

I-DEAS TDM Data Management Cheat Sheet

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It has become apparent that users often forget the defined rules for managing data within the I-DEAS TDM. This document covers the established rules and provides links to available detailed information where needed. These are not in any progressive order.

1. If you discover an error – fix it or notify Tony Parker to get it fixed – it will have to be corrected eventually.
2. All 3D Items must have a BORN node as the first history node; this includes items created from standard I-DEAS catalog parts, STEP, and IGES imports. If your history tree shows only one node (Orphan), use the Create Part command to get a BORN and then join your Orphan part. This gives you the flexibility to modify or change the part and maintain its usage in any existing assemblies without using replace part in the assembly. STEP imports also contain attributes that should be cleaned before the part is placed in the Library – Help for processing these files is located at: <code>\\fermi-cadsrv-1\CADENV\STEP_IMPORT_HELP\PPD_STEP_import_cleaning_notes_060908.txt</code>
3. All 3D parts should have a material to allow for weight and center of gravity. Hint: If you're making parts that can start with the Create Part command, i.e. just a named BORN. You can get and apply your material to the empty part and save it (only in your Model File) as [MaterialName]_template. You can re-use this base part by copying it to a new part name and building your part on it. It will retain the material.
4. All 3D items should have a Description, see MS-OfficeSharedCustomDictionarySetup_112608.pdf this requirement will change if we implement the FAE 3D Attribute Editor that would eliminate the need to enter a description manually, I will notify users and revise this document if this happens. NOTE: <u>Some rules on TDM descriptions.</u> <ol style="list-style-type: none">Descriptions are version specific so they could be different for each library version - WE DO NOT WANT THIS. Set it and forget it.Once a description is defined for a version it will propagate to the next version so you only have to define it once.Limit your description to 80 characters or less the same as DWG_TITLE, I-DEAS will not transfer strings longer than 80 characters to external files so we're stuck with this.All UPPER CASE, same rule as drawing title. It's just too much work to massage these in Excel since we use so many acronyms.NO commas (Excel problem) NO (" – double quotes) use IN instead, or leave units off, causes problems exporting from Excel. These can be

accommodated but cause more trouble than the value they add to a description.

- f. NO punctuation - not really valuable - use dashes to separate terms
- g. Consistent naming for all items, i.e. all MAIN INJECTOR, or MI or FMI or MAIN INJ - only one of the choices.
- h. Only abbreviations defined in the online list. If you need one added, submit it. See setting up MS Office dictionary below. Including the abbreviation list in your Custom Dictionary file the Description should pass Spell Check in Excel - there will probably be a few exceptions due to length etc.
- i. Use REAL English dictionary words or acronyms defined in the master list. These should pass a spell check if the dictionary file includes the master acronym list. These will be used for queries in Teamcenter so abbreviations and non standard acronyms will not help. Do Not string your words together with (_ - underscores) this requirement is only for item naming.
- j. If you are using I-DEAS standard catalog parts they will enter a default generic description, please edit the text to add size information etc. and rename according to current naming rules.

Example: Adding a standard 1/4 set screw **Name:**

SCR-SET_250-20_X_4_LG

Default Description: ANSI Hexagon Socket Oval Point Set Screw

Edited Description: ANSI HEXAGON SOCKET OVAL POINT SET SCREW 1/4-20UNC X 4 IN.

There is additional information available on www-cad regarding naming and managing TDM data. You can also view the current items in COMMON_PARTS to see how like items are named, you should be looking there first to see if the item already exists. If you have suggestions, hints, problems etc. let me know ASAP.

5. Whether you agree with the naming rules or not, **THEY ARE THE RULES AND EVERYONE MUST FOLLOW THEM.**

6. Since the 32-character name limit is very restrictive, consistency is extremely important to aid all users in identifying TDM items by name. Follow the naming rules and adhere to the approved abbreviations and acronyms. The current naming rules have existed for several years and are located at [I-DEAS TDM Item Naming & Numbering Rules RevA.pdf](#). You may also want to copy and try this:

\\beamssrv1\mechdrft.bd\public\IDEAS_ItemNameGenerator\ItemNameGenerator.xls See the Help worksheet in the file.

7. Follow the naming rules, all capitals, no spaces, 32 characters max. and none of the following characters "!@#\$%^&*.,':;`~+V{ }[]()=|<>? The ItemNameGenerator.xls file above includes a macro for checking names.

