

Sample SOLIDWORKS Test Plan

Description

The purpose of these tests is to perform common operations in SOLIDWORKS and PDM that users complete in a typical workday and identify if the changes made to your computing environment introduced any new problems that were not present prior to the change. Users who perform these tests shall report the details of their test results by describing whether the test procedure resulted in any error messages or unexpected software behavior in a shared document.

Refer to the following [KB article for best practices](#) for developing an overall Test Plan for SOLIDWORKS:

Test Procedure

Create a New SOLIDWORKS Part

1. Launch SOLIDWORKS.
2. Go to Tools > Add-Ins and make sure SOLIDWORKS PDM is enabled as an Active Add-In.
3. Go to File > New.
4. Make a new part.
5. Create a simple extruded block or other simple shape.
6. Save the part file in the test folder in the PDM Vault.

Create a New SOLIDWORKS Part Drawing

7. Go to File > Make Drawing from Part.
8. Add drawing views.
9. Add dimensions.
10. Add notes.
11. Save the drawing file in the test folder in the PDM Vault
12. Go to File > Save As and save the drawing as a PDF file.
13. In SOLIDWORKS, go to File > Print and save a PDF file of the drawing.

Create a New SOLIDWORKS Assembly

14. Go back to the simple part that you created in SOLIDWORKS.
15. Go to File > Make Assembly from Part.
16. Save the assembly file in the test folder in the PDM Vault.
17. Insert components into the assembly.
18. Mate the components together.



Create a New SOLIDWORKS Assembly Drawing

19. Go to File > Make Drawing from Assembly.
20. Add drawing views.
21. Add dimensions.
22. Add notes.
23. Add a Bill of Materials.
24. Save the drawing file in the test folder in the PDM Vault.
25. Go to File > Save As and save the drawing as a PDF file.
26. In SOLIDWORKS, go to File > Print and save a PDF file of the drawing.

PDM Check-In of New SOLIDWORKS Files

27. Check-in the newly created part, assembly, and drawing files into the PDM vault.

Work with Existing SOLIDWORKS Files

28. In SOLIDWORKS go to File > Open.
29. Open any existing assembly.
30. When prompted, select to Check-out the assembly and referenced files.
31. Make a change to the assembly or a part file in SOLIDWORKS.
32. Create a drawing of the assembly or part in SOLIDWORKS.
33. Go to File > Save As and save the drawing as a PDF file.
34. In SOLIDWORKS, go to File > Print and save a PDF file of the drawing.
35. In SOLIDWORKS, Save all files.

PDM Check-In of Existing SOLIDWORKS Files

36. Check-In all the files to PDM.
37. Close all files in SOLIDWORKS, leaving the SOLIDWORKS application open.

PDM Operations in Windows File Explorer

38. Open Windows File Explorer.
39. Navigate to the test folder in the PDM Vault.
40. Select any assembly in the folder.
41. Check-out the assembly and referenced files.
42. Make a change in the data card for the assembly or a part file.
43. Save the data card.
44. View the tabs Version, Bill of Materials, Contains, and Where Used for an assembly or a part file.
45. Check-In the assembly and referenced files that were checked-out in step 41.

Performance Benchmark Test

46. Reboot the computer. This benchmark test can take up to 30 minutes. Do not use your computer for other applications during the test.
47. Go to Windows Start menu > SOLIDWORKS Tools 20xx > SOLIDWORKS Rx 20xx.
48. In SOLIDWORKS Rx, go to the Benchmark tab.
49. Click the "Start Benchmark" button.

