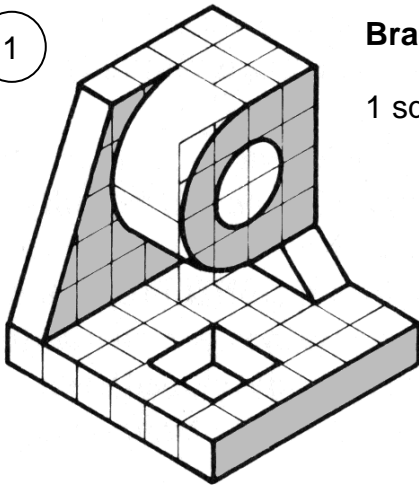


Introduction to SolidWorks - worksheet.

1 Extrude.

1



Bracket.

1 square=10mm

Material: ABS Plastic
Mass: (84g)

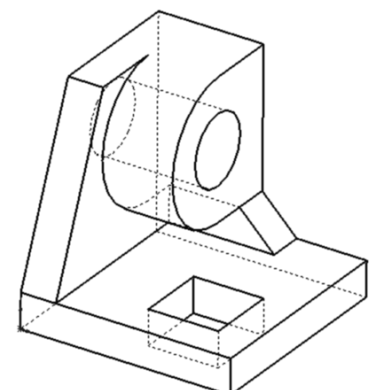
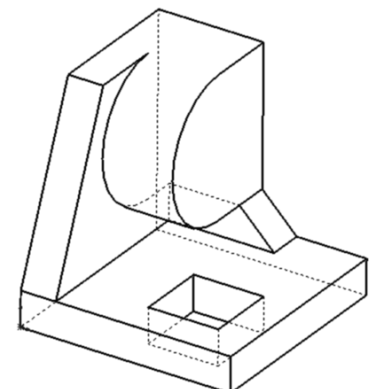
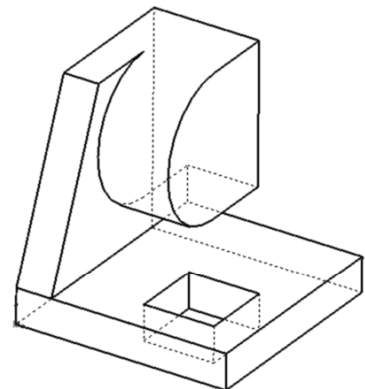
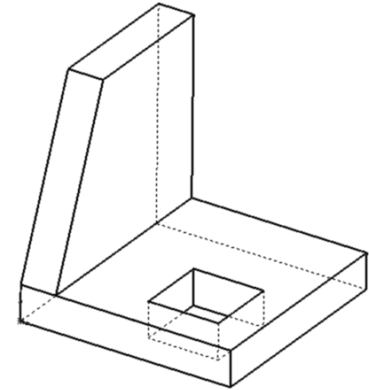
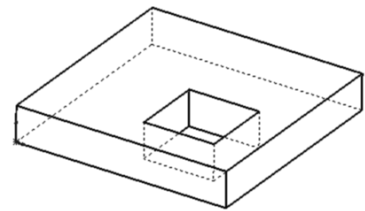
For this example we will build up the model in the stages shown on the right here.

The first four stages will each involve the following operations:

- selecting surface or plane
- creating a **sketch** on that surface/plane
- creating an **Extruded Boss/Base** solid feature using that sketch.

The last stage will be the same only:

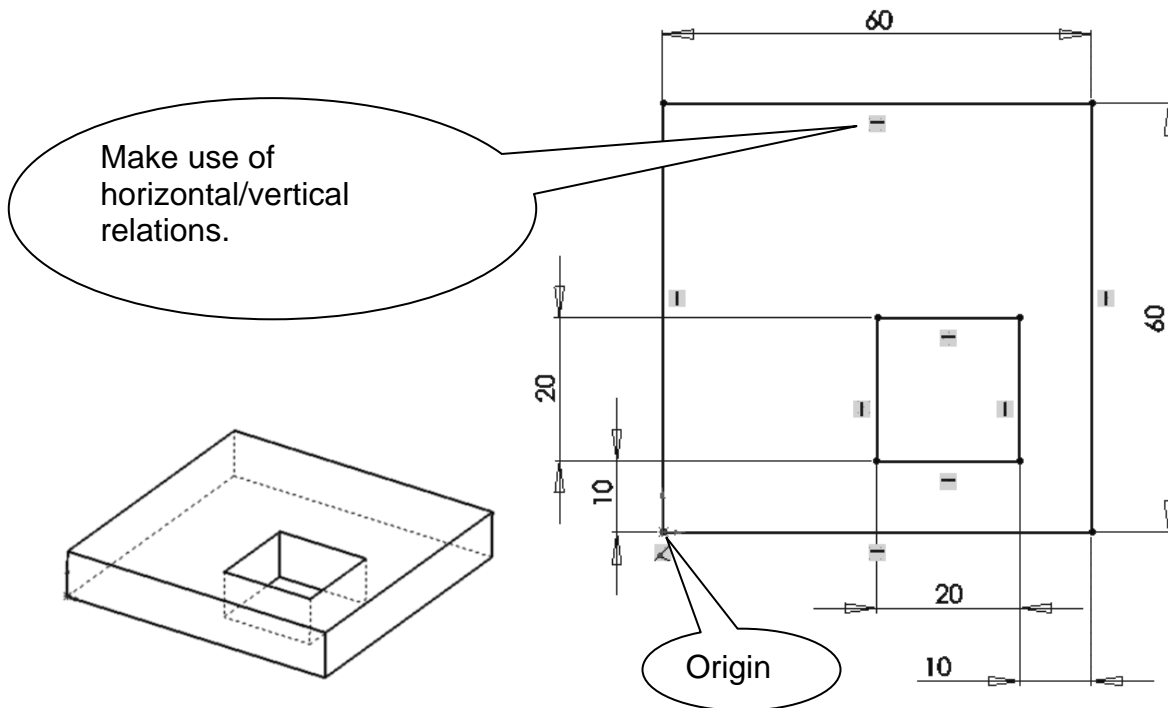
- creating an **Extruded Cut** feature, where solid material is cut (removed) from the model. This makes the hole.



