

# Practical usage issues with Toolbox

There are two add-in programs for SolidWorks Toolbox. One is simply called SolidWorks Toolbox. This contains the beam and bearing calculators, structural steel shapes, grooves and cams. The subject of this paper is Toolbox Browser, which is the library of hardware with the part insertion and configurator interface.

Most of this document is written assuming you are trying to use configurations for your Toolbox parts.

## How Toolbox Works out-of-the-box

### Installation

By default, Toolbox installs on the local machine, with the library data going to the directory C:\Program Files\Common Files\SolidWorks Data. The user enters this information during the “Shared Files” part of the installation. The Toolbox part files as well as the Toolbox database, called SWBrowser.mdb are contained in the library data.

During an installation where all the default settings are used, Toolbox is set up to use a separate SolidWorks configuration for each size, but only a “default” configuration exists for each part after install. Using this scenario, size configurations are added as requested.

### Why is there a database?

There are three aspects to Toolbox. One is the library of parts, the second is the program that contains the interface and builds the various sizes of each fastener type, the third aspect is the database which contains all of the size information for each part. When a given part size is requested through the interface, Toolbox adds a configuration to the part using data from the database if the size does not already exist.

Think of the “program” aspect of Toolbox as being a “part configurator” in which you specify the dimensions of the part you are looking for, and Toolbox builds it.

## Sharing Toolbox Data

Toolbox is designed to work best on a single workstation where assemblies are not shared with other users. In this scenario, it works well as installed with default settings out-of-the-box. As you digress from that scenario, there will be several issues which you will need to know how to accommodate for. In general, it is *possible* for Toolbox to be used correctly under most common shared data arrangements, but the steps needed to accomplish this are not always obvious, especially for new users.

### Missing configurations

When multiple users on a single network use a default Toolbox installation where the top level assembly files are located on the network and the Toolbox parts are local, the main difficulty is that one user may have created configurations which another user has not. The second user will see Toolbox parts that cannot find the configuration the first user put into the assembly, and will instead get the “default” configuration, which is typically the largest size available. This is a condition affectionately known as “Huge Screws”.

## Shared network library

This multi-user issue points to a shared library as a solution, so that the configurations one user creates are available to all users. Physically doing this is easy, but what is not easy is doing it correctly so that no user data is lost.

To set up a shared network library:

1. Move one user's "SolidWorks Data" directory to the desired location on the network (contains folders called Browser, Copied Parts, lang, and PartLibs)

The SolidWorks documentation for doing this type of set up tells you to "Identify the computer whose user who has used SolidWorks Toolbox most extensively." This is good advice, but what it does not tell you is that if User 1 has created sizes 1, 3, 5, 7 and User 2 has created sizes 2, 4, 6, and you use User 1's Toolbox library as the basis for the network library, none of User 2's data will be in the new library.

Aside from manually re-creating previously existing configurations, there is no way to take two parts and add the configurations of one to the other without using Design Tables. Toolbox parts do not use Design Tables. The best way to combine all the configurations would be to use the auto-create option for Design Tables, and copy the appropriate rows from one spreadsheet to another.

Another thing which needs to be mentioned here is that the Toolbox database is contained in the "lang" folder. The database includes several settings made in the "Browser Configuration" dialog accessed through the Toolbox menu in SolidWorks. When you copy one user's library to the network, you are also copying that user's Toolbox settings for everyone. Just be aware of this and make sure that the settings you are copying are appropriate for all users. Once the network library is working, any user can change the settings used for everyone, so it should be a matter of policy that users do not change Browser Configuration settings without consulting the CAD admin or lead user.

2. Use Notepad to edit Toolbox.ini (C:\Program Files\SolidWorks\toolbox\toolbox.ini) so that the last line gives the path to your network Toolbox location. Mapped drives are not recommended, you should use the UNC path name. For example, if the network drive on the server is mapped to N:\, avoid using:  
N:\SolidWorks\SolidWorks Data\

instead, use the UNC path name:

\\ServerName\SolidWorks\SolidWorks Data\

where "SolidWorks" is a shared directory on the server called ServerName.

