



A Libertarian Guide to Implementing Corporate Standards

Jason Pancoast
Engineering Manager



Who am I?



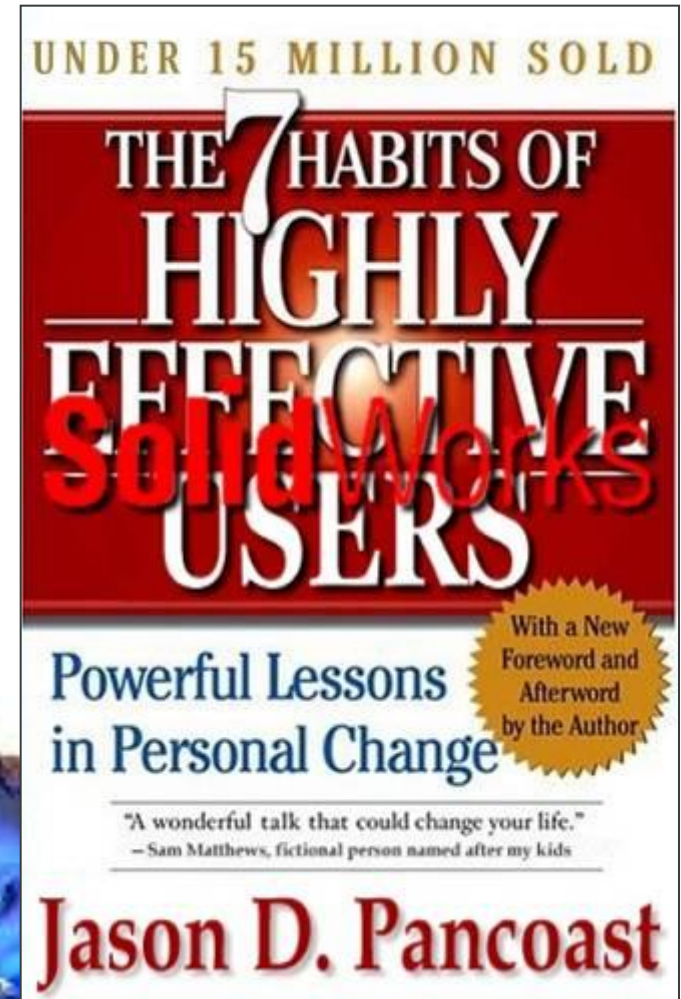
- Engineering Manager
 - In charge of the support and services team at CAPINC.
 - Been supporting SolidWorks since 1997.
 - Worked briefly in aircraft engine design and in manufacturing process improvement.
- Elite AE, class of 2007
 - Advanced Certification in every SolidWorks product.
 - Seven-time SWWorld presenter.
- CAPINC
 - SolidWorks and Stratasys dealer serving New England.
 - 4 offices in MA, NH, and CT.
 - Founded 1989.
 - Awarded Number One in SolidWorks Customer Satisfaction in North America, 2006 and 2008.



The 7 Habits of Highly Effective SolidWorks Users

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1. Customize the User Interface.
2. Implement Corporate Standards.
3. Don't Skimp on Hardware.
4. Buy and Use Partner Products.
5. Capture and Communicate Design Intent.
6. Manage Data and Collaborate.
7. Get Training.



Jason D. Pancoast, CAPINC Engineering Manager. Manchester, NH, 2008.

Habit 2: Begin with the End in Mind



"Begin today with the image, picture, or paradigm of the end of your life as your frame of reference or the criterion by which everything else is examined."

---Stephen R. Covey

Habit 2 of Highly Effective SolidWorks Users:

Implement Corporate Standards

- A clear understanding of your destination
- By design or by default?
- Do not violate the criteria you have defined

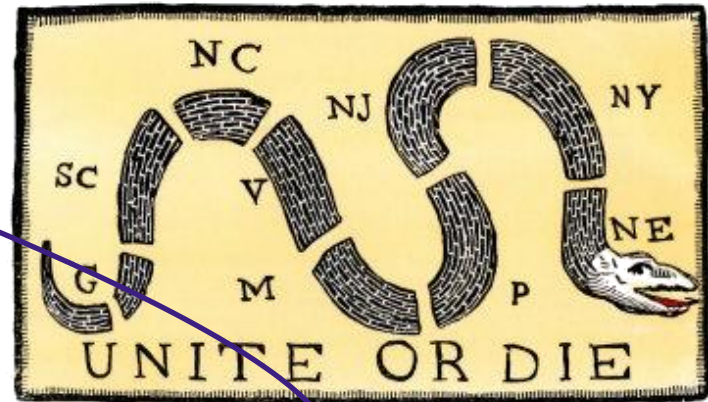
Design the settings, templates, and conventions used with SolidWorks files.

