

SolidWorks Tools and PDMWorks Vault Tutorial:

1. Download LIGO Customised Tools from the vault/DCC
2. Draw up ECD Attachment Arm on LIGO_Part Data Templates
3. Define ECD Attachment Arm Custom Properties
4. Insert ECD Attach. Arm and 4x4 array into the D040xxx_Test Mass Assembly and add LIGO Customised Bolts.
5. Create .JPG of assembly
6. Create Engineering dwg of Assembly using CAD dwg Templates
7. Using Macro copy across custom properties from part to drawing.
8. Create Bill of Materials (BOM) in Assembly Drawing
9. Create .PDF of Drawing
10. Upload the files to Vault, noting features of the vault.

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Referenced Documents (All on DCC and PDMWorks Vault)

- LIGO-D030382-04
 - A Summary of the Drawing & Data Templates, Macros, Bill of Materials and Customized Toolbox created for SolidWorks & an Introduction to the LIGO Caltech PDMWorks Vault
 - .pdf and a .zip file
- LIGO-T030143-03
 - Designing in Solidworks: Customised Tools for Design & Documentation of LIGO Parts, Assemblies & Drawings
- LIGO-D030383-04
 - Customised Toolbox for SolidWorks
 - .pdf and a .zip file
- LIGO-D030384-08
 - Bill of Materials (.pdf and a .xls)

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- **File names** File names must be unique.
- **Project names** Both project names and project descriptions must be unique.
- **File storage** Only one copy of a document is stored in the vault, no matter how many other documents reference it.
- **Revisions** Revision numbers can only go forward. Each revision of a document is kept in the Vault.
- **Open** Anyone with access can open & download a document or print after previewing it from the vault
- **Ownership** When you own a document, you can modify it and check it into the Vault. Only the owner can check a modified version of a document into the Vault. Only one person can own a document at a time. When you check a document out of the Vault, the document must have no owner.
- **References** are logical links between documents in the Vault. For SolidWorks files, references are automatically based on assembly hierarchy. You can also assign references manually for non-SolidWorks documents.








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- PDMWorks compares the age of the local copy of a document with the vault copy, and displays the differences between them:

	White line through a blue circle	The document does not exist in the vault.
	White upward arrow in a green circle	The local view document is newer than the document in the vault.
	White downward arrow in a red circle	The local view document is older than the document in the vault.
	White equals sign in a gray circle	The local view document is equal to the document in the vault.
	White star in a green circle	This is a new document or new document reference.
	Hollow icons	The document is loaded in memory but not in an open window.
	Document name in bold text.	You need to save the document.

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Document Status

The document icon indicates the document status.

-  Magenta-colored circle on document icon You checked out this document.
-  Cyan-colored circle on document icon Another user checked out this document.
-  Shortcut icon on or next to a document icon This is a shortcut reference to the actual document that resides in another project.

Security of the Vault

- The Vault directory is accessible only to the administrator, (he/she installs updates, deletes files, maintains passwords, etc ...) Users should not have any access to the Vault through Microsoft Windows Explorer.
- The vault exists on a server that is backed up onto tape. Each machine with SolidWorks has Client software
 - Access tested from US and UK locations
 - Vault exists on 131.215.115.155
 - Files can be viewed / printed via via the web portal

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