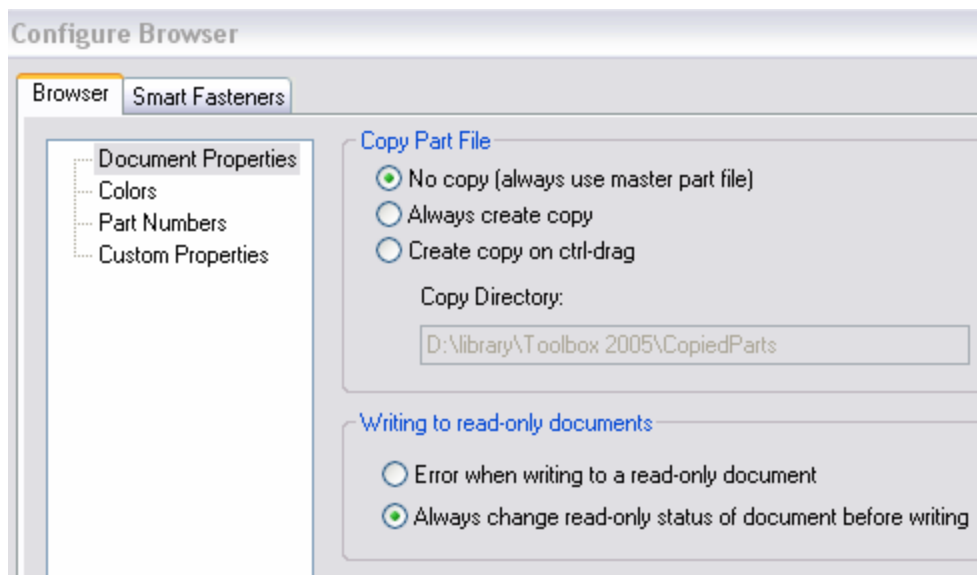
In the end, the text inside the Toolbox.ini file should look like this:

```
[Toolbox environment configuration file]
Creation=9/24/2004 at 11:13:26
[configuration]
ToolboxPartFolder=\\ServerName\SolidWorks\SolidWorks Data\
```

- Using Windows Explorer, set this network directory along with all the subdirectories and files to Read-Only access for all Toolbox users. This sounds counter-intuitive, but Toolbox will manage the permissions behind the scenes.

The reason for this is that if it is set to Read-Write for everyone, the first user to open an assembly with Toolbox parts would get access to the data and no one else would be able to add configurations to the parts.

- Copy this Toolbox.ini file to the other computers and overwrite their existing Toolbox.ini
- Rename the local SolidWorks Data folders on each user's computer as a back up.
- Remove any references to the SolidWorks Data folder in Tools, Options, File Locations, Referenced Files
- In SolidWorks, in the Toolbox menu, go to Browser Configuration



