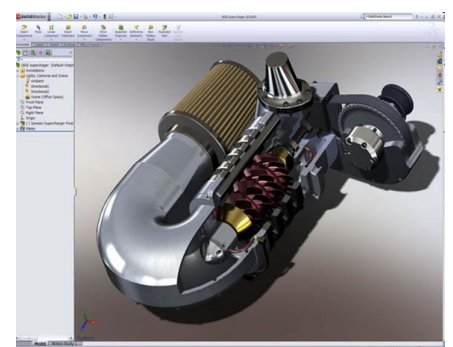
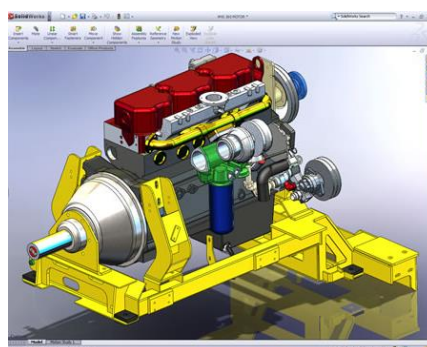
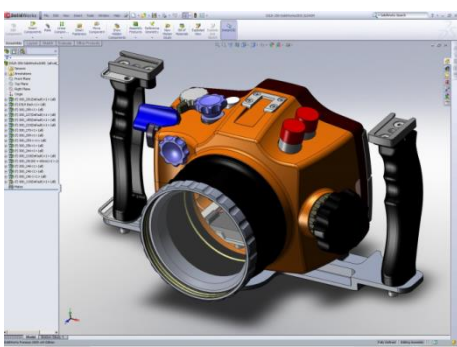
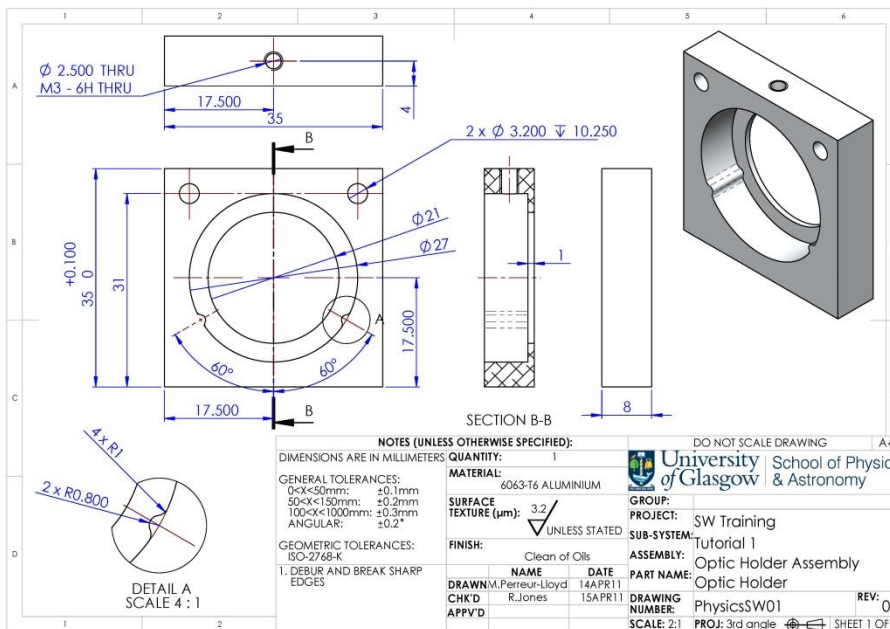
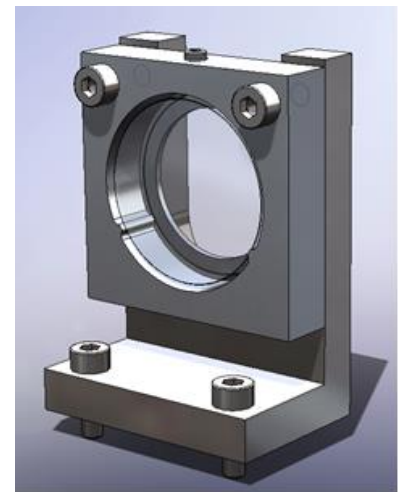
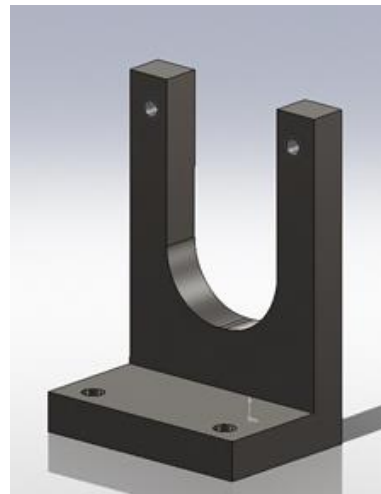
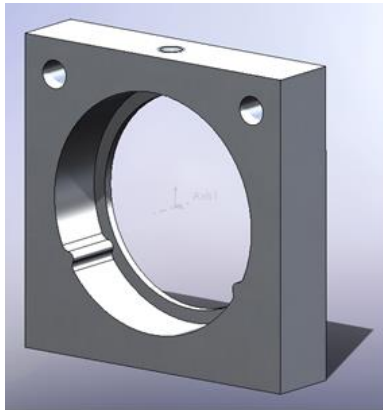


# SolidWorks Training



Introductory course for staff and students from the School of Physics and Astronomy



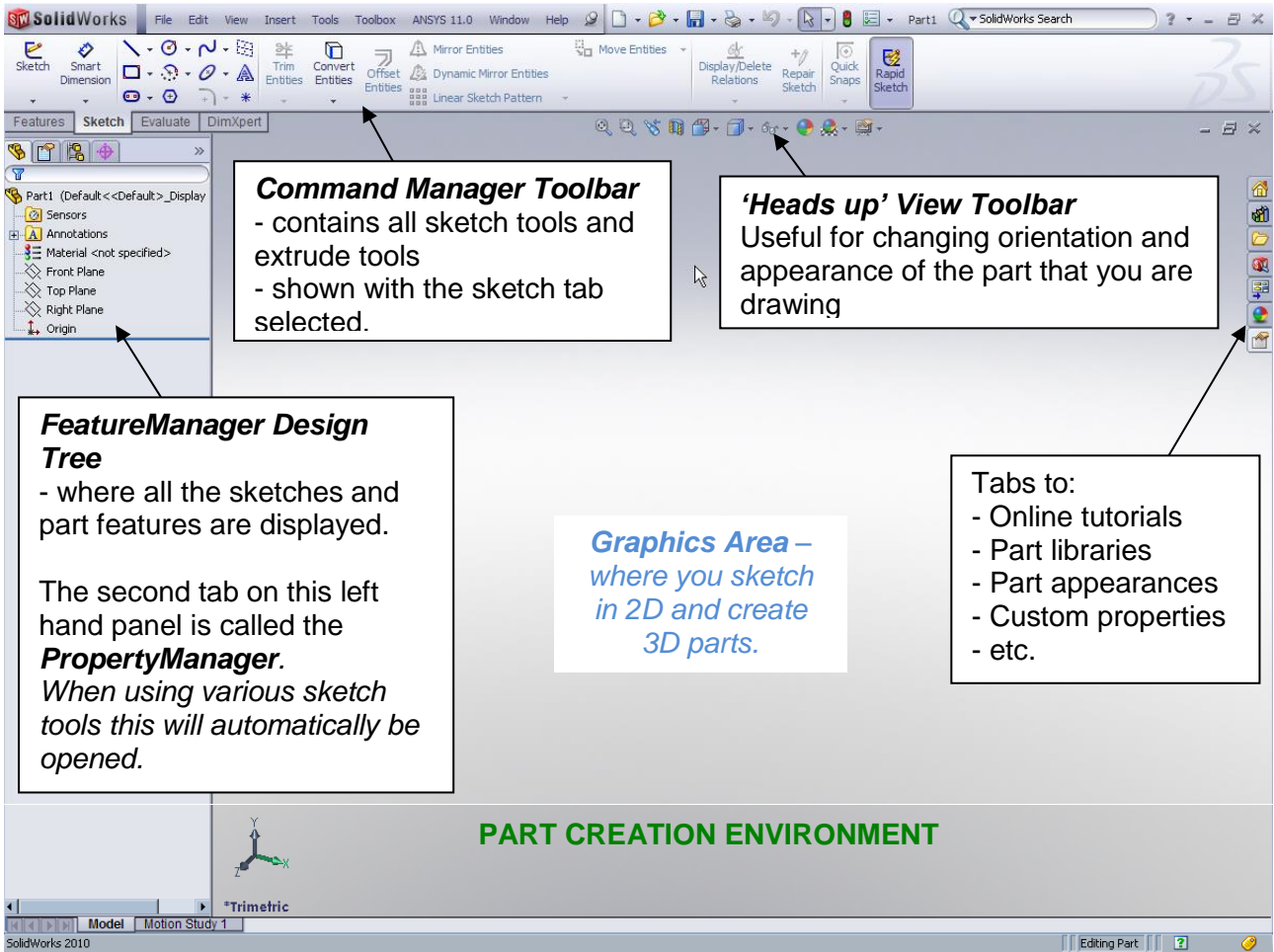


**SolidWorks Training for School of Physics & Astronomy Staff and Postgraduates**  
Introductory course for beginners and false-beginners

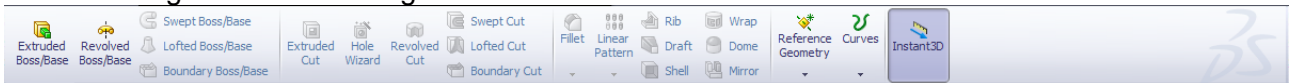
**i) Introductory presentation – ‘SolidWorks Training’**

**ii) The SolidWorks GUI**

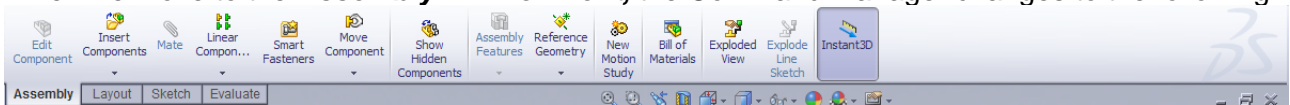
The SolidWorks Graphical User Interface (GUI) is shown below. Specifically this is how the GUI looks when using the Part Creation Environment:



When in the **Part Creation Environment**, the Features menu tab of the Command Manager toolbar changes to the following:



When we move to the **Assembly Environment**, the Command Manager changes to the following



When we move to the **Drawing Environment**, the Command Manager toolbar becomes this:



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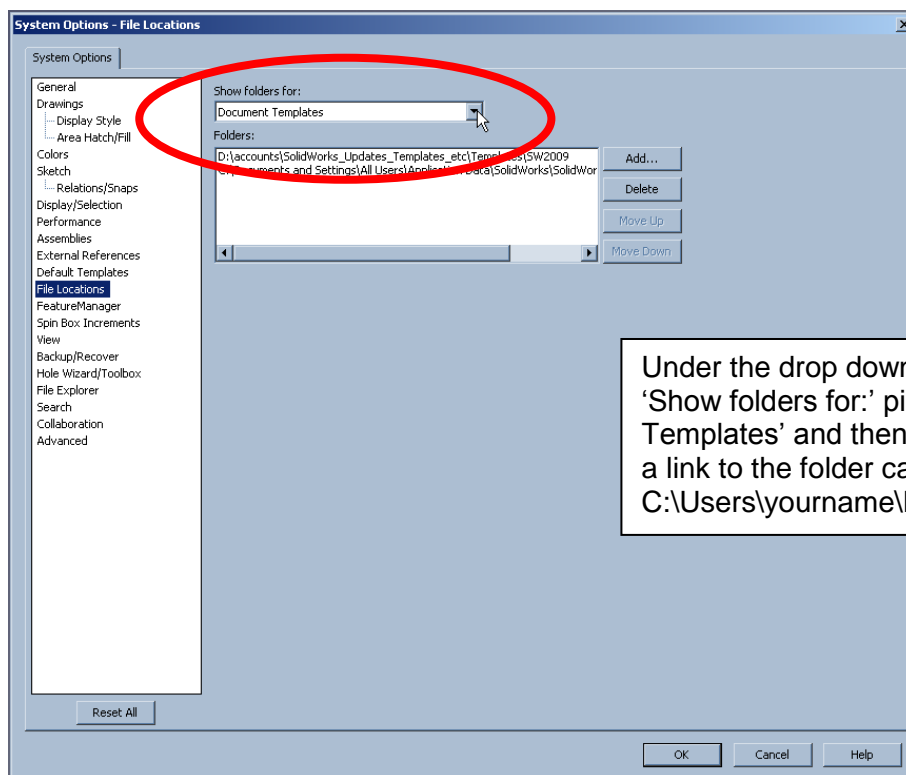
## iii) Setting-up customised SolidWorks tools

Before starting we are going to set up the SolidWorks software with several customised tools that are useful in design created in the School of Physics & Astronomy. These custom tools allow for parts to be drawn up with additional custom properties that are automatically linked to and displayed on engineering drawings (more detail later!).

- Open a web browser and go to <http://www.physics.gla.ac.uk/~mployd/Eng-SW>
- Download the 'UGL Tools' zip file to C:\Users\yourname\Documents\ then extract the files (right click on zip file and select Extract All...)
- Open SolidWorks
- Select the 'Options' menu – as shown below (this menu can also be found under Tools>Options)



- Select 'File Locations' from the right hand menu tree.



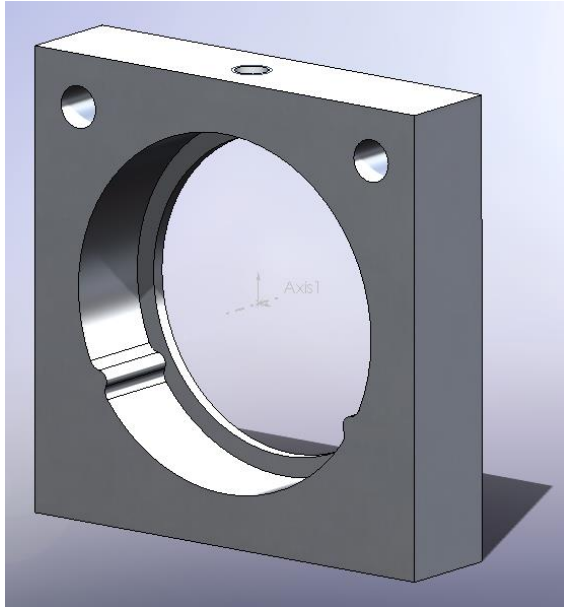
- Repeat this for the following categories in the drop down menu:

- Document Templates
- BOM Templates
- Custom Property Files *[Delete the default folder before adding a new folder link]*
- Drafting Standards

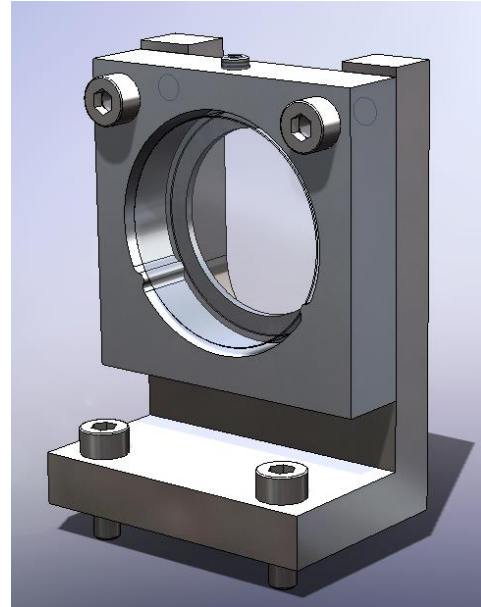
- Sheet formats

## TUTORIAL PART 1 - Creating Parts

The first tutorial is an exercise that demonstrates the most commonly used functions when creating a 3D solid part. We are going to create a simple optic holder, shown below-left, which will form part of an optical mount assembly shown in the picture below-right.

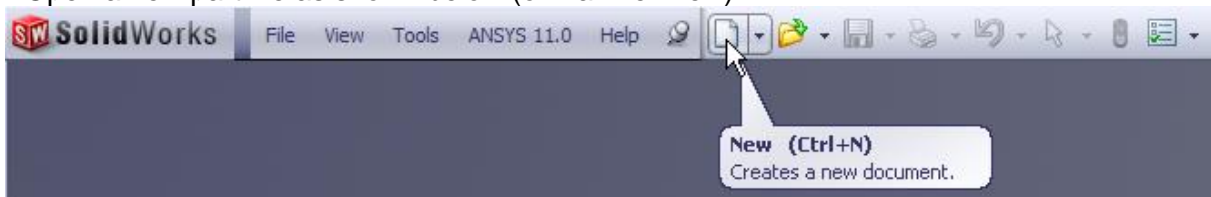


Optic Holder

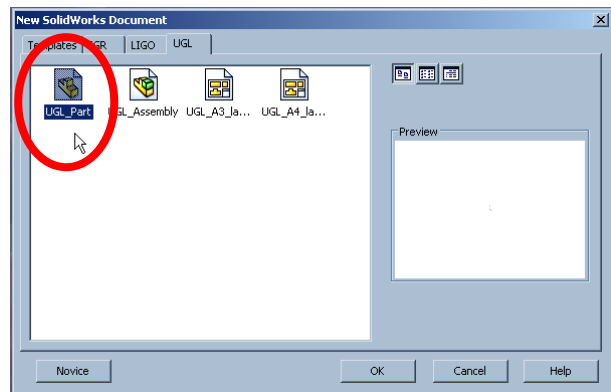
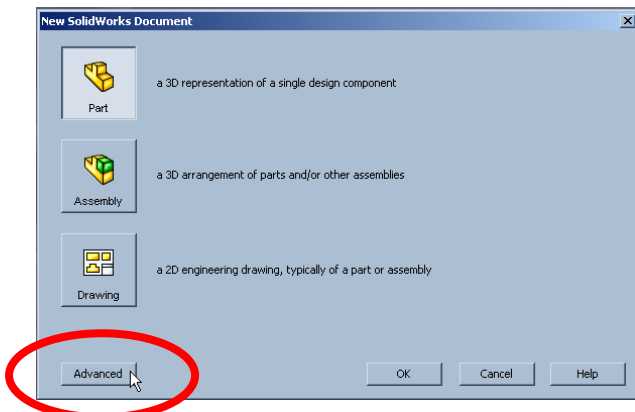


Optical Mount Assembly

1.1 Open a new part file as shown below (or via File>New)



1.2 Select 'Advanced' using the bottom right hand corner button then UGL\_SW\_Templates\_Etc

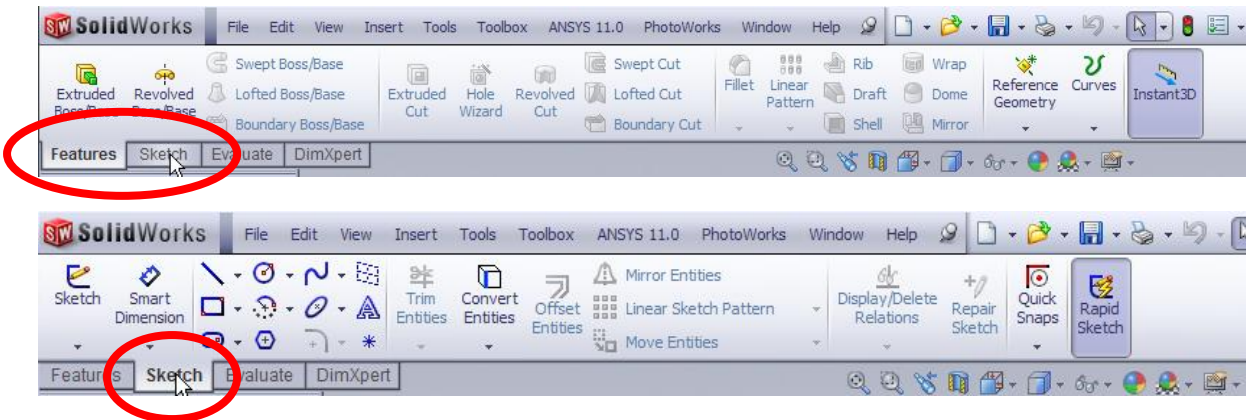


A new part window shall open with the most commonly used tools in the tool bar, named the 'Command Manager' along the top of the screen.

