SOLIDWORKS Large Assembly Guide

Suggestions and best practices for working with large assemblies and their referenced files.

Contents

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Introduction

This guide is intended to be a best practice guide for users who wish to optimize their large assembly and large assembly drawing performance. Generally, all large-scale design uses complex assemblies. SOLIDWORKS allows users to easily build complex assemblies consisting of many components, which can be built from parts and subassemblies. Drawings will also need to be produced for these large designs. Due to the size and nature of these assemblies, you may need to introduce different modelling techniques and System Options etc. to improve performance.

In this document we will discuss these options and techniques. Not all techniques will apply to your design environment but if you take the time to read and understand these techniques, you can find which work best for you and your designs.

SOLIDWORKS is serious about helping you get your job done fast and accurately. Continuous improvement of large assembly and drawing performance is a major focus with every software release. Through the addition of new enhancements, and the refinement of existing functionality, SOLIDWORKS now provides unmatched large assembly and drawing performance for your products.

Large Assemblies

What is a large assembly? A large assembly can be any assembly that is complex enough to max out your system resources and harm your productivity. There are usually lots of contributing factors to why an assembly is affecting your performance including:

- General Settings (Options)
- Hardware (CPU, memory, graphics card, OS)
- Environment (Network, retrieval methodology)
- Data Management (Manual data management, PDM)
- File size
- Number of components, mating, use of configurations and external references in an assembly
- Assembly Design Methodologies (Top-Down, Bottom-Up)
- Drawings (Lightweight or resolved, configurations, views, display quality)
- Parts (Sketches, external references level of detail)

It is usually quite apparent when you are experiencing problems because of a large assembly. You may experience performance degradation in all or some of these areas:

- Slow and painful opening and saving
- Slow rebuild times
- Slow drawing creation and manoeuvrability
- Slow and stuttering rotation and viewing of the file
- Slow mating ability

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Assemblies in SOLIDWORKS

SOLIDWORKS assemblies can be made up of many components, which can be parts of other assemblies, called subassemblies. For most operations, the behaviour is the same for both types of components. Adding a component to an assembly creates a link between the assembly and the component. When SOLIDWORKS opens the assembly, it finds the component file to show it in the assembly. Changes in the component are automatically reflected in the assembly.

The Assembly FeatureManager Tree is usually solved in this order:

1. Reference geometry is solved. Then sketches that are listed before parts in order, at the top of the design tree
2. Individual parts are rebuilt as necessary
3. Mates and are solved
4. In-context features in parts are solved
5. Reference geometry and sketches listed after the mates are solved
6. Assembly features and component patterns are solved (assembly dependent items)
7. Loop to step 3 to solve mates that are connected to anything that was solved after the first round of mate solving
8. Continue to loop until complete

Large Assemblies – Main Points

The following points are some of the main topics that we will be covering in more detail. There will also be many others to discuss.

- Keep physical file size down. For very large files, consider splitting the file into multiple pieces i.e. sub-assemblies
- Consider the structure of your file. Maybe include more sub-assemblies inside of the top-level
- Reduce the number of top-level mates in any assembly
- Be aware of circular references and circular mates. Mating to a common reference, ideally a plane, is better than a string of mates
- One sure fire way to increase performance is to remove (or ideally suppress) all mates. Especially if movement is needed, and changes to geometry are not going to be made. Note: This is only to be used in extreme circumstances
- Utilize SpeedPak configurations
- Utilize Simplified configurations
- Be sure to use Assembly Visualization to check Graphical triangles, open times, rebuild times and to get an idea of problem components
- Take advantage of the SOLIDWORKS open time property

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Assembly Design Methodology

The modelling techniques listed below are powerful, but each has advantages and disadvantages. Make sure you use these techniques with care and only use the techniques that you feel are most appropriate to your own situation.

Bottom-up Designing

Bottom-up design is the most traditional method. You first design and model individual parts, then insert them into an assembly and use mates to position the parts. To change the parts, you must edit them individually. These changes are then seen in the assembly. Bottom-up design is the preferred technique for previously constructed, off-the-shelf parts, or standard components like hardware, pulleys, motors, etc. These parts do not change their shape and size based on your design unless you choose a different component.

Top-down Designing

Top-down design is also referred to as "in-context design" in terms of SOLIDWORKS software. In Top-down design, parts, shapes, sizes, and locations can be designed internally to the assembly.

For example: You can model a motor bracket, so it is always the correct size to hold a motor, even if you move the motor. SolidWorks automatically resizes the motor bracket based on the existing assembly design, even if it changes. This capability is particularly helpful for parts like brackets, fixtures, and housings, whose purpose is largely to hold other parts in their correct positions. You can also use top-down design on certain features (such as locating pins) of otherwise bottom-up parts.

Sketches can also be used to control part designs within an assembly. The design of a system such as a photocopier can be laid out in a layout sketch, whose elements represent the components of the photocopier. The 3D components are created based on the sketch. As elements are moved or resized in the sketch, SolidWorks automatically moves or resizes the 3D components in the assembly. The speed and flexibility of the sketch allow you to try several versions of the design before building any 3D geometry, and to make many types of changes in one central location.

The advantage of top-down design is that much less rework is needed when design changes occur. The parts know how to update themselves based on the way you created them. You can use top-down design techniques on certain features of a part, complete parts, or entire assemblies. In practice, designers typically use top-down techniques to lay out their assemblies.

Creating a Part in an Assembly

You can create a new part in the context of an assembly. That way you can use the geometry of other assembly components while designing the part. The new part is saved internally in the assembly file as a virtual component. Later, you can save the part to its own part file if required. You are also able to create a new sub-assembly in the context of the top-level assembly.

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To save a virtual component to its own external file, right-click the component and select Save Part (in External File). Alternatively, when you save the assembly, you can select to save the part either inside the assembly or to an external file.

Skeleton Part Technique

This technique is similar to the Assembly Layout Sketch technique in top-down designing, except the sketch is created at a part level rather than at the assembly level. This technique is most effective if you want your layout sketch to be able to be inserted directly into a part or into any other assembly as a first part. Think of it as a reusable layout sketch for parts and assemblies.

First, the part that that represents the overall simplified layout of the assembly is created. This is the skeleton part. Most time this is a 2D sketch, kind of like a cross-section of the design. It may contain sketches, planes, points, surfaces, whatever is needed to define the skeleton to the level of detail you desire. You can use Sketch Blocks functionality to more easily create the skeleton and even create mechanisms that move to test kinematics.

This skeleton part can then be added as the first part to any assembly or be inserted directly into a part. Sub-assemblies can also use the skeleton part as the first component so that even if you open just a sub-assembly, you can still see and select references from the skeleton part representing the overall critical references of the entire assembly. Parts can be assembled to the skeleton part in the sub-assembly or at the top-level assembly.

<u>Note:</u> If you create in-context references to the skeleton, or assembled pieces of a mechanism to the skeleton, you won't be able to free-drag the components. Component position will update if the skeleton changes.

Multi-Body Technique

If you have parts in an assembly with extremely intricate part relationships (i.e. laminated wooden guitar body, mold cavity, etc.) you may consider using multi-body modelling. Parts can be "split" out to create an assembly – the location of parts in the split assembly is automatically controlled by the multi-body part.

<u>Note:</u> individual parts are completely dependent on the multi-body model. Some changes to parts should occur in the multi-body part. Features added to the individual "split" parts (hole wizard holes, shell, etc.) are not sent back to the multi-body part.

"Insert Part" or "Master Model" Technique

If you have a design with only a few intricate relationships between the parts you may opt to model only the shared geometry or references. This technique is often used in designing consumer products, automobile bodies, and other designs with intricate relationships among the parts and complex shapes. This design technique is more appropriate if you usually start your design with a surface model which represents the outside shape of the product.

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The surfaces, datum planes or other reference geometry that multiple parts will reference should be modelled first. New parts are then started - this reference part can be added into each model as the first feature. If the part inserted into the design is updated, all parts update. Note: these have similar cautions to those of the multi-body parts. They still have an external reference to the inserted part. Changes to these features must occur in the inserted part.

General Guidance for Large Assembly Design

The following points cover general recommendations and best practices you can consider when working with large assemblies.

Managing File Sizes

Where possible, it is recommended that physical file size is kept down to down to a minimum of 2MB. Files larger than this can have a significant effect on performance. If you have an assembly file that is much larger than this, you should consider analysing it to reduce the file size. These are some of factors that you can take into consideration to achieve this.

- Consider splitting the main file up into individual files. This can be done by either creating smaller sub-assemblies or consolidating many parts into one. This is particularly relevant for weldment structures.

- Use relations to control the sketch over the use of dimensions. Usually this will be simpler, neater and faster with the bonus of a much-reduced file size.

- Consider using Features instead of Sketches i.e. use Fillet features instead of Sketch Fillets. This also applies to patterning. It is much better to pattern Features rather than inside a sketch where possible.

Network v Local

When an assembly is slow to open and save, you are running into an issue with loading a large amount of information into "RAM." If the SOLIDWORKS files are stored on a fixed disk in a network location, it will take longer to open and save the files than it would if the files were stored on a local fixed disk (i.e., stored in your laptop or tower). The reason behind this is the information being loaded into RAM may be bottlenecked by a variety of networking factors. These bottlenecks will not exist if you are opening the files from a fixed disk that is stored directly in your laptop or tower.

To cache files locally, you may find it useful to utilize the PACK AND GO command to make a copy of the entire assembly to save it in local location on your local fixed disk while you work on it.

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Data Management software

You may want to consider utilizing data management software such as SOLIDWORKS PDM if you are keen to use a network location. PDM will help manage your engineering data in a centralized location. The data is archived on a server, and files are cached locally on a user's machine on check out. Users will then work with those files that are stored on their local drives, eliminating the need to work across a network (which as previously discussed are slower). This means better performance, especially during Opening, Saving and Rebuilding operations. Alternatively, if you would prefer to use a cloud solution, we would recommend 3DEXPERIENCE Cloud Services.

With data management, all users will have direct access to the latest versions of files, that are stored in the central location. As all files are stored centrally, the chance of data loss is greatly reduced. This includes having your work overwritten, by a co-worker saving files to an unmonitored and unmanaged shared network drive.

Autosave, Backup and Recover

If using these tools, you should not try to use save locations on a network. If the locations aren't local, SOLIDWORKS will try to be constantly working and trying to write over the file in the network location. A poor network connection can cause slowness and performance issues. Alternatively, turn these options off when working on particularly large assemblies. But this will be at the detriment of the functionality.

The Backup option to put the backups in the same place as the original file can double the issue if you're working on files saved the network and not locally. It is better that you set the locations to your C drive instead.

Unused Add-ins

Many SolidWorks utilities and tools are designed as modules that can be enabled or disabled whenever required. These are called Add-ins and can be accessed via Tools > Add-ins.

You may have different add-ins available depending on your type of SolidWorks installation. Some of these add-ins are commonly used (such as Toolbox) while many others are only used very rarely. As each add-in requires additional memory and places a further load on your computer, it is strongly advised to keep most add-ins disabled during general SolidWorks usage, only enabling when required.

When looking to disable/enable an add-in, you will notice in the Add-ins dialog that there are both a left-hand (LH) and right-hand (RH) column that can be checked (checked in On, empty is Off). The LH checkbox determines if the add-in is currently enabled for this session of SolidWorks, while the RH checkbox determines if the add-in will be automatically enabled the next time you launch SOLIDWORKS.

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Previous Version Files

In versions of SOLIDWORKS previous to 2020, it is always recommended that to improve file performance, that files are saved using the latest version of the software you are using. However, in SOLIDWORKS 2020, most assemblies and drawings that were saved in a previous version will open nearly as fast as those saved in SOLIDWORKS 2020.