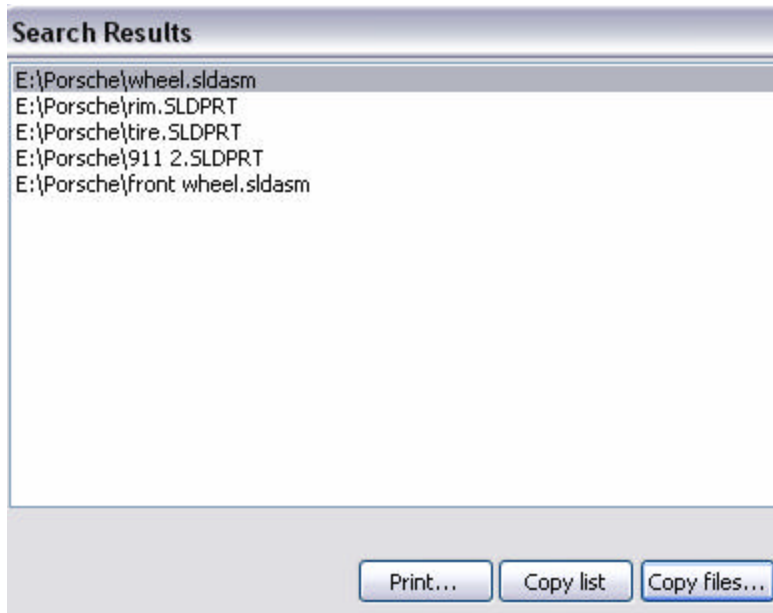
- Make sure "Always change read-only status of a document before writing" is turned "ON"
- Restart SolidWorks for the settings to take effect
- The following types of data will be lost for users other than the user whose SolidWorks Data was copied to the network:
  - Configurations
  - Add My Parts Wizard parts
  - Custom Standards
  - Browser Configuration settings
- The moral of this story is that it is much easier to set this up correctly right out of the gate rather than going back to fix it after it has been in use for a while.

#### Sharing assemblies using Toolbox parts with users not on the same network

When you are sharing assemblies using Toolbox parts with SolidWorks users outside your network, for example other divisions of your company or outside contractors, it is actually more problematic to share with people who are also using Toolbox than it is to share with people who are not using Toolbox. If they

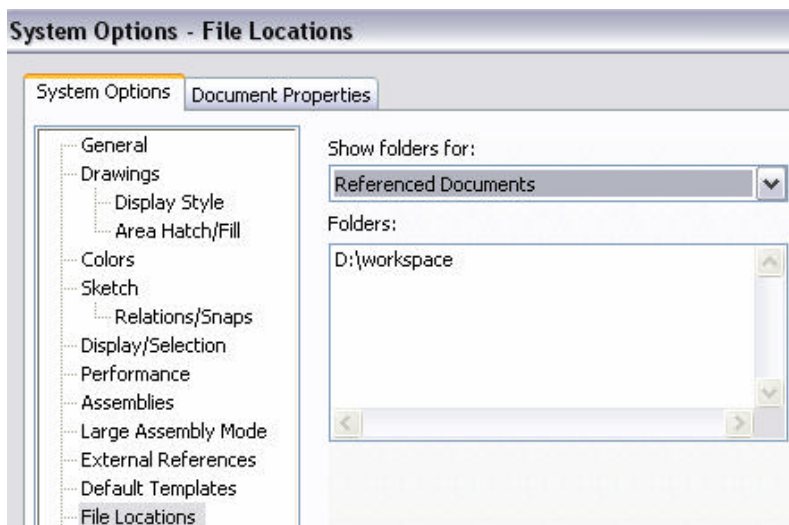
are also using Toolbox, you have to make sure that they use your Toolbox parts instead of their own, or they will risk getting “Huge Screws”. When you open their assemblies, it will be best if you can do it using their Toolbox parts.

To collect your assembly and all the parts related to it regardless of where they are located into a single location, go to File, Find References, and hit the Copy Files button.



This will copy all the files (including your Toolbox parts) to a single location, ready to be sent to the outside SolidWorks user.

When the outside SolidWorks user gets the files and they also have Toolbox, if they want to use your Toolbox Parts, they will need to make an entry in their Tools, Options, File Locations, Referenced Documents to point to the folder where your Toolbox parts are.



When they are done and want to use their own Toolbox parts again, they will need to remember to remove this reference. What this does is to tell SolidWorks to look in the specified path first for any documents referenced by an assembly, so that where there are duplicate file names, the ones in this directory are chosen first.

