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DESIGNING IN SOLIDWORKS

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

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DESIGNING IN SOLIDWORKS

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1. INTRODUCTION

This document outlines how to attach information 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[1] 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[2] 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 will make time to focus on 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.

[1] E030350-00-D Mechanical Drawing Guidelines, http://www.ligo.caltech.edu/docs/E/E030350-00.pdf, D Coyne
[2] D030384-07_LIGOBOM.xls, http://www.ligo.caltech.edu/docs/D/D030384-07_LIGOBOM.xls, M Perreur-Lloyd and C Torrie

2. PART PROPERTIES FOR THE BOM

ITEM NO	REQ.	SPARE	TOT.	PART NUMBER	DESCRIPTION	MATERIAL

2.1. Item Number Column

2.1.1. 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.

2.1.2. 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).

2.2. 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).

2.3. 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.

2.4. 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.

2.5. 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

2.6. 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 Custom Properties:

e.g. D020123: Enter 'number' into the custom properties, followed by the part or assembly's DCC number.

2.7. 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.

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

To create a Bill of Materials, Drawing Tree or Materials List by using the LIGOBOM, 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:

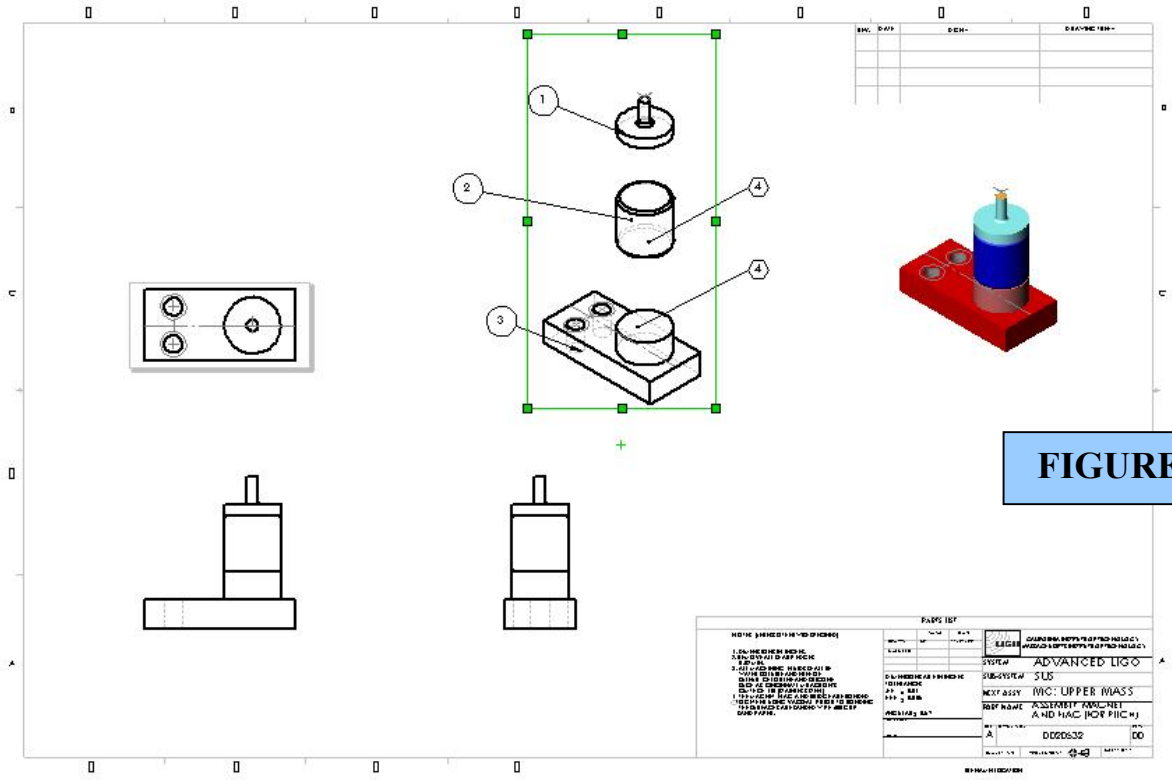


FIGURE 1

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. At this point it is required that the file: D030384-07_LIGOBOM.xls is located.

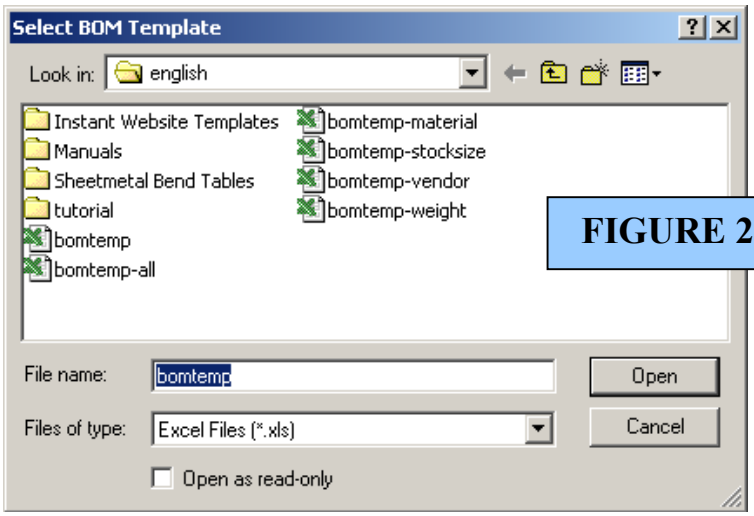


FIGURE 2

Once the D030384-07_LIGOBOM.xls is located, the following window shall open up on your screen:

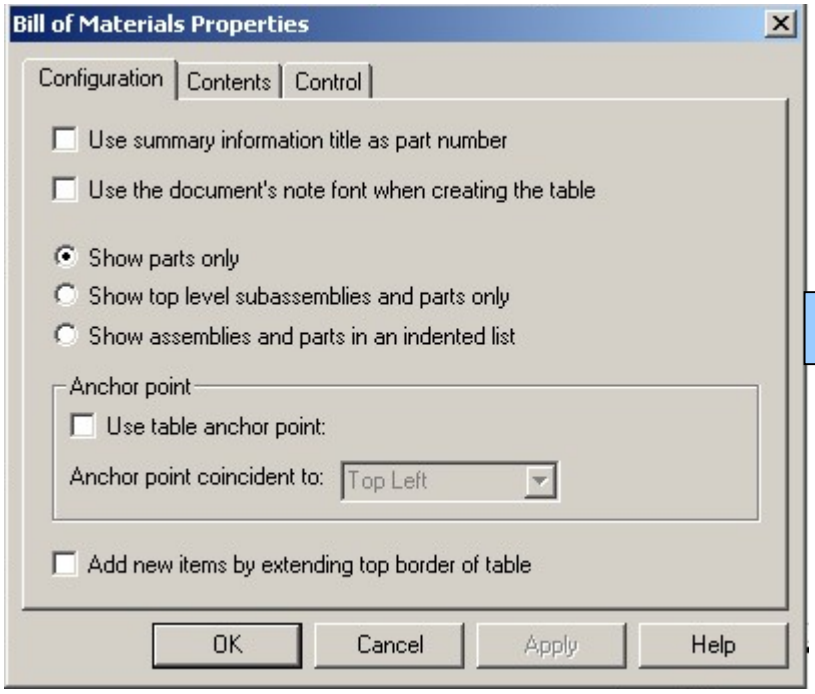


FIGURE 3

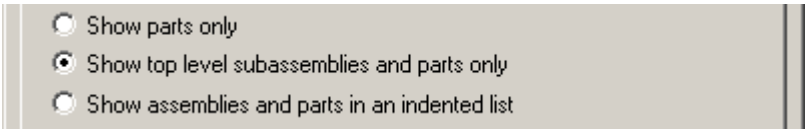
And it is by using this interface that the selections can be made to tabulate a BOM, Drawing Tree or Materials List.

3.1. 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 table 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 left unchecked

3.2. Creating a Bill of Materials

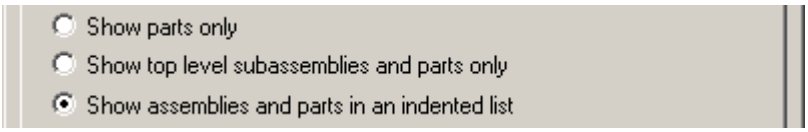
To create a bill of materials, the box that should be checked on the 'Configuration' area of the 'Bill of Materials Properties' is:



And on the 'Contents' area of the 'Bill of Materials Properties' uncheck 'Display labels at top'

3.3. Creating a Indented Drawing Tree

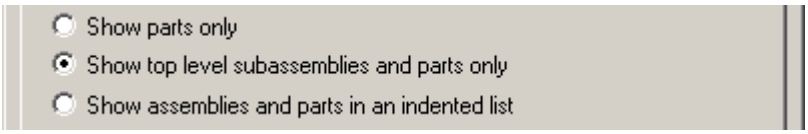
To create an Indented Drawing Tree, the important boxes that should be checked are:



3.4. 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:



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

3.5. Finishing off

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)

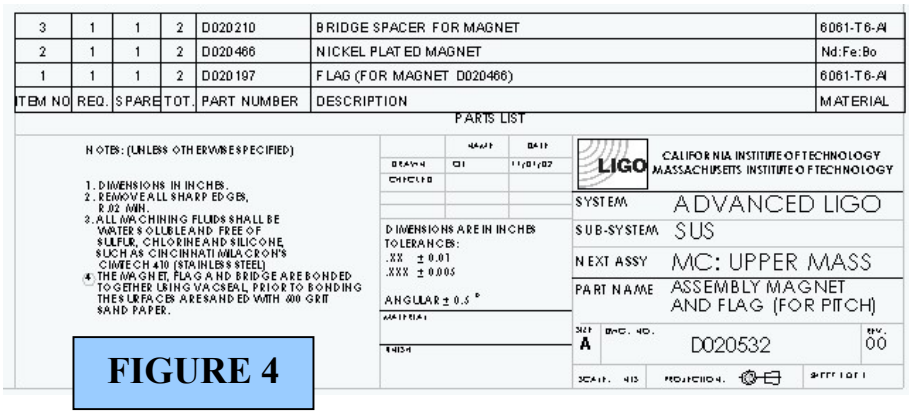


FIGURE 4

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.

4. 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.

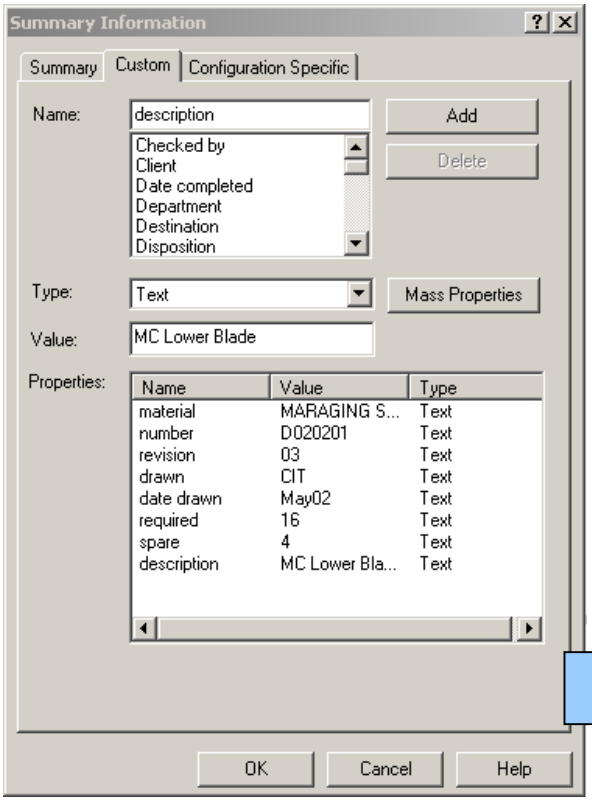


FIGURE 5

4.1. 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.

4.2. 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

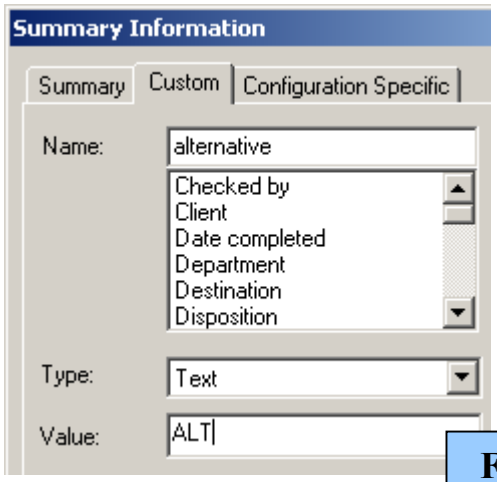
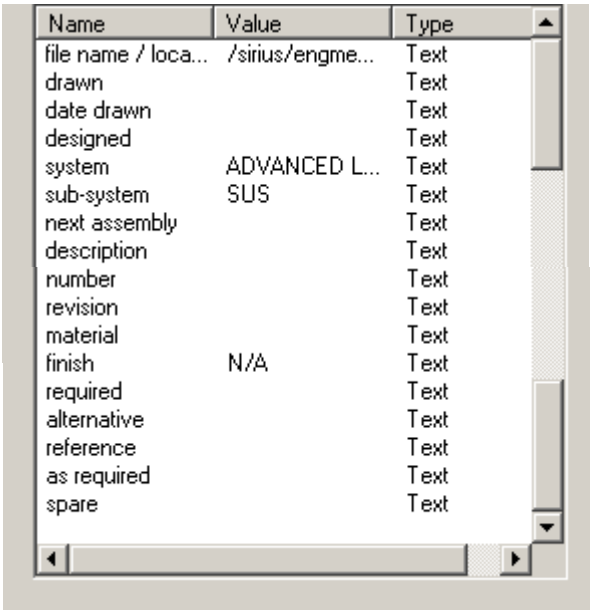


FIGURE 6

4.3. 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 5. Smart CAD and Data Templates).



