



**Work Smarter, Not Harder:  
Your Quick Fix Guide to Managing  
Large Assemblies in SOLIDWORKS**



A compilation of best practices that'll improve  
your large assembly load time and performance

# Introduction

## About the Author



### **Elise Moss, ME and CAD Instructor**

With more than 20 years of engineering design experience, Moss is both a practicing mechanical engineer and a CAD instructor. She holds 12 certifications in both SOLIDWORKS and Autodesk and has published more than 100 CAD design textbooks. Moss is currently working as a contractor at Google and she's the sole proprietor for Moss Designs, an engineering services consulting business. Moss is also currently a CAD instructor at Laney College. She has worked in the automotive, nuclear, petroleum, medical, data center, and consumer electronic industries.

## Improve Your Large Assembly Performance

Working with large assemblies in SOLIDWORKS can be challenging. The files are often slow to load and slow to react to even minor zooms and rotations. Because of these inefficiencies, it can be very frustrating for an engineer to work with large assemblies.

Common challenges engineers experience when working with a large assembly:

- Opening the file takes too long
- Regeneration when the view changes slowly
- Rotating the model is slow and choppy
- SOLIDWORKS software crashes
- Notified that system resources are running low

However, there are several ways you can improve the performance of large assemblies by fixing errors and updating system settings and configurations. In this e-guide, we'll explain steps you can take to make working with large assemblies faster and easier.

# Table of Contents

No-Fuss Fixes for Improving Large Assembly Performance.....	1-2
How to Organize Your Assembly.....	2-11
How to Troubleshoot Large Assemblies.....	12
How to Effectively Open Large Assemblies .....	12-13
Upgrade Your Hardware.....	13
How to Manage Your Resources.....	13-15
About SolidProfessor.....	16

# No-Fuss Fixes for Improving Large Assembly Performance

## Check the “Don’t show again” box

Check the **Don’t show again** box found at the bottom of most dialogs. This speeds up your throughput because it eliminates one more dialog box you need to advance through to get to what you want to do. See Fig. 1.

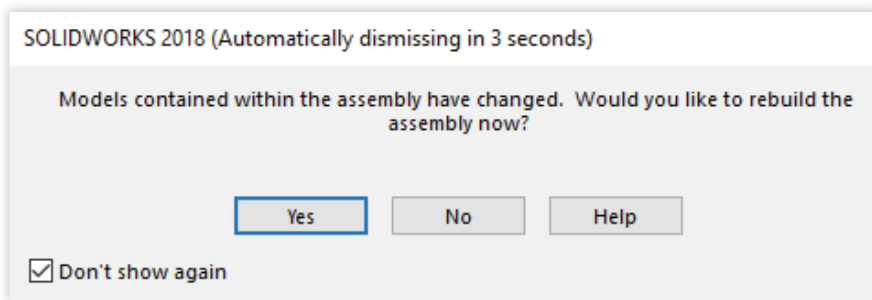


Fig. 1

## Update Your SOLIDWORKS Software

Update any components that are an older version than the release you’re working in. If you’re using older version components, SOLIDWORKS will constantly prompt you to update and monitor them, which slows down just about everything. If you don’t want to save to a newer release because a vendor or client needs them to stay in the older release, save a copy with a new file name in your release version.



### Consider This: Mates

- Mates that reference assembly features rebuild slower than mates that reference component geometry.
- Mates that reference instances of components that are generated by a component pattern rebuild slower than mates that reference non-patterned components.
- Use faces when mating fasteners and avoid using axis to mate.
- Do not create “domino-style” mates. For example, don’t mate the face of Component 1 to Component 2 flush, then mate the face of Component 3 flush to Component 2, and so on. Designate one part as the base/main part and try to mate all the sub-components to the main part.

## Eliminate Mate and Equation Errors

Correct mate and equation errors by eliminating the warning icons in your browser tree. Many designers choose to ignore those glaring red and yellow icons in the browser tree, but they have a negative impact on performance. See Fig. 2.

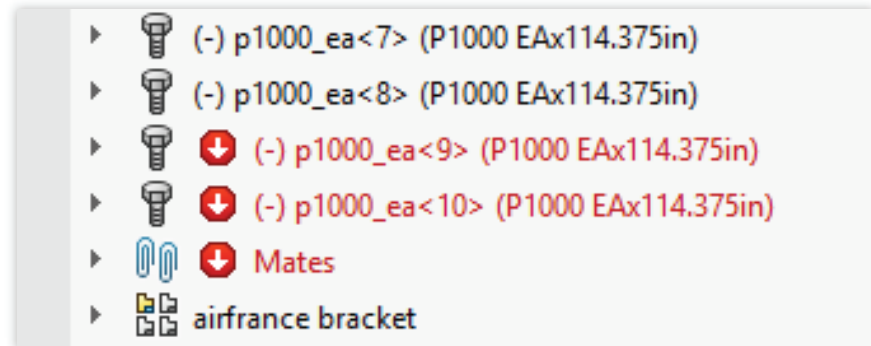


Fig. 2



## Consider This: Assemblies

- Advanced mates, such as width, limit, and hinge, can create a load on your resources. Only use them if you absolutely need to. Flexible sub-assemblies can really slow down your assembly, so use them sparingly.
- If you have missing references or components, this slows SOLIDWORKS down because it has to keep track of them and it's constantly trying to relocate them.
- The graphic display can control performance. Viewing the model in **Shaded mode** is the fastest while viewing in **Wireframe mode** is the slowest.
- Eliminate degrees of freedom in components. If your assembly is fully defined, its performance will improve. Use the **Lock Rotation** option when placing fasteners.
- Save a downloaded assembly as a part.
- Modify the display settings to turn off shadows and turn off real view.

# How to Organize Your Assembly

## Organize Your Assembly Using Treehouse

If you can reduce the number of components and assemblies in the top-level assembly, you can improve its performance. Remember: SOLIDWORKS doesn't rebuild the components or mates in sub-assemblies. To get started, create a rough drawing tree of any large assembly to help organize your assembly and provide a game plan of how to build it up. This will also help you figure out the best way to create simplified configurations of each part and sub-assembly. See Fig 3.

1. Navigate to the **Start** menu.
2. Select **SOLIDWORKS Tools**.
3. Select the **SOLIDWORKS Treehouse**.
4. Open a file and browse to locate the assembly.

**Note:** A drawing tree will be automatically created. The tree allows you to see how you have your assembly organized and shows you ways you can reorganize the assembly to improve performance.

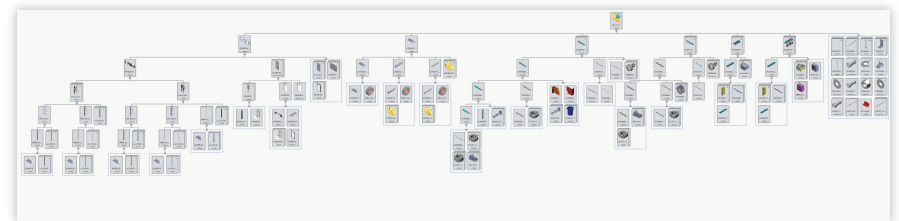


Fig. 3

5. Use the **Assembly Visualization** tool on the **Evaluate** ribbon to better understand how the assembly is being organized. You can change how you see the assembly using a nested (Fig. 4) or flat (Fig. 5) view.

