
SOLIDWORKS BEST PRACTICES

The ideas/techniques in this document are NOT original. Many ideas come from other Solidworks users and user forums.

The document is arranged in the systematic order of how Solidworks would be used:

- 1) Sketch/model
- 2) Assembly
- 3) Drawings
- 4) Tips and Tricks

Following these guidelines will make parts, assemblies, and drawings more similar and easier to work with.

K.I.S.S. (Keep It Simple Silly) Use system default setting as much as possible.

Don't try to out think the software.

1) Part/Sketch/Modeling

- a) ALWAYS Model in real world orientation (use the main planes).
 - Ex: TOP panels should be sketched on the TOP Plane, SIDE panels should be sketched on the RIGHT Plane, BACK/FRONT panels should be sketched on FRONT Plane.
- b) Sketch on the three MAIN planes (TOP, FRONT and RIGHT) as much as possible.
- c) Use the part origin as a starting point. DO NOT "FIX" (lock) geometry.
- d) K.I.S.S. Keep sketches simple, preferably one sketch for each feature when applicable.
- e) When constraining sketch geometry, add geometric relations **1st**, then dimensions.
- f) Constraint order should be: 1) Origin/Main Planes, 2) Previous sketches, 3) Current sketch geometry,
- g) Use "Convert Entities" and "Power Trim" instead of sketching new lines. "Convert" automatically adds a constraints.
- h) Use mirrors and symmetry (about centerlines) whenever possible.
 - An infinite centerline is a good visual reference. A full screen zoom will only include drawn geometry (not an infinite centerline)
- i) Sketches should ALWAYS be "fully defined" when completed.
- j) Use the Hole Wizard for ALL HOLES and SLOTS. (Hole wizard features must be 100% matching to appear as a single callout)
 - Use the appropriate Hole Type (C'sink, C'bore, Slot, Drill, Tap Drill, Screw clearance)
 NOTE: The standard drills in our CNC are 3mm, 5mm, 8mm and 12mm drills. All other sizes are routed.
 Standard Predrill is 1/8 (3mm) x ¼ deep.
 - If a hole is required for a threaded insert, add the insert to the part as derived part.
- k) Pattern Features or Bodies rather than Sketch entities.
- l) NAME FEATURES so they might be easily found and understood by others.
- m) Handed parts should use Mirror Body>Delete Body(feature) about a main plane as the last feature in a tree. (see [1/a](#)) This practice makes assemblies more initiative. Handed features are symmetrical. Additional features require a new part #.
 - When modifying ALWAYS roll the Feature Manger back before the mirror to making modifications. When finished, roll to end.
- n) Part Configurations should be named as it will appear in a drawing assembly BOM. (also see ENG Part No. Log)
- o) Handed Parts should have a LH/RH suffix in the configuration name.
- p) Part Configurations should be kept minimal (rule of thumb: scrolling through configurations BAD!)
 - Avoid Derived configurations!
- q) Parts should be UNIQUE TO SPECIFIC PRODUCT LINES (and products) even if part/features are identical (EX: doors, drawers).
- r) Assign a material in the Feature Manager using the pulldown menu. DO NOT manually enter a material in the File Properties tab.
- s) File Properties must be filed out for all configurations (Configuration Specific), as they populate Drawing Title blocks and assembly BOMs. (All the fields in the "Custom" Tab should ALWAYS be empty)
 - Description: Product Line, type, part, size, LH/RH EX: ZORI CREDENZA, SIDE PANEL, 36in, LH
 - Revision: All part configurations must be the same. (drawing revs may be "NO CHANGE THIS SHEET")
 - Drawn by: (3 letter initial) Drawn Date: (date of part creation)
 - Weight: "SW-Mass@@___.SLDPRT"
- t) When complete, a part should:
 - Hide all sketches.
 - Purge all unused sketches and features (In feature manager RMB->purge unused).
 - Be saved in a ISO, full screen view. (thumbnail view will be correct)
 - NO rebuild errors

