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Summary of Results of CAD File Format Compatibility Tests

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1. INTRODUCTION

A series of tests have been run to find out the best way of transferring files from Pro/Engineer to SolidWorks, the primary CAD package used by the LIGO group, and to Ansys. There are a number of reasons why we need to do this and these are as follows:

- RAL[1], the institute responsible for the design of Quad Noise Prototype for Advanced LIGO and also involved in the development of the Quad Controls Prototype, use Pro/Engineer.
- SolidWorks is unable to directly import Pro/Engineer files from both the 2001 or WildFire editions, which RAL will be using.
- ANSYS is the primary Finite Element Modelling package that will be used on the Advanced LIGO project therefore it would be useful to also have a format that could interact directly with ANSYS without further conversion.
- All assemblies, including parts and sub-assemblies from RAL, will be assembled in SolidWorks and stored on the Caltech PDMWorks Vault.
- All drawing files stored on the Caltech PDMWorks Vault will include a pdf version with an embedded CAD file of the Universal File format that we choose. The thought behind this is so that in the future, should SolidWorks or Pro/E no longer be used by the Advanced LIGO group, the parts will still be able to be opened by whichever CAD package is adopted.

To find the format that would be suit the above criteria Ian Wilmut of RAL, Calum Torrie and I tested a number of file types. The files were tested for the following:

- Compatibility
- Export file size
  - Part
  - Assembly
- File quality when Imported
  - Part
  - Assembly
- Ease of use

[1] Rutherford Appleton Labs, Oxford, England

2. COMPATIBILITY

To begin the tests, a table was drawn up listing and comparing all available import and export file types from each of the respective programs (see Appendix 1). In Appendix 2, a score has been allocated to each of the formats according to their compatibility.

The table shows that IGES was the only universal file format that could be used across all three platforms. Ideally we would like to find a format that bridges all three programs, however the latest version of ANSYS can directly import SolidWorks files so once a file had been imported to SolidWorks it could then be used automatically in ANSYS. Four file types in the test fitted this criterion - STEP, VRML, STL and VDAFS.

There were some misgivings by the design team as to the quality of IGES files as past tests using this format had shown it to be of very poor quality. It must be noted however that these misgiving may in relation to an older IGES format, the latest being IGES 5.3 – thus tests were run using this later format. The remaining formats, outside of those mentioned above, are un-usable for one reason or another, as highlighted in the table, Appendix 1, and so were not tested any further.

3. EXPORT FILE SIZE

A smaller file size is understandably better than a large one when designing complex assemblies that have many parts and that have to be stored for later use. Furthermore, the time taken to extract a file is directly linked to the size of the file hence this is a key factor in our decision of what file type to use.

The score for file size shown in the table (appendix 2) is a rank from smallest to largest file size. In some cases a small file size means a poor quality file and the score given in this category may be contradicted by the score for quality. The STL and VRML file formats come out best in this category for the export of a part file however as you can see no assembly was tested. As stated earlier this was due to the very poor quality of the imported file.

4. FILE QUALITY WHEN IMPORTED

As can be seen by the screenshots below (Figure 1), the file quality can vary greatly between formats. The user can see visually that there is a difference in quality and when it is incorporated into an assembly further problems arise due to its inability to mate to other parts. Beyond opening the part file in SolidWorks, a number of a larger assembly with were opened but with varied results (see Appendix 2)