Feature 1)

Making the flat bottom slab of the bracket. This will be the base or starting feature.

- Select the **top** plane for sketching and start a sketch.
- Use the **origin** as the start point of sketching, and draw the sketch below.
This sketch will define the cross section which will be extruded to create the base.

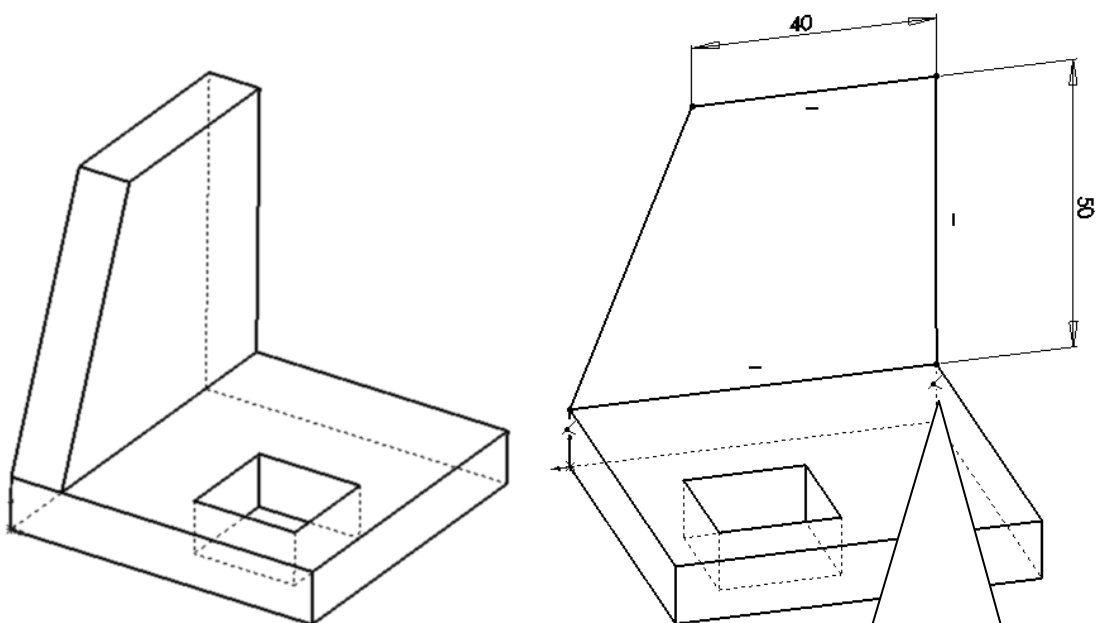


- Create the **Extruded Base/Base** feature using this sketch.

Feature 2)

Making the vertical web.

- Select the appropriate back face of base slab for sketching.
- Create the sketch shown here. Make good use of relations as you sketch.
- Create the **Extruded Base/Base** feature using this sketch.



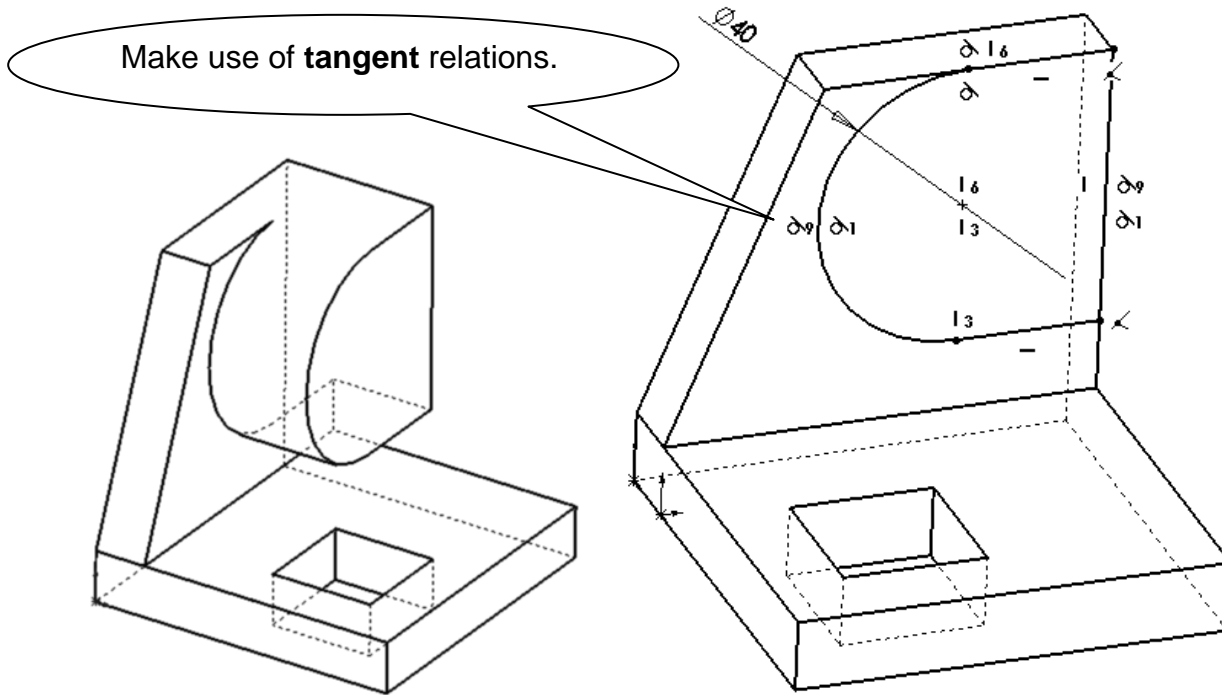
Make use of **coincident** relations with existing geometry.

Feature 3)

Making the lug.

- Select the front face of the vertical web for sketching.
- Create the sketch shown here. Make good use of **relations** as you sketch.