2) Assemblies

- a) Build an assembly in the order it would be built in on the floor. (see [1\)a\)](#)
 - Top Level assembly names/numbering should always match the catalog price list numbering.
- b) 1st part insistence of an assembly should always be fixed (part origin to assembly origin)
 - When starting an assembly, use the green check mark in the feature manager. This will automatically fix the part origin to the assembly origin. DO NOT click on the work area as this will fix the part at a random location.
- c) All proceeding parts should be mated in the following order (as much as possible):
 - 1) To Assembly Main Plane/Origin (Assembly/Part Main Planes are the most stable geometry)
 - 2) To another common parts Part Main Planes/Origin (mirrored or repeated part)
 - 3) Largest surface to largest surface
 - 4) Edge, Point, Vertex (coincidences) should be avoided.
- d) Always work TOP DOWN as it would be built physically. Group common components together in the Feature Manager.
- e) AVOID Assembly Mirror Component! Handed components may not mirror symmetrically.
- f) Use subassemblies for common/repeated groups of parts (EX: bolts, nuts, washers, divider/shelf combos, drawer boxes).
 - Internal subassemblies may be used ("Promote" the components in "configurations manager" of the subassembly).
 - "unmeasurable" parts (Glues, tape, paint, fabric, etc...) that should appear in a BOM may be added using Internal components (empty parts). These components have no features (complete the property tab). Internal components exist only in the top level assembly (not in PDM).
- g) Handed/Mirrored parts should be mated:
 - 1) Coincident the Main Planes of each Mirrored Part (Front to Front, Top to Top, Right to Right).
 - 2) Fully constrain the 1st instance (in the feature manager) of the Mirrored Parts.
 - 3) Add a Symmetrical using the parts Main Planes OR Width constraint using surfaces of a mirrored part.
- a) Handed Assembly constraints should be labeled "RH" or "LH" prefix.
 - Handed constraints may be placed in "RH" or "LH" folders for quicker isolation.
- h) If Profile or Concentric mate is used, use "Lock Rotation" when possible.
- i) Bolts, nuts, and washers should be mated to each other 1st, then to the hole feature. This simplifies the mating scheme of models and is easily modifiable.
- j) When patterning common parts (example: levelers, screws)
 - Fully constrain the 1st instance than:
 - Use a component pattern (Pattern Driven Component Pattern or Sketch Driven Component Pattern)
 - Pattern Driven Component Pattern require Hole wizard features. (This will not work on a mirrored hole/body)
 - OR constrain all common planes of all the components to the 1st instance, then each component to the secondary part.
- k) File Properties must be filed out for all configurations (Configuration Specific) as they populate drawing Title blocks and assembly BOMs. (Always clear all fields in the "Custom" Tab)
 - Description: Product Line/type/ASM/size, LH/RH EX: ZORI CREDENZA 36in LH
 - Revision: All Assembly configurations must be the same. (drawing revs may be "NO CHANGE THIS SHEET")
 - Drawn by: (3 letter initial) Drawn Date: (date of part creation)
 - Material: " SEE COMPONENTS "
 - Weight: "SW-Mass@@___.SLDASM"
- l) When complete, an assembly should:
 - Hide all sketches.
 - Purge all unused components and features (In feature manager RMB->purge unused).
 - Be saved in a ISO, full screen view. (thumbnail view will be correct)
 - NO rebuild errors (RMB "rebuild all assemblies" is always a good idea)