### Nested View

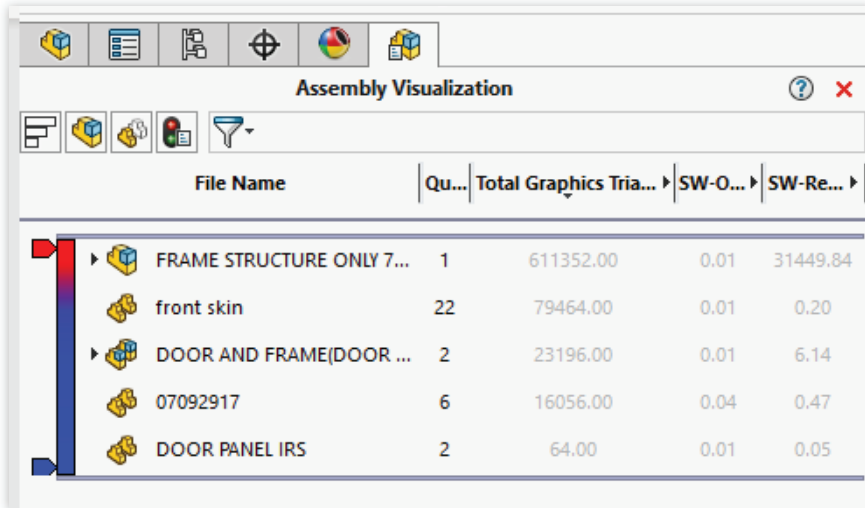


Fig. 4

### Flat View

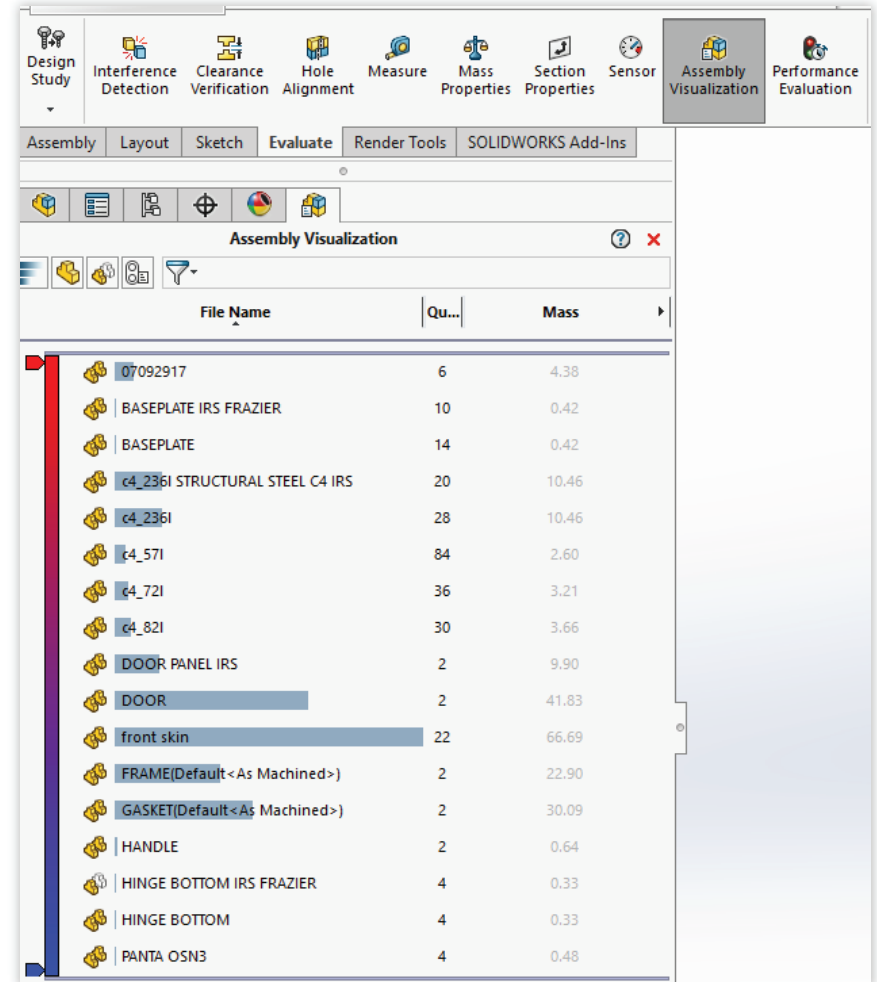


Fig. 5

6. Use the **Performance Evaluation** functionality on the **Evaluate** ribbon to help identify which files might be bogging down the assembly the most. See Fig. 6.

**Open Performance**

**Document Open File Details:**

File Name	Configuration	Open Time
BASEPLAT...RAZIER	Default	8.89
BAY_IRS...LENGTH	Default	8.11
VERTICAL...RAZIER	Default	1.95

[Show These Files](#)

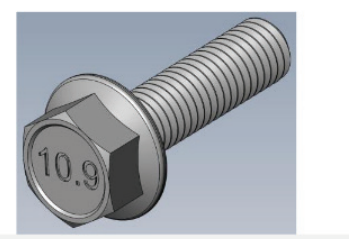
**Previous Version References**  
The files for all the components of the assembly have been updated to the latest version of SOLIDWORKS.

Fig. 6

### Case in Point: Reducing Bolt Details

Note that the total rebuild time is 0.58 seconds in the bolt in Fig. 7. Once the threads and the other unnecessary features were eliminated, the bolt's rebuild time reduced to 0.03 seconds. See Fig. 8. In this particular assembly, there were 302 bolts, so the benefits of reducing that level of detail added up quickly.

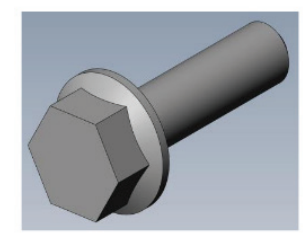
302 bolts x 0.58 seconds = **175.2 seconds**  
 302 bolts x 0.03 seconds = **9.1 seconds**  
 Time savings = **166.1 seconds (2 minutes 46 seconds)**



98093A548  
 Features 24, Solids 1, Surfaces 0  
 Total rebuild time in seconds: 0.58

98093A548, 2016  
 Features 24, Solids 1, Surfaces 0  
 Total rebuild time in seconds: 0.25

Fig. 7



98093A548  
 Features 9, Solids 1, Surfaces 0  
 Total rebuild time in seconds: 0.03

98093A548 Simplified, 2016  
 Features 9, Solids 1, Surfaces 0  
 Total rebuild time in seconds: 0.01

Fig. 8

## Organize Your Assembly by Creating Simplified Configurations

1. Create a configuration for each component that suppresses the holes.
2. Create a configuration for each sub-assembly that suppresses the hardware and uses the simplified configurations without holes.

**Tip:** By creating simplified versions of each part, you can significantly reduce the amount of time it takes to load the assembly.

**Tip:** The simplified configuration method requires a certain amount of pre-planning and setup. You have to create the simplified representations for each component and sub-assembly as you go.

For example, in a framing structure that uses Unistrut components, each component has a series of holes. To create a simplified configuration of each component, suppress the holes and fillets. See Fig. 9.