- File Properties
 - Title Blocks
 - BOM
 - PDM
 - Equations
 - Search/Select
- Document Templates
 - Detailing Standard
 - Sheet Formats
 - Title Blocks
- File Locations
- Configurations
 - Yes/No
 - Names
- File Management
 - Names
 - Locations – PDM?



Things to Standardize

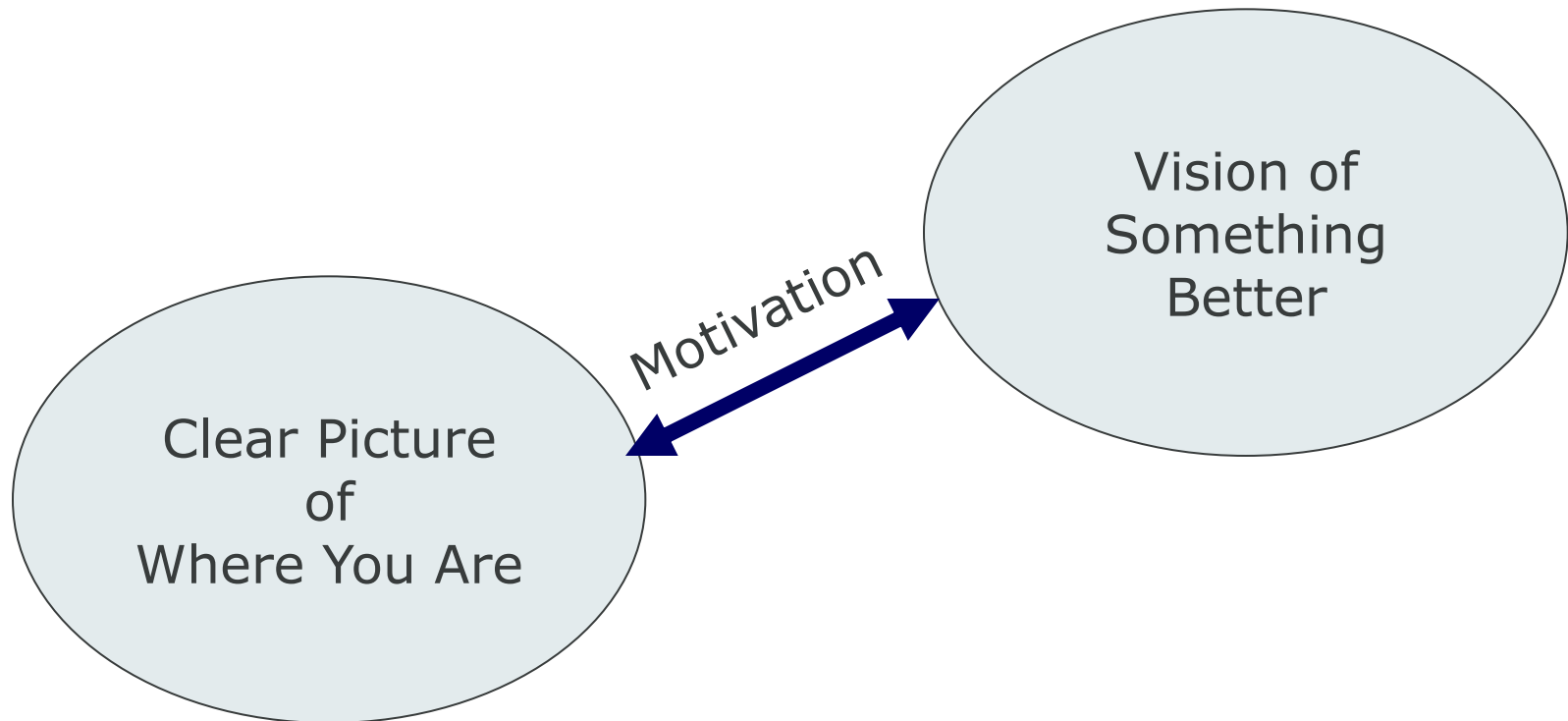
- File Properties
 - Property tab
- Document Templates
- File Locations
- Drafting Standard
 - DesignChecker
- Naming conventions
 - Files
 - Configurations
 - File Properties
- Configurations and Display States
 - To be or not to be?
- File Management



Best Practices?

- Specific details about how to USE SolidWorks.
- Advice and preferences for approaching specific tasks, designing specific products.
- Not the topic of this presentation.