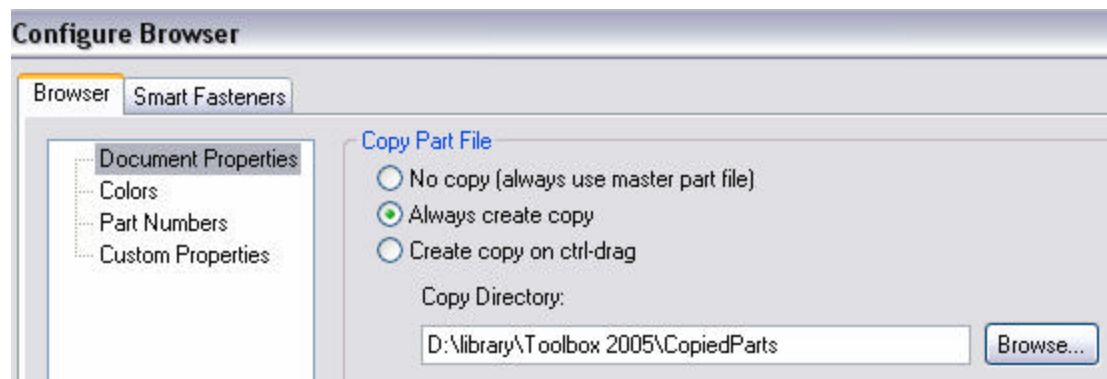
If the outside SolidWorks user does not have Toolbox, then there should be nothing to worry about, and they can just open the assembly and all the parts will come up as you saved them.

### Alternative to configurations

As you can see, configurations present some challenges. It would seem best if Toolbox were able to install right out of the box with all of the configurations available immediately. This would cut down on so much lost data that it would definitely be worthwhile, but nevertheless, this is not an option. The only thing you can do is hire an intern and let him/her spend the summer creating all the possible sizes of all Toolbox parts. Then you have to make sure that anyone you share data with knows how to use your parts or has done the same thing.

Fortunately, there is an alternative to using configurations. You can tell Toolbox to save out each size as a separate file. You cannot control the names of these files. The advantage of using separate files is that the size of the part is built right into the file name. Also, when a configuration is missing, the part is still added using the last saved or in use config, and if saved that way, there is no way to go back to the intended configuration. When a part is missing, it is simply suppressed. Missing is better than incorrect, and easier to fix. Once the correct file has been located and put in the correct path, SolidWorks can find it and the part can be unsuppressed (resolved), regardless if the assembly was saved with the part suppressed or not.

The option to create separate files for each size is called “Copy Parts”, and it can be set in the Browser Configuration dialog, Document Properties page.



It is recommended that if you do this, the Copy Directory should be on the network. It will need to be read-write so that people can create new sizes.

### **Don't mix Copy Parts with Configurations**

If you have started using configurations, and then switch to copy parts, the copied parts may have configurations in them which may cause problems. The best thing to do if you make this switch mid-course is to remove all configurations from Toolbox parts, copy over a fresh Toolbox library or just delete the local library and reinstall Toolbox.

Moving the other direction, while not advised, is actually less problematic. If you start using Copy Parts, and switch to Configurations, there is no overhead from the copied parts aside from the parts in the directory and the references to those parts in any assemblies you have put them into.

### Drawbacks of the network based Toolbox installation

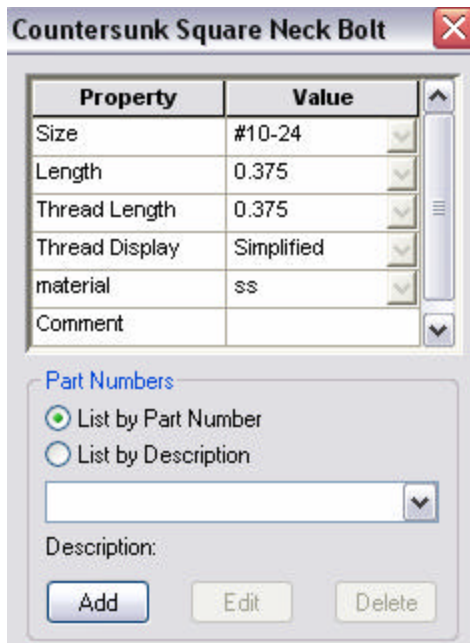
In a network installation, the database resides on the network, not locally. This means that it has to be opened across the network when any settings are changed or parts are inserted. The default freshly installed SWBrowser.mdb file is just over 50 Mb. If you add custom standards, this can be over 350 Mb. And it opens across the network. This will cause a very noticeable performance lag. It is not uncommon for part insertion to take 2 minutes mainly due to database size and network speed.

