# Organizing CAD Data

The approach you use to organize your CAD data will depend on the size and make-up of the group in which you work. If you work in the context of a large group of departments who share files and data across a formally structured network, then you will use one approach. If you work as a single group that is virtually independent, then you will use another approach. If you work on your own or in a *virtual company* model, then another approach should be used. This monograph is aimed at those who work in that last (*sole* or *virtual company*) context.

## General Overview:

A large company with multiple departments sharing a file-server with *fixed* location paths, will use an approach where all design or analysis files are located in specific *directories* (*folders*) that will be the same for all users. Thus, *standard parts*; such as fasteners, bearings, electronic components, and the like; will have the same *file path identity* for all users. Under these conditions, it make sense to leave these *standard parts* in the *standard part location* where a system manager can oversee and maintain their geometry and supporting data.

A similar, if somewhat less complicated approach will work well in the case of a single engineering or design group working off of a locally networked file-server. The difference will be one of scale and rigor of management. The advantage of this approach is the unity of shared data.

However, when one is either working alone or connected through internet-based file sharing, require a localized and individual file structure approach. Under these conditions, each user will be responsible for maintaining their *standard parts libraries* and such parts, when used in a model, **must** be saved to the *project directory* to avoid their getting lost when moved from user to user as the design evolves. The disadvantage of this approach is that, if there is a change to a given *standard part*, then each design file that uses that part will have to be individually updated to incorporate such a change.

## File-Path Approach:

The first question that needs to be addressed is **where** will you maintain your CAD files on your local drives? The "standard" created by Apple and Microsoft of using a *\Users\User\Documents* directory on a (usually *C:*) local hard drive has its faults. One of these is that using the *C:* drive means that your programs and data are all in the same place. Another is that, should somebody wish to *crunch* (the correct term rather than the inaccurate *hack* promoted by the ignorant) your system, using this scheme makes it easier for them to do so. Personally, I find that using a *non-sequential* drive letter (i.e. making my *data drive* a *T:* drive with no intervening drive identifiers) adds to my data security. This approach will, generally, keep out a tyro-type *syscruncher* (there being *nothing* that will stop a true professional level *syscrucher*). It also makes it easier to perform data back-ups – locally and *off-site* (an aspect that many overlook).

Not giving anything particularly harmful to me away because I have *published* so many *libraries* that include *standard parts* along with *spreadsheet indexes*, I use my *J:* drive as my primary *data drive*. The structure I use is: *J:\Lews_Data\Designs* as my foundation. Off of this, I create various *XXX Standards* directories where *XXX* is the CAD system's identity. Thus, off of my *Designs* directory, I have *Alibre_Standards*, *SolidWorks_Standards*, and others. [I started this practice back in the days when having spaces in a directory name could, and often did, cause problems. This is no longer true, but making such a change for me today would involve a lot of donkey-work.] Off of each *XXX_Standards* directory will be other directories detailing such things as: *Structural Shapes*, *Fasteners*, and the like.

My *J:\Lews_Data\Designs\XXX_Standards* directory is where I locate all my *spreadsheet index* files that provide a set of *hyperlink* selections as well as applied design reference information to make finding and selecting/loading the specified *standard part* quick and simple. Once loaded into my CAD system, the *Save As* command allows me to store that part (renaming, if necessary) into my *project* directory and then inset it into my *working design*. Thus, when I have complete the project in question, I can merely *compress* the entire project to create an *archive* of it that contains all the CAD (and CAE) files for future reference.

## Spreadsheet Indexes:

To be clear, **I** use the *LibreOffice* program set and have as little to do with *Microsoft Office* as possible. Several of the specifics listed in this section will not work without *tweaking* in *Excel*, though they work just fine with *SCalc* (the *LibreOffice* spreadsheet). This statement constitutes *fair warning*.

My *spreadsheet index* files break down into: *Index*, *Instructions*, *Selector*, *Collector*, and *System Values* worksheets.

The *Index* worksheet provides the starting point for the index. It provides *hyperlinks* to the various *Selector* worksheets within the library. Thus, in the case of "cap screw" parts, it will break down into *material* (typically alloy steel or stainless steel), *grade*, and *pitch-class* of the parts. Thus, in the case of *SAE Graded Bolts*, the choices will be: *Grade 2 UNC Thread*, *Grade 2 UNF Thread*, *Grade 5 UNC Thread*, etc.

The *Instructions* worksheet provides useful information on installing and using the system.

The *Selector* worksheets are the actual working *selection index*. They provide an appropriate matrix of information such as size, availability, and the like to help the user make their choice. What application information is supplied depends upon the components being selected. In the case of screws and bolts, things such as maximum design tensile load and shear load, and recommended torque are supplied. In the case of double-acting pneumatic and hydraulic cylinders, extension and retraction pressure areas are supplied. Etc. Care in developing this portion of the index will pay you back in spades!

The *Collector* worksheets are where the information presented in the *Selector* worksheets is collated and prepared. They are, in essence, a *rough layout* of the *Selector* worksheet.

The *System Values* worksheet is where information such as *file paths* and *file extensions* and other data generally useful in generating the index is stored. Setting the *file paths* allows the *concatenate* function to update to the user's choices without a lot of effort. Allowing the user to adjust the *file extension* will allow a user to copy the *spreadsheet index* and apply it to a different CAD system. I have, over the years, learned to extend the utility of the *System Values* worksheet greatly.

## Organizing Working Parts Libraries:

As noted above, my system bases off of my *J:\Lews_Data\Designs\XXX_Standards* directory. The part libraries are first broken into general categories: *Bearings*, *Structural Shapes*, *Fasteners*, *Electronics*, and the like. These are further broken down (using *Fasteners* as my guide) into: *Inch Fasteners* and *Metric Fasteners*. These in turn are broken down (using *Inch Fasteners* as my guide) into: *Dowel Pins*, *Washers*, *Socket Head Cap Screws*, *SAE Graded Bolts*, *General Hardware Bolts*, and the like. Creating a sensible *tree* of such directories ends up being an ongoing *work in progress*, but a bit of time planning for your needs is worthwhile.

## Conclusion:

Hopefully, this monograph helps you get your data organized in a sensible and useful manner.