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## Introductory course for beginners and false-beginners

1.3 From the 'Command Manager Toolbar', select the 'Sketch' tab (see below)

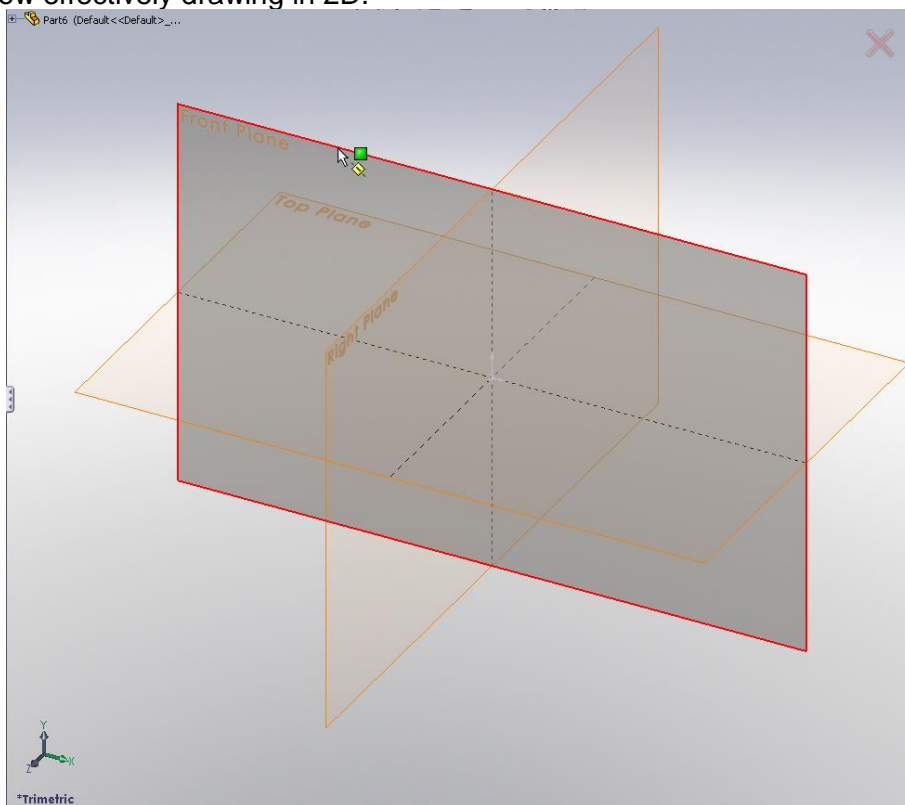


## 2 Drawing the Part




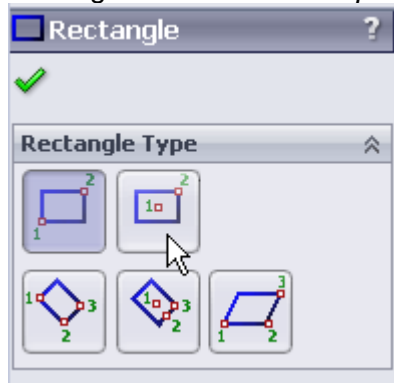
2.1 Select the 'Sketch' tool. Three planes should now be shown in the main drawing window (as per the figure below).

2.2 By selecting the front plane, the view should rotate such that the front plane is facing you and you are now effectively drawing in 2D.



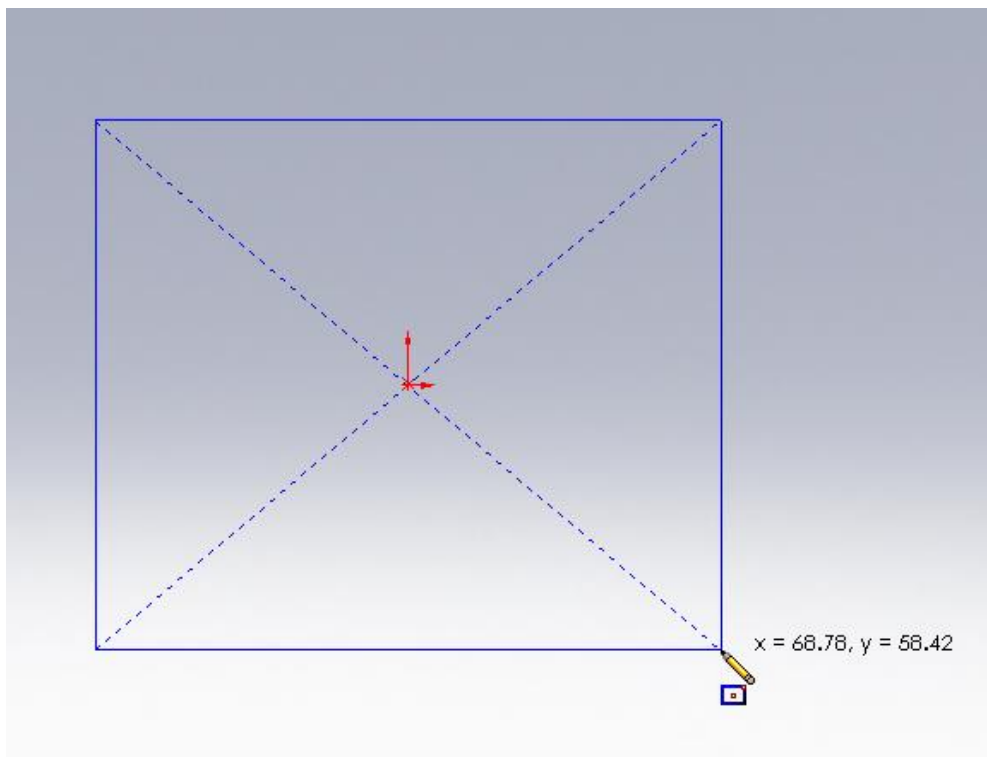
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2.3 Select the 'Rectangle' tool . Notice that there are several options on how to draw the rectangle shown in the *PropertyManager* panel on the left hand side of the GUI



. We will select the 'Center Rectangle' tool.

2.4 Hover the cursor over the origin point and using the left mouse button (press once but do not hold) select the red origin point of the sketch, pull out a rectangle of any size by moving the mouse laterally, then click the left mouse button again to complete the action.



*Should you draw something that you do not wish to have, press **Escape** to exit the rectangle tool. The unwanted sketch can be deleted by dragging the cursor across the rectangle using the left mouse button, then pressing **Delete** on your keyboard. Alternatively individual lines of the sketch may be selected using the mouse and Deleted one by one.*

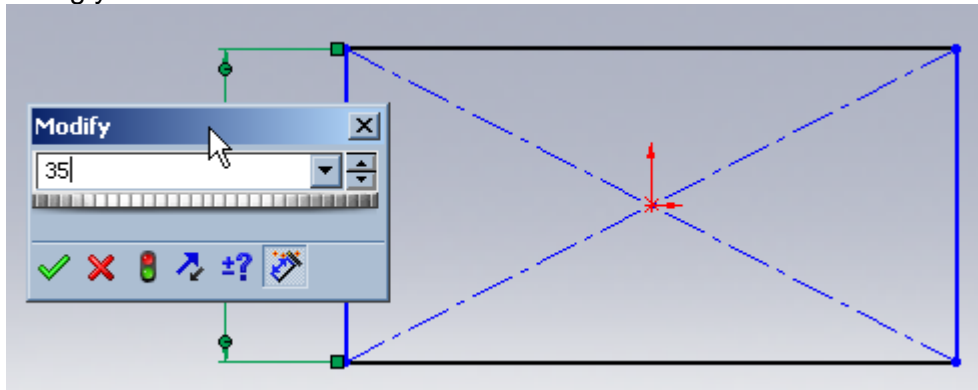
2.5 We are now going to define the line lengths of the sketch by dimensioning each line.




Select the 'Smart Dimension' tool

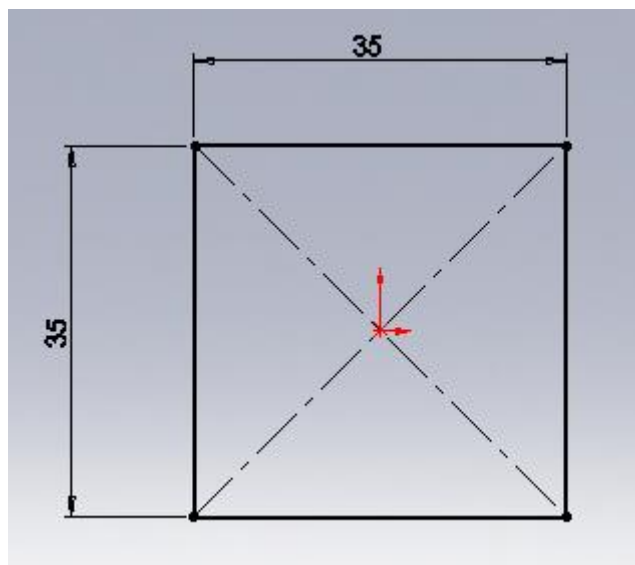
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- 2.6 Select the vertical line in your sketch, drag out the dimension so that you can see both the line and the dimension clearly, and then click the left mouse button. A small 'Modify' window shall pop up allowing you to enter a dimension. Enter 35mm.



- 2.7 Repeat for the horizontal dimension and again enter 35mm as the length.

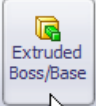
To close the 'Dimension' Tool, either press the green tick  at the top left corner of the *PropertyManager* panel, or simply press the 'Esc' button on the keyboard.



Now that we have defined our overall dimensions in 2D we now want to add the 3<sup>rd</sup> dimension to create a solid part.

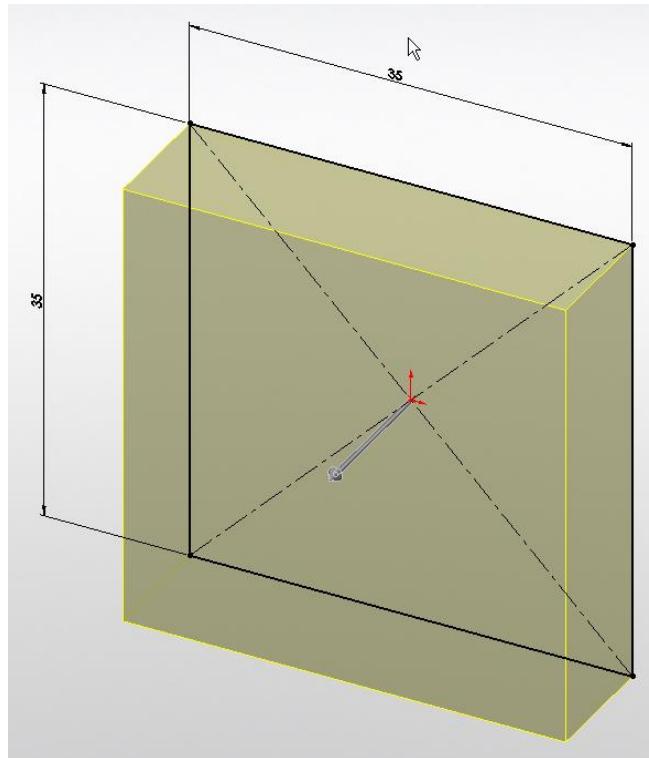
*To edit a dimension, if it needs to be changed or was entered incorrectly, simply double click on the number in the graphics area and enter the correct dimension.*

- 2.8 Select the 'Features' tab from the *CommandManager Toolbar* along the top of the SolidWorks

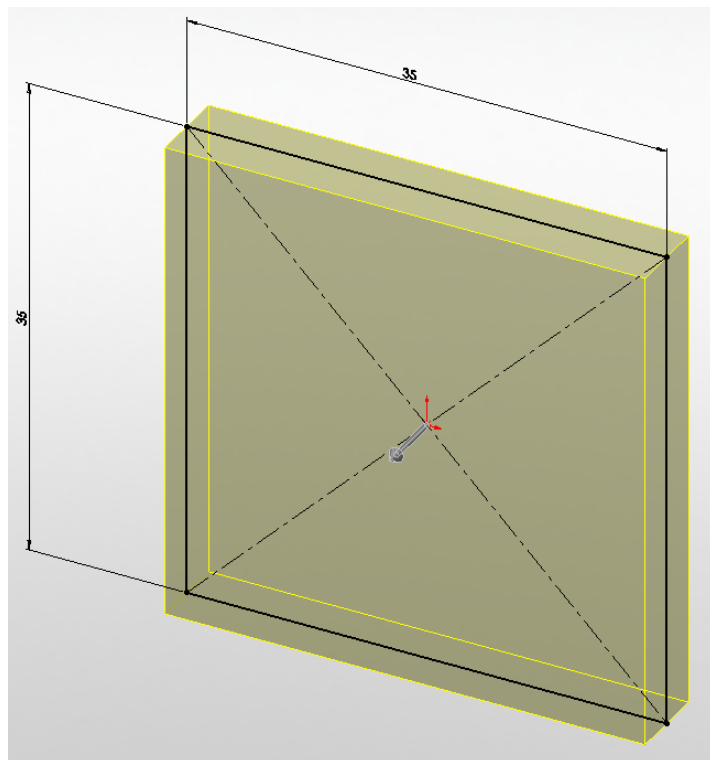
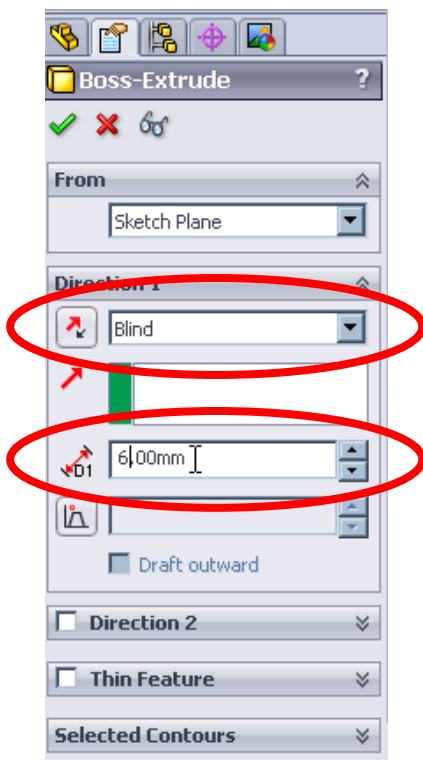
window. Then select 'Extruded Boss/Base' .



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The part should rotate to an Isometric view with a silhouette of the solid part. By default the part thickness value in the *PropertyManager* panel is set to 10mm. This should be changed to 6mm and also 'Mid Plane' should be selected from the drop down menu instead of 'Blind'



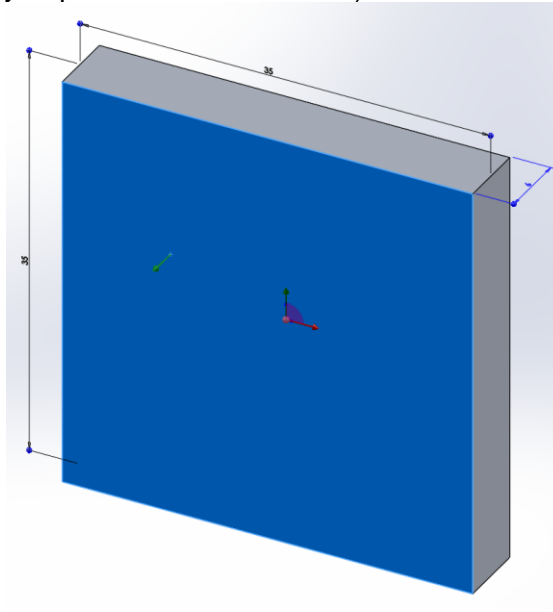
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To confirm the action, click the green tick (either in the *PropertyManager* panel  or in the top right hand corner of the *Graphics Area*  ) to leave a solid cube of dimensions 35x35x6mm.

### 3 Circular holes

3.1 Select the front face of the cube with the left mouse button. (Notice how a face is highlighted with an orange outline as you pass the mouse over it).



