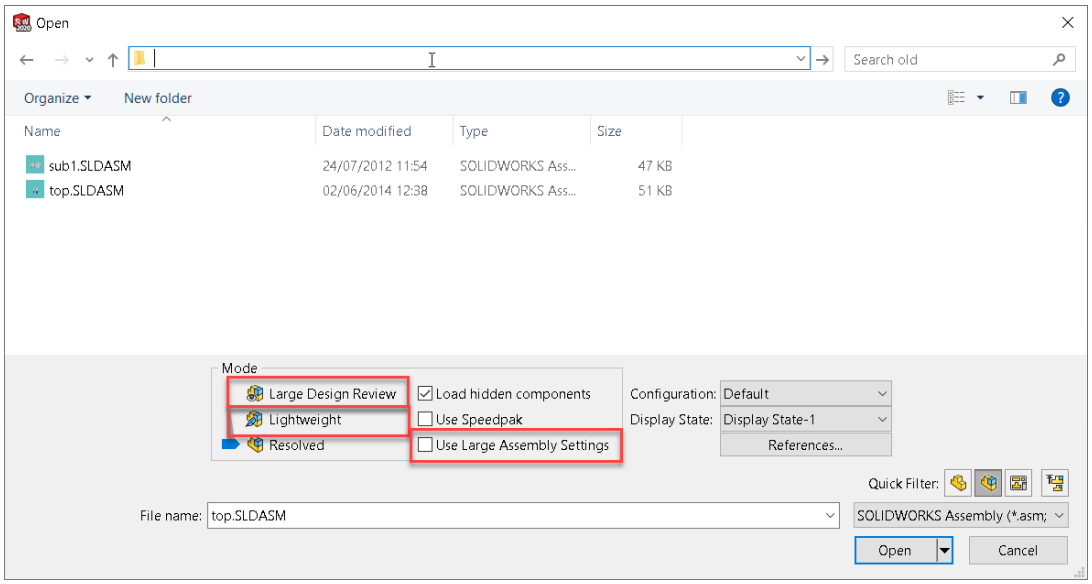
While you work with SOLIDWORKS drawings, consider disabling the following options under **Tools** > **Options** > **System Options** > **Performance** to provide you with more control and efficiency:

- **Show contents while dragging drawing view**
- **Allow auto-update when opening drawings**
- **Save tessellated data for drawings with shaded and draft quality views**
    - This may shorten the save time and decrease downstream operations because it reduces the resulting file size.
    - **CAUTION**: The resulting file may be inaccurate with viewers that rely on display data such as eDrawings®.

Also, consider right-clicking the drawing within the FeatureManager® Design Tree and disabling the **Automatic view update** feature. This behaves similarly to the **Allow auto-update when opening drawings** feature, except this applies to changes that affect the drawing when the drawing is already open.

This gives you the flexibility to update any or all of the views at your convenience while actively making changes to your design data.

b) Work in Lightweight, Large Assembly Mode, Large Design Review Mode, or Detailing Mode



Lightweight mode loads a subset of data from assembly files. This data includes the 3D body data, mates, and reference geometry. For the vast majority of assemblies, Lightweight is the best choice of assembly modes, limiting the initial amount of data in transit and in memory when opening and beginning to work within a larger assembly or drawing. In Lightweight mode, you can add mates, create assembly features, calculate interference detection, add annotations, evaluate measures, and more.

Large Assembly mode loads assemblies as lightweight and activates additional SOLIDWORKS system options that improve performance.

Large Design Review mode loads only the display data, which is even lighter than lightweight mode. However, you can still obtain rough measurements, isolate necessary components, add and edit mates, insert components, and create section views and camera walk-throughs. If this is enough for your present task, it may really increase your design efficiency. This is also useful for navigating and visualizing the structure of an assembly in a lightweight mode without the overhead of loading all of the feature data.

Generally, use *Resolved mode* for a new or small assembly, or if you need to edit most or all its components.

Similar to Large Design Review, *Detailing mode* in drawings allows you to open large drawings quickly, without loading the model data. Detailing mode is particularly useful when it is necessary to make minor changes to the drawings, such as adding new dimensions or new annotations that do not require model information. This can drastically reduce opening and saving time.

For more information about the assembly modes described here, refer to the SOLIDWORKS Help and to the Large Assemblies Training in MySolidWorks.

c) Decrease the graphics overhead

The graphical data of your CAD files often influence the time it takes to rebuild and update models. The graphics data also influences open and save times.

Consider applying the following settings to reduce the time it takes to "paint" the display during updates and to improve the dynamic manipulation of your models (zoom, pan, and rotate):

- Move the image quality slider (**Tools** > **Options** > **Document Properties** > **Image Quality**) to the **Low (faster)** position for the display style that correlates to your model.
- Turn off **HLR Edges** in the **Shaded** display mode.
- Turn off **RealView** mode.
- Limit the use of transparency.
- Hide items from the **View** menu that you do not need.
- Move the **Level of detail** slider (**Tools** > **Options** > **System Options** > **Performance**) to the **Less (faster)** position.
- Disable the **Verification on rebuild (enable advanced body checking)** option (**Tools** > **Options** > **System Options** > **Performance**).