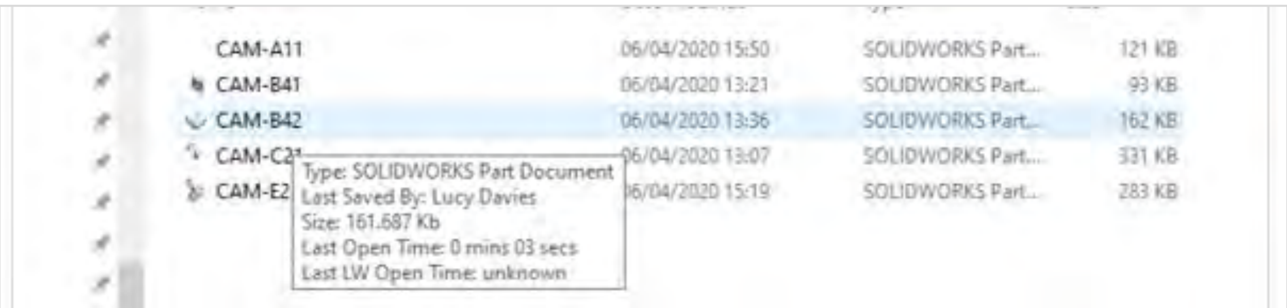
Previously, some assemblies and drawings that had not yet been saved in the current version, took longer to open and save. This was particularly true for assemblies and drawings with reference components in multiple configurations.

A system option, **Force referenced documents to save to current version under External References** now improves save performance. When you clear the option, only documents that have been modified in the current session are saved to the current version of SOLIDWORKS. This reduces save times significantly on the first save of large assemblies and drawings.

For example, if a SOLIDWORKS 2019 assembly with SOLIDWORKS 2019 parts is opened in SOLIDWORKS 2020 and has a mate added to the top-level assembly. The file is saved. When the option is cleared, only the top-level assembly is saved because it has been modified by adding a mate. Only the top-level 2019 assembly file has been converted to 2020. When you select this option, the assembly and its references are all converted to 2020 when saving the assembly. This is the standard behaviour in versions before SOLIDWORKS 2020.

SOLIDWORKS Open Time Property

The file property, Last Open Time, appears in a tooltip when you hover the pointer over a SOLIDWORKS part, assembly, or drawing document in File Explorer. Last Open Time displays the time it took for the SOLIDWORKS software to open the file, the last time it was opened. This file property is helpful for managing your time when you have large assemblies or data sets that are time consuming and slow to open. We can quickly identify the files that take the longest to open to narrow down causes of issues.
The time displays in minutes and seconds, and is available for models that are opened directly from disk. For assemblies opened in Lightweight mode, the tooltip displays Last LW Open Time. Note: Last Open Time does not update in reference files when they are saved while open in memory, but does update when the reference files are saved while open in their own window.



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You can add a column in the Details view of File Explorer to display SW Open Time. (You can identify files saved in earlier versions of the SOLIDWORKS software by adding a column for SW Last Save with.)

To add columns for SW Open Time and SW Last Save with in File Explorer: open a folder with SOLIDWORKS documents so the contents of that folder appear in the Details view of File Explorer. In the Details view, right-click on the header. Typically, the header displays columns for Name, Type, and Size. Then in the context menu, select More. Under Details in the dialog, scroll to SW Last Save With and SW Open Time. You can select one or both options.

System Options and Document Properties

SOLIDWORKS contains various performance options that allow users to handle large assemblies faster and more productively. While handling Large Assemblies, ensure you have considered the following options carefully.

- *Verification on Rebuild.* This option turned on checks every new or modified feature against all existing faces i.e. the faces that share an edge with the face(s) of the feature it is building. For example, when building a fillet, SOLIDWORKS checks to see if the face of the fillet fits with the faces the fillet shares edges with ("connects to"). When it fits, SOLIDWORKS moves on, but when it doesn't an error is generated. When this option is turned off, new and modified features will be checked against adjacent surfaces.
  Essentially this controls the level of error checking when you create or modify features. For most applications, the default setting (cleared) is adequate and results in a faster rebuild of the model. Checking Verification on Rebuild will increase rebuild times, but the amount of time required to fix a complex part with multiple rebuild errors may be far greater.

- Check your *system option > Assemblies > Do not rebuild when switching to assembly window*. You can control when to rebuild. You will not waste time rebuilding after moving between windows. Use Ctrl + Q to rebuild manually.

- Look at checking your *system option > Performances > Level of detail (Less)*. This allows you to specify the detail level during dynamic view operations in assemblies, multi-body parts, and draft views in drawings. Note: this option is not available when Large Assembly Settings is on.

- Check your *system option > Performances > automatically load components lightweight*. This loads all the individual components and subassemblies in assemblies that you open as lightweight. When a component is lightweight, only a subset of its model data is loaded in memory. The remaining model data is loaded on an as-needed basis.

- Check your *system option > Performances > No preview during open (faster).* This will speed up the opening of a file.

- Set your *system option > Performances >Rebuild assembly on load to "Prompt".*

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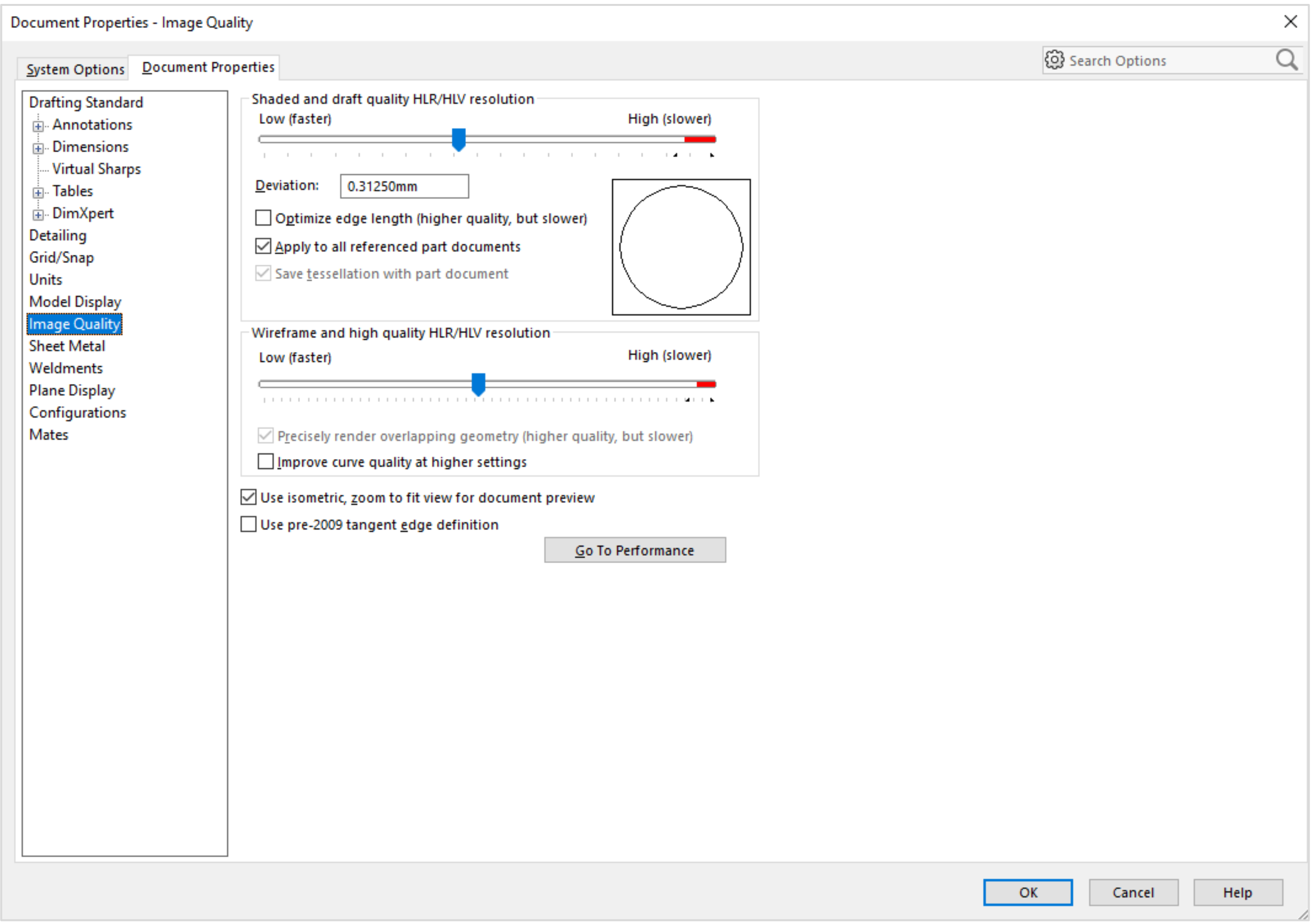
- Set your *system option > External References documents to "Prompt".*

- Set your *Document Properties > Image quality > Shaded and draft quality (Low) & Wireframe High quality HLR/HLV Resolution (Low)* for faster performances.

- *Adjust Image Quality Settings.* Try and refrain from using too High of an Image Quality. A higher image quality can lead to very poor performance, particularly when this component is used in multiple locations in an assembly. Shaded and Draft Quality HLR/HLV resolution controls the tessellation of curved surfaces for shaded rendering output. A higher resolution setting results in slower model rebuilding but more accurate curves. Wireframe and High-quality HLR/HLV resolution control the image quality of model edges in drawings. A higher resolution setting results in a slower screen redraw but a higher display quality. It is recommended to keep your image quality set to a reasonable value. To change these settings, go to *Tools > Options > Document Properties > Image Quality*.



- *RealView Graphics and Shadows.* Using both RealView and Shadows with your SolidWorks model can make it look more realistic, but does require additional graphical resources. It is typically recommended to work within SolidWorks with these toggled off, and only toggling these back on when more realistic visual features are required. These can both be toggled on or off via View Settings on the Heads-up View Toolbar.

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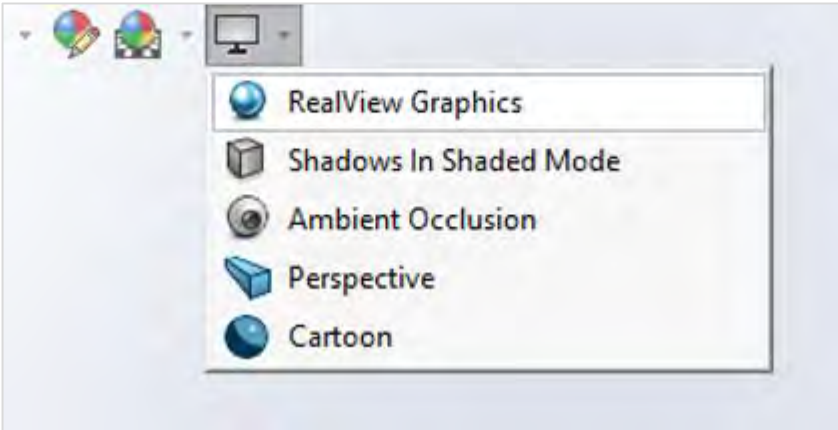

- *Dynamic Highlight.* Items in the graphics area are highlighted when you select them, or dynamically highlighted when you move the pointer over them. Dynamic highlighting can be really useful because it lets you tell where your mouse is before you actually click anything, but it can be also very taxing on the graphics card. In *Options > Display, uncheck "Dynamic highlight from graphics view*. Similarly, you can go to System *Options > Feature Manager > and uncheck the option for Dynamic highlight.* For this point, there is a balance is here between getting the best possible performance and the convenience of having that dynamic highlight.

- *Shaded with Edges Removed.* From the Heads-up toolbar, we can change the display style to shaded with edges removed. This is probably going to give you the best performance from your assemblies as far as display style goes (compared to shaded with edges shown or wireframe or hidden lines visible or hidden lines removed). This is the least taxing on your graphics card so should reduce any lagging you are experiencing.

Guidance for Large Assemblies

The following points cover recommendations and best practices you can apply to your large assembly files. Not only will these impact the behaviour of the assemblies themselves, they will be sure to impact the behaviour of any large assembly drawings.