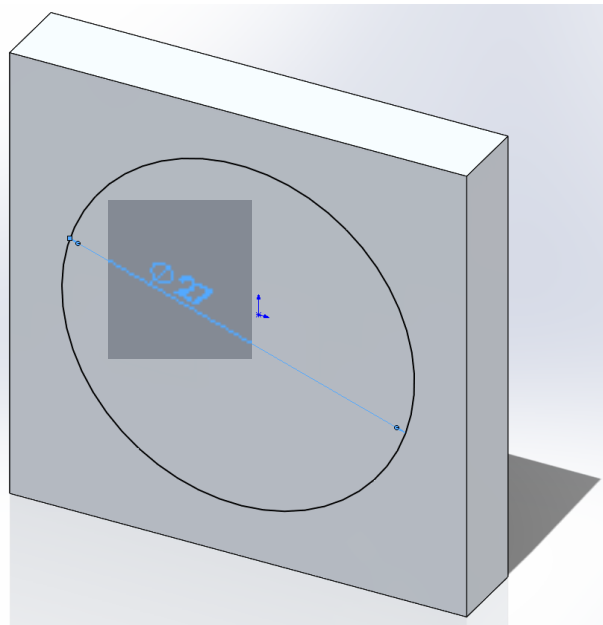
3.2 Select the 'Sketch' tab from the *CommandManager* toolbar and then select 'Sketch'. A new sketch shall be started on the front face of the cube.

3.3 Select the 'Circle' tool  and draw a circle, centred on the red origin on the front face of the cube. Again, the circle can be dragged to any size – we will define this by adding a dimension, next.

3.4 Exit the Circle tool by pressing Escape or the green tick in the *PropertyManager* panel.

3.5 Select the 'Smart Dimension' tool  and dimension the circle with diameter 27mm.

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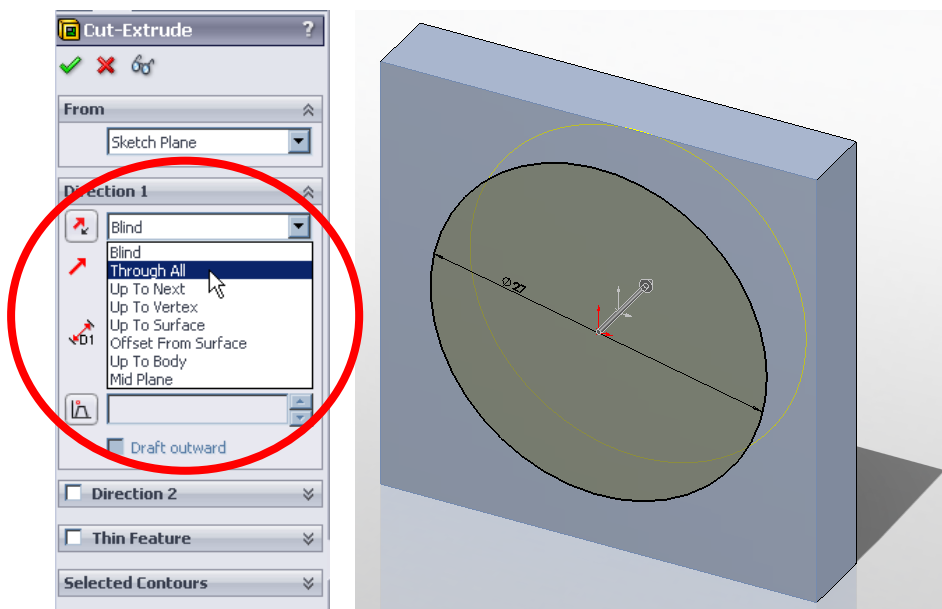


3.6 Now, to make a circle-shaped cut through the part, select the 'Features' tab in the Command



Manager toolbar, then select the 'Extruded Cut' tool. The circle should now have a silhouette of a hole removed from the cube (as per figure below right).

3.7 In the PropertyManager panel, under the top drop down menu in 'Direction 1', change the hole from being 'Blind' (and 6mm long) to going 'Through all'



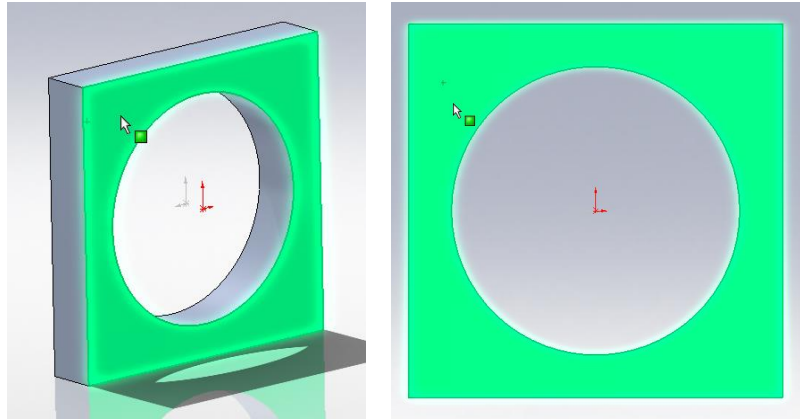
Whilst this action (changing it to Through All rather than Blind/6mm) will achieve the same end result, it is better in this instance to use the 'Through All' action in case you wish to change the thickness of the part at a later stage.

3.8 Rotate the part to show the reverse side of the cube.

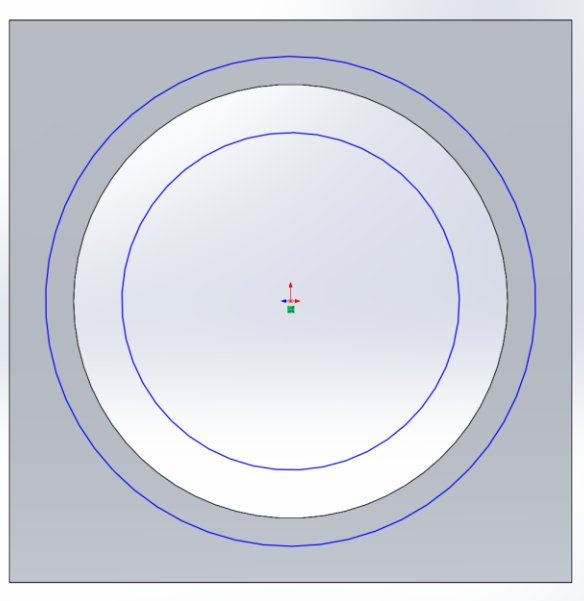
*As a reminder, the part can be rotated by pressing and holding the middle mouse button and moving the mouse. Zooming in or out can be done by scrolling the middle mouse button.*

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3.9 Now select the back face of the part and select the 'sketch' tool under the 'sketch' tab of the Command Manager toolbar.



3.10 Sketch two concentric circles with different diameters centred on the red origin of the back face of the part.



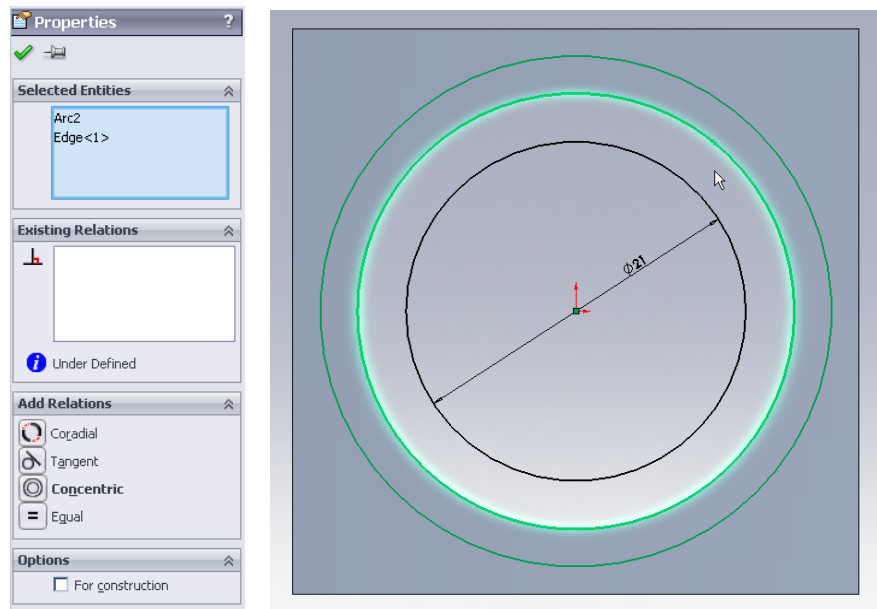
3.11 Using the Smart Dimension tool, dimension the smaller of the circles with a diameter of 21mm, after which exit the Smart Dimension tool.

The second circle is to be identical in diameter to the cylindrical bore through the part. There are two possibilities for this, either a dimension made be applied exactly the same as the diameter of the circular bore or, much better, particularly if the part may change size at a later stage, a 'Relation' may be added to match the diameter of the bore feature to the circular sketch. To add a relation, do the following:

3.12 Select the edge of the circular bore and then, whilst holding down the 'Ctrl' button on your keyboard, select the second circle in the sketch.

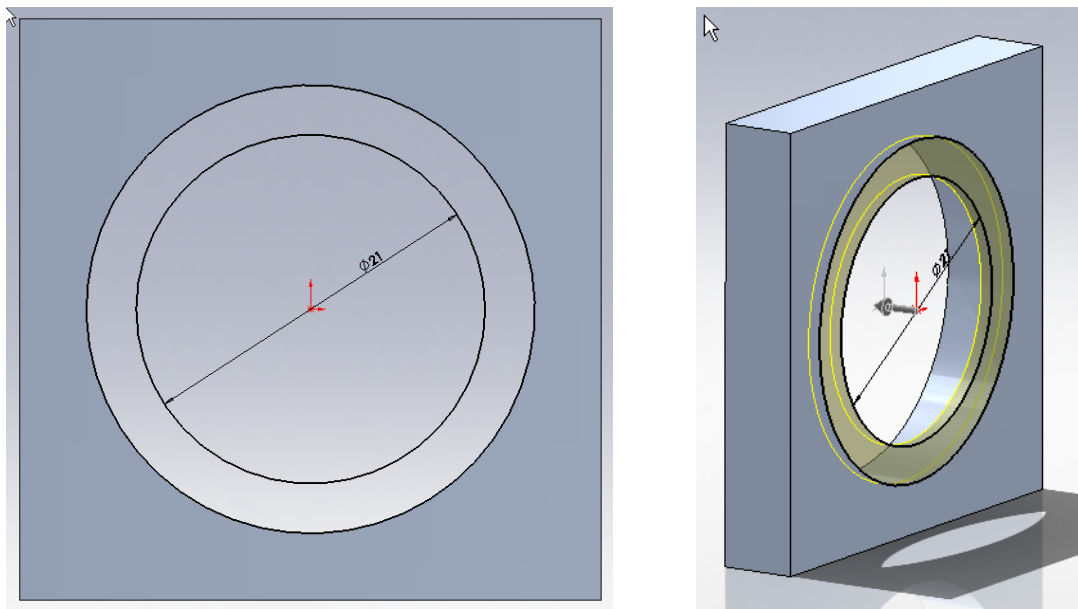
Notice that a 'Properties' area will open in the *PropertyManager* in the left hand panel.

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3.13 In the 'Add Relations' sub-section of the *PropertyManager*, select the 'Coradial' relation.

The sketched circle should now sit coradial to the edge of the bore hole and will have changed colour from blue to black. This line colour change indicates that the circle is fully defined.



3.14 Now, extrude these two circles 1mm in towards the part (i.e. so that this does not protrude from the part). To see this better the part should be rotated as per the right hand figure above.

The direction of the extruded sketch may be reversed by using the 'reverse direction' button



in the *PropertyManager* panel.

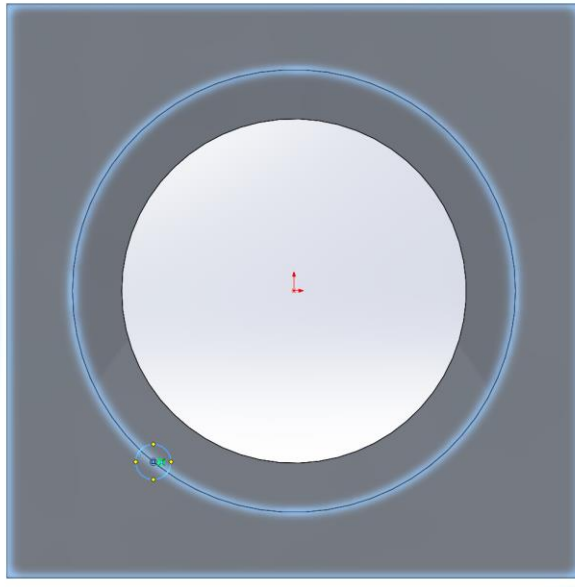
3.15 Confirm the action by clicking the green tick.

*This is probably a good time to Save your work. Save the file as Optic\_Holder.sldprt in the folder that you created at the start of the tutorial.*

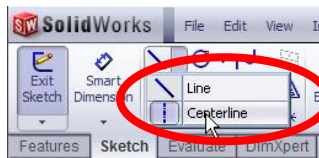
## 4 Sketches with Construction Lines

We are now going to add two features that form the two points in the kinematic mount for our optic. The third point in this three point mount shall be defined by a grub (or 'set') screw.

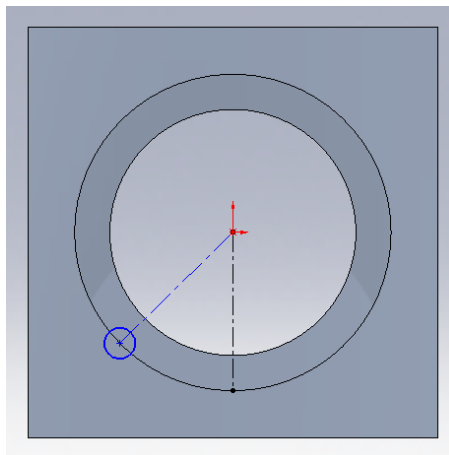
- 4.1 Select the front face of the cube, and draw on it one small circle at approximately 7 o'clock and with the centre of the circle aligned with the circular edge of the large bore hole.



If the small circle is drawn without its centre point being aligned to the circular edge of the bore, this can be realigned by selecting both the edge of the bore hole and the circle centre, whilst holding down the 'Ctrl' button, and adding a 'coincident' relation.

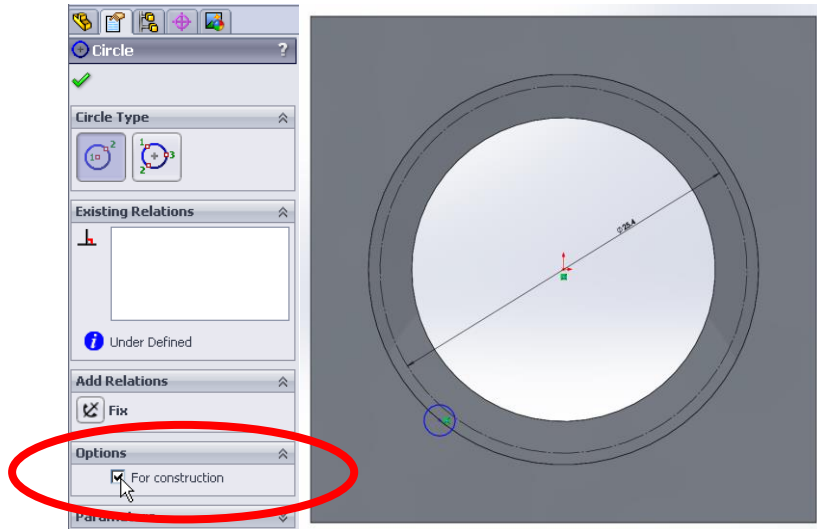


- 4.2 Additionally draw two centerlines. The first should be from the centre of the small circle that you have just drawn to the origin of the part, then a second line from the origin down to the edge of the bore (at 6 o'clock). Double click away from the part to stop sketching lines.



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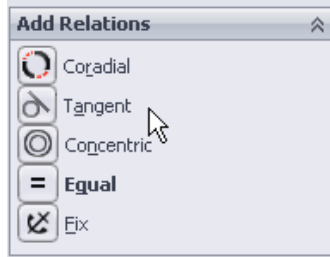
- 4.3 Now, in the same sketch, add a circular sketch centred on the part origin.
- 4.4 Additionally, change the line to being a construction line by checking the box in the Circle *PropertyManager* menu.



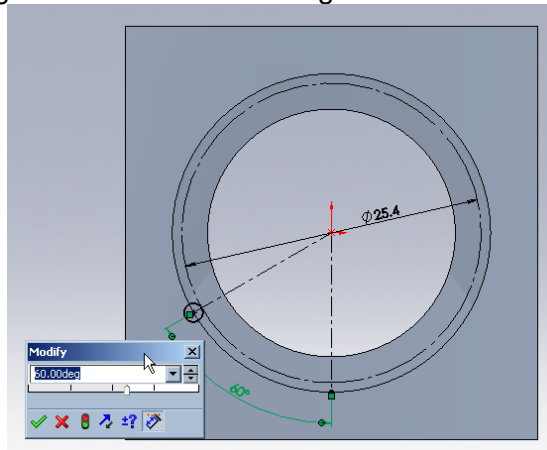
- 4.5 Dimension this circle to be 25.4mm.

*This dashed, circular construction line simulates the position where a one inch cylindrical optic will sit in the optic holder. We shall use this construction line to help us define the kinematic features around the optic, to be the correct size.*

- 4.6 Now, to define the diameter of kinematic contact point, the small circle should be aligned tangentially with the 25.4mm diameter construction line. To do so, select both the small circle and the construction circle (hold down the Ctrl button) and set a Tangent relation.



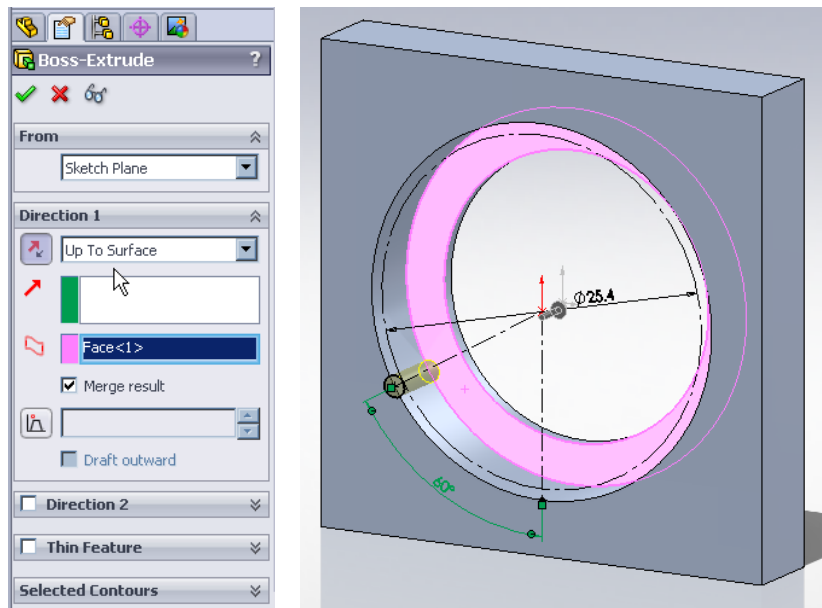
- 4.7 Finally, dimension the angle between the two straight centrelines as 60 degrees.