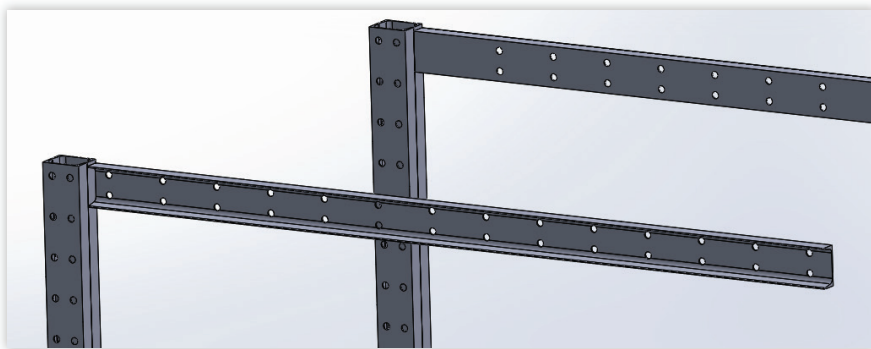


Fig. 9

3. Open each part and select **Add Configuration**. See Fig. 10-11.

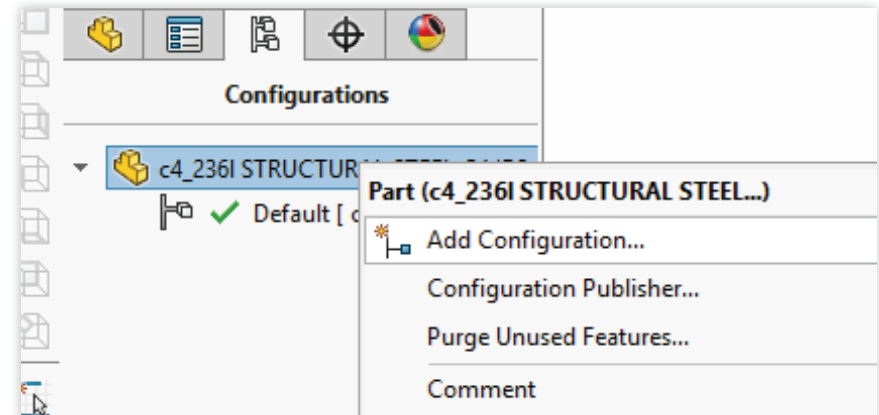


Fig. 10

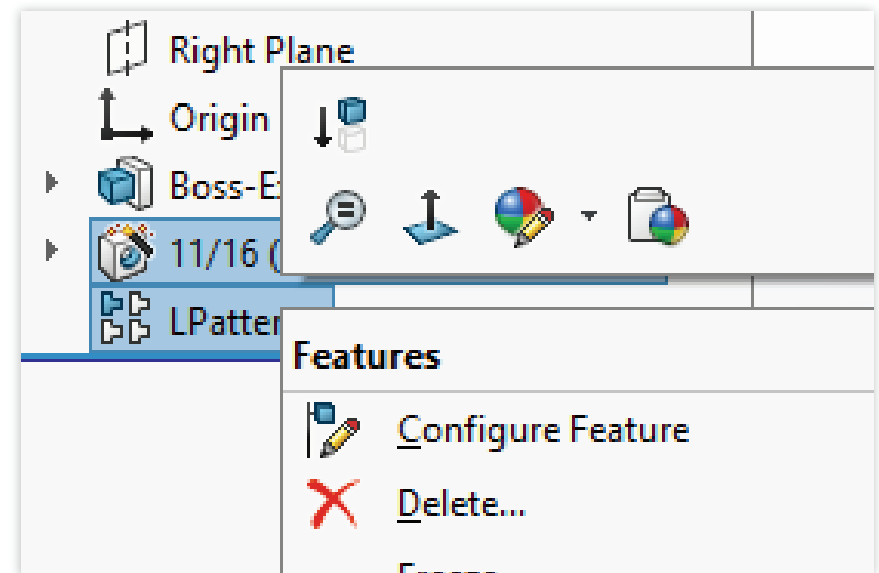


Fig. 11



- Highlight the features to suppress in the browser. Right click and select **Configure Feature**. See Fig. 12.

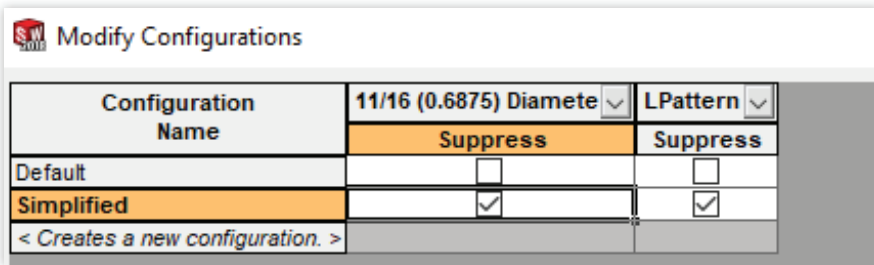


Fig. 12

- In the table, enable **Suppress** to eliminate the undesired features. See Fig. 13.

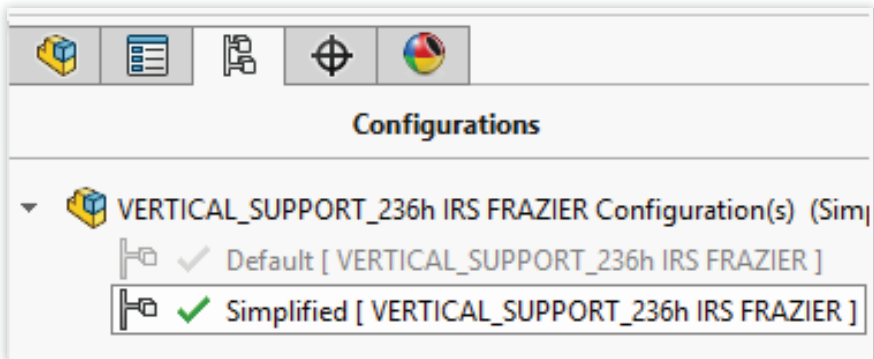


Fig. 13

- Go to the assembly where that component is used and create a simplified configuration. See Fig. 14.

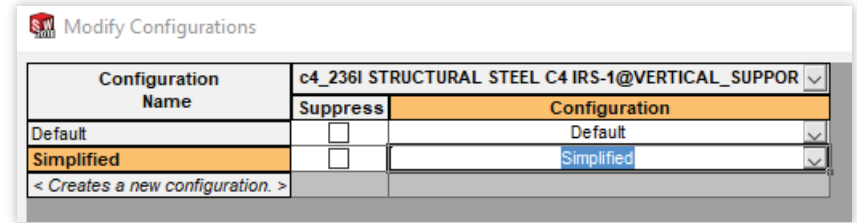


Fig. 14

- Set the simplified version of the assembly to use the simplified version of each component. This way, the holes are no longer visible and are not included in rebuilds. See Fig. 15.

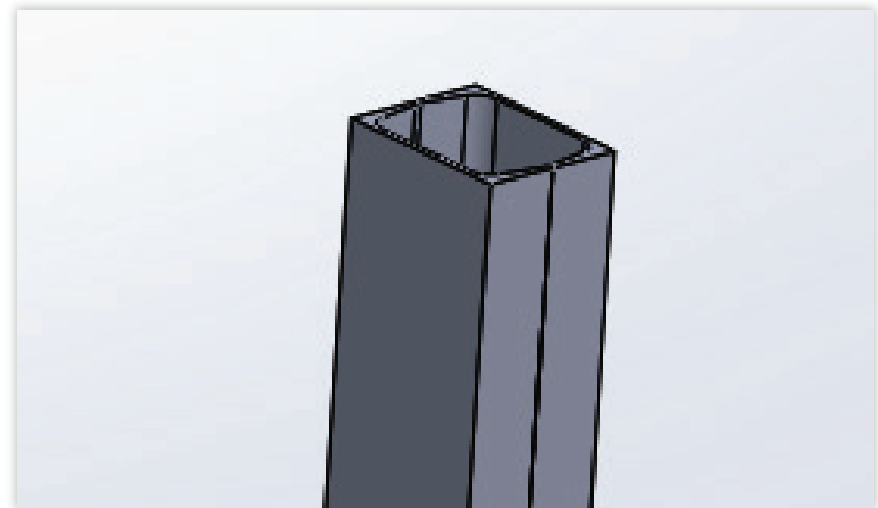


Fig. 15

- As you move up the assembly tree, keep adding simplified configurations using the simplified versions of each sub-assembly and component. See Fig. 16.

