LASER INTERFEROMETER GRAVITATIONAL WAVE OBSERVATORY

LIGO Laboratory / LIGO Scientific Collaboration

LIGO-T030143-03   *ADVANCED LIGO*   February 5th 2004

DESIGNING IN SOLIDWORKS

Customised Tools for Design and Documentation of

LIGO Parts, Assemblies and Drawings

Michael Perreur-Lloyd (IGR), Calum I. Torrie (CIT)

Distribution of this document:
LIGO Science Collaboration

This is an internal working note
of the LIGO Project.

California Institute of Technology
LIGO Project – MS 18-34
1200 E. California Blvd.
Pasadena, CA 91125
Phone (626) 395-2129
Fax (626) 304-9834
E-mail: info@ligo.caltech.edu

LIGO Hanford Observatory
P.O. Box 1970
Mail Stop S9-02
Richland WA 99352
Phone 509-372-8106
Fax 509-372-8137

Institute for Gravitational Research
University of Glasgow
Kelvin Building,
Glasgow G12 8QQ, Scotland
www.physics.gla.ac.uk/gwg

Massachusetts Institute of Technology
LIGO Project – NW17-161
175 Albany St
Cambridge, MA 02139
Phone (617) 253-4824
Fax (617) 253-7014
E-mail: info@ligo.mit.edu

LIGO Livingston Observatory
P.O. Box 940
Livingston, LA 70754
Phone 225-686-3100
Fax 225-686-7189

http://www.ligo.caltech.edu/

DESIGNING IN SOLIDWORKS

Customised Tools for Design and Documentation of LIGO Parts, Assemblies and Drawings

CONTENTS

DESIGNING IN SOLIDWORKS

Customised Tools for Design and Documentation of LIGO Parts, Assemblies and Drawings

1.   INTRODUCTION

This document serves to describe in detail the usage of a series of SolidWorks customised tools created by Mike Perreur-Lloyd and Calum Torrie. The document is written to be used in conjunction with a sister document[1], which gives an initial outline of the customised tools and instruction on the downloading and integration of these from a central location to the users desktop.

2.   BACKGROUND INFORMATION

The document details how information can be attached to SolidWorks parts, assemblies and drawings to aid the production of consistent engineering drawings, bill of materials, drawing trees and materials lists.

The Bill of Materials [BOM] and Drawing Tree are essential documents required by LIGO that follow all designed parts from manufacture through to assembly. The LIGO Mechanical Drawing Guidelines document[2] indicates that it is also useful, when sending a drawing pack for manufacture, if the major assemblies have an overall BOM (a, so called, materials list). In past LIGO projects it was necessary to create all of these as separate documents in programs such as Word or Excel, however by implementing the SolidWorks 'Bill of Materials' function it is possible to produce the documents automatically.

To help the automation of these, a LIGO Bill of Materials has been designed that is tailored to ensure that the information detailed in the Mechanical drawing guidelines is correctly tabulated. Alongside this, 'Smart' CAD templates, 'Smart' Data templates, and the Customised Toolbox have been developed to automate the creation of parts, assemblies and engineering drawings.

By establishing an array of customised tools it becomes easier to standardise the information shown in every part, assembly, and engineering drawing, thus improving the quality of the documentation. And by automating the production of the supplementary documents through entering information during the design stage, we make time for other areas of the project.

Following the description and explanation of the LIGO Customised Tools, a step-by-step tutorial provides an example of the procedures used to produce a part, an assembly and a mechanical drawing with a the BOM, Materials List or Indented Drawing tree. The tutorial is concluded by an example of how a file can be uploaded to the Caltech PDMWorks vault.

---

[1] D030382 "An Introduction to the LIGO PDMWorks Vault and a Top-Level Summary of the Drawing & Data templates, Macros, Bill of Materials and Customised Toolbox created for SolidWorks".
[2] E030350 "Mechanical Drawing Guidelines; D Coyne; available on the LIGO DCC".

3.   PART PROPERTIES FOR THE BOM

ITEM NO	REQ.	SPARE	TOT.	PART NUMBER	DESCRIPTION		MATERIAL
					The above table shows the LIGO Bill of Materials column headers	**FIGURE 1**	

The functions used in creating an accurate Bill of Materials are outlined below. Further instruction on how to use these is contained in the appended Tutorial (Appendix 1).

3.1.   Item Number Column

3.1.1. Default Properties

SolidWorks has a number of default settings to each of the columns in the BOM. In the case of the Item number it will automatically choose an item number depending on the order that parts are entered into an assembly. These are the same numbers shown when balloon numbers are added to a drawing. In some situations, however, we need to call up parts without showing them in an assembly drawing therefore the we have designed the LIGO BOM with several new functions that will over-write the default information with a more customised form.

3.1.2. Hidden Parts and Assemblies

In an assembly it is often necessary to have additional parts or sub-systems that are not eventually used in the final system, for example (in terms of a suspension prototype): a wire jig; an extra mass; an angled blade clamp. In the bill of materials for an assembly it is necessary to differentiate between these components depending upon their usage in the assembly. In this situation these should be referred to as one of the following:

ALT: 'Alternative', e.g. an angled blade clamp. This component would only be used in a suspension assembly should a blade be deflecting above or beyond the optimum height.

REF: 'Reference', e.g. a wire jig. This sub-assembly was used in the in the construction of the suspension assembly, however is not part of the assembled suspension and hence need not be displayed on the assembly drawing document.

A/R: 'AsRequired', e.g. an extra mass. This part is only required should the optic of a suspension assembly, once fully suspended, be hanging too high. In this situation an 'extra mass' or a number of 'extra masses' are added, as required.

All of the above component types are entered into the custom properties of a drawing. SolidWorks custom properties are explained in section 4, 'Setting Custom Properties in SolidWorks' and further relayed in the appended tutorial (see Appendix 1).

3.2.   Required Column

SolidWorks automatically calculates the quantity of each part from the number of parts entered an assembly. However, when we need to specify that more than one assembly is to be built we need to be able to multiply the automatic defaults by however many assemblies we are having manufactured. Again this can be done in the Custom Properties:

REQ: 'Required', the total number of parts required to build an assembly or a number of assemblies (not including spares).

3.3.   Spare Column

Again SolidWorks has a default setting, that is: the number of Spares is equal to the number of parts (i.e. 10 parts made, 10 spares made). As it is likely that this will not be the requirement for all parts it is possible to override this setting using the function:

SPARE: If 'Spare' is entered in the custom properties along with the desired value, this will override the default.

3.4.   Total Column

The 'Total' column is far more self-explanatory and is purely a sum of the 'Required' and 'Spare', detailed above. No custom properties are set for 'Total'.

3.5.   Material Column

Material is a column that has no default value as it has been created specifically for use in the LIGO BOM. It is important to enter a material custom property for all parts as the BOM will be the sole place that this information will be contained (individual engineering drawing of parts do not show material). The number of character spaces in the Material column is limited so below are some examples of typical material acronyms.

300 SSTL: 300 Series Stainless Steel
6061-T6-Al: 6061-T6 Aluminium (US specification)
6082-T6-Al: 6082-T6 Aluminium (UK specification)
PEEK [NAT]:   PEEK natural or virgin grade

3.6.   Part Number Column

The LIGO BOM is set up such that the seven-digit part number associated with the LIGO Document Control Centre (DCC), e.g. D020123, will automatically be picked up from the 'Summary Properties' beside the heading 'title':
e.g.   D020123:   Enter this number next to Title on the 'Summary Properties of the part.

3.7.   Description Column

Again the LIGO BOM has been set-up to automatically pick up part description from the Custom Properties:
e.g.   Angled Blade Clamp: Enter 'description' into the custom properties followed by the part name for this to link in to the BOM automatically**.**

4. CREATING A BILL OF MATERIALS, DRAWING TREE OR MATERIALS LIST

To create a Bill of Materials (BOM), Drawing Tree or Materials List by using the LIGOBOM[3], it is simply a matter of selecting the appropriate settings in the SolidWorks Bill of Materials function. All of these functions are available when a drawing (.slddrw) of an assembly is open. For example:
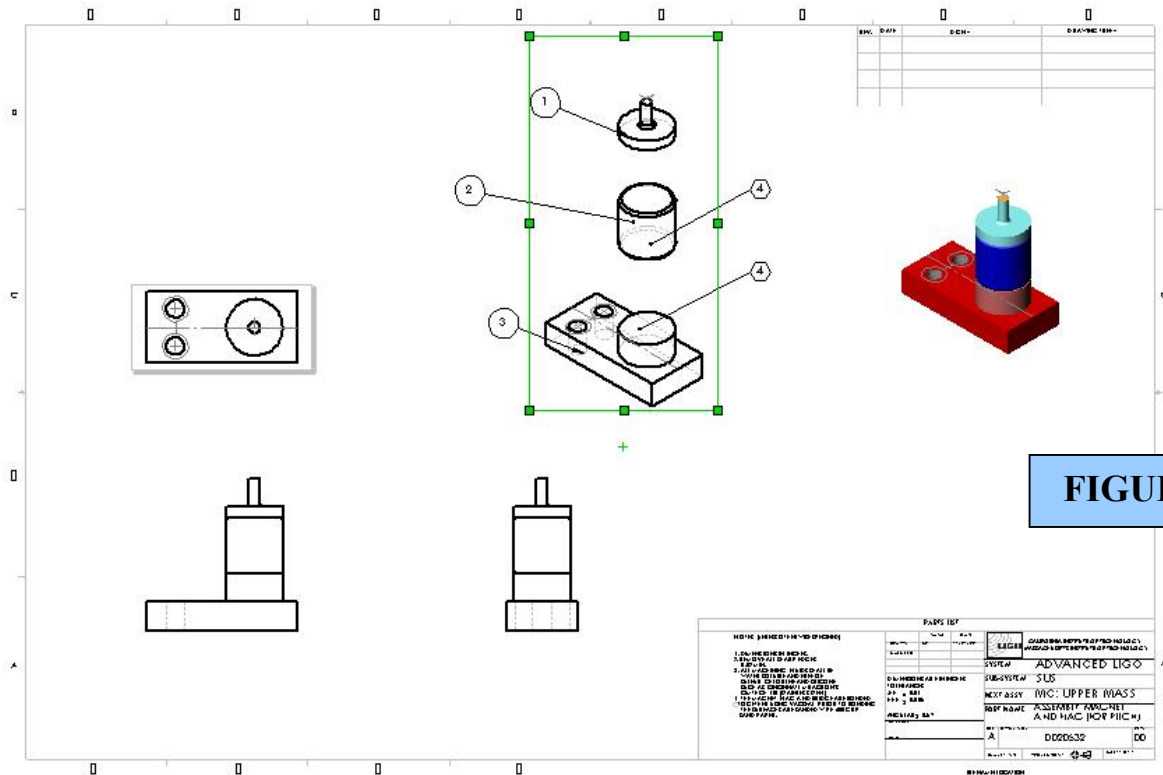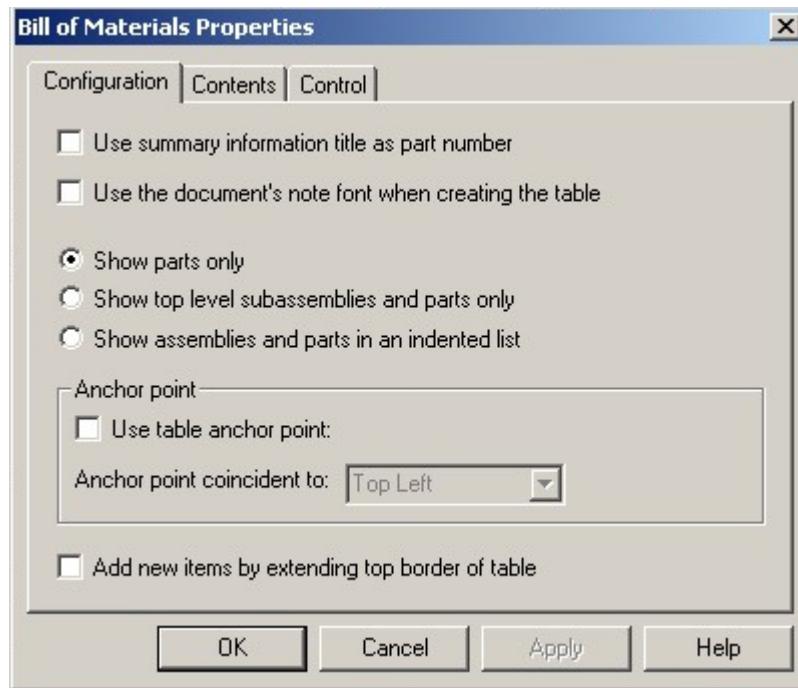

FIGURE 2

By selecting any of the assembly views on the drawing sheet and then, from the Main Heading Toolbar, Insert>Bill of Materials (You will notice that this is only highlighted when your assembly view is highlighted) you will open up the. Bill of Materials function

This will then prompt you to open up a BOM.xls file from a list of files in SolidWorks.

---

[3] D030384.xls, refer to D030382 for information on locating and downloading this file.

Once the LIGOBOM is located, the following window shall open up on your screen:


FIGURE 3

And it is by using this interface that the selections can be made to tabulate a BOM, Drawing Tree or Materials List.

4.1.   General Notes

- It is advisable for all tables that the first two boxes are left unchecked, i.e. Do not 'Use Summary Information title as part number' and Do not 'Use the document's note font when creating the table'. The LIGO BOM is set-up such that the font size in the excel spreadsheet will create a table with a width equal to the note box on the bottom right corner of a drawing sheet.
- The 'use table anchor point box' and the 'Add new items by extending top border of table' should also be left unchecked

4.2.   Creating a Bill of Materials

To create a bill of materials, the box that should be checked on the 'Configuration' area of the 'Bill of Materials Properties' is:
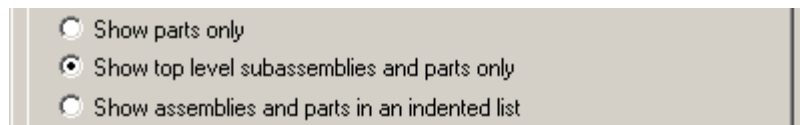
And on the 'Contents' area of the 'Bill of Materials Properties' uncheck 'Display labels at top'

4.3.   Creating a Indented Drawing Tree

To create an Indented Drawing Tree, the important boxes that should be checked are:

○ Show parts only
○ Show top level subassemblies and parts only
◉ Show assemblies and parts in an indented list

4.4.   Creating a Materials List

Creating a materials list is now very similar to creating a LIGO bill of material (Section 3.1) with exception 'display labels at top' should be checked. Where a Bill of Materials is used on every assembly and sub-assembly, a Materials List is used only on Major assemblies and generally created prior to a drawing package being sent out to tender.

To create an Indented Drawing Tree, the following box should be checked:

○ Show parts only
◉ Show top level subassemblies and parts only
○ Show assemblies and parts in an indented list

 And, to reiterate, 'Display labels at top' under the 'Contents' Tab should also be checked.

4.5.   Completing the Engineering Drawing

Once the correct boxes are checked and OK pressed, a BOM table will appear on the Drawing sheet. It is likely, however, that none or very little information is shown at first. The reason for this is that the added functions - the equations from Excel - have not yet been solved. You can update these functions by double clicking the table (which triggers Excel to open and calculations to be done) and then by clicking back on the drawing sheet again, this will close.

The Table can then be moved into position, above the Drawing sheets PART LIST title box, by simply dragging the table to this location (as shown in Figure 4)

3	1	1	2	D020210	BRIDGE SPACER FOR MAGNET	6061-T6-Al
2	1	1	2	D020466	NICKEL PLATED MAGNET	Nd:Fe:Bo
1	1	1	2	D020197	FLAG (FOR MAGNET D020466)	6061-T6-Al
ITEM NO	REQ.	SPARE	TOT.	PART NUMBER	DESCRIPTION	MATERIAL

PARTS LIST

NOTES: (UNLESS OTHERWISE SPECIFIED)

1. DIMENSIONS IN INCHES.
2. REMOVE ALL SHARP EDGES, R.02 MIN.
3. ALL MACHINING FLUIDS SHALL BE WATER SOLUBLE AND FREE OF SULFUR, CHLORINE AND SILICONE, SUCH AS CINCINNATI MILACRON'S CIMTECH 410 (STAINLESS STEEL).
4. THE MAGNET, FLAG AND BRIDGE ARE BONDED TOGETHER USING VACSEAL, PRIOR TO BONDING THE SURFACES ARE SANDED WITH 600 GRIT SAND PAPER.

CALIFORNIA INSTITUTE OF TECHNOLOGY
MASSACHUSETTS INSTITUTE OF TECHNOLOGY

SYSTEM   ADVANCED LIGO
SUB-SYSTEM   SUS
NEXT ASSY   MC: UPPER MASS
PART NAME   ASSEMBLY MAGNET AND FLAG (FOR PITCH)

DIMENSIONS ARE IN INCHES
TOLERANCES:
.XX   ± 0.01
.XXX ± 0.005
ANGULAR ± 0.5 °

SIZE DWG. NO.   REV.
A   D020532   00

FIGURE 4

All information in the Bill of Materials, Drawing Tree or Materials List can be either kept in the SolidWorks drawing sheets or copied to an Excel Spreadsheet for inclusion in another document. It is important to note that, to keep the information up-to-date, it is advisable that the tables remain in SolidWorks until a point where all drawings are finalised i.e. just prior to Manufacture.

5.   SETTING CUSTOM PROPERTIES IN SOLIDWORKS

SolidWorks custom properties are very useful as they encourage us to embed important information into the part, assembly and drawing files and more importantly this information can be then be linked to other SolidWorks tools such as the Bill of Materials function and the Drawing Sheet.

The Custom and Summary Properties window can be opened by going File>Properties.
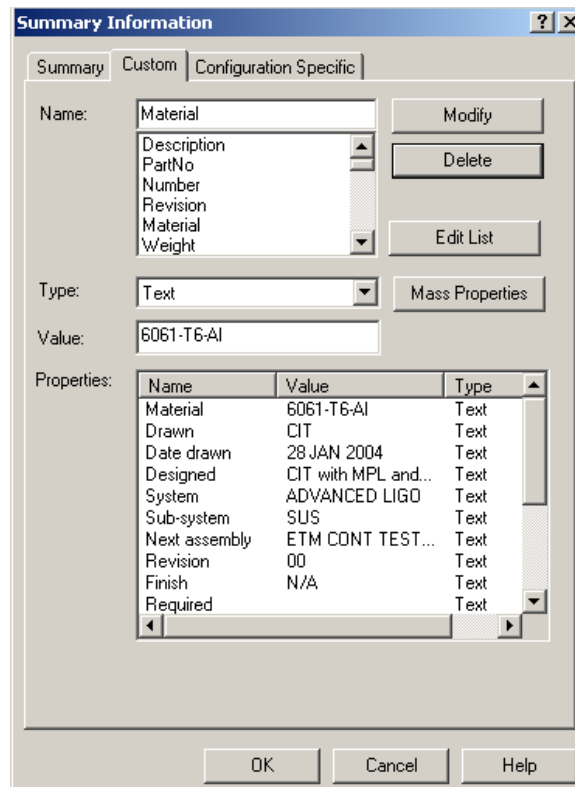
FIGURE 5

5.1.   Summary Properties

The first window in File>Properties will show the Summary Properties. This area is like any Microsoft file where you can store useful information about the file and when it was created, etc. Although it is not a necessity to store information here it will is a good exercise to practice. By clicking the top 'Custom' Tab we can now begin editing the custom properties.

5.2.   Custom Properties

This is the key area in which you can enter the important information that links to the Bill of Materials and Drawing sheet. As shown in Figure 5 (Right) above the information described earlier about Required, Spare, Material, etc can be entered here.

In the case of Alternative, Reference and As Required parts these should be entered as follows (see figure 6):

e.g.   Name: Alternative
       Type: Text
       Value: ALT

       Name: Reference
       Type: Text
       Value: REF

       Name: As Required
       Type: Text
       Value: A/R
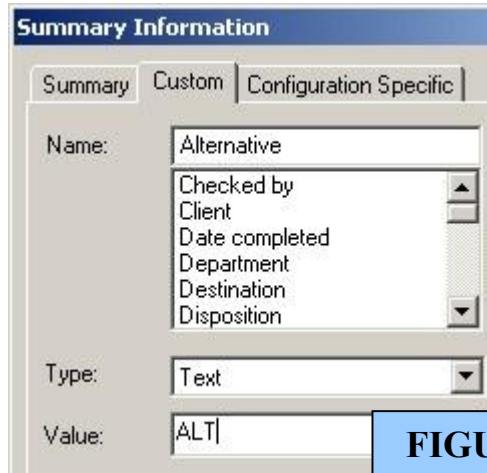
FIGURE 6

5.3.   Additional Custom Properties

A number of additional properties that link to the latest Drawing Template can also be added in the Custom Properties further automating our process of design to manufacture of a design in SolidWorks (see figure 7 below). With the development of Smart Data Templates, it is not necessary to add all of these properties (See section 6. Smart Data and CAD Templates).

FIGURE 7

6.   'SMART' DATA AND CAD TEMPLATES.