- Often the de facto approach to implementing corporate standards anyway...
- “Sell” standardization to your peers based on its benefits.
 - Less work. Easier work.
- Allow users to maintain their own custom environment.
 - One size fits most.
- Enforce standards as necessary to prevent harm to others.
 - One drop of sewage...
- **IMPROVES ADOPTION AND COMPLIANCE.**

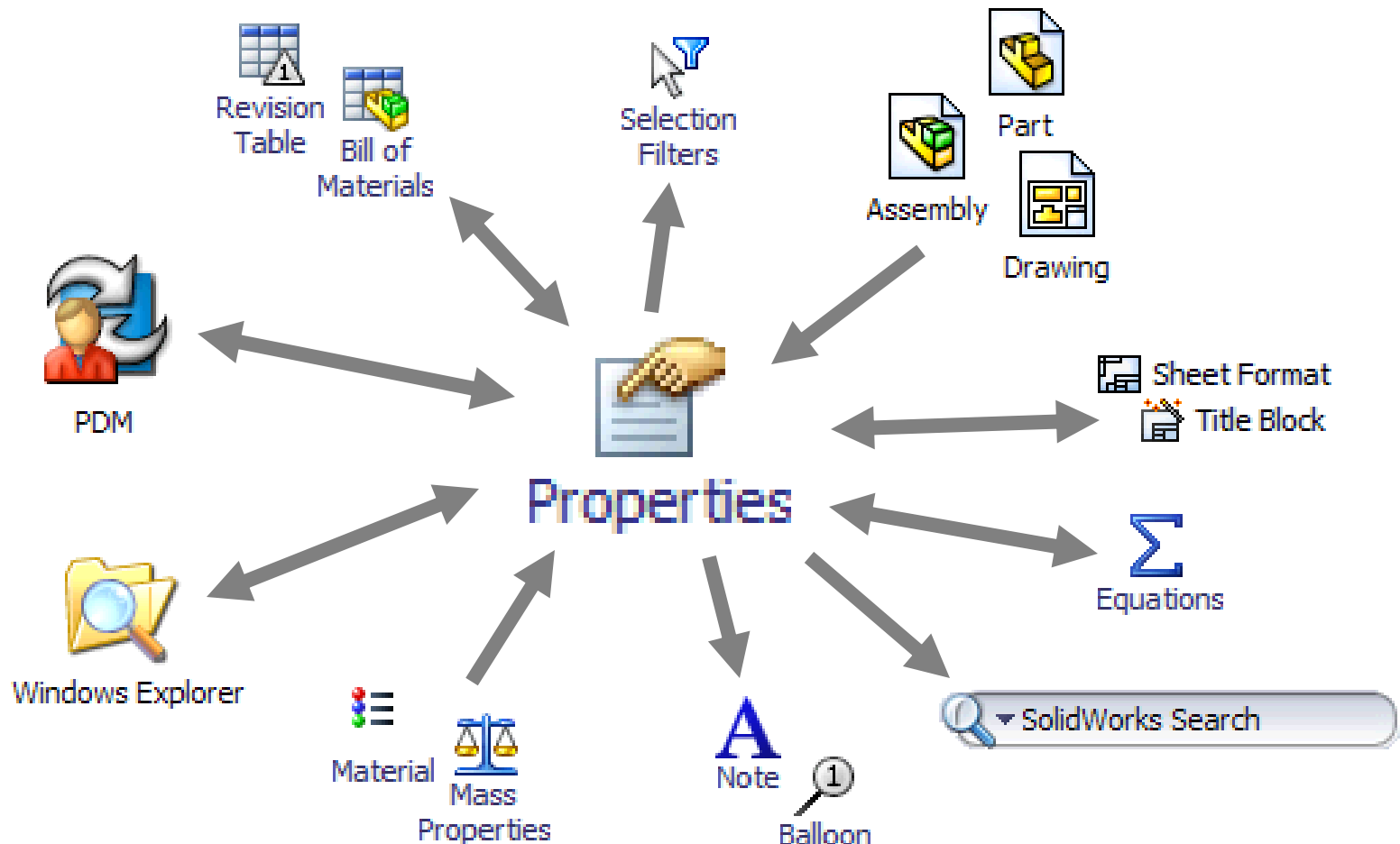




File Properties

File Properties

File Properties can be linked to many other objects, and many other objects can read (and often write) File Property values. Implement internal standards for Property names and values.