3) Drawings

- a) Templates and Drafting Standards must be used for all drawings.
 - Drawings preference is B Size. 3 place decimal for components and 2 place for assemblies.
 - **DO NOT MODIFY SHEET FORMATS OR SYSTEM VARIABLES.** Templates are designed to populate based on the part properties.
- b) **ALWAYS** use the default MAIN VIEW. (A main view will have an "Orientation" option, child views will not)
 - Change the orientation/rotation of MAIN view to get the desired projections. **DO NOT delete the default main view.**
 - The main view should always be inked to the sheet scale. All other views should ALWAYS be projection of the main view. All view properties (Scale, Display Style,...) must be "Linked to parent" (the main view).
 - Sections and detail views scale and orientation may differ from their parent view but this change must be noted in the view label.
 - ISOMETRIC view are for visual reference and may be scaled independently.
 - Drawings should use the least number of views to completely define the part. Remove unnecessary views.
- c) Use PDM stored blocks for standard text, symbols and shapes, as much as possible.
- d) Symbols/text that apply to a specific view must be place in that view (use view focus/lock)
- e) All text should be upper case (exceptions are "mm" & "x" (variable))
- f) Drawings may be Ordinate or Baseline dimensions. Avoid mixing dimensioning styles.
 - Ordinate should be the 1st choice of dimensioning styles and should be used for parts with a single configuration.
 - The 0,0,0 drawing origin must be consistent through all projected views!
 - Base line should be used for parts with multiple configuration that will include tabled dimensions.
- g) Drawings may contain TABLED dimensions (within reason)
- h) The Hole Call Out feature should be used for all holes. This will pull in Hole wizard dimensions.
- i) Holes required for a threaded insert should have the insert in the part model (See [1j](#)) and should be noted with the insert description and part number in the drawing.
- j) HOLE TABLES may be used for parts requiring more than 3 different sizes holes.
- k) When modifying old drawings, templates and sheet formats should be updated to the current PDM stored format.
- l) Drawings MAY be multi sheet.
 - All sheets will have the same drawing name in the title block and will be distinguished by sheet number ("X of X").
 - Multi sheet drawings have ONLY 1 Rev block (on the 1st sheet)
 - Rename each sheet name in the feature manager for reference (EX: BOM, BUILD, DIMS)
- m) Assembly drawings may be multi sheet (All views must be linked to the main view across all sheets!)
 - 1st sheet should isometric view with a top level BOM.
 - 2nd sheet should an exploded isometric view with a BOM of panels only.
 - 3rd sheet should be overall dimension.
 - BOM must be:
 - Sort in numeral order
 - All purchased components/hardware should be moved to the bottom on the BOM.
 - Verify part revisions (remove revisions from purchased components/hardware)
 - 5) Top level assemblies may also contain a note explaining catalog variables [a](#)) EX: LNGB66@#%+%^&*!
- n) Drawing Revisions are linked to part/assembly properties (see [1j](#)) & [1k](#)) All part/assembly revisions must be the same.
 - Drawings Revisions may be "NO CHANGE THIS SHEET" to reflect configurations.
 - DATE: (date of drawing modification) DESIGNER: (3 letter initial)
- o) When complete, a drawing should:
 - be saved in a full screen view.
 - (PDM) Drawings should have PDF/DXF, Copy/Paste as reference attached to the. SLDDRW
 - A copy of the PDF should be placed in the appropriate FTP folder.

4) Tip/Trick & General Rules of Thumb (“P” = Part, “A” = Assembly, “D” = Drawing)

1. **ALL** K.I.S.S. There are MANY ways to do the same thing. Use default setting as much as possible. Don't try to outthink the software. Don't model in such a way that features are nested (buried) in configuration specific options.
2. **ALL** When starting a new part, use the existing template parts and modify from there.
3. **ALL** When using copies of existing products, do not perpetuate bad practices. Correct issues as they are found. Consult with SR Engineers and implement the Best Practices found here.
 - **ALL** Use the Engineering Part Numbering Log (Refer to the “Logic” page for numbering and naming)
 - When using variables in **PART NUMBERS** (for sizes or options), use a single (lower case) “x”
 - Part numbering and description should follow the catalog Variables (in the same order as the catalog).