A series of 'Smart' CAD and Data Templates[4] have been created that will also help automate the design of parts and assemblies. These smart templates make the job of adding custom properties and making full use of them becomes even easier.

6.1.   'Smart' Data Templates

A data template is the background file that is opened when a 'New' file is created in SolidWorks.
i.e When you go to File>New or click ⬜ the icon. The following screen will then appear:
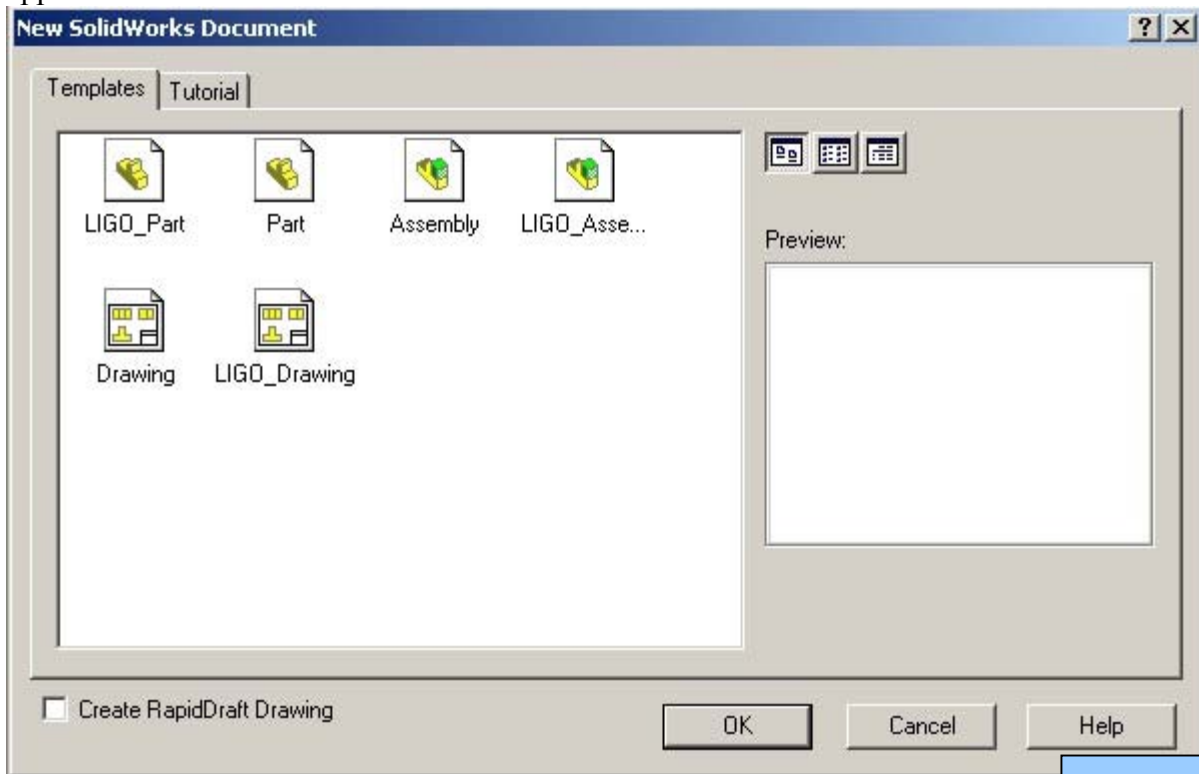
FIGURE 8

      These are the Data Templates

Notice above that the new 'Smart' Templates, named LIGO_Part, LIGO_Assembly and LIGO_Drawing, are already downloaded here and can be edited as new files.

What makes them 'Smart'?

A 'smart' template is a template that has LIGO-personalised custom properties already embedded and, from the moment you start a SolidWorks drawing, it will prompt you for the information used in the LIGO BOM and LIGO CAD templates.

---

[4] See document D030382 for information as to locating the latest version of these files

If you open one of the above 'Smart' Templates as a new file and go to the Custom Properties. You will notice that the following information is already programmed into the file:

FIGURE 9

All of this information is linked to the BOM and Drawing Templates. And if filled in completely will save much time in creating the engineering drawings for the parts and assemblies.

6.2.   'Smart' CAD Templates

Smart CAD templates are LIGO-personalised borders for engineering drawings intrinsically linked to the above properties. And by using these no text editing need be done to an engineering drawing once the part or assembly has been inserted. Programmed into the LIGO drawing sheets are links to the Assembly, Drawing Name, Material, Finish, Drawn by, etc (detailed in section 5. Setting Custom Properties) now entered in to the Customised Properties. This means that all drawing information is no longer edited by using 'edit sheet format' function on a sheet but by adding information to the Custom Properties of a file.

7.   CUSTOMISED MACRO FOR SYNCHRONISING PART AND DRAWING PROPERTIES

A more recent development of the customised tools has been the addition of a macro which copies all properties entered into the SolidWorks Part or assembly across to its associated drawing file.

There are two great benefits of being able to do this:
   7.1.   It saves time having to type up the same data into both part (or assembly) and the associated drawing files custom properties.
   7.2.   The properties act as Windows Metadata, which is then searchable through Windows Explorer (and, in future, Internet Explorer).

A consequence of this is that all of these properties are transferred across when we create a pdf of the drawing, therefore enhancing the quality of the documentation submitted to the DCC

An example scenario of how this will help us could be as follows:

Sometime in the future a member of the installation team needs to know the specifications of a cantilever blade but they do not know what the part number of the blade is. They do know that however that it is a part from the Mode Cleaner Upper Mass and that Mike Plissi probably designed the blade. Therefore, in this example, without trawling through numerous pdf drawings, we could search the list for all files with 'MVP' or 'MC Upper Mass' in the properties.

The macro, called "GOE PropSynch.swp", is contained as an attachment to this documents sister paper[5] and will be available on the PDMWorks vault. An example of this macro being used is in the appended tutorial.

---

[5] D030382

8.   SETTING OTHER PART PROPERTIES

When designing suspensions it is important that we can predict accurate mass and moments of inertia of each of the masses so that the actual manufactured suspension can fit the necessary noise requirements.

For SolidWorks to calculate an accurate mass we need to input the density into each part. The area in which this information can be entered can be accessed a number of ways: a) by right clicking the part name in the FeatureManager Design Tree and selecting Properties, b) by clicking the GRID icon on the 'SKETCH' toolbar (see Figure 8), or c) Tools>Options>Document Properties

GRID icon
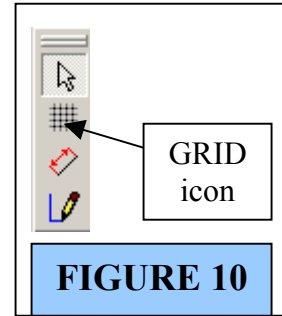
FIGURE 10

This will then bring up the Document Properties window under which the following values of density can be entered under the 'Material Properties' heading (see figure 11).

8.1.   Examples of Material Densities

Alu	0.0027 grams/mm^3
Stainless Steel	0.0078 grams/mm^3
Titanium	0.0045 grams/mm^3
Fused Silica	0.002202 grams/mm^3
PEEK	0.00132 grams/mm^3
Glass (BK7)	0.0025 grams/mm^3
Alu	0.09754 lb/in^3
Stainless Steel	0.28179 lb/in^3
Titanium	0.16257 lb/in^3
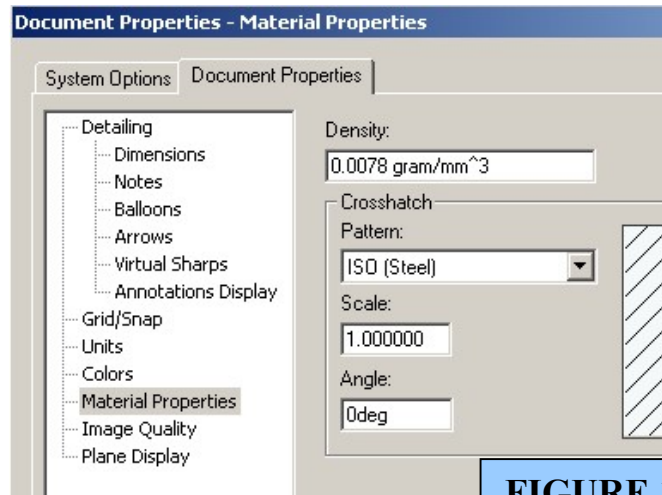
FIGURE 11

9.   CUSTOMISED TOOLBOX.

The customised toolbox[6] was created following problems found with the SolidWorks Toolbox when passing assemblies between various engineer's computer terminals. Following its creation, the customised toolbox has since been expanded to include a full range of nuts, bolts and washers.

Once located, the toolbox should be downloaded in its entirety (preferably, or as required) to a local drive for usage in all Advanced LIGO assemblies. For more information on downloading this Toolbox, refer to document D030382

9.1.   Background to Development of the Toolbox

The reason for undertaking the exercise of creating this new toolbox as opposed to continuing to work with the SolidWorks Toolbox is because assemblies that use the SW version always refer to the location of the toolbox on the workstation that is being used. This works well if assemblies and designs are never moved from that one machine, however with the suspension team being based between GEO in the UK and LIGO in the US, it has been found to be problematic when transferring and sharing files.

When SolidWorks cannot find the existing location of it's toolbox on, say, the C:/ drive of a workstation, it will automatically prompt the user to locate the whereabouts of this file. Upon opening the file it will then revert back to what it calls 'the last saved configuration' of the bolt (for example) and that, despite the fact you specify a #4-40 bolt, could be a #½-20 bolt!

9.2.   Using the Customised Toolbox

The customised toolbox consists of a full set of Imperial and Metric nuts, bolts, washers and associated tools (wrenches, interlocking masses, etc) in both Stainless Steel and Silver-plated Stainless Steel currently used by suspensions team.

The customised toolbox has been designed so that when a bolt is added to an assembly it is added as an actual part. By so doing, once that part has been introduced to the assembly as a certain size it will not change, unless the user physically changes it. To introduce a new Stainless steel bolt, for example, into an assembly the SolidWorks user would 'Open' the file SST_SocketHeadCapScrew.sldprt, select the appropriate configuration, then drag and drop it in to the assembly. Once in the assembly the user then 'Mates' the bolt to the interfacing parts.

---

[6] D030383, Refer to document D030382 for information as to the location of the latest version of this.

The customised toolbox has the luxury that, once a bolt is inserted in to an assembly, it is very easy to change its configuration. For example, if a hole size increases or decreases, a change is simply a right click away. Figure 12 shows how a bolt can be changed:

FIGURE 12
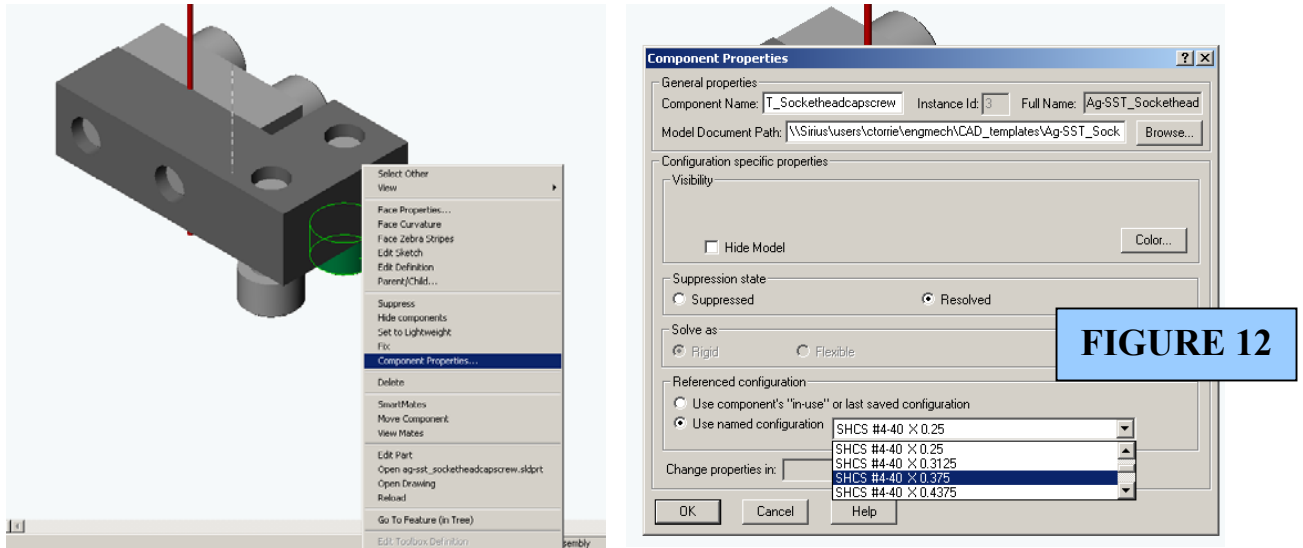
10.   **SETTING UP A WORKSTATION WITH LIGO CUSTOMISED TOOLS**

All Customised Tools in this document can be found on the DCC however the latest versions of all tools are available on the Caltech PDMWorks vault. Please refer to the top-level document D030382 that describes where all customised tools are located and instructs as to how and where these should be downloaded.

11.   THE PDMWORKS VAULT

It is the intention of the suspensions team that, from the publication of this document onwards, all part, assembly and engineering drawing documents in SolidWorks, pdf and universal CAD format (SAT) are all stored on the CalTech PDMWorks vault. This replaces Sirius as the storage location for future controls and noise prototypes, and finalised suspension designs. It has become apparent through testing at CalTech, MIT and Glasgow that this Data Management tool provides the solution to the problem of designing suspension assemblies and their associated surrounding architecture, safely and securely across a number of locations and countries.

The following sections highlight some points on PDMWorks usage and general convention for naming design files specific to the Advanced LIGO project. For full instruction as to how to use PDMWorks please refer to the help files.

11.1.   Rules for Creation and Naming of Files for the Vault

10.1.1. Part files should be named like this:
D040xyz_`Suspension'_`Sub-System'_`Description'.sldprt;
e.g.   D040036_ETM_TEST_MASS.SLDPRT.
10.1.2. Assembly files should be named like this:
D040abc_`Suspension'_Assembly_`Sub-System'.sldasm;
e.g.   D040038_ETM_Assembly_Test_Mass.SLDASM.
10.1.3. Drawings should be named as per their associated part or assembly (sldprt):
e.g.   D040038_ETM_Assembly_Test_Mass.SLDDRW;
or   D040036_ETM_TEST_MASS.SLDDRW.
10.1.4. Drawings should, once completed, be converted into .pdf format with and embedded .SAT file. This can either be done using BlueBeam[7] or using Adobe Acrobat (full version), which allows SATs to be embedded in pdf files.
10.1.5. For assemblies it is also encouraged that an isometric rendering in jpeg format is also checked into the vault. These will be very useful when preparing presentation material or writing report and papers.

11.2.   Management of the Vault

The management of the vault at the Administrator level will be undertaken by Mike Pedraza of Caltech. The vault will be backed up every day so that should something go wrong we can check back to the previous days file structure.

Other than this PDMWorks vault manages itself as it automatically identifies old or new files being uploaded, or indeed the same unchanged files being uploaded and issues revision numbers when necessary. Files can be 'owned' by any member of the vault – where this is the case, specific files *can* be downloaded but only as read only. If however the file is resaved locally by that new owner and uploaded, it will be checked in with a new revision number.

---

[7] Bluebeam PDF - www.bluebeam.com have created an icon based pdf creation tool which simplified the process of creating pdfs and embedded SAT files. Now also available from SolidWorks (www.SolidWorks.com)

11.3.   Checking files in to the Vault

11.3.1. Files should be checked in with the appropriate naming convention (see 10.1.)

11.3.2. The correct 'Project' should be selected for the file being checked in. The 'Project' is the directory or folder under which the file will be listed.

11.3.3. Non-CAD files such as pdf's, jpeg's, word documents, and excel spreadsheets can be referenced to other files on the vault should their information be linked somehow.

11.4.   Other features of the Vault

11.4.1. SolidWorks files and their revisions. In the custom properties of the Smart Data templates there is a reference to a file's 'Revision'. All Smart CAD templates are set up to automatically use the PDMWorks revision information, however should you wish to over-ride this function – your specified revision number can be manual entered into the custom properties of the file.

11.4.2. Revisions occur every time an item is are checked out, edited and checked back in to the PDMWorks vault.

11.4.3. To re-iterate, all of the customised tools described in this document will be available in the Caltech PDMWorks vault. Please refer to the document D030382 for advisory notes on locating and downloading the files.

12.   CONCLUSIONS

This document overviews how to enter the particular information that is required to use the LIGO BOM to its full potential and how to enter all the information that the Suspension team has found useful to incorporate so far. It is possible to further tailor the BOM or Custom Properties to suit many more needs, but a balance must be made on how time consuming this over and above the future benefits of doing this.

The improvements in the document are comprehensive but hopefully, once set-up, are straightforward and well integrated. It is important to note that they have currently been set-up to match the requirements of the suspensions team. Other design teams within the LIGO collaboration may find that these improvements need to be expanded or further customised to match their exact needs. The authors would be interested to hear of any new suggestions or further customisation of the above tools.

Authors:
Michael Perreur-Lloyd:
Institute for Gravitational Research, University of Glasgow, mplloyd@physics.gla.ac.uk
Calum Torrie:
LIGO Project, California Institute for Technology, ctorrie@ligo.caltech.edu

DESIGNING IN SOLIDWORKS

Customised Tools for Design and Documentation of LIGO Parts, Assemblies and Drawings

APPENDIX 1

Tutorial Example

This tutorial demonstrates the functions used in creating a typical test mass assembly with the required information to produce an automatic Bill of Materials. This tutorial is designed to integrate with D030384_08_LIGOBOM.xls and the LIGO Customised Tools described earlier in this document. Please ensure that all necessary updates have been completed as this tutorial is designed to make full use of the new functionality.

NOTE: Some of these tutorials screenshots were taken prior to the most recent document revision please refer to the main document (which has been fully updated) should you notice anything unusual. (e.g. the custom properties shown in the tutorial - pages 6 to 8 - have small letters)