- SolidWorks comes with a list of suggested properties, that you can edit.
- Here are some common File Properties you might want to establish:
 - MakeOrBuy/Purchased
 - Material
 - Weight/Mass
 - Finish
- In the model or in the Drawing or both?
 - PartNo/Number/Part Number
 - DrawnBy
 - DrawnDate
 - Revision



Properties

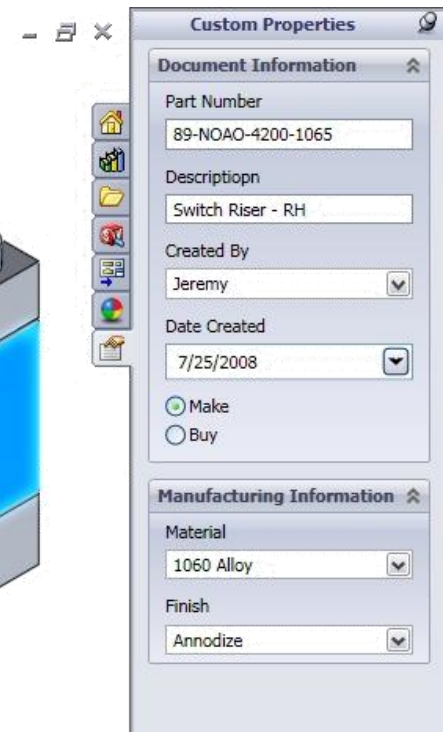
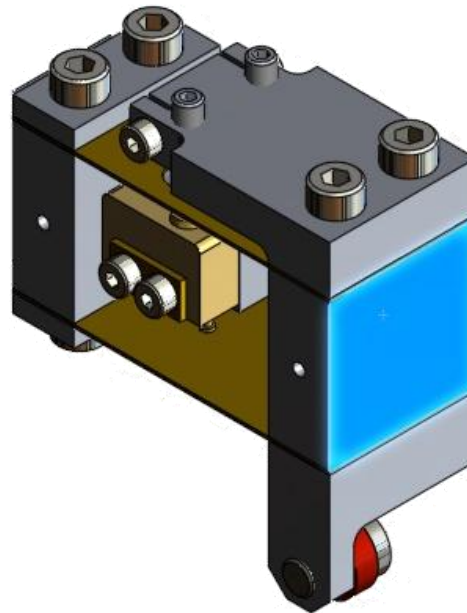


VS.



Faster File Properties

- Add a keyboard shortcut (ex:Shift-P).
- Pre-populate your Document Templates.
- Design and use Property Tabs.
 - One tab for each: Part, Assembly, Drawing
 - Be careful of accidental write.
 - Configuration-specific?



Custom Properties

Document Information

Part Number
89-NOAO-4200-1065

Description
Switch Riser - RH

Created By
Jeremy

Date Created
7/25/2008

Make
 Buy

Manufacturing Information

Material
1060 Alloy

Finish
Anodize



- Agree on the name of each property you would want linked to a Title Block, or BOM, or used with Assembly Visualization.
- Decide which properties will exist in the Drawing.
- Decide which properties will be configuration-specific (to work with the Property Tab).
- Design and distribute 3 Property Tabs (see “File Locations”).

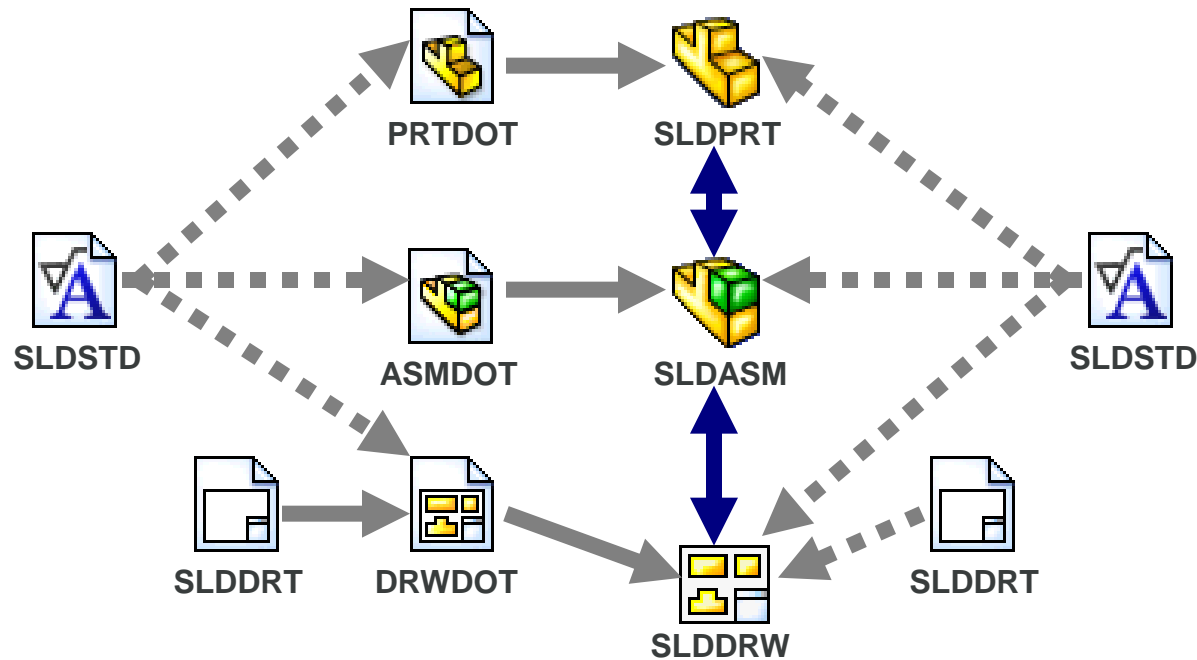


Document Templates

Document Templates, etc.