This sketch is a bit more complicated. To add sketch **relations**, such as to make two lines/arcs **tangent**, select both of them at the same time, using the **Ctrl** key and select the appropriate relation from the Properties window.

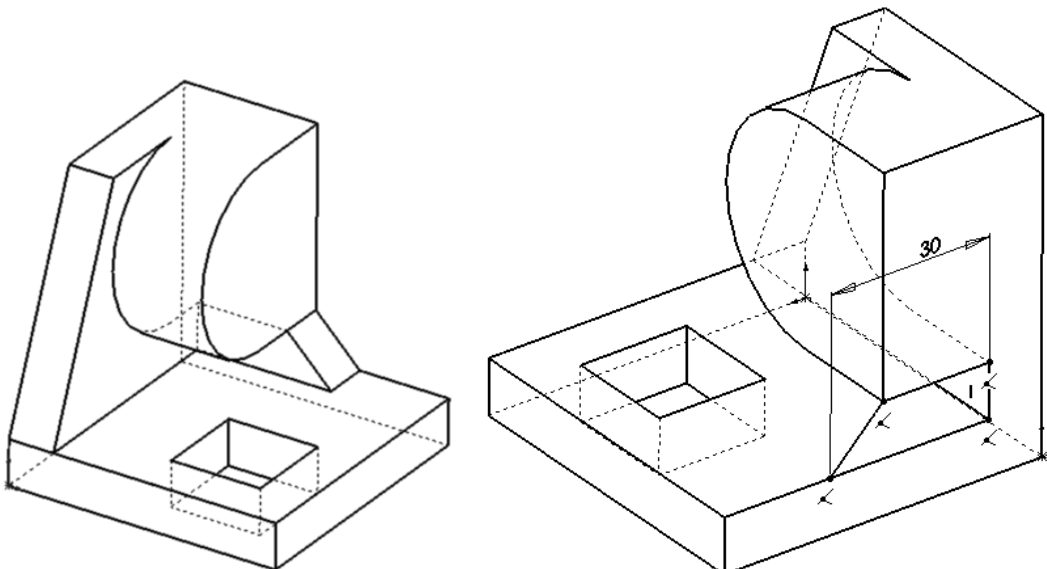


- Create the **Extruded Boss/Base** feature using this sketch.

Feature 4)

Making the small web connecting the base to the lug.

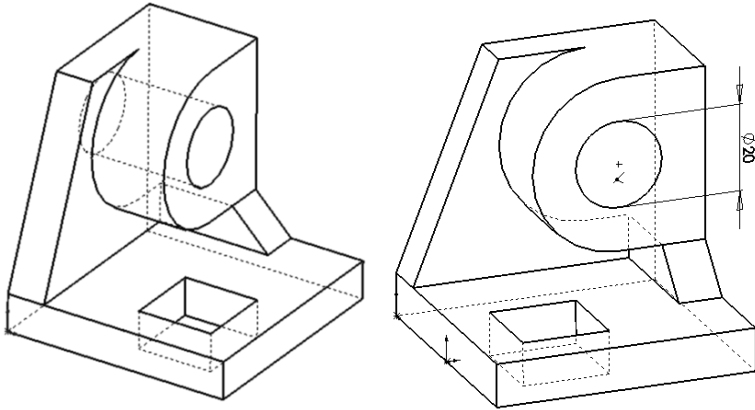
- Select the back face of the lug for sketching.
- Create the sketch shown here. Make good use of relations as you sketch.
- Create the **Extruded Boss/Base** feature using this sketch.



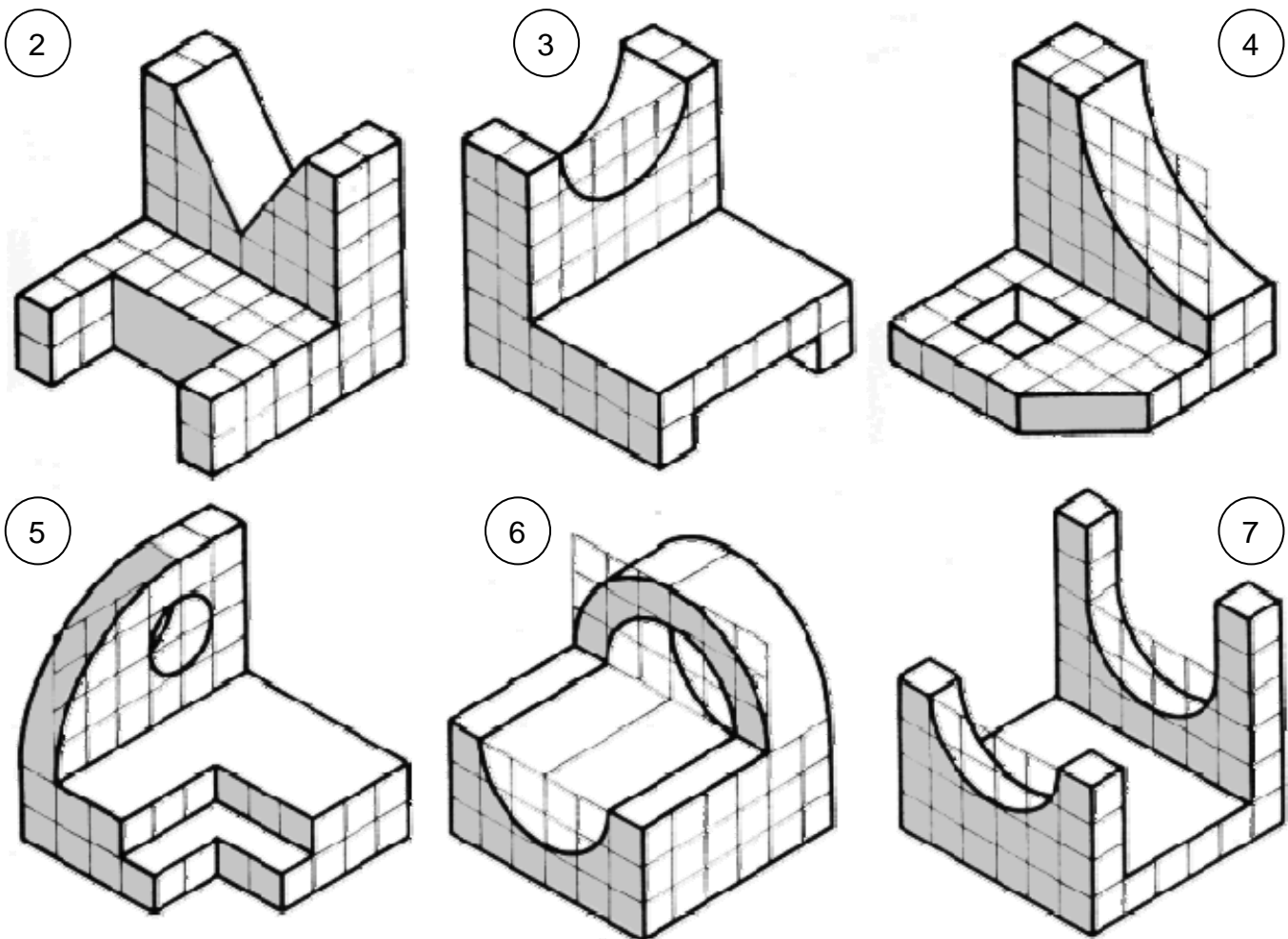
Feature 5)