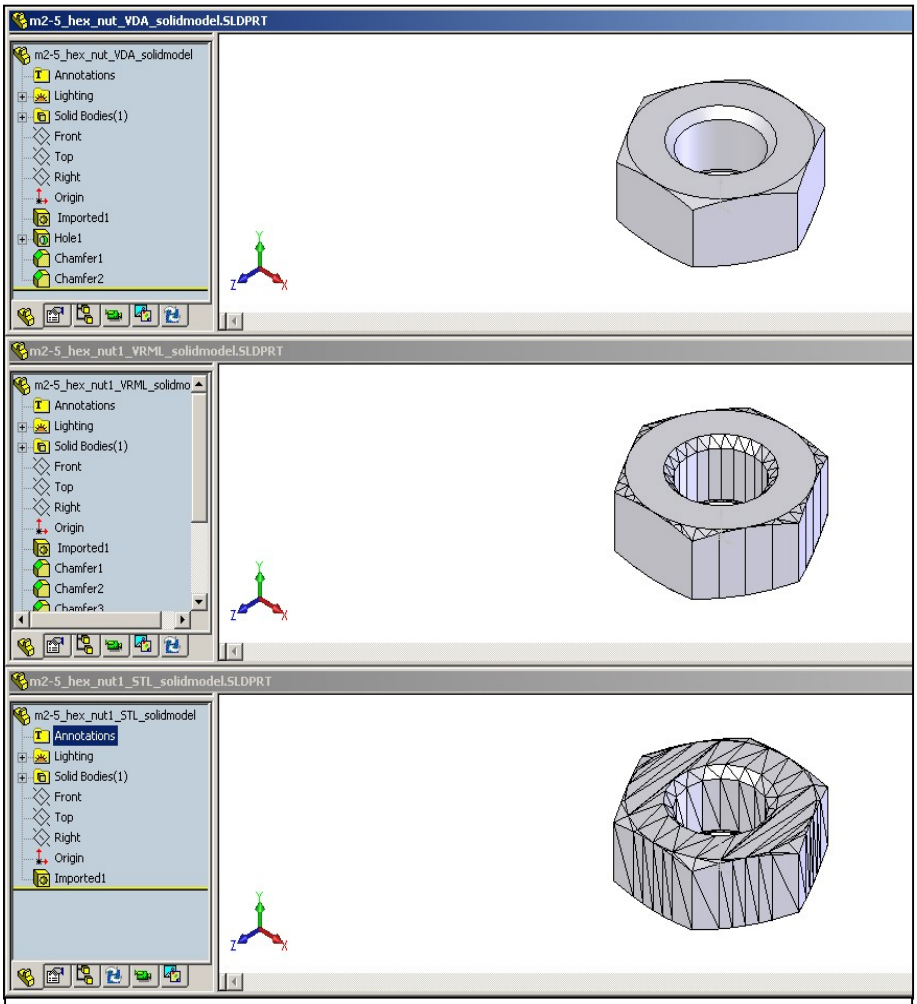

Fig 1: Screenshots from SolidWorks showing VDA, VRML & STL (from top to

5. EASE OF USE

The final category that the file formats were tested on was their ease of use. The ease of use category is the ability for a file to be sent and received with the same desired quality level every time. The ideal file format is one that has a small number of export settings and one produces a high quality output.

When a file is exported as an IGES format, for example, there is a long list of settings as to how we want to send the file (see figure 2).