To begin, open a new LIGO_Part:
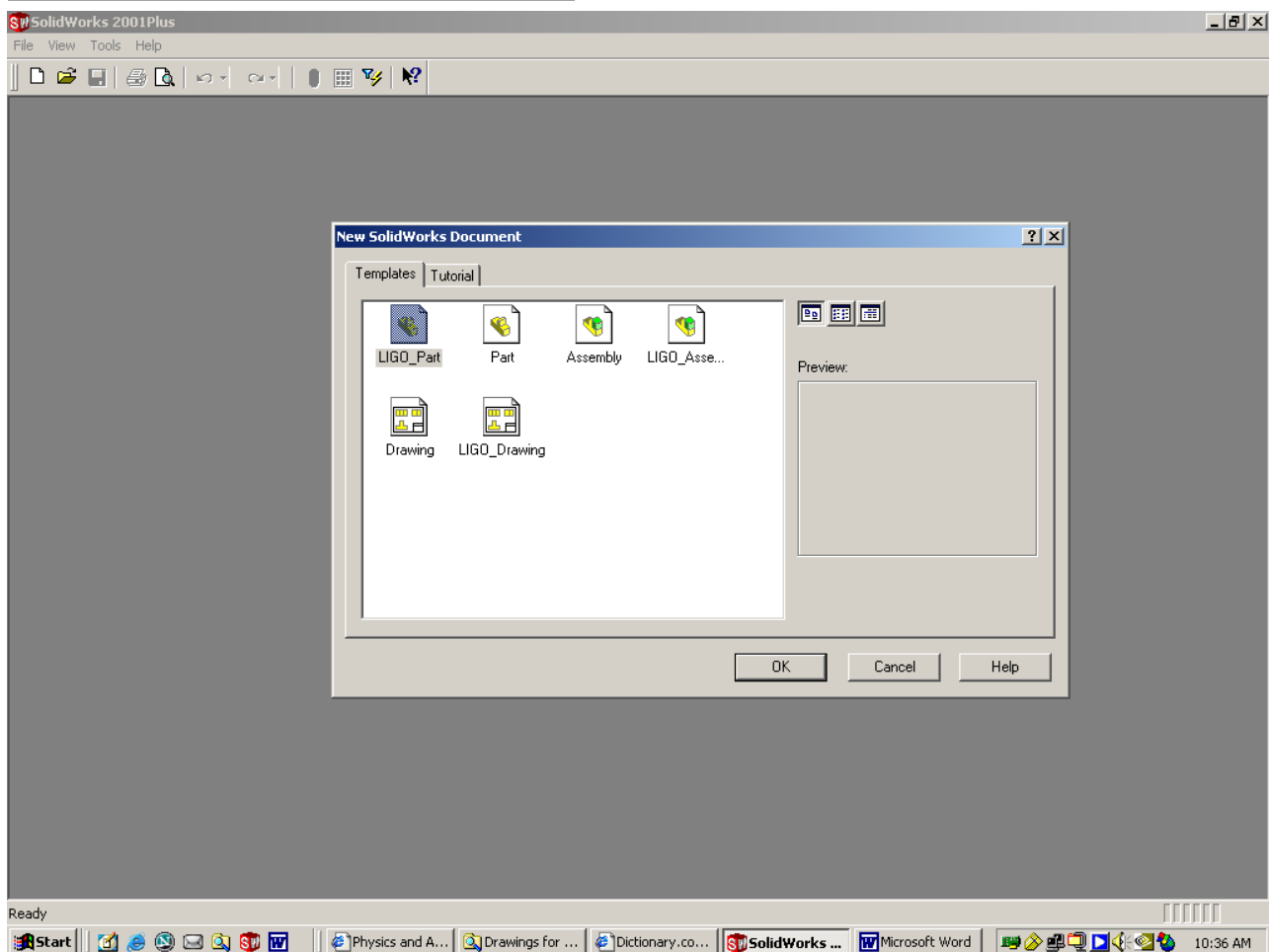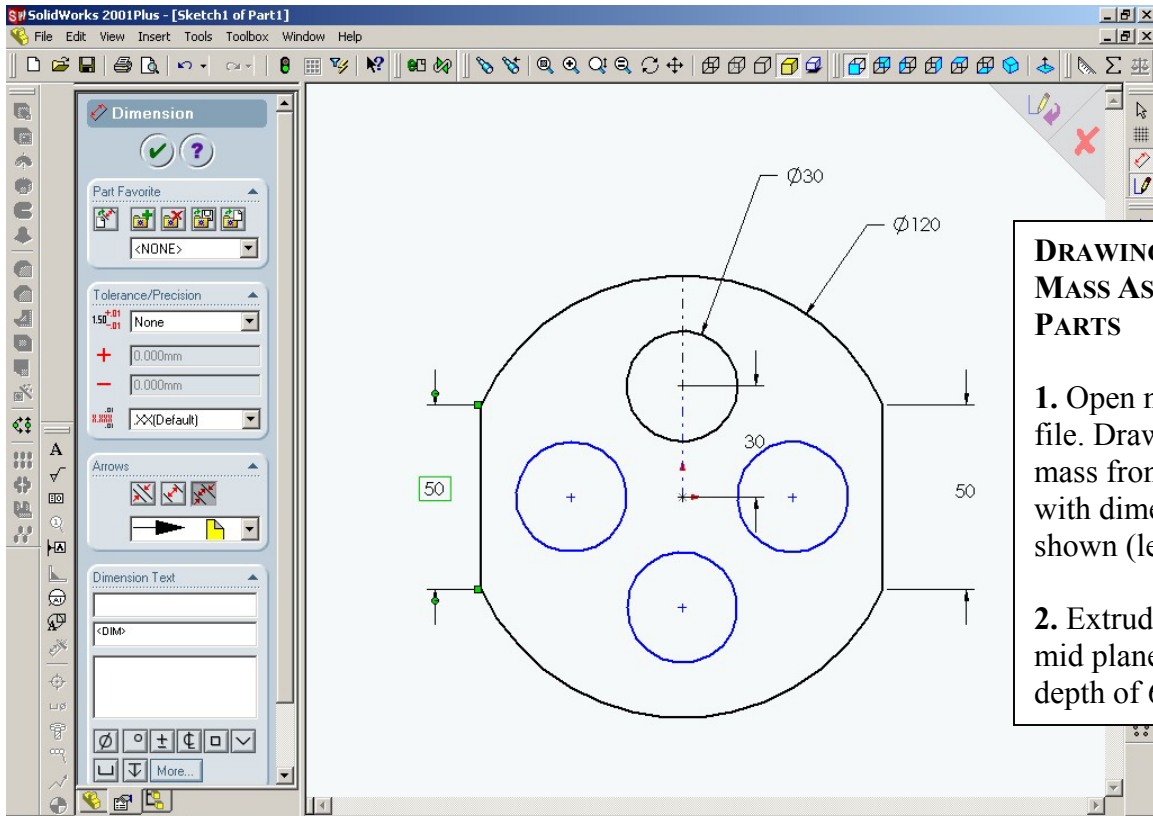
DRAWING TEST MASS ASSEMBLY PARTS

1. Open new part file. Draw up test mass front view with dimensions as shown (left)

2. Extrude-Boss, mid plane to a depth of 60mm
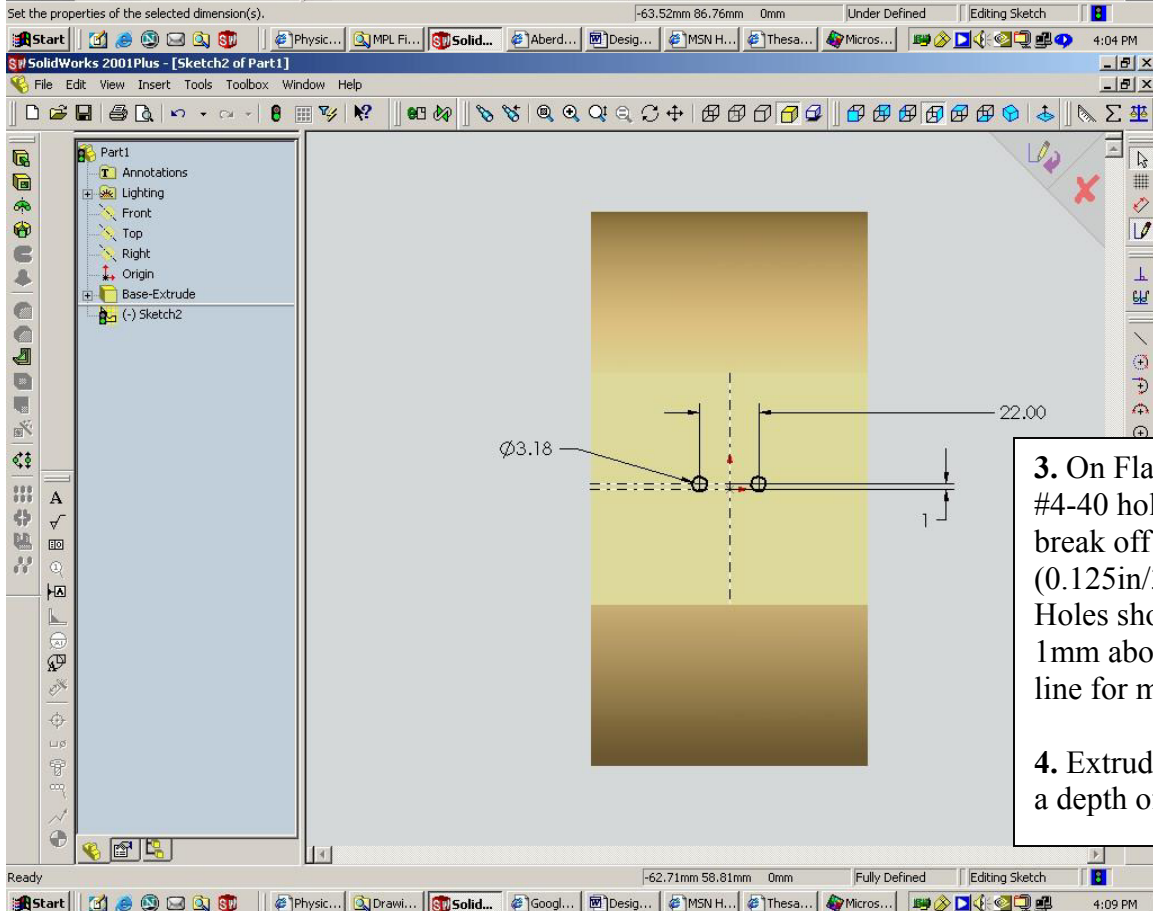
3. On Flat, Draw #4-40 holes for break off ears (0.125in/3.18mm). Holes should be 1mm above centre line for mass.

4. Extrude-Cut, to a depth of 8mm.
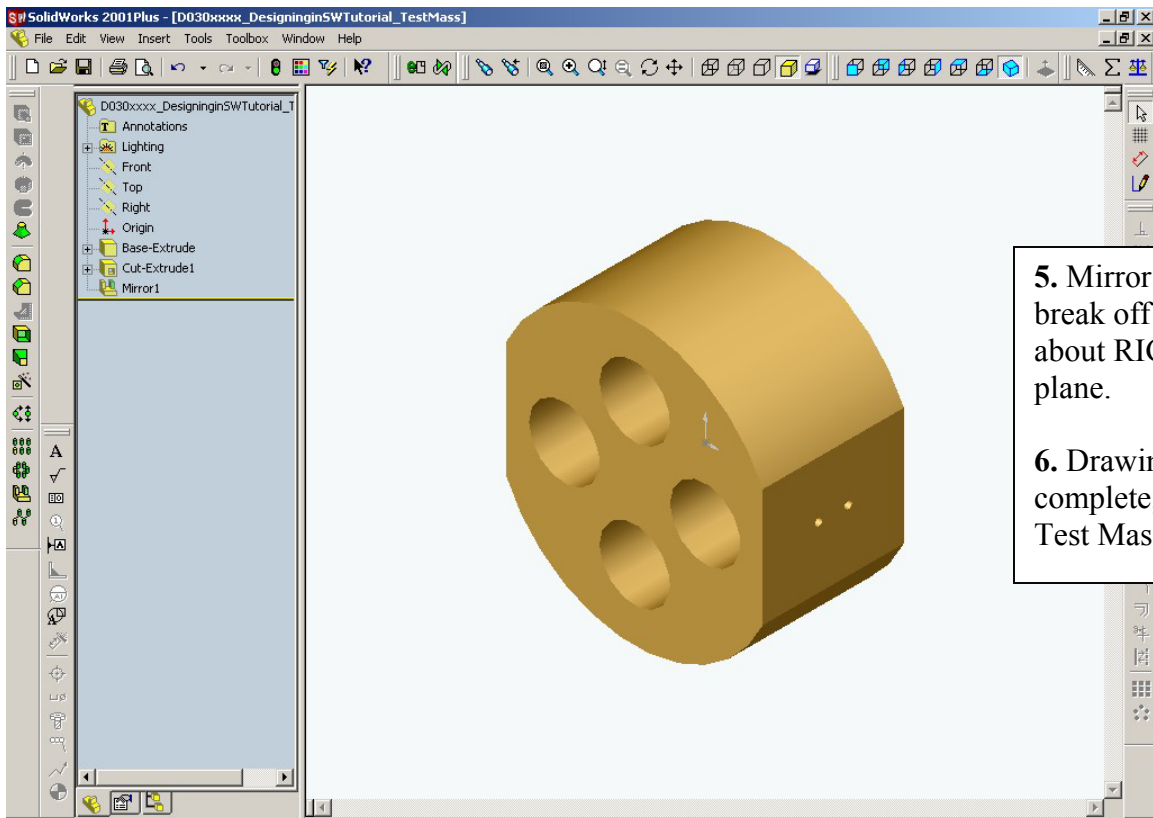
5. Mirror holes to break off ears about RIGHT plane.

6. Drawing complete, save as Test Mass.sldprt
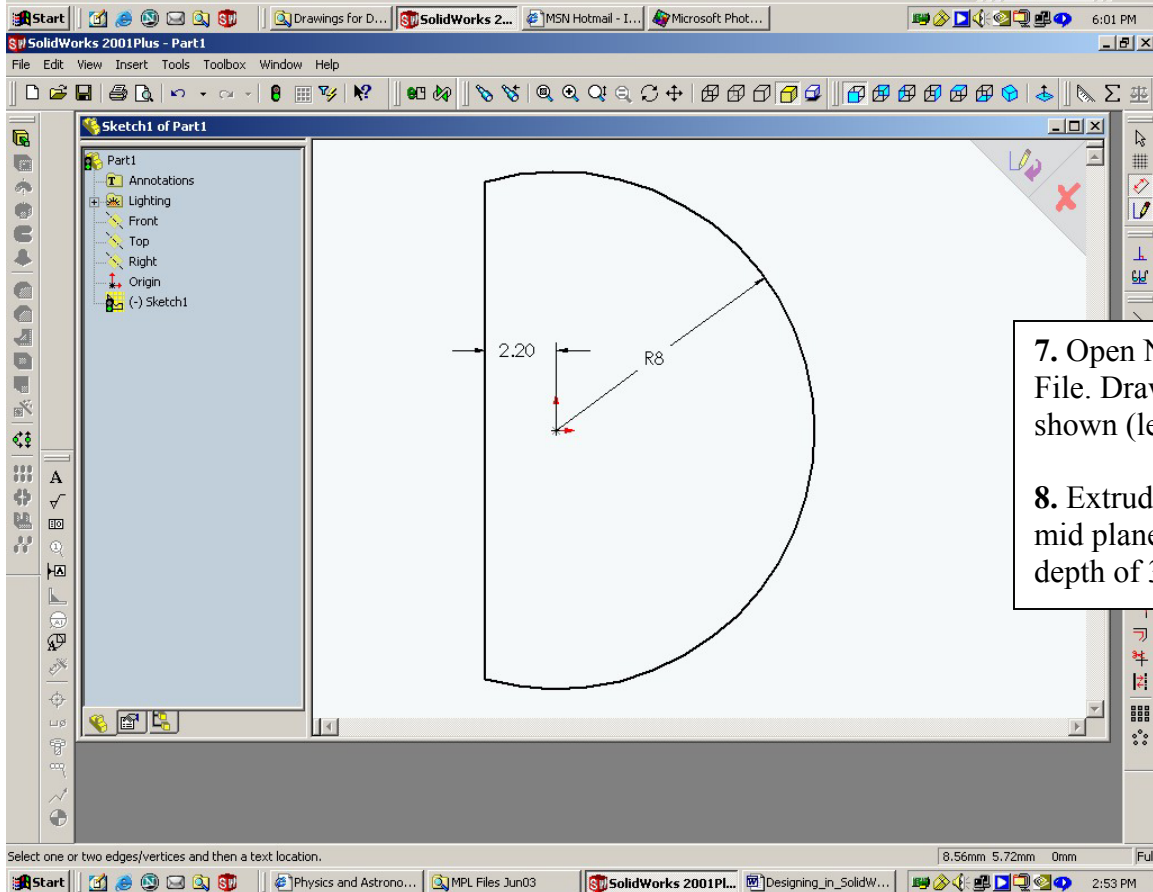
7. Open New Part File. Draw profile shown (left).

8. Extrude-Boss, mid plane to a depth of 30mm.
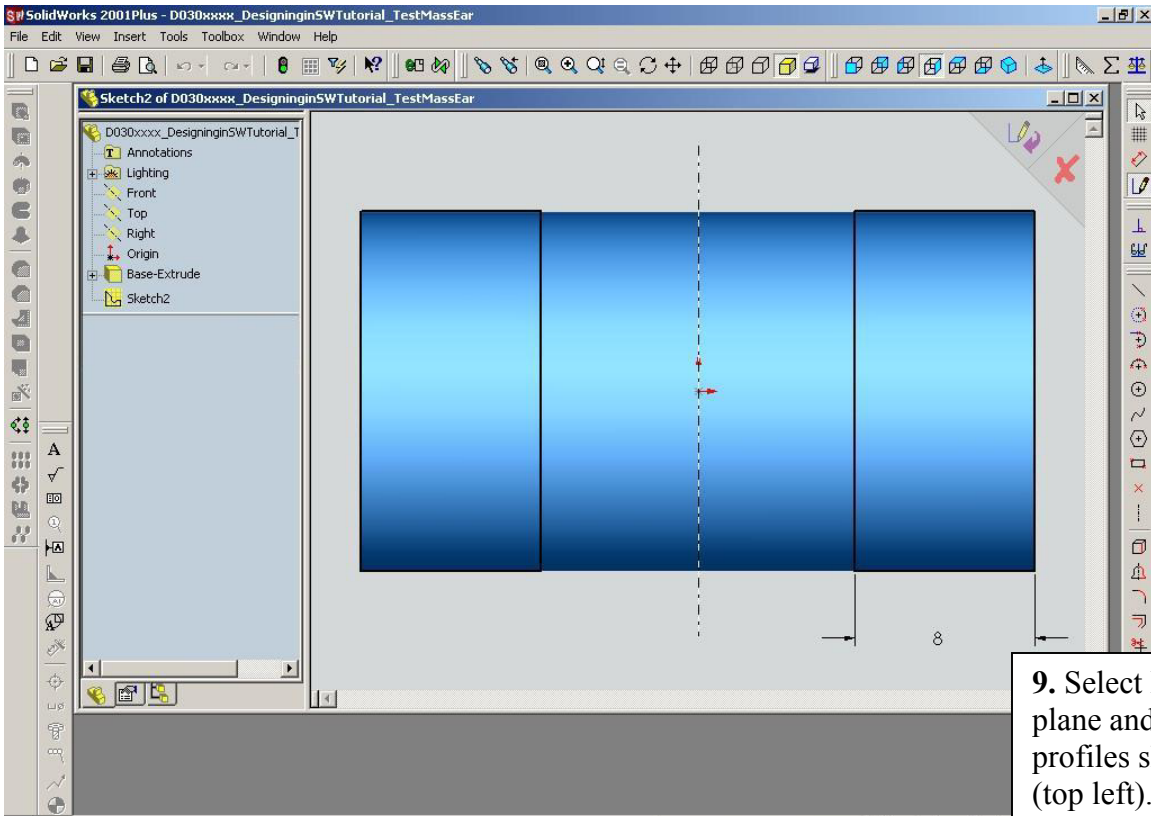
9. Select RIGHT plane and draw the profiles shown (top left).
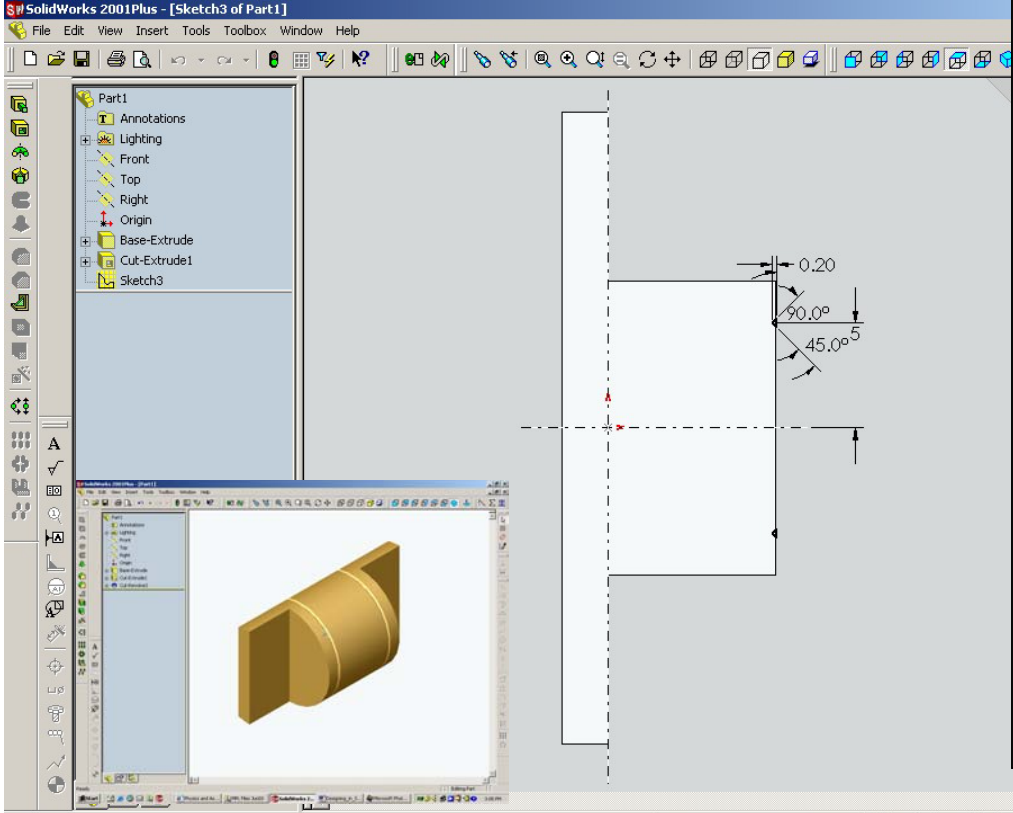
10. Extrude-Cut, to a depth of 8mm to get profile shown in the following screen capture (inset bottom left).

11. Select TOP plane. Draw V-shaped wire groove making the vertical v-line co-incident/co-linear with the 'silhouette edge'. Mirror about Centreline.