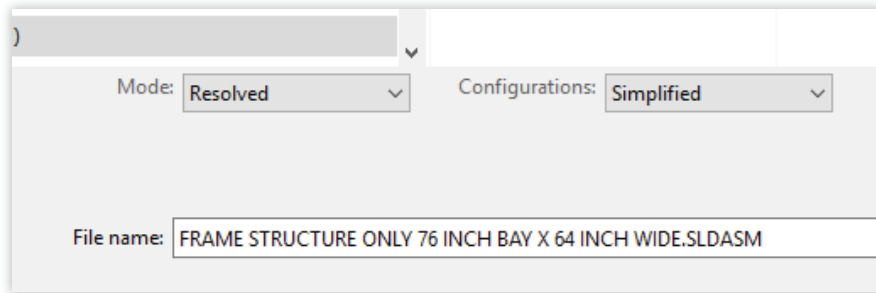


Fig. 16

- When you open the assembly, you can open it using the **Simplified Configuration** and reduce the amount of time it takes to open and load the file.
- If it is not detrimental to your assembly model, suppress fillets and chamfers in your parts in the simplified configuration.
- Create a folder for fillets and chamfers to make it easier to suppress and unsuppress. This is also useful for finite element analysis (FEA). If you're going to do that, don't use fillet or chamfer features for any mates.
- If you have a flexible sub-assembly, make it rigid for the simplified configuration.

**Tip:** Consider suppressing any internal components for the simplified configurations of assemblies.

## Organize Your Assembly Using SpeedPak

Use SpeedPak to create a simplified version of any large assembly. The SpeedPak version uses all the exterior faces and eliminates any internal features.

- To create a SpeedPak version of an assembly, go to the **Configuration Manager** tab in the assembly browser. See Fig. 17.

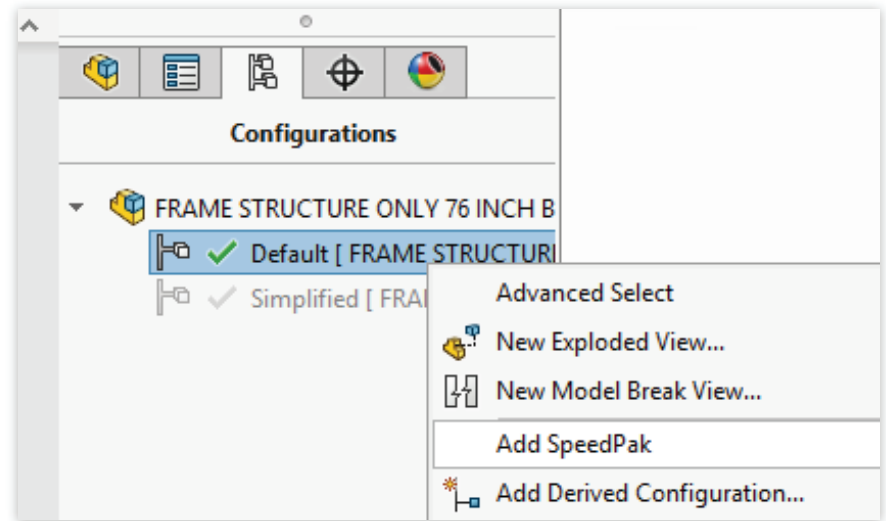


Fig. 17

2. Highlight the **Default** configuration, right click, and select **Add SpeedPak**. See Fig. 18.

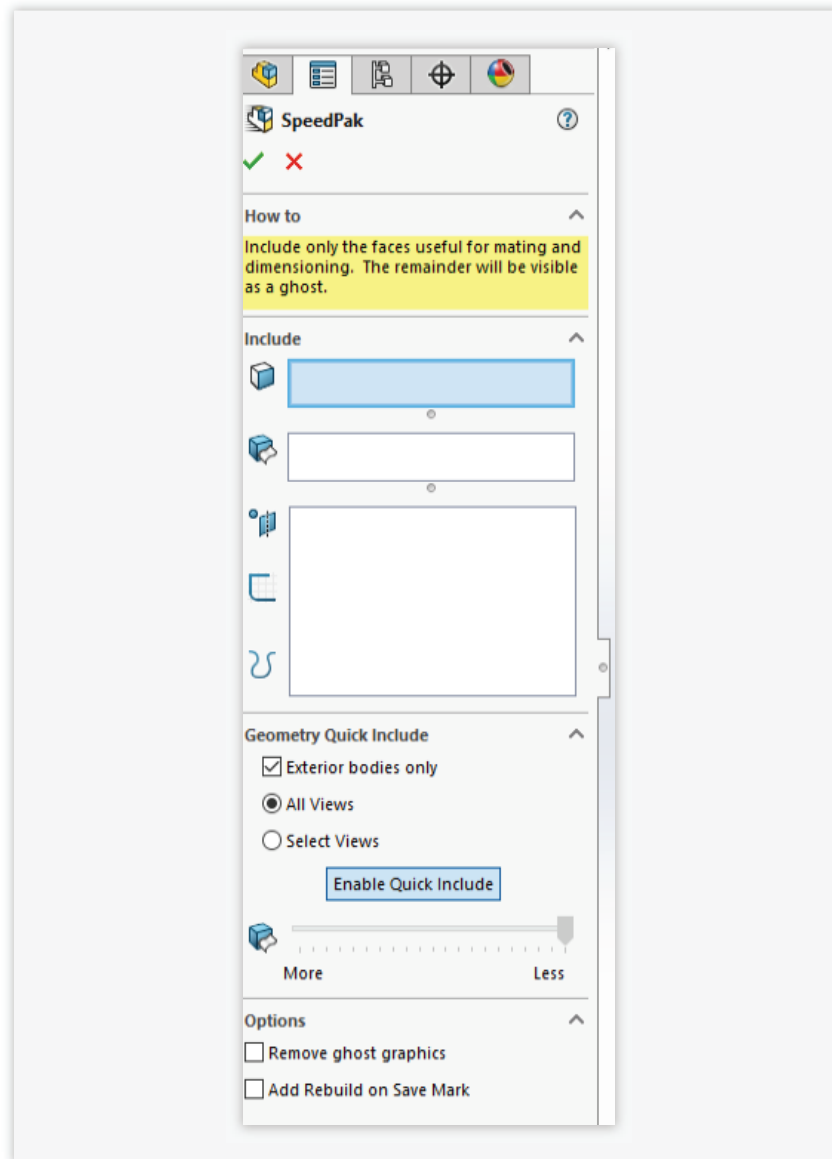


Fig. 18

3. Select the **green check mark** when you're done defining what you want to include or exclude from the operation. See Fig. 19.

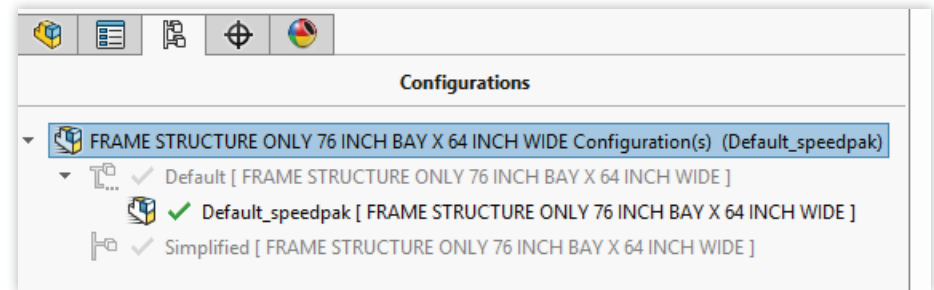


Fig. 19

4. The **SpeedPak** configuration is displayed under the **Default** configuration. See Fig. 20.

**Note:** SpeedPak will put you in a **magnifier viewing mode**. To exit that mode, press **Alt+S** or go to **Tools > Options > Display**. Disable the Display SpeedPak graphics circle.