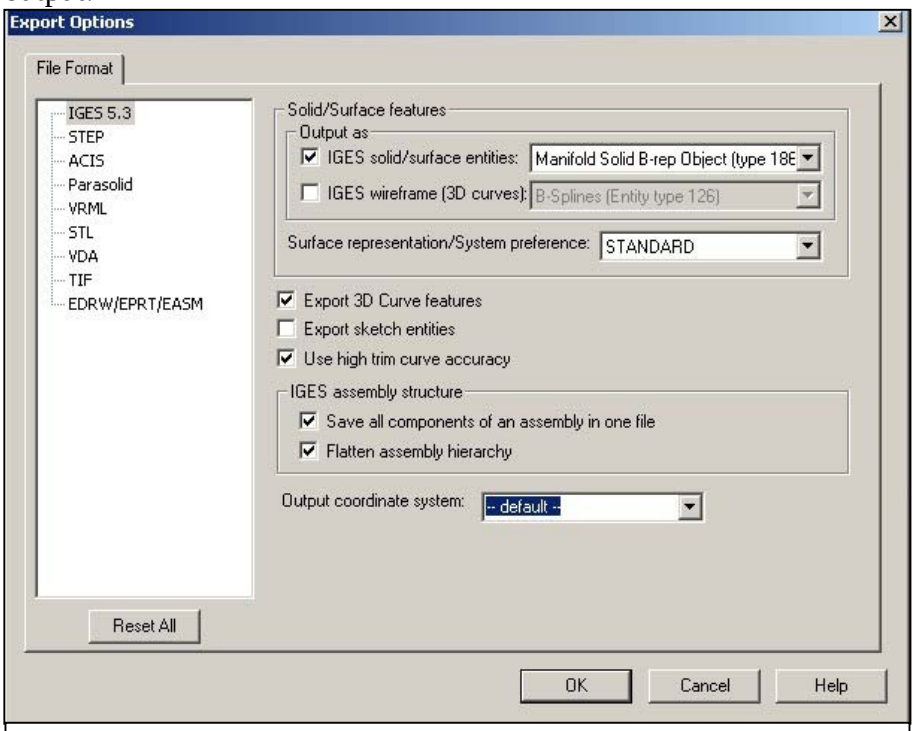

Figure 2: IGES Export Settings (screenshot from SolidWorks 2003)

Alternatively, figure 3 shows that STEP only offers a small number of options to create a file, thus along with its smaller file size seems better than IGES in this category

The weighting for this factor lower is than the other categories as we can work around the problem of having a large number of settings. By setting up a default or a macro, we can standardise the settings and thus minimise the chance of a poor quality file being created. As suggested earlier though, it would be nicer that we find a foolproof export format.

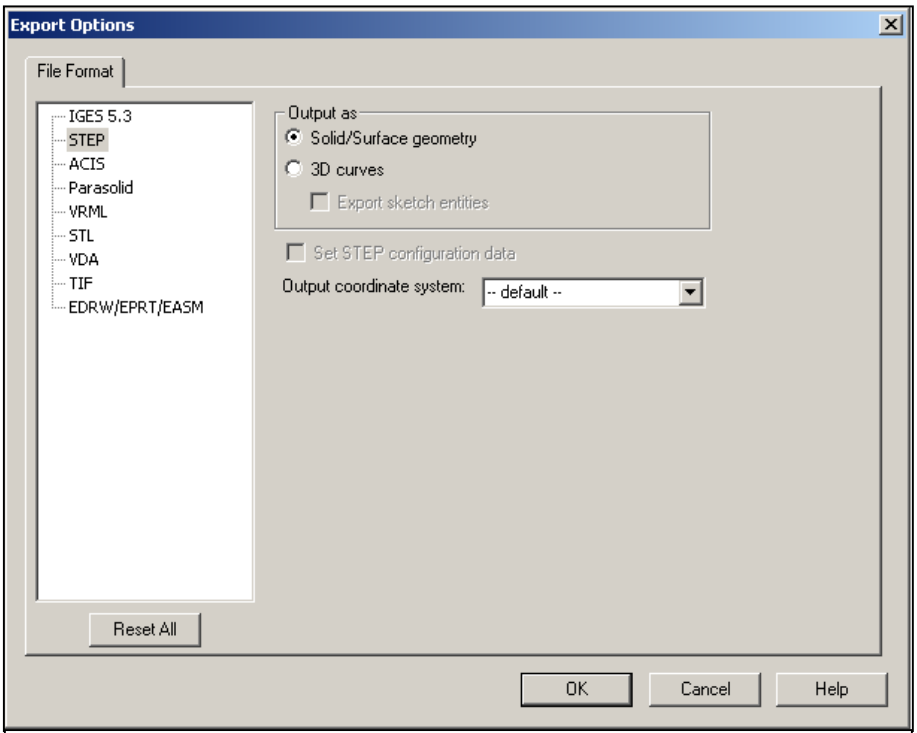

Figure 3:STEP Export Settings (screenshot from SolidWorks 2003)

6. CONCLUSIONS

The weightings have been set such that the compatibility between the three programs is of highest importance – a file format unable to interact between the programs is clearly useless to us. The File export size and import quality have been given the same weighting, as the format we choose has to have a balance of these two factors. Finally, the ease of use is of least importance as we can set-up a macro or a default setting to ensure that complicated settings are kept the same on every export/import should we need to. The total score for each file format is a sum of the score under each category multiplied by its weighting.