12. Extrude-Cut-Revolve around vertical axis shown (bottom left)
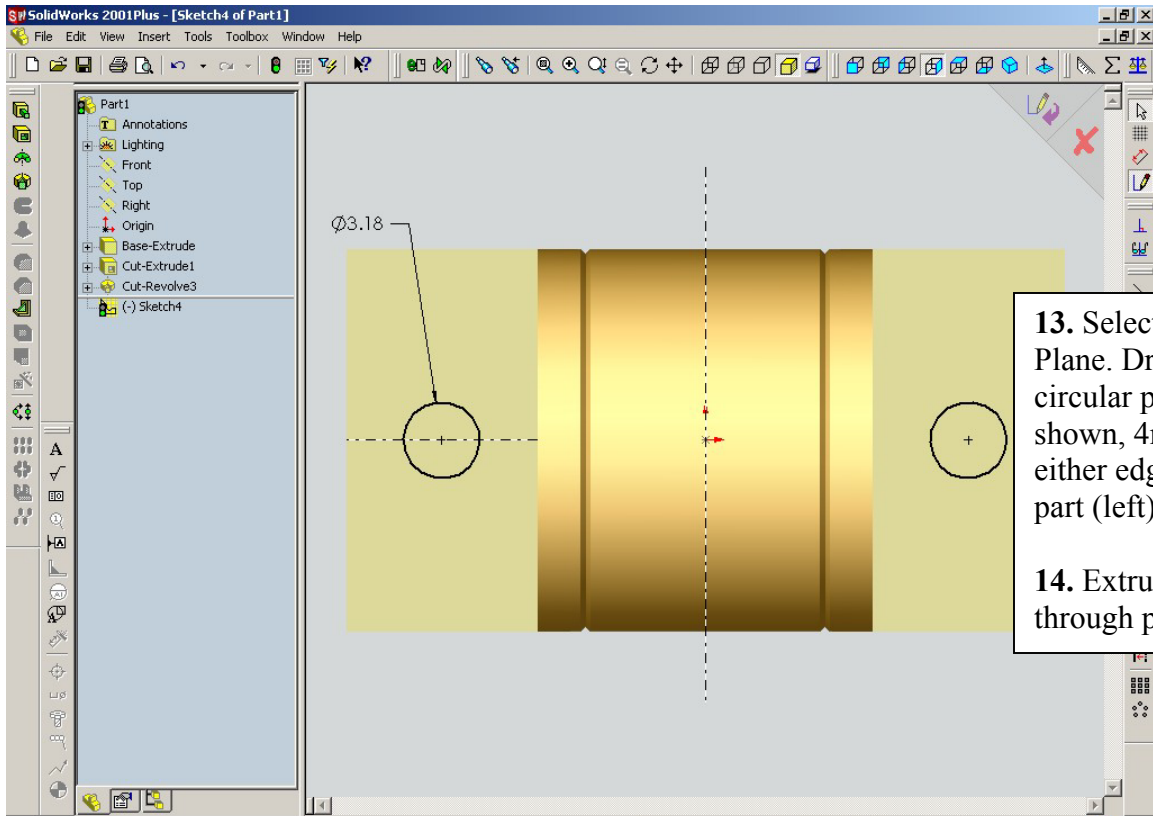
13. Select RIGHT Plane. Draw circular profiles shown, 4mm from either edge of the part (left).

14. Extrude-Cut, through part.
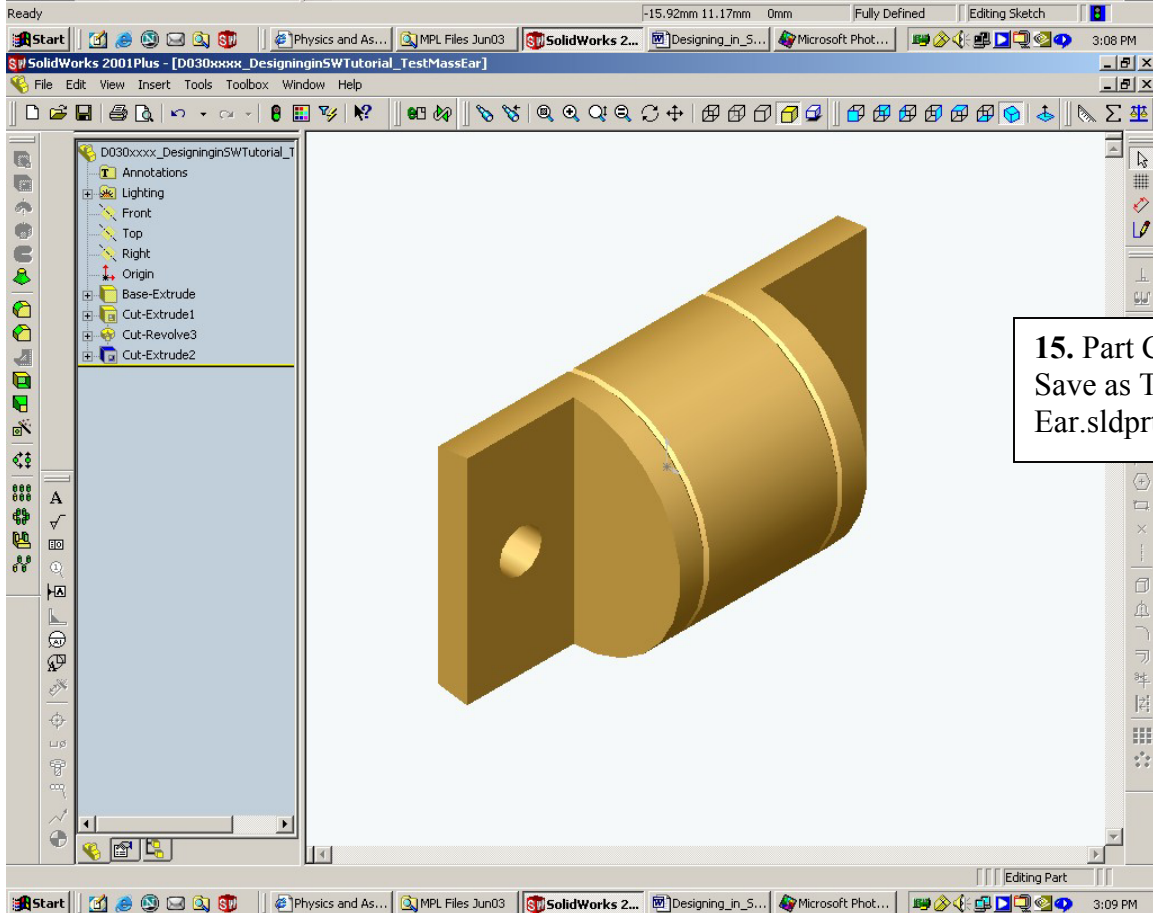
15. Part Complete. Save as Test Mass Ear.sldprt
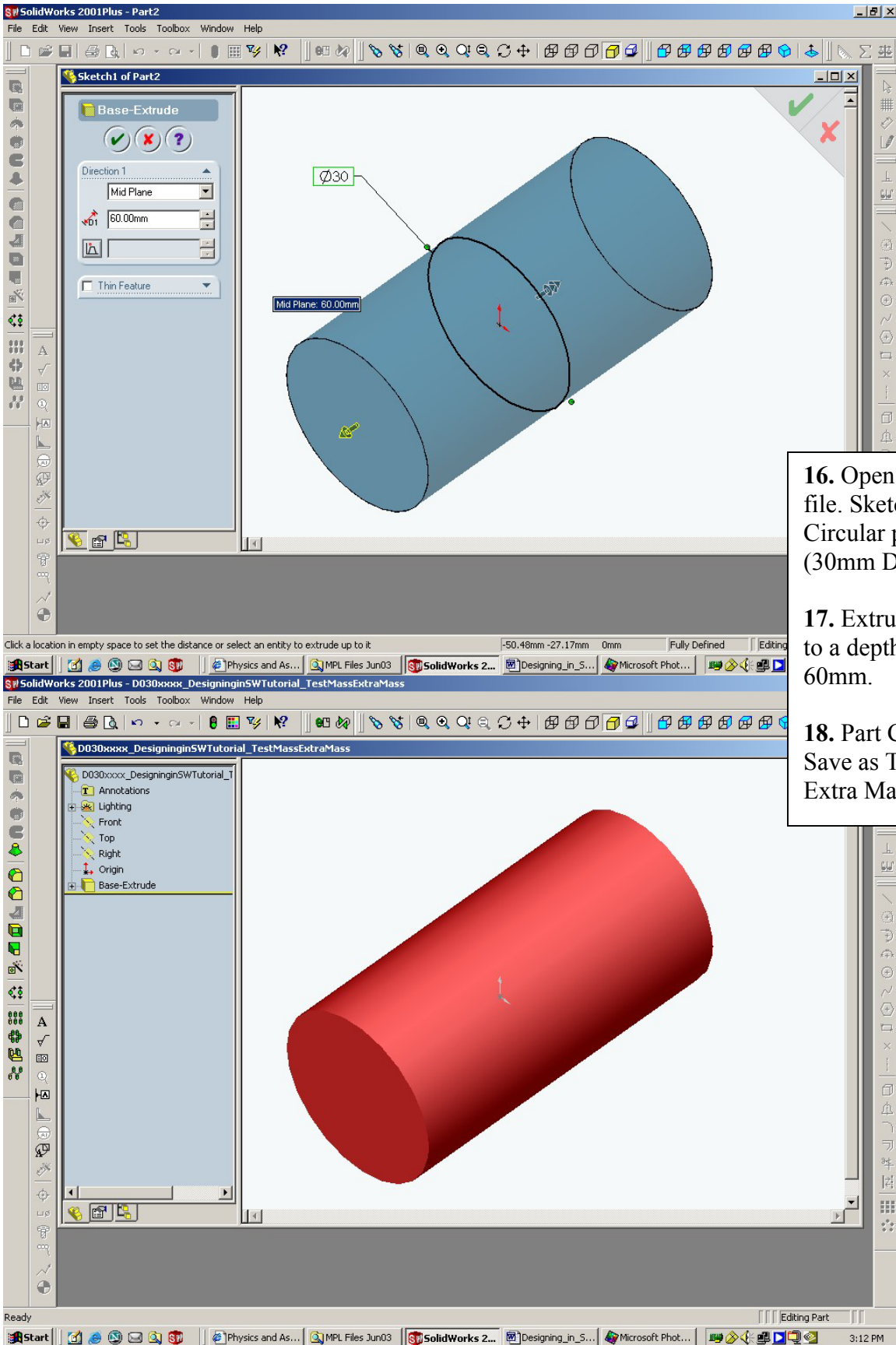
16. Open New Part file. Sketch Circular profile (30mm DIA.).

17. Extrude-Boss, to a depth of 60mm.

18. Part Complete. Save as Test Mass Extra Mass.sldprt
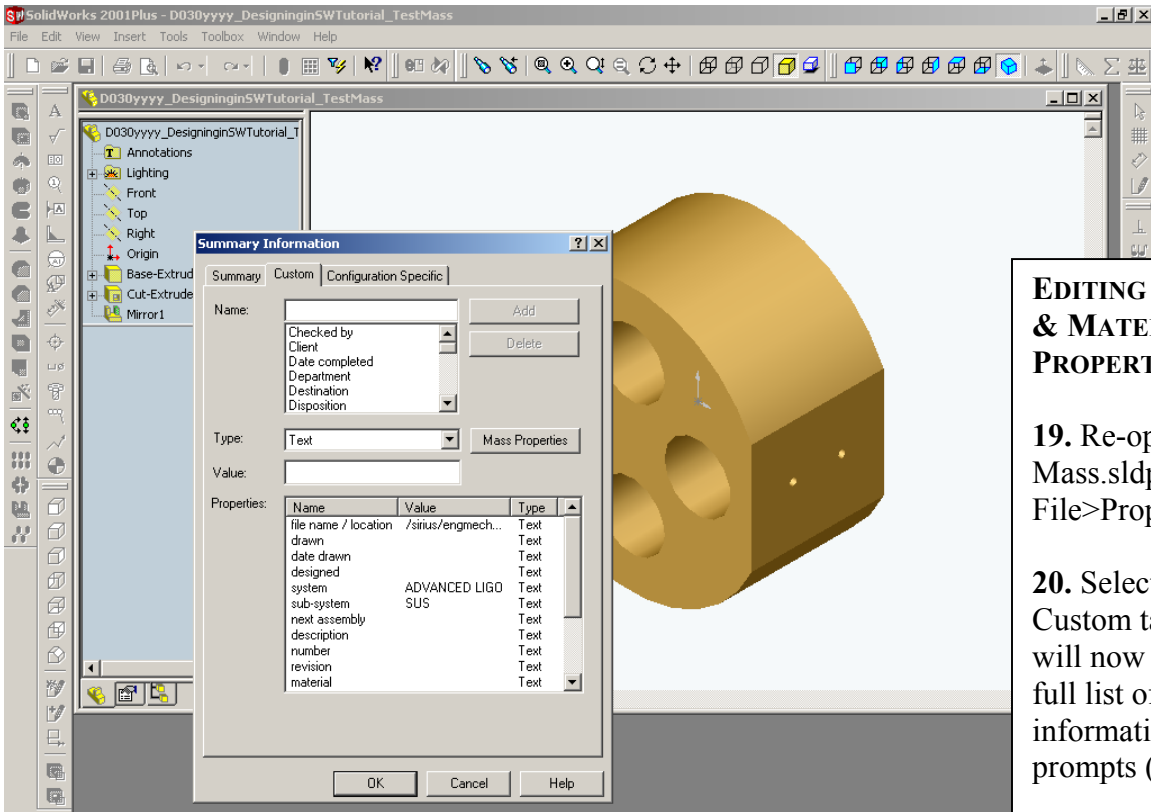
EDITING CUSTOM & MATERIAL PROPERTIES

19. Re-open Test Mass.sldprt. Go to File>Properties.

20. Select the Custom tab - you will now notice a full list of custom information prompts (top left).

21. Enter the information shown (bottom left). Make sure that the 'Modify' button is pressed after each entry.

This information will be picked up by the BOM and Smart Drawing Template when opened (later in the tutorial).
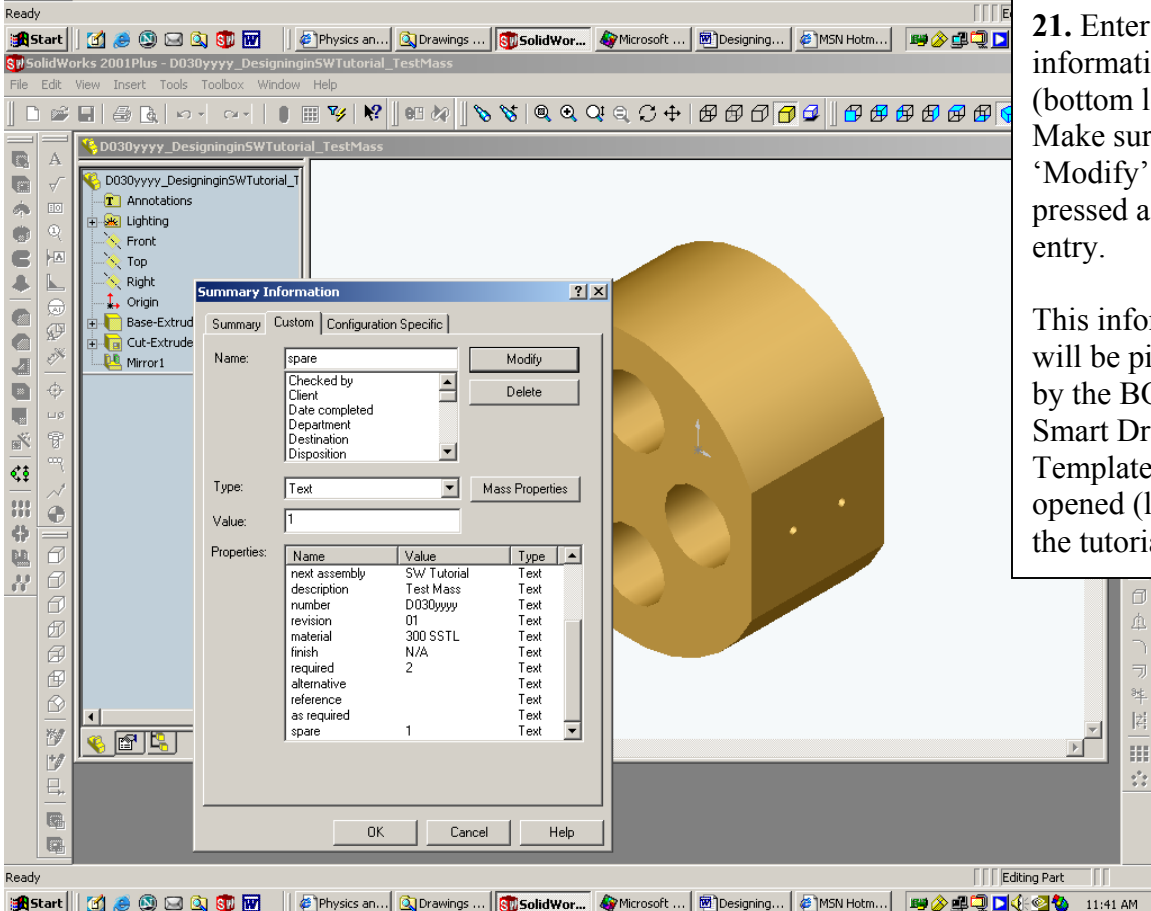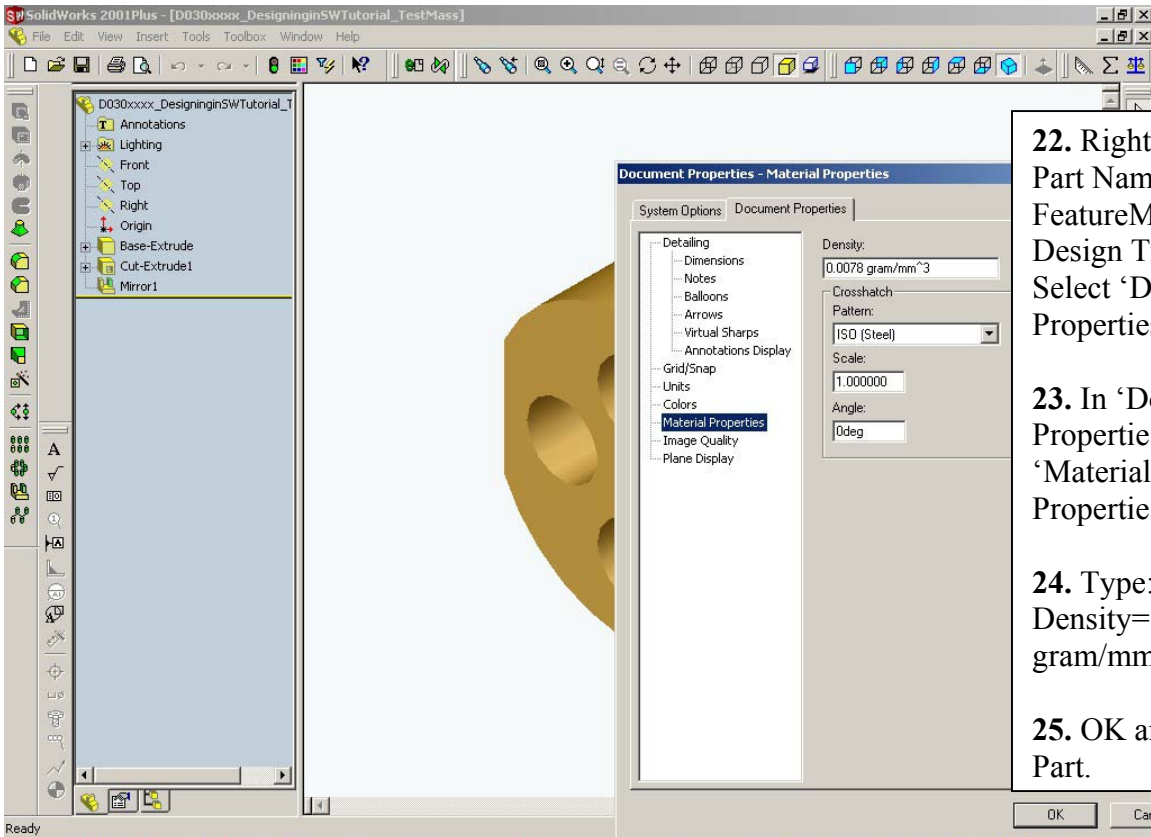
22. Right-click Part Name in FeatureManager Design Tree. Select 'Document Properties'.