Ex.: Part number **LNL-1162-PLT-x** could contain a configuration **LNL-1162-PLT-84-H-R-A**. The description for that configuration would be “**LIGOURI PO PLINTH, 84, HAD, RH APPLICATION, ACRYLIC DOOR**”
4. **ALL** All parts should be PRODUCT specific.
 - EX: DOORS - Although doors are repeatable/similar, a Standard Credenza door should NOT be used in a Buffet Credenza assembly. Each assembly should have separate part files.
5. **ALL** Use CAPS for everything (exception are variables and unit suffixes)
6. **ALL** Only open what is needed! Avoid have 10-20 parts/asm/drawings open!
 - Use Ctrl>tab to move between screen. Or use split screen vert/hort.
7. **PA** When starting a PROJECT from scratch;
 - **P** Start on a plane which would place the part in real world (assembled) orientation. The sketch should control the panel width and height (the extrude feature should control the panel thickness).
 - **P** Use a rectangle in the sketch for overall size.
 - **P** Extrudes should be used for panel thickness.
 - **A** Place the Part in assembly (to determine panel overlaps)
 - **P** Go back to the Part to place screws and dowels.
 - Cams should always be the driver of feature location. Screws are the result.
 - **P** Add any other features
8. **P** SKETCH it is always recommended to start a sketch with a vertical and horizontal reference line (coincident to the origin). This establishes a relationship to the origin to be used for sketch entities and part relationships.
 - These lines can also be used for mirrors and pattern.
9. BOX SELECTION (sketch entities, parts & assemblies)
 - FENCE (Box select from left to right) Selects only items completely within the box.
 - WINDOW (Box select from right to left) items crossing the box boundaries are selected, in addition to the items within the box.
10. **PA TIP** Parts that contain LH/RH configuration (and related holes), Consider the 1st extrusion direction! Mirrored surfaces are recognized as a different surface and constraints to these surfaces will fail in RH/LH assembly configurations. Extrude in the direction that allows Hole Wizard features that extrude **AWAY** from the main plane. This allows parts in the assembly to be constrained to the **main planes** rather than the surfaces (which may change RH/LH).
11. **P** Inside Radii (panel cut outs) should be 1/4” (cutter used is 1/2”)
12. Lines can be sketch continually by just picking points. Lines can switch from line to arcs just by touching the endpoint of the previous segment. This will additionally add constraints such as vertical, horizontal, coincident, tangent... as the line is sketched.
13. **ALL** Always work top to bottom in the Feature Tree
14. **ALL** If a file is being used for reference and move things around and a “RESET” to the original is required, use "File -> Reload" rather than close and open. This does the same thing without having to wait.
15. **PA** For part construction and assembly constraints, Always think: 1) Main Planes, 2) Geometry, 3) Dimensioning
16. **P** Use 3 or 4 plc decimal (to accommodate common fractions) in models. Round numbers to 4 places.
 - Type the dimension needed: “3/16” for a fraction, “2” for 2.000 inches, 2mm, etc.
17. **P** When starting a sketch, start from a rectangle and use “Add Construction Lines” -> “From Midpoints” OR make 2 construction/Centerlines (infinite) for the X/Y access

18. **P** When sketching rectangles, dimension X/Y to the centerlines (not line to line). This allows trimming of the geometry without affecting the driving dimensions.
19. **P** Try to make panel as symmetrical as possible (Cams, Hinge holes, shelf pin holes). This make assembly easier.
P If asymmetry is required on a panel (up/down, LH/RH) offset a Cam & Dowel in even 1" increments.
20. **P** When creating a part based of a metal extrusion (or anything with a base part that is cut to length), create it using a mid-plane extrusion. The Extrude length can be configured to be the cut length OR a configured cut feature maybe used to show part length. If a cut feature is used, make it symmetrically constrained about the main plane (to keep the origin in the center of the part).
 - Name the part based on the extrusion.
 - The configuration should be the CUT length because a cut length can be reused in other applications.
 - The Configuration Properties -> Bill of Materials Option should pull from the Document Name (Not Configuration)
 - EX. Part Number: LAN-2004-x
 - Configurations: 12, 13, 14 etc.
 - Configuration Properties -> Bill of Materials Option
 - Use the appropriate configuration in the ass and in the Drawing BOM the part will read "LAN-2004-x"
21. **P** Don't draw unnecessary geometry (use the base planes/origin whenever possible).
 - Exception - Centerlines for reference in sketches.
 - Make them infinite length (this will make them stand out but zoom extent will only be to the other geometry)
 - Constrain centerlines to the origin
22. **PD** When dimensioning, use a Line to Line selection 1st. This has a higher likelihood of reattachment if a sketch is changed.
23. **P** Dimensions may be entered as Standard or metric (Using a unit Suffix "in" or "mm" when entering a value)
 - Let Solidworks do the math. Units may be mixed using suffix (ex. .975 in. + 2mm).
24. **P** Dimensions drawn to a centerline will give 2 options; TO the centerline is a single dimension, across the centerline will double the dimension (as if it is a mirror)
25. **P** When sketching (with the intention of mirroring)
 - Sketch a centerline (Infinite and coincident to the origin)
 - Sketch and constrain 1 side of the geometry
 - mirror
 - fully constrain the resulting geometry
26. **P** Fully contained sketch geometry is BLACK. If geometry is blue, select it, and try to move it.
27. **P** Use Dynamic Visual Reference Visualization (RMB in the feature manager). This will make Parent/Child relationships visible.
28. **P** Equations/LINKED dimensions may be used for often occurring dimensions in Sketches.
29. **PA** Constraint order in a sketch: 1) Base Planes/Origin, 2) sketched geometry, 3) previous sketches/Features
30. **P** When creating multi piece parts, **USE SPLIT (Not an extrude cut .00001!)**
31. **P** Before modifying a part, use a roll back to understand part construction.
 - ALWAYS modify TOP down. Group logically related features together.
32. **P** Locate features in the feature manager by selecting a surface of a feature. It will highlight in the feature manager.
33. **PA** When repairing errors, repair them in feature tree order. Fixing from the top down.
34. **PA** Use predrill when possible. Slots should be predrilled to slot center.
35. **PA** Try to avoid display states in parts (display states may be used in assembly for view creation)
 - Uncheck "Link display state to configuration"
 - Delete all but Display State 1.
36. **PAD** Unlink DISPLAY STATES. (Configuration TAB->Display States->)
37. **P** Use Feature pattern, up to reference for features
 - Feature Patterns are configurable. Sketch pattern are not!
38. **P** Common features are located in:
 - Part templates (File ->New) Common panels for a good starting point.
 - PDM\06 Engineering Reference Files\01 ENWORK DESIGN LIBRARY (Library features)
 - Dissolve to features after use.