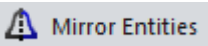
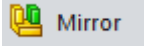
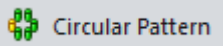
All lines in your sketch should now be black – identifying that they are all fully defined.

4.8 Next, extrude a solid cylinder using the small circle up to the lip on the hole. To do so, select 'Extrude Boss/Base' from the *Features* toolbar and extrude 'Up to Surface', selecting the surface now at the base of the bore.

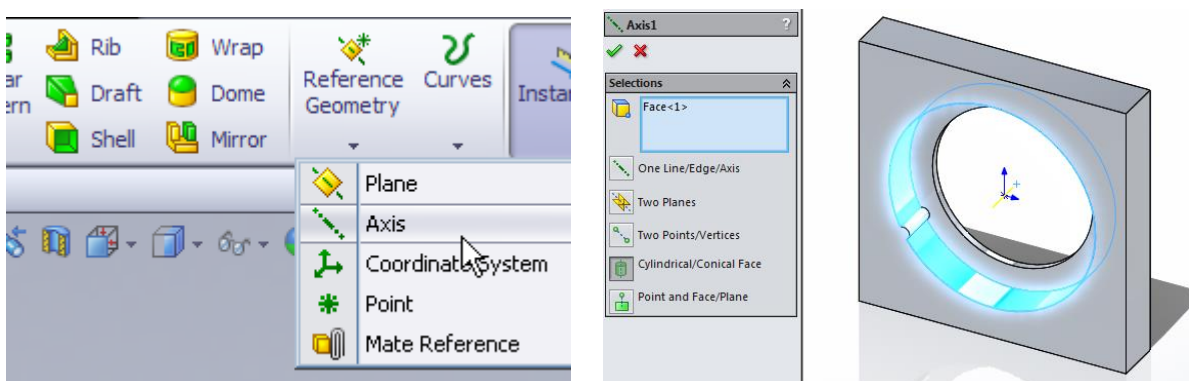
You may wish to rotate the part to make it easier to see which direction of the extrusion.



## 5 Patterning Features

The first of the two contact point features of the isostatic mount is now in place. This feature could simply be mirrored in the sketch  or mirrored as a Feature  however, to introduce another useful tool, for the second we will use the 'Circular Pattern' tool . To aid the use of this tool a reference 'Axis' will be added to the 3D part to pattern our feature around.

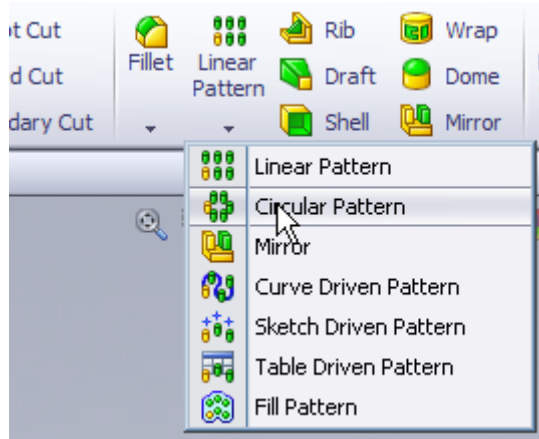
5.1 Select the 'Axis' tool (under the Reference Geometry drop-down menu) and then select the internal cylindrical face of the first bore that was cut from the part. SolidWorks should automatically select the correct tool function to create an axis down the centre of the bore, as highlighted in the figure below right.





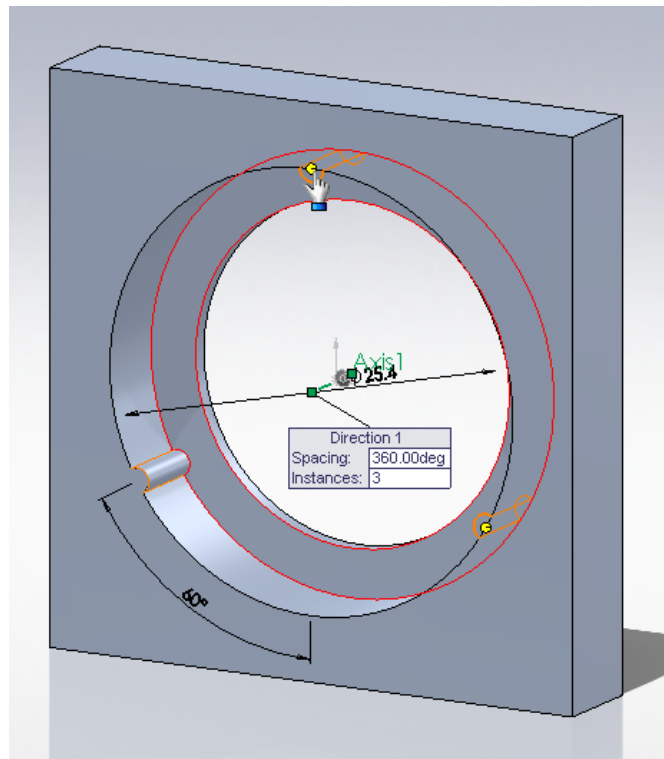
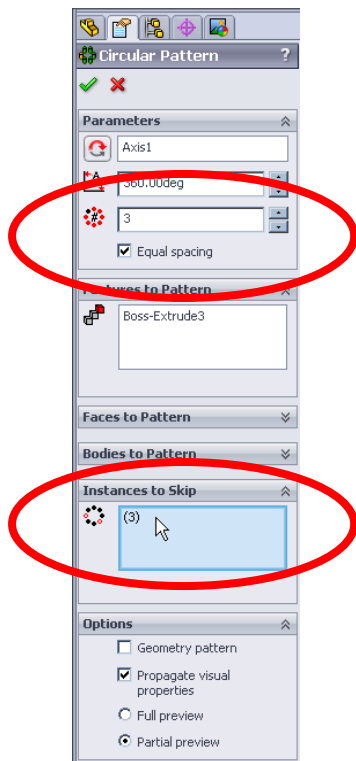
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5.2 Now, in the *FeatureManager Design tree*, select both the last feature that was made and the new 'Axis' (Use the Shift or Ctrl key to select both items), and then open the 'Circular Pattern' tool.





Automatically SolidWorks shall preview several further identical features.

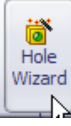
5.3 In the Circular Pattern tool, change the 'number of instances' to 3, select Equal Spacing, and also, under 'Instances to Skip', select the node at the top of the part (as shown below).




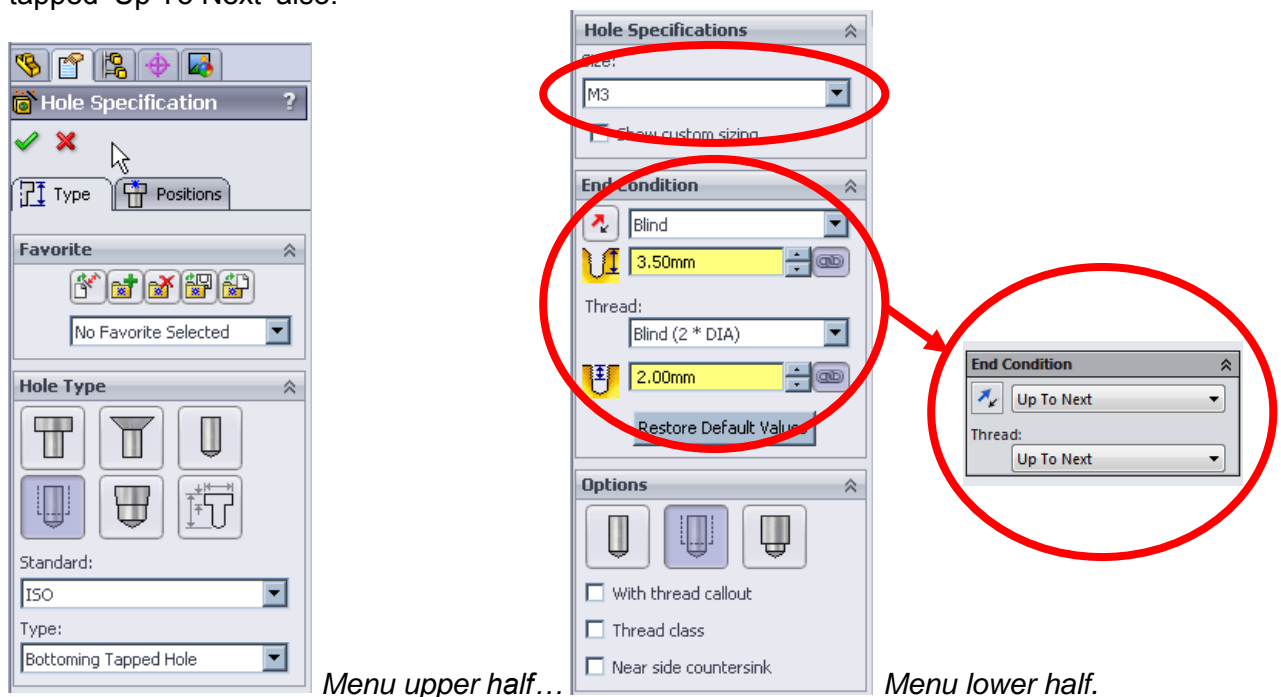
## 6 Adding holes with the Hole Wizard

Now we will add a threaded hole to the top face of the optic holder.

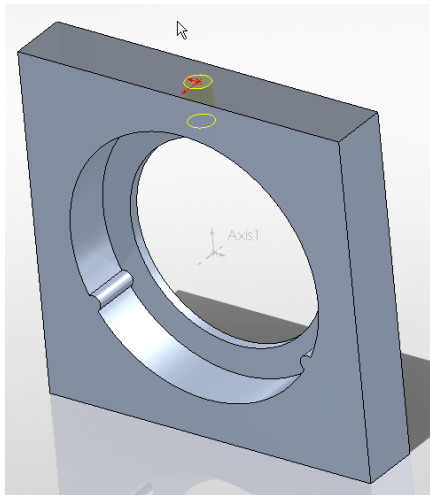
- 6.1 Select the top face of the part and rotate the part such that that top face is pointing towards you (press the Space-bar and click the 'Normal To'  or 'Top View'  button).


- 6.2 Next, open the Hole Wizard tool  in the Features toolbar. A menu will open up in the FeatureManager Design Tree with a number of options allowing you to draw a variety of hole types from a large library within SolidWorks.

We want to draw a tapped M3 hole so this can be selected by clicking the 'Straight tap' button  and selecting M3 from the drop down menu under 'Hole Specification'. Additionally, under the 'End Condition' for the hole, we wish the hole to go 'Up To Next' and it should be tapped 'Up To Next' also.




- 6.3 Next, select the 'Positions' tab at the top of the *PropertyManager* panel and then press click once on the part origin to place a point at the centre of the top face. After that, click the green tick to confirm the action and exit the tool.



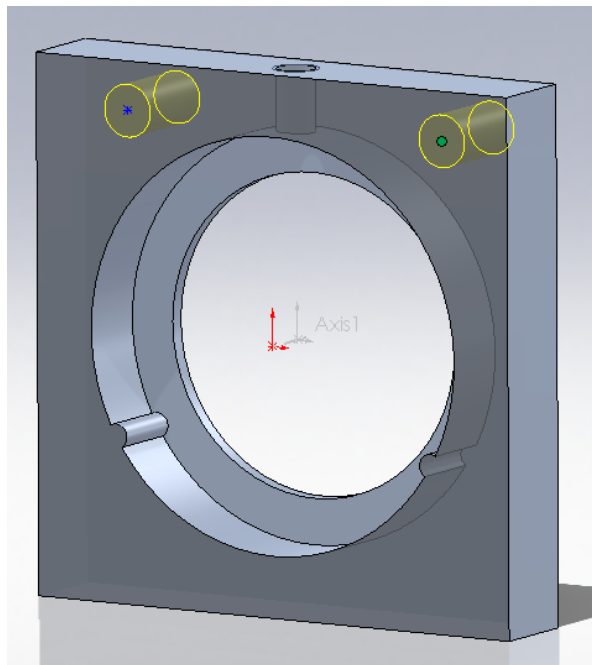
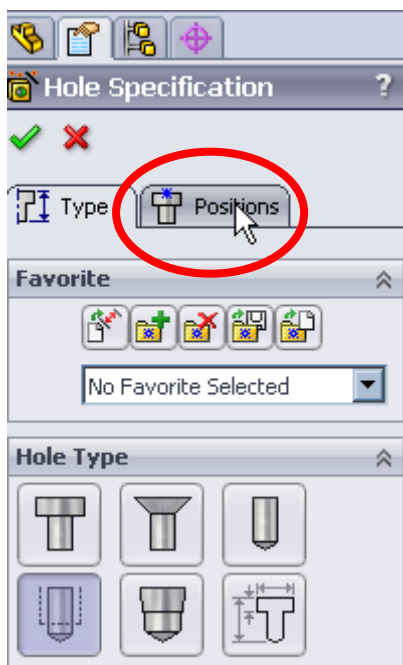
If you happen to have created two or more point sketches on the top surface, to remove the extra points first deselect the Point tool  by clicking on the button in the 'Sketch' toolbar then, secondly, select the supplementary point(s) on your sketch with the cursor and delete (either by pressing delete on your keyboard, or right-clicking and selecting delete from the pop-up menu).

We are now going to add two further 'clear' holes on the front face of the part that are used to attach the optic holder part to its bracket. Clear holes are those that give a clearance to a threaded bolt or screw, e.g. a normal clear hole for an M3 screw is 3.4mm in diameter.


6.4 Select the front face of the part and open the 'Hole Wizard' tool, this time selecting the 'Hole

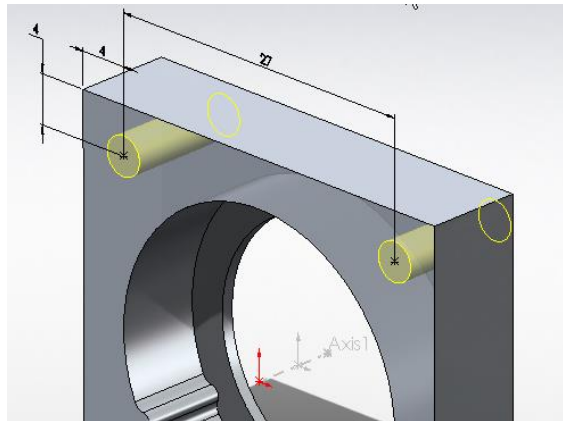
Type as 'Hole' . Under the 'Type:' drop-down menu select 'Screw clearances' and finally make the hole size M3 and give it a 'Through all' End Condition.

6.5 In the *PropertyManager* panel, select the top tab marked 'Positions'. Add two points on the front face of the part then deselect the point tool by clicking on the 'Point' button in the Sketch toolbar.




6.6 Now we are going to define the location of these holes by adding dimensions. Select the

'Smart Dimension' tool  and then dimension to the holes with respect to the edges shown in the picture below.



Note that the right hand point is still blue in colour. This is because it is not yet fully defined. This could be defined by adding a further dimension but a better way to do this is to co-align the point horizontally with the left hand point that is already defined in this direction.


6.7 To align the holes horizontally, deselect the 'smart dimension' tool and select both of the points whilst holding down the Ctrl key. The relations menu will now appear in the Feature

Manager Design tree and a 'Horizontal' relation  Horizontal should be selected.

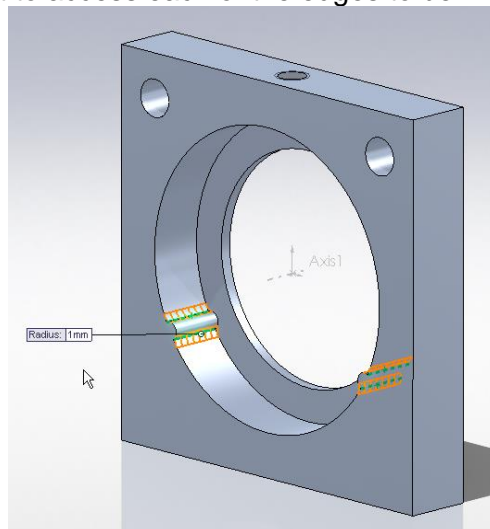
6.8 Complete the Hole Wizard action by clicking the green tick.

## 7 Finishing Touches

As a finishing touch to the part, and a useful thought towards the manufacturability of it, we will add a radius, or fillet, at the edges of the two isostatic features of the optics mount. This is a simple

operation using the 'Fillet' tool .

7.1 Select the 'Fillet' tool. Change the radius of the fillet to 1mm. then select the four edges. You will need to rotate the part to access each of the edges to be filleted.



7.2 The part is now complete. Save the part as Optic\_Holder.sldprt in the assigned folder in C:\Users\yourname\Documents\.