23. In 'Document Properties', select 'Material Properties'.

24. Type: Density= 0.0078 gram/mm^3

25. OK and save Part.
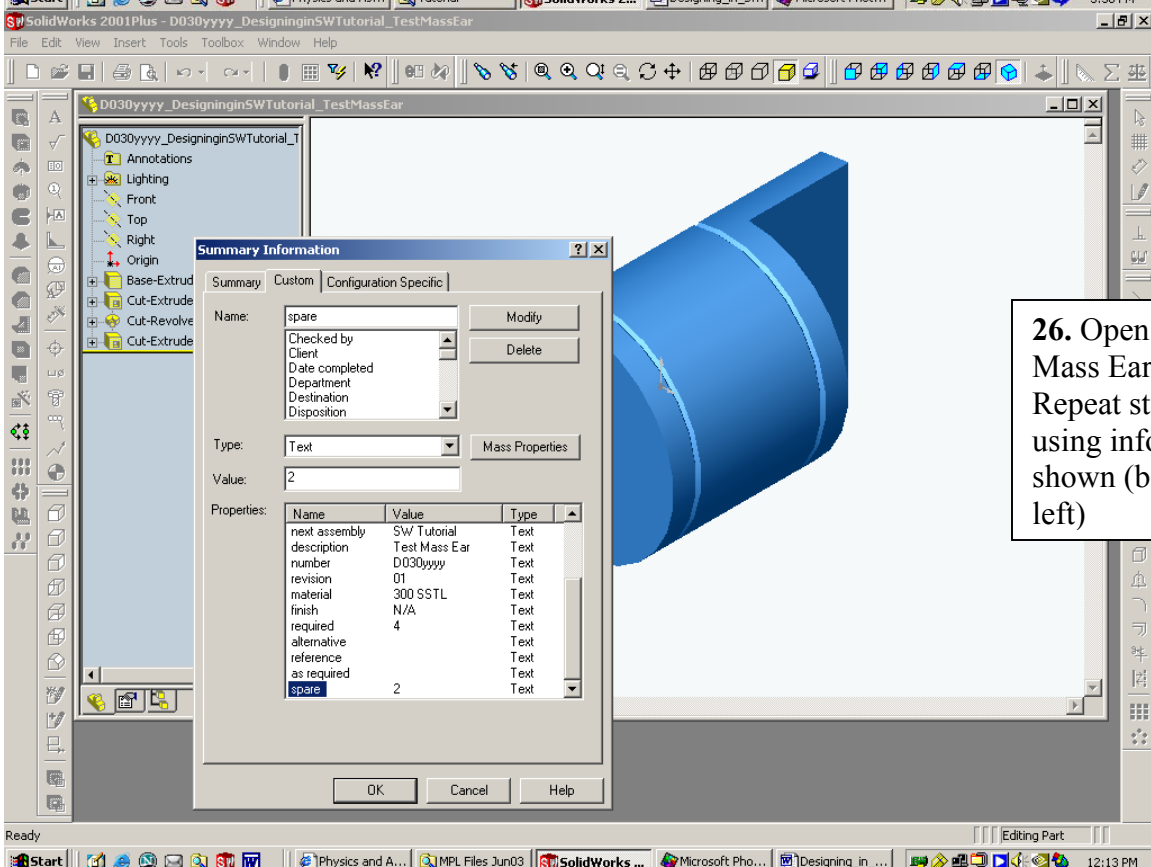
26. Open Test Mass Ear.sldprt. Repeat steps 19-25 using information shown (bottom left)
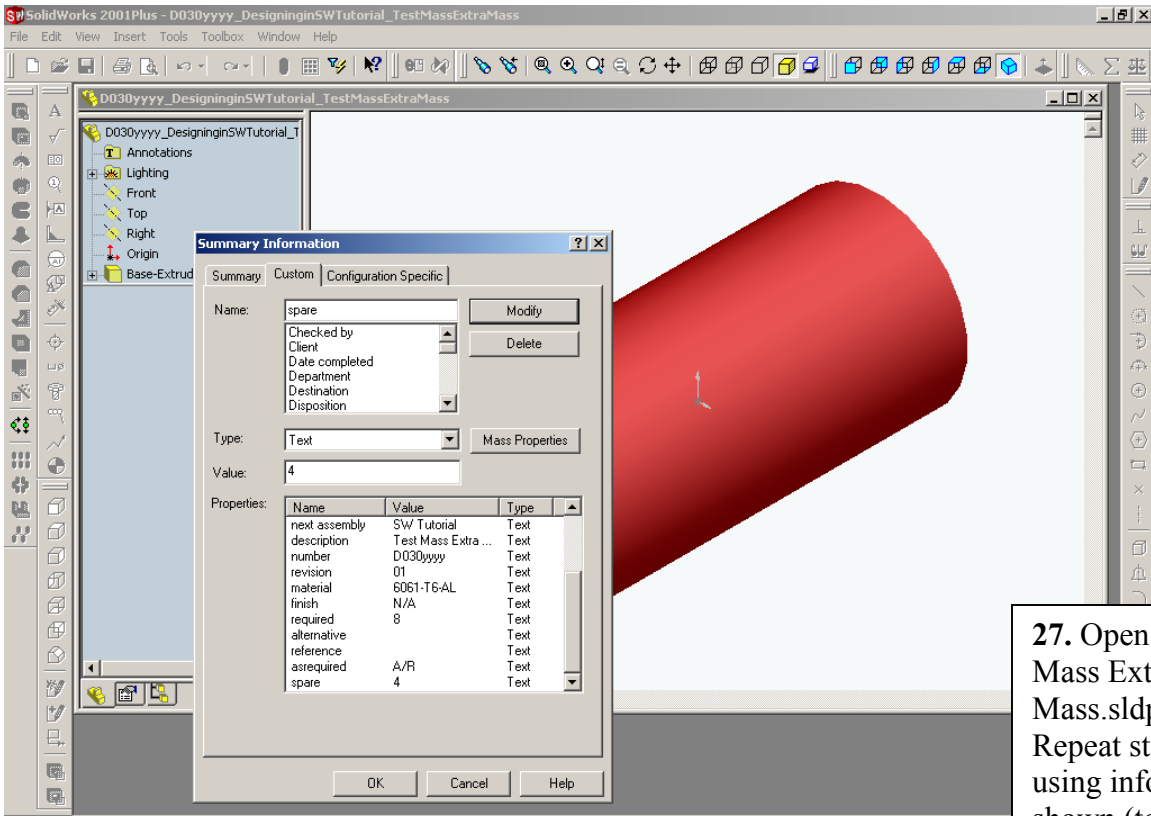
27. Open Test Mass Extra Mass.sldprt. Repeat steps 19-25 using information shown (top left & bottom left).

NOTE: an extra instruction 'asrequired: A/R' has been added.

NOTE: the extra masses are Aluminium and should have a Density of 0.0027g/mm^3.
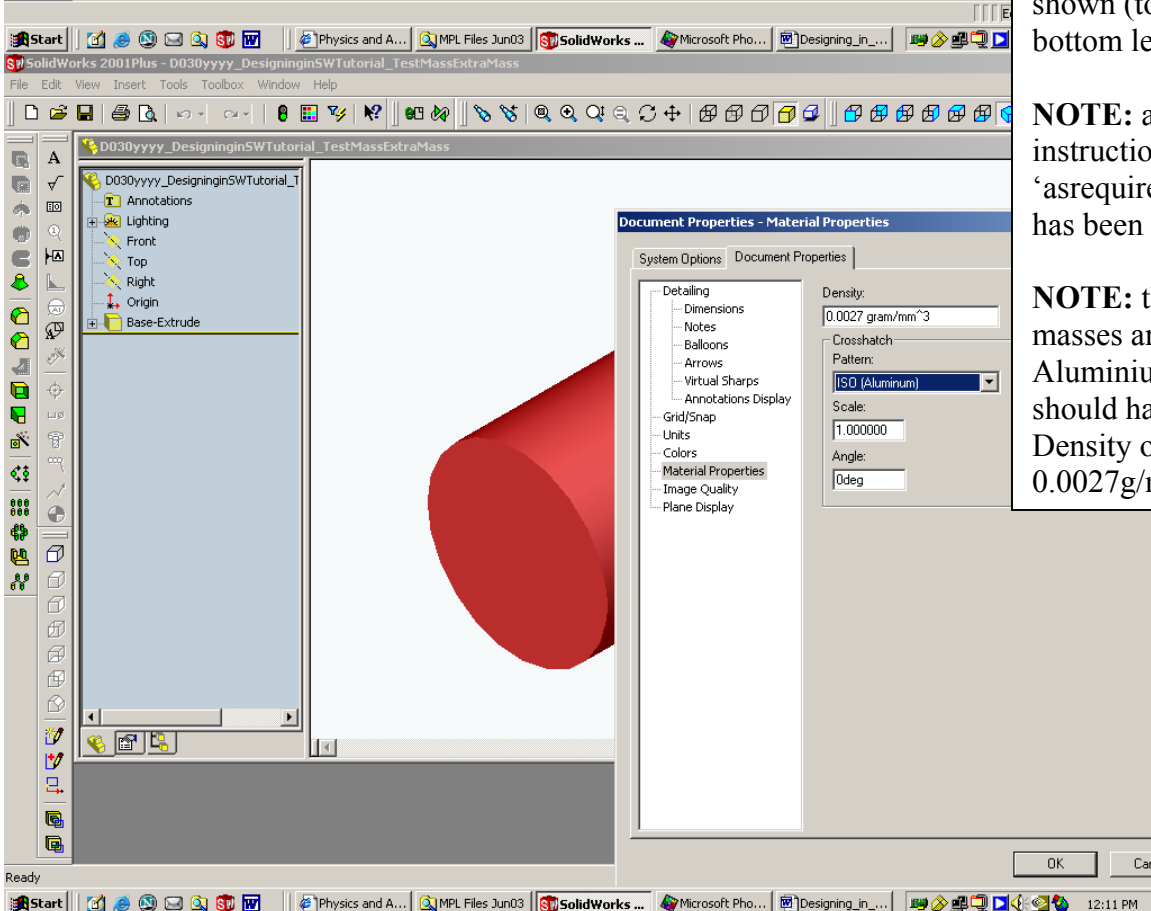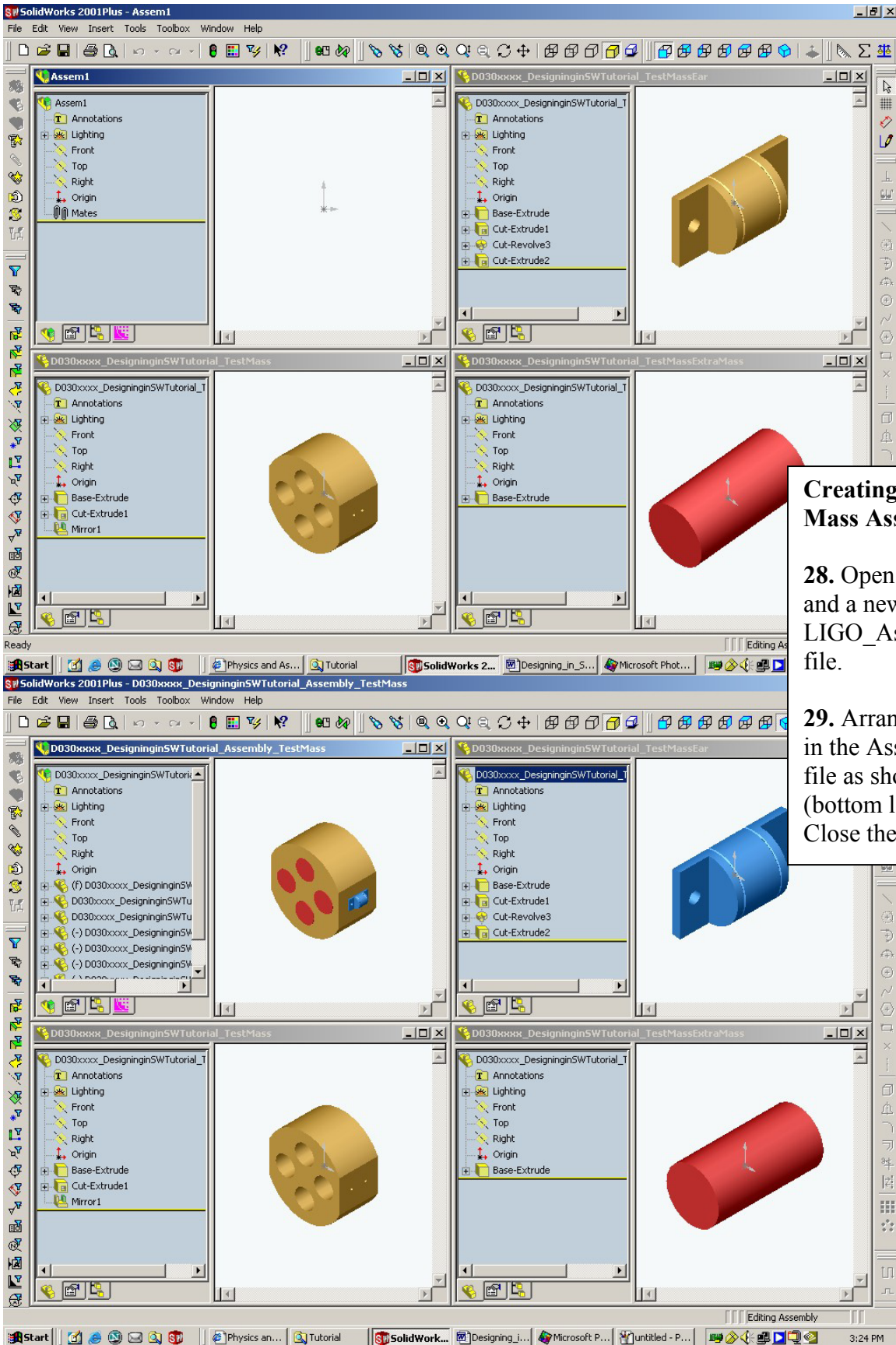
Creating the Test Mass Assembly

28. Open all parts and a new LIGO_Assembly file.

29. Arrange Parts in the Assembly file as shown (bottom left). Close the part files
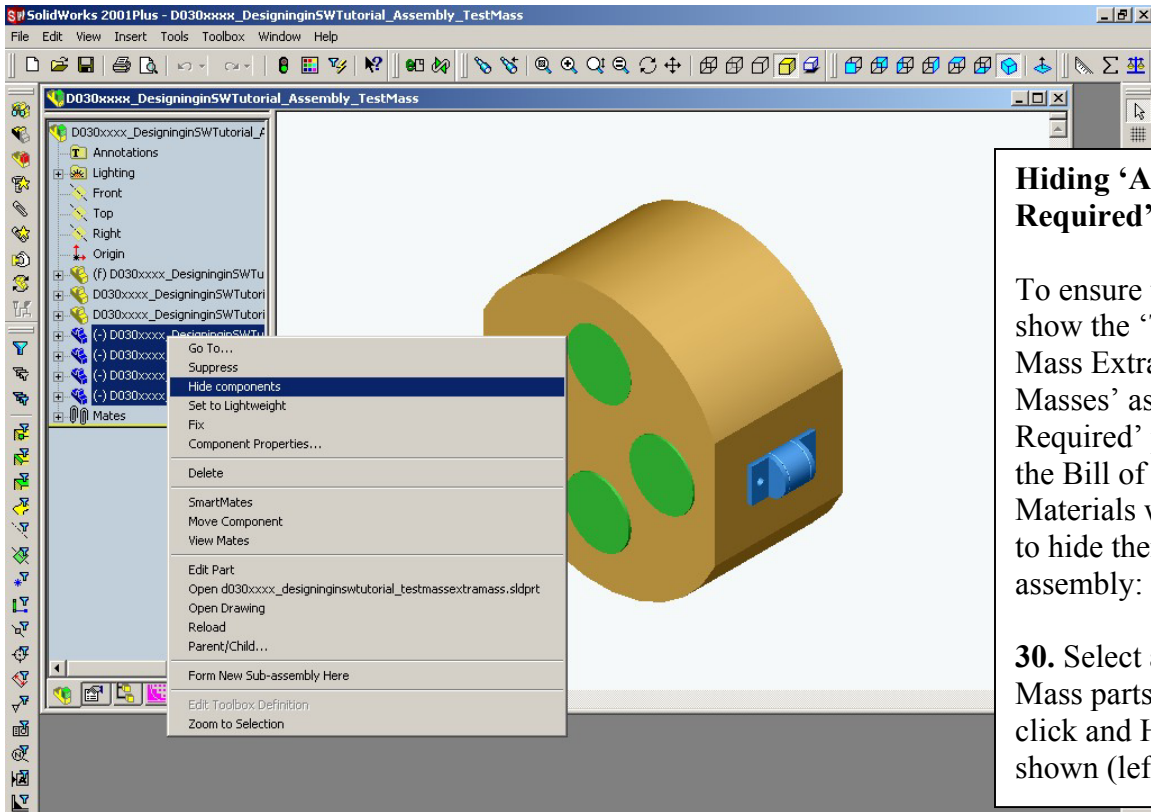
Hiding 'As Required' Parts

To ensure that we show the 'Test Mass Extra Masses' as an 'As Required' part in the Bill of Materials we need to hide them in the assembly:

30. Select all Extra Mass parts, right click and Hide as shown (left).
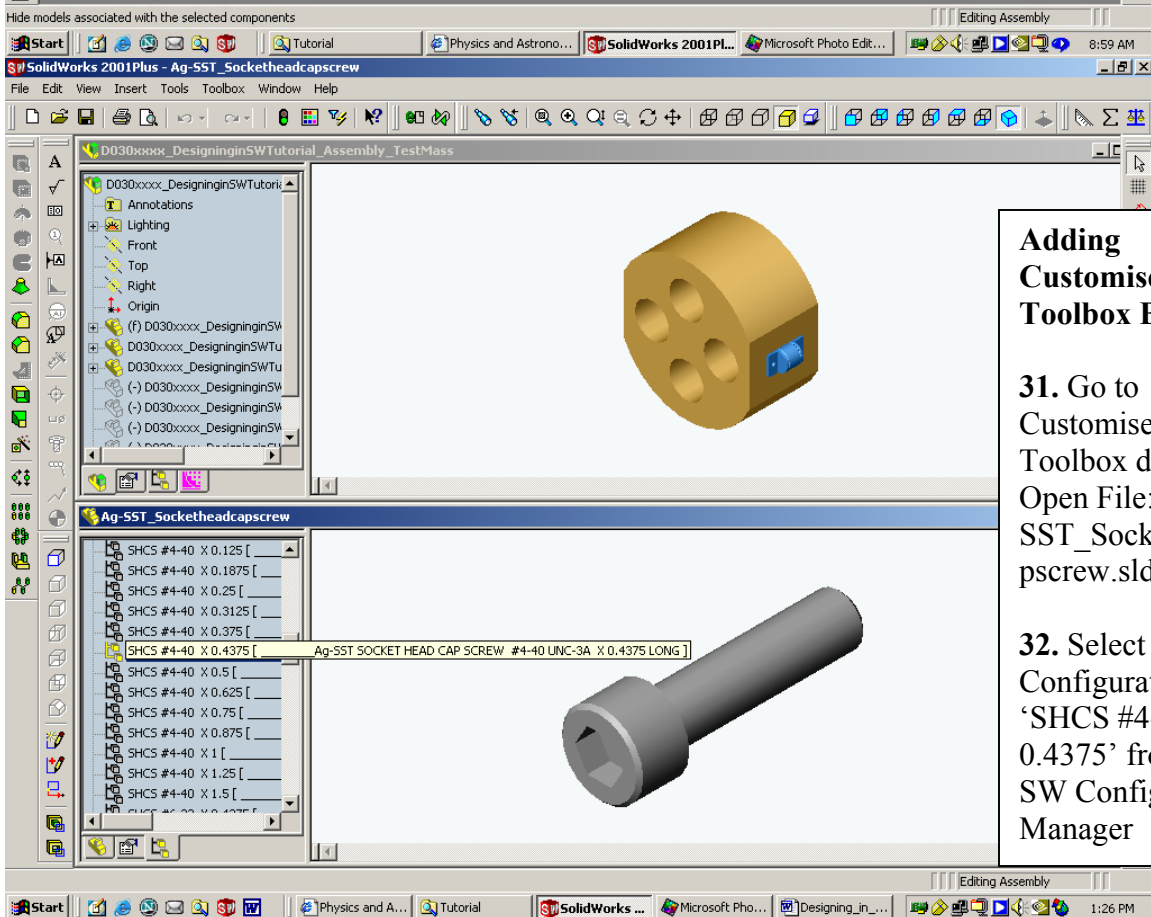
Adding Customised Toolbox Bolts

31. Go to Customised Toolbox directory, Open File: Ag-SST_Socketheadcapscrew.sldprt

32. Select Configuration 'SHCS #4-40 X 0.4375' from the SW Configuration-Manager
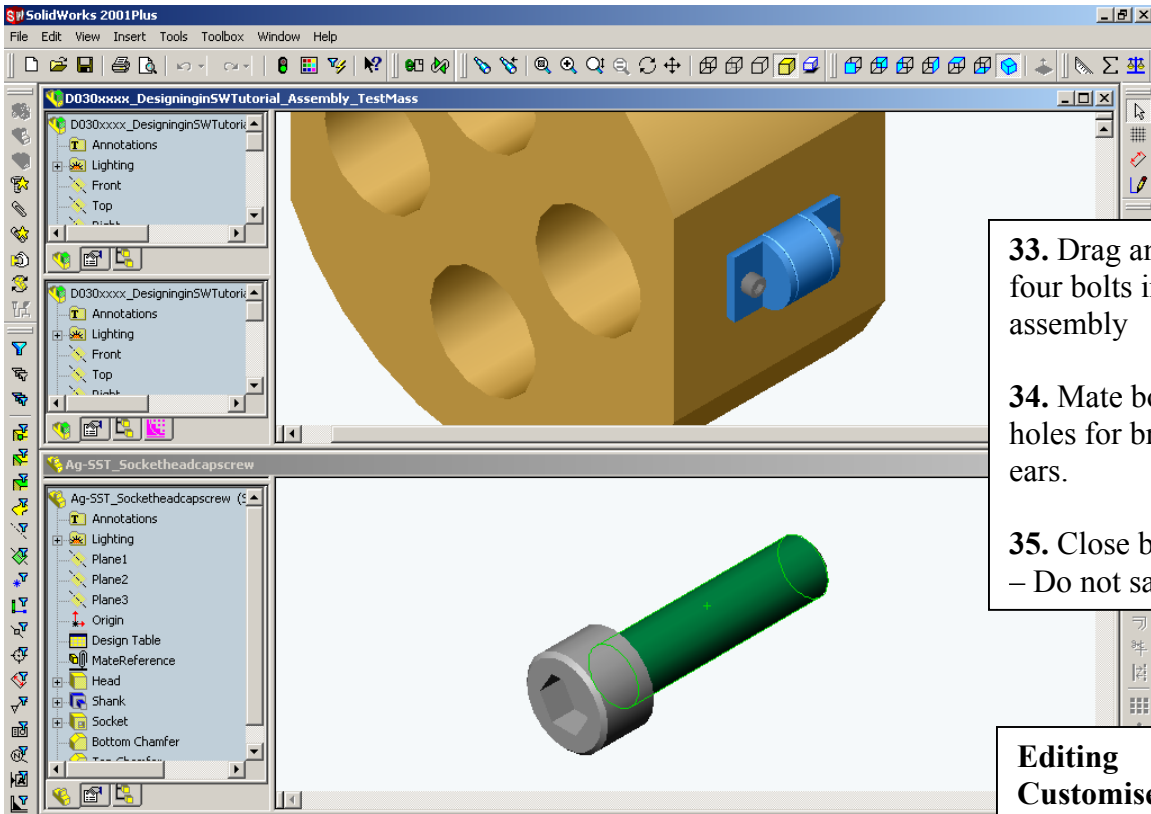
33. Drag and drop four bolts into assembly