A Document Template (*.????DOT) determines the initial conditions of a new Part, Assembly, or Drawing File.

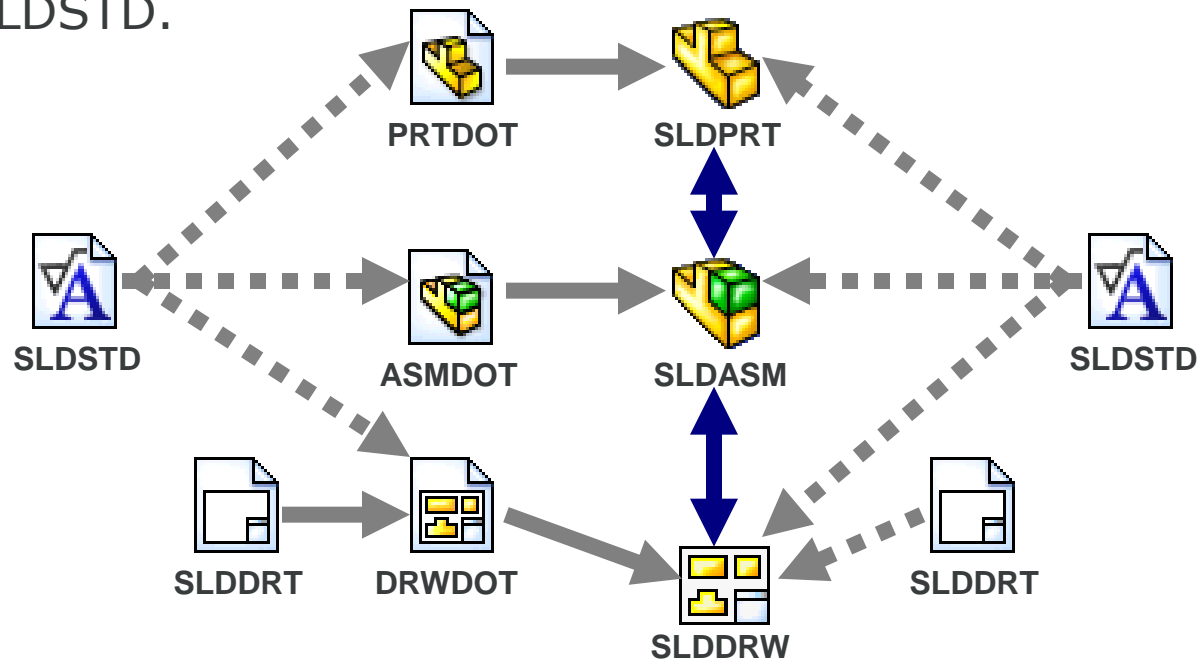
- Sheet Formats (*.SLDDRT) are often included within Drawing Templates but should also be saved separately to apply to existing files and additional sheets.
- A custom Drafting Standard (*.SLDSTD) can be applied to any Document Template or existing file!



- Defines the initial conditions at the moment a new file is born.
 - Document Properties
 - Units
 - Drafting standards
 - Image quality
 - File Properties
 - Color
 - Features
 - Layers (DRWDOT)
 - **Everything!**
- A Document Template is really nothing but an automatic **Save As...**



- Don't go overboard with your customizations.
- Typical minimums:
 - Parts and assemblies: 1 PRTDOT, 1 ASMDOT.
 - Drawings: 1 DRWDOT and 2 SLDDRT's per sheet size.
 - Double the above if you use inch and metric.
 - Optional:
1 SLDSTD.



How To...

- PRTDOT/ASMDOT/DRWDOT:
File – Save As...



PRTDOT



ASMDOT



DRWDOT

- SLDDRT:
File – Save Sheet Format...



SLDDRT

- SLDSTD:
Tools – Options...
Document Properties – Drafting Standard...