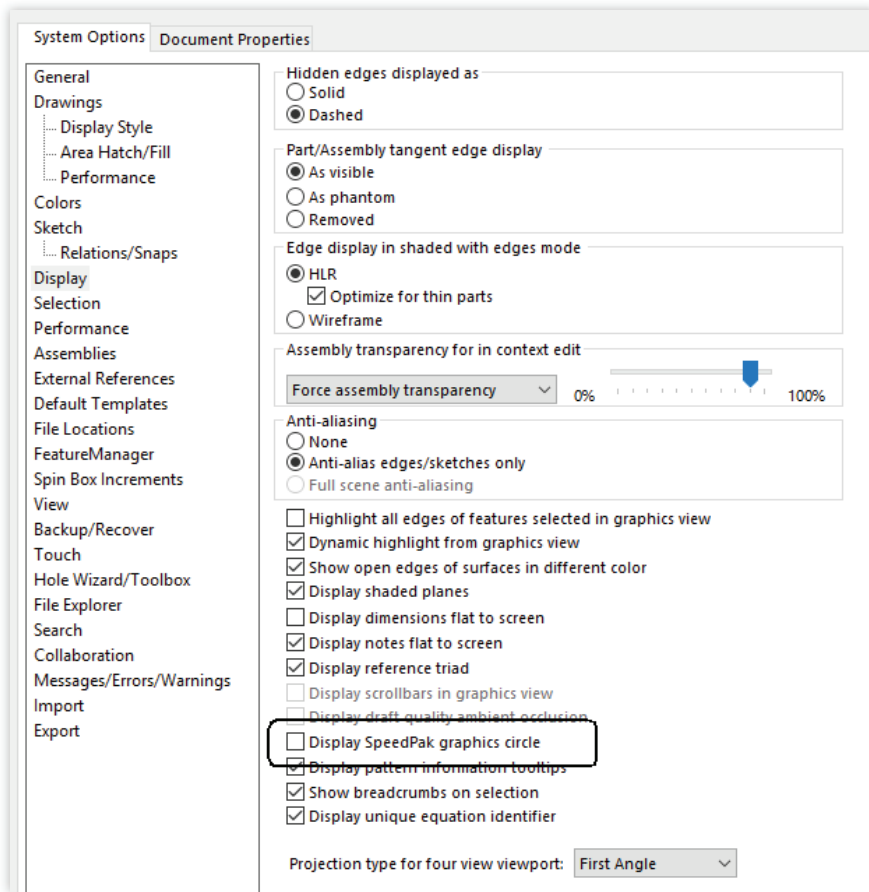


Fig. 20

## Organize Your Assembly Using the Defeature Tool

This technique is similar to creating a simplified configuration. The Defeature tool removes selected faces or features. The file is then saved as a “dumb solid,” or a body without feature definition or history. The downside is that, like any dumb solid, there aren't any features so the component or assembly can be difficult to modify.

**Tip:** This is a great tool to use if you want to protect intellectual property. For example, if you're emailing an assembly to a client to use to check for fit, you might not want them to copy your design. You can use the Defeature tool prior to emailing the file to the client.