Large Assembly Mode and Lightweight Assemblies

It is highly recommended that Large Assembly and Lightweight Modes are used when opening and using Assemblies. Large assembly mode is a collection of system settings that improves the performance of assemblies. They can be turned on at any time or a threshold can be set. When the threshold for the number of components to automatically open the assembly in Large Assembly Settings has been reached, the mode will be activated. This threshold is user defined.

To turn Large Assembly Settings on or off, click Large Assembly Settings (Assembly toolbar) or go to *Tools > Large Assembly Settings*. When Large Assembly Settings is on, Large Assembly Settings appears on the status bar.

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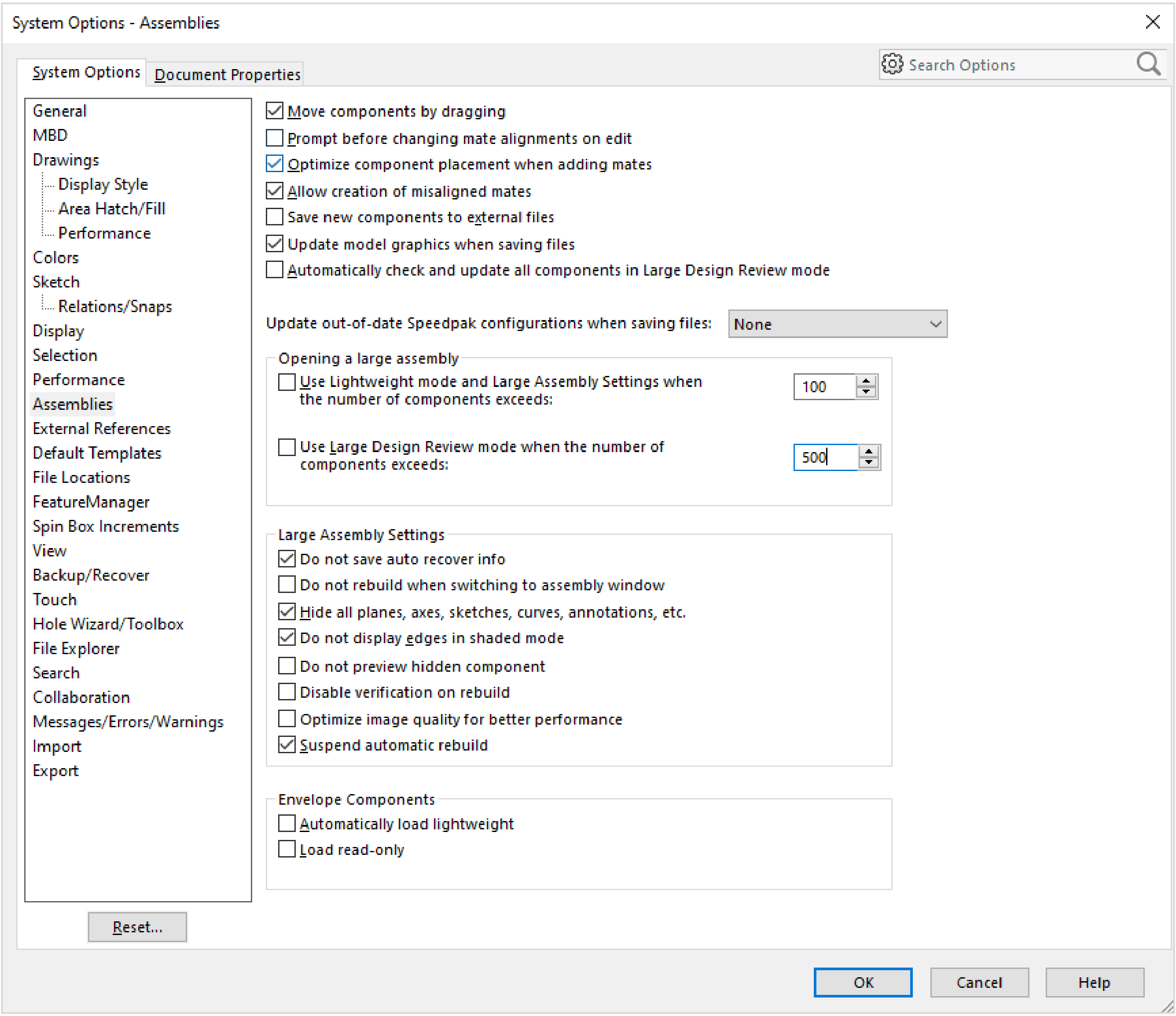
When activated, Large Assembly Mode makes certain Options settings un-editable on their System Options page or toolbar. These and are automatically set as described on this SOLIDWORKS help page: https://help.solidworks.com/2025/english/SolidWorks/sldworks/r_Large_Assembly_Mode_SWassy.htm?id=7.12.3.7 When you turn off Large Assembly Settings, the options return to their previous settings.



An assembly can be loaded with its components either fully resolved or lightweight. Both parts and subassemblies can be lightweight. When a component is fully resolved, all its model data is loaded in memory. When a component is lightweight, only a subset of its model data is loaded in memory. Primarily the graphical information for the components and their reference geometry are loaded into memory, but the features that define the part are not. These features cannot be edited or shown in the feature manager design tree. The remaining model data can be loaded on an as-needed basis.

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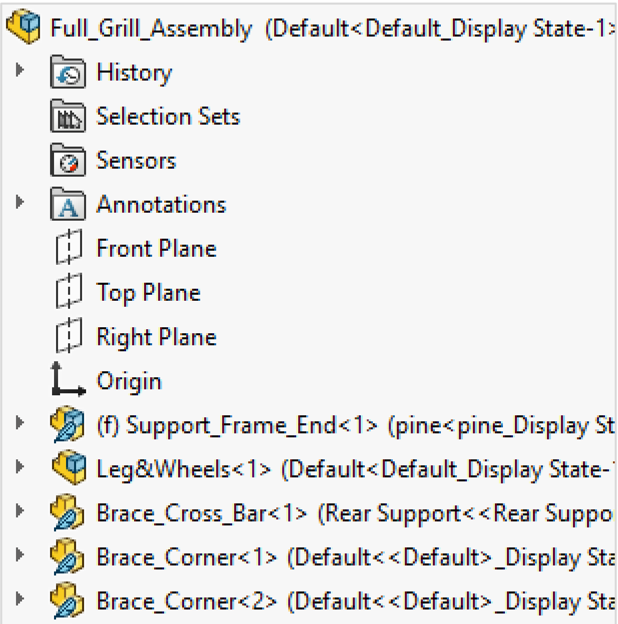
The performance of large assemblies can be significantly improved by using lightweight components. For example, loading an assembly with lightweight components is faster than loading the same assembly with fully resolved components. Assemblies with lightweight components also rebuild much faster because less data is evaluated.

To manually open an assembly with lightweight components, click *Open or File > Open*. In the dialog box, select the assembly you want to open, and then in Mode, select Lightweight. You can also set a system option to open assemblies in lightweight mode by default by clicking *Options or Tools > Options*. On the *System Options tab, select Performance. Under Assemblies, select Automatically load components lightweight.*

In an open assembly, you can set resolved components to lightweight individually. For a single component, right-click the component and select *Set to Lightweight*. This process also works for the reverse. Right-click the Lightweight component and select *Set to Resolved*. This will load the additional model data so that you can edit the features etc.

The components that are Lightweight have a blue feather on their icon as show in the image below.



For a quick summary of those and some additional opening modes for different files, see the points below:

- **Resolved** – (Parts, Assemblies and Drawings): In resolved mode, all components are fully loaded in memory.
- **Quick view** – (Parts): It opens the part only for viewing. You can able to select the configuration, but not the display state. You can move, scale, or rotate the model, but you cannot make changes. If any changes needed means switch to edit mode by the right-click in the graphics area and select the Edit command.

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- **Lightweight** – (Assemblies, Drawings): Loads only a subset of model data into memory. The remaining model data loads on need basis. Opening in lightweight mode improves the performance of assemblies and drawings.
- **Large Assembly Mode** – (Assemblies): It contains a collection of settings that improves the performance of large assemblies.
- **Large Design Review** – (Assemblies): It opens very large assemblies quickly, so whenever we need to open the entire assembly with all the components and measurements you do not need to open the entire assembly through fully resolved mode.

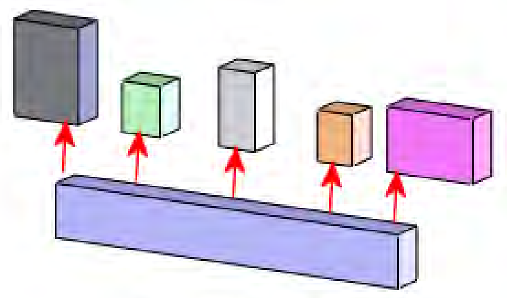
Fix Missing and Broken References

It is highly recommended that you spend time fixing missing and broken references. A good quality assembly ensures assembly accuracy and speeds up rebuilds. It is also a good idea to fix mate errors as soon as they occur. Adding mates never fixes earlier mate problems.

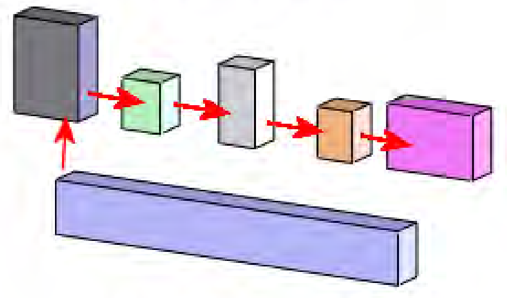
Mate Recommendations

Try to reduce the number of top-level mates in any assembly. Sub-assemblies are smaller and less complex to work with than top level assemblies. Keeping mates inside subassemblies, will also minimize top level mates and features which leads to faster solving. Also, wherever possible, minimize the use of flexible assemblies unless necessary.

Mating many components to a common reference is better than chain (chain mating is mating lots of different parts together. Parts are only controlled by the position of other parts). For example, to mate 3 screws to a flat plate, mate the bottom of the heads to the top surface of the plate rather than mating the top of screw B to the top of screw A, the top of screw C to the top of screw B, etc.



Good mate scheme



Mate scheme to avoid

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Be mindful of the types of mates you are using inside of a larger assembly. The mate performance in order of speed (fastest to slowest) is as follows:

- Relation Mates (Coincident, Parallel, etc.)
- Logical Mates (Width, Cam, Gear)
- Distance Mates
- Limit Mates

An extreme method that will improve performance is to remove but preferably suppress all mates. Especially if movement is needed and changes to geometry are not going to be made. Note: this should only be used in extreme circumstances.

Circular References

Circular Reference can occur when two or more assembly components share an external reference. For example, a rebuild of component1 requires the rebuilding of component2, but component2 then requires a rebuild of component1, and so on. These should be avoided.

Common symptoms of circular references are assemblies requiring more than one rebuild in one way or another. A Circular Reference can manifest itself as the presence of reoccurring rebuild symbol in the Feature Manager Tree. Rebuilding the assembly will cause the rebuild symbol to move from one component to another. Further rebuilds of the assembly may cause the rebuild symbol to cycle back to the original component. With each rebuild, the rebuild symbol will loop through all the affected components. However, the rebuild symbol will not be present if one of these parts is opened in a separate window.

These errors most commonly occur when mating to in-context features but can also occur when mating to component patterns.

Assembly Structure - Subassemblies

Where possible, breaking down a large assembly into subassemblies will allow for a better performing top-level assembly. Not only are subassemblies smaller and less cumbersome to work with than the main top-level assembly, but as mentioned previously it will also minimize top level mates and features (which leads to faster solving).

Ideally the subassemblies would be incorporated during the main assembly build. However, it is not always possible to have the foresight in terms of how your assembly will finally be constructed. Sometimes it may be necessary to build up your assembly using all of the relevant parts first. This however can result in poor performance. If it is not possible to construct your assemblies with as many subassemblies as possible initially, they can be added later.