Making the round hole.

- Select the front face of the lug for sketching.
- Create the sketch shown here. Make use of coincident or concentric relations as you sketch.
- Create the **Extruded Cut** feature using this sketch. Use the **Through All** option.



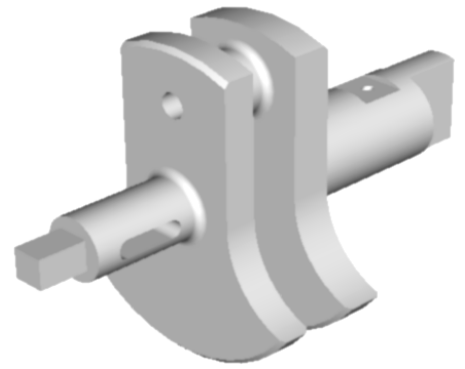
Now model each of these blocks. For practise make sure that you use both extruded **boss/base** & **cut** features. Remember to use symmetry in sketches/features where possible.



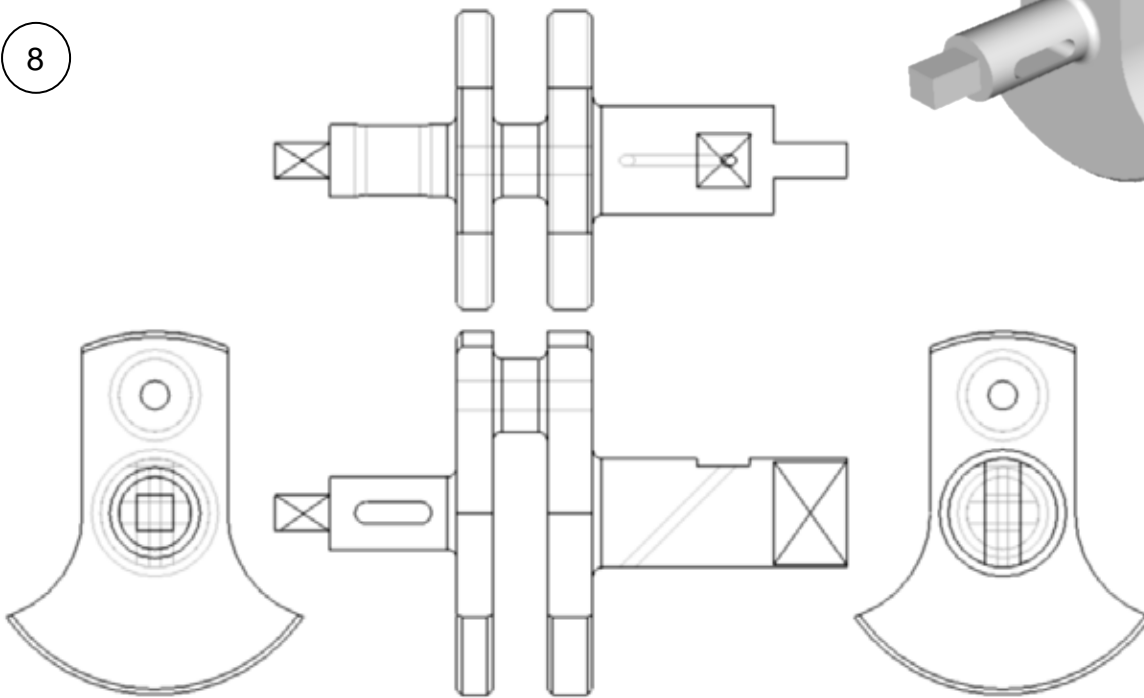
One square = 10mm

What is the minimum number of features needed to model each of these, just using extruded features?

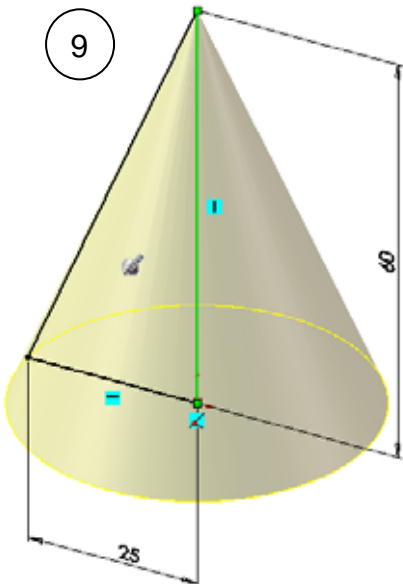
Now try this crankshaft. Make up your own dimensions.
 Again, only use extruded **boss/base** & **cut** features. Consider the most useful position for the crank shaft model origin.