1. To access the tool, go to **Tools > Defeature**. See Fig. 21.

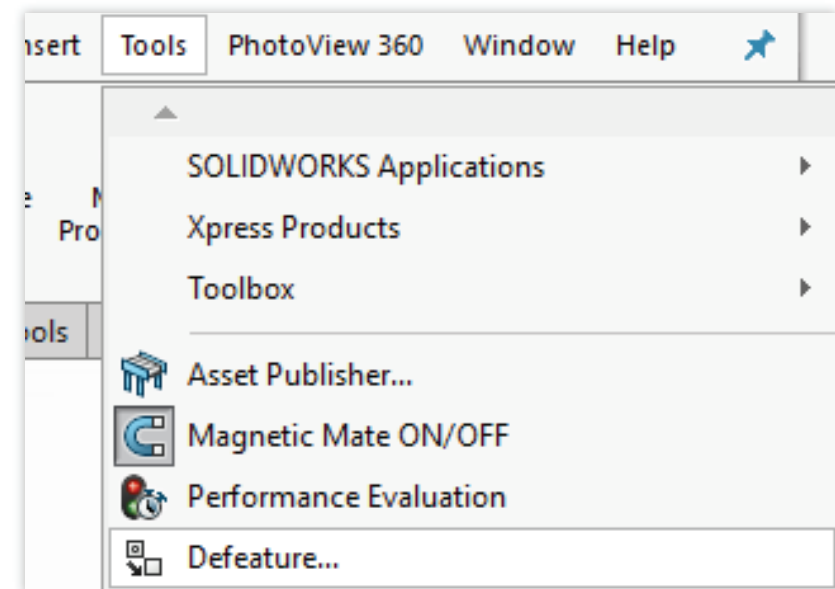


Fig. 21

2. Enable to remove all internal components. See Fig. 22.

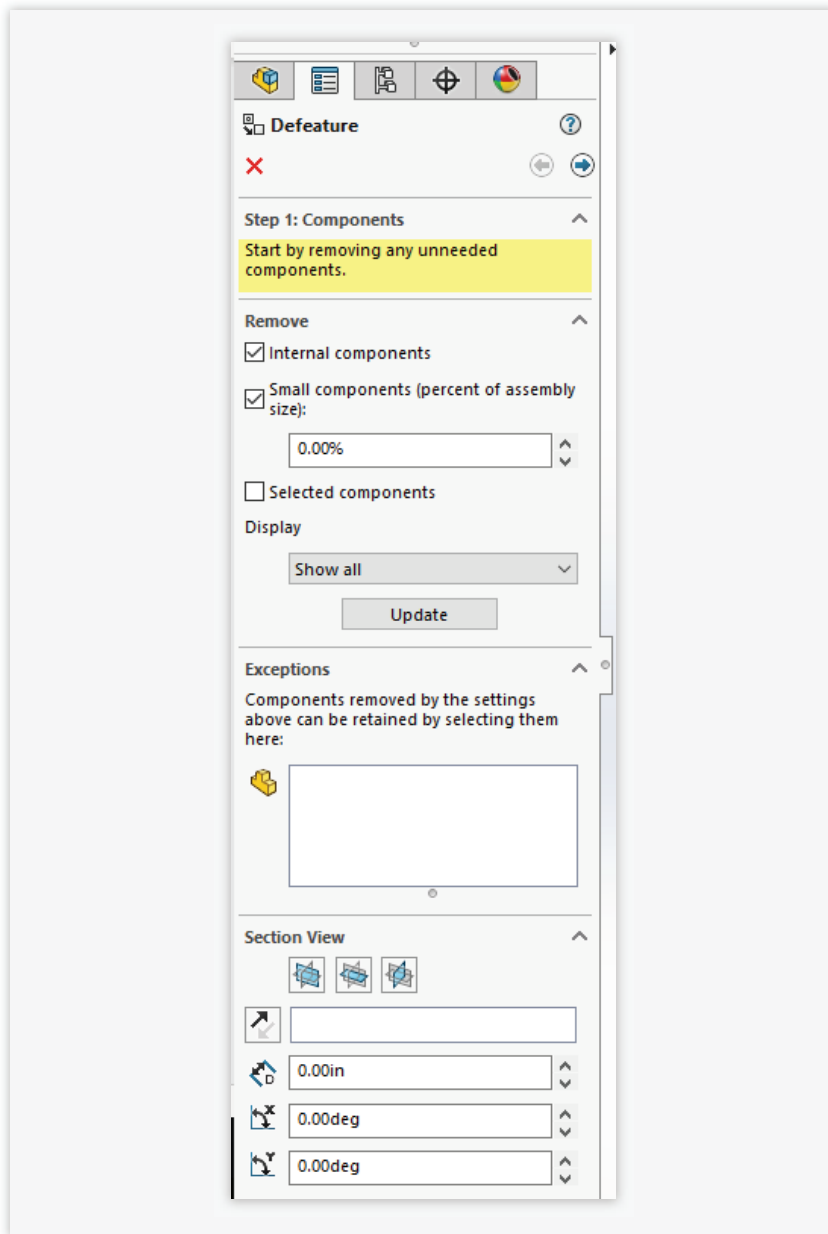


Fig. 22

3. Enable and Save to a new file.

4. Select the **green check mark**. See Fig. 23.

**Note:** You now have a new part you can use in an assembly as a dumb body or to send to a vendor/client.

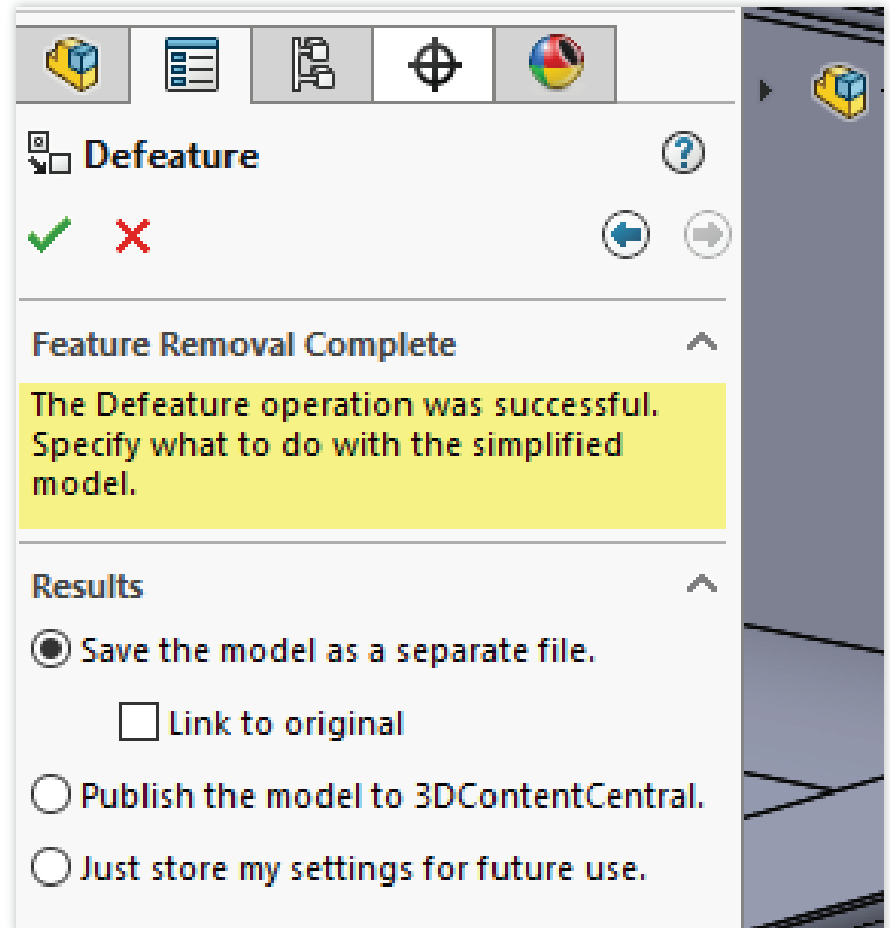


Fig. 23

## Organize Your Assembly by Creating a Folder for Your Hardware

Use folders as another way to organize an assembly to make it easy to suppress components that don't need to be visible. See Fig. 24.

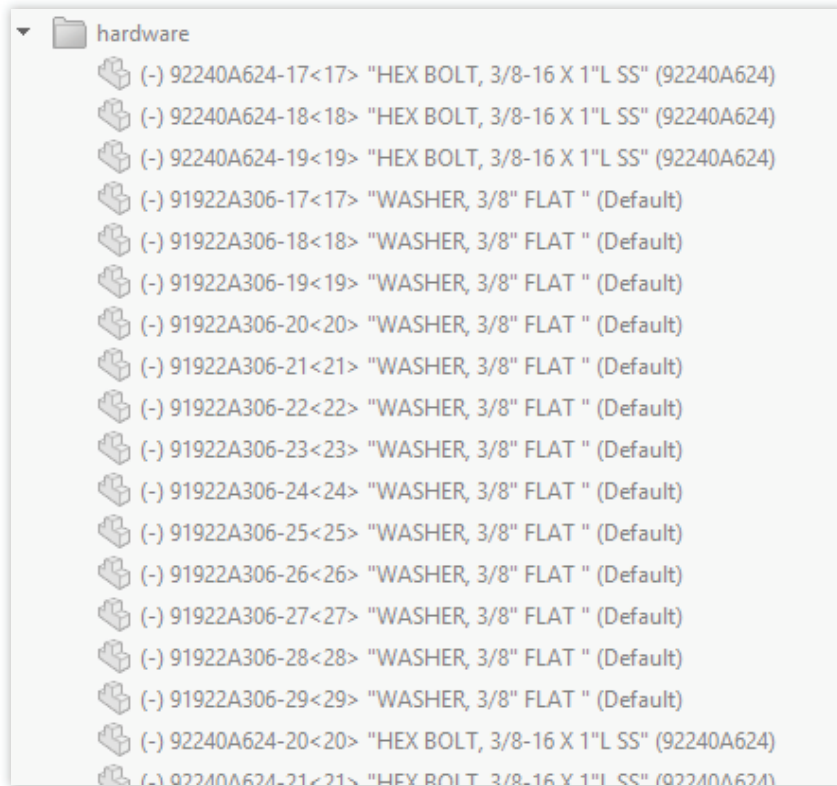


Fig. 24

1. To create a folder, highlight one of the components you want to place in the folder. See Fig. 25.
2. Right click and select **Add to New Folder**. See Fig. 25.
3. Drag and drop any additional components into that folder.

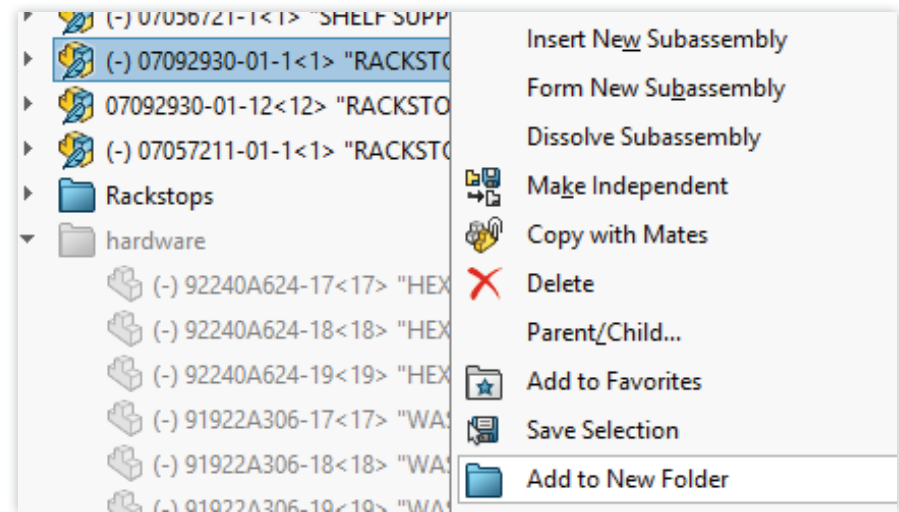


Fig. 25

To highlight the folder, right click and opt to suppress or unsuppress all the items in the folder. See Fig. 26.

