



Assembly Templates

- **SolidWorks**
 - COSMOS
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The correct use of Assembly Templates can streamline your workflow and help you avoid problems when updating your configurations. Many users incorrectly assume that the software is "buggy" when unwanted parts keep popping up in other configs. Avoid unnecessary frustration by tuning your configuration settings in the default template.

In this article, Keith sheds enlightenment on the nuances of past versions of SolidWorks, and the value of using mapped folders containing your company standards and templates.

About KAP

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Keith Pedersen has a BSME from Clarkson College, and an MSME from Boston University. After a stint at General Electric in Burlington, VT, Keith was the lead Applications Engineer for Advanced Surfacing products for Matra Datavision USA, including EUCLID-IS, UniSurf, and STRIM. He joined CAP in 1998 to support advanced surfacing applications in SDRC I-DEAS and joined our SolidWorks group one year later. Keith has extensive industry and consulting experience in non-linear Finite Element Analysis and Computational Fluid Dynamics in addition to surfacing applications. He is a Certified SolidWorks Professional (CSWP) and certified to train and support COSMOSWorks.

About CAP

Computer-Aided Products

With offices throughout New England, Computer-Aided Products is one of the largest resellers of CAD/FEA/PDM software, rapid prototyping systems, training, consulting, and support in the Eastern US. Our engineering staff includes experts in design, analysis, drafting, PDM, and Windows networks. Our partnership with the best solutions in the industry, along with over 16 years of practical experience, provides a solid foundation for your mechanical engineering and design process.

As I visit SolidWorks customers throughout New England, I find that when the topic of conversation turns to Assembly Configurations, users are strongly polarized into two camps. The majority of users are enthusiastic about using Configured assemblies, and the discussion usually centers on establishing company-wide conventions for configuration names and properties, etc. However, it is disturbing how often I run into users who have decided to avoid configuring at the assembly level entirely. As I often say in my training classes, "If you're not using Configurations, then you're not really using SolidWorks".

When I encounter a site that has foresworn the use of Configurations, I always dig deeper to get at the root of the dissatisfaction, and I find that in most cases it stems from a single source – badly configured assembly templates.

Before we discuss the button-pushing, let's take a very high-level look at how your work-flow might impact your use of Configurations. I'll compare two different approaches. For illustration sake, I'll label these two workflows the "Pre-Configure" and "Post-Configure" approach.

Pre-Configure:

KAP's Tip: If you've pre-configured the assembly, then as you start adding parts, you want the majority of these parts to be added in common to all the other configurations.

Let's imagine that you are designing an electrical device, one that could end up being shipped to the UK, Canada, USA, or France. The power supply will have to be different for each of these assemblies, but let's assume that the rest of the parts will be common to all four designs... If you are planning ahead, you might open a new assembly file, and create a configuration for each country, prior to placing any component parts.

Now as you start adding parts to the assembly, the vast majority of parts will be common across all Configurations. Let's assume that you are working first in the USA configuration. In the early

going, at least, parts added to the USA config. Also want to show up in the bill of materials for the UK, CAN, and FRA configurations. You can set the Configuration Properties to behave this way by default. The effect you seek is that, any part added to the Active Configuration, should ALSO be added to the configurations that are not currently active.

Post-Configure:

KAP's Tip: If you've post-configured the assembly, then as you start adding parts, you want the majority of these parts to be unique to the current configuration.

Here's a different work-flow, one that I think reflects the habits of the majority of users. Assume that you open an assembly, and (in the DEFAULT configuration) you build the machine, until it is substantially complete. At this point, you want to use this machine as the base-line for creating the four country variations. So you re-name the Default configuration to be USA, and then create three more configurations, names UK, FRA, and CAN.