Name	Value	Type
file name / loca...	/sirius/engme...	Text
drawn		Text
date drawn		Text
designed		Text
system	ADVANCED L...	Text
sub-system	SUS	Text
next assembly		Text
description		Text
number		Text
revision		Text
material		Text
finish	N/A	Text
required		Text
alternative		Text
reference		Text
as required		Text
spare		Text

FIGURE 7

5. SMART CAD AND DATA TEMPLATES.

A series of 'smart' CAD and Data templates 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.

5.1. 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:

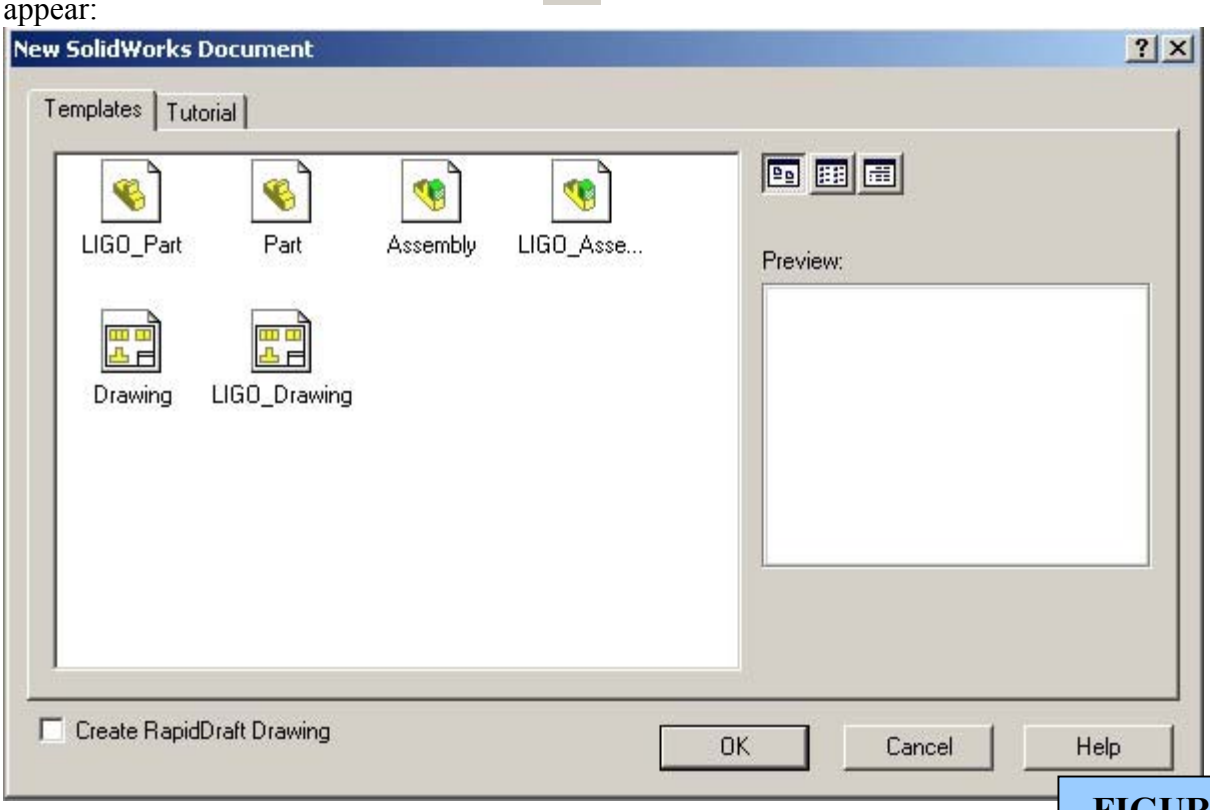


FIGURE 8

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.

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:

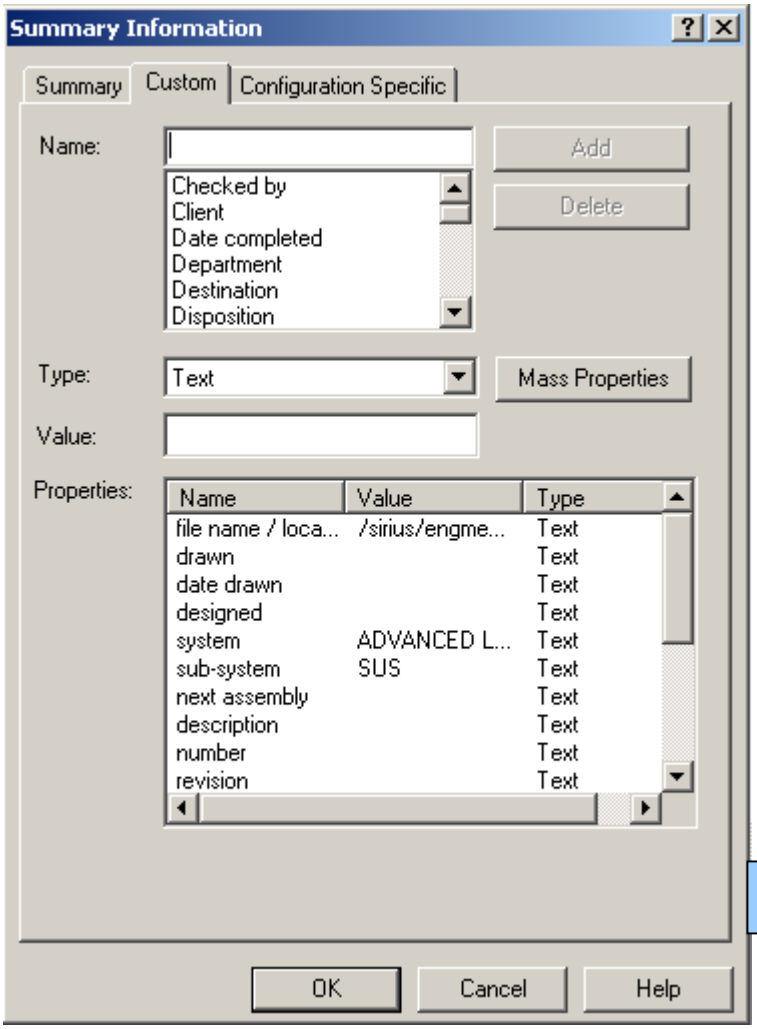


FIGURE 9

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.

5.2. 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 (D030382) are links to the Assembly, Drawing Name, Material, Finish, Drawn by, etc (detailed in section 4. 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.

6. SETTING OTHER PART PROPERTIES

When designing suspensions it is important that we can predict accurate mass and moments 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 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

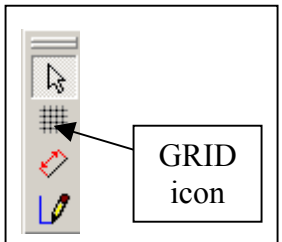


FIGURE 10

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).

Alu	0.0027 grams/mm^3
Stainless Steel	0.0078 grams/mm^3
Titanium	0.0045 grams/mm^3
PEEK	0.00132 grams/mm^3

Alu	0.09754 lb/in^3
Stainless Steel	0.28179 lb/in^3
Titanium	0.16257 lb/in^3

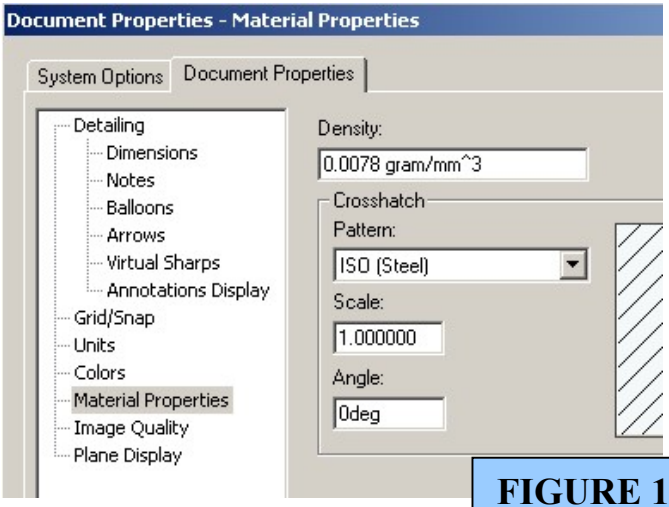


FIGURE 11

7. CUSTOMISED TOOLBOX.

The customised toolbox (http://www.ligo.caltech.edu/docs/D/D030383-03) was created following problems found with the SolidWorks Toolbox when passing assemblies between the engineers in the US and the UK. Following it's creation, the customised toolbox has since been expanded to include a full range of nuts, bolts and washers incorporated in LIGO Suspension design further automating the design process.

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 reason for undertaking the exercise of creating this new toolbox as opposed to continuing to work with the SolidWorks version is because assemblies that use the SW toolbox 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 already to be found to have problems 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!

To ensure these problems do not occur again, 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. The Customised toolbox also has the luxury that the bolts are very easy to change once inserted in to an assembly such that, if a hole size increased or decreased a change is simply a right click away. Figure 12 shows how a bolt can be changed:

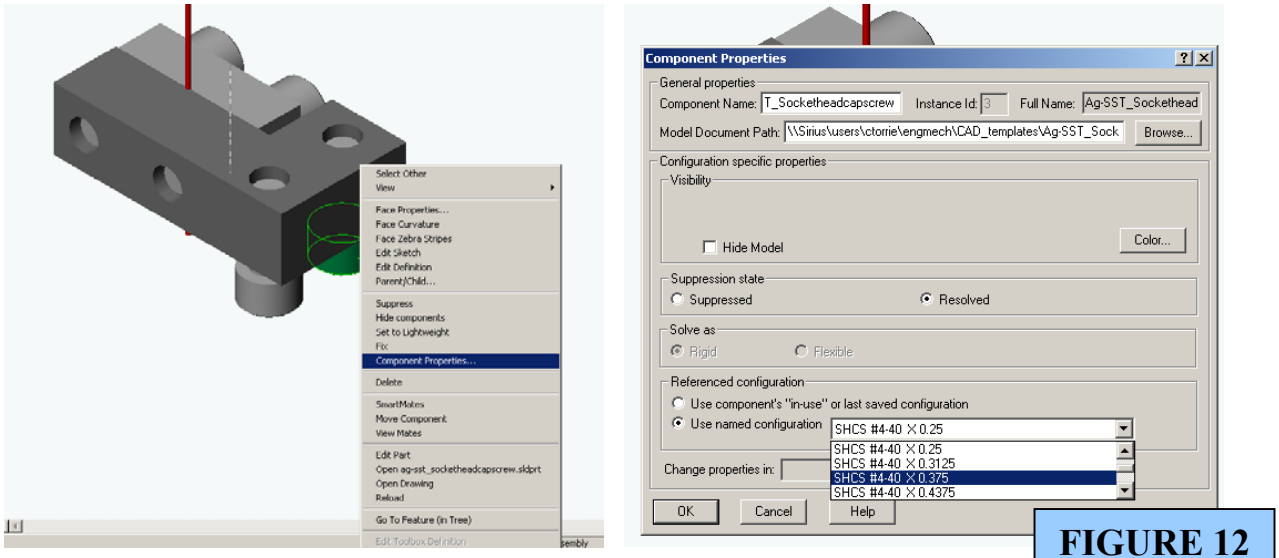


FIGURE 12

The Customised toolbox can be found on the DCC.

8. SETTING UP A WORKSTATION WITH LIGO CUSTOMISED TOOLS

All aforementioned tools and customised SolidWorks functions have been allocated LIGO DCC numbers and can be found in this Database. All of the tools should be download to the appropriate location at the same time and should ideally be used at all times during the design of parts for the Advanced LIGO project. All of these tools have been saved to the DCC as .zip to make downloading of the files very straightforward.

PDF Documents have been created alongside the tools zip files giving instruction of where the tools should be downloaded to and details of what tools are included in the zip files. The DCC numbers are as follows:

D030382-03_Customised Smart CAD and DATA Templates for SolidWorks.zip
D030382-03_Customised Smart CAD and DATA Templates for SolidWorks.pdf
D030383-03_Customised Toolbox for SolidWorks.zip
D030383-03_Customised Toolbox for SolidWorks.pdf
D030384-07_LIGOBOM.xls

In the case of the Bill of Materials, D030384-07_LIGOBOM.xls, it is advisable for ease of use that this should be downloaded to:

//Program Files/Solidworks/lang/english/

as this is where all of the other SolidWorks Bill of Materials templates are stored.

Note: Older revisions of the Customised tools are also located on the DCC and may be found during a DCC search. Where this is the case, locate and download the latest revisions of these. The latest revisions at time of press are those seen above.

9. 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 however a balance must be made on how time consuming this is above the future benefits of doing this.

The improvements in the document are comprehensive but hopefully, once setup, are straight forward and well integrated. It is important to note that they have 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.