You just need to right-click on the relevant part you want to be included within the subassembly and select *Form New Subassembly* as seen below.

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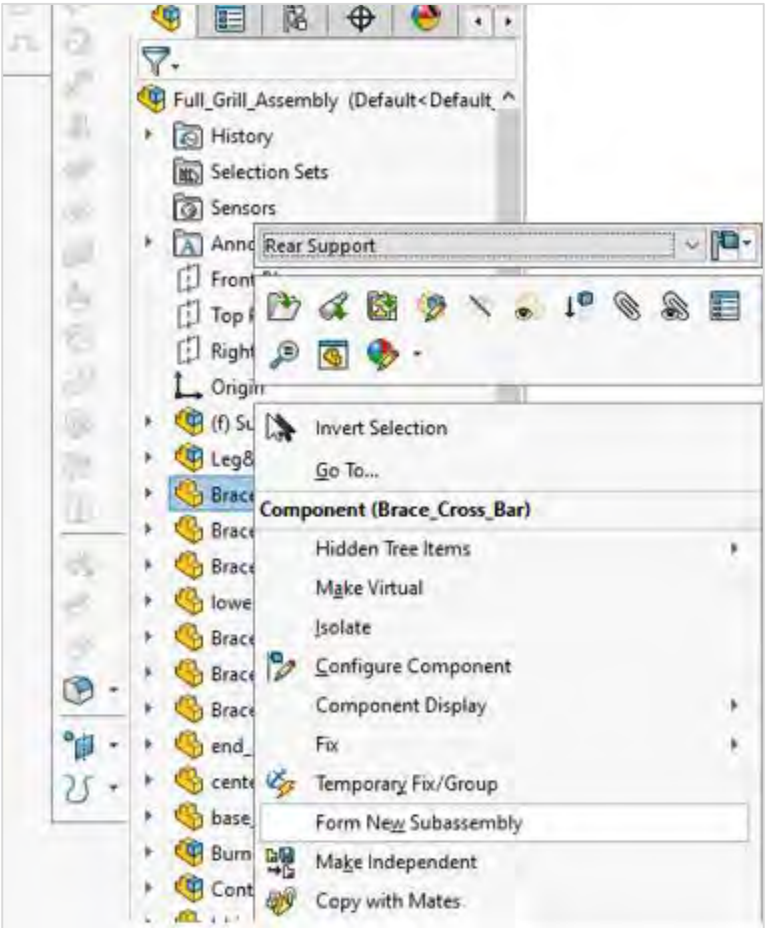

This then creates a new virtual Subassembly within the main assembly. Additional parts from either the main assembly or another subassembly can just be dragged and dropped into this new subassembly. Something to note: be careful when moving components into virtual subassemblies, as features created inside the main assembly (such as holes etc) may not be transferred. Also, components referenced via mates, patterns or through external references will be broken unless all associated parts are brought along.

SpeedPak

SpeedPak in SOLIDWORKS creates a simplified configuration of an assembly without losing any references. If you work with very large and complex assemblies, using a SpeedPak configuration can significantly improve performance while working in the assembly and its drawing.

Unlike regular configurations, where you can simplify an assembly only by suppressing components, SpeedPak simplifies without suppressing. Therefore, you can substitute a SpeedPak configuration for the full assembly in higher level assemblies without losing references. Because only a subset of the parts and faces is used, memory usage is reduced, which can increase performance of many operations.

A SpeedPak can be used when you want to insert a complex large assembly into a higher-level assembly, especially if you want to see the entire SpeedPak assembly but need to mate and

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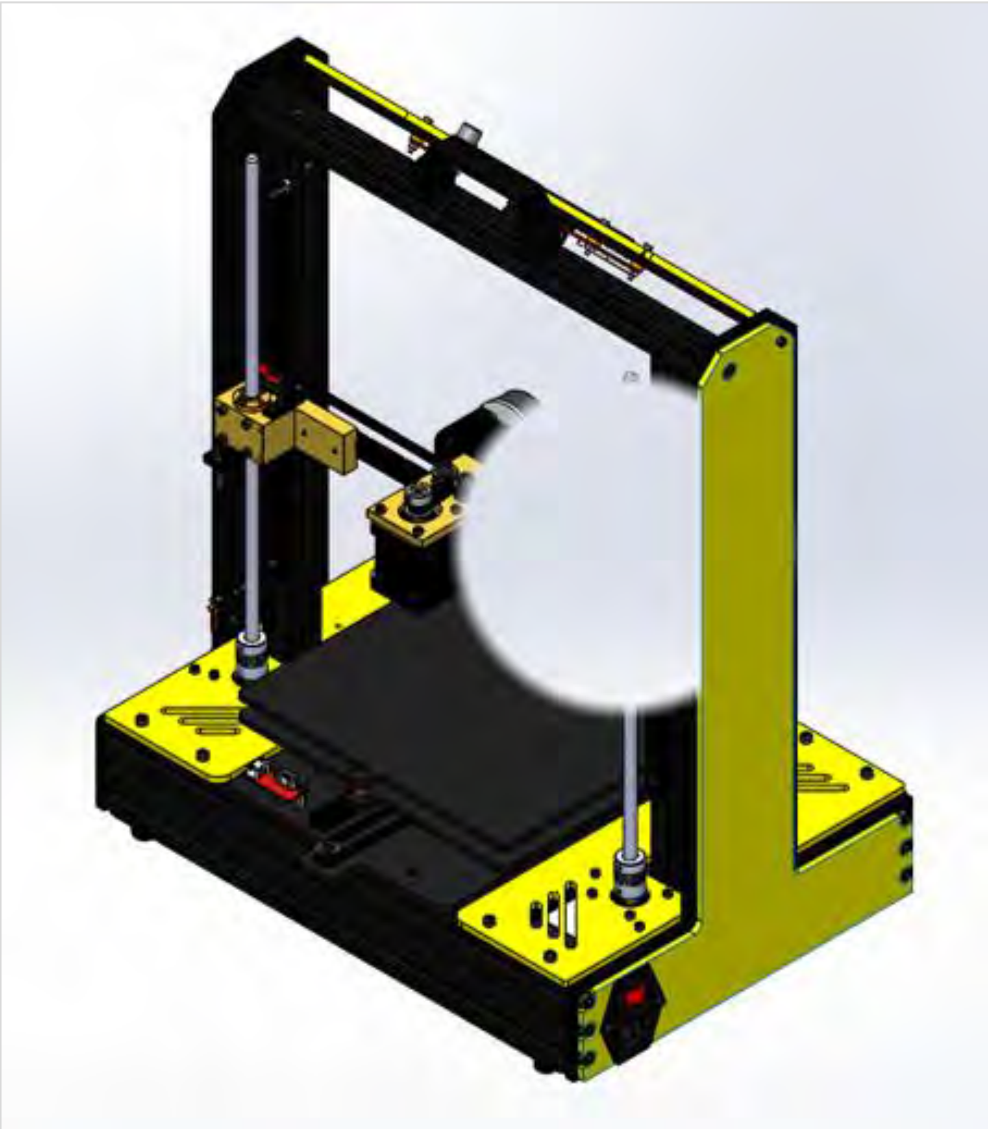
dimension only to relatively few locations. You can also use SpeedPak to facilitate file sharing. The SpeedPak information is saved entirely within the assembly file. Therefore, when sharing an assembly, you can send just the assembly file. You do not need to include component files.

In an assembly file, you can derive a SpeedPak configuration from an existing configuration. To create a SpeedPak, on the ConfigurationManager tab, under Configurations, right-click an existing configuration and click Add SpeedPak. Alternatively, in the PropertyManager select the faces, bodies, reference geometry, sketches, and curves that you want to be selectable in the SpeedPak configuration.

The image below illustrates a SpeedPak configuration. In the graphics area, when you move your pointer over the assembly, only the faces, bodies, reference geometry, sketches, and curves that you selected for the SpeedPak are visible and selectable in the region surrounding the pointer. Only the mating faces or bodies that are required are loaded into memory.



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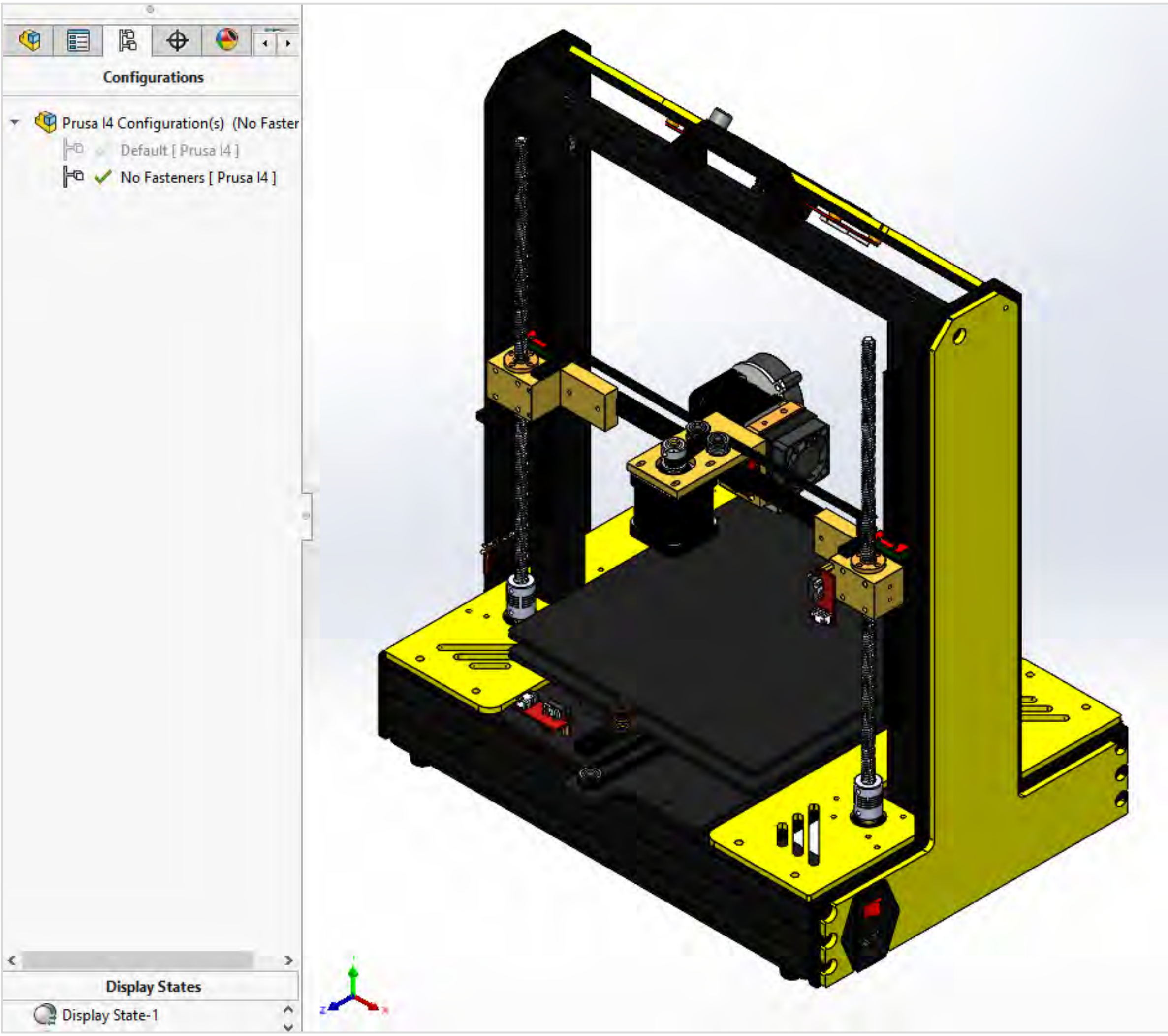
Simplified Configurations

Potentially one of the biggest contributing factors to poor performance in a large assembly could be some components in the assembly contain too much detail. Where possible, use configurations of parts that have a reduced level of detail.