<p>8. Since I-DEAS does not correctly handle ‘.’ (Decimal point) in names, it was decided by representatives from each group to use ‘-’ to represent the decimal point.</p>
<p>9. All items should have a revision entered using the defined sequence -,A,B,C,D...ZZ excluding the letters I,O,Q,S,X, and Z per ASME. Even un-numbered 3D items. Hint: When you create new parts you need to perform at least one check-in-keep and enter – in the revision field. This will propagate so every check-in will keep this revision level until you correctly make a revision using the New Version command described below. You can select multiple new parts, select check-in-keep, type – as the New Revision on the first part and then Check-in-All. All parts will get the – revision. If you fail to do this and the part gets checked in because it’s used in an assembly or drawing, the revision will be blank.</p>
<p>10. The correct procedure for revising a 3D drawing is:</p> <ol style="list-style-type: none"> a. Create new version of the 3D item and increment the revision to the next level. Even if you are not changing the 3D item. b. Create new version of the drawing. c. Select new drawing version created and use Update in Library to get it connected to the new 3D item version. d. Check out the drawing and optionally the 3D item. e. When you get the drawing from bin a revision will be made. <p>You should manage 3D/2D associated items as if they were a single item, i.e. whatever you do to one should also be done to the other. Every RELEASED 3D drawing requires a unique version of the associated 3D item.</p>
<p>11. Before releasing a drawing, always verify that you have all attribute data entered correctly. This also includes correct entry of user and date information in the Revision block. Since we don’t store paper files the revision information is lost unless it appears on the drawing. You can use the LIST ALL SYM ATTRIBUTES to easily check the values. I have modified the macro so it will automatically display the results in notepad.</p>
<p>12. When you release a 3D drawing, the 3D item should also be approved to RELEASED state in the library, verify that the part numbers and revision levels match. Also, verify that the 3D item has a correct Description.</p>
<p>13. Releasing a drawing for a different user. To maintain correct revision information if you need to release a drawing when you were not the user that performed the work, you must preset the REV_BY value before running the release macro. # (Enter Calculator mode) REV_BY="I.NAME" (Users assigned string Initial.LastName) Done You can run Update IDM Attributes to make sure the value is applied and use LIST ALL SYM ATTRIBUTES to verify.</p>
<p>14. DWG_TITLE – These are normally locked after first release of the drawing, however, when you work with a drawing and see that the DWG_TITLE is not correct, FIX IT using calculator mode.</p> <ol style="list-style-type: none"> a. # Enter calculator mode b. DWG_TITLE = “NEW CORRECT TITLE” c. Run Update IDM Attributes. d. Email the corrected DWG_TITLE so I can fix the web query files.

15. Create all Model Files in the 000_MODEL_FILES Project. This makes management much easier and allows for quicker access since the Project stores no library data, only lists of Model Files. If you follow the naming rules of starting all files with your unique initials or name, you can easily use the filter with your initials* to show only your files.

16. Macro Errors – All ADMS nodes run a cleanup process nightly that removes garbage from the TDM scratch directories. The following file extensions are removed: STOP_SDRC*.CMD, .STOP*, APPREC*, error*.out, cerror*.out, clog*.out, cerror*.out, clog*.out, errtp*.out, exec*.cmds, ideas*.app, ideas*.aux, ideas*.dsp, ideas*.psf, ideas*.sf1, ideas*.sf2, ideas*.und, ferror*.out, foutput*.out, output*.out, coutput*.out, nsfError*.out, exec*.bat, *_var.lis, *.log, num_*.out, *.bak, *.aux. When you get an error from a macro, rename the .log file so it will not be removed. Without the actual log file, it takes much longer to diagnose the actual problem.

17. Clean and remove all old Model Files when you're not actively working in them, references to items you have in the file prevent other users from purging their data in the Libraries. In addition, they are just surplus overhead for the system.

Tools:

1. ModifyTDM is a very useful tool for managing TDM metadata. Help is available at C:\UGS\ideas12\ulib\modifytdm\ modifytdm_user_guide.txt, and detailed descriptions of usage from the 2007 PLM World conference are located at \\fermi-cadsrv-1\CADENV\UsingModifyTDM the detailed Help document http://www.dmscad.fnal.gov/caduser/TP_using_modifyTDM_to_match_3Ditems_to_drawings-2v1.pdf.

The basic procedure to run the tool is:

- a. From the Modeling Task
- b. In the Prompt Window Type: **/oaxx start OITERMRUN**
- c. *This should start a CMD shell in your TDM scratch directory.*
- d. **Set PATH=%PATH%;%SDRC_INSTL%\ulib\modifytdm**
- e. **modifytdm [parameters]**
- f. See the modifyTDM_User_Guide in C:\UGS\ideas12\ulib\modifytdm for Valid parameters.

2. Matching Names and Numbers: You can easily match names and numbers of 3D associated drawings using modifytdm using this procedure:

- a. From the Modeling Task
- b. In the Prompt Window Type: **/oaxx start OITERMRUN**
- c. *This should start a CMD shell in your TDM scratch directory.*

- d. Set PATH=%PATH%;%SDRC_INSTL%\ulib\modifytdm
- e. modifytdm -a -t 2 -v latest -p [Project] -l [Library] -o TAB-delimited-output