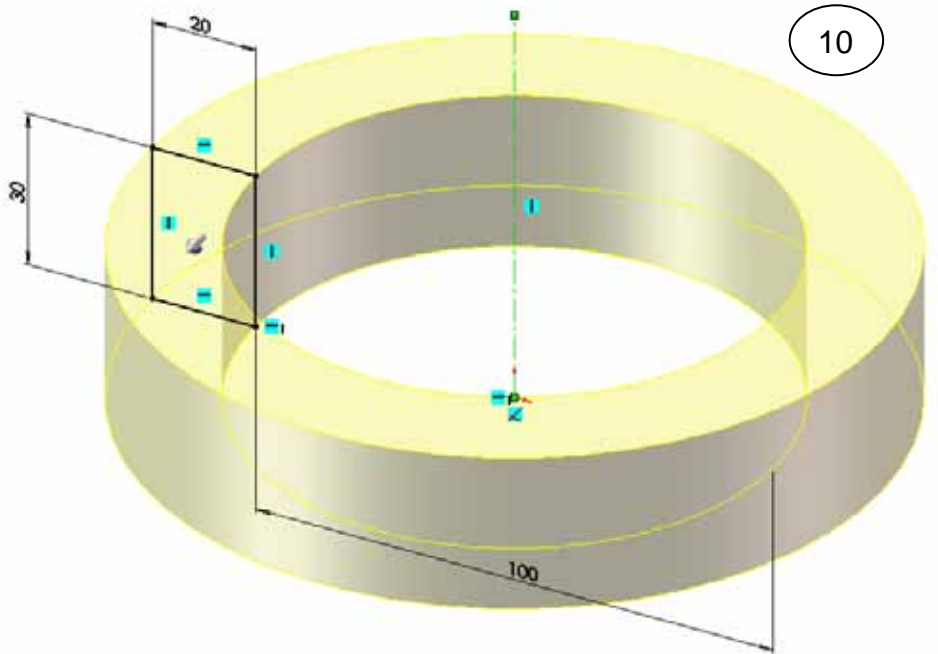
8



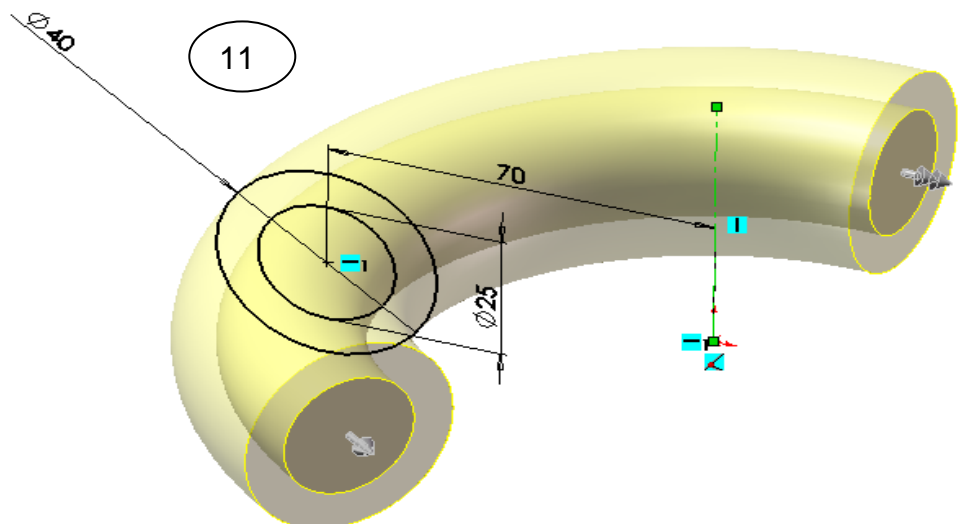
2 Revolve.