Fortunately there is a work-around for this. Since we have already established that the installation default of using configurations is very problematic, and the relative advantages of separately saved parts solve many of those problems, we will go ahead and just say that the “Copy Parts” option is the recommended one. With the “Copy Parts” option, you can choose where to save the copies, which means that the copies don’t have to be in the same place as the rest of the library, which in turn means that Toolbox can be installed locally, and the copied parts can go on the network.

As it turns out, this is probably the best way to use Toolbox. The main difficulties with this method are that you need to make sure that all users are using the same Browser Configuration settings. To do this, get it set up on one computer, then copy the entire database to each computer, and make it a matter of policy that users should not change the database settings without consulting the admin or lead user.

### Custom Part Numbers, Descriptions

Many companies use custom part numbers or descriptions for Toolbox parts because the quirky Toolbox names are possibly not the best way to display data in the SolidWorks BOM. In Toolbox, the way that Part Numbers and Descriptions are entered is in the interface dialog ... one size at a time.



If you find that you can use configurations, you can back door this somewhat. You could auto-create a Design Table, and use the design table to build part numbers and descriptions, but that assumes that you have pre-built all of the sizes already.

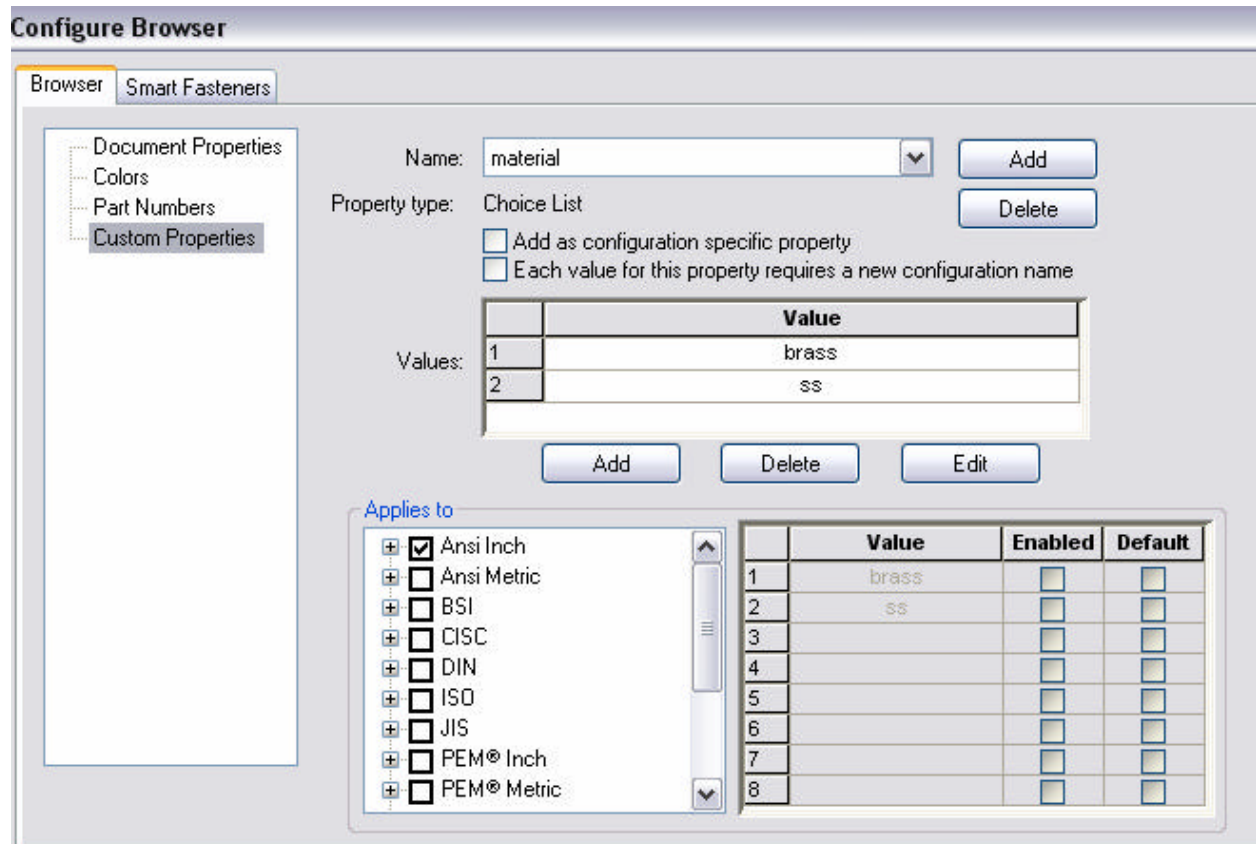
If you cannot use configurations, you are stuck with keying part numbers and descriptions into the interface one by one. Time to hire another intern.

