



## Synchronizing the Toolbox Library

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- *SolidWorks*
  - *COSMOS*
  - *PDM*
  - *Rapid Prototyping*
  - ***Office Productivity Tools***
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Have you ever tried to open a coworker's assembly, only to find that all of the fasteners in his assembly have defaulted to some huge size? Have you ever taken work home over the weekend, and found that when you open an assembly on your home computer, some of the fasteners have become substituted for different, seemingly random sizes? These problems stem from sharing data that reference more than one Toolbox fastener library. This installment of KAP's Corner will show you why these problems occur, and how to avoid them in the future.

### **About KAP**

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Keith Pedersen has a BSME from Clarkson College and an MSME from Boston University. After working for General Electric in Burlington, VT, and Lynn, MA, Keith became 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 also has extensive industry and consulting experience in Finite Element Analysis and Computational Fluid Dynamics. He is a Certified SolidWorks Professional (CSWP) and certified to train and support both Solidworks and COSMOSWorks.

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## ToolBox is a *Grow-as-you-Go* Database

There are a number of different third-party hardware libraries available for Solidworks, but they all fall into one of two organizational schemes. The most common scheme is to purchase a number of CDs that are pre-populated with CAD files. The advantage of this approach is that the library is static. That is, once the files are installed, the database will neither grow nor shrink in size. The disadvantage of this approach is that the database starts out with a maximum footprint on your hard drive. There may be thousands of files within this database that your company may never need. The second, and less common scheme, is to create the CAD files on-the-fly, as the user requests new types and sizes of hardware. The Toolbox Library falls into this second scheme.

On-the-fly file creation means that your hardware library starts out very small, and though it will grow over time, it will always be as small as possible, containing only those parts your company has actually utilized. This is also what allows you to customize the Toolbox library with your own components. The disadvantage of this method is that the hardware library is never really 'Closed' – you will always need some level of Write-Access to add new hardware types and sizes to the library folders. This can particularly be a problem when your CAD database is placed under the control of an overarching, company-wide document control database. Since the Toolbox Library is itself already a database, you have to carefully consider how Toolbox, and the company wide database, will stay up-to-date with each other. Just as an example, the PDM system “Adept” from Synergis Corp., allows you to identify what it calls “External Library Folders”, and so it would seem that you could put the Toolbox folders under the control of the PDM. But the utility for doing this only does a one-time scan of the folders you identify. It establishes links in the PDM database to only those files that exist at that moment, and assumes that the folder contents will never change.

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*KAP's Tip: In the Design Library Panel, the **Add To Library** icon allows you to add your own components to the Toolbox Library.*

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The most common synchronization problems are caused by two or more users each having their own Toolbox Library. Sometimes the problem can be easily solved by combining these libraries into one, shared location. In some companies, however, the need to collaborate with CAD users in different divisions, or even from different companies, is a daily fact of life. Such collaboration means that designs will often employ hardware files that originate from different library locations, and so the Toolbox library will have to be tuned differently to still work under these circumstances.

## How Will I Be Using My CAD System?

Although no two companies are exactly alike, we find that the way a company utilizes Solidworks can be classified under one of these three descriptions:

- 1) Out-of-the-box Solidworks will install as a single-user, single machine CAD system. This sort of installation is the simplest, and the Toolbox Library will land on the user's local hard drive. Typically, the user's output of CAD documents will reside on his local hard-drive as well.
- 2) Users may choose to collaborate with each other over the network. This will require more careful tuning of the CAD system, and the Toolbox Library should be installed once on a network drive. Each CAD client is then directed to adopt that single database. This model of use is referred to as the play-it-where-it-lies method, or sometimes the 'Sandbox' approach, because all of the users are opening and sharing files in common folders over the network. This approach can be employed successfully by small, co-located design teams that value fast, flexible collaboration.
- 3) Larger design teams, and companies that need to collaborate across departments or between different locations, will require more formal work flow controls. They will need a central storage vault for archiving and versioning, and this is usually done by installing a supervisory program, (Product Data Manager, PDM, or Team Data Manager, TDM, or Enterprise Data Manager, EDM, etc.) that will control access to the Vault. However, the user's modeling tasks are performed on copies of the files in a 'scratch' location, usually on his/her local hard-drive.