34. Mate bolts into holes for break off ears.

35. Close bolt file – Do not save.
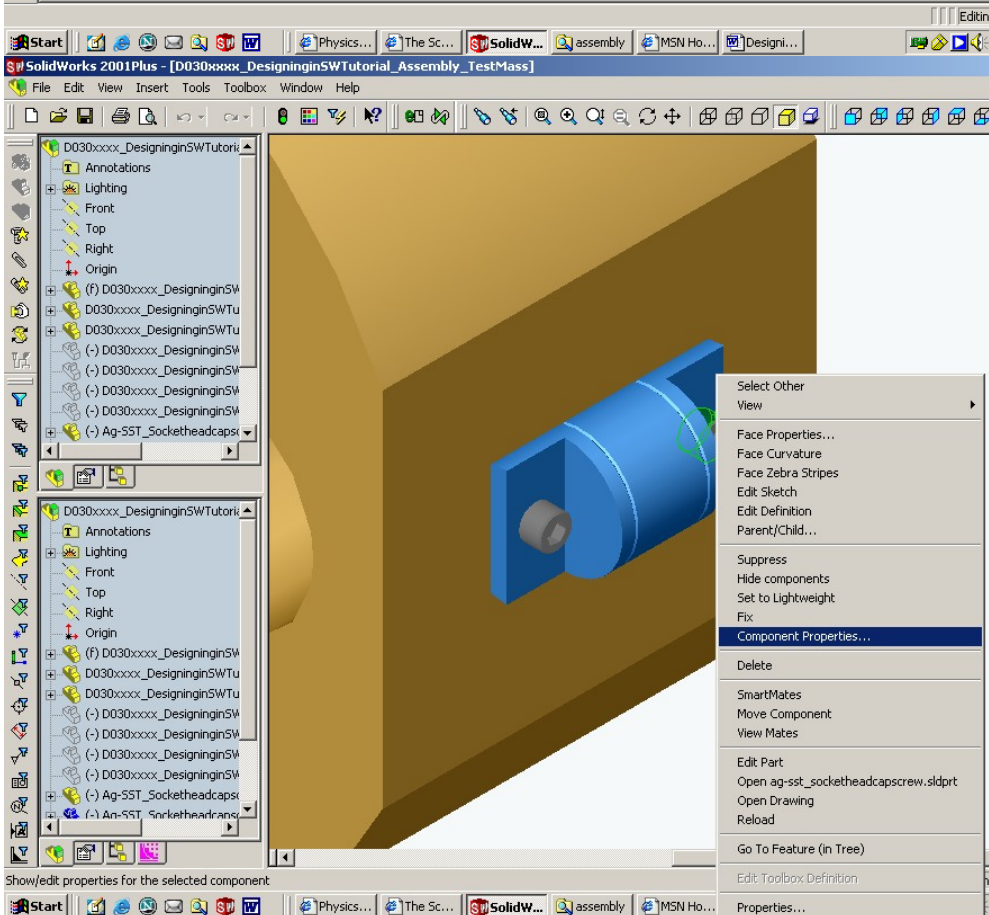
Editing Customised Toolbox Component Properties

If we decide that the bolt we have inserted is the wrong size this can easily be altered:

36. Right-click surface of bolt head or it's name in the feature manager design tree, select 'Component Properties…'

37. Change all bolt configurations to #4-40 X 0.375 (see bottom left and top screenshot on the next page)
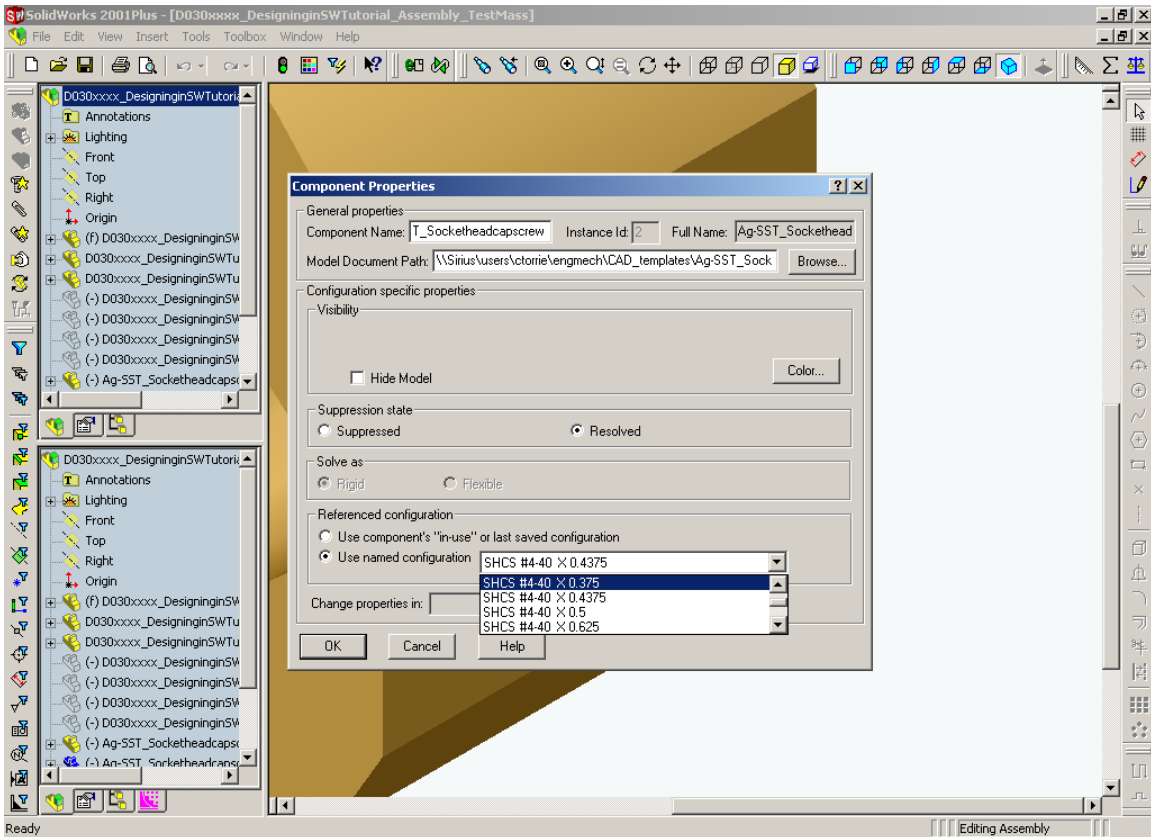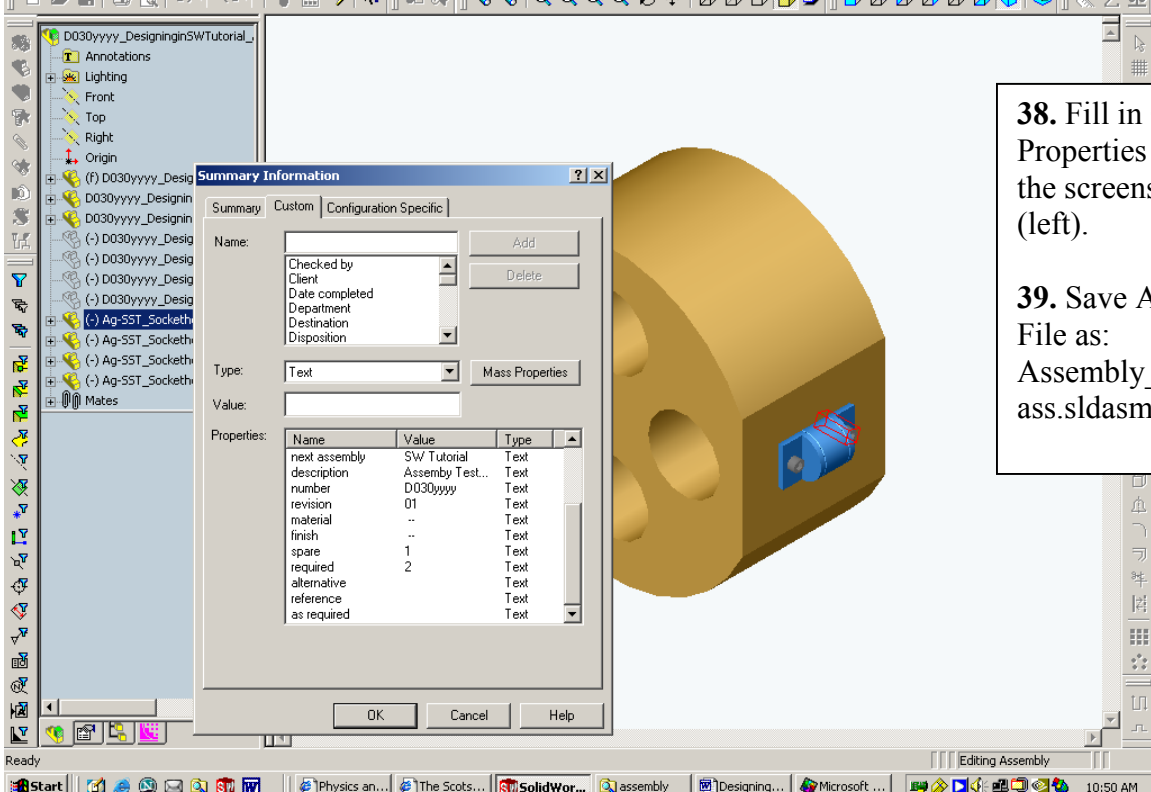
38. Fill in Custom Properties as per the screenshot (left).

39. Save Assembly File as: Assembly_Test_Mass.sldasm
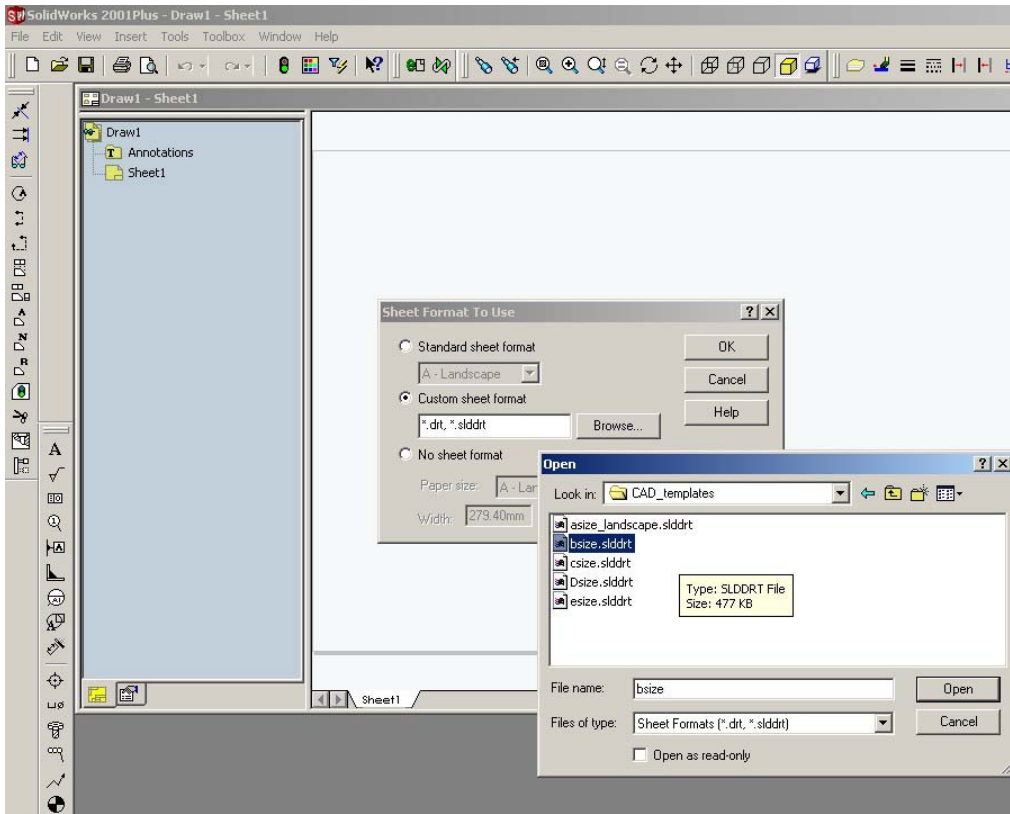
CREATING A FULLY AUTOMATED ENGINEERING DRAWING

40. Open a new LIGO_Drawing file

41. Insert a 'Custom Sheet format' and select the LIGO 'bsize.slddrt' template.
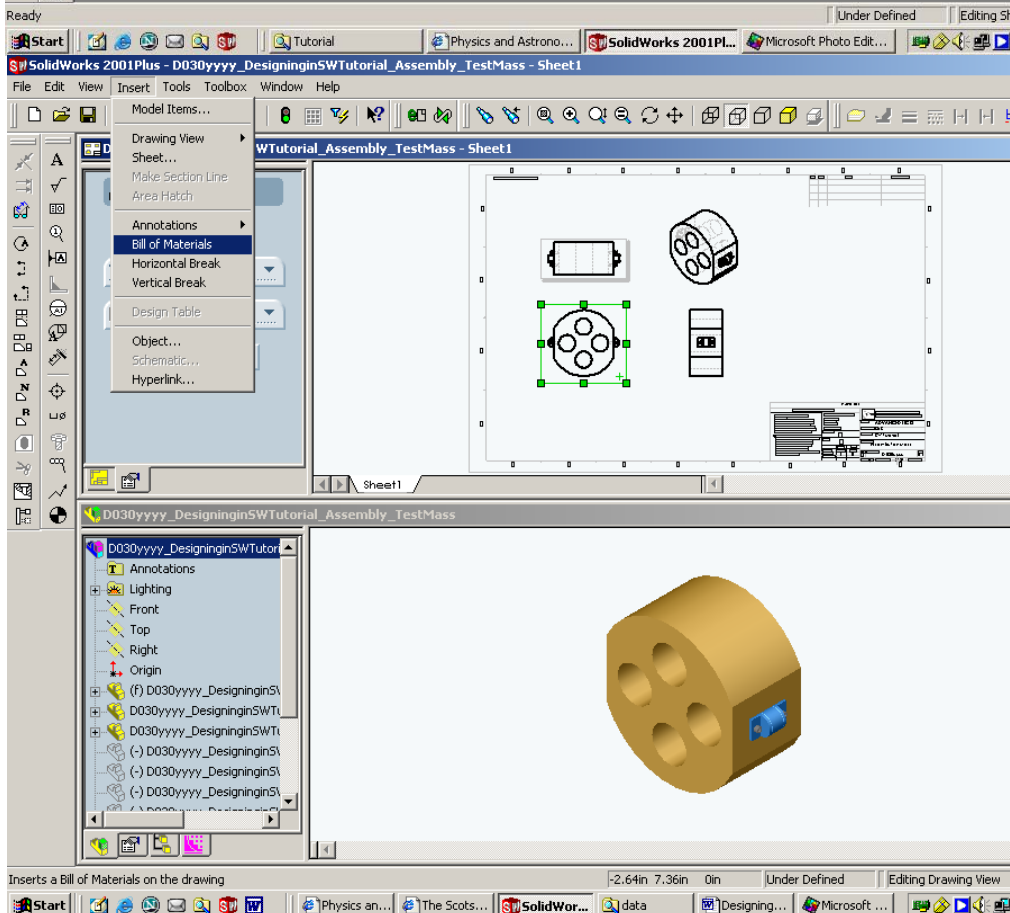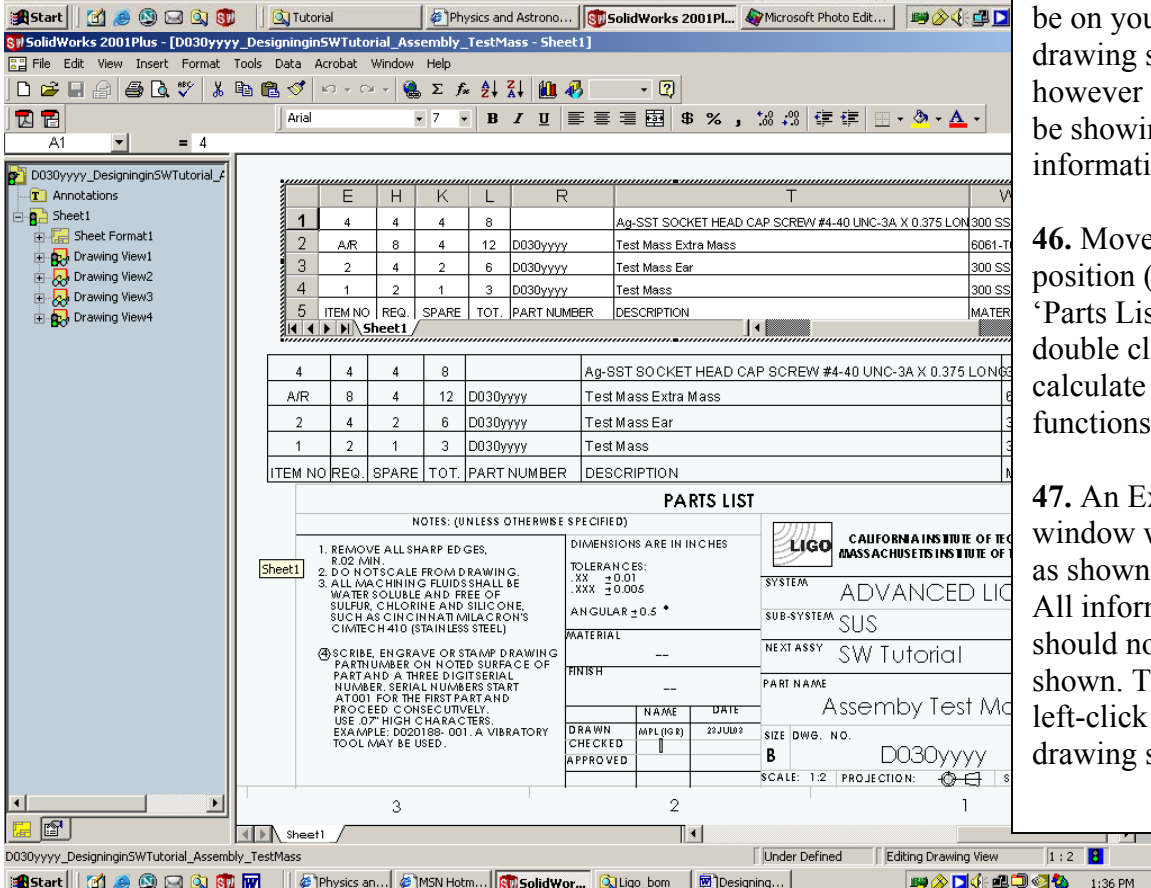
ADDING A BILL OF MATERIALS

42. Attach the assembly by dragging it from the assembly window into the drawing window. (This is best done by holding CTRL and dragging the assembly name from the top of the Featuremanager design tree)

43. Select any view of the assembly in the drawing and go to: Insert>Bill of Materials as shown (top left). Open 'D030384_07_LIGOBOM.xls'.
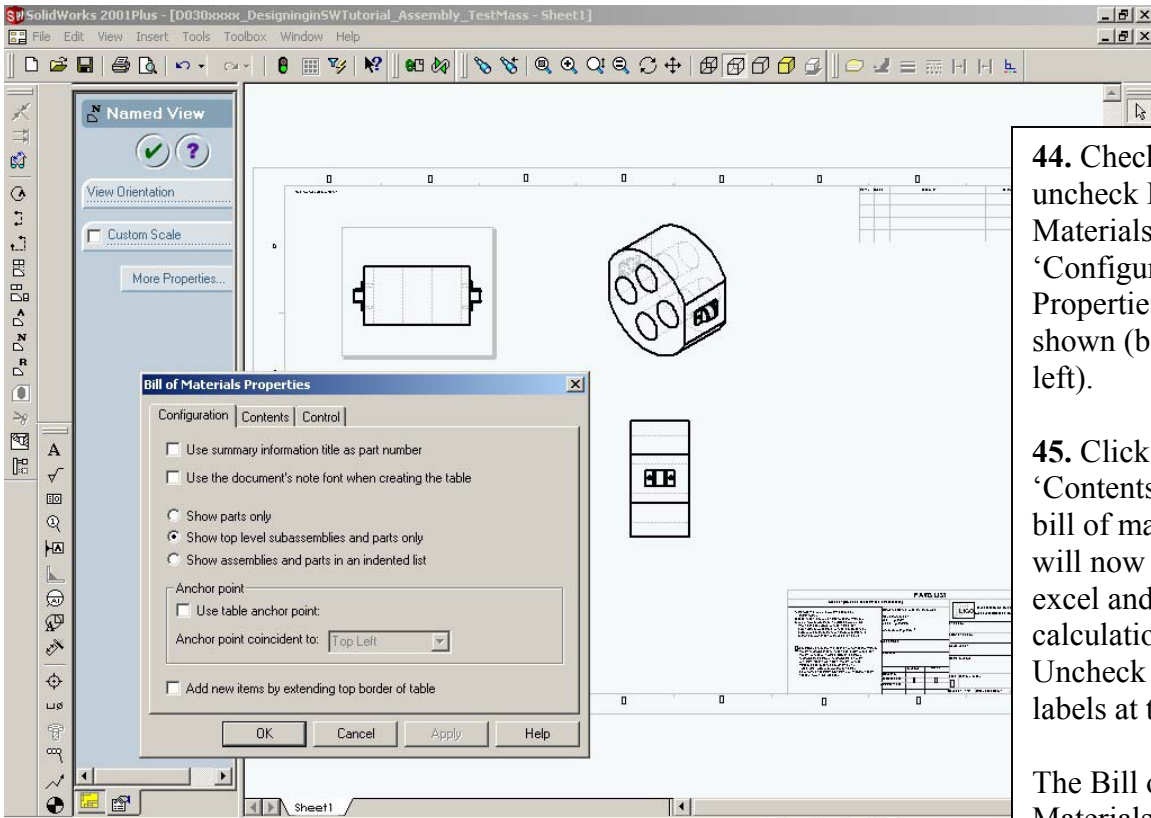
**44.** Check and/or uncheck Bill of Materials 'Configuration' Properties as shown (bottom left).