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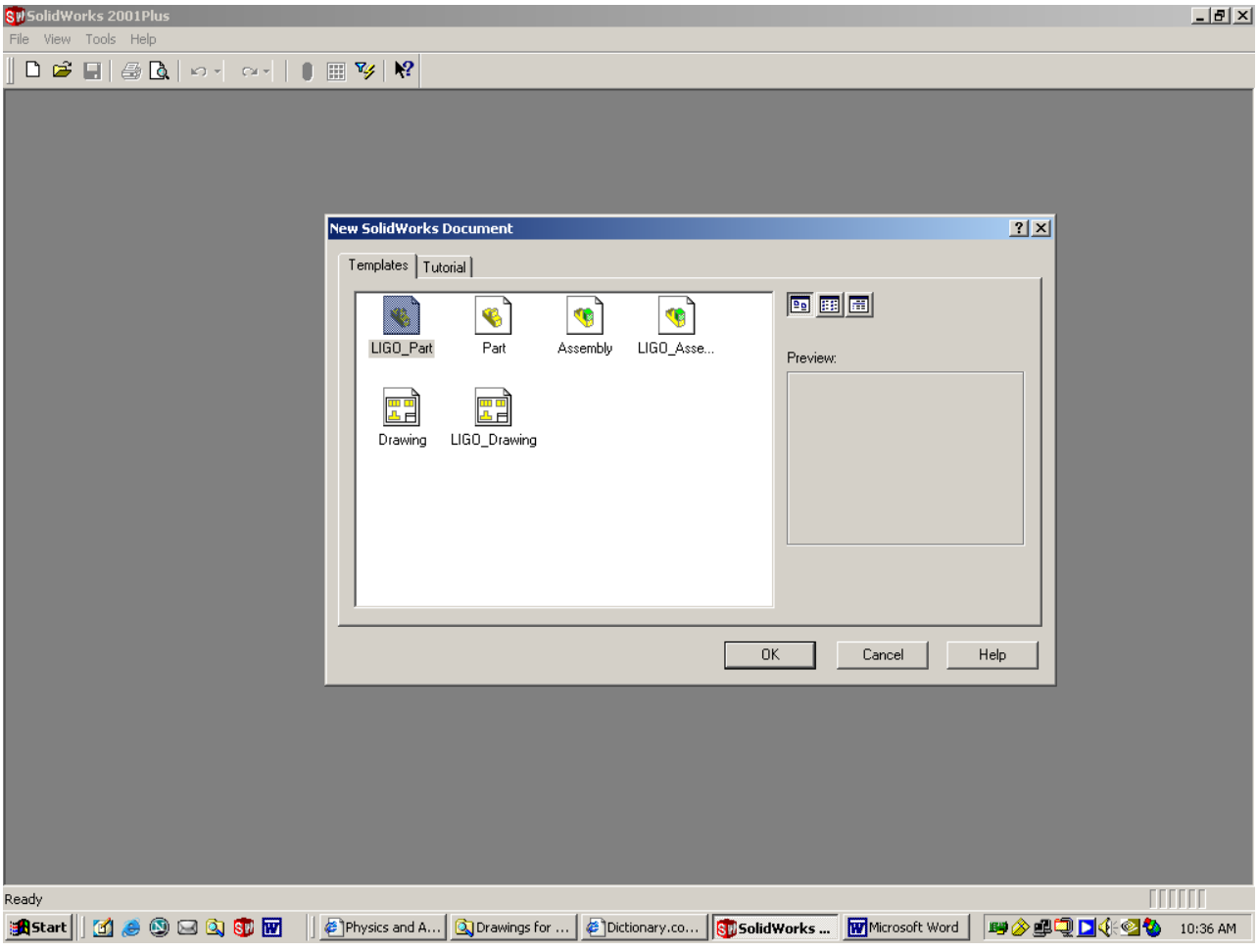
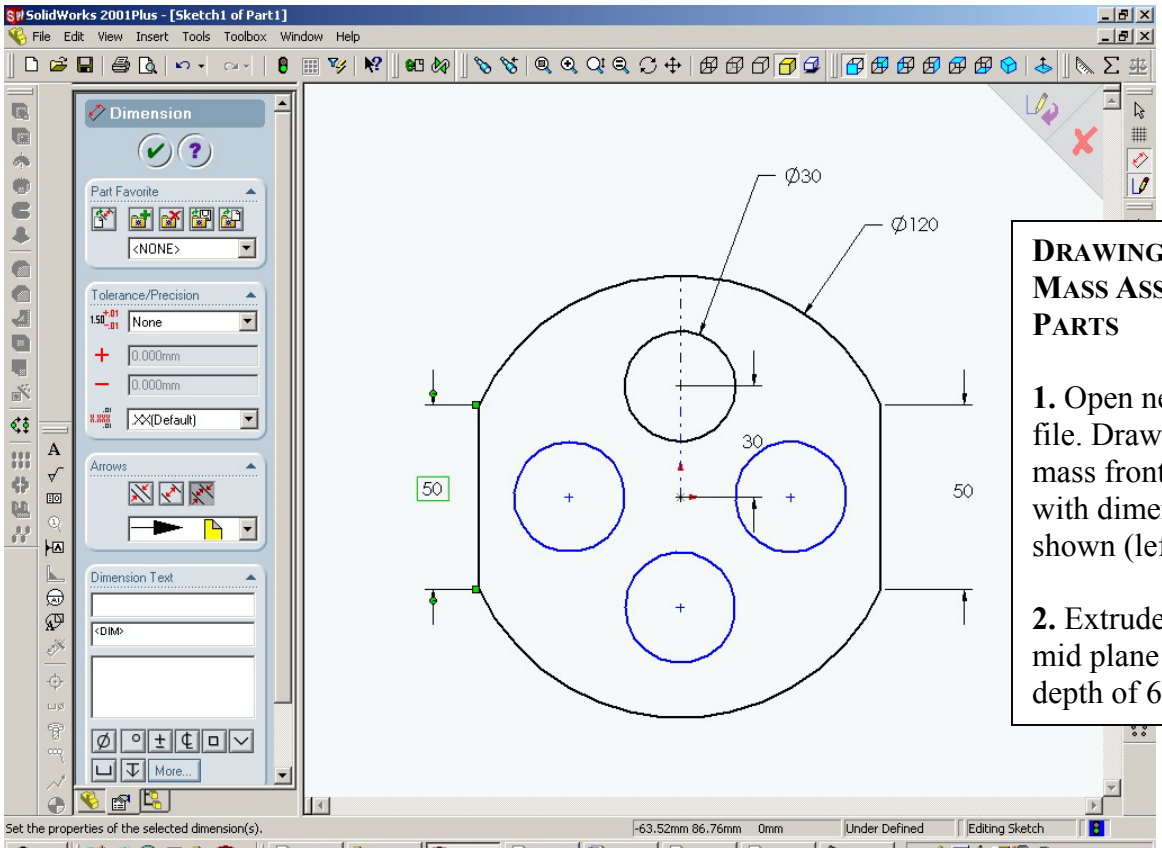
APPENDIX 1

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_07_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.

To begin, open a new LIGO_Part:

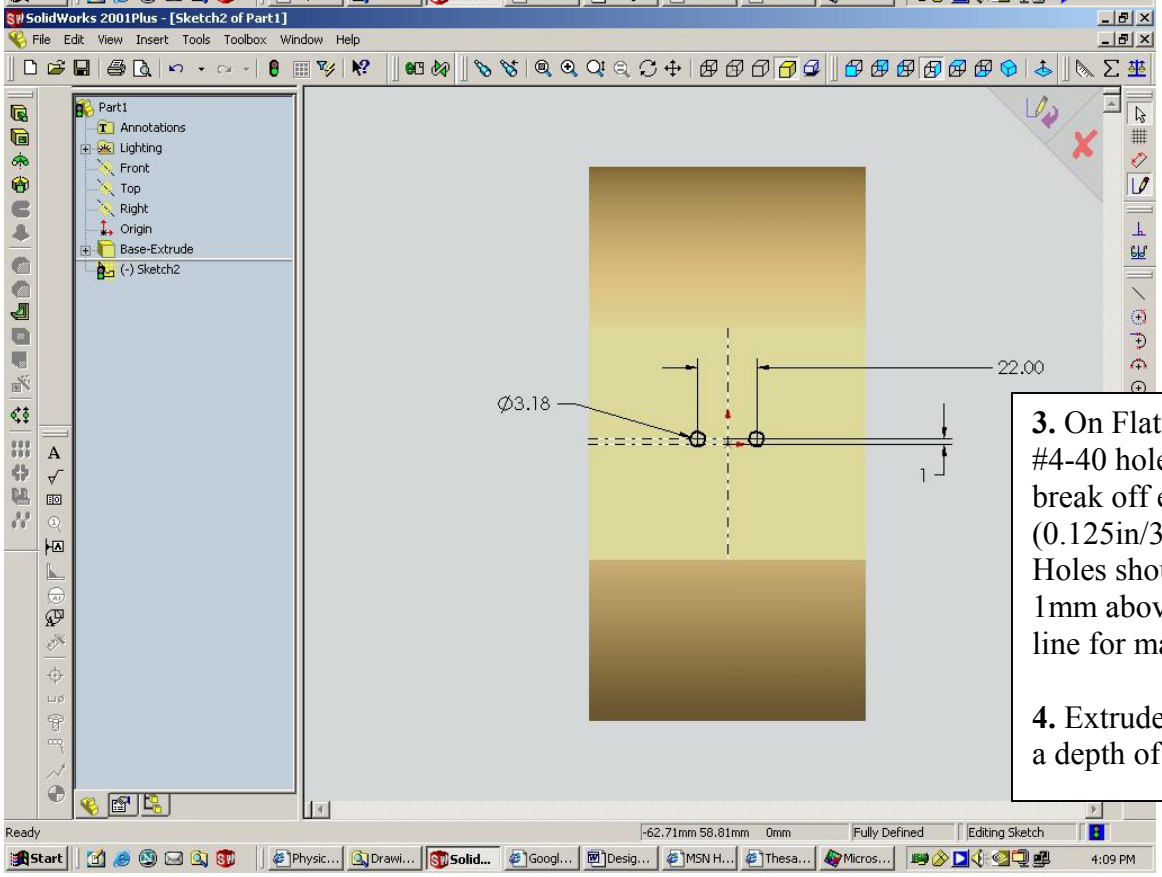




DRAWING TEST MASS ASSEMBLY PARTS

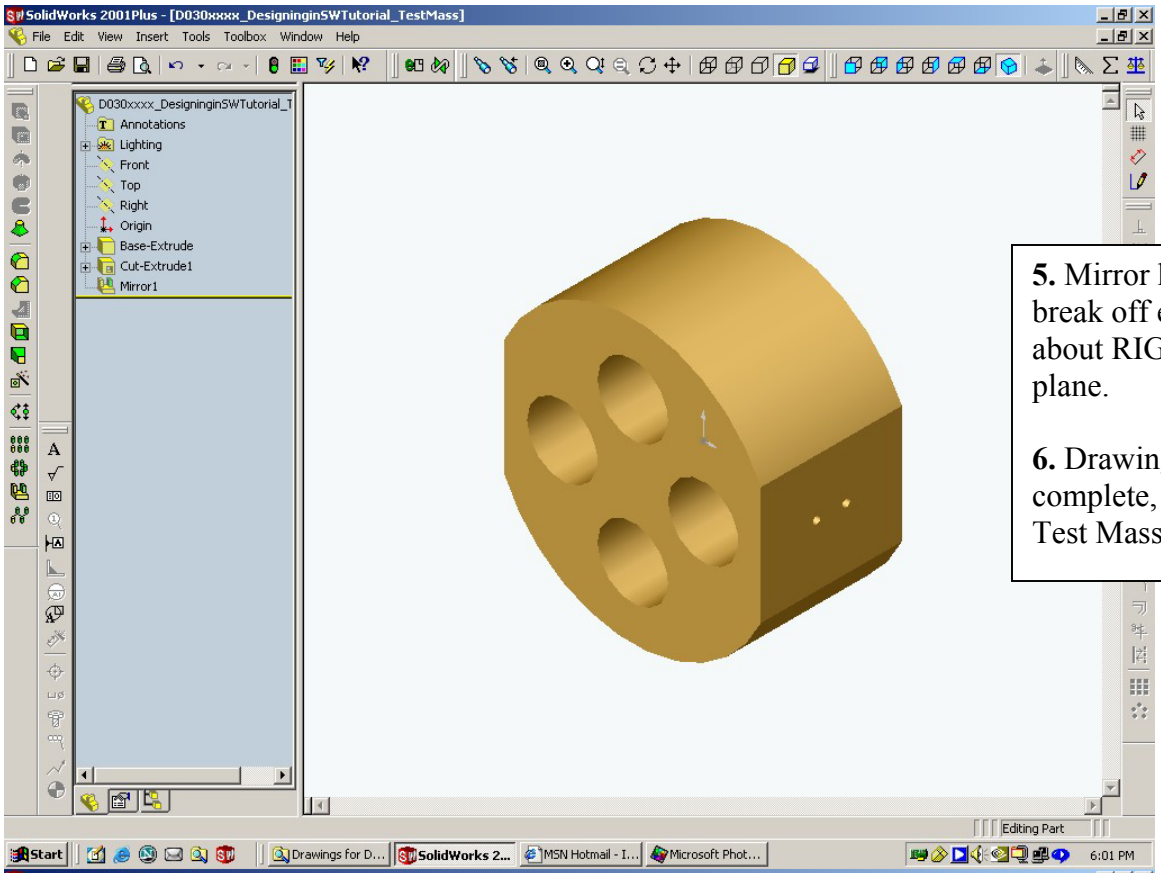
1. Open new part file. Draw up test mass front view with dimensions as shown (left)