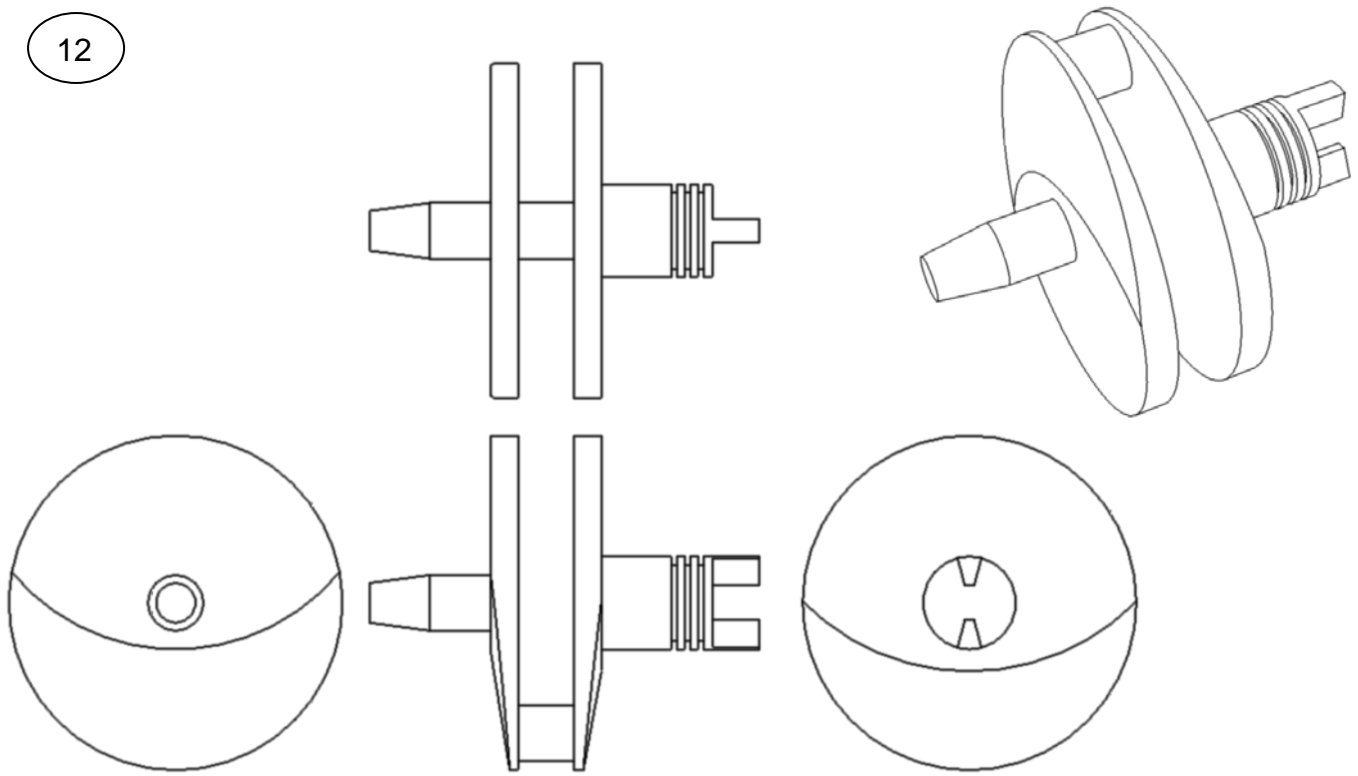
9



10



11



3 Extra features.

Round, chamfer, hole wizard, mirror, pattern, shell & rib.

3.1 **Rounds** and **Chamfers**: from examples 2 – 7, experiment with the round and chamfer features. Try to add 2mm radius rounds to all internally angled edges, and 1mm chamfers to a selection of externally angled edges.

3.2 Hole Wizard.

We will model a motorcycle wheel hub:

1) The first feature.

Sketch:

Right Plane

Circle, centre coincident with origin, Smart Dimension diameter of 130mm.

Extruded Base/Base:

Midplane (for symmetry) 120mm.

2) Add a brake disc hole using the Hole Wizard.

Insert > Features > Hole > Wizard

Leave as default except for:

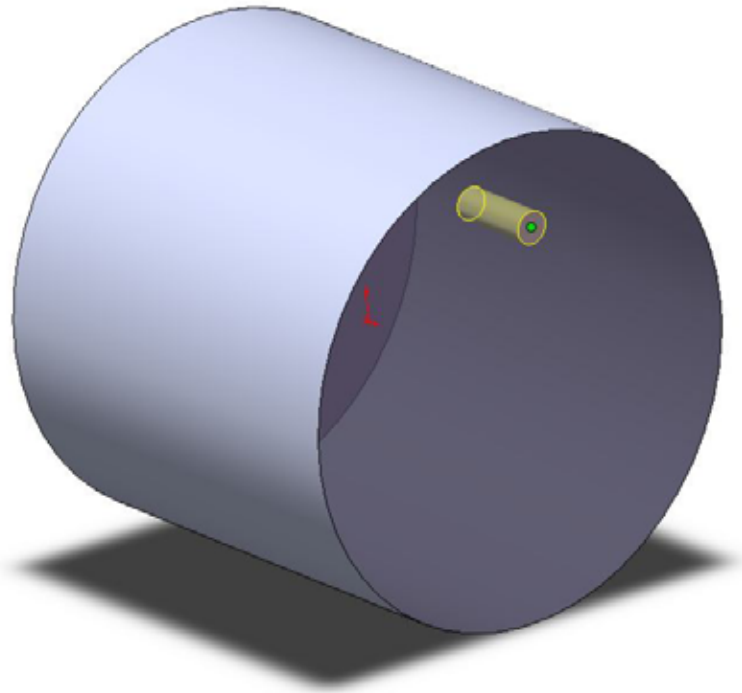
ANSI metric

Bottoming tapped hole

M10 x 1.25

Click on **Positions** tab and click on one side circular face of hub. This places the hole on that face. Click green tick.

13



To accurately position that hole, expand feature in Design Manager, and edit the 3D sketch.

Select the point and change the co-ordinates appropriately:

X	60.00
Y	50.00
Z	0.00

3.3 Pattern the hole.

First display centre axis of hub.

View > Temporary Axes

Insert > Pattern/Mirror > Circular Pattern...

Select the hub axis, then the hole as the **Feature to pattern**, and set **5** instances of the pattern.

3.4 Mirror the holes.

Insert > Pattern/Mirror > Mirror...

Mirror Face/Plane – Select the **Right Plane**, expand the model tree in graphics area.

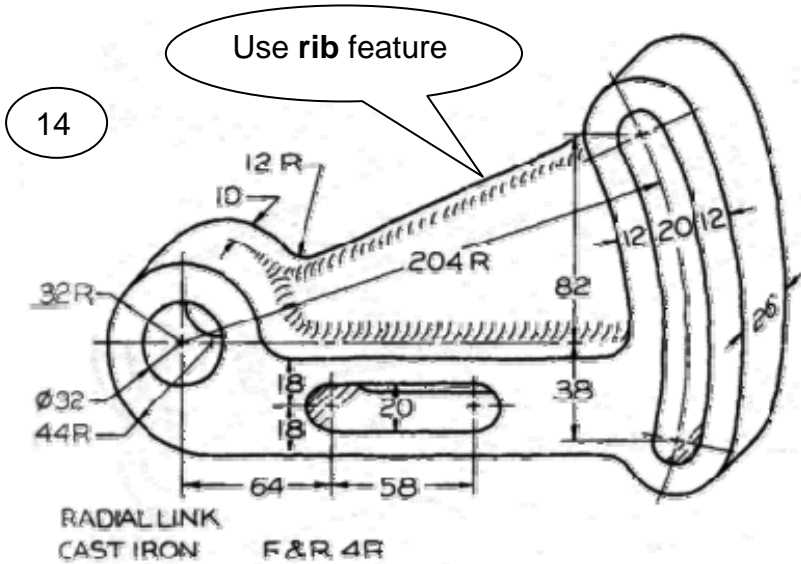
Select the **Circular Pattern** feature to mirror.