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*KAP's Tip: How to tune your ToolBox Library when you know you will be sharing assemblies with other SolidWorks –using companies.*

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The problems arise when users take all (single-user) installation defaults (1) for their Solidworks, but then use the CAD collaboratively by way of the 'Sandbox' or the Vault methods. This could be because the file-management and system tuning decisions are too abstract or arcane for the user in his first few weeks of learning the CAD. Or, frequently, a company will start small, with just one or two Solidworks users, and so have limited collaboration and file management concerns. As their database, and perhaps their user-base, grows over time, they eventually exceed the limits of their initial data management scheme.

## How It Works

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*KAP's Tip: The Toolbox catalog of all available hardware standards is stored in an integral Microsoft Access database*

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Toolbox contains a Microsoft Access database, which contains a list of standards for every possible in size and length of fastener. The Access database is also used to keep track of which of these sizes have been put to use. Toolbox also installs a folder with Master CAD files, one for each type of hardware. These master files initially contain only a single, template size of hardware. Imagine now that a user has just finished installing his CAD system, and his first job requires an assembly that contains 1/4-20 UNC Hex Bolts, that are 1.5" long. Toolbox will first add this entry to the Microsoft Access database. What happens next depends on how the Toolbox Library is tuned.

If the Toolbox Browser is configured to the **default** settings, the new CAD data for the 1/4-20 Bolt will not be stored in a new file - it will instead be added to the Template as a new Configuration. Each time you request a new length of this bolt, the Template will expand to contain that new Configuration. The Access database will log the fact that each possible length of 1/4-20 Bolt is stored in a single file, and the name of that file will refer not to a particular size fastener, but to the entire size family. This use of Solidworks Configurations makes for more efficient use of computer resources. But it also puts all the eggs in one (or rather, relatively few) baskets. *This is where the collaboration problems can start.*

A less efficient but more robust way to tune the library is to have the system spawn a new unique Part file for each newly created type and length of hardware. In this case, the Master (template) files for each type of hardware item will remain unchanged.

**SolidWorks 2006:** Open an assembly file on your desktop, so that you can activate the TOOLBOX menu. Select: **TOOLBOX ->**

**BROWSER CONFIGURATION ->**

**BROWSER -> (Document Properties)**

Change the selection "Copy Part File" from the default '**NO COPY**' to '**ALWAYS CREATE COPY**'.

**SolidWorks 2007:** Select:

**TOOLBOX ->**

**CONFIGURE ->**

**SETTINGS**

Change the "Files" control from '**CREATE CONFIGURATIONS**' to '**CREATE PARTS**'.

## What Could Go Wrong

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*KAP's Tip: Have you ever experienced opening a Word document with a font substitution? If so, you'll understand the complexities of library substitutions when referenced elements do not exist.*

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The problems arise when users share their assemblies, and there is more than one Toolbox library.

For example, imagine that our user, named Alf, completes his assembly (which required a 1/4-20 UNC Bolt x 1.5" long) and saves it to a network folder. Now imagine that a second user Bart, browses to this network folder and opens Alf's assembly. If Alf and Bart had both directed their Toolbox Libraries to be shared from a common location, there would be no problems, as both refer to the same Bolt Master file.

However, let's assume that Bart has installed his Toolbox Library on his local hard drive and has never used a 1/4-20 UNC Bolt x 1.5" long before; so it does not yet exist in his database. In SolidWorks 2006, when Bart opens Alf's assembly, he would get a missing reference. ToolBox would substitute the next available fastener size that was found in Bart's local ToolBox database. Obviously, this could cause some serious problems, especially if not detected right away.

Now imagine a new user, Chuck, who works in a different company, perhaps as the machining vendor for Alf. There is no chance that these two users will share a common Library. If Alf does not provide a copy of his Bolt Master file to Chuck then it is possible that a size substitution will occur. But, if Alf does share a copy of his bolt Master file, there is the danger that it will overwrite Chuck's Master file, and so lose any size configurations that Chuck had already deployed.

