

Best Practices for Parallel Installations

What is a “Parallel Installation” and why are they used?

A “Parallel Installation” of SOLIDWORKS is any time more than one version (i.e. 2011, 2012, 2013, 201X, etc.) of SOLIDWORKS is installed and operating on a single system. This setup does not require additional licenses as both versions operate under the same serial number and the same license runs both installations. When working with multiple companies, each company may have their engineering department frozen at a particular version of SOLIDWORKS. It is always best to match their version of SOLIDWORKS to ensure design changes can freely flow back and forth between companies.

What is the benefit of setting up “Parallel Installations” in a best practices manner?

The reason a “Parallel Installation” is setup in this way is to prevent a messy file structure on a CAD computer. A messy file system can have several consequences:

- ❏ Linking each installation of SOLIDWORKS to the same “File Locations” reference material
- ❏ Updating the Hole Wizard/Toolbox database or linking to the wrong location
- ❏ A folder structure that contains many duplicate folders

Each of these items contributes to lost productive time due to causing:

- ❏ Compatibility errors within the various SOLIDWORKS installations
- ❏ Making it difficult to determine which folders may be removed when finally removing obsolete SOLIDWORKS installations

Ultimately, two simple edits during the initial installation can alleviate many of the problems. It is recommended to use only the version of SOLIDWORKS and not include the service pack number as it is common to upgrade an existing installation between service packs rendering the installation directory an inaccurate description of the installation.

What needs to be done?

Each SOLIDWORKS installation on a single machine should go through the following process:

1. Initiate the installation from the installation disc or downloaded data set
2. On the ‘Welcome’ screen specify the type of installation to be “Individual (on this computer)” then proceed ‘Next’

Individual (on this computer)

3. On the 'Serial Number' screen enter the assigned serial number or leave the existing serial number and proceed 'Next'

SolidWorks Standard, Professional, Premium or SolidNetWork License

4. If the 'System Check' finds an existing SOLIDWORKS installation a special 'Installation Options' screen will appear. In this screen select to "Create a new installation of SOLIDWORKS 201X SPX.X" and proceed 'Next'

Create a new installation of SolidWorks 2013 SP2.0.

5. On the 'Summary' screen is where most the changes will be now taking place.

- a. Find the 'Installation Location' section and elect to 'Change' it

Installation Location CHANGE 

- b. Set the folder for the installation to include the version of the installation. For example the default installation directory is this:

"C:\Program Files\SolidWorks Corp"

So, set it to:

"C:\Program Files\SolidWorks 201X"

- c. Once changed go 'Back to Summary'
6. Now, on the 'Summary' screen again.
 - a. Find the 'Toolbox/Hole Wizard' section and elect to 'Change' it

Toolbox/Hole Wizard Options CHANGE 

- b. The options should continue to be set to "Create a new Toolbox"
- c. Set the folder for the Toolbox/Hole Wizard to include the version of the installation. For example the default Toolbox/Hole Wizard directory is this:

"C:\SolidWorks Data"

So, set it to:

"C:\SolidWorks Data 201X"

- d. Once changed go 'Back to Summary'
7. With these two items set the installation is ready for 'Download and Install'
8. Once the installation is complete repeat steps 1 through 7 for additional parallel installations