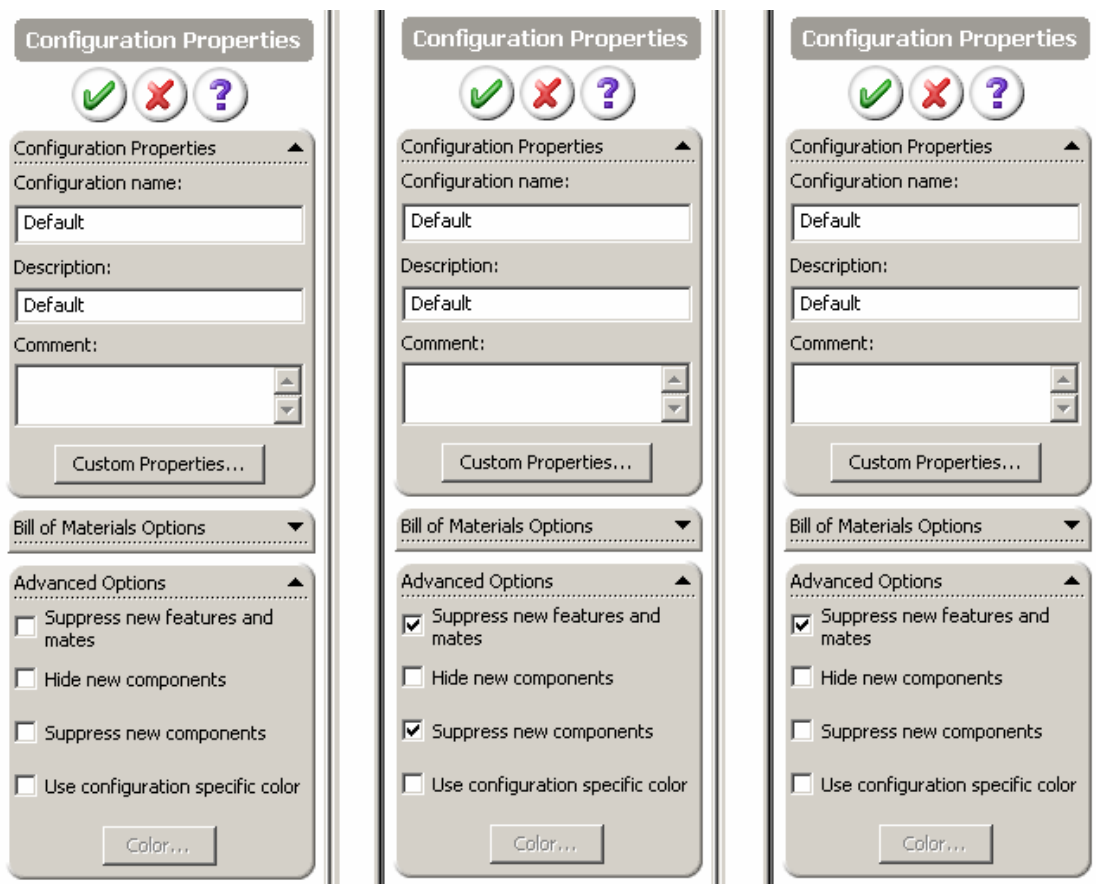
At this point, as you activate each configuration and drop in the appropriate power supply, you want the newly added parts, mates, and assembly features to be unique to this country's design. You can set the Configuration Properties to behave this way by default. The effect you seek is that, any part added to the Active Configuration, should NOT be added to the configurations that are not currently active.

Default Settings:

Every Assembly configuration starts out with at least one configuration, named "Default". Any new Configuration will copy its property settings from the current configuration, so the Default configuration is in essence the parent of any future configurations. You want to make sure the Property settings of the Default config accurately reflect your typical work flow. Let's check these settings now.

KAP's Tip: The check-boxes pictured here control what will happen within this configuration, when it is not active.

Launch a SolidWorks session, and start a new assembly, from whatever template you most commonly use. Activate the configuration tab, then right-mouse click over the default configuration and select "Properties". Compare the settings in the three panels pictured below, paying attention to the check-box settings for "Advanced Options".



1. Pre-Configured

2. Post Configured

3. Alternate Positions

The first panel shows a configuration in which newly added components and mates will neither be hidden, nor suppressed. Therefore, any changes made in some other configuration, will be reflected here also. The third panel shows how to set the panel when doing configurations that represent the same bill-of-materials, but with alternate positions in space. So newly added components will appear in every configuration, (will not be suppressed), but newly added Mates will only take effect in the active configuration, (because they will be suppressed in other configs).

With the release of Solidworks version 2001-Plus, the default assembly templates changed to the settings shown in panel 3. Customers with existing templates were mostly un-affected, since their templates would have been adopted by the new version while upgrading.

Default Assembly Templates:



By far, the majority of users I encounter add detail to their assemblies in a Post-Configure mode. Therefore, the best settings for the DEFAULT configuration in your assembly template(s) should be the settings show in panel 2, above. In fact, from 1995 thru 2001, the default Assembly templates that shipped on SolidWorks CD's were configured as shown in panel 2.

If unwanted parts keep popping up in other configs, you can feel like you're playing a game of "Whack-a-Mole". Tune your configuration settings in the default template to avoid unnecessary frustration. We at CAP are always careful to discuss the importance of setting up your template files in our training classes.

Conclusion:

Regardless of whether you are a new user, or a long-term SolidWorks site, take a few minutes to open a new file from each Assembly template you have, and check the Default configuration properties. Decide which pattern of use describes you best, and tune the "Advanced Options" panel to suit. Then, to lock-in your changes, use FILE -> SAVE AS, and change the "Save As Type" option to say "Assembly Templates, (*.asmdot)".

Keep in mind that you can change these settings, at any time, for any configuration. So if you have been building an assembly in the "Pre-Configured" mind-set, but have now come to a point where you want to add parts unique to each, then you simply click on the Configuration Management tab, and edit the properties of each configuration to the settings shown in Panel 2, above. And, you may want to add parts in common between three related configurations, but not in any others, so prior to the additions, you set those three configurations' properties to look like Panel 1.

KAP's Tip: An ounce of prevention is worth a pound of cure. Take advantage of CAP's training courses and our support Hotline to learn how up-front customization of your CAD environment can shorten the learning curve and pay big benefits in productivity.

If your assembly templates are distributed from a common mapped folder as a company standard for use by all users, Good for you! You only have to change the template files in one place. If you haven't directed all users to a common mapped folder, then you will have to make this change, for every template file, and for every CAD-using machine.

And if you're not sure how to go about directing all your users to mapped folder containing your company standards and templates, then you should call your CAP support engineer or account manager - we should talk. This is one area of the CAD implementation where a little effort up-front will equate to great manpower savings later on.