39. **P** Predrills should be used when possible.
40. **P** Common features for lock, grommets and door hinges are stored as blocks in PDM.
- Door hinges have centerline to aid in alignment in assembly.
41. **P** Hole Wizard:
- defaults to the last hole inserted. If it's changed to a flat bottom drill, it will continue to create flat bottoms until it is reset to default (118° point)
 - Use Hole Wizard Favorites (PDM\07 Solidworks Templates\Hole Wizard Favorites Database)
 - If a part requires a threaded insert, insert a derived part.
 - Identical copies of Hole wizard features can be created by CTRL + drag to new surfaces.
42. **D** When using Ordinate dimensions:
- Keep the 0,0,0 corner consistent in all projections.
 - Keep dimensions on the 0 side as much as possible.
 - Dimension leader that cross the midline should be moved to the opposite side (break alignment)
43. **D** Centerlines are a good visual reference. Use this to establish symmetry. Drawing note "Part is symmetrical about centerline, unless otherwise noted."
- Sketch a Centerline in the views for reference and label them "☉".
44. **D** Naming Views is helpful. (Flat, Detail, BOM, DIMS, etc.). Label Views with name, scale, and rotation. (use a link! \$PRP:"SW-VIEW NAME(VIEW NAME)")
45. **D** Annotations can be edited in a group. (Text may be added to multiples dimensions) EXPLANATION NEEDED
46. **P Purchased Components**
- If creating a part from a metric drawing, features should be modeled using metric values. (3mm (.11811) ≠ .125)
 - When using models from suppliers (iges or step)
 - Save them as a Solidworks part (Use "save as" "part")
 - Save in real world orientation (see [1/a](#)) Part/Sketch/Modeling a) Origin and planes on logical mating orientation. Use Move Body to locate them.
 - If the part is an assembly of components, create configurations that may be useful in its applicable. (EX drawer slides: OPEN/CLOSED)
 - **Verify dimensions to a physical part!**
 - Complete the part properties!
47. **P** Suppress a sketch rather than the feature (Parent child relationship will suppress the sketch). Suppressing the feature may not suppress the sketch.
48. **PAD** Multi body part should be avoided with the exception of WELDMENTS. A WELDMENT is a single orderable part number. DO NOT make an assembly for a weldment. EX: weldnuts on a plate are a single orderable part number. Create as a multibody part.
- **P** When creating a multi_part use the rules for creating standard parts and assemblies:
 - KISS!
 - Use main planes as much as possible.
 - Use construction sketches (3D overall dimension, sections work well)
 - **D** A drawings of a weldment may be multi-page.
 - 1st page should be a completed part with 1 rev block
 - Each subsequent page may be individual components of the weldment (No rev block)
 - (i) Each part drawing will be distinguished by page number ("x of x")
 - (ii) The Title block "Title" will always refer to the 1st page (the complete part)
 - (iii) For drawings views, use "Select Bodies" to isolate components
46. **D** Sheet metal drawings may have a "Flat" view (used for quoting) but it must be noted "FOR REFERENCE ONLY" (in the common metal note) and only show overall blank dimensions. The logic is: This is the part I need supplier to make. Ask the Supplier to help get there!
- *ASK THE SUPPLIER for advice on forming!*
47. **D** When adding table, select Use Anchor Points. When used with the current sheet format they will locate automatically.