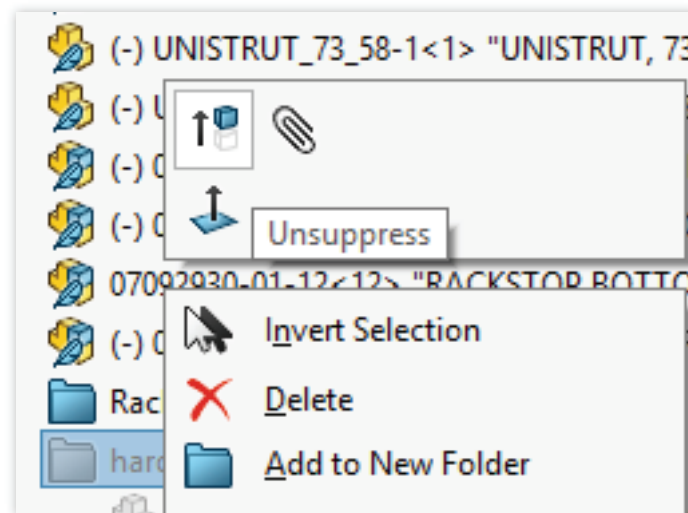


Fig. 26

# How to Troubleshoot Large Assemblies

1. Use the **Assembly Expert/Performance Evaluation** tool.
2. Look at the number of mates. If your assembly is entering the 150-200 mates realm, consider how to reorganize the file structure to create more sub-assemblies. Sub-assemblies don't have to rebuild their components and mates.
3. Check to see if you have any older (non-updated files) loaded in the assembly.

# How to Effectively Open Large Assemblies

## Consider the Mode

There are four modes available to open assembly files within SOLIDWORKS:

- Resolved
- Lightweight
- Large Assembly Mode
- Large Assembly Design Review

**Note:** All modes allow you to switch to Resolved. Simply select the components you wish to manipulate, right click, and select **Resolved**.

When opening a large assembly, use the **Do not load hidden components** option.

## Resolved Mode

By default, assemblies open in the Resolved mode. This automatically loads the assembly and all the associated file information. A resolved component is fully loaded into memory, fully functional, and accessible for editing.

## Lightweight Mode

Lightweight opens the assembly file but doesn't load the associated files. An icon appears next to the components within the assembly in the design tree to indicate it's in Lightweight mode. All the graphic and geometry data is loaded but not the actual features.

You can perform the following assembly operations on lightweight components without having to resolve or fully load them.

- Add and remove mates
- Select edge or face
- Measure
- Mass properties
- Create exploded views
- Check for interference
- Collision detection
- Add dimensions
- Create section views

## Large Assembly Mode

Large Assembly mode is similar to Lightweight mode but restricts additional processes. You can control which options are available in Large Assembly mode by going to **Tools > Options > Assemblies**. If the number of components exceeds the number specified in Options, the mode is automatically set to Large Assembly mode.

## Large Assembly Design Review Mode

Large Assembly Design Review mode is the best option for viewing the assembly and checking distances. This is a good choice when doing a design review with a colleague or a client. It allows the assembly to load very quickly. You can zoom in and out, rotate the model, and take measurements.

In Large Design Review mode, you **can**:

- Navigate the design tree
- Measure distances
- Create cross sections
- Hide or show components
- Create screenshots

In Large Design Review mode, you **can't**:

- Make file changes
- Save
- Make drawing from model

## Upgrade Your Hardware

Pan, Zoom, and Rotate are operations that have the most impact on performance. These operations are not managed by memory or RAM. These operations are mostly CPU driven. So, if the system is too slow, throwing more RAM in the motherboard might not solve the problem. Instead, consider upgrading the CPU or motherboard.

## How to Manage Your Resources

### Update System Options and Document Properties

1. Check what add-ins you're using and unload the add-ins you aren't using or don't need.
2. Unload any ribbons you don't need or don't use.
3. Go to the **System Options** dialog. On the **General** page, uncheck **Show thumbnail graphics in Windows Explorer** and **Show latest Technical Alerts and News in Welcome dialog**. Testing has shown that if the Freeze Bar is enabled, there's an improvement in file open times.



4. On the **Performance** page, uncheck **No preview during open (faster)**. This ensures your assemblies will open faster. See Fig. 27.

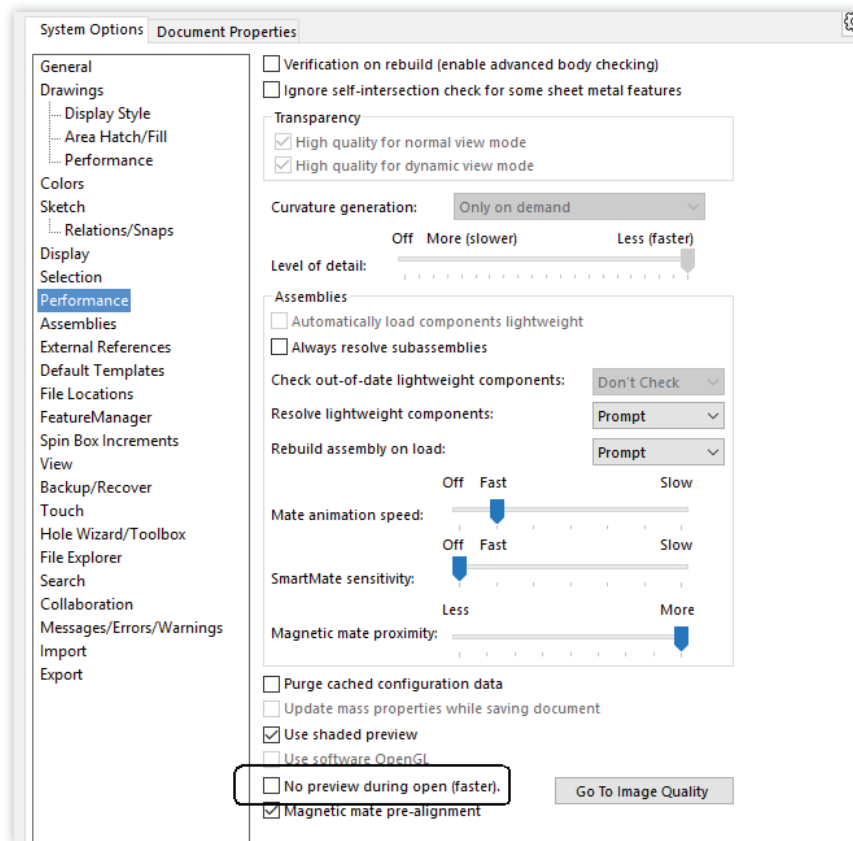


Fig. 27

5. Open the **Assemblies** page. In the middle of the page, check the **Use Large Assembly Mode to improve performance when the assembly contains more than this number of components** box. See Fig. 28.