The reason that no further tests were run on the STL and VRML formats after the part files were exported and imported was due to the quality of the imported part file. It was instantly clear (see figure 1) that we could not use a file of this quality, as when mated with other parts in SolidWorks Assembly or when imported to Ansys they would give very inaccurate results.

From the table of weighted scoring we can see that the STEP file format wins over the IGES and VDA formats. Although the IGES format could be used across all three programs, the file size (especially for Assemblies) and the number of settings involved in getting a good quality export file type proved less useful than the hassle free STEP format. The STEP format is of the desired quality and is not only very compact but has the least number of set-up variables prior to export.

Table of File Types and Compatibility

SolidWorks	Pro/E WildFire	Ansys 8.0
SolidWorks (*.sldprt,*.sldasm,*.slddrw)		SolidWorks (*.sldprt,*.sldasm)
IGES(*.igs,*.iges)	IGES (*.igs)	IGES(*.igs,*.iges)
Parasolid (*x_t,*.x_b,*.xmt_txt,*xmt_bin)		Parasolid (*x_t,*.xmt_txt)
ACIS (*.sat)		ACIS (*.sat)
UGII (*.prt)		Unigraphics (*.prt)
STEP AP203/214 (*.step,*.stp)	STEP (*.stp)	
VRML (*.wrl)	VRML (*.wrl)	
STL (*.stl)	STL (*.stl)	
VDAFS (*.vda)	VDA (*.vda)	
Pro/E Part/Assembly (*.prt,*.prt*,*.xpr,*.asm,*.asm*,*.xas) ##	Pro/E Part (*.prt*,*.asm*) **	Pro/Engineer (*.prt*,*.asm*) ##,**
Catia Graphics (*.cgr) @	CATIA (*.ct,*.cat,*.model) @	Catia [V5] (*.catPart,*.catProduct) @
Inventor Part (*.ipt) ++	Inventor (*.iv) @	Inventor (*.ipt,*.iam) @
DWG (*.dwg) ++		Mechanical Desktop (*.dwg)
SolidEdge Part (*.par) ++		Solid Edge (*.par,*.asm,*.psm)
JPEG (*.jpg) X3D	JPEG (*.jpg) X3D	
TIFF (*.tif) X3D	TIFF (*.tif) X3D	
CADKEY (*.prt) ++	SET (*.set)	
DXF (*.dxf) ++	Neutral (*.neu)	
Reality Wave ZGL (*.zgl)	PATRAN (*.ntr)	
HCG (*.hcg)	Cosmos (*.ntr)	
HOOPS HSF (*.hsf)	3DPaint (*.obj)	
edrawing (*.eprt,*.edrw,*.easm)	Render (*.slp)	
	SuperTab (*.unv)	
	ECAD (*.emn,*.evs,*.mdb,*.edn,*.emp,*.edp)	
	Optegra Vis (*.gbf)	
	Medus (*.asc)	
	XPatch (*.facet)	
	Shaded Image (*.shd)	
	EPS (*eps)	

KEY

Compatible format for all programs
Compatible between SW & Ansys
Compatible between SW & ProE
Not available with this program
Incompatible across programs

NOTES

Pro/E 2001 or earlier ##
Pro/E WildFire **
Export Formats differ @
Import only no export ++
Not a 3d Model Format X3D

APPENDIX 2

	Compatibility	Export Size		Quality		Ease of use		Total Score
Weighting	3	2		2		1		
		Part	**Assembly**	**Part**	**Assembly**			
IGES	5	2 [166kb]	2 [50Mb/100Mb*]	5	5 / 1**	3		46
STEP	3	3 [45kb]	5 [3.6Mb]	5	5	5		50
VRML	3	4 [32kb]	-	1	-	-		N/A
STL	3	5 [18kb]	-	1	-	-		N/A
VDAFS	3	1 [413kb]	1 [176Mb]	5	1**	3		28

NOTES

Weightings are on a scale from 1 to 3, where 3 is most important

Scores are on a scale of 1 to 5, where 5 is best

* = Different IGES settings used

** = Large file caused SolidWorks to crash