Another technique you can use to improve performance is to create a configuration within the assembly, suppressing some unnecessary associated parts and subassemblies. For example, many configurations may be needed for a different combination of the components within the main assembly.

Assembly configurations allows you to suppress parts and substitute "simplified" configurations of parts for more complex finished models, which frees up more RAM. Just make sure when simplifying parts be sure you are not suppressing surfaces that are needed for mates.

The image below illustrates an assembly with configurations. To improve performance, a configuration excluding toolbox fastener has been created.



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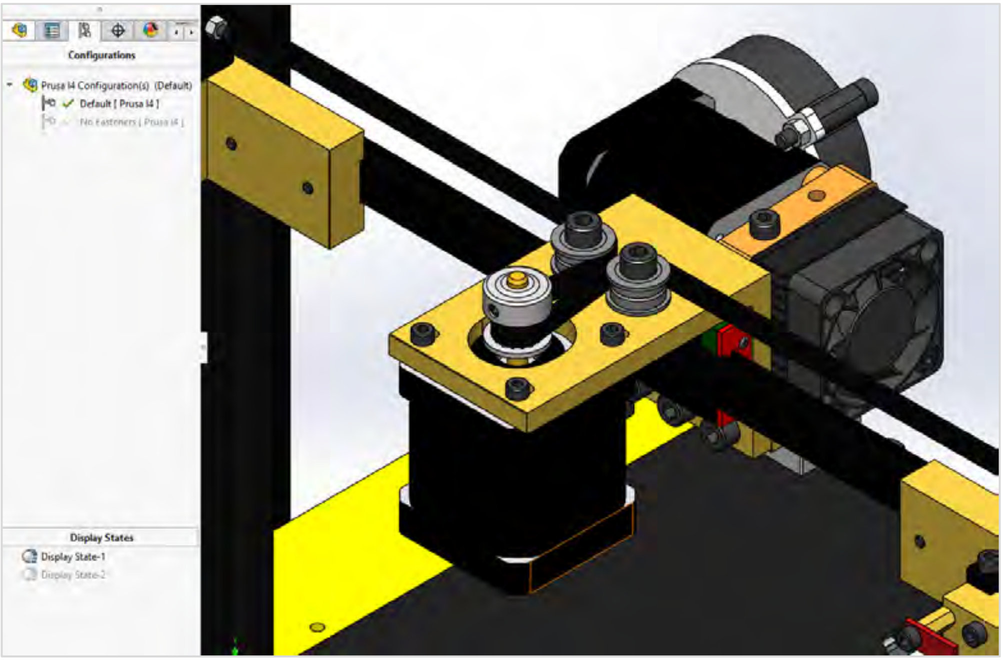
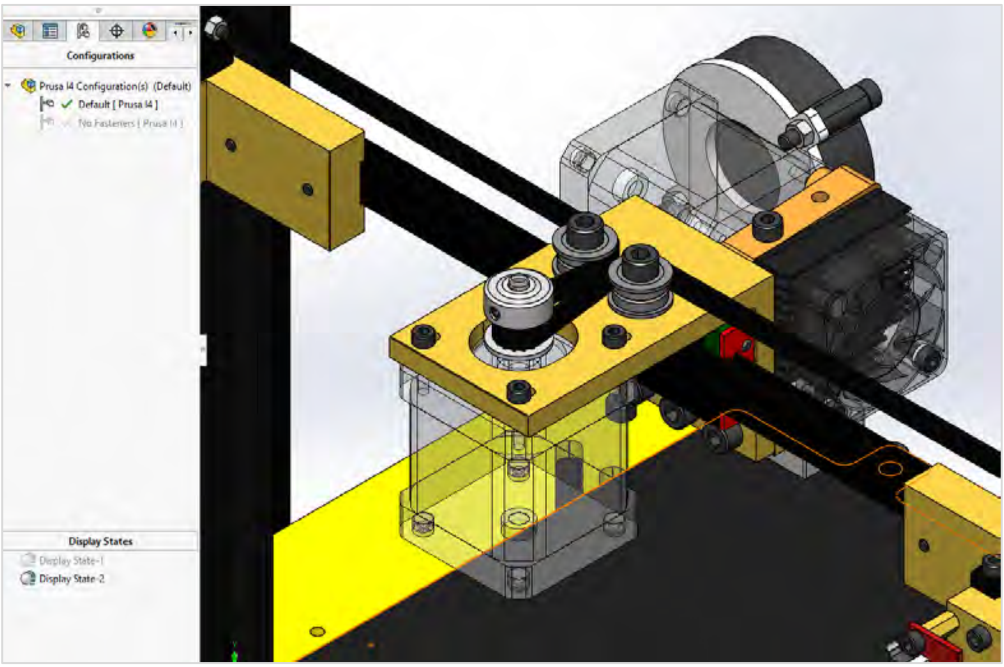
Display States

Configurations should be used if you are going to keep the configuration simplified in the assembly or if it is for design purposes (i.e. a new design configuration). Configurations should not be used to Hide/Show parts because configurations always require a rebuild (Display States never require rebuilding). To Hide/Show components Display States should be used instead.

Display states are used to showcase a component differently within one assembly. In Display States we can modify the display of a component by changing the Transparency, Appearance, Display Mode, and Hiding bodies/components as mentioned previously.





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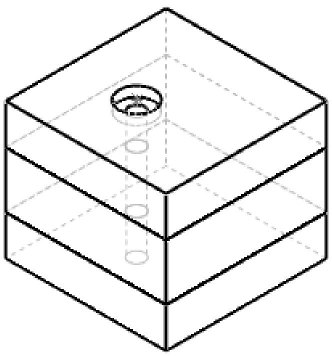
Defeature

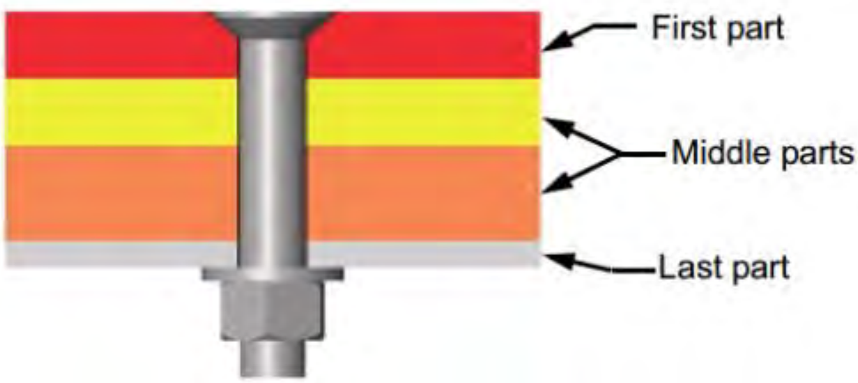
Another technique similar to creating a simplified configuration is the Defeature tool. This tool removes the selected faces, features, etc. For example, the performance of large assembly with hundreds of versions of the same bolt can be greatly improved by removing the threads. This saves the assembly from generating the graphic triangles of the threads (of the hundreds of iterations) which can't be seen anyway.

The file can then be saved as a "Dumb Solid", which is a file without any feature definition or history. This is also a great for protecting intellectual property whilst maintaining a good understanding of the geometry. Defeature can be found via: *Tools > Defeature*.

Hole Series

The SOLIDWORKS Hole Series is an assembly feature that creates hole features in individual components of the assembly. A Hole Series will cut each unsuppressed component in the assembly that intersects the axis of the hole (the components do not have to touch). Unlike other assembly features, the holes exist in the individual parts as externally referenced features (in-context). If you edit a Hole Series within the assembly, the individual parts are modified to suit. It can also automatically add fastener stacks using Smart Fasteners. <u>Note:</u> this feature does create external reference because a hole sketch point is created.





This feature guarantees holes will line up and correct fasteners will be added to holes correctly. In a large assembly this will help rebuild times. Only one assembly level feature will need to be rebuilt as opposed to many part level features.

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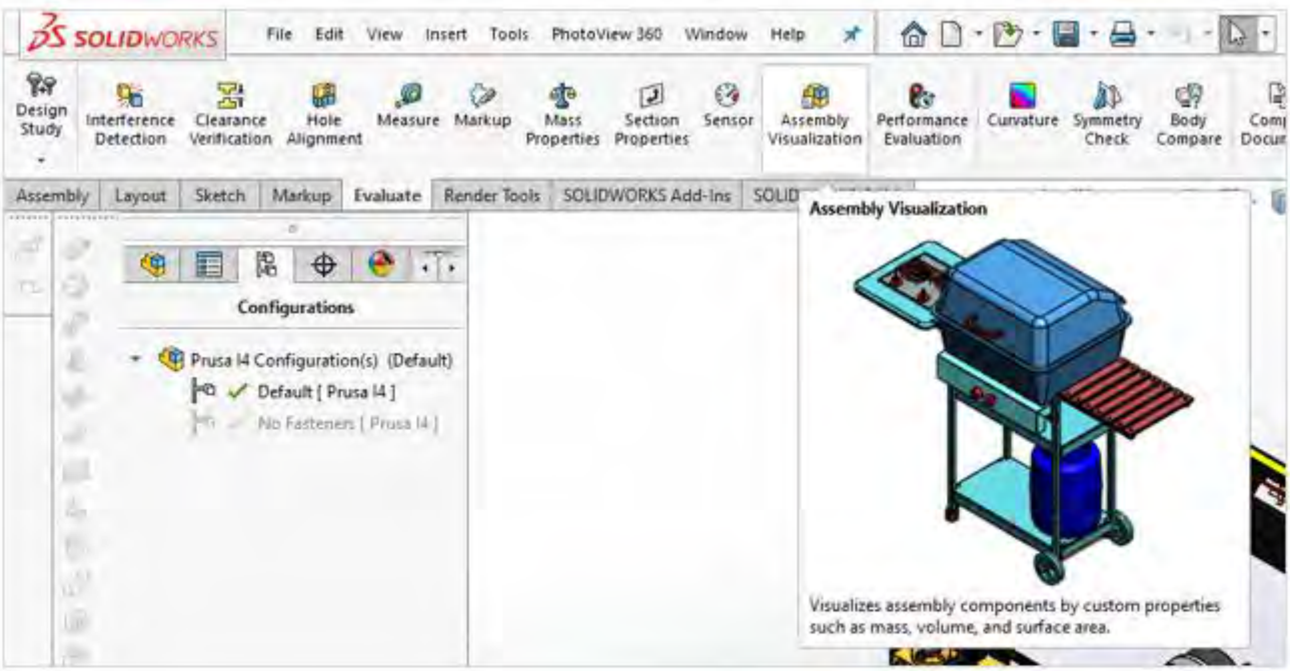
Assembly Visualisation

Assembly Visualization provides different ways to display and sort an assembly's components in a list and graphically in the graphics area. You can use it to illustrate basic numerical data such as component mass, density, and volume.

This tool is also very useful in improving the performance of large assemblies in SOLIDWORKS. There are three Assembly Visualization properties that can help to find the likely causes of slow performance in a large assembly. These are: SOLIDWORKS Rebuild Time (SW-Rebuild Time), SOLIDWORKS Open Time (SW-Open Time) and Total Graphics Triangles (Graphics-Triangles).
The tool can be found on the Evaluate tab of the CommandManager.