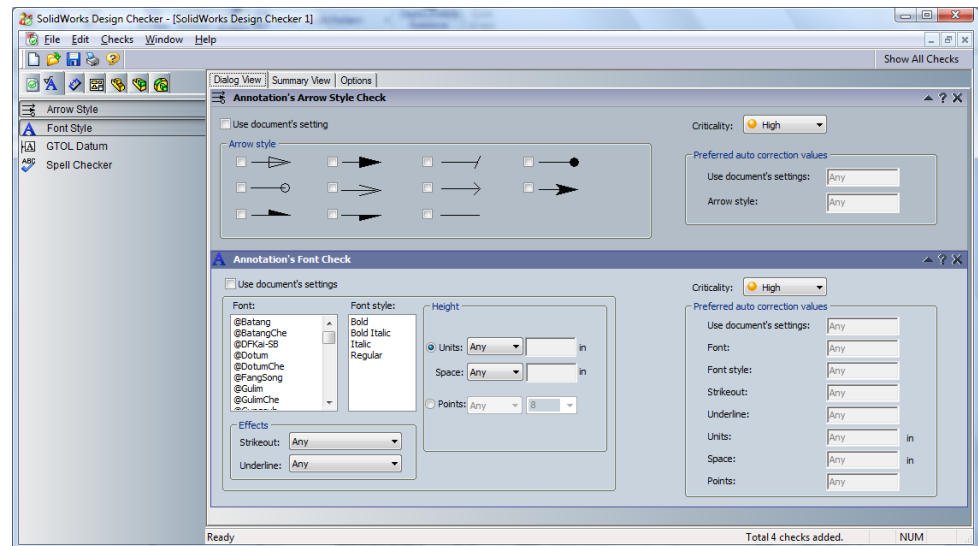
SLDSTD



- Make the effort as soon as possible to build a Sheet Format with an intelligent Title Block for each of the sizes you use.
- Define and distribute at least one Document Template of each type (PRT, ASM, DRW).
- Existing SolidWorks files cannot import a new Document Template, BUT you **can** load a new Sheet Format and/or a new Drafting Standard.
- The Design Checker can be used to verify that existing files are in compliance for many setup and property values.

(sidebar: Design Checker)

- Tool available in SolidWorks Professional.
- Checks (and helps correct) Drawings, Parts, and Assemblies for compliance with corporate standards and general good hygiene.
 - Document's drafting settings.
 - Sheet Format and values.
 - Title Block and values.
 - Layers.
 - Material.
 - Dangling dimensions.
 - Rebuild errors.
 - AND MUCH
 - MUCH
 - MORE!





File Locations

File Locations settings can ensure that all users are working with the same templates and other defining files used in the SolidWorks environment.

- A shared network folder is the most common place to put these files.
- Typically these files reside outside of a PDM vault, but in some cases it is possible to keep them in the vault.
- Tools... Options... System... File Locations...



There can be only one...

- Only **one path** is allowed for these locations:
 - Custom Property Files
 - DimXpert Callout Format File
 - Hole Callout Format File
 - Hole Wizard Favorites Database
 - Sheet Metal Bend Line Note File



- You can point these to a network location...
OR
Overwrite the default local data.
- Local path is preferred. Keep backups on LAN.
- All other File Locations can accept **multiple paths**.

You can't change your mind...

- You **cannot** browse on demand to select an alternative for these locations:
 - Document Templates
 - Referenced Documents
 - Color Swatches
 - Material Databases
 - Search Paths
 - Spelling Folders
 - Weldment Profiles
 - Weldment Property File
- Plan ahead and add the desired locations to your options lists.