### Materials and Finishes

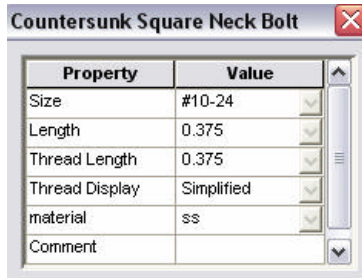
Materials and finishes are a very common property of Toolbox parts. Unfortunately, there is no built-in way to work with them. Again, with configurations, this is something that you might be able to accomplish using the Design Tables.

You could make custom standards in Toolbox for materials, but this turns out not to be a very good approach. Besides the issue of custom standards bloating the size of the database to unwieldy proportions, early versions of 2005 have a bug if you use custom standards and have some standard standards (?) turned off, screw sizes for Inch fasteners will show metric sizes.

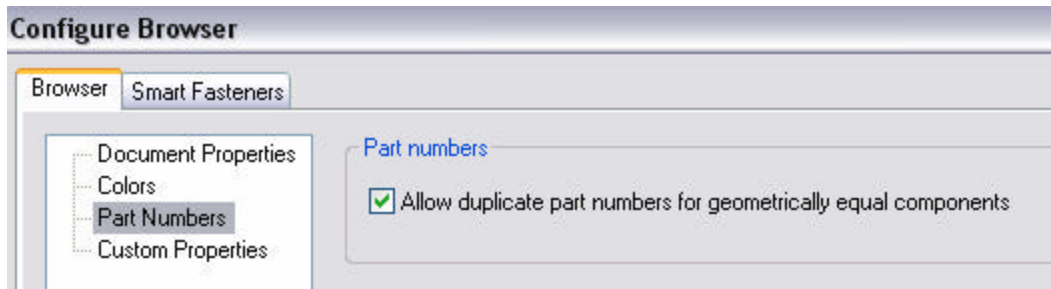
The best way of doing this is probably to use the Custom Properties page of the Browser Configuration menu.



This causes a material drop down to show up in the interface window as such:



If you do this, remember to set the “geometrically equal” switch:



If you are using the Copy Parts option, you will get file names that end with “V1”, etc. for additional materials.

There may be ways of manipulating the database to get things like this to work out better, but they are neither recommended nor supported.

In short, if you are concerned about efficiency, part numbers, descriptions and materials, I strongly recommend that you create your own library driven by Design Tables with lookup tables or heavy use of the Concatenate function. This will take time, but has several advantages over the alternatives.