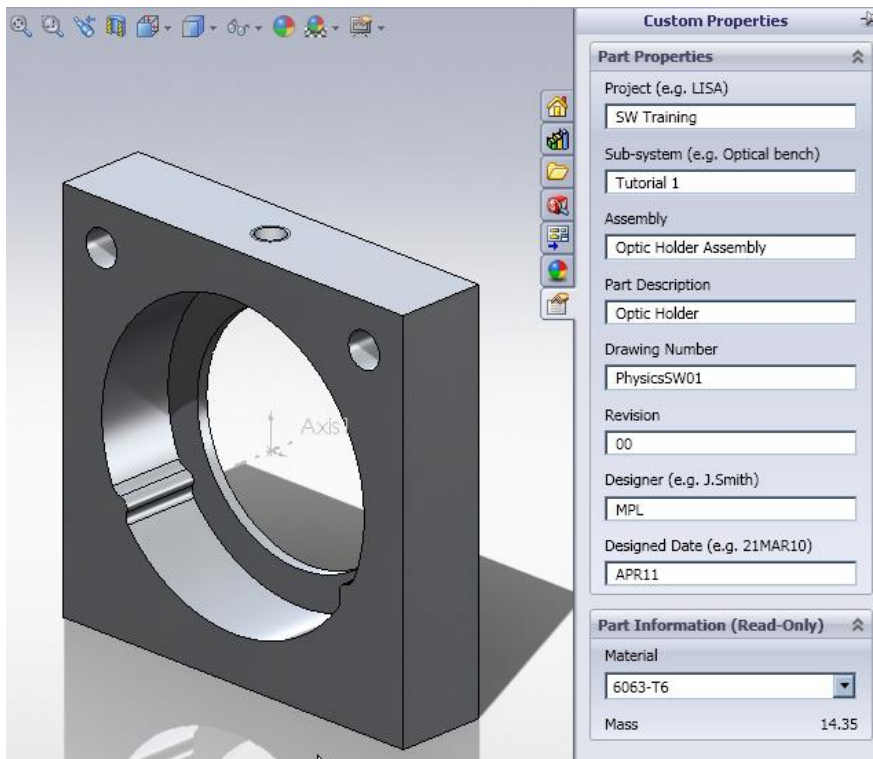
**END OF TUTORIAL PART 1**

## TUTORIAL PART 1 Extras

### A.1 Adding Custom Properties

When creating the parts or assemblies it is possible to add properties to the part. These properties will subsequently show in the engineering drawings.

A.1.1 Select the Custom Properties tab from the right hand tabs on the SolidWorks GUI. Then fill in the tabs with the appropriate information.

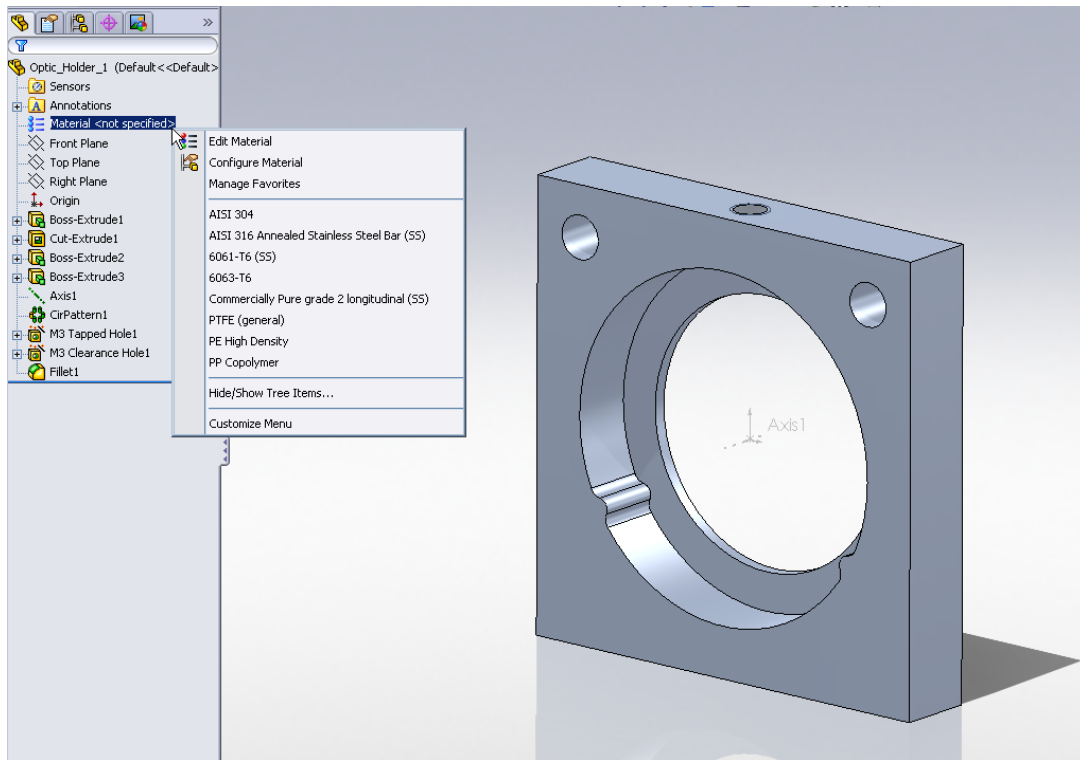


A.1.2. The drawing number should be an alphanumeric code of 10 digits or less that will allow you to easily and quickly identify you part if it is out for manufacture.

### A.2 Applying Materials

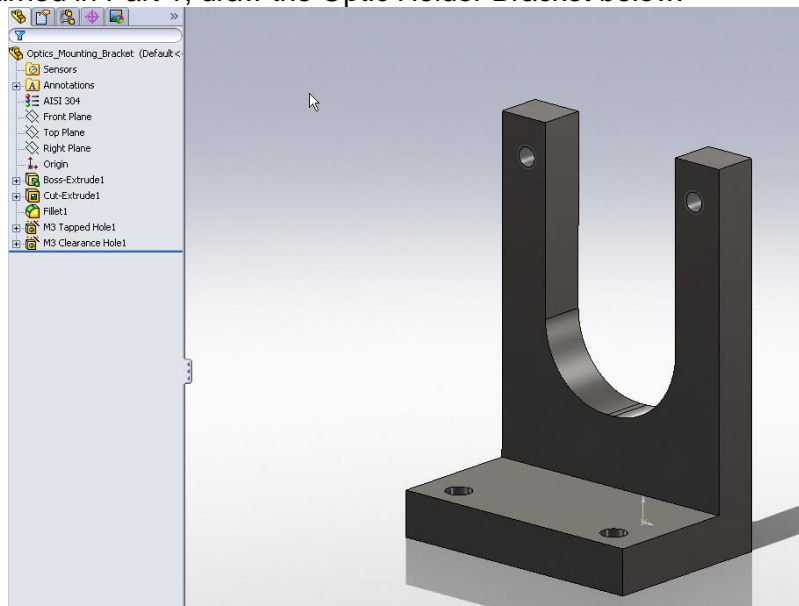
Under the 'FeatureManager Design Tree', you will notice that there is an area for adding a material to the part (see picture overleaf). This is a useful tool such that material information can be displayed on engineering drawings or if the mass of the part is required. By right-clicking on Material <not-specified> you can bring up a pop-up menu, and from there you can add a multitude of materials.

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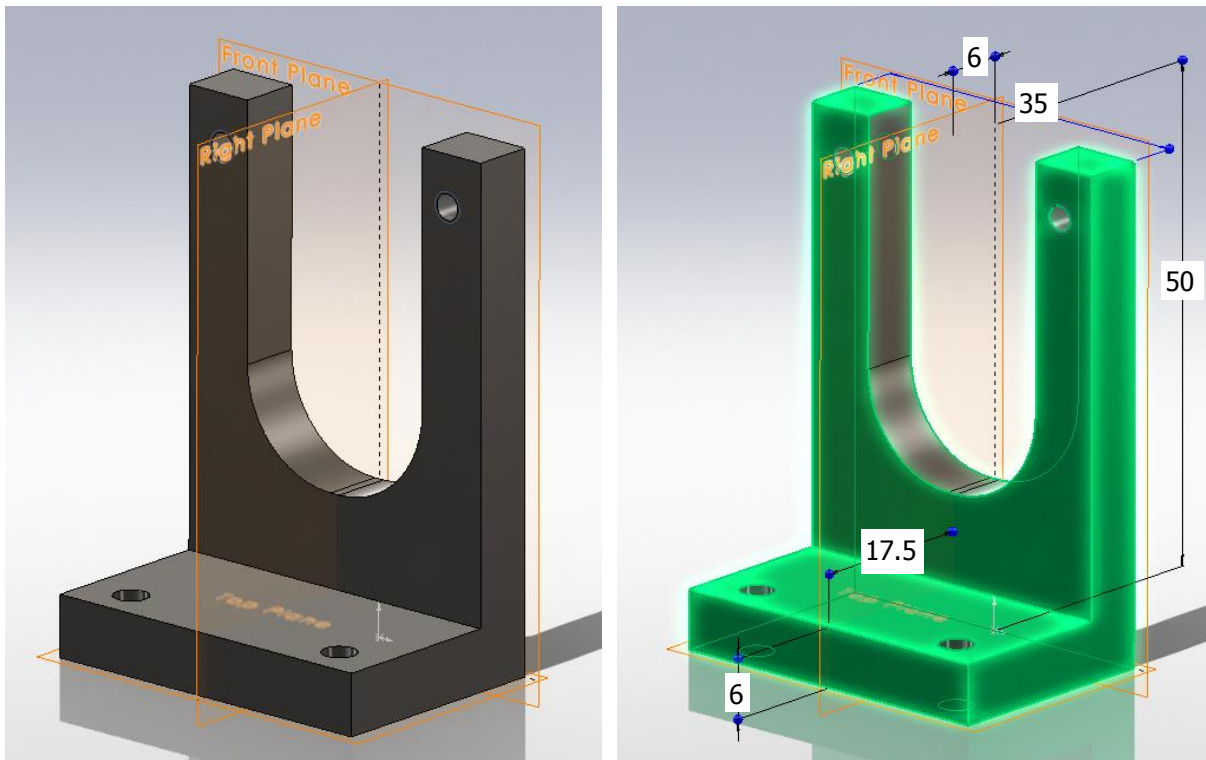
### A.3 Draw up the bracket for the optic holder

Using the tools learned in Part 1, draw the Optic Holder Bracket below:

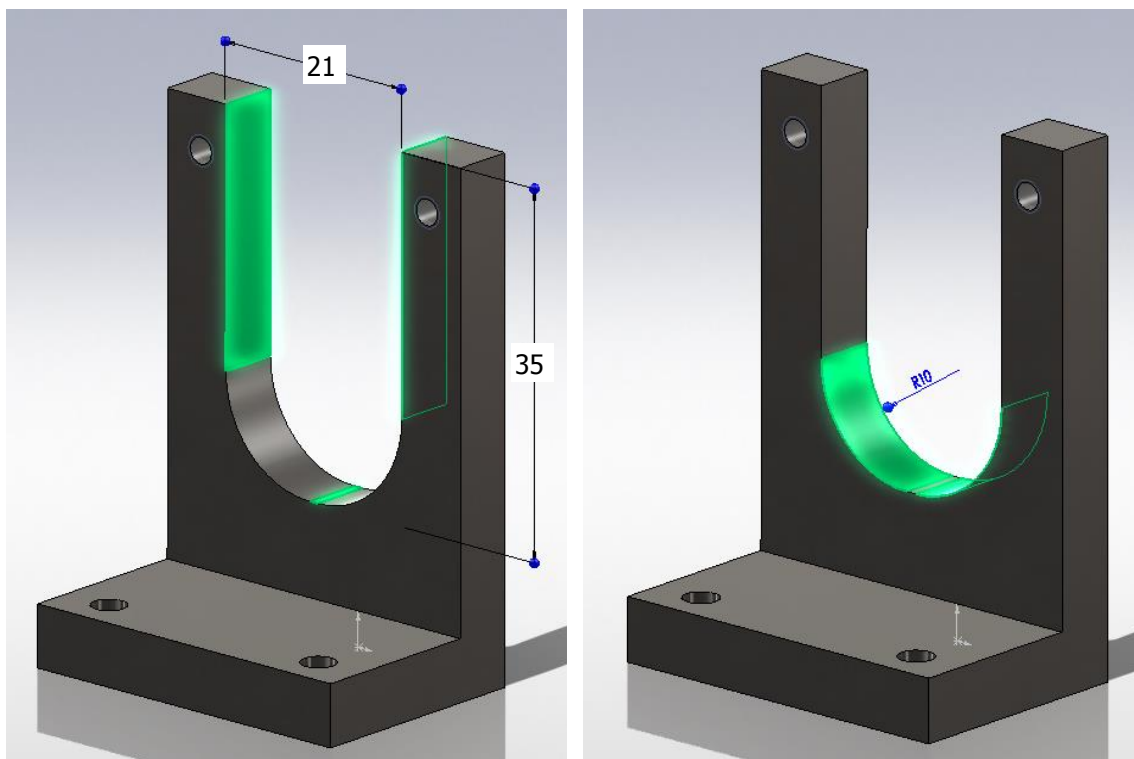


(More pictures and dimensions overleaf)

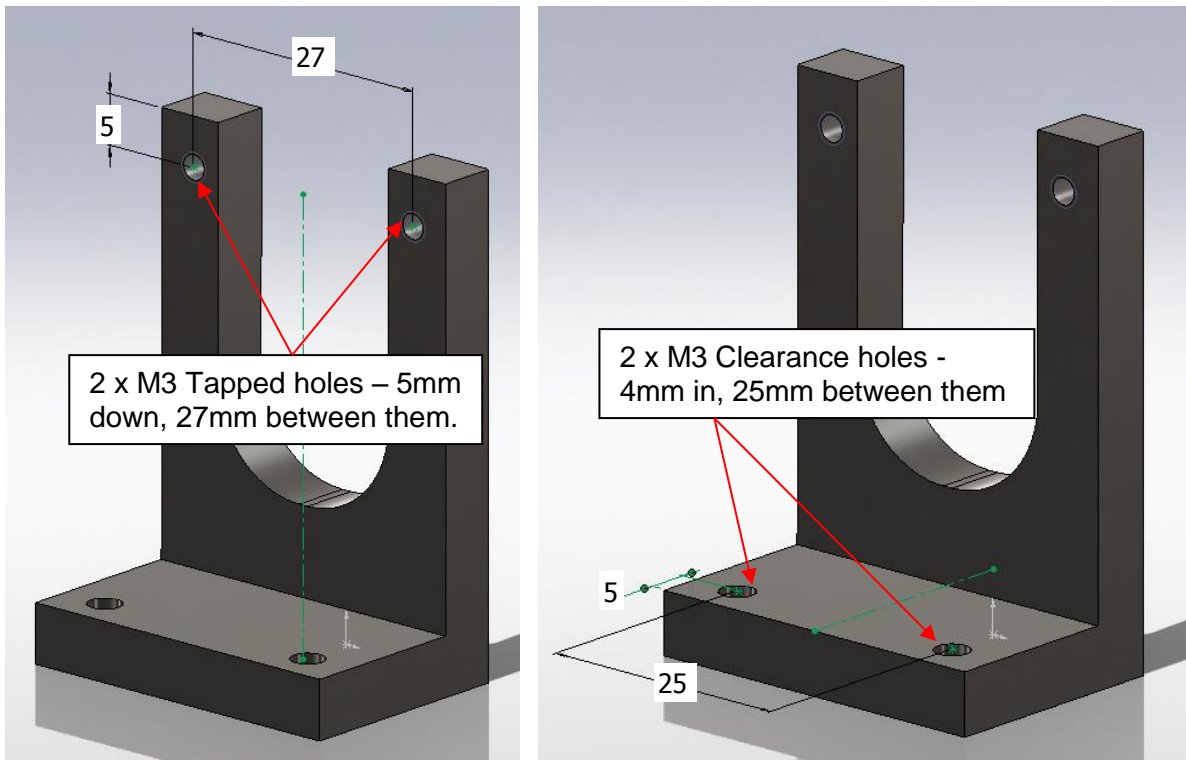
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Hint: Draw L-profile on the right plane then do a mid-plane extrude.

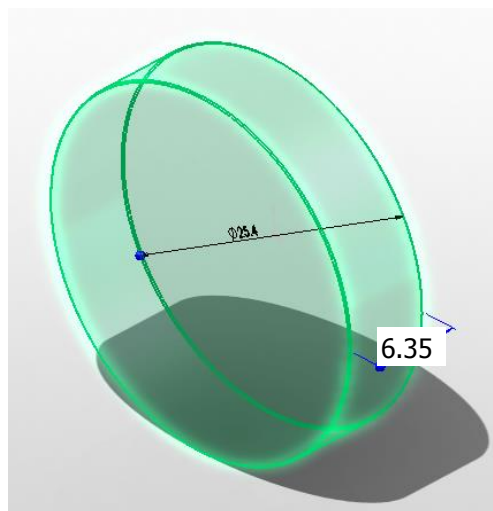


Hint: Cut a rectangular hole, then create fillet to radius the bottom two edges of the hole.



Save this as Optics\_Mounting\_Bracket.sldprt

## A.4 Draw up a one inch optic



Hint: Chamfer edges 0.1mm. Apply a material – ‘glass’.

Save this as: One\_Inch\_Optic.sldprt

## A.5 Add Custom Properties to the parts

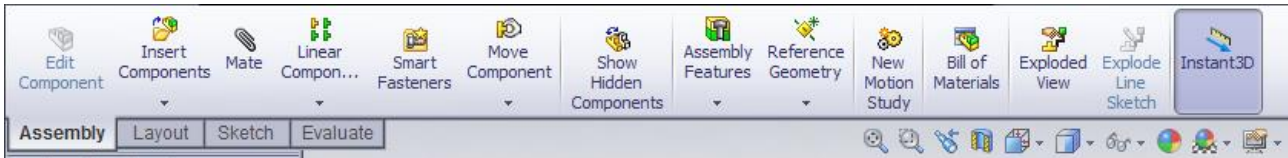
Add custom properties to the mounting bracket and one inch optics parts.