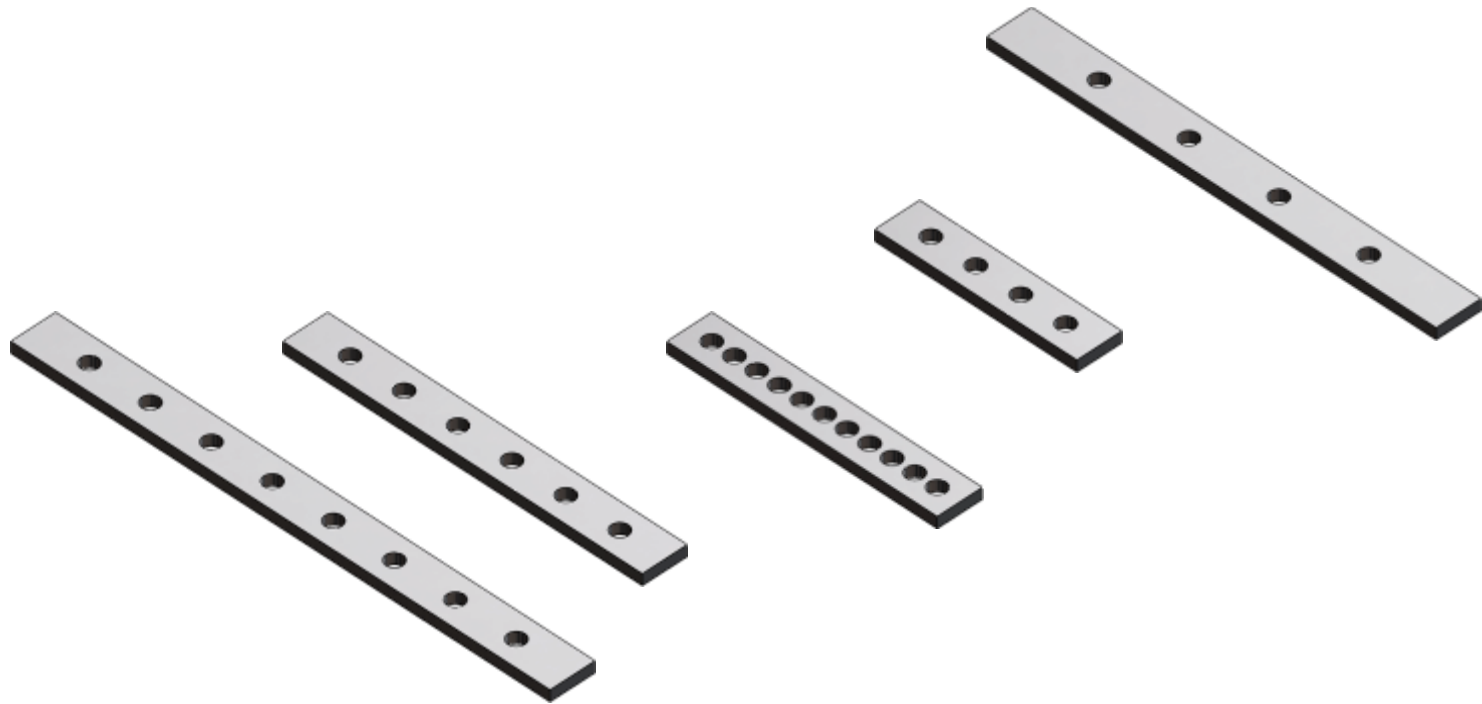
- Customize the pointed-to files if needed, and distribute them to all users.
- Share a network folder with subfolders for all other File Locations.
 - Users can add these locations to their own lists.
 - Users can populate these folders for team use.
 - Careful use of Copy Settings Wizard can automate changing File Locations.
- As needed, backup and/or vault customized files.



Configurations

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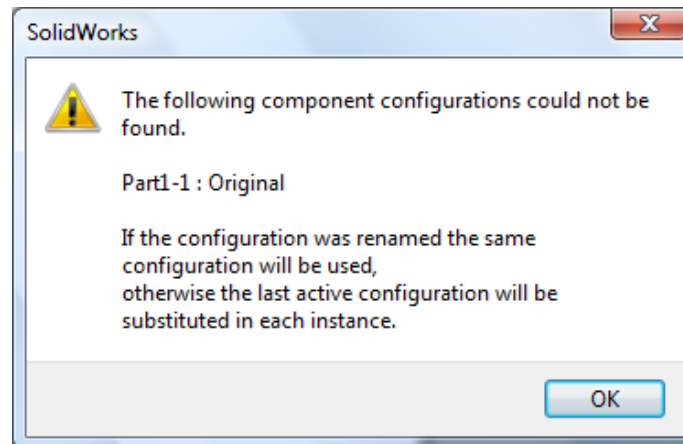
Configurations capture variations of what a model is.
Display States capture variations of how a model appears.
Decide if/when to use these technologies. Establish naming conventions for typical uses.



- Some companies do not use configurations and prefer to use multiple files for similar designs.
 - Because PDM is always done per document.
 - Because configurations bloat the file.
 - But can't use Design Tables.
 - But edits to similar designs require more work.
- You might still want configurations for:
 - Analysis
 - Performance
 - Drafting tricks
 - As Installed
 - Mfg Operations
- Establish a naming convention.
 - Especially for large assembly work.



- Decide if/when configs will be used.
- **Don't** use configs for Toolbox. Use copied parts.
- Agree on a handful of common config names.
- Renaming existing configurations is OK.