2. Extrude-Boss, mid plane to a depth of 60mm



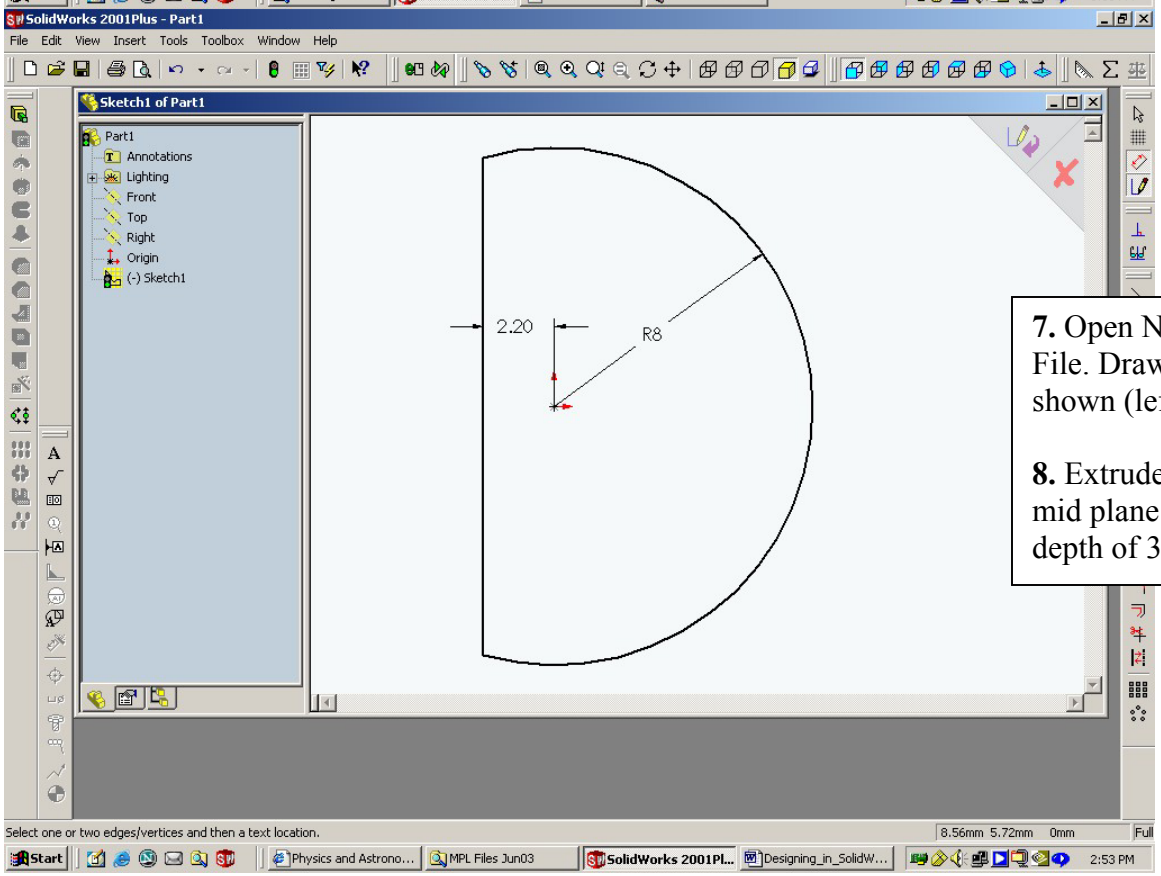
3. On Flat, Draw #4-40 holes for break off ears (0.125in/3.18mm). Holes should be 1mm above centre line for mass.

4. Extrude-Cut, to a depth of 8mm.



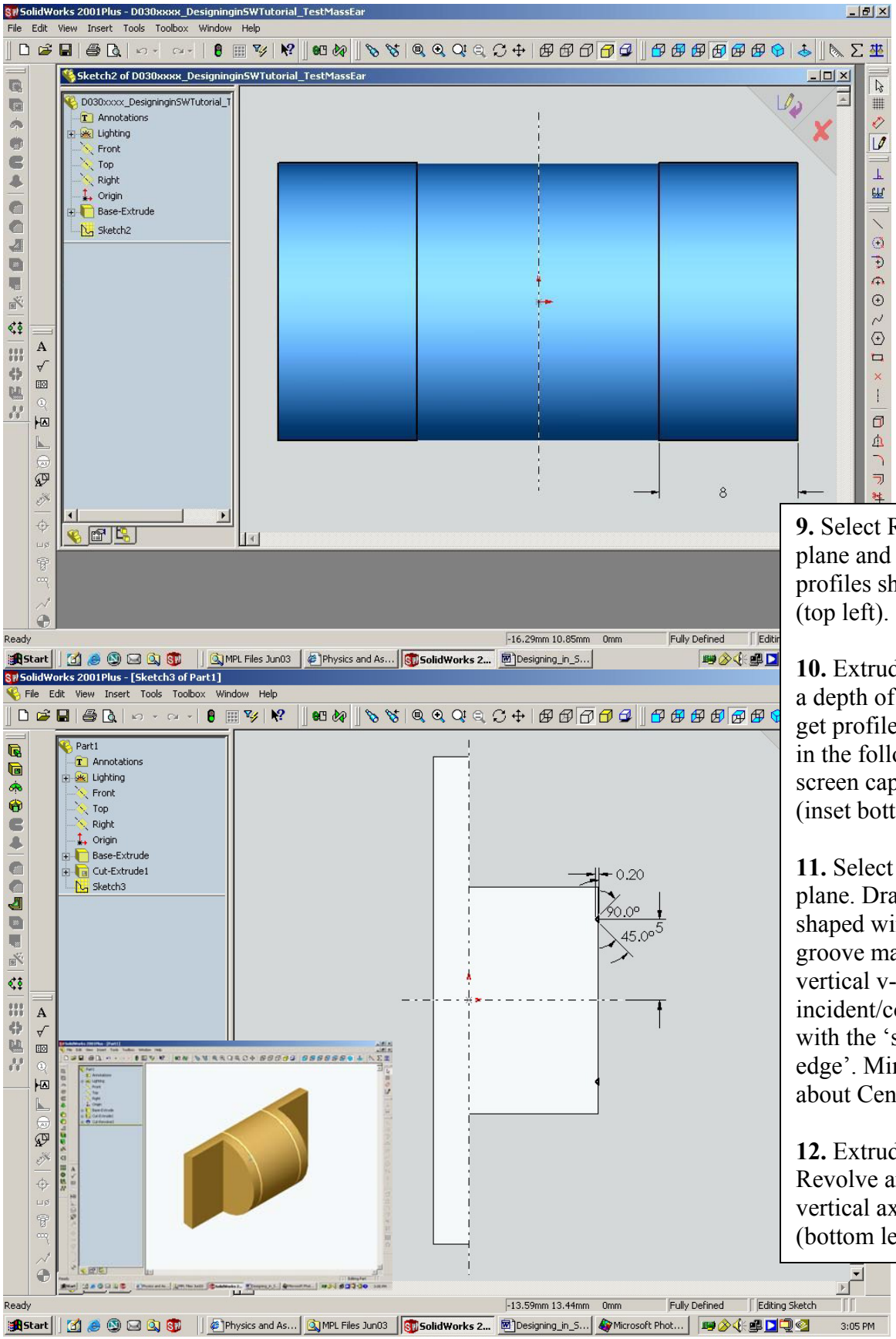
5. Mirror holes to break off ears about RIGHT plane.

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.

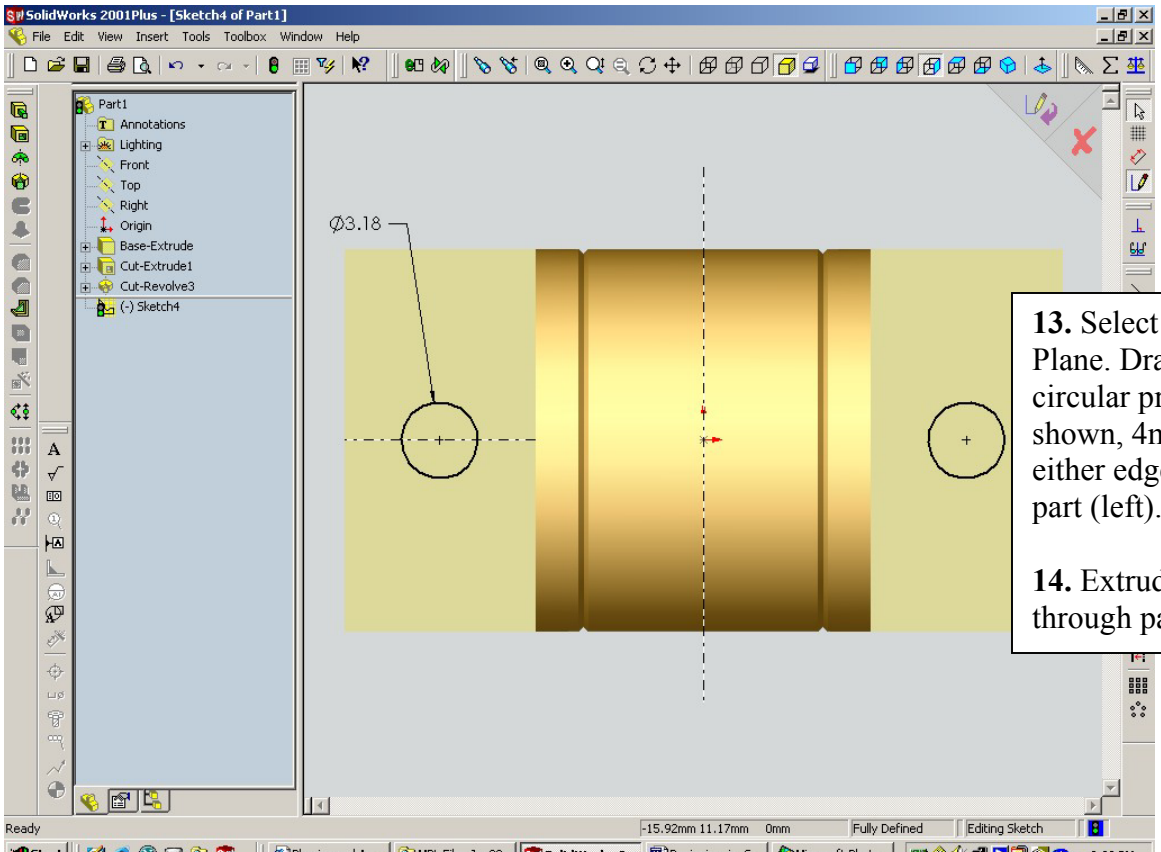


9. Select RIGHT plane and draw the profiles shown (top left).

10. Extrude-Cut, to a depth of 8mm to get profile shown in the following screen capture (inset bottom left).

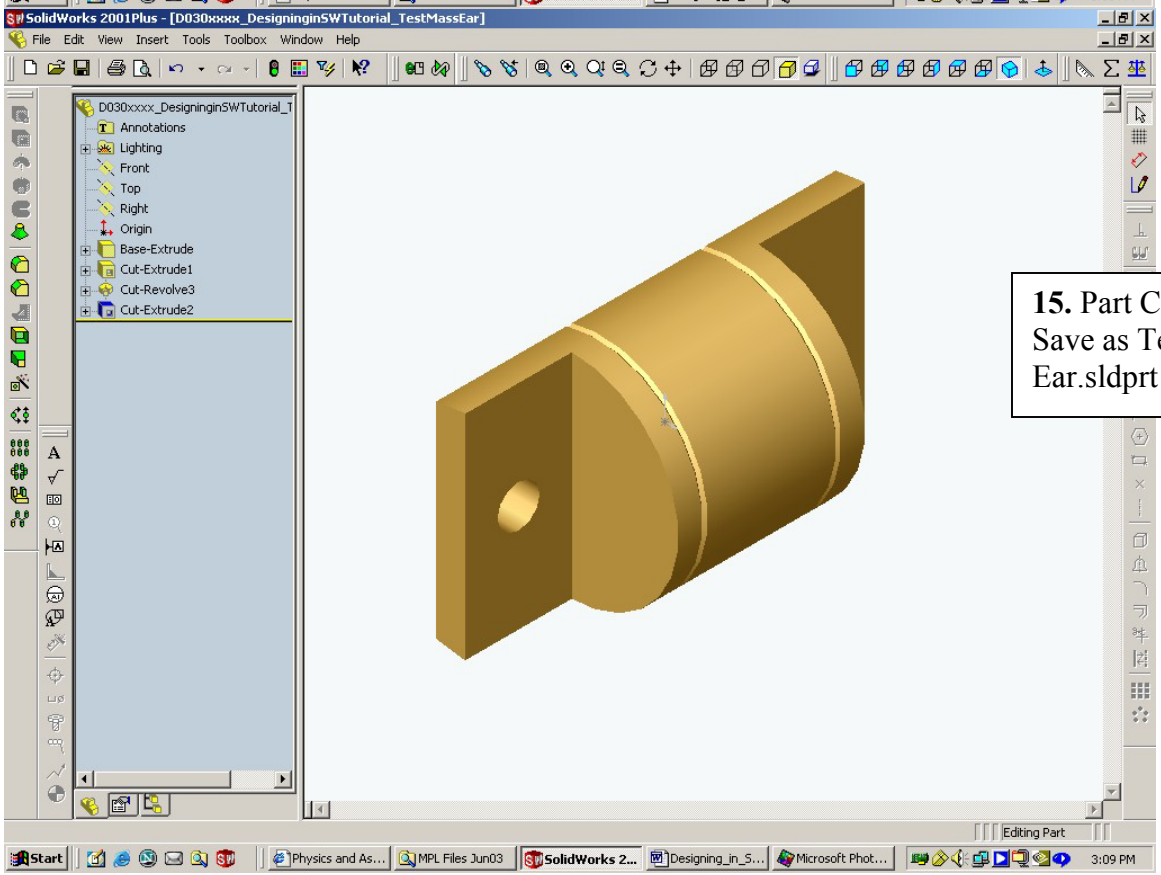
11. Select TOP plane. Draw V-shaped wire groove making the vertical v-line co-incident/co-linear with the 'silhouette edge'. Mirror about Centreline.