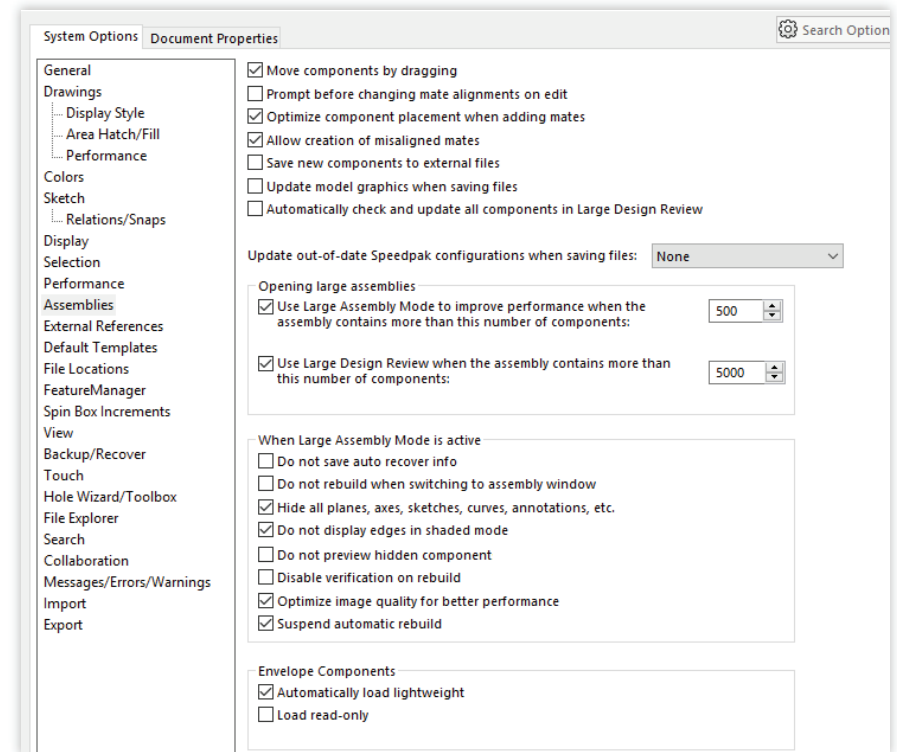


Fig. 28

6. Select the **Document Properties** tab. Then, select the **Image Quality** page. Reduce the quality in both sections to speed up performance. Check the **Apply to all referenced part documents** box. See Fig. 29.

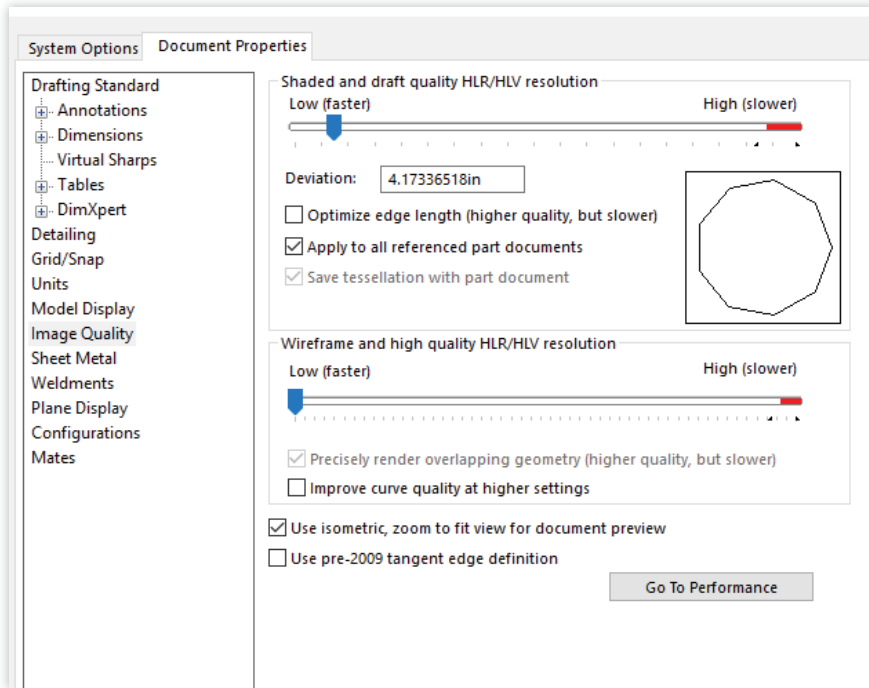


Fig. 29

7. Open the **File Locations** page. Under the **Show Folders for** drop-down list, select **Referenced Documents**. Do not add any paths for the referenced documents as this will slow down performance. See Fig. 30.

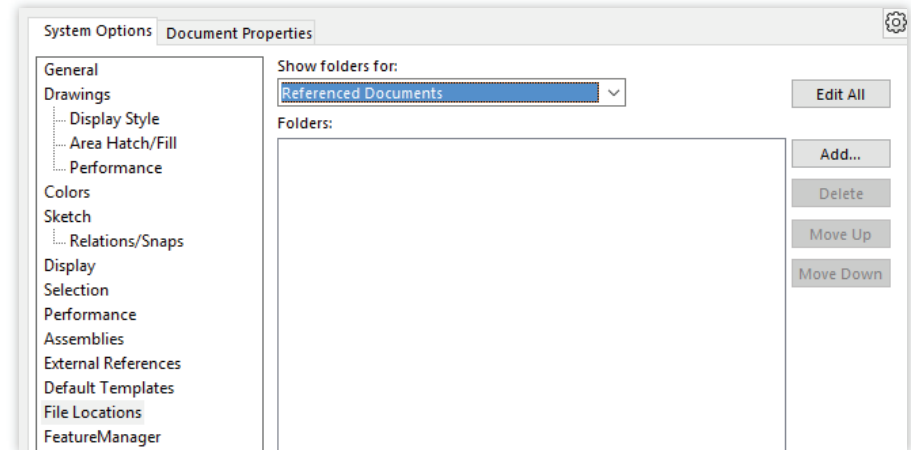


Fig. 30

8. Open the **External References** page. Check the **Don't prompt to save read-only referenced documents (discard changes)** box. In the **Search external references** section, check the **Reference Documents specified in File Location** box and **Include sub-folders** box. See Fig. 31.

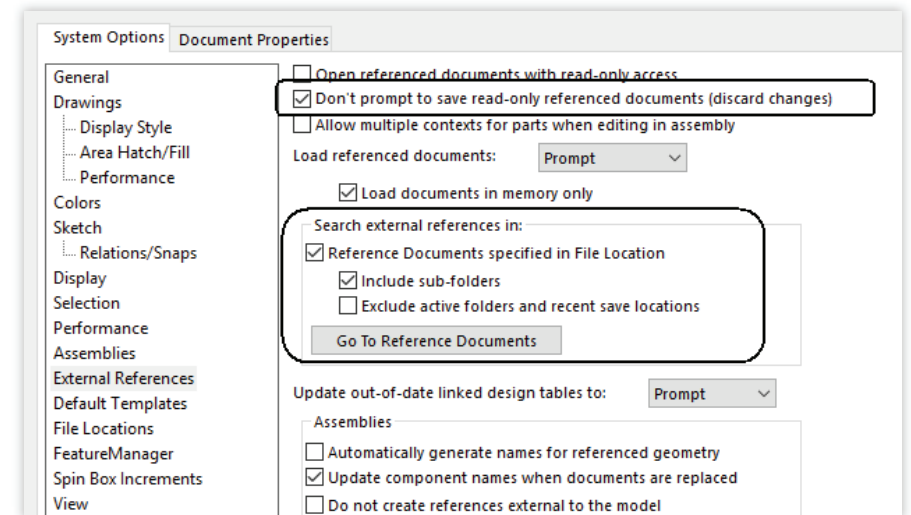


Fig. 31

# About SolidProfessor

At SolidProfessor, we do things differently. Rather than thinking of training as a one-time event, we aim to create a culture of continuous learning by empowering engineers and designers to take full control of honing their knowledge and skills with online tutorials.

## Take advantage of the tools you need to succeed



Prepare for SOLIDWORKS CSWA, CSWP, and CSWE with guided Learning Paths



Earn SolidProfessor Technical Certificates to boost your resume



Gain unlimited access to our library of 240+ SOLIDWORKS courses and 5,000+ other CAD tutorials



Target opportunities for performance improvement with SOLIDWORKS skills assessment tests



Diversify your skill set by diving into 15+ software programs and engineering design theories

**Explore SOLIDWORKS  
Tutorials »**

**Become a Member »**

## Questions? Contact Us.

 [www.solidprofessor.com](http://www.solidprofessor.com)

 +1 (619) 269-8684

 [Facebook](#)

 [Twitter](#)

 [LinkedIn](#)