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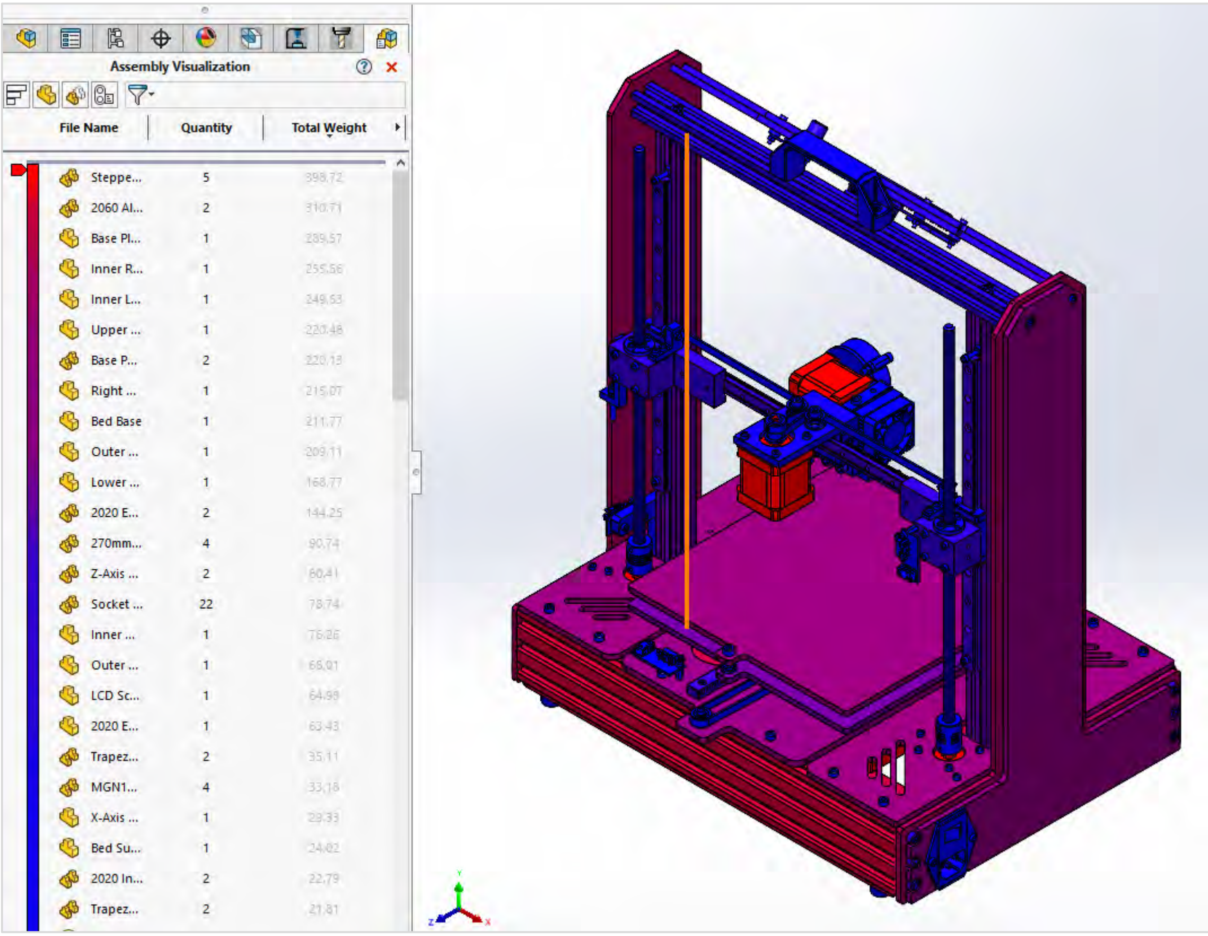
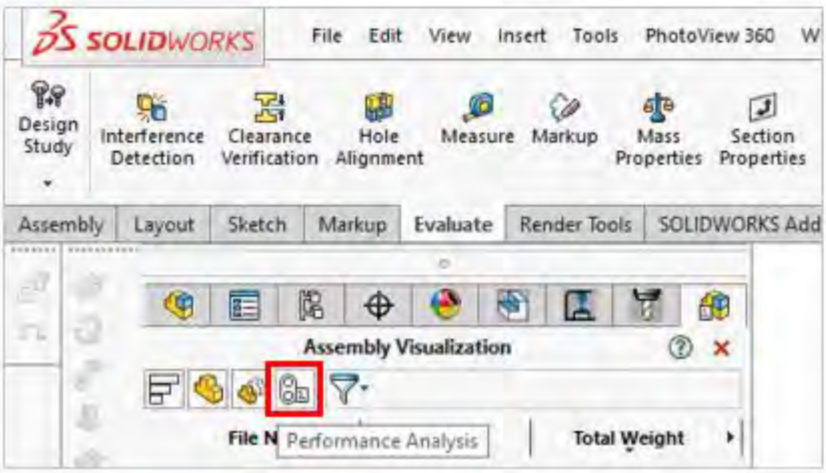
By default, the visualisation tool will show standard properties such as quantity and weight when the tool is first opened. This is shown in the image below.



To use the properties that help diagnose likely causes of performance issues, you can choose the Performance Analysis button. This brings up the 3 diagnostic properties.



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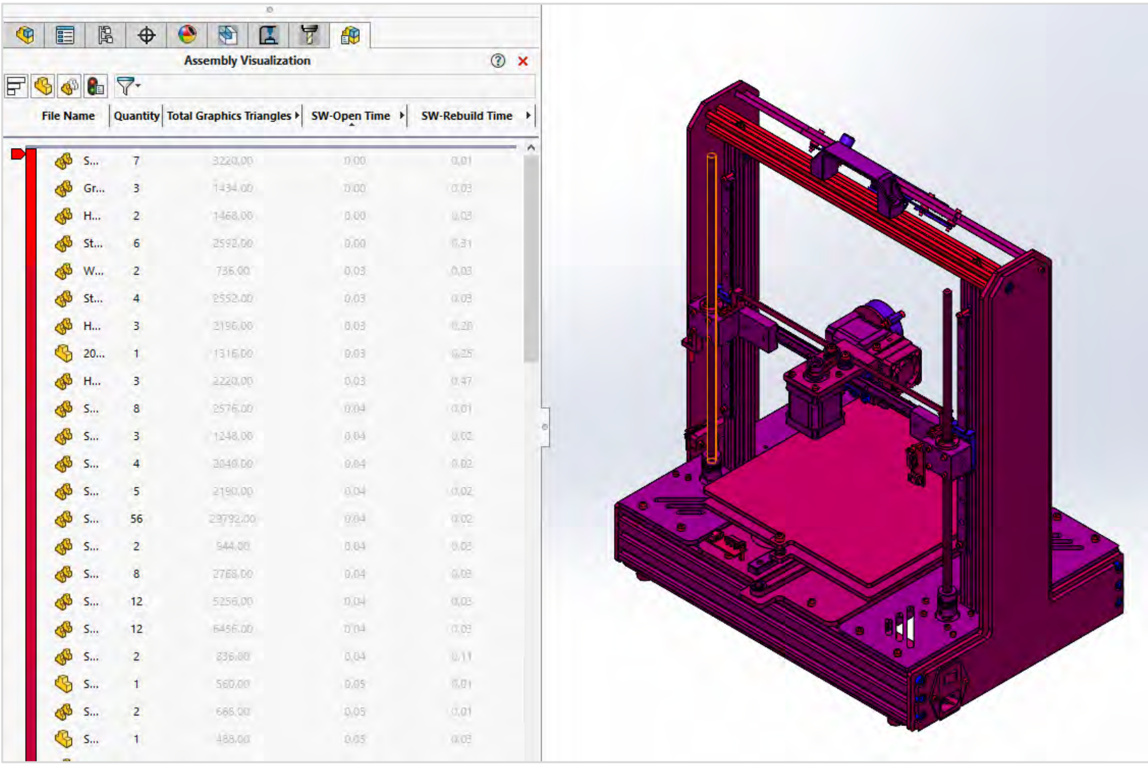
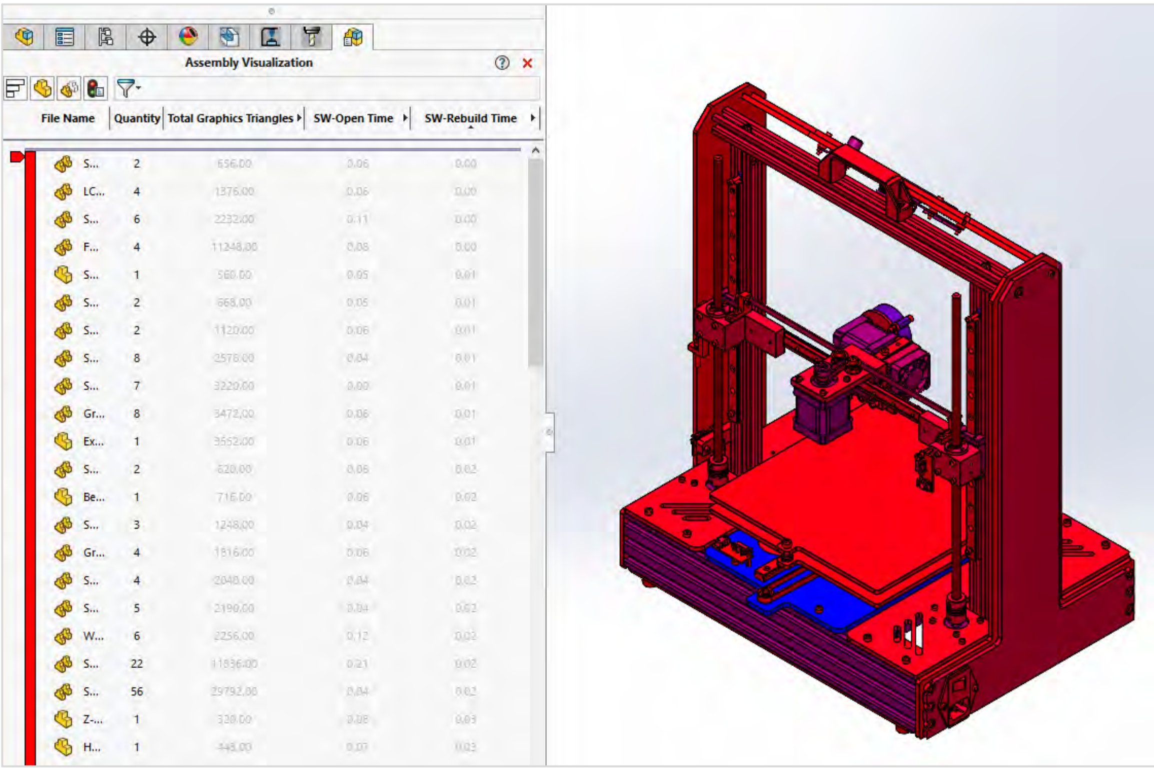
The component list can then be sorted as per the property values selected. The colours of the components in the graphics area will also update accordingly. The image below shows SW-Open Time graphically. The parts in red take the longest to open so we can narrow down slow opening to those components. We can edit them if needed.





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Similarly, if we sort by SW-Rebuild Time we can identify the parts that take the longest to rebuild. By being able to see these properties graphically, we are quickly and easily able to identify the areas of the assembly we need to analyse and work on to make the assembly perform better.

General Guidance for Parts Used in Large Assemblies

Parts are the foundation of any assembly and drawing. If we use the best methods and workflows to improve performance at the part level, this will propagate through to the assemblies and drawings, which collectively, can have a big impact on opening times and performance. We will now go through some modelling techniques that you can apply to new and existing parts.

Note: Using the performance evaluation as shown above can help quickly identify problem components which can potentially be improved upon to increase performance.

Fix Rebuild Errors

Errors in parts, sub-assemblies or top-level assemblies will slow down SOLIDWORKS. This should be the first step to take if you have errors within the assembly, even if you are not suffering with a drop in performance.

