

December 12, 2017

How to troubleshoot slowness in SOLIDWORKS when working with Large Assemblies

The Fisher Unitech team knows that working with large assemblies can sometimes lead to SOLIDWORKS slowing down. We have created a 12 part video series to address these slowdowns and to share some great tips and tricks on how to avoid them altogether. This checklist is a supplement to our video series on working with Large Assemblies, which can be found on the Fisher Unitech youtube channel.

1. Faster data transfer speeds will result in assemblies opening more quickly

- ACTION ITEM – Cut the network out of the loop**
- ACTION ITEM – Switch from mechanical hard drive to SSHD**

2. Less data loading into RAM will result in assemblies opening more quickly

- ACTION ITEM – Reduce file size of large part files**
- ACTION ITEM – Utilize the option to LOAD COMPONENTS LIGHTWEIGHT**

3. Follow best practices for file conversion and error management

- ACTION ITEM – Make sure every part and sub assembly is up converted**
- ACTION ITEM – Resolve/Remove any errors in mates, features or imports**

4. The setting for SYSTEM OPTIONS>FILE LOCATIONS>REFERENCED DOCUMENTS can make your assembly slower to open/save

- ACTION ITEM – Examine file locations using FILE>FIND REFERENCES**
- ACTION ITEM – Evaluate your assembly using Performance Evaluation
and/or Assembly Visualization**

5. Certain system settings will decrease the taxation on your graphics card

ACTION ITEM –

Heads up toolbar settings:

- DISPLAY- Turn off REAL VIEW GRAPHICS, SHADOWS
- IN SHADED MODE, AMBIENT OCCLUSION, PERSPECTIVE
- HIDE/SHOW Menu- Hide All Types
- BACKGROUND- Simple static image
- MODEL DISPLAY- Shaded with NO EDGES shown

System Options:

- DYNAMIC HIGHLIGHT- Turn off (2 places)
- TRANSPARENCY QUALITY - Dial down (2 places)
- EDIT PART IN CONTEXT- Keep all parts OPAQUE
(Set color options for assembly in context edit)
- LEVEL OF DETAIL- Set to FASTEST setting
- VIEW ANIMATION and HIDE/SHOW ANIMATION- Turn off
- LARGE ASSEMBLY MODE- Understand how to use it

6. Overly detailed parts will INCREASE the taxation on your graphics card

ACTION ITEM –

Identify Overly detailed Parts :

- YOUR EXPERIENCE- Use what you know about the project -
This is especially true when you are building around imports
- ASSEMBLY VISUALIZATION – Start with HIGH QTY parts
Then go back and get the HIGH GRAPHICS TRIANGLE parts

Create SIMPLIFIED Configurations:

- Use the DERIVED CONFIGURATION option
- Consider affects/impact on MATES in assemblies
- Create a SIMPLIFIED configuration at each Sub-Assembly level
- For HIGH QTY Parts create a “Detailed” derived configuration

7. Addition Tools for Large Assembly Management

ACTION ITEM – Learn about TOOLS>Find/Modify>Simplify

ACTION ITEM – Learn about Large Design Review Mode

Conclusion

When it comes to working with a large assembly there can be several factors that adversely affect the SOLIDWORKS performance. By following the seven steps above you should see and feel an improvement to your overall SOLIDWORKS assembly performance.

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If there are any questions/comments on this tech tip, please contact the Fisher Unitech SOLIDWORKS Technical support team at the following:
Phone: 1-800-816-8314
email: Support@fisherunitech.com