49. **D** Hole tables can appear out of order or skip order. Table may be reset in the feature manager by changing from Letters to Numbers and then back again.
50. **A** Locks, hinges and drawer slides may be placed in assemblies.
- Slides clearance should be on the high side (loose fit) of the tolerance. Shims may be added but material cannot be remove.
 - Add predrills.
51. **A TIP** When starting and cabinet assembly from scratch, start with blank (featureless) panels 1st (roll back/suppress if necessary) Construct the cabinet with offsets in mind and then place connecting hardware.
52. **PA** Face to Face models are **OK** but be aware of clearance! Model in clearances when appropriate.
53. **PA** To determine how a part is free when constraining, simply grab the part/geometry and try to drag it in any direction.
54. **PA** When making parts/assemblies with multiple configurations use “Configure Feature” for modifying multiple dimensions/features (using the Excel like table)
55. **PA** When making parts with multiple configurations, use a base “dummy” configuration to be used in a drawing.
- EX CYM-1000-x.sldprt (Metal part with multiple configurations)
 - Create configurations CYM-1000-x, CYM-1000-10, CYM-1000-12, CYM-1000-14
 - Link the LENGTH SPECIFIC part properties to “configuration”
 - Link the GENERIC “x” part properties to “file name”
56. **PA** When making Parts/Assemblies with multiple configurations that may be used in a tabled drawing, use “dummy” configuration(s) of the smallest size to be used in a drawing.
- EX CYM-1000-x.sldprt (Metal part with multiple configurations)
 - Create Part Configurations: CYM-1000-x, CYM-1000-10, CYM-1000-12, CYM-1000-14
 - Link the part properties to “configuration”
 - Create Assemble configurations: ASM-1000-x, ASM-1000-10, ASM-1000-12, ASM-1000-14
 - Link the assembly properties to “configuration”
57. **PA** Design tables may be used to set up multiple configurations but should be deleted after use. **DO NOT save and/or check in a part with a design table driving the part.**
56. **D** When dimensioning drawings, try to dimensions edge to edge/line to line (2 selections) rather than line (single selection) or point to point. If changed, an edge to edge dimension has a higher likelihood that it can be reattached to other geometry.
57. **DA** When creating exploded views for a drawing. Exploded view should be in the configuration. Exploded view can be copied between configurations (pick and drag). Warning: when coping drawings, exploded view a can be present in a drawing view that is NOT actually in a configuration.
- **A** Explode as it would be assembled
 - **D** Use Auto Balloon. Auto Balloon WILL NOT balloon item that are not visible in the view.
 - Once placed, move balloons to a logical location. Attach to edges NOT faces (This will display an arrow head)
- **PDM**
 - Use PDM functions in File Explore 1st, rather than the Solidworks PDM “Blueberry”
 -
 - Try to check out **ONLY what is intended to be modify.**
 - PDM will always prompt to check out when opening. **DON’T** check items out until they have to be modify.
 - **TIP** Check out one at a time.
 - To modify a part contained within an assembly:
 1. Open the assembly (do not check out)
 2. Open the part from the assembly (check out only the part) and modify.
 3. Open the associated drawing, check out, and update.
 - a. When <OPEN> Filter for parts and drawing and see what is currently checked out.
 4. Check out the assembly and drawing.
 - **TIP** When checking out, UNSELECT everything 1st and then reselect only the items needed for check out.

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- DO NOT check in a part/assemble with rebuild errors! This includes under constrained sketches in parts and under constrained parts in assemblies.
- Drawings should have PDF/DXF, Copy/Paste as reference attached to the .SLDDRW (drawing must be checked out)
- Always leave a comment when checking in. (ANYTHING so that there is some record of what is done)
 - One of my Favorites is “THANKS TWB”
- Use **Copy Tree** when making copies of Parts/Asm/Drawings. This will allow items to be copied as a group and keep relations. It also allows for immediate rename.
- Files that are Checked out and Modified (parts/assemblies/drawings) are ONLY SAVED ON THE LOCAL COMPUTER where it was checked out on. Files are **NOT SAVED in PDM** (or the network) UNTIL they are checked in back into PDM. To avoid data/work loss, it is suggested to use the Check in/Keep checked out to save work to PDM at major design changes.
- **DO NOT** create layers of file folders. Folders should be Product Family/Product Type (EX: Zori\Flip Table) there is NO NEED for creating sub folders for assemblies\components\drawings.