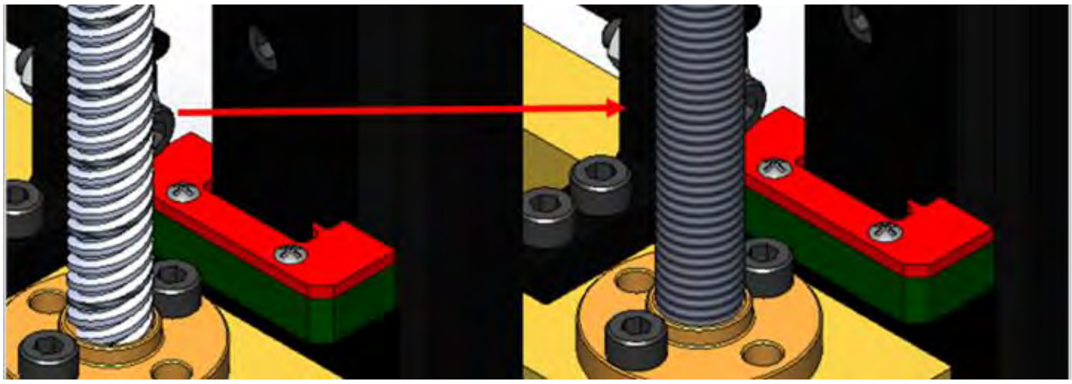
Fully Define Sketches

It's always good practice to fully define all sketches, but SOLIDWORKS solves fully defined sketches faster. Fully defined sketches entities will appear black. Underdefined sketches entities will have some degrees of freedom – these appear blue.

Do Not Model Threads Unless Necessary

You should remove or supress non-essential threads or replace them with a cosmetic thread if a visual effect is needed. Cosmetic threads describe the attributes of a specific thread, so you need not add real threads or add actual thread geometry to the model. They contain all the essential thread information.

It is recommended that only functional threads should be modelled. Even then, these should be suppressed at the top-level assembly where possible. You can use configurations to create simplified versions as needed.



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Physical thread

Feature Order	Time %	Time(s)
CirPattern1	85.13	6.80
Cut-Sweep1	12.32	0.98
Cut-Revolve2	0.79	0.06
Cut-Revolve1	0.78	0.06
Sketch3	0.58	0.05
Helix/Spiral1	0.20	0.02
Sketch 5	0.20	0.02
Sketch 1	0.00	0.00
Boss-Extrude1	0.00	0.00
Sketch2	0.00	0.00

Physical thread with Geometry Pattern

Feature Order	Time %	Time(s)
CirPattern1	71.60	3.03
Cut-Sweep1	23.25	0.98
Cut-Revolve2	1.84	0.08
Cut-Revolve1	1.84	0.08
Sketch3	1.09	0.05
Helix/Spiral1	0.38	0.02
Sketch 1	0.00	0.00
Boss-Extrude1	0.00	0.00
Sketch2	0.00	0.00
Sketch 4	0.00	0.00

Cosmetic thread

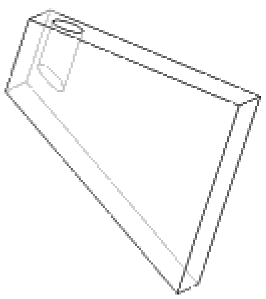
Feature Order	Time %	Time(s)
Sketch3	75.41	0.05
Sketch 5	24.59	0.01
Sketch 1	0.00	0.00
Boss-Extrude1	0.00	0.00
Sketch2	0.00	0.00
Helix/Spiral1	0.00	0.00
Cut-Sweep1	0.00	0.00
CirPattern1	0.00	0.00
Sketch 4	0.00	0.00
Cut-Revolve1	0.00	0.00

We can see from the three different performance evaluations above the difference each step can make to rebuild times.

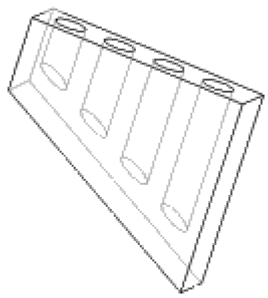
Minimise Any Unnecessary Details

- **Fillets** – If possible, fillets should be added into the part last. Equal fillets should also be combined into one feature where possible. These should be suppressed in a new configuration when not necessary.

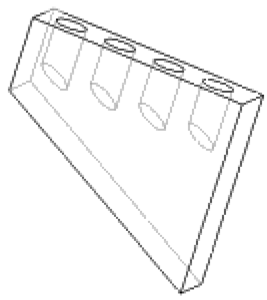
- **Patterns** – Avoid showing large complex patterns at top level assembly (E.g. Fill pattern) and try to use ''Geometry Pattern'' option. A geometry pattern copies, but does not solve, the patterned feature. Each instance is an exact copy of the faces and edges of the original feature. They rebuild much faster as End conditions are ignored.



Offset from Surface cut feature



Geometry Pattern Not Selected – the end condition is continued



Geometry Pattern Selected – the end condition is ignored

- **Springs** – Do not model spring detail or helixes. Instead use a cylinder to visually represent them.

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- **Perform Performance evaluation on parts** - This should be performed on imported parts is important as some can have excessive amounts of triangles/faces which will slow things down. This is especially relevant for imported parts such as PCBs.

- **Model appropriate assemblies as parts. E.g. Bearings** – This can be done using Save as inside of the assembly and selecting sldprt (SOLIDWORKS PART) instead of the default option of assembly.

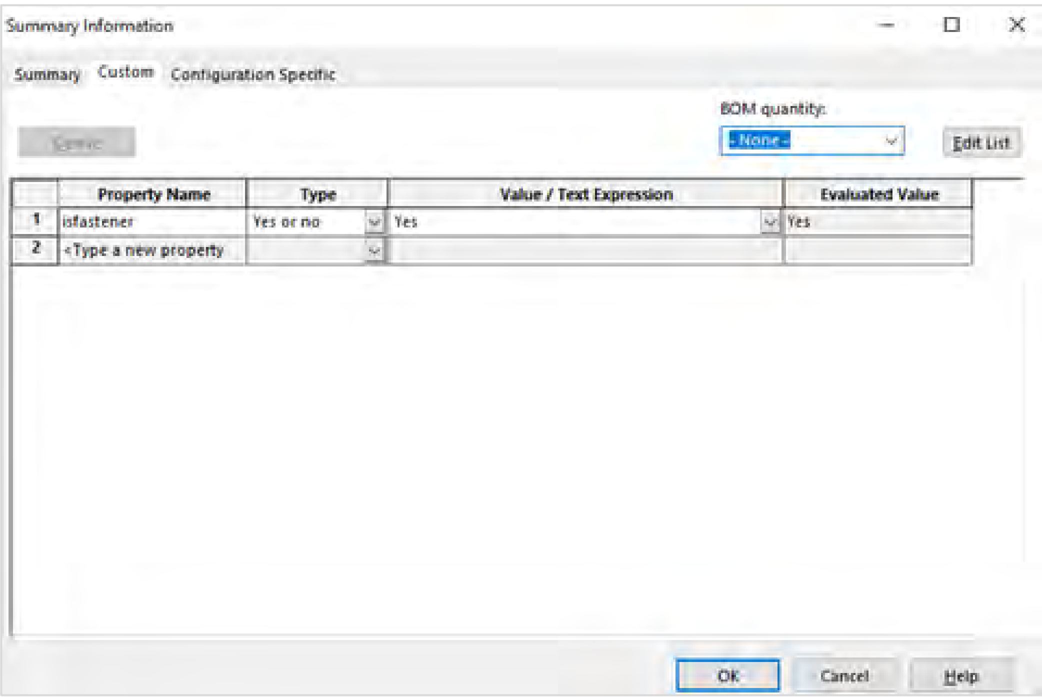
- **Avoid using text for features** - Simplified sketches can be used as an alternative if the detail must be shown.

- **Avoid using decals if possible** – If parts need decals for important illustrative purposes, create display states with them hidden to be used in the top-level assembly.

Document Templates

Use different templates to store document settings for components. This will help to speed up interference checking. By using a template to utilize ''*IsFastener*'' for example, you can set the components to be fasteners. With this property, SOLIDWORKS interference check will ignore the interference created for these components i.e. the interference created by the thread being altered to a cylinder.

This can be done by going into the properties of a part, creating a custom property ''*IsFastener*'' and setting this to yes. Or setting up the property name as part of the template, then only yes or know will have to be selected.



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Imported Geometry

To save imported geometry as SOLIDWORKS file types:

- If using FeatureWorks, only create features that are necessary i.e. those that you wish to edit. The FeatureWorks software recognizes features on an imported solid body in a SOLIDWORKS part document. Recognized features will be the same as features that you create using the SOLIDWORKS software. You can edit the definition of recognized features to change their parameters. For features that are based on sketches, after you recognize the features, you can edit the sketches from the SOLIDWORKS FeatureManager design tree to change the geometry of the features.
- If you do not need to edit any features, just leave the model as imported geometry. However, it is important to make sure all surfaces are fully knitted together to create a good quality solid.

To leave imported geometry as its native file types:

- Use 3D Interconnect to work with third-party native CAD data in SOLIDWORKS. 3D Interconnect allows you to insert proprietary CAD data directly into a SOLIDWORKS assembly without converting it to a SOLIDWORKS file. It also keeps its associative link to the original part. Using this function will mean that the part won't be converted to potentially hundreds of surfaces, reducing file size and potential for faulty surfaces.

General Guidance for Large Assembly Drawings

We can improve the performance and speed that large assembly drawings can be opened and worked on. Remember, all parts/assemblies contained in the drawing must be loaded into memory, so usually the best place to start when looking to improve the performance of drawings lies within the parts and assemblies it is referencing. If you have read through those topics and applied them, then we can now look at how to improve performance within the context of the drawing environment.

Opening Modes

Opening Mode is also a relevant factor when it comes to drawings. We have multiple options available when opening a drawing. Resolved, Lightweight, Detailing and Quick View.

Just like the context of parts and sub-assemblies, drawings can be loaded fully resolved or lightweight. Resolved bringing through all data related to the part/sub-assembly and lightweight, bringing through a sub-set of this data with the remaining data loaded in when needed. It is important to consider whether you want to open your assembly in Resolved mode. The majority of the time this will load in large amounts of unrequired data, slowing down the drawing. Choose a more suitable mode for what you need to do. By default, assemblies in large assembly mode will load drawings in lightweight.

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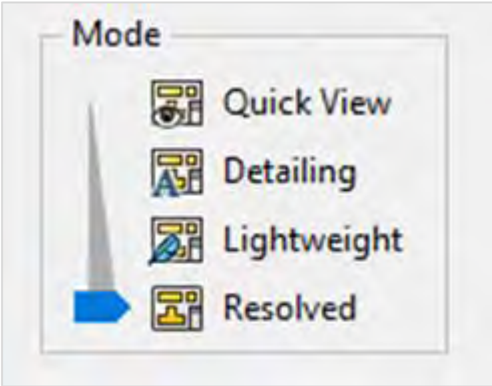

Detailing Mode can offer a huge improvement to performance to large, complex drawings. Unlike other modes, detailing mode does not load up the model. This means it will open much faster and can be worked on with little to no performance compromises.

Detailing Mode does however have some restriction. Some capabilities have been listed below

• Add annotations such as notes, balloons and more
• Print and Save As
• Create sketch entities
• Add and Edit dimensions

When in detailing mode, anything not greyed out in the command manager can be used. A great tool which can be used to review a design and make minor edits. When opening in Quick View mode you cannot make any edits to the drawing as it will open a read only copy. Useful for when you need to quickly view drawings, but don't need to make any changes. For example, a manager to confirm the drawing is correct and change the state of the file to approved/released through PDM.

Another option comparable to detailing mode, is detached drawings. Saving a drawing as a detached drawing splits all of the links to the parts, allowing it to be opened without access to them. It is similar to detailing mode in its capabilities but available in versions prior to 2020.

Also, you can select to only open specific sheets while in the open dialog box. This can be useful in only viewing sections of the drawing which you need to.

The mode you select to open the drawing in should be dictated by what you want to achieve. Selecting the correct mode can improve the opening time of these drawings.

Performance Evaluation

Just like within an assembly, we can utilize the Performance Evaluation Tool. This can help to identify the areas which are taking the most time to load in the drawing environment. This isn't restricted to the drawing environment; it also points us towards the assemblies/parts contained within the drawing that need attention/have the largest loading times etc. We can also use this to identify any problem parts which contain complex features, excessive faces or may even be corrupt.