**Now, progress to Tutorial Part 2.**



## TUTORIAL PART 2 – Assemblies

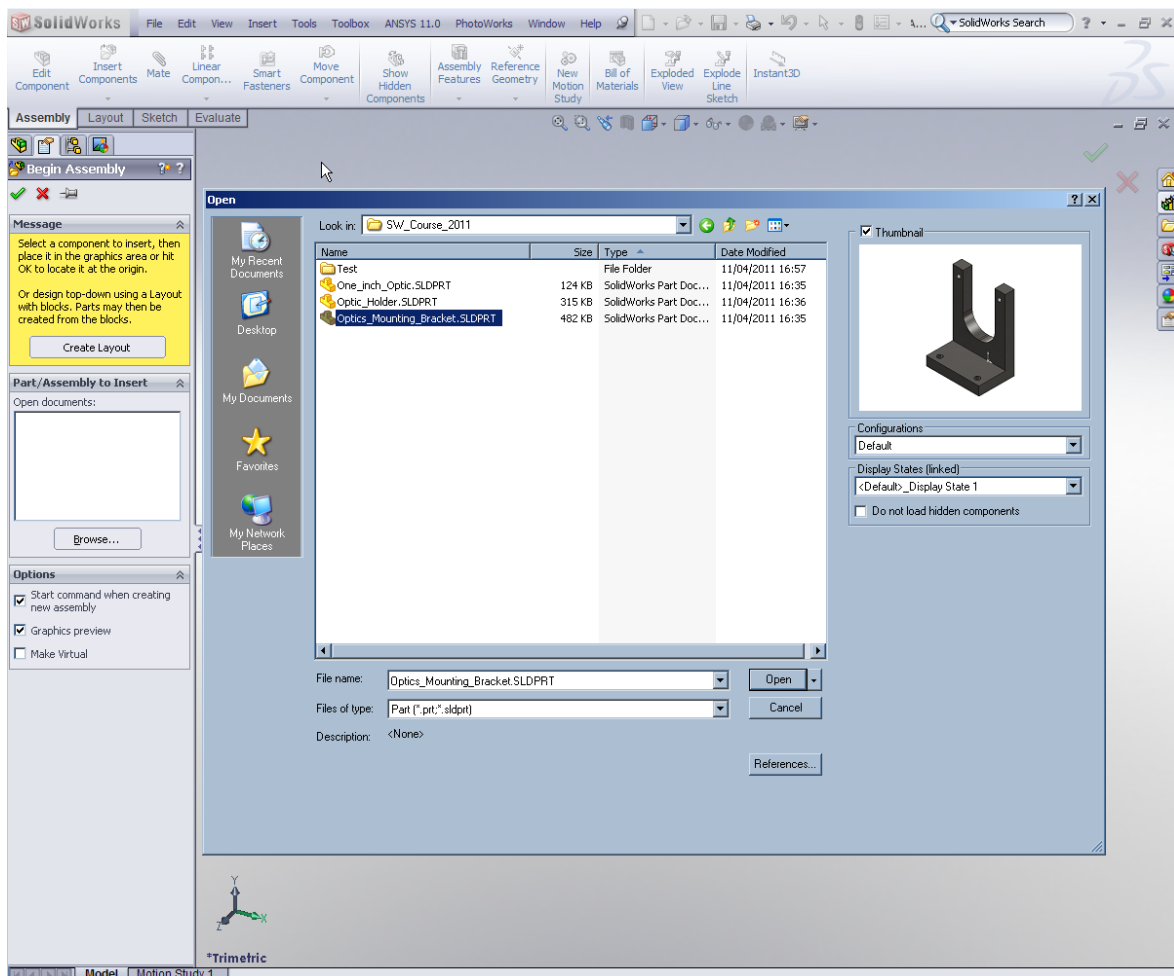
SolidWorks Assembly Environment interface is very similar to the Part Creation interface, however the tools are somewhat different. Whilst it is possible to create parts and sketches within this environment it is not typical to do this so the main toolbars that you will use in this section will concentrate on manipulating pre-existing parts such that can be aligned or 'mated' together to form a visual representation of an assembled mechanism.



1.1 Open a 'New Assembly', in the same manner as you opened a part previously, only selecting UGL\_Assembly this time around.

As you begin in the Assembly Environment SolidWorks will automatically prompt (in the left hand *PropertyManager* Panel) to open a Part or Assembly to Insert.

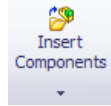
1.2 'Browse...' for the first part of you assembly called Optics\_Mounting\_Bracket.sldprt



1.3 Click 'Open' and then click the green tick button to confirm the action. This will fix the part to the assembly axis.

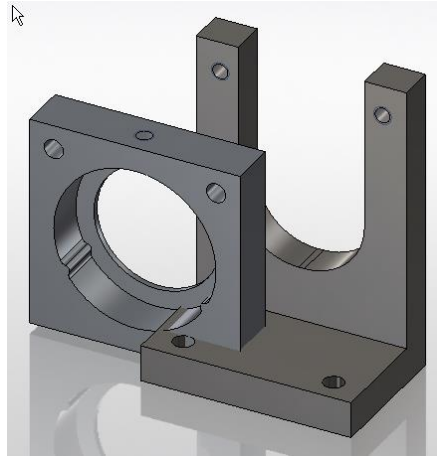
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1.4 Next we would like to bring the second part in to our assembly. To do so, select the 'Insert



components' tool from the Command Manager toolbar.

1.5 Again, 'Browse...' to find the Optic\_Holder part and insert it anywhere in the assembly. Note that the part in its current unconstrained state may be dragged anywhere on the screen with by clicking and holding the left button of the mouse. Notice too that the part can be placed such that it interferes with the bracket, as per the figure below.



## 2 Mating Parts

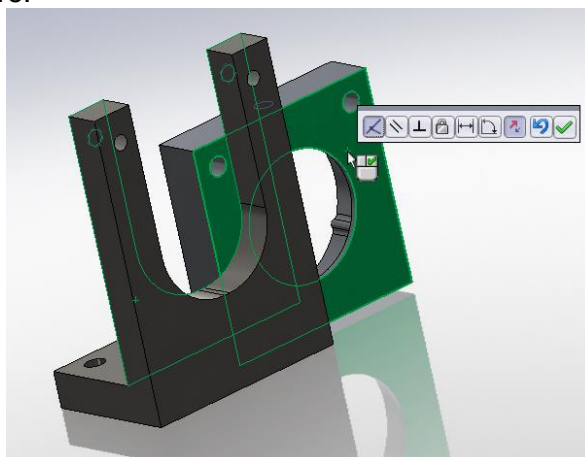
Now we are going to define the position of the optic holder with respect to the bracket using the 'Mate' function.



2.1 Select 'Mate' from the *CommandManager* toolbar.

2.2 Select both the front vertical (U-shaped) face of the bracket and the back face of the optic holder. You will need to rotate the assembly to select the back face of the optic holder.

Notice that the part automatically moves to align the faces of the two parts. Also notice that in the *PropertyManager* panel, SolidWorks has automatically selected a Coincident Mate which is the correct Mate here.

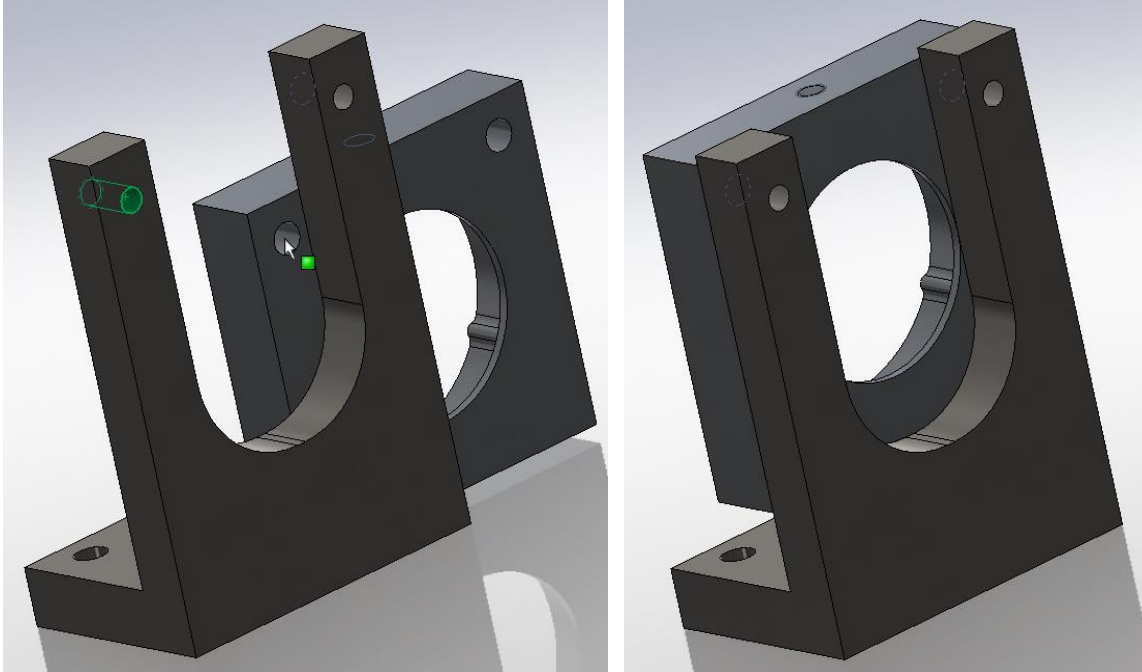


Click the green tick to confirm the action.

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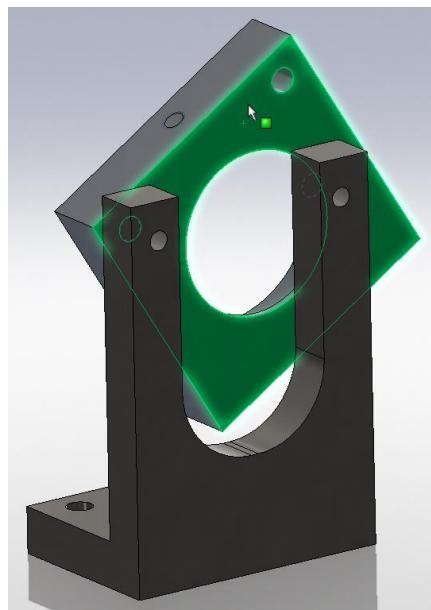
Now we are going to align the screw holes that we created such that the assembly is fully constrained.

- 2.3 Using the Mate tool, select the cylindrical bore of one of the tapped holes in the bracket part. Also, select the corresponding cylindrical bore in the optic holder.



- 2.4 Again the parts should move together automatically and SolidWorks shall select the most likely Mate type – in this case a Concentric Mate.

- 2.5 With one Mate remaining you can now 'grab' and pivot the optic holder about the bracket by left-clicking and trying to pull it away from the bracket. The optic holder should rotate about the one constrained hole and in plane with the front face of the bracket. Doing this will make it easier to see both of the remaining, unconstrained holes.




- 2.6 Finally, to Mmate the two remaining holes on the optic holder and bracket, select each one of them and again add a Concentric Mate.

### 3 Adding screws

3.1 SolidWorks has a large library of standard parts which can be found under the design library in the (hidden) right-hand window panel.



3.2 By selecting the 'Design Library' tool  and then the 'Toolbox' icon you can find a large array of bolts, washers and other mechanical parts for many different international standards.



BSI or ISO Metric standards are the most commonly used in the School of Physics & Astronomy. Those who work on US projects, such as LIGO, may however adopt the ANSI Inch system.

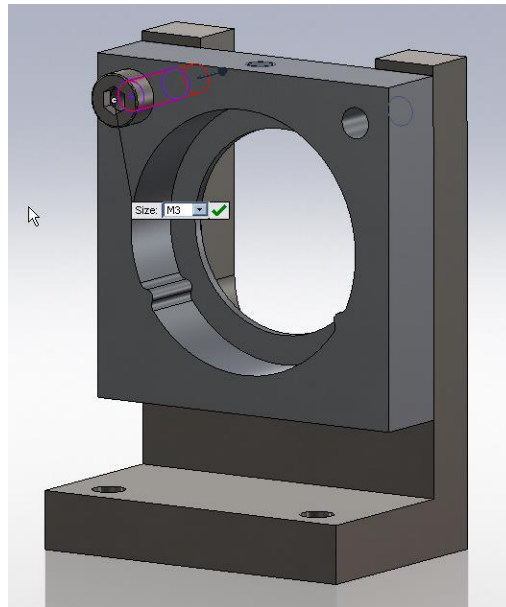
3.3 Go to the ISO section of the design Library, select 'Bolts and Screws'>'Hexagon Socket Head



Screws'> Hex Socket Head ISO 4762

3.4 Press and drag the icon across on to the graphics area, adjacent to one of the two holes bolt holes. When an impression of a screw shows then let go of the mouse button. A menu will open in the *PropertyManager* panel with an interface to adjust the screw to your required size. M3 x 12mm should be suitable for your assembly.

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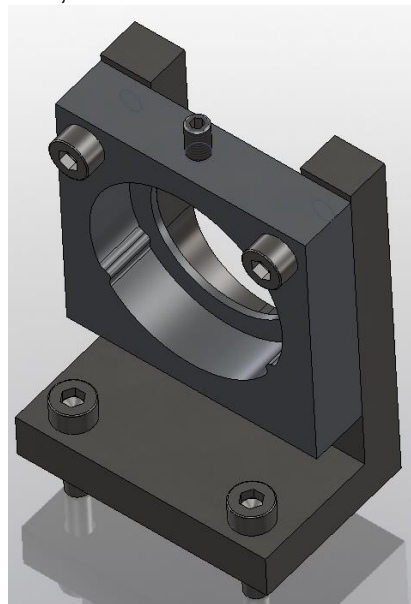
- 3.5 Once the first screw has been added (confirming by clicking the green tick), SolidWorks will automatically give you another screw of an identical size to be added to the assembly. Add this screw by hovering over the second screw hole.
- 3.6 Repeat this action to add two more screws in the two holes on the base of the bracket (these are for attaching the bracket to an optical bench).

To finish, click the red cross or press the Esc key.

- 3.7 We now want to add a set-screw for holding the optic in place so through the design library

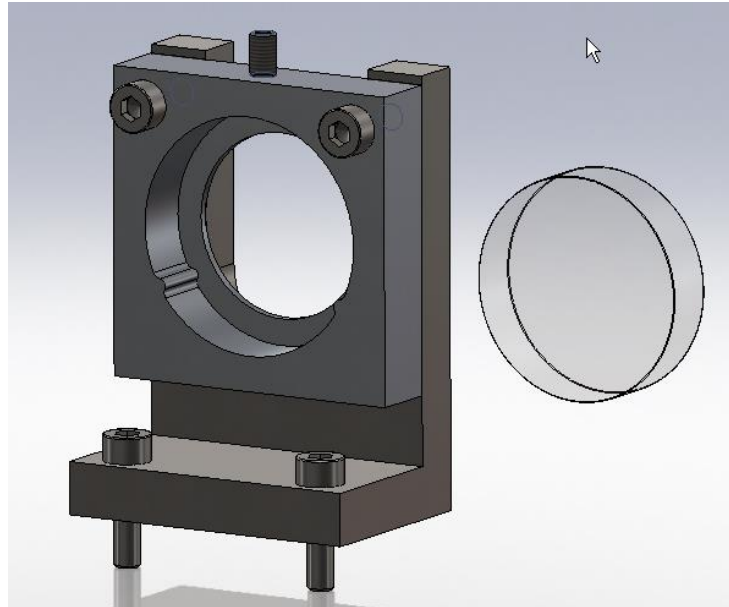


select an ISO 'Socket' Set-screw with a cone point and insert it in the threaded hole at the top of the optic holder. It is best to hover the part over the central bore of the hole on this occasion and it may be beneficial to rotate your assembly and zoom-in so that you can easily highlight the hole. Again, to finish, click the red-cross.



## **PART 2 Extras**

B.1 Add the One Inch Optic to the assembly. Trying first mating using the concentric mate with the large bore of the optic holder. Secondly, suppress or delete that concentric mate and try using 'tangential' mates with the two isostatic mount features (MORE DIFFICULT!) then a second mate between the tip of the set-screw and the barrel face of the optic.



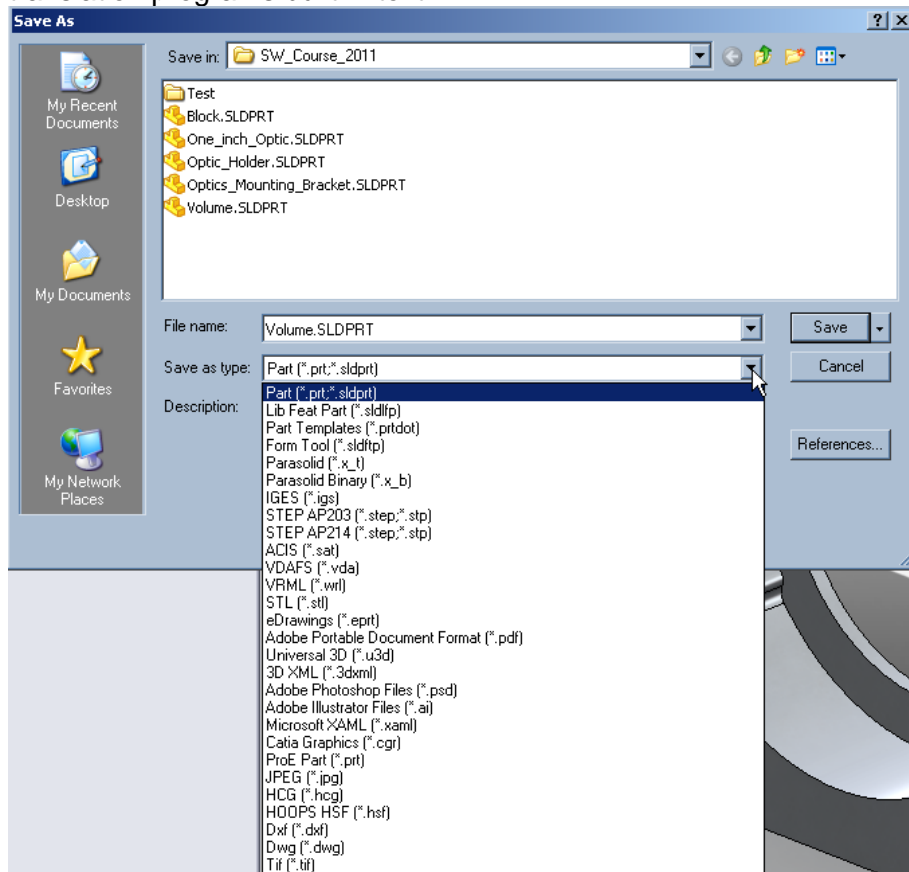
4

B.2 Adding Custom Properties. As per in the parts we can add custom properties that are displayed on the engineering drawing.

**END OF TUTORIAL PART 2**

## TUTORIAL PART 3 – Exporting to ANSYS, CAM or for 3D printing

It is relatively simple to export from SolidWorks for use in other programs. SolidWorks has a number of file translation programs built in to it.



### 1.1 Exporting to ANSYS

If you have both SolidWorks and ANSYS installed on your machine, there are two ways of exporting in to ANSYS.

1.1.1 For ANSYS Workbench, the cleanest way to export is by opening ANSYS directly from SolidWorks. There is an ANSYS add-in in SolidWorks which can be activated by going to the Tool> Add-Ins... menu.

1.1.2 For ANSYS Classic and ANSYS Workbench on a machine that does not have SolidWorks installed, the SolidWorks part or assembly can be saved as a .SAT file by simply using the File>Save As... function.