Instructing Toolbox to create a unique part file for each fastener size does help, but does not completely solve this problem. Even though user Alf could share with user Chuck copies of his hardware files (one file for every size) merely placing these files in the Toolbox folder does not cause Chuck's Access database to be updated, so these files will not be adopted by his Library. And, the presence of these files could cause conflicts if user Chuck does try to initialize those particular hardware sizes later on.

## The Solution(s) ...

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*KAP's Tip: SolidWorks 2007 saves the day – at least for newly-created assemblies*

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Good news! This problem is partially solved by the release of Solidworks version 2007. From now on, when Toolbox cannot find a particular fastener type and length already present the library; it will **automatically initialize that size**, instead of performing a substitution. Any two users who collaborate, while working from different Toolbox libraries, will no longer experience any problems as long as their assemblies were **created** in version 2007.

However, if an assembly was last saved in Solidworks 2006 or prior, then opening the assembly in the 2007 version will *not necessarily* manufacture the missing references. Partly, this is because the older versions of Solidworks, (particularly 2004 and prior), stored less information with each component reference than 2007 does. But even when migrating from 2006 to 2007, I was able to create some test cases that continued to perform unwanted substitutions. And, in some cases where Toolbox did properly create the new referenced sizes, it was unable to save them because of the way the system now automatically manages the read/write properties of the library files.

So, if you do have two or more users each running their own Toolbox Library, NOW is the time to fix this problem. You should pick one of those libraries to be the starting point for building a unified, shared database, and fill in the gaps in this database relative to the other users, prior to upgrading to the next version. Once you have done this, the improvements in the latest version should be sufficient to ensure that the problem does not ever resurface.

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*KAP's Tip: Use a dummy assembly as a placeholder for initializing hardware components*

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How do you fill in the gaps in your chosen common database, to contain all the desired hardware sizes? The best procedure is very manual, but also very easy. While connected to the desired, common Library, you simply create a new dummy assembly. Then use the Toolbox browser to drag and drop into this assembly one instance of every hardware component that you want to have initialized. Don't worry about positioning these components to each other or to anything else - just the fact that a fastener size has been used once, and then saved, will be enough to initialize that component in the database. (This would be a good job for a co-op student, perhaps). It could take as much as two hours to get in one of every type and size, but once you save this assembly, all the component references will be saved and initialize in the Toolbox Library as well. Then you can simply delete the dummy assembly.

Finally, you will want to make sure that all of your CAD installations are now pointing to this common Library. In SolidWorks 2006 and prior, this is done by editing a text file in one of the installation folders. Use Notepad or WordPad to edit the following file:

C:\Program Files\Solidworks\Toolbox\Toolbox.ini

The last line in this file will be the path to the Toolbox Library. Make sure that all your users are pointing to the same folder on the network.

SolidWorks 2007 allows you to make this change more easily from inside the CAD session, by going to:

TOOLS ->

OPTIONS ->

System Options ->

Hole Wizard/ToolBox

Enter the path to the desired folder on the network.

## Conclusion

Having a library of pre-modeled hardware is extremely useful – so make sure you optimize to take advantage of it. If you have had problems with your libraries being out-of-sync before, NOW is the time to fix it.

0) Decide if you should re-tune your Toolbox to create new part files for each item, or to continue adding Configurations to the master.

1) Choose a network folder location to serve as the Toolbox library for everyone at your site.

2) Decide whose Toolbox library currently has the most complete. If nothing else, you can look at the path: C:\Program Files\Solidworks\ToolBox\Browser, and see which user's folder holds more data (by looking at the folder PROPERTIES). Copy this user's Toolbox folder to the network location.

3) Add to this folder any hardware types and sizes that may be missing, by building and populating a dummy assembly.

4) Point every other Solidworks installation to this same location

5) Upgrade to Solidworks 2007 – so that, even if assemblies do come into your company from outside CAD sites, your library will automatically initialize any needed sizes.

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*KAP's Tip: Follow these steps to maximize Toolbox efficiency with SolidWorks 2007*

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Have a Solidworks bone to pick? Want more tips in a specific area of the CAD? Keith is looking for requests from users for future KAP Corner topics. Email your suggestions to; [KAP@CAPINC.COM](mailto:KAP@CAPINC.COM)