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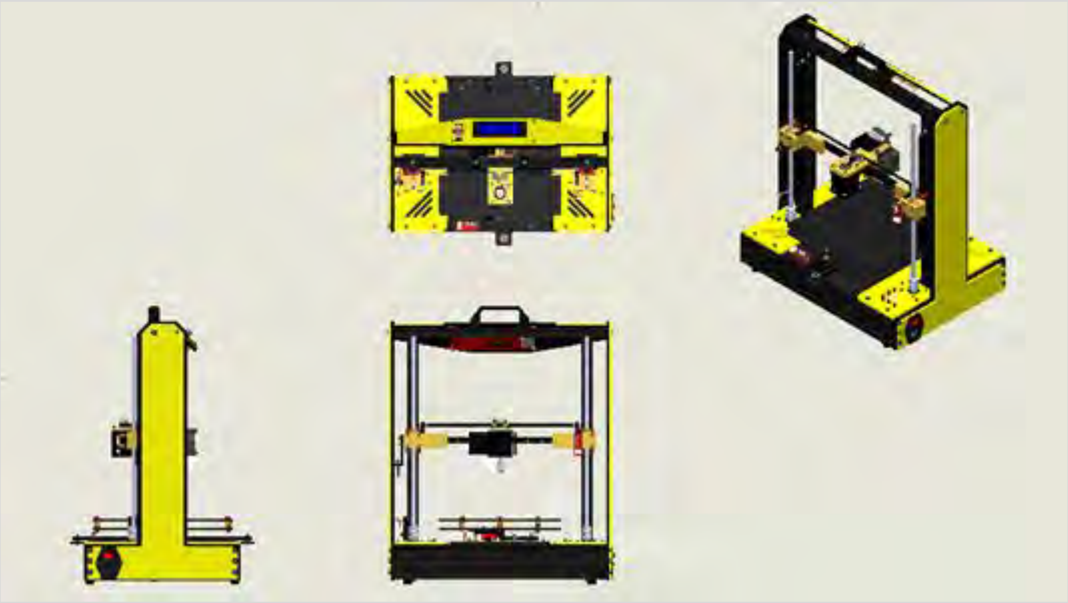
Drawing Views

- Always use Draft Quality views where possible - Only minimum model information is loaded into memory. Some edges may appear to be missing and print quality may be slightly degraded.

- Use shaded view to layout drawings onto the sheet - You can use system setting to alter the default Tools>Options>Display Style, Shaded. Shaded only loads graphical data whereas shaded with lines loads graphical data as well as parametric data.



- Use less views per drawing sheet - Use multiple sheets or drawings instead.

Configurations and Display States

Configurations should be utilized to minimize un-necessary detail in drawings. However, there may need to be a trade-off. The number of different configurations per drawing should be limited. Each configuration is treated as a sperate part, so the more configuration views within an individual drawing, the more time it will take to update. Try using separate drawings if you have many different or separate configurations.

Display States are preferred over Configurations if you are just hiding components/sub-assemblies. Configurations must be rebuilt whereas display states don't. If you have configurations set up specifically just to hide parts, use display states instead. This will increase performance greatly. For more information see the assembly configuration section.

For Bill of Materials, it is recommended that you use the BOM configuration source. This can be set within the BOM properties, just select each configuration you wish to display. The balloons link directly to the BOM and BOM numbering can be retained as well as balloon numbers effectively.

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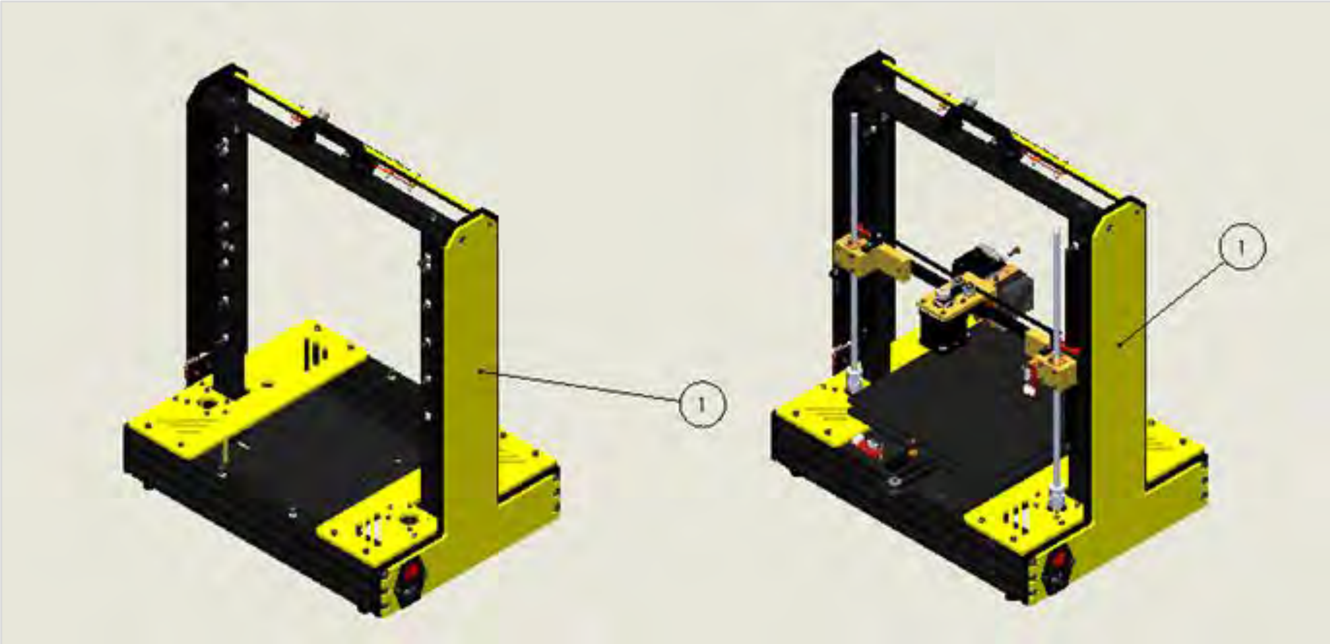

The image above shows two different configurations displaying the same balloon number for the same part.

Drawing Sheets

It is important to consider how many drawing sheets and views you are using. Where possible, use less views per drawing sheet. It is better to use multiple sheets instead. However, this could result in a drawing with many sheets. It is even better to consider using separate smaller drawings instead of one larger file with countless sheets. If this isn't an option, try opening only the required sheets in the drawing file.

Where possible, try to keep the number of files that a sheet of a drawing is referencing to a minimum (best practice is 1). However, this is a little harder when large assemblies are considered. One file per drawing sheet means SOLIDWORKS isn't referencing multiple locations/files at once to display an individual sheet.

Suspend Automatic Rebuilds

For large assemblies, select this feature to prevent the rebuild of the assembly every time you switch between opened models. *Tools>Options>Assemblies>Large assembly settings>Suspend automatic rebuild.*

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Large Assembly Hardware Requirements Guidance

It goes without saying that having a computer capable of running SOLIDWORKS is essential for ensuring smooth performance and stability, especially where large assemblies are concerned. The following section details recommended specifications. For additional information, visit our website.

Where do I find hardware requirements for running SolidWorks?

Current hardware requirements are listed on the SolidWorks website under System Requirements. https://www.solidworks.com/sw/support/SystemRequirements.html

How much RAM do I need?

The most important requirement for handling large assemblies is having enough RAM. The minimum requirement of RAM indicated on the SOLIDWORKS System Requirements page is at least 16GB. However larger models and simulation studies may require more memory. To avoid running out of memory for growing models, consider installing 32GB. It's best to split this between fewer memory chips to leave room for possible expansion in the future, for example install two 16 GB memory chips.

Will I need a 32 bit or 64-bit machine?

The main reason to go to 64bit machines (and 64bit OS) is for more memory. 64-bit machines will be faster for assemblies that exceed the 2Gb limit of the 32bit systems. 32-bit machines have a 2Gb RAM size limit – once exceeded, you will be swapping to disk, which is much slower.

What Processor (CPU) is recommended?

SOLIDWORKS supports both Intel and AMD processors. The rebuild process in SOLIDWORKS is inherently linear (parent/child relationship of features) and therefore can only use a single core. Rebuild performance can be increased by having a faster clock speed of the CPU.
Be aware that the Intel and AMD Boost speeds represent the max speed that can be hit for a period. However, it may not reach these speeds in all scenarios or for an extended period. It must remain in specification limits for workload, temperature and power.
Keep in mind that the fastest machine is no substitute for good design practices. Minimizing in-context references, utilizing large assembly modes and other performance tools will reduce the load on the processor.

What type of Hard Drive is recommended?

For best performance, a Solid-State Drive (SSD) can provide up to 10x faster performance for open/save tasks compared to a standard Hard Disk Drive (HDD). SSDs have come down in price over the years and are worth the investment. Also note that opening files from a local SSD drive compared to opening from a standard disk drive on a network server can be up to 100x faster. Be sure to maintain enough hard drive space after installing for Windows to run effectively. Keep at least 20GB or 10% of your hard drive capacity as free space.
For even greater performance, you can consider using a NVMe/PCIe SSDs rather than standard SATA SSDs. These have a much faster interface, but are more expensive.

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Also, you may wish to research into using RAID storage (Redundant Array of Inexpensive Disks) to improve performance and security.

What type of Graphics Card is recommended?

A good graphics card is one of the most important elements for running SOLIDWORKS efficiently, however, it's important to make sure there is an effective balance between Graphics Card and CPU. A high-end graphics card will not run effectively if paired with an entry-level CPU as it would be waiting around for data to be passed from the processor.

To run SOLIDWORKS at its optimal and help prevent system instability, requires the use of a Professional Grade Workstation Graphics Card such as those in the links the below.
The card must be capable of running the **OpenGL** engine to function correctly and have the most suitable SOLIDWORKS-certified Software Drivers installed as these have undergone more stringent testing. A graphics card with hardware OpenGL acceleration will provide superior performance and stability, especially in 3D model viewing (refresh, rotate, zoom, pan). Historically many people would have recognised the branding of Nvidia Quadro or ATI/AMD Firepro but as new cards are released branding changes.
The manufacturer websites below show the latest Graphics Card ranges and specifications.

Nvidia RTX ADA Series Nvidia Workstation Cards
AMD Radeon Pro AMD Workstation Cards

Currently the branding of certain workstation cards can be difficult to determine, **please be sure to check the most recent currently supported cards on the SOLIDWORKS hardware certification website: Solidworks.com/support/hardware-certification**. Always ensure that you are using the most appropriate graphics driver for your card. You will also need a certified card to take advantage of RealView graphics.

While a high-end graphics card sounds like it will give much better performance, the amount of money involved won't see a dramatic boost. Investing in a faster CPU and SSD drive will provide much greater gain in performance.
Tip: If your assemblies are running close to your max available RAM, then a lower-level graphics card would be better because a 512Mb card will map to 512Mb of RAM.

Although it is possible to run SOLIDWORKS on mainstream "Gaming Graphics Cards", these types of cards are not always suited to the workloads they would be undertaking in a professional environment. These cards are optimised for performance in gaming and use Software Drivers that are tested for stability in DirectX rather than OpenGL.

Users may also experience frequent graphical glitches and find that features of the software such as Realview Graphics will not function correctly, if at all. SOLIDWORKS therefore does not officially support the use of these types of graphics cards and may not be able to help if there are problems with the software

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Conclusion

Utilizing this guidance will increase opening speeds and performance while working on models within the context of assemblies, drawings and in some cases, parts. All the methods above can be implemented into both new and existing projects. Although all the above point may not apply to each of your assemblies, you can implement what does. Minimising the amount of unnecessary information displayed, loaded and read by SOLIDWORKS within an individual assembly or drawing will improve performance.

For additional information and guidance, please don't hesitate to use our support services.

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