### Smart Fasteners

Smart Fasteners functionality is only available with Toolbox parts, and not available with Add My Parts Wizard parts, Toolbox Custom Standard parts or “homegrown” libraries. Smart Fasteners is also limited to Screws and Bolts, basically any of the types listed in the Hole Wizard Screw Type lists, and cannot populate pins, PEMs, etc. The biggest benefit of using Smart Fasteners comes when using Hole Wizard Series Holes in the context of an assembly, where Smart Fasteners populate the holes which are driven through multiple parts at once immediately upon creation.

Several limitations of Smart Fasteners are listed in the SolidWorks Help file under “Smart Fasteners, Unsupported Hole Types”. While this is not an exhaustive list, it begins to help you to know what to expect from Smart Fasteners. There are times when Smart Fasteners puts screws in random cylindrical shapes which may or may not include a hole. In general, on all but the most carefully controlled design situations, the edit rate may be as high as 30-40%. Editing Smart Fasteners is more time consuming than judicious use of component patterns.

You will need to be your own judge about whether or not Smart Fasteners will save you and your company time and money.

## Missing Toolbox content

There are a few types of hardware that are missing from the Toolbox library, such as sheetmetal or wood screws, nyloc and vented screws, cotter pins, diamond pins and many metric parts including bushings, lock washers, pins, many kinds of nuts and screws. Also missing is any reference to drive types other than “cross”, “slotted” and “socket”. In short, before deciding to commit resources to implement Toolbox, you need to make sure it has the content that you need.

## Modeling problems with Toolbox parts

There are some modeling problems with Toolbox parts which result in parts with more compute overhead than necessary or which may cause parts to be difficult to place with Smart Mates. I'm quite sure that a more thorough investigation of these models would show a dozen or more issues which could be improved, but these are issues that I have noticed while working with the Toolbox screws and bolts.

- Inconsistent use of edge breaks on the threaded body of the screw (machine screws have none, socket head screws have fillets and chamfers)
- Some screws (SHCS and Shoulder Bolts) have fillets under the head where you would otherwise be tempted to grab it to do a Smart Mate (fillet edges can't be used for Smart Mate)
- Inconsistent availability of thread options
- Cross head uses a lofted cut and then a circular pattern, which together account for 87% of the rebuild time (Feature Statistics)
- Many of the parts have unnecessary small edges or faces which have a negative effect on graphics performance, especially when the screws are used in large numbers.
- There is no option to show the thread at the root diameter to avoid interference when put in a threaded hole.

## The dangers of Updating Toolbox

You need to be careful about how you update Toolbox from one version to the next. The first thing to be careful of is that you don't accidentally delete your previous Toolbox library. If you do this, you lose all of the configuration data, copied parts, custom property data, custom standards, add my parts wizard parts and other types of data. Your Toolbox library has developed over time, and there is a lot of time invested in making it useful. Be careful not to delete or overwrite the files.

Bottom line on Toolbox updates is to back up the library parts and database before doing the update. Reading the SolidWorks installation notes is not a bad idea either.

## Toolbox and Version Compatibility

Many companies for one reason or another need or choose to use two versions (such as SolidWorks 2004 and 2005) in parallel. Remember that SolidWorks is not backward compatible when it comes to versions. So even if you update your Toolbox properly, if you are also using a previous version of SolidWorks, you will not be able to use new version parts in an old version of SolidWorks. In this case, you would need to make a parallel Toolbox install and copy the old library to overwrite the new one.