12. Extrude-Cut-Revolve around vertical axis shown (bottom left)

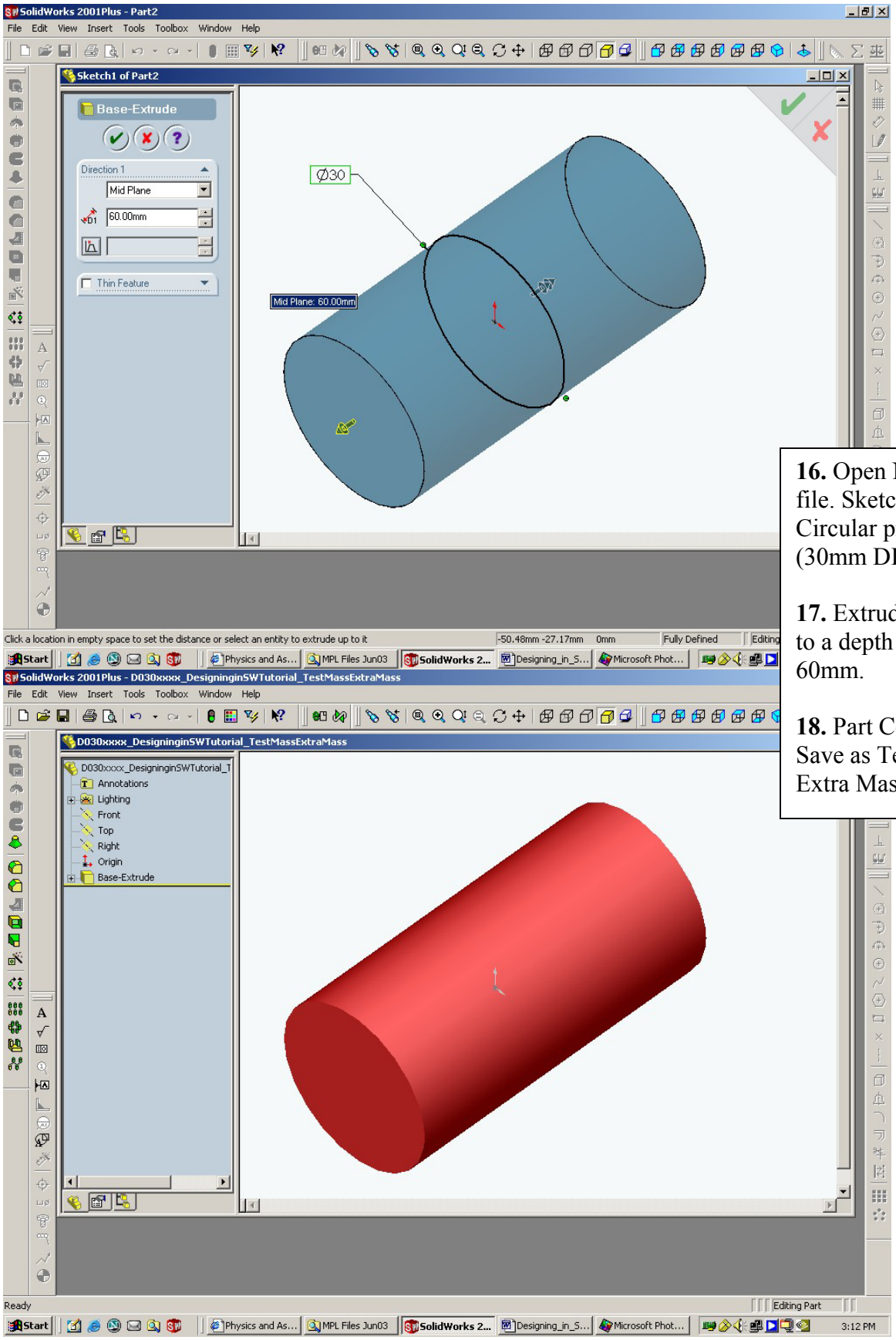


13. Select RIGHT Plane. Draw circular profiles shown, 4mm from either edge of the part (left).

14. Extrude-Cut, through part.



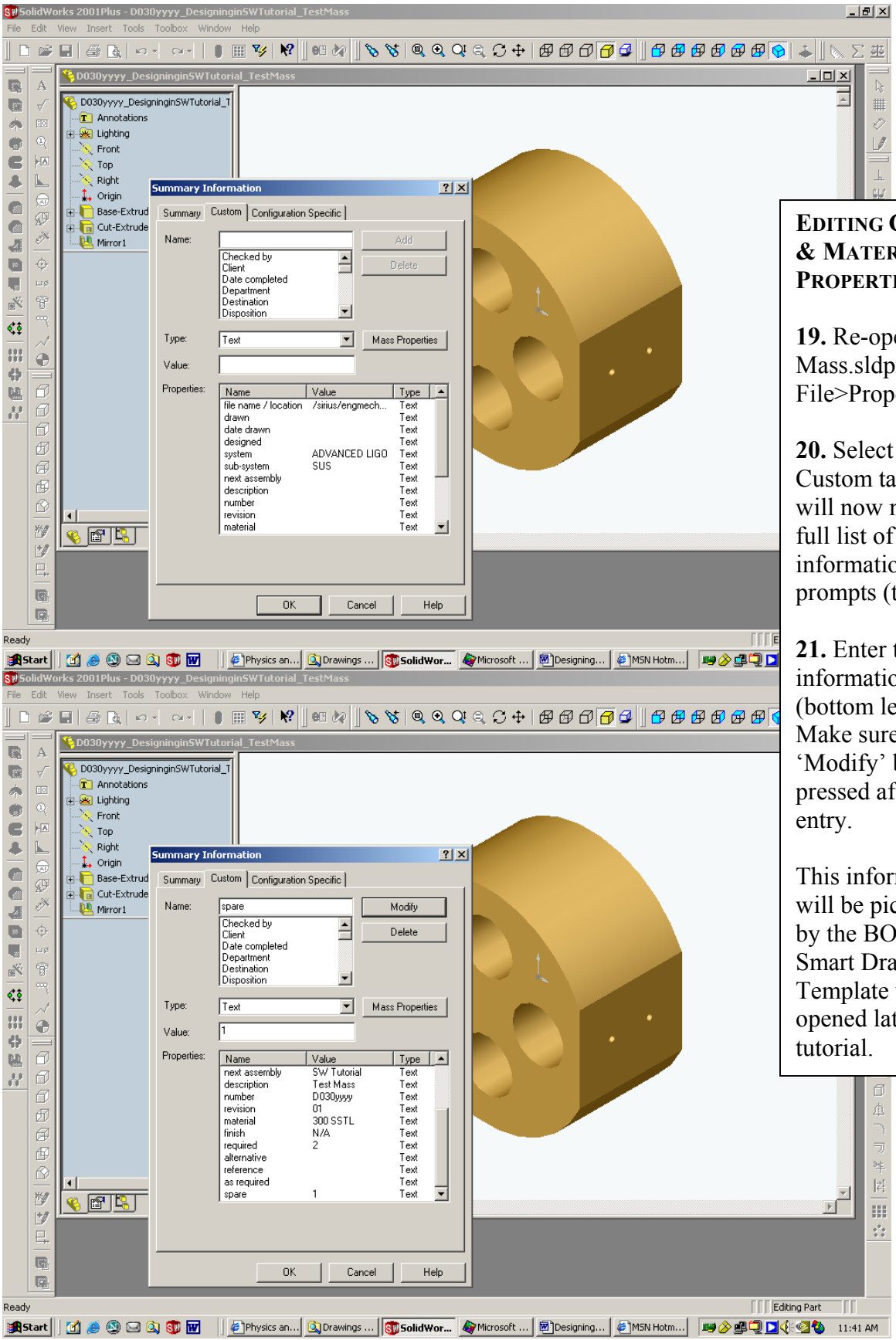
15. Part Complete. Save as Test Mass Ear.sldprt



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



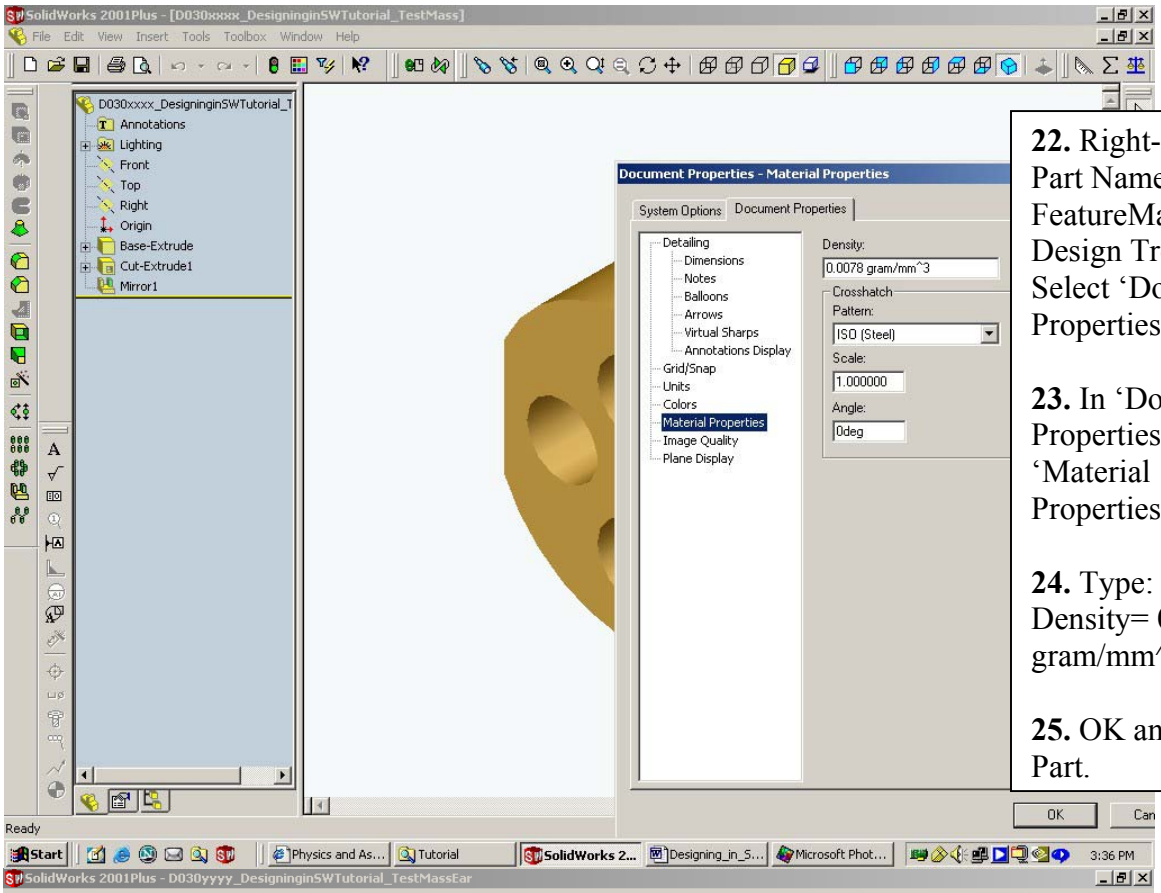
EDITING CUSTOM & MATERIAL PROPERTIES

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.