### 1.2 Exporting for Computer Aided Manufacture (CAM) or 3D Printing

For CAM manufacture, files should be saved as STEP, IGS, STL or DXF. Note that DXF is a 2D format so you will be asked which view of the 3D part that you would like to export (i.e. TOP, SIDE, BACK, FRONT, etc). An STL file is a 3D format commonly used in 3D printing.

Notes: - Material properties from SolidWorks shall not automatically transfer to ANSYS.  
- Make sure, if exporting as .SAT file the settings are such that the export is completed in meters, otherwise the part/assembly will come in to ANSYS with the wrong scale. The 'settings' can be found as you Save the part as a .SAT file.  
- It is good practice to check that your exported file has been created correctly. This can simply be done by reopening it in SolidWorks.

END OF PART 3

## TUTORIAL PART 4 – Engineering Drawings


The engineering drawing is an important document that allows your part to be manufactured to the dimensions, tolerances and finish standards that you require. All jobs submitted to a manufacturer, whether this is done electronically using a 3D solid model file format (sldprt, SAT, IGES, STEP), an engineering drawing should also be supplied. The problem with 3D solid model formats is that they do not typically contain information about dimensions or tolerances so for example, if a file translates incorrectly or if a manufacturer uses a different tolerance level to what you require, you may end up with a part that does not have the same dimensions that you had intended it to have.

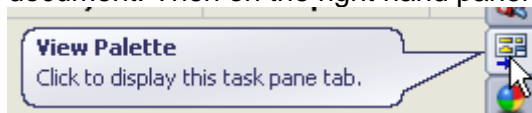
SolidWorks makes engineering drawings relatively easy as it recognises the various international drawing standards such as ISO (International), BSI (British), ANSI (American), DIN (German), JIS (Japanese), etc. These can be set-up in the SolidWorks Options menu but typically for the work done in the School we will use the ISO standard.

All engineering drawings should have at least three views, e.g. front, top and one side view, but given that you have created a 3D model it is useful to add an Isometric view of the part too to make it easier for the manufacturers to visualise.

### 1 Starting an engineering drawing

There are two ways of beginning an engineering drawing:

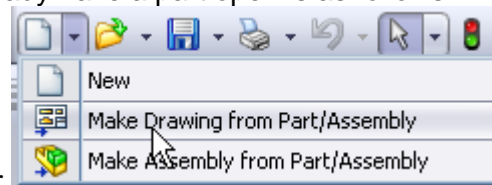
- 1.1 Go to File>New> (or ) and select the UGL\_A4\_landscape (or UGL\_A3\_landscape) document. Then on the right hand panel of your screen select the 'View Palette' tab




and browse for the part that you want to draw. Front, Top, Side views of the part you select should now appear in the right hand panel. You should now drag and drop one of these views on to your drawing sheet and then other views will automatically appear from this view.

OR

- 1.2 the second way of starting a drawing is if you already have a part open is as follows:

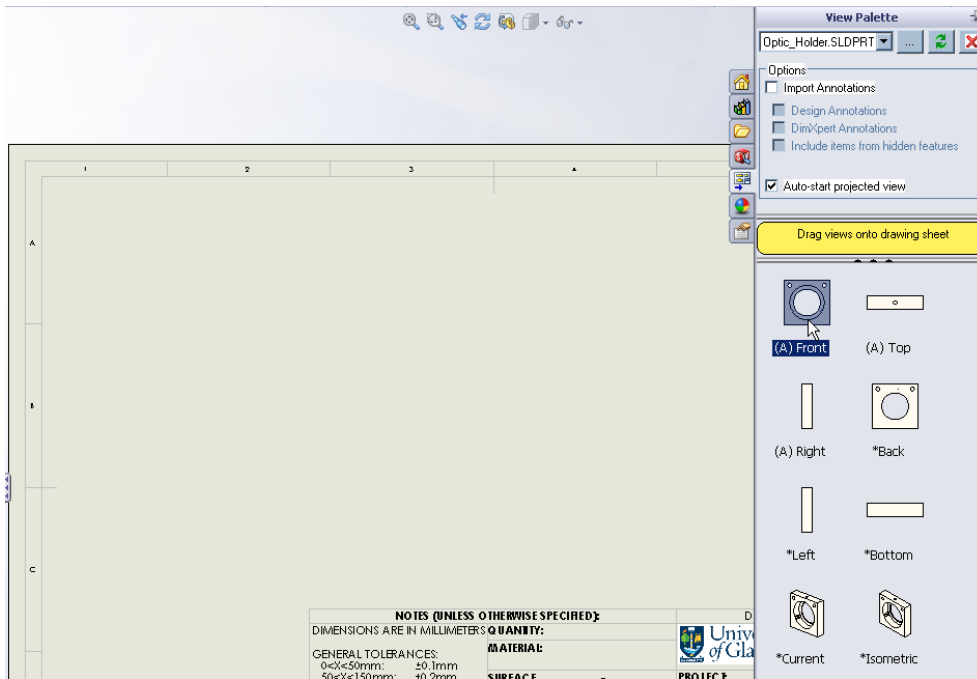


- 1.3 Go to File>Make Drawing from Part/Assembly (or  from the drop down menu), select the drawing type that you want and then the views should open up automatically in the right hand panel. As per the first method for beginning a drawing, drag one of these views on to the drawing and then layout the other views that you require.

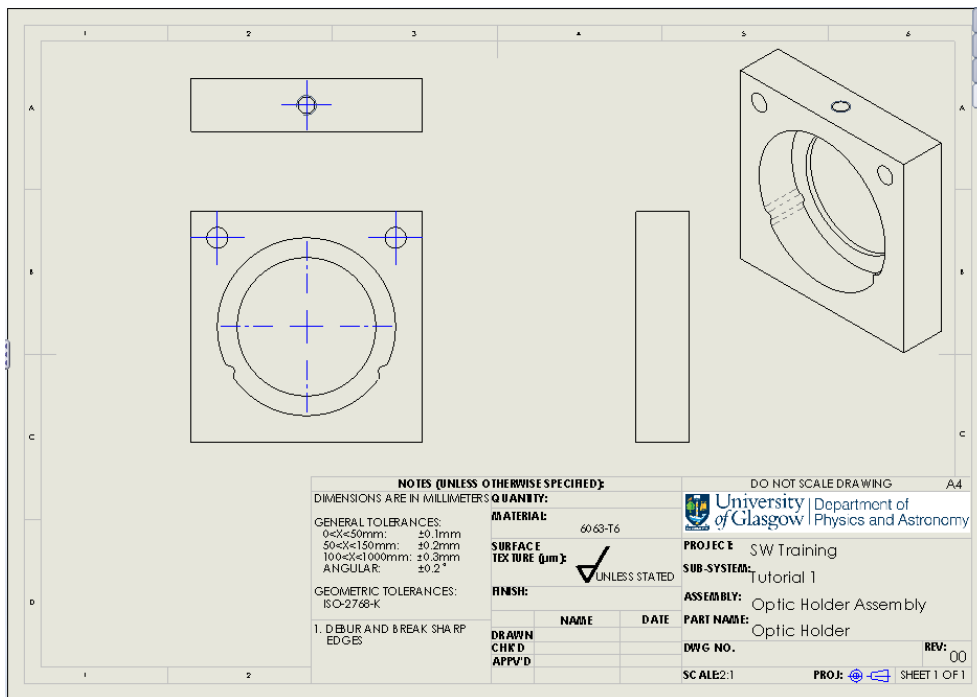


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The 'View Palette' panel – drag and drop views from here




Front view and auxiliary views laid out on the drawing sheet

You are now ready to begin dimensioning your part. The key to dimensioning any part is not to over-dimension – make sure that the position of all features are determined from one clear datum but do not dimension to the same feature twice! Also, make sure that the drawing has been completed on the correct size of paper for the part or assembly – if what you are drawing up has many fine details, scale up and use a larger drawing size, or use detail views to show fine detail.

Before submitting an engineering drawing for manufacture it is always best to have it checked by an experienced engineer or draftsman to be sure that all dimensions and details have been included.

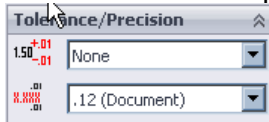
## 2 Adding linear dimensions

2.1 Linear dimensions are added like they are in the part creation environment of SolidWorks.

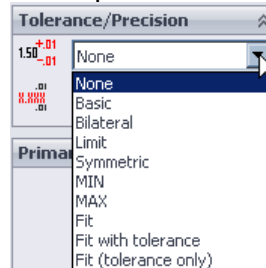
Select 'Smart Dimension'  from the 'Annotation' menu of the *Command Manager* toolbar. Now, simply select edges, vertices or centrelines to add dimensions to your part.

If you want to add a tighter tolerance than those shown in the 'General Tolerances' in the title area of the drawing this can be done in the following way:

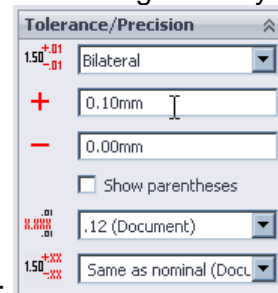
2.2 Firstly deselect 'Smart Dimensions' and select a line or feature to be dimensioned. When selected, you will notice that a set of tools will appear in the PropertyManager panel.



2.3 Under Tolerance/Precision there are two pull down menus, the first of



which allows you to choose a variety of tolerance types. Most commonly used are 'Symmetric' or 'Bilateral' which essentially let you set how much you would like to have the part over and under the nominal size. Lets say for this part our height is very critical

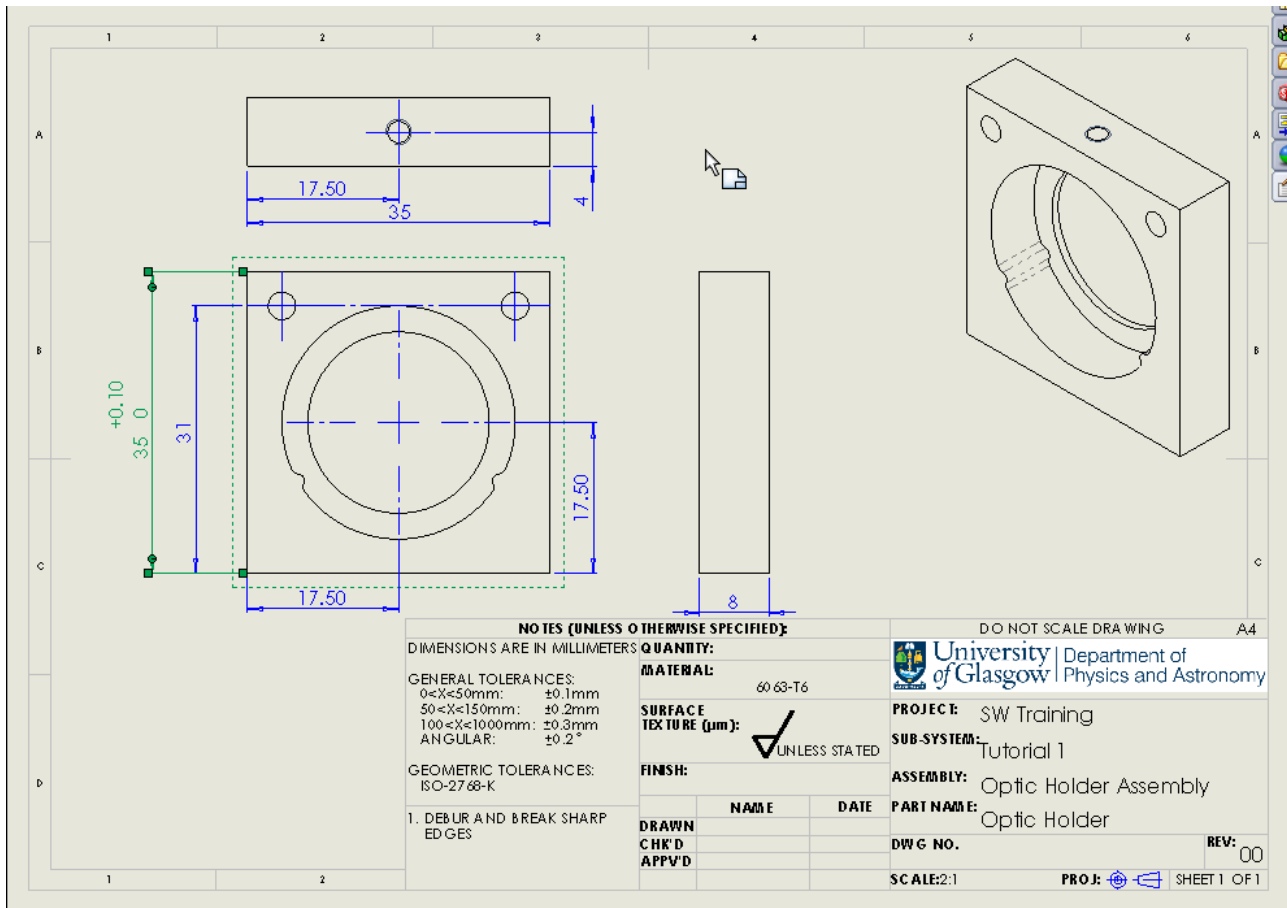


so we shall set a bilateral tolerance of +0.1, -0.0, as shown here:

Once the linear dimensions have been added the drawing should look something like the figure below:

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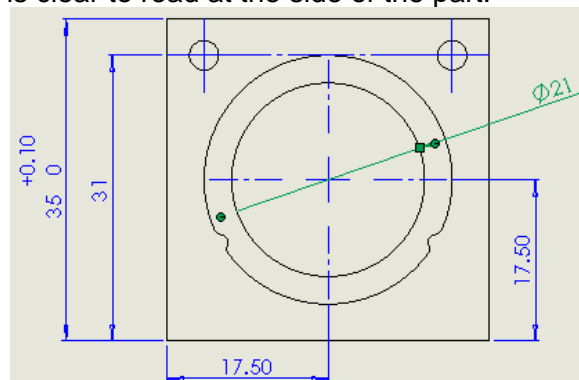
*Drawing with Linear Dimensions added*

### 3 Adding radial or angular dimensions

Adding radial or angular dimensions works in the same manner as adding linear dimensions as SolidWorks, essentially, will know if the line that you select requires a radial or angular dimension. We may, however need to add some extra centre marks and construction lines to define the angles between the parts.

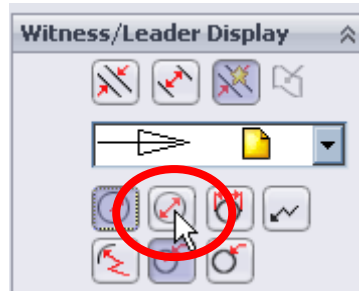
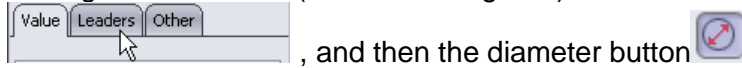
#### 3.1 Radial Dimensions

Select the 'Smart Dimension tool' and then select the inner bore hole and drag out the dimension such that it is clear to read at the side of the part.

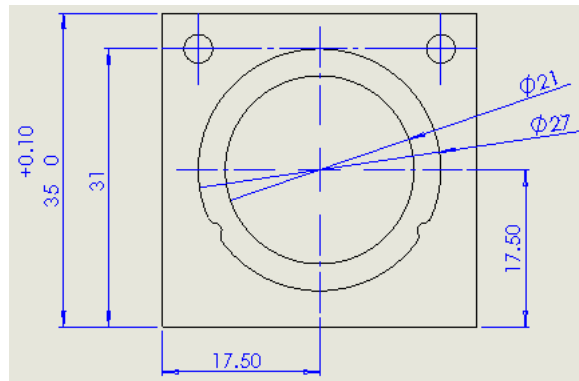


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3.1.1 The next, larger bore hole with the isostatic features. You will notice when you try to dimension this a radius appears rather than a diameter this is because it is not a complete cylinder. Whilst having a radius dimension here is okay, we can change this to a diameter for more clarity by selecting the dimension (it should turn green) and then in the LHS panel, select the 'Leaders' tab



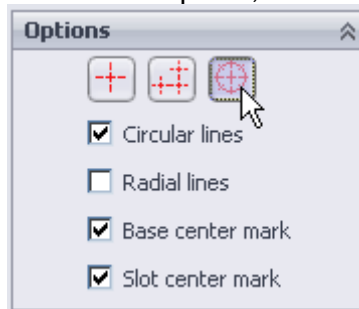
3.1.2 The sketch should now look like this:



### 3.2 Angular dimensions

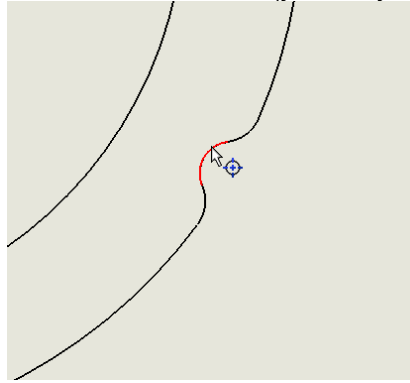
To dimension the two features that make up two of three points of our isostatic mount we need to add some centre marks. As these have cylindrical profiles on their outer surface, we can apply centre marks and by using the correct tool we can include circular construction lines and radial lines that can then be dimensioned.

3.2.1 Select the 'Centre Mark'  tool under 'Annotation' **Annotation** of the Command Manager toolbar. In the left hand panel, select the 'Circular Centre Mark' tool .