Disable the following options:

- Under **Tools** > **Options** > **System Options** > **Display/Selection**:
  - **Highlight all edges of features selected in graphics view**
  - **Dynamic highlight from graphics view**
  - **Anti-aliasing**
- Under **Tools** > **Options** > **System Options** > **FeatureManager**:
  - **Dynamic highlight**

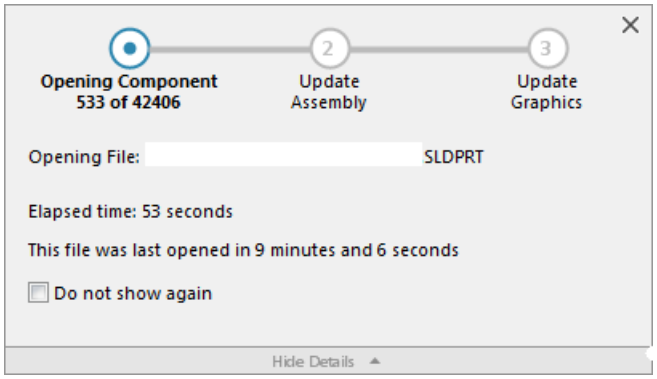
Many of these options are set automatically when you enable Large Assembly mode.

For more information about these settings and options, refer to the SOLIDWORKS Help and to the Large Assemblies Training in MySolidWorks.

6) Use the Assembly Open Progress indicator and Performance Evaluation Information

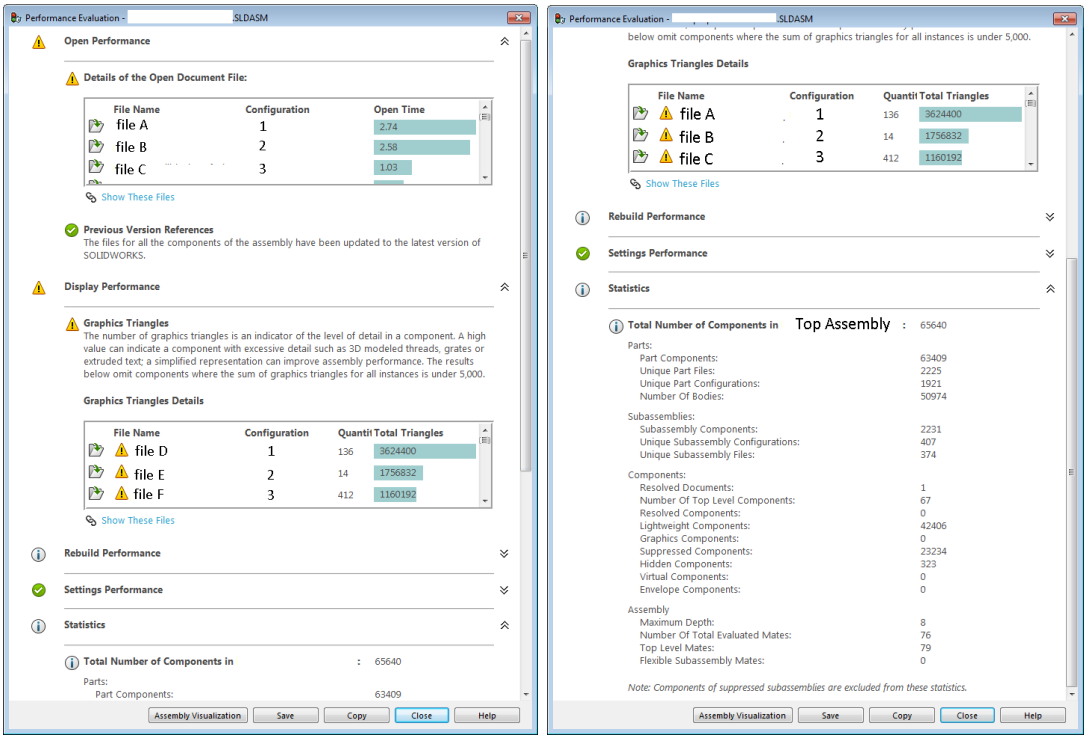
You can use this information to obtain detailed information about a large assembly or large drawing.

a) Assembly Open Progress indicator



The Assembly Open Progress indicator provides details about the status of the three main operations that run while you open an assembly or drawing. These are opening (loading) the components, updating the assembly/drawing, and finally updating and repainting the graphics. Determining which of the operations take more time while opening your assembly or drawing could indicate a potential area of your design, where to port modifications for performance improvements.

b) Performance Evaluation



The **Performance Evaluation** displays information about how your assembly or drawing is performing. This analysis includes open performance, rebuild performance, and display performance of the models in your assembly. The evaluation also displays the settings performance, highlighting particular options to improve the file performance. Furthermore, it provides statistics about the referenced parts and subassemblies within the top-level assembly.