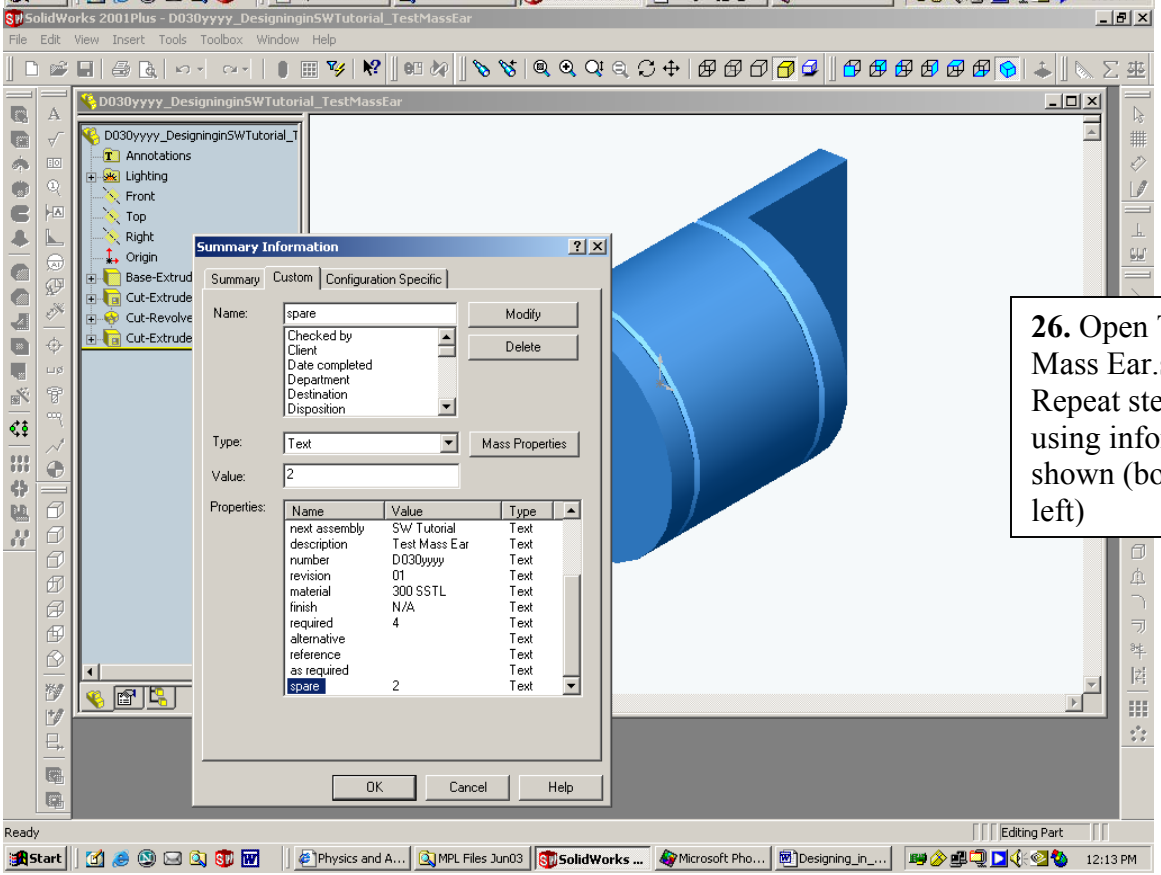


22. Right-click Part Name in FeatureManager Design Tree. Select 'Document Properties'.

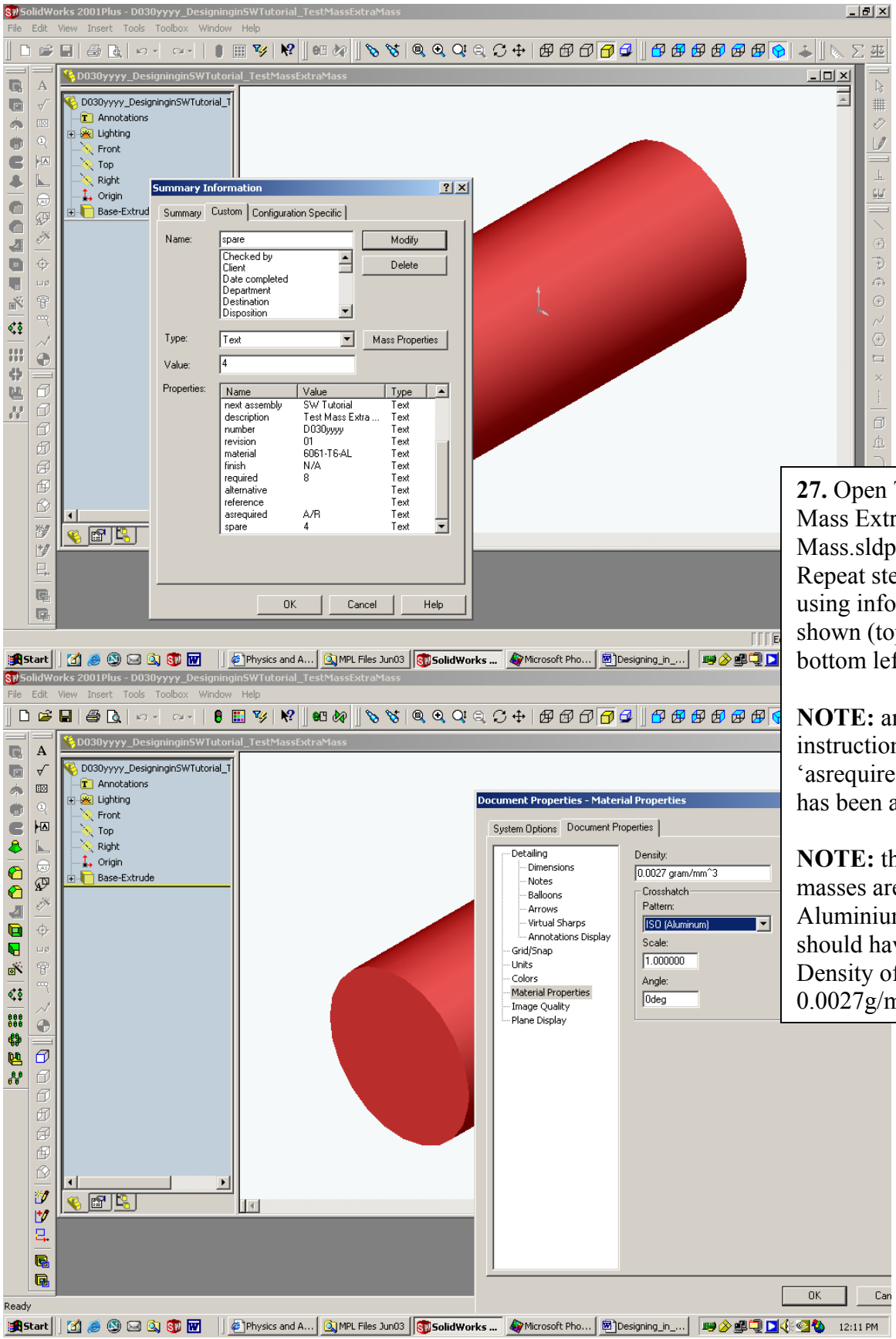
23. In 'Document Properties', select 'Material Properties'.

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

25. OK and save Part.



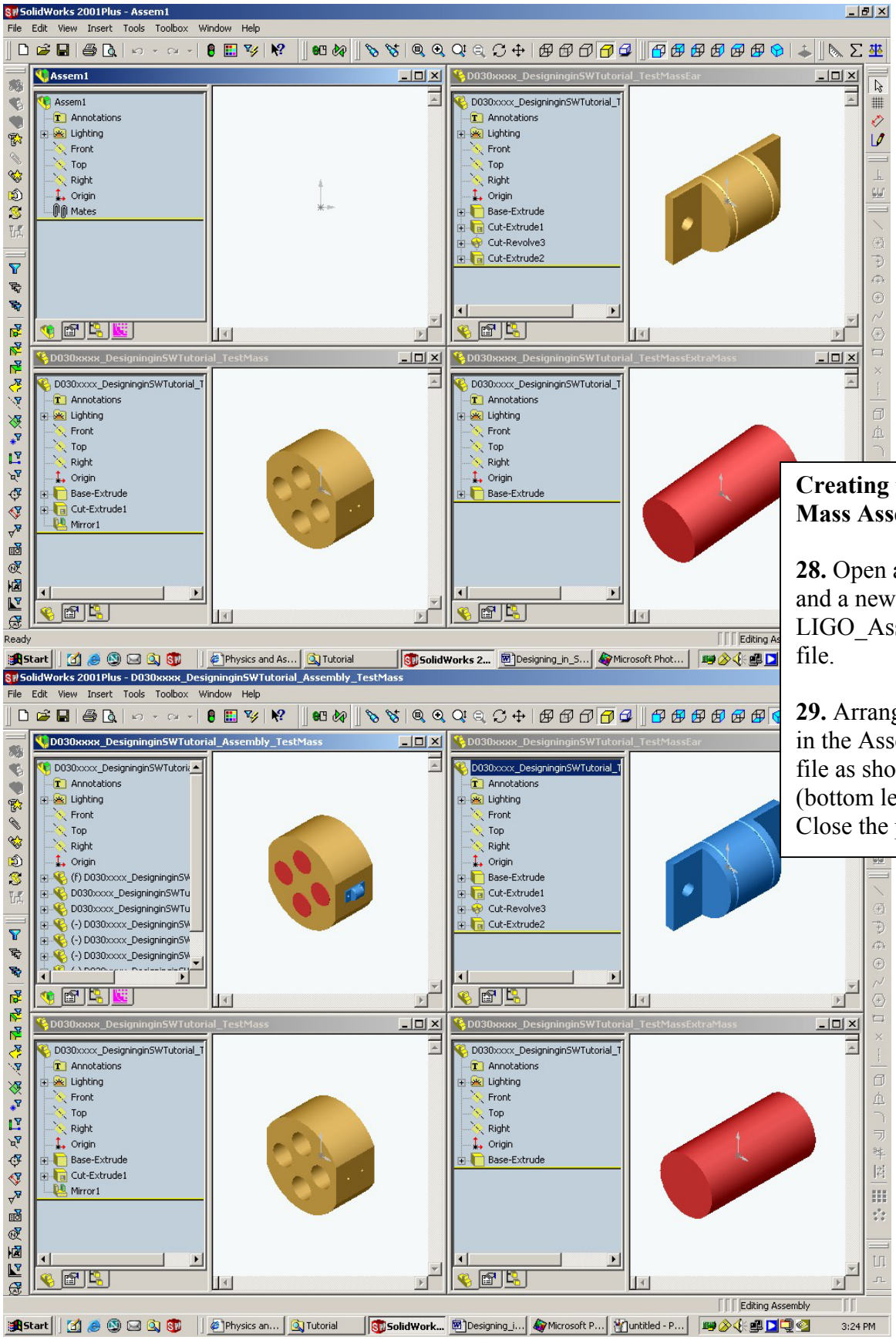
26. Open Test Mass Ear.sldprt. Repeat steps 19-25 using information shown (bottom left)



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.

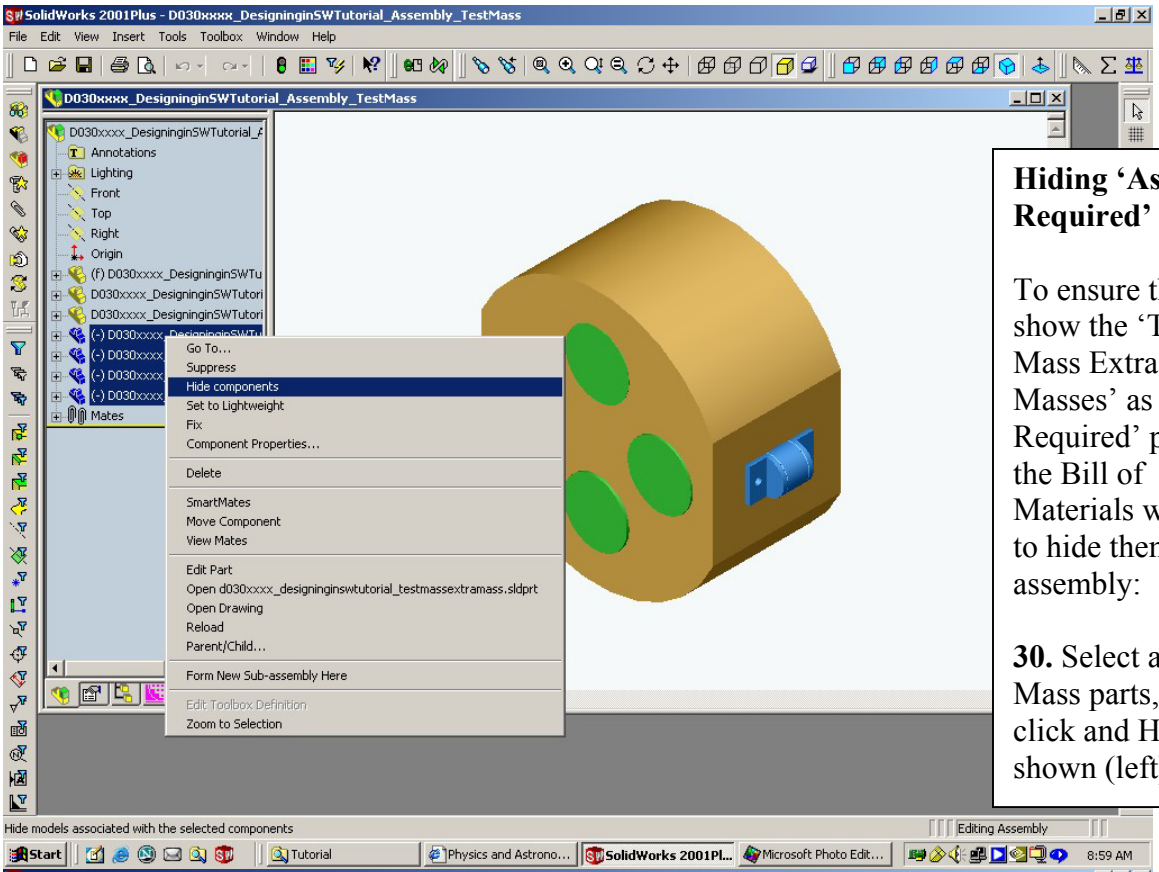
NOTE: the extra masses are Aluminium and should have a Density of 0.0027g/mm^3.