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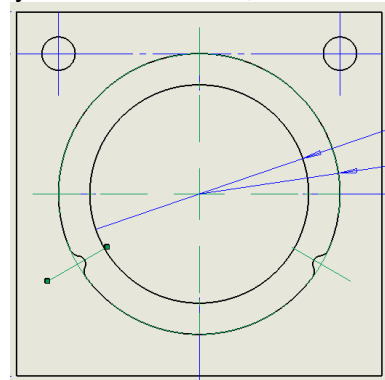
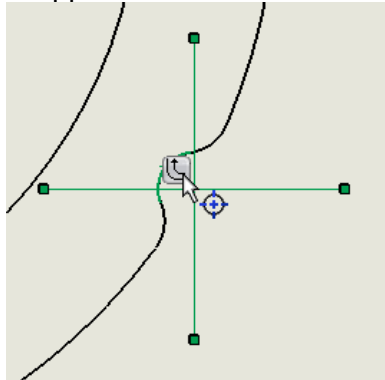
3.2.2 Now, select the outer edge of isostatic mount feature (you may wish to zoom-in to



make sure you catch the correct edge)

and then select

the propagate icon  that appears above the centre mark you have inserted, to make



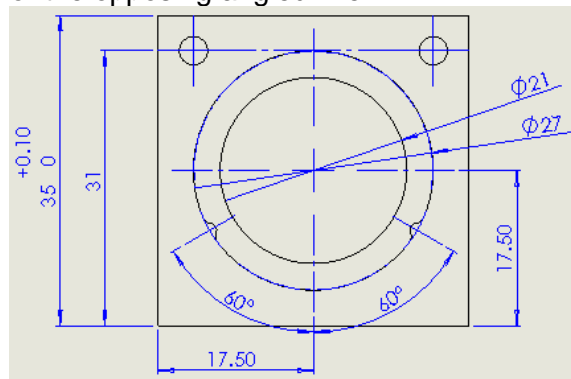
your radial centre marks.

->

3.2.3 Press the green tick to confirm the action.

3.2.4 You may now dimension to the radial lines of these features to show what angle they are positioned with respect to the vertical centre line of the large bore. To do so, select 'smart dimension' and then select both the vertical line and the radial line. This should now pull out a dimension of a 60 degree angle.

3.2.5 Repeat this for the opposing angled line.



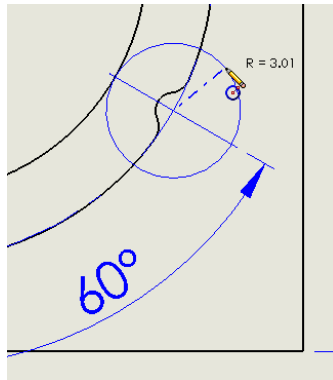
## 4 Section and Detail Views

Now that there are a number of dimensions on this view of the drawing the kinematic points are quite hidden behind the dimensions. For dimensioning these smaller features it is best to add a Detail View (a zoomed-in view of a small area of the drawing).

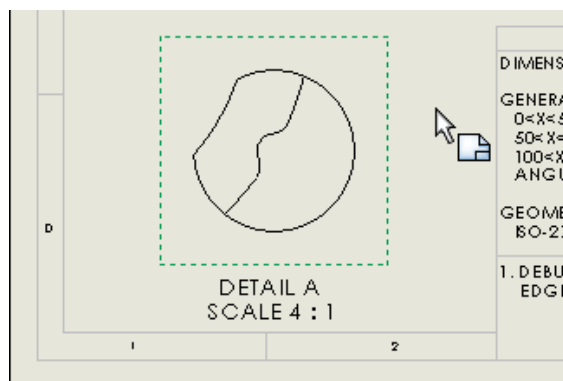
### 4.1 Detail Views

4.1.1 To add a detail view click the 'View Layout' tab  in the Command

Manager Toolbar and then find the 'Detail view' tool . Now, with this tool, draw a circle

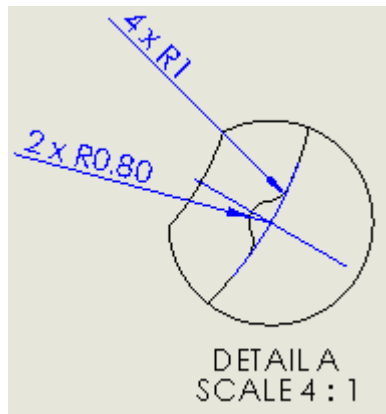


over the isostatic mount point and then drag the detail view in to free space. SolidWorks will choose the scale for you however if you prefer something different, this can be edited in the left hand panel.



4.1.2 Now add dimensions to this view using the 'smart dimension' tool. Again, for clarity you should add a centre mark in the detail view, too.

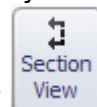
4.1.3 To save doing a second detail view for the opposing isostatic point you may add the note '2x' to your dimensions in your first detail view. This will be interpreted accordingly by the manufacturer.



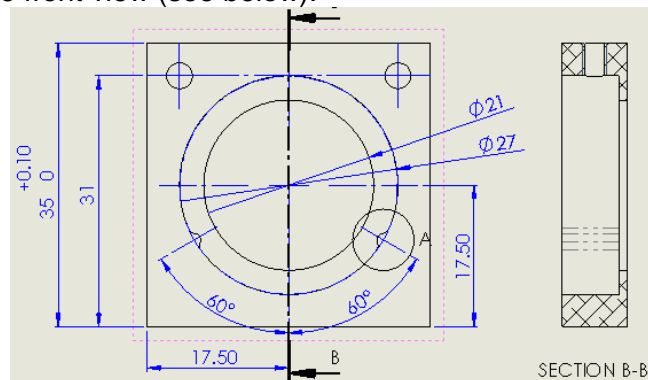
4.1.4 One detail of our drawing that the manufacturer cannot see from the three views of the optic holder, is the thin disk at the base of the hole. To highlight and dimension this feature we can use a cross-section tool.

## 4.2 Section Views

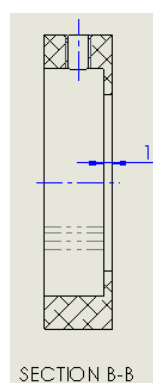
4.2.1 To add a section view click the 'View Layout' tab  in the Command



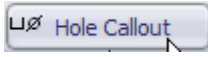
Manager Toolbar and then find the 'section view' button. Next, using this tool, draw a vertical line from a little above the centre point of the top edge of the front view, to a little below the bottom edge of the front view (see below).



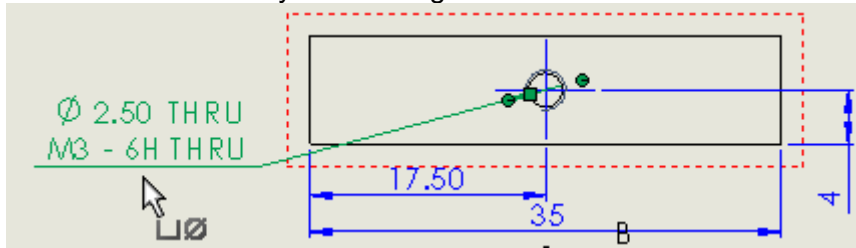
4.2.2 Now dimension the thin disk feature. Centrelines should be added to show the cylindrical profiles in the section view.



## 5 Dimensioning holes created using the 'Hole Tool'

Three holes were created using the hole tool in this part. By doing so, SolidWorks adds the necessary detail on the engineering drawing for the hole to be manufactured appropriately. To display this information we use the 'Hole Callout' tool  under the 'Annotation' Tab of the Command Manager.

- 5.1 Select the Hole callout tool. Select the hole at the top of the part. Drag the dimensional instruction out to a clear area on your drawing sheet.

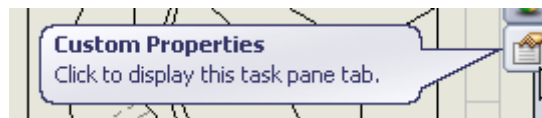


- 5.2 Repeat for the one of the two holes on the front face of the part. Note that as both of these holes were created together and are of the same size, SolidWorks groups the instruction for them in one annotation.

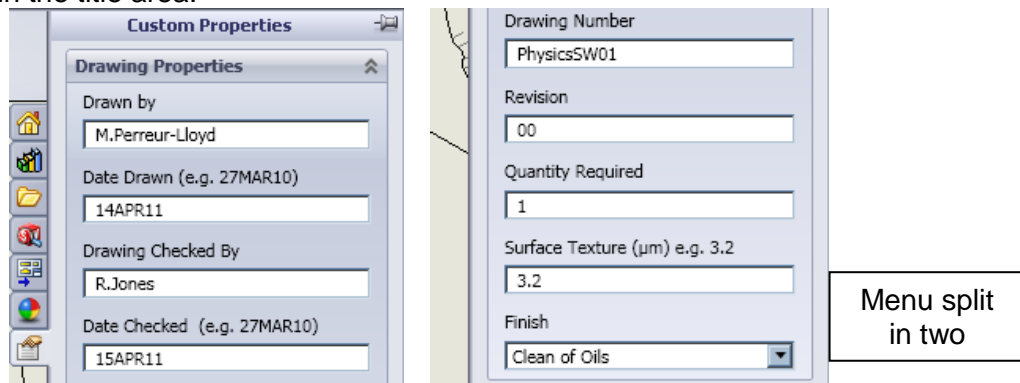
$2 \times \text{Ø } 3.20 \text{ } \nabla 10.25$

## 6 Adding Custom Properties

Like when creating the parts and assemblies, from the right hand tabs on the SolidWorks GUI you may add custom properties to the engineering drawing that shall automatically fill-in to the fields in the title area.



- 6.1 Click the Custom Properties Tab, and fill in all details. Notice that when the fields are filled in they appear in the title area.

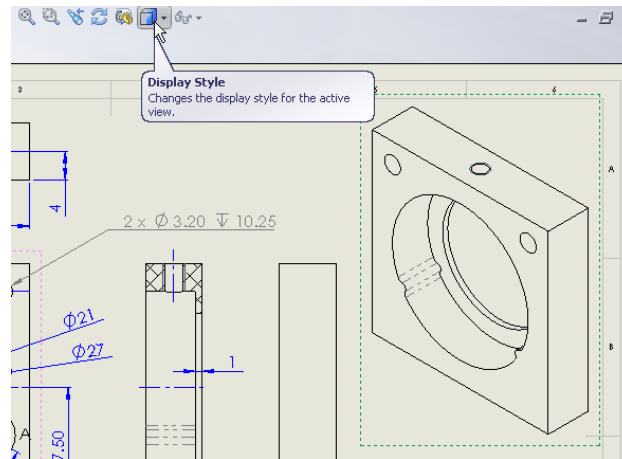





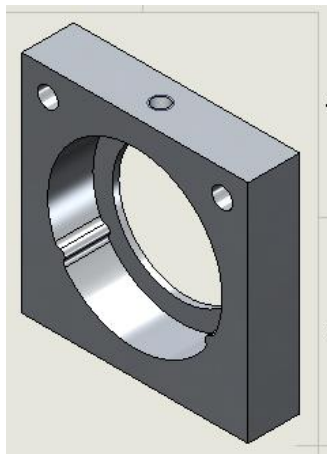
## 7 Final touches


The drawing is finished as far as the dimensioning and detailing is concerned but there are a couple of things that can be done to make things even clearer. Firstly we will change the isometric view to give it solid colour and secondly we will change the colour of the centre lines to make them stand out a little better.


7.1 Select the Isometric view, and then select the 'Display style' icon at the top of the drawing area.



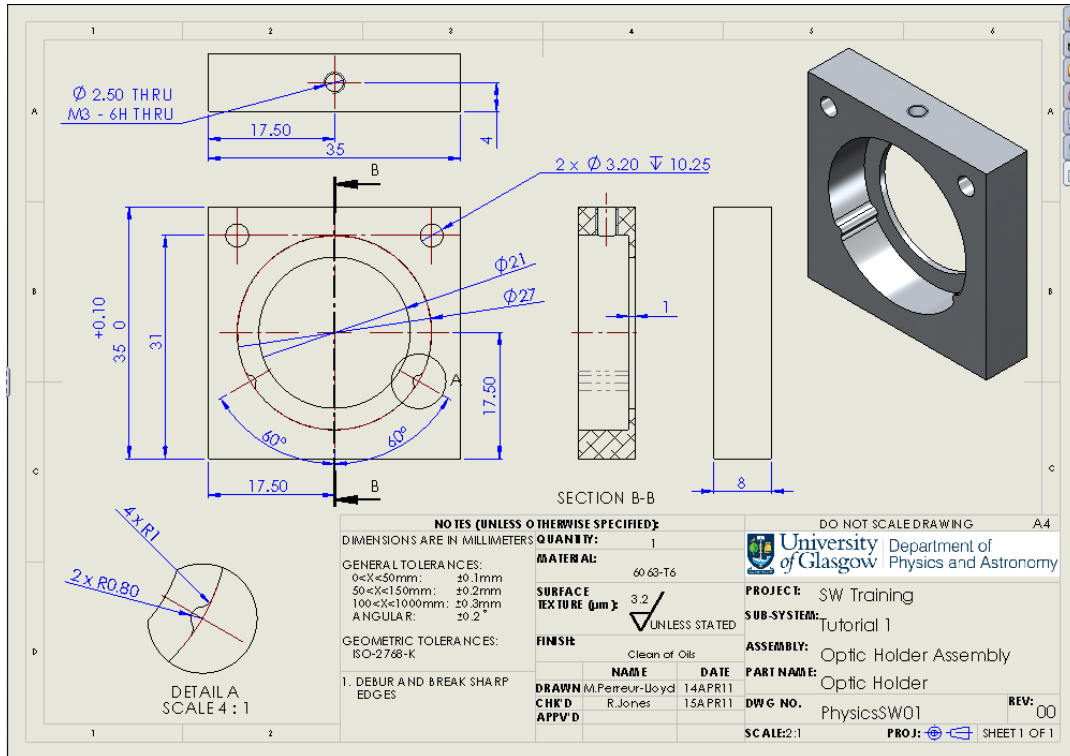
7.2 Pull down the 'Display style menu and change it to the solid colour with lines . The Isometric view shall now look solid making it easier for a manufacturer to understand the geometry.



7.3 To change the centre line format we firstly need to open a new toolbar. To do so, right click any menu area and select the 'Line format' toolbar  (alternatively, go to Tools>Customize... and open the toolbar this way).

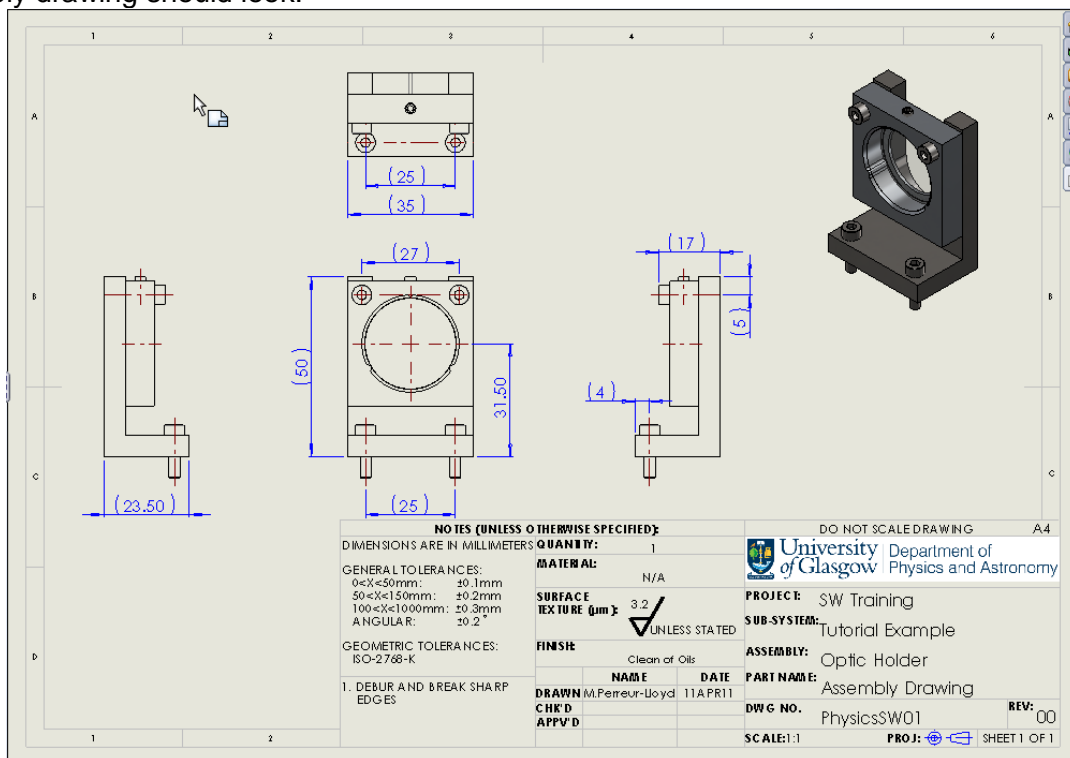
7.4 Now select all of the centrelines, you should hold the shift button down as you do this. Then, finally, select the 'line colour' icon  and change the centre marks and lines to, for example, red.

7.5 Lastly, change all dimensions to blue (if they are not already). Your drawing is now complete. And should look like this (see below):




## 8 Drawings of Assemblies

Assembly drawings do not need to have quite so much detail as part drawings. This is because, in general they are not used for manufacture and are more there to show the overall layout and assembly of the parts in a formal manner. The drawing below gives an example of how an Assembly drawing should look.



# SolidWorks Training for School of Physics & Astronomy Staff and Postgraduates

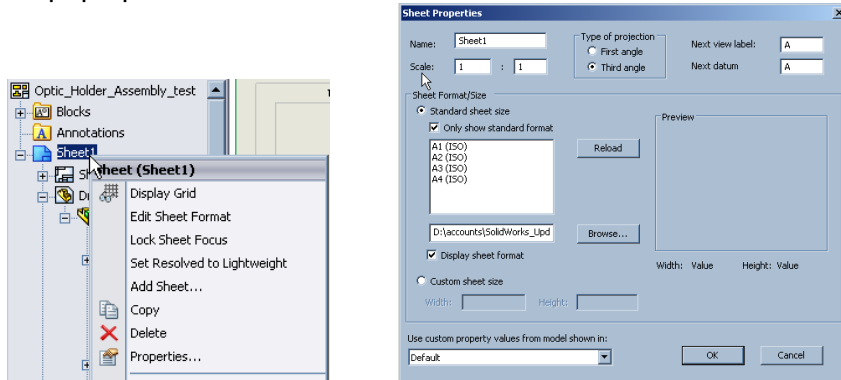
## Introductory course for beginners and false-beginners

Notice that all dimensions are shown with parentheses (brackets) to signify that they are reference dimensions and are not to be used to manufacture the parts. These parentheses  can be added from the left hand side bar when the dimensions are added to the drawing.

## 9 Changing the drawing scale

SolidWorks automatically chooses a drawing scale for the parts on the drawing. Should the drawing scale not be best suited to the parts/assemblies being dimensioned this can be changed in the following way.

9.1 Right click 'Sheet1' in the *FeatureManager Design Tree*. Select 'Properties' and then change the scale in the pop up window.



**END OF TUTORIAL PART 4.**