Likewise, you can use the Performance Evaluation to point out areas of your design or of your SOLIDWORKS settings to change to improve performance.

7) Considerations for Working Directly Over the Network

Many options are taken for granted when working within a production environment with access to high-speed internet and optimal hardware resources. However, working from home can expose bottlenecks that may not be clearly visible in the optimal production environment.

Here is a list of common SOLIDWORKS functions/commands/options that may have the potential to influence your experience in SOLIDWORKS if you need to work with SOLIDWORKS data through your company network from your home.

a) File > Open

i. Assemblies and drawings file structure

SOLIDWORKS uses a file reference structure. When you open an assembly or drawing resolved, you not only open the drawing or assembly document, but you also open the referenced files in memory. This is why you should expect a greater impact in SOLIDWORKS when working over the network as compared to working in Microsoft Word when you are working with assemblies and drawings. As you can imagine, if you open these files over VPN from your home, you can expect to experience slower performance than you are used to seeing where you work. Using the strategies mentioned in the previous section can help minimize the amount of data transferred over the network. This helps to improve the open time if you need to open assemblies and drawings over the network using a VPN solution.

ii. Legacy files versus converted files

When opening older version files in a new version of the software, it may be necessary to update the model data to benefit from the latest software enhancements. This can increase opening times. After converting the files, this step is no longer necessary.

To ensure that files open faster, convert the files to the latest version of SOLIDWORKS in use. Use SOLIDWORKS Task Scheduler to convert files in bulk.

iii. Tools > Options > System Options > External References > Referenced Documents specified in File Location > Include sub-folders

This option can significantly affect the time it takes to open an assembly document if the folder is a network location with many subfolders, and the assembly document cannot locate referenced documents in their last saved location. The impact could be greater at home if accessing your work network over VPN.

iv. Image quality

At higher quality settings, the display (tessellation) data could also generate low performance. A SOLIDWORKS part file, assembly file, or drawing file that contains shaded data using the **Tools** > **Options** > **Document Properties** > **Image Quality** > **Shaded and draft quality HLR/HLV resolution** slider set at a high level can cause delays during the open procedure. This occurs because SOLIDWORKS is attempting to display the preview in the graphics area before loading the files for users working over VPN at slower internet connection speeds. This can also affect downstream operations that rely on viewing the display data.

b) File > Save

i. Assemblies and drawings file structure

When you save assemblies and drawings outside of a Product Data Management (PDM) environment, you may also be saving other reference documents that have pending modifications. If the referenced files are large file sizes, you may see delays as the file is saved over the network.

This can be magnified when working over the network, especially over VPN.

ii. Image quality

The **Tools** > **Options** > **Document Properties** > **Image Quality** > **Shaded and draft quality HLR/HLV resolution** option and subset options control the quality of tessellation of curved surfaces in shaded displays.

The higher the value, the larger the file becomes after saving. This can affect the time it takes to save the file over the network (if the file is stored in a folder in a network location), or to copy and paste the file from a local folder to the server (if working mostly locally but sharing files with colleagues in a central office network location).

c) SOLIDWORKS Toolbox

If you need to access a shared network Toolbox from home over VPN, you may experience longer delays when editing or creating hole wizard holes or inserting toolbox fasteners than what you would see in your production environment. Most of the delays occur when SOLIDWORKS makes calls to the *SQLite database* over the network that contains the metadata of your Toolbox.

Leave the Task Pane pinned and the Toolbox node expanded while you browse and insert fasteners while working in SOLIDWORKS. This reduces the number of calls to the SQLite database while browsing and inserting fasteners into your assemblies.

You may also experience delays when working with other features that rely on the SLQLite database:

- Editing or creating **Hole Wizard** features
- Editing or creating **Advanced Hole** features
- Editing or creating **Thread** features

If it is not a strict requirement to maintain a centralized Toolbox folder in the office. It may be considered to copy the Toolbox folder locally, for each SOLIDWORKS user, and specify that location in the **Tools** > **Options** > **System Options** > **Hole Wizard/Toolbox** > **Hole**

Wizard and Toolbox Folder path. Doing so can reduce the network communication and improve working performance.

d) Launching SOLIDWORKS

If you are using SolidNetWork Licensing (SNL), you may experience delays launching SOLIDWORKS or while attempting to enable add-ins in your home environment.

Possible reasons for delays include:

- Attempting to communicate with the license server over VPN.
- Having multiple license servers in your server list. If multiple license servers are present in the **Server List** tab of your SNL software client, and some of those servers are no longer in service, remove the servers that are no longer in service. This helps to improve the performance.
- The network going in timeout while attempting to communicate with a server in your server list.