## Toolbox Alternatives

Through all of this, remember that Toolbox is supposed to be a *library*, a library of parts which have been defined by Machinery's Handbook for *decades*. The geometry of the parts is static and well defined, as is the non-geometrical data such as materials, finishes and drive styles. All of this while a very powerful yet

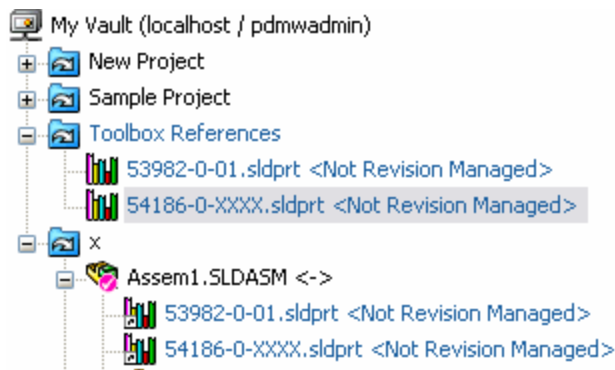
simple tool (Design Table) begs the question why this has been so rigidly difficult and incomplete for so long.

There are a few alternatives to Toolbox:

1. Use Toolbox to build every available size of fastener, store them in a library and send this library to all of the outside SolidWorks users you work with.
  - a. This could be done with either “Copy Parts” or Configurations.
  - b. Once it is done, auto-create a Design Table to populate the custom part numbers, descriptions, materials, drive types, etc.
  - c. Model any necessary missing hardware and drive with Design Tables taken from manufacturer’s, distributor’s or Machinery’s Handbook’s tables.
  - d. Change the names of the parts to reflect your company’s standards.
  - e. Turn off the SolidWorks Toolbox Browser add-in
2. Purchase a library from one of the partners on the SolidWorks partners web page, be sure to get a trial of it first to make sure that it is an improvement and has what you are looking for.
3. Create your own home grown library.
  - a. There is not much more work involved in creating your own parts from scratch than there is in making the necessary changes to make the Toolbox parts useful.

## PDMWorks

If you are using Toolbox or any library including Feature Palette with PDMWorks, you have several options. Library parts can be checked into the vault or not. Even if they are not checked into the vault, with PDMWorks 2005, they can still be tracked in the vault, showing in a special project called “Toolbox References” where any part in the path specified on the Vault Admin tab automatically is moved. Library parts show as shortcuts under the assembly where they are put.



This is a great function which I hope can be expanded to other types of files in future releases. The parts are not moved into the vault, although it looks like they are there. The only thing in the vault is a bitmap image of the part. Even though the library parts are not in the vault, they can still be dragged and dropped from the vault window into the assembly window to Smart Mate them into place.

The image shown above uses parts from a custom library, but this approach will also work with Toolbox parts. I do not recommend checking in Toolbox parts unless the “Copy Parts” setting is used.

## **Conclusion:**

I have met several people who have been pleased with how SolidWorks Toolbox works. These people are generally just looking to put a “looks like” piece of hardware of approximately the correct dimensions into their assembly more or less for illustrative purposes. For this it works well.

Generally speaking, Toolbox is not well suited for sharing data between users on a network or between remote users who both have Toolbox. It is ideal for a single user in a stand alone design environment. Toolbox by default installs with the worst possible combination of settings for sharing data in multi-user installations, so if you are new to this, you might want to ask your reseller to recommend an installation scheme, or you can follow mine below.

If you are looking for good BOM data, including company specific part numbers, descriptions, finishes/materials or special drive types, you are going to spend a good deal of time working with the Toolbox parts to get them to fulfill your needs.

## **Recommendation**

If you plan to work with Toolbox, here is the most reliable scheme:

- 1 Local installs of Toolbox, database copied to each computer
- 2 Use “Copy Parts” setting
- 3 Network copied parts directory
- 4 Set Tools, Options, File Locations, Referenced Documents to point to the copied parts directory on the network
- 5 Use “geometrically equal” part number setting
- 6 Use “change read only status” document property switch
- 7 Use Custom Properties for material/finish and special drive types as necessary
- 8 If you travel with a laptop and want to do design work on the road, you will need to copy the copied parts directory to your laptop before you go and change the Referenced Documents path to the local location. When you return you will need to copy any newly created sizes to the network copied parts directory.

Matt Lombard  
October 5, 2004