45. Click 'Contents' tab (the bill of materials will now open excel and begin calculation), Uncheck 'Display labels at top'. OK.

The Bill of Materials will now be on your drawing sheet, however it will not be showing all information.

46. Move to position (above 'Parts List') then double click to re-calculate Excel functions.

47. An Excel type window will open as shown (top left). All information should now be shown. To get out left-click the drawing sheet.
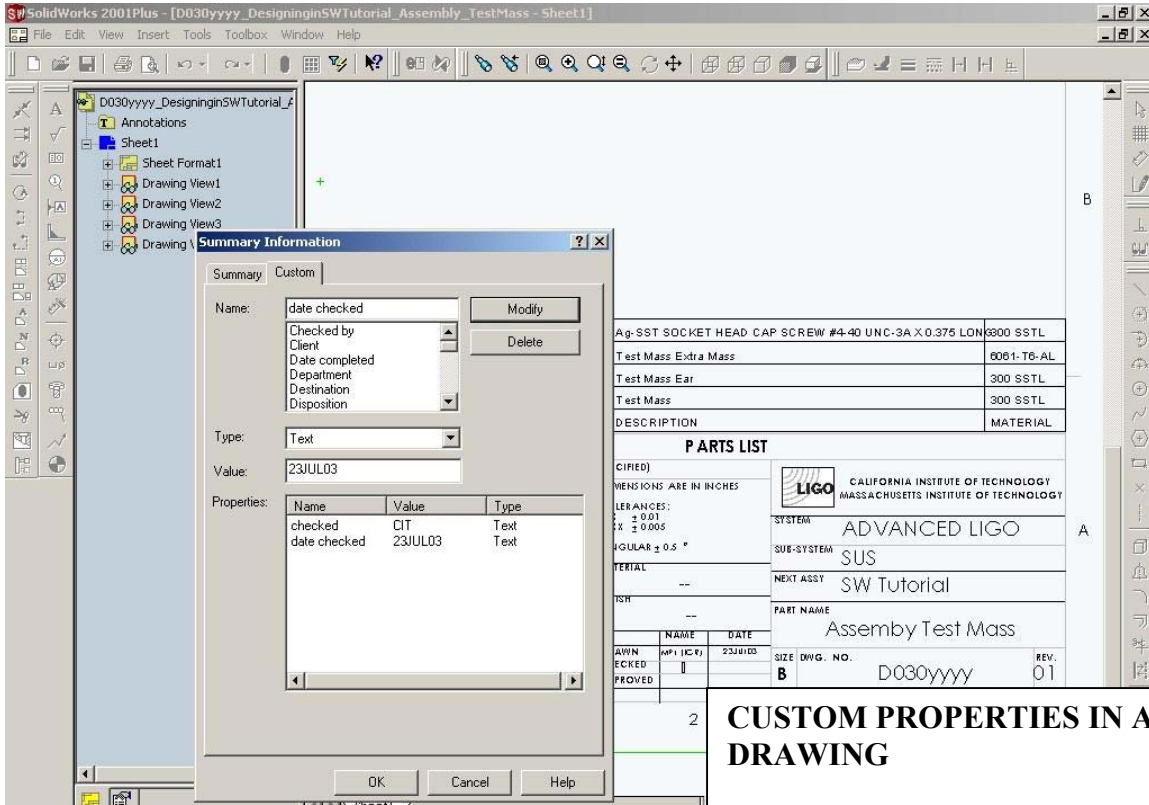
CUSTOM PROPERTIES IN A DRAWING

To indicate that a drawing has been checked. This should be done in the drawing's custom properties.
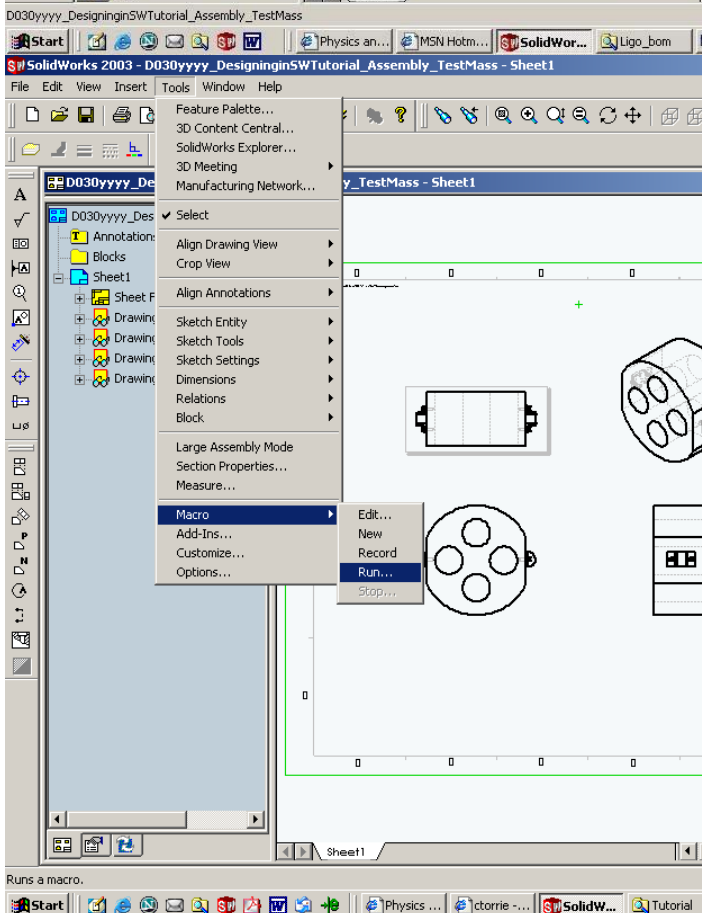
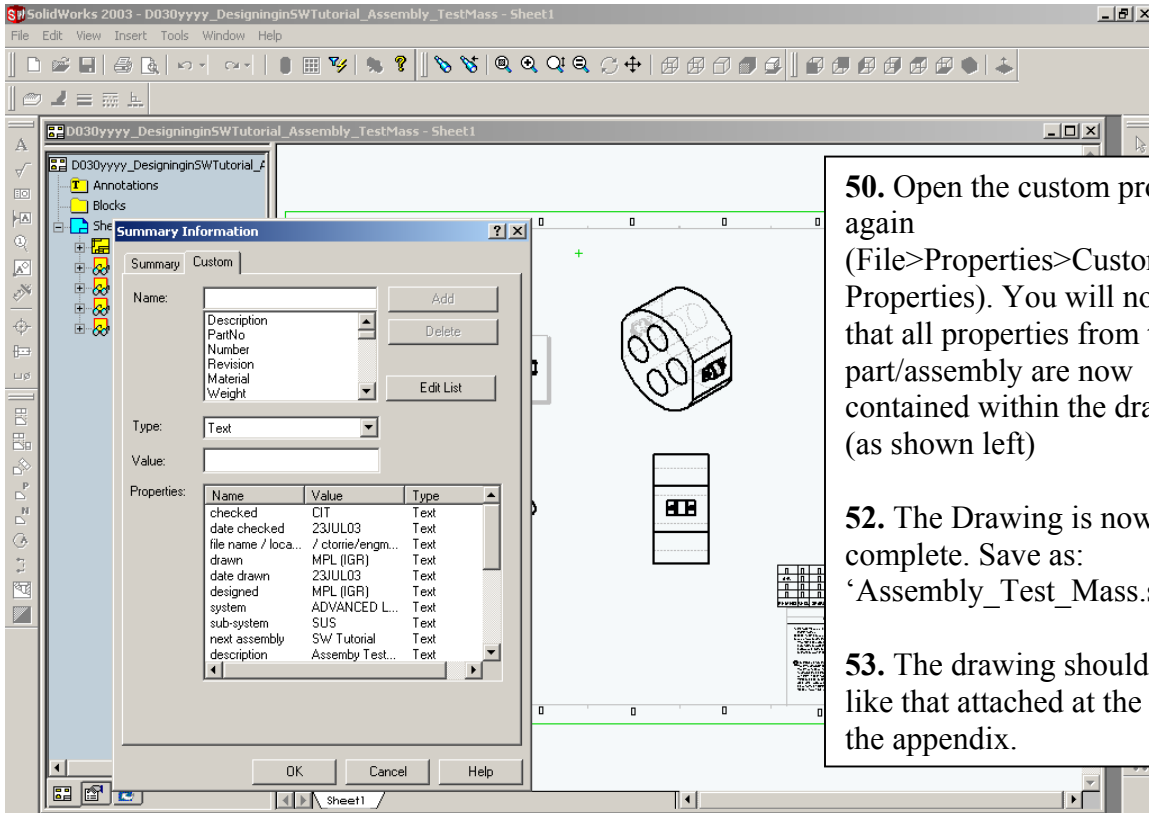**48.** Go to File>Properties>Custom Properties. Enter the checker's name and date.

Should you require to add further notes to the drawing, e.g. manufacture notes, this can still be done by right clicking the drawing and pressing 'edit sheet format'.

**COPYING ACROSS CUSTOM PROPERTIES FROM A PART (OR ASSEMBLY) TO A DRAWING**

49. Go to: Tools>Macro>Run… Select the GOE PropSynch.swp macro and 'Open'.
Now, when you save your drawing a window should pop us asking whether you wish to 'Synchronise document properties?' Select 'YES'.

50. Open the custom properties again (File>Properties>Custom Properties). You will notice that all properties from the part/assembly are now contained within the drawing (as shown left)

52. The Drawing is now complete. Save as: 'Assembly_Test_Mass.slddrw'

53. The drawing should look like that attached at the end of the appendix.
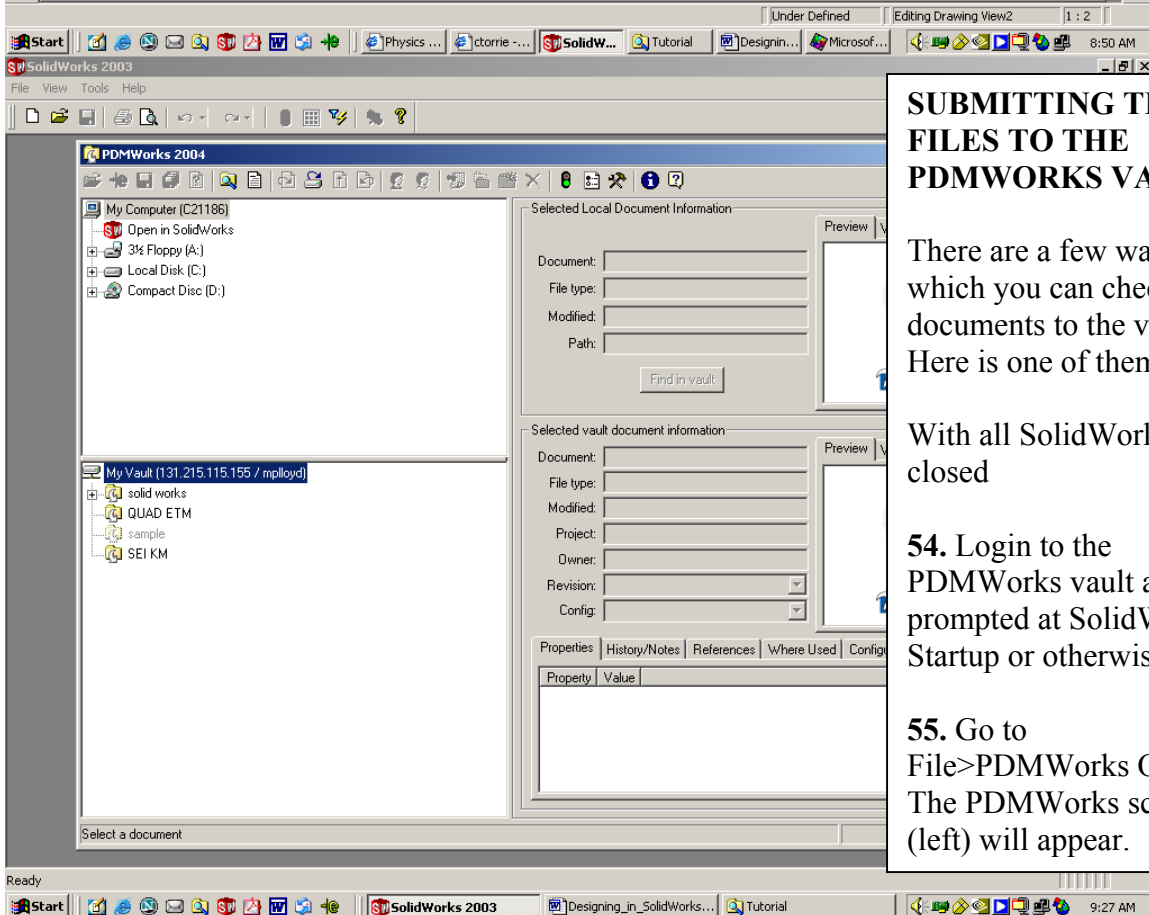
**SUBMITTING THE FILES TO THE PDMWORKS VAULT**

There are a few ways in which you can check-in documents to the vault. Here is one of them:

With all SolidWorks files closed

54. Login to the PDMWorks vault as prompted at SolidWorks Startup or otherwise.

55. Go to File>PDMWorks Open… The PDMWorks screen (left) will appear.
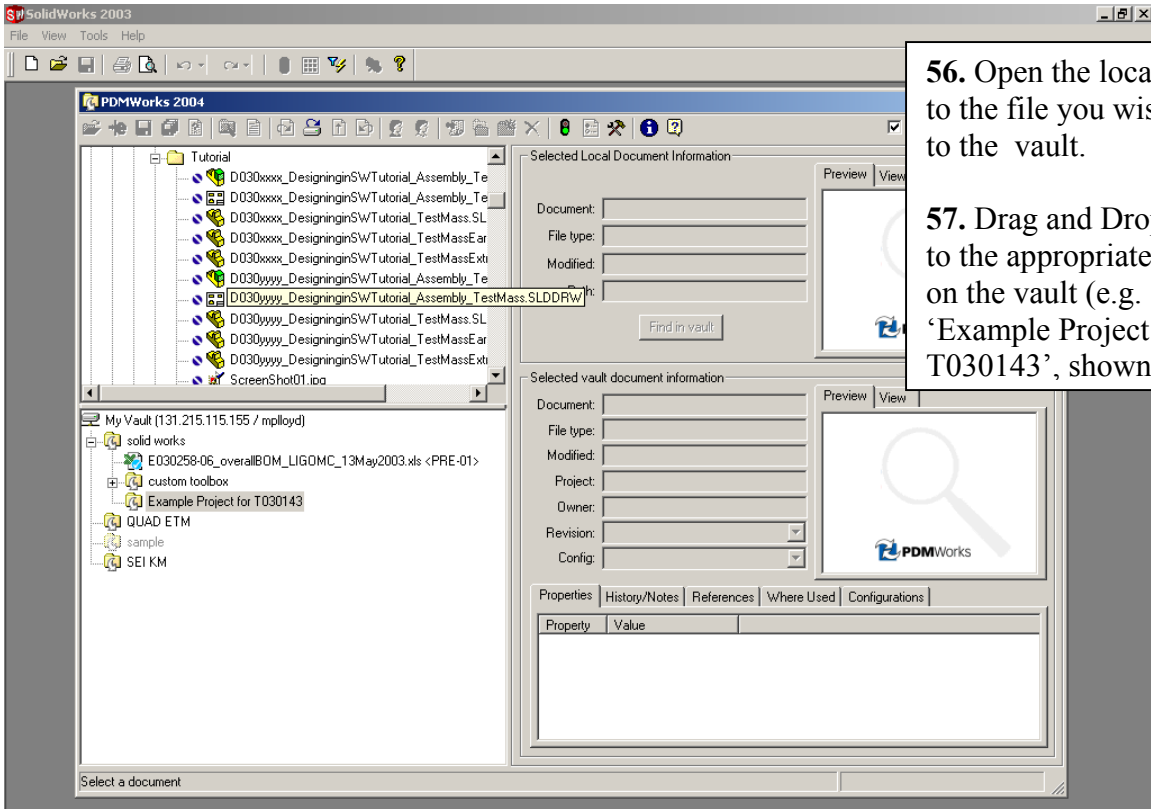
**56.** Open the local folders to the file you wish to add to the vault.

57. Drag and Drop the file to the appropriate folder on the vault (e.g. 'Example Project fro T030143', shown left)
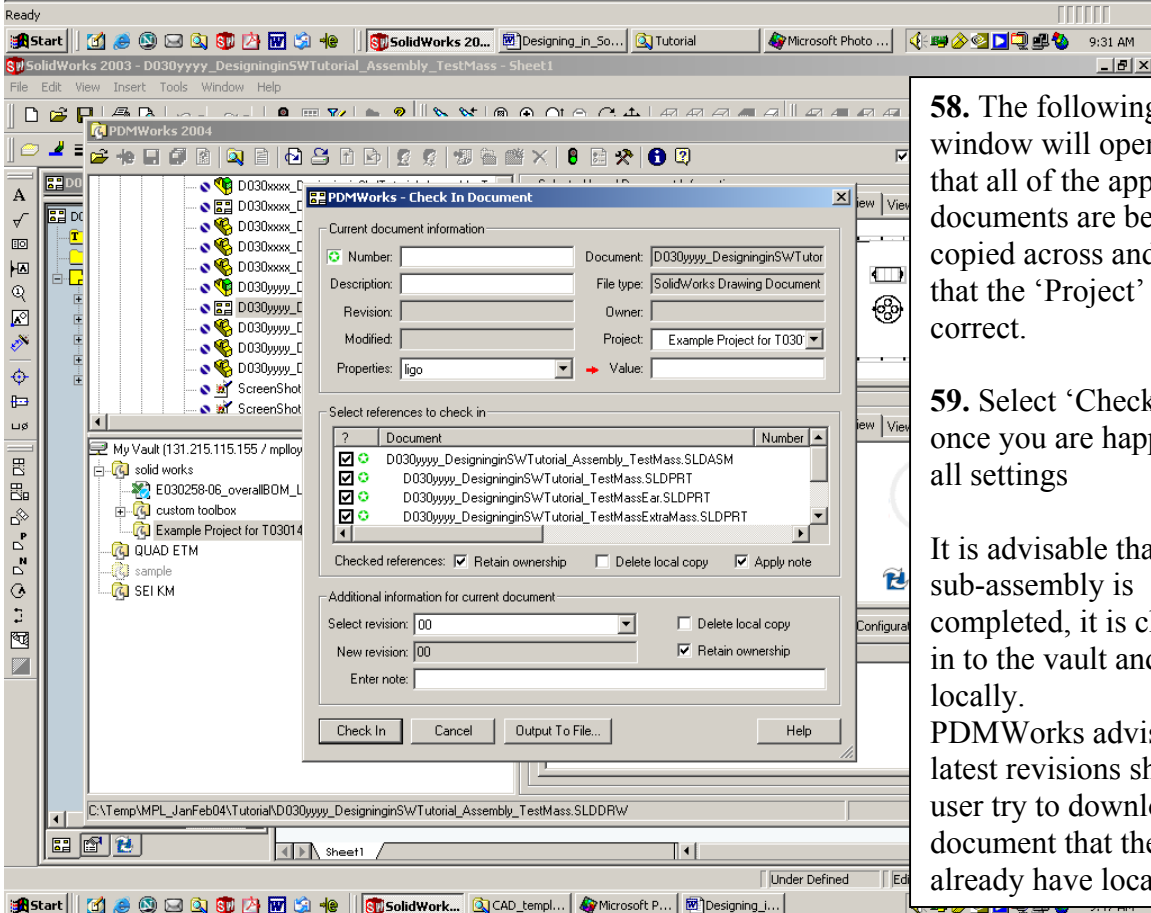
58. The following window will open. Check that all of the appropriate documents are being copied across and check that the 'Project' is correct.