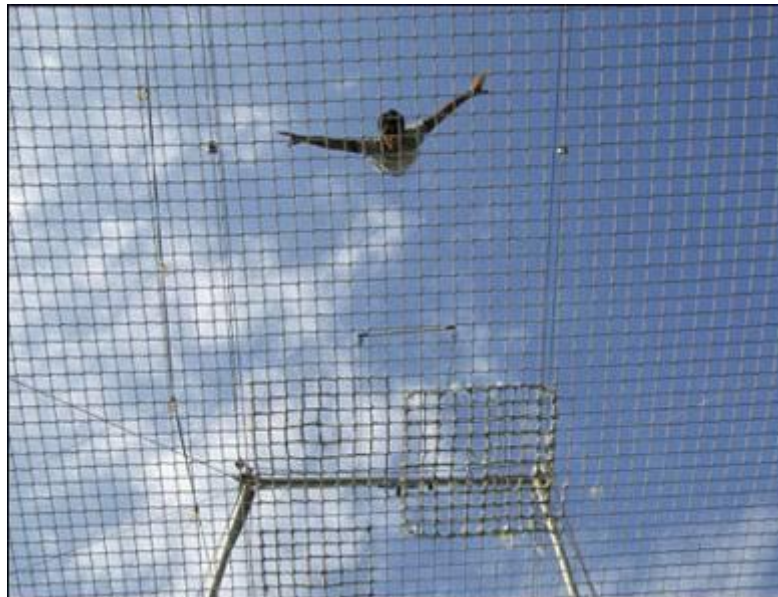




File Management

File Management

If you're working alone or with somebody, you **need** to define an organized method to create, store, retrieve, and edit your data.

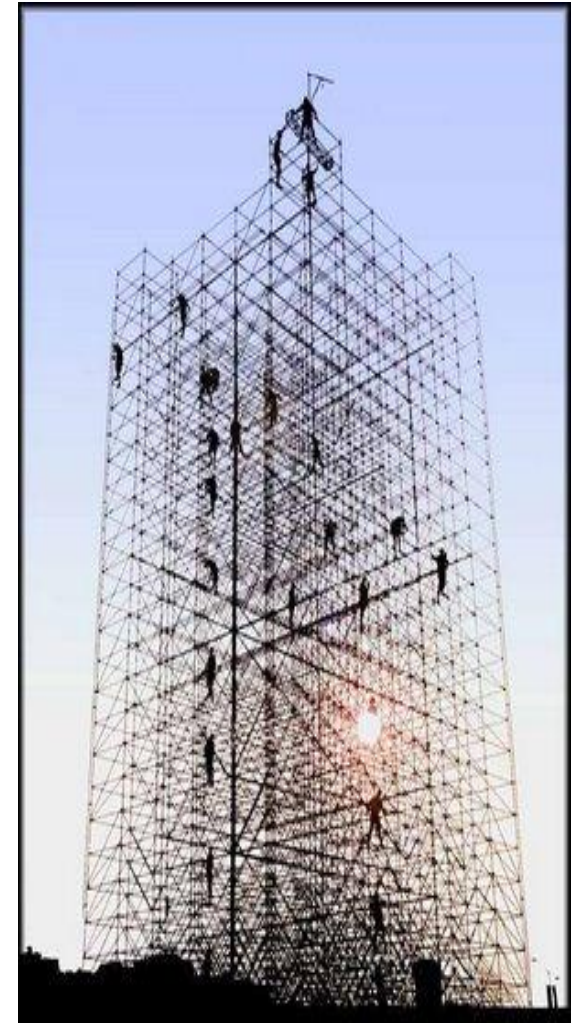


1. By God, use a PDM system.
 - If you own WPDM and aren't using it...
 - EPDM is for people who actually care how their data is managed.
2. Use **unique** file names.
 - File **name** is the only way SolidWorks finds referenced documents.
 - Unique names ensures you will never open and edit the wrong file by accident.
3. Understand how SolidWorks finds referenced documents.
 1. RAM.
 2. File Locations – Referenced Documents.
 3. Active document's current folder.
 4. Last found file's folder.
 5. Where last saved.



"Golden Rule" by Jerry Hanks

- What folder structure will you use?
 - Released projects
 - Development projects
 - Purchased parts
 - Other libraries
- How far up the ladder will you follow a revision?
 - Keep parts and their assemblies in sync?
 - Keep models and their drawings in sync?
- What is your naming convention?
 - For revisions?
 - For bringing work home?
 - For sending files to others?





- Decide what your ideal file management environment would look like.
- Start by adhering to those standards from now on!
- Develop a plan for migrating existing data into the new standard.
 - Get it under control.
 - Then keep it under control.



Summary



- You have some decisions to make in these key areas:
 - File Properties
 - Document Templates
 - File Locations
 - Configurations
 - File Management
- Everywhere, naming conventions.
- If you lack leadership, go grassroots.
 - Get everyone involved = self-governing
 - If not you, who?
 - If not now, when?



Thank You!

Jason Pancoast
jpancoast@capinc.com

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