For even more practise:

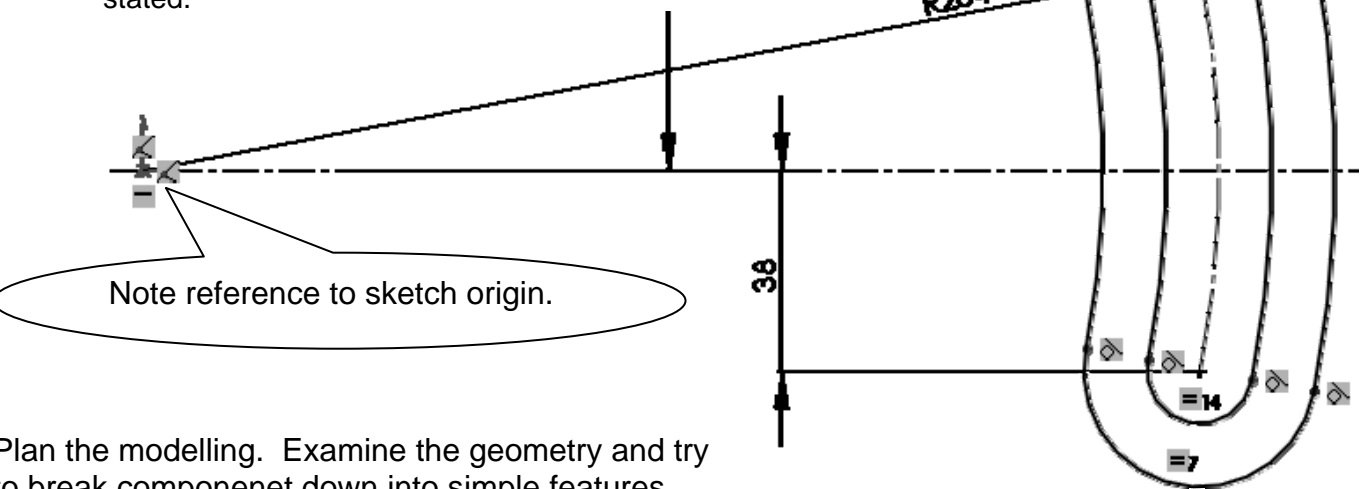
Model each component below, 14, 15 & 16, assign the correct material, right click **Material** feature in design tree, and then obtain the component mass using SolidWorks:

Tools > Mass Properties...

Material: Plain Carbon Steel
Mass: (2890g)



All fillets & radii 4mm unless otherwise stated.



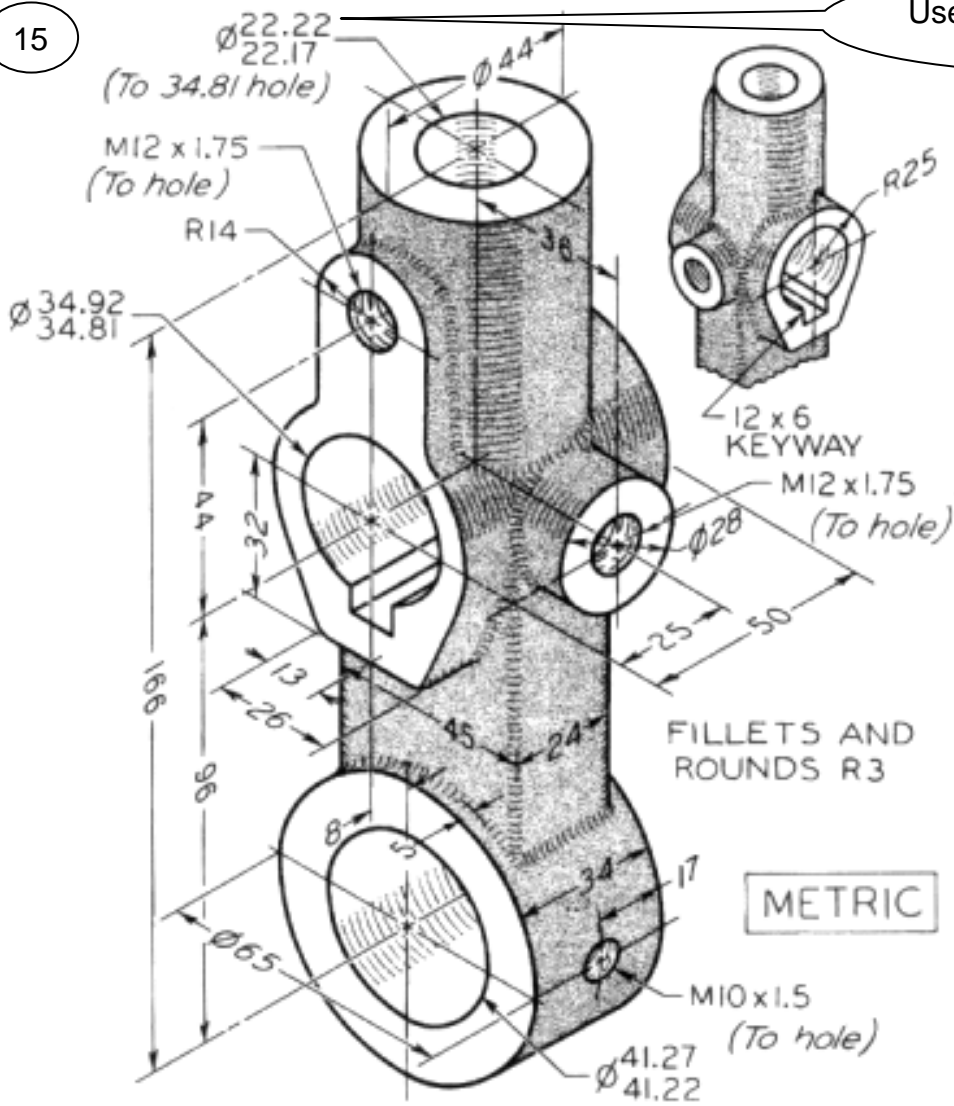
Plan the modelling. Examine the geometry and try to break component down into simple features.

You may decide to start by creating this sketch on the **Right** plane to create the 26mm extruded curved slot as the base feature.