**59.** Select 'Check-in' once you are happy with all settings

It is advisable that once a sub-assembly is completed, it is checked in to the vault and deleted locally.
PDMWorks advises of latest revisions should the user try to download a document that they already have locally.
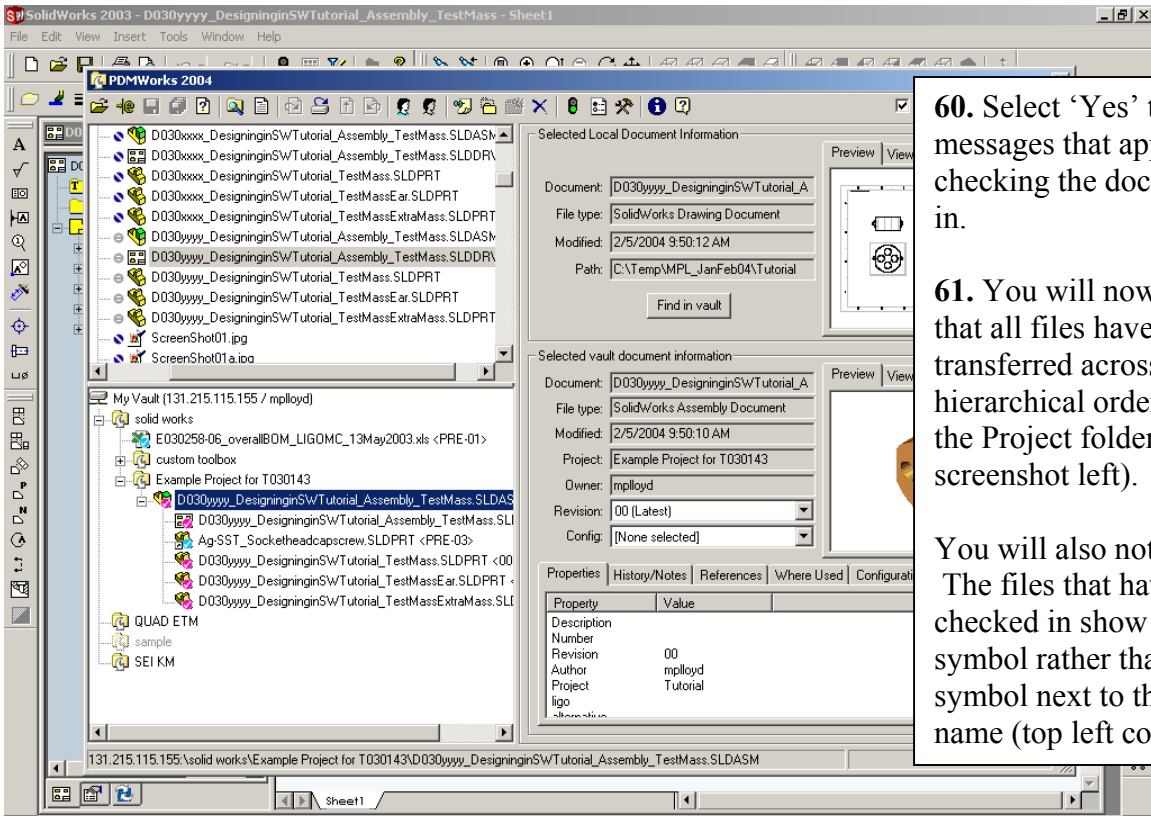
60. Select 'Yes' to the messages that appear on checking the documents in.

61. You will now notice that all files have been transferred across in a hierarchical order below the Project folder (see screenshot left).

You will also notice that The files that have been checked in show a grey symbol rather than a blue symbol next to the file name (top left corner)
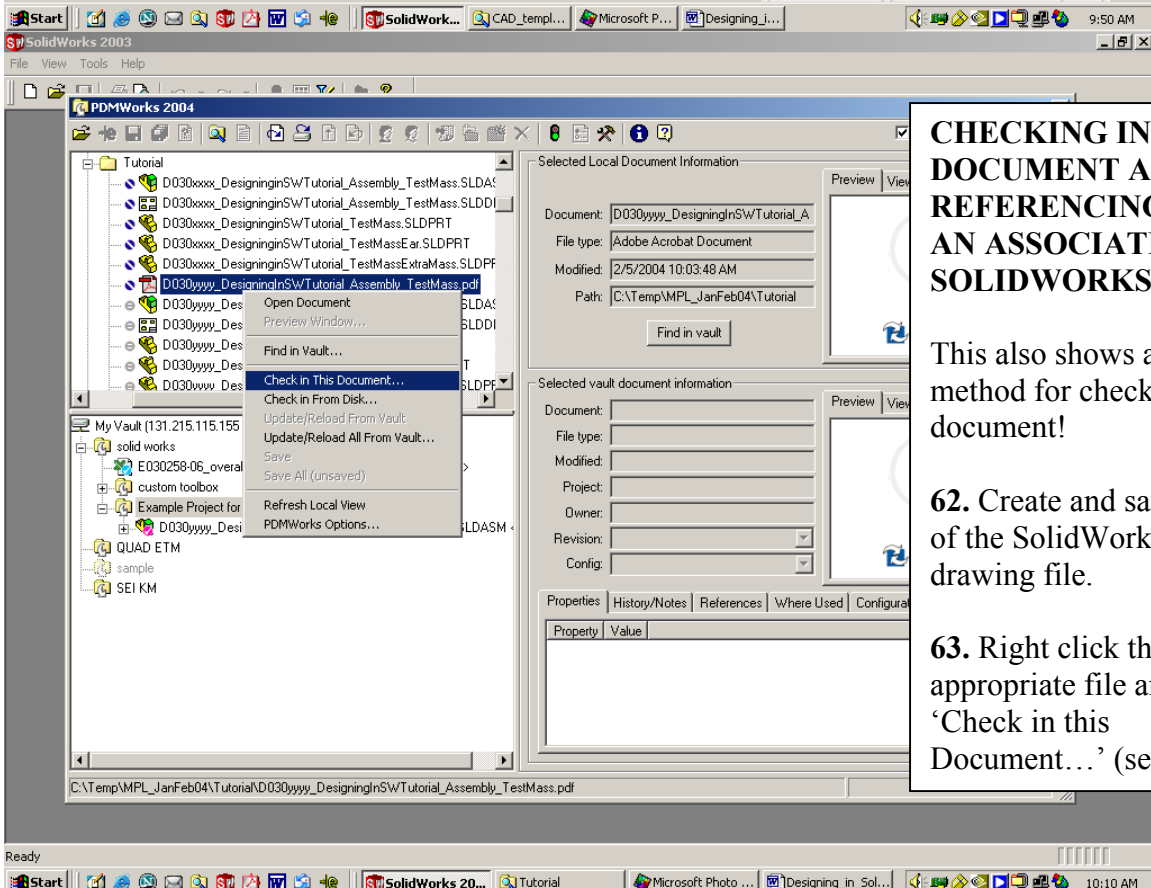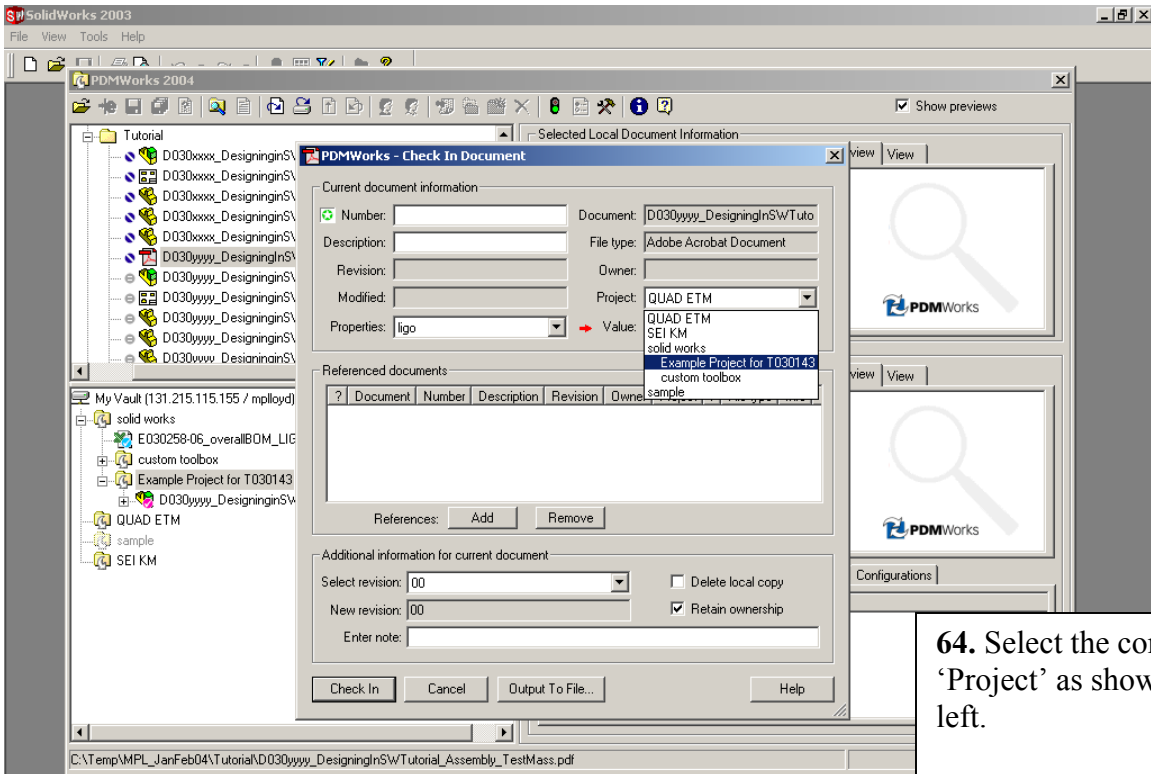

CHECKING IN A PDF DOCUMENT AND REFERENCING TO AN ASSOCIATED SOLIDWORKS FILE

This also shows another method for checking in a document!

62. Create and save a PDF of the SolidWorks drawing file.

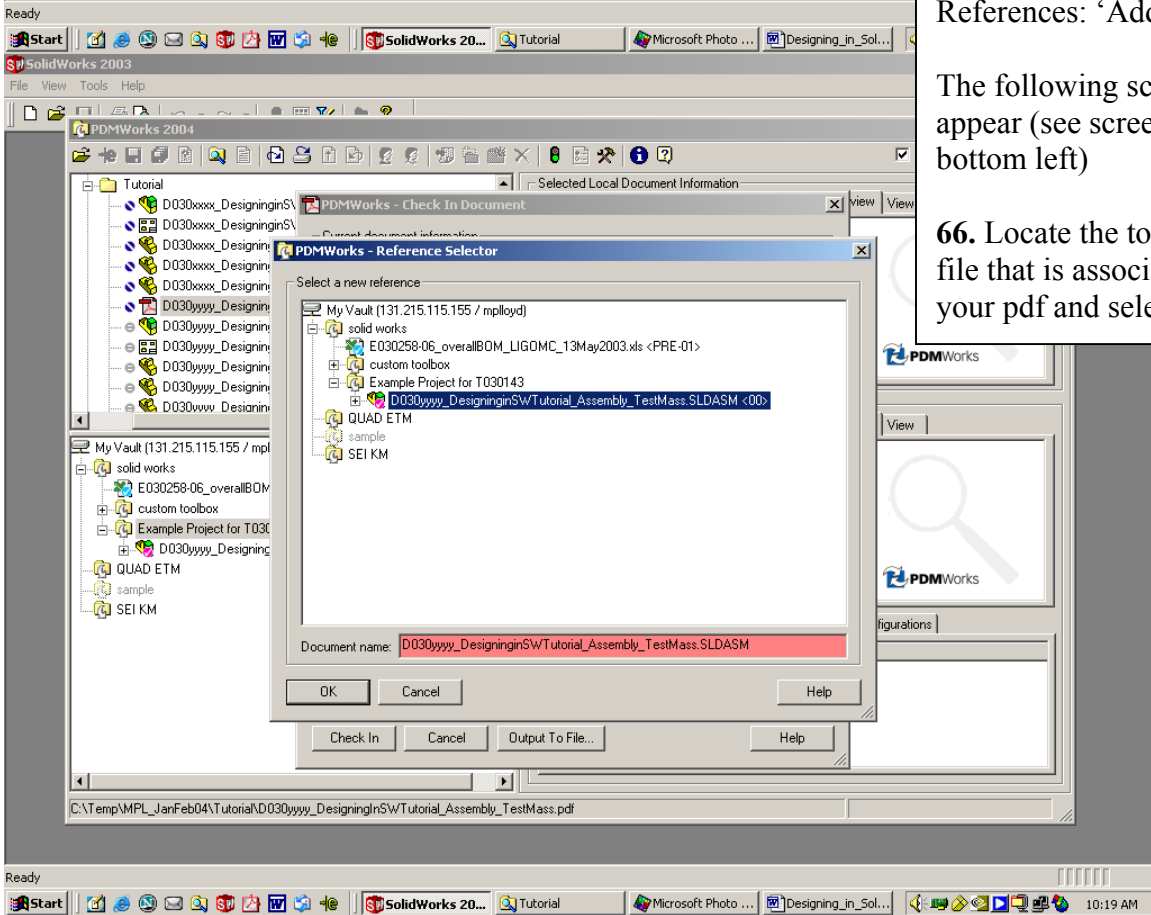**63.** Right click the appropriate file and select 'Check in this Document…' (see left)
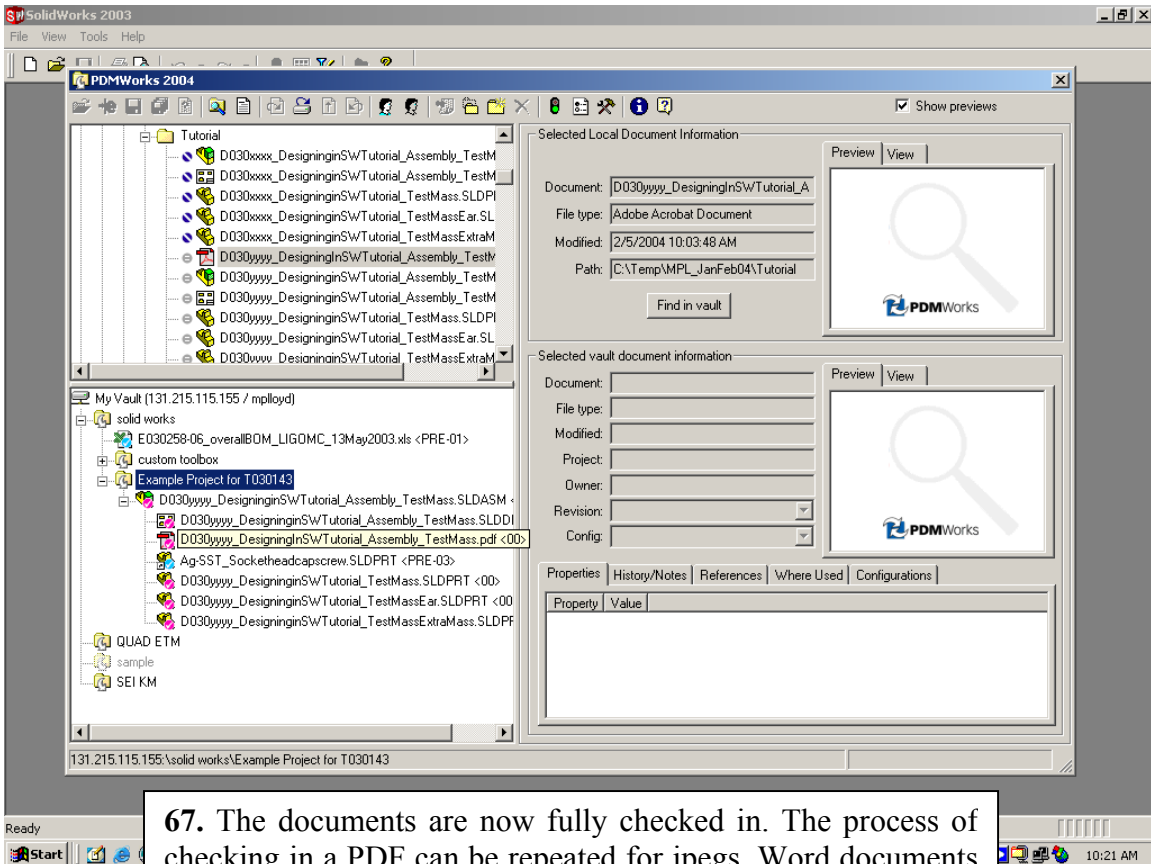
64. Select the correct 'Project' as shown top-left.

65. Next, click the box References: 'Add'

The following screen will appear (see screenshot bottom left)

66. Locate the top-level file that is associated with your pdf and select OK.

67. The documents are now fully checked in. The process of checking in a PDF can be repeated for jpegs, Word documents Excel spreadsheets, etc.
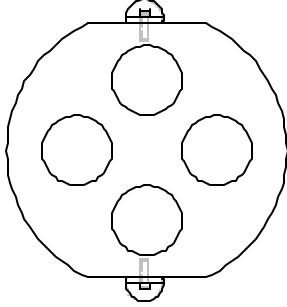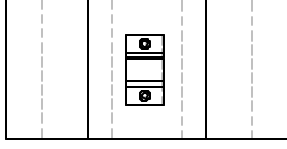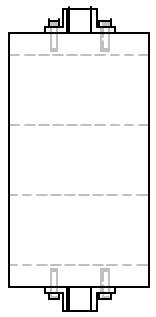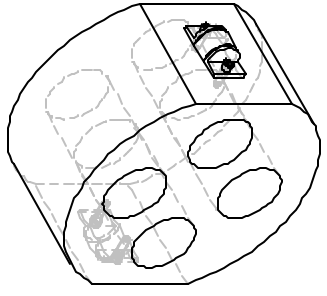
END.

See over for the completed engineering drawing of Assembly_Test_Mass.

DRAWING TREE #

REV. | DATE | DCN # | DRAWING TREE #

PARTS LIST

ITEM NO	REQ.	SPARE	TOT	PART NUMBER	DESCRIPTION	MATERIAL
	A/R				Aq-SST-SOCKET-HEAD-CAP-SCREW #4-40-UNC-3A-X-0.375-LONG	300-SSTL
	12			D030yyy	Test Mass Extra Mass	6061-T6-AL
				D030yyy	Test Mass Ear	300-SSTL
				D030yyy	Test Mass	300-SSTL

NOTES: (UNLESS OTHERWISE SPECIFIED)

1. REMOVE ALL SHARP EDGES,
   R 0.3 MAX.
2. DO NOT SCALE FROM DRAWING.
3. ALL MACHINING FLUIDS SHALL BE
   WATER SOLUBLE AND FREE OF
   SULFUR, CHLORINE AND SILICONE
   SUCH AS CINCINNATI MILACRON'S
   CIMTECH 410 (STAINLESS STEEL.)

SCRIBE, ENGRAVE OR STAMP DRAWING
PART NUMBER ON NOTED SURFACE OF
PART AND A THREE DIGIT SERIAL
NUMBER. SERIAL NUMBERS START
AT 001 FOR THE FIRST PART AND
PROCEED CONSECUTIVELY.
USE 07" HIGH CHARACTERS.
EXAMPLE D020186-001, A VIBRATORY
TOOL MAY BE USED.

DIMENSIONS ARE IN INCHES

TOLERANCES:
XX ± 0.01
XXX ± 0.005
ANGULAR ± 0.5°

| MATERIAL | -- |
| FINISH | -- |

	NAME	DATE
DRAWN	MPL (IGR)	23/JUL/03
CHECKED	Cit	23/JUL/03
APPROVED		

CALIFORNIA INSTITUTE OF TECHNOLOGY
MASSACHUSETTS INSTITUTE OF TECHNOLOGY

ADVANCED LIGO

SYSTEM	SUS
SUB-SYSTEM	SW Tutorial
NEXT ASSY	
PART NAME	Assemby Test Mass

SIZE	DWG. NO.	REV.
B	D030yyy	01

SCALE: 12 | PROJECTION: | SHEET 1 OF 1