Creating the Test Mass Assembly

28. Open all parts and a new LIGO_Assembly file.

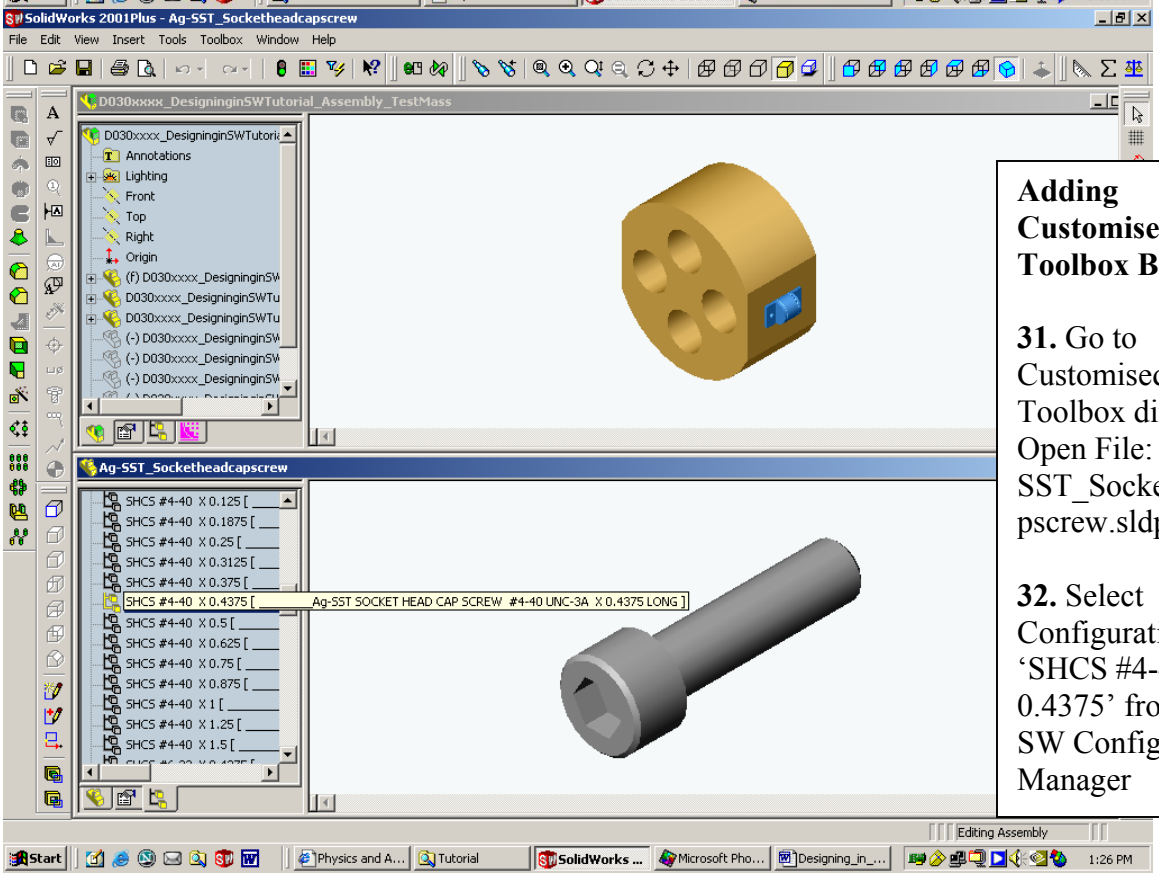
29. Arrange Parts in the Assembly file as shown (bottom left). Close the part files



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:

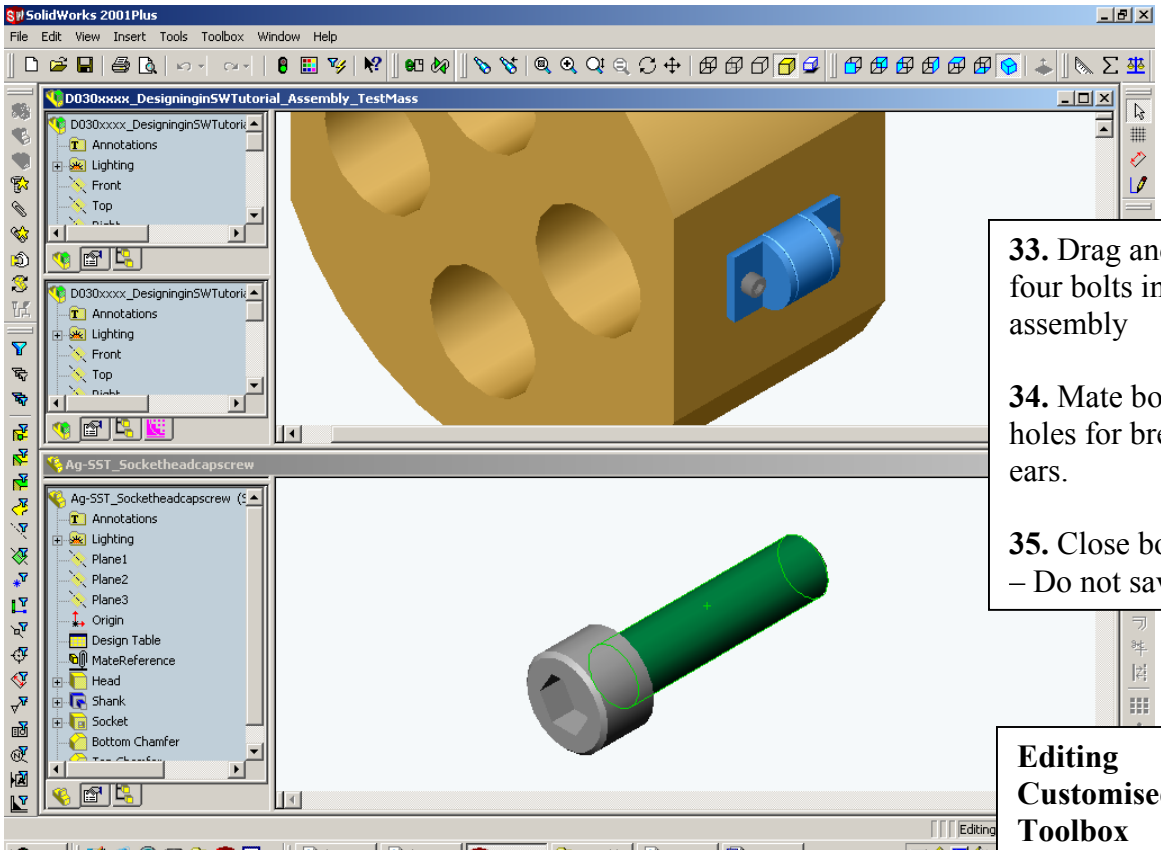
30. Select all Extra Mass parts, right click and Hide as shown (left).



Adding Customised Toolbox Bolts

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

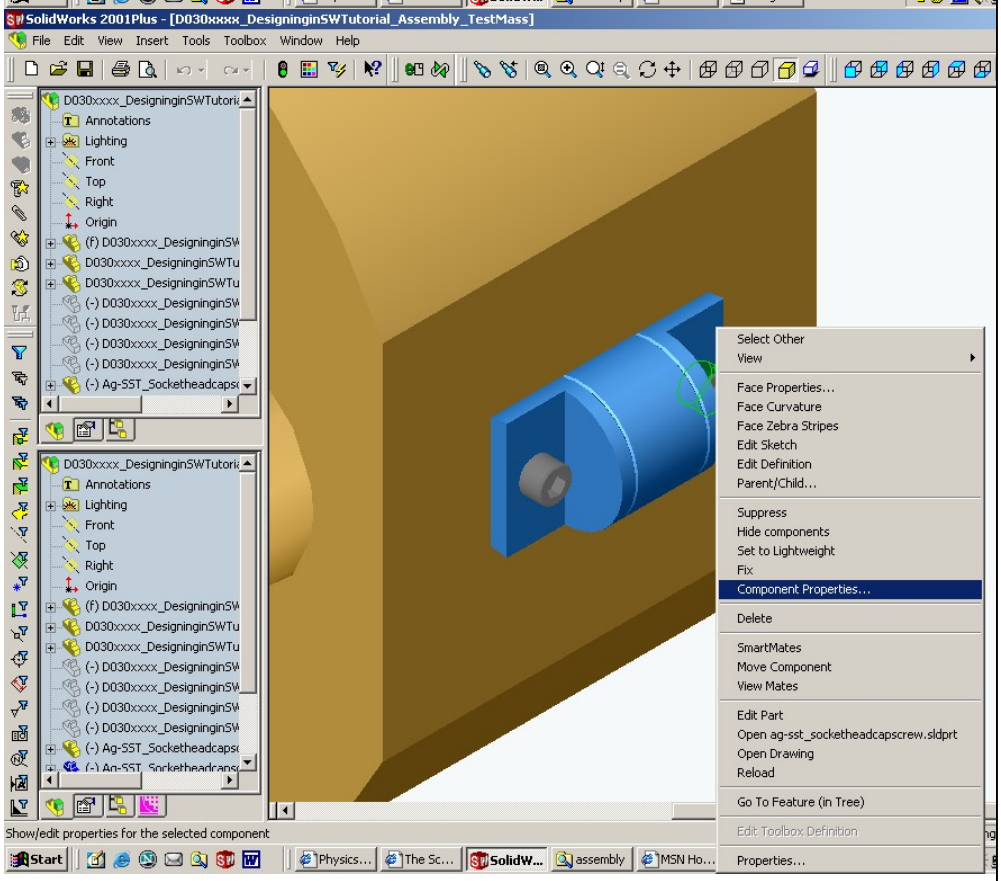
32. Select Configuration 'SHCS #4-40 X 0.4375' from the SW Configuration-Manager



33. Drag and drop four bolts into assembly

34. Mate bolts into holes for break off ears.

35. Close bolt file – Do not save.

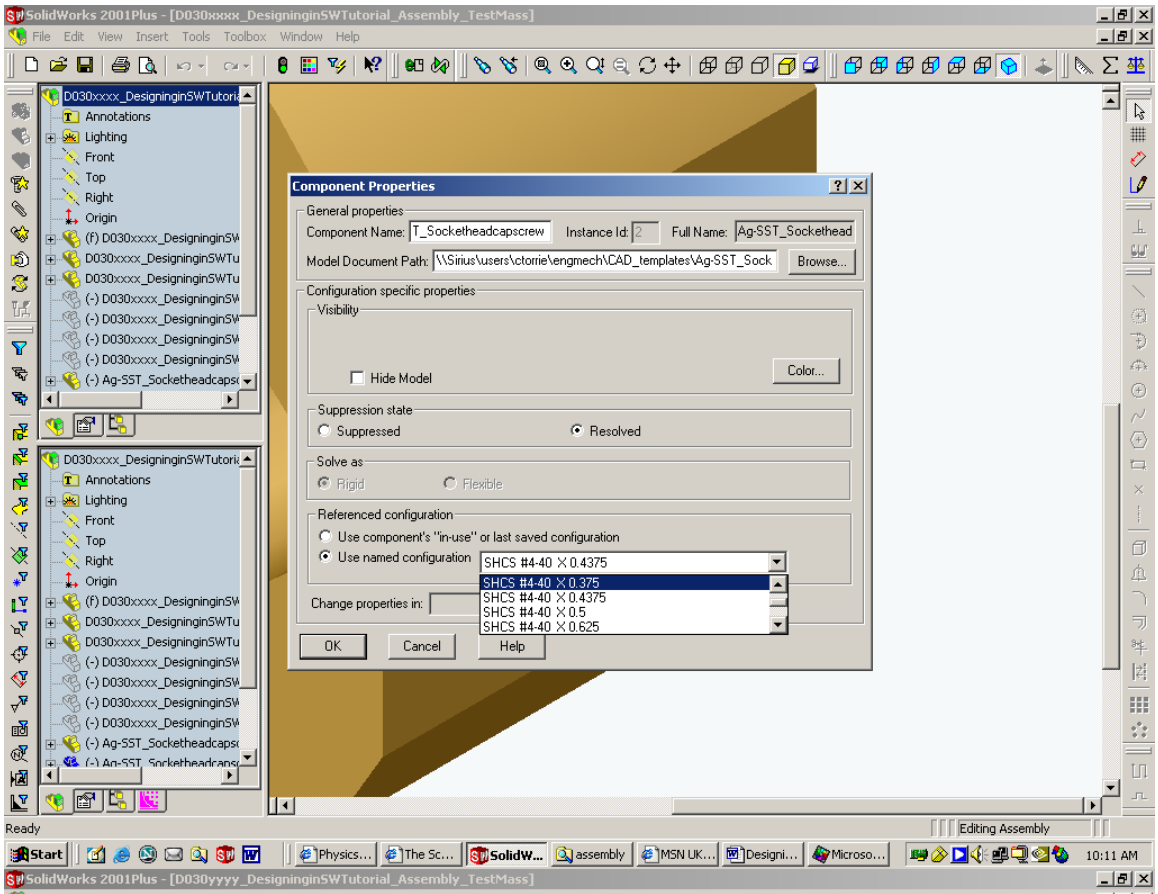


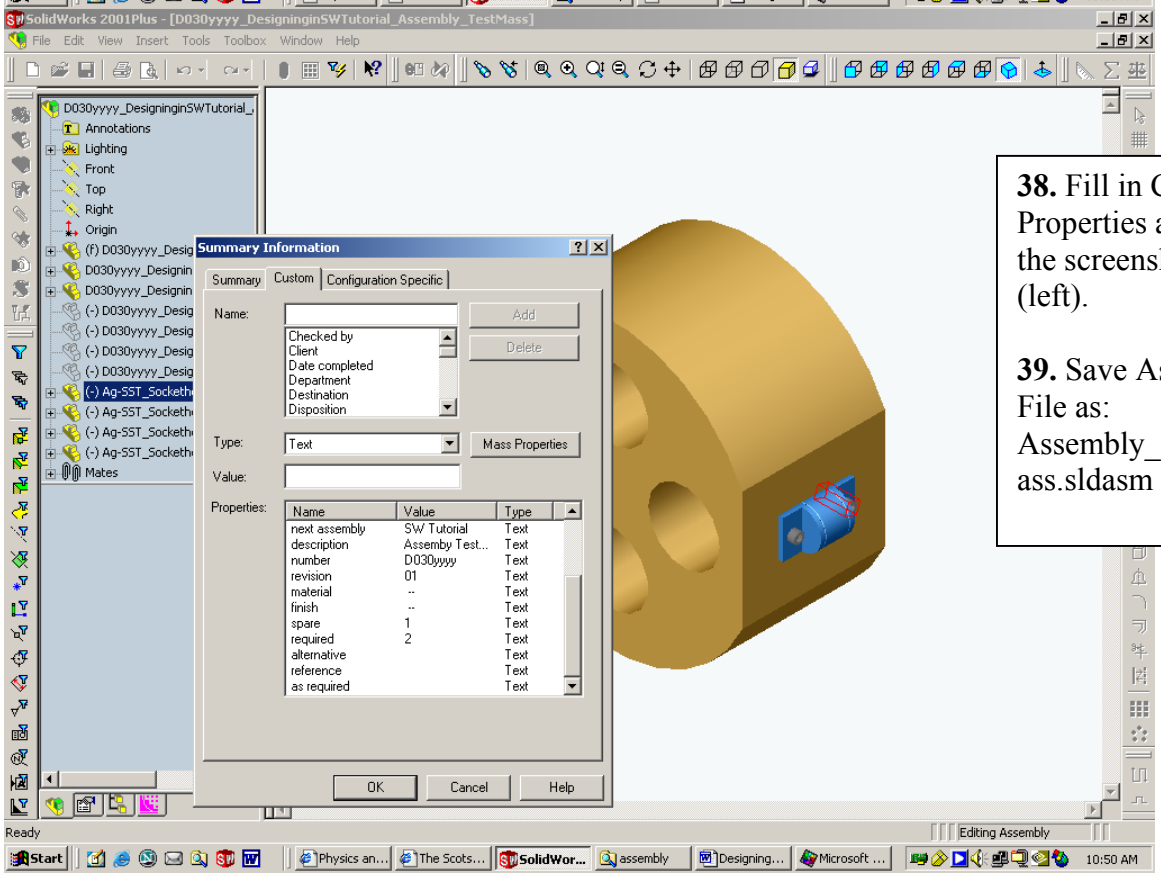
Editing Customised Toolbox Component Properties

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…'

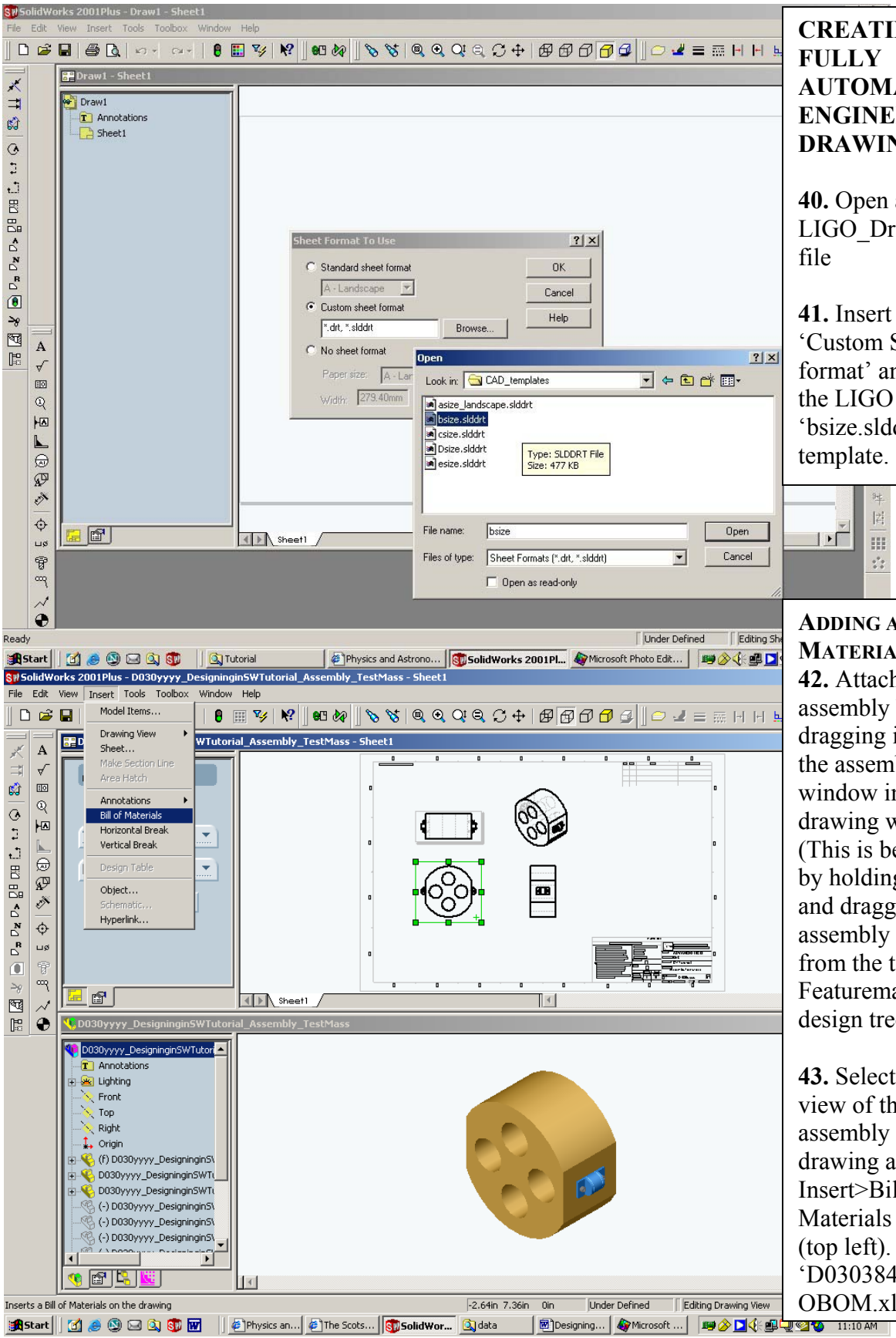
37. Change all bolt configurations to #4-40 X 0.375 (see bottom left and top screenshot on the next page)





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

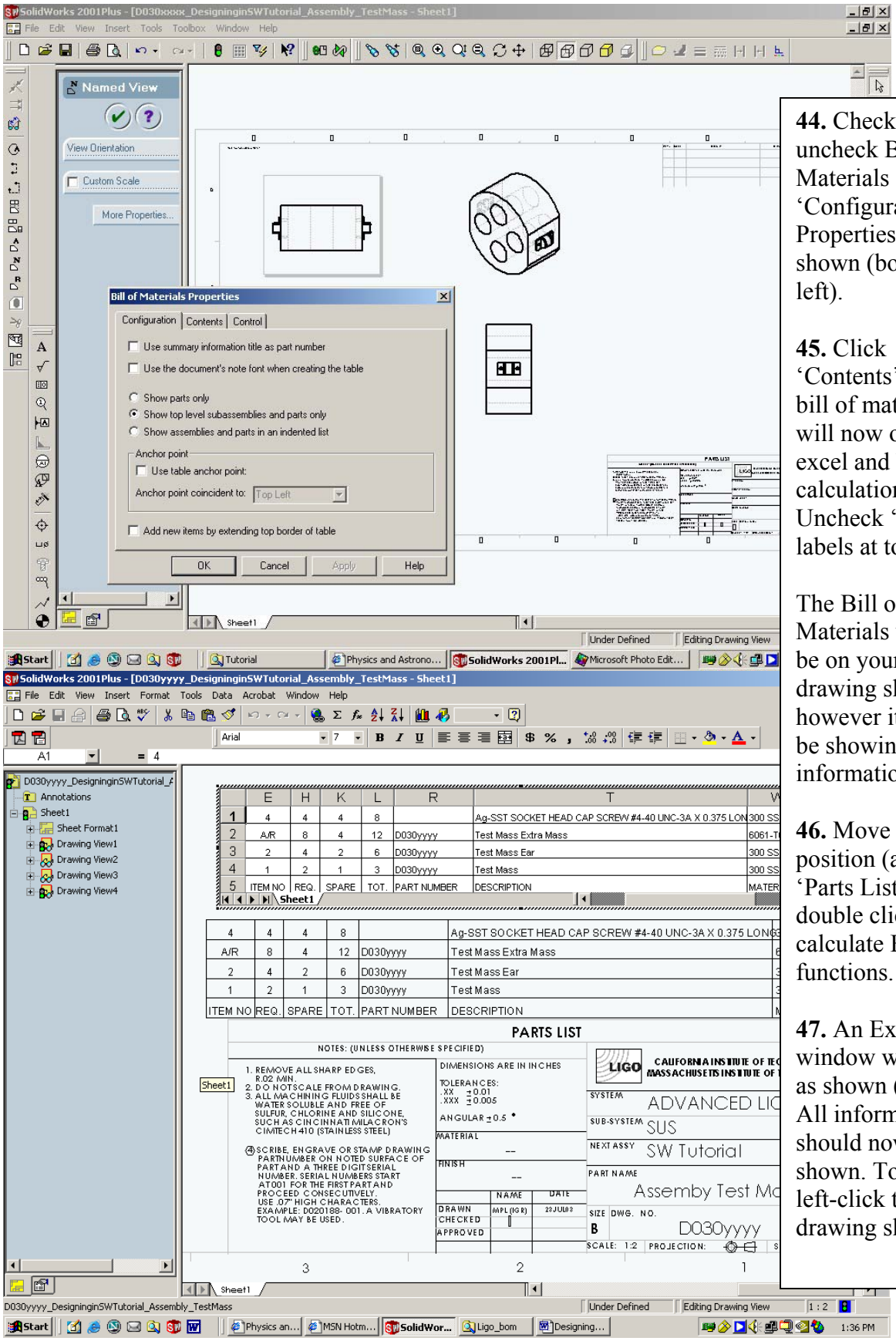
40. Open a new LIGO_Drawing file

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'.



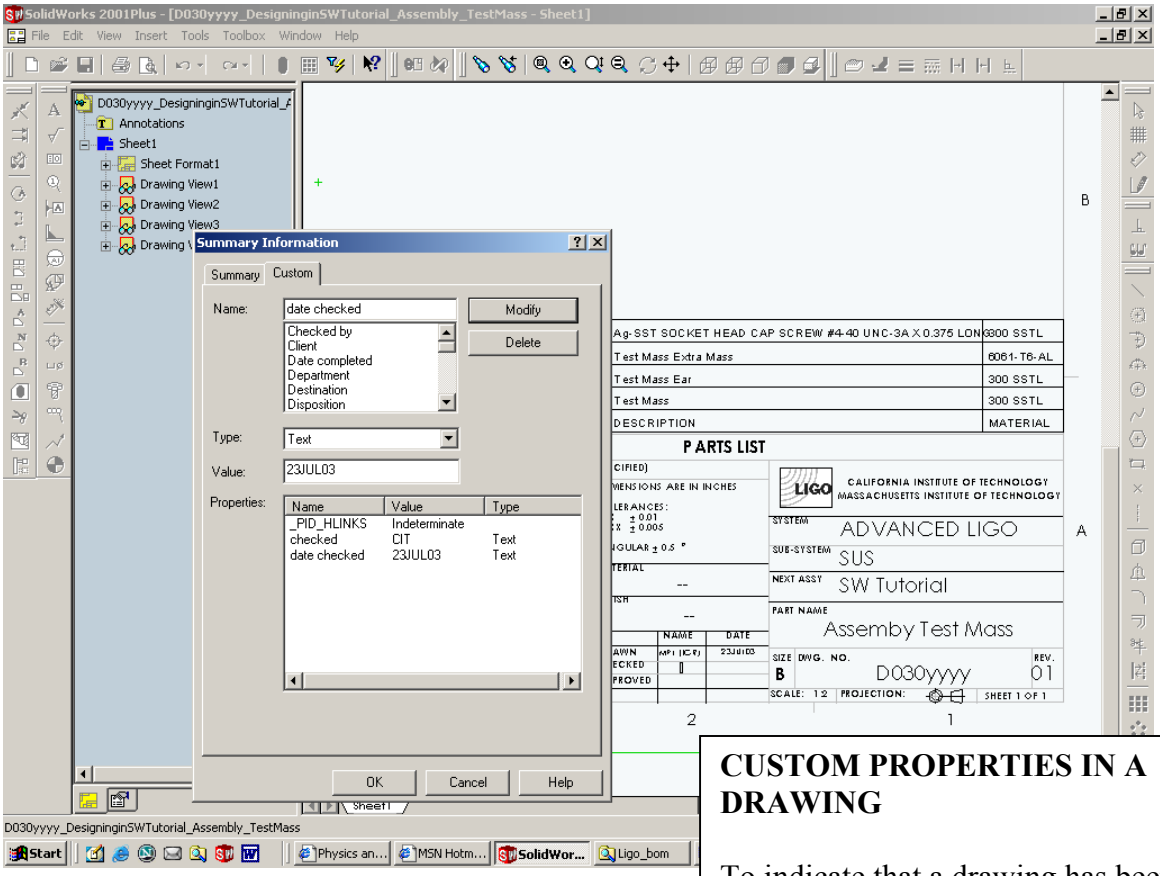
44. Check and/or uncheck Bill of Materials 'Configuration' Properties as shown (bottom left).

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 e.g manufacture notes to the drawing this can still be done by right clicking the drawing and pressing 'edit sheet format'

49. Drawing Complete. Save Drawing as: Assembly_Test_Mass.slddrw

50. The completed drawing should look like that overleaf.

END.