Leave fillets and remaining holes until last.

15

Use lower limit values



Material: Brass
 Mass: (2001g)

Extrude either **Blind** in **Direction 1** and **Direction 2** or use the **Mid Plane** option.

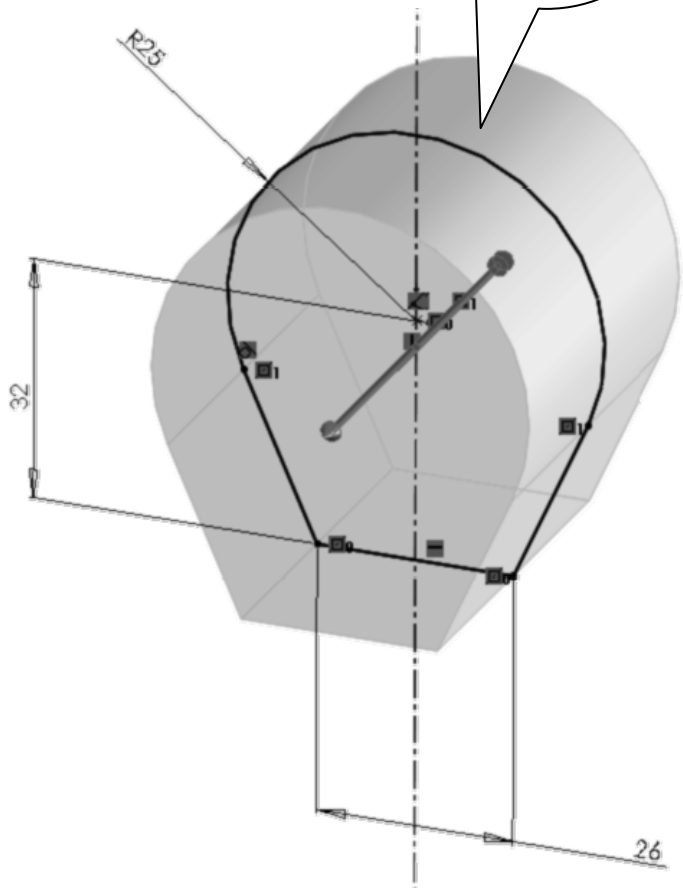
Examine the component geometry.

You may make the following observations:

- The basic component can be made up using a series of horizontal and vertical extrusions.
- There is symmetry for the major features in both of the vertical planes. This should be utilised when modelling.
- Details such as holes and fillets (rounds) exist, which can be added last.

Plan the modelling.

This is up to you. There are many ways achieve the same end result of course. You could start by modelling the central lug as the base feature, sketching on the **Right** plane.



to

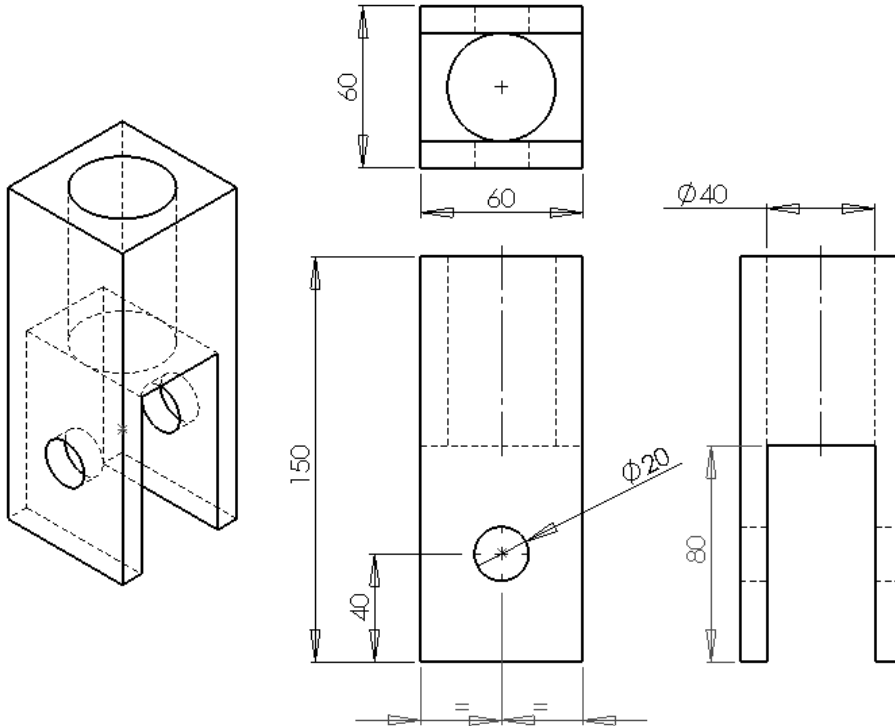
4 Assembling parts.

4.1 Preparation.

You will need to download this zipped file and extract files:

http://www.tech.plymouth.ac.uk/dmme/cad/cad_files/motion.zip

Model and save this part in same folder as above downloaded files:



4.2 Making the Assembly.

Create a new 'Assembly' file.

View > Origins

Insert the block, modelled above. Make sure that the **origins** are aligned by pressing Return to place the component, or... move pointer over assembly origin and click.

Insert crank part. Use appropriate **mates** to position the crank in the block.

Insert remaining parts appropriately, again using **mates**.

4.3 Creating an 'exploded' view.

Configurations allow us to capture different set ups of the same model/assembly file.

Click on **ConfigurationManager** tab.

Right click on the **Default** configuration feature and then click on **New Exploded View...**

To set up the exploded view simply select a component and drag it to the exploded position and click the **Done** button. This completes this step. Now repeat for the other parts. When finished click the **OK** (green tick) button. To edit this exploded view, select it from tree and right click Edit Feature, etc.

You can make as many different configurations/exploded views as you like. See the **Help** menu for more information.

5 Lofts (& sweeps).

5.1 Simple loft.

Create a cylinder: Insert an **Extruded Boss/base**, sketching on **Front** plane. Circle diameter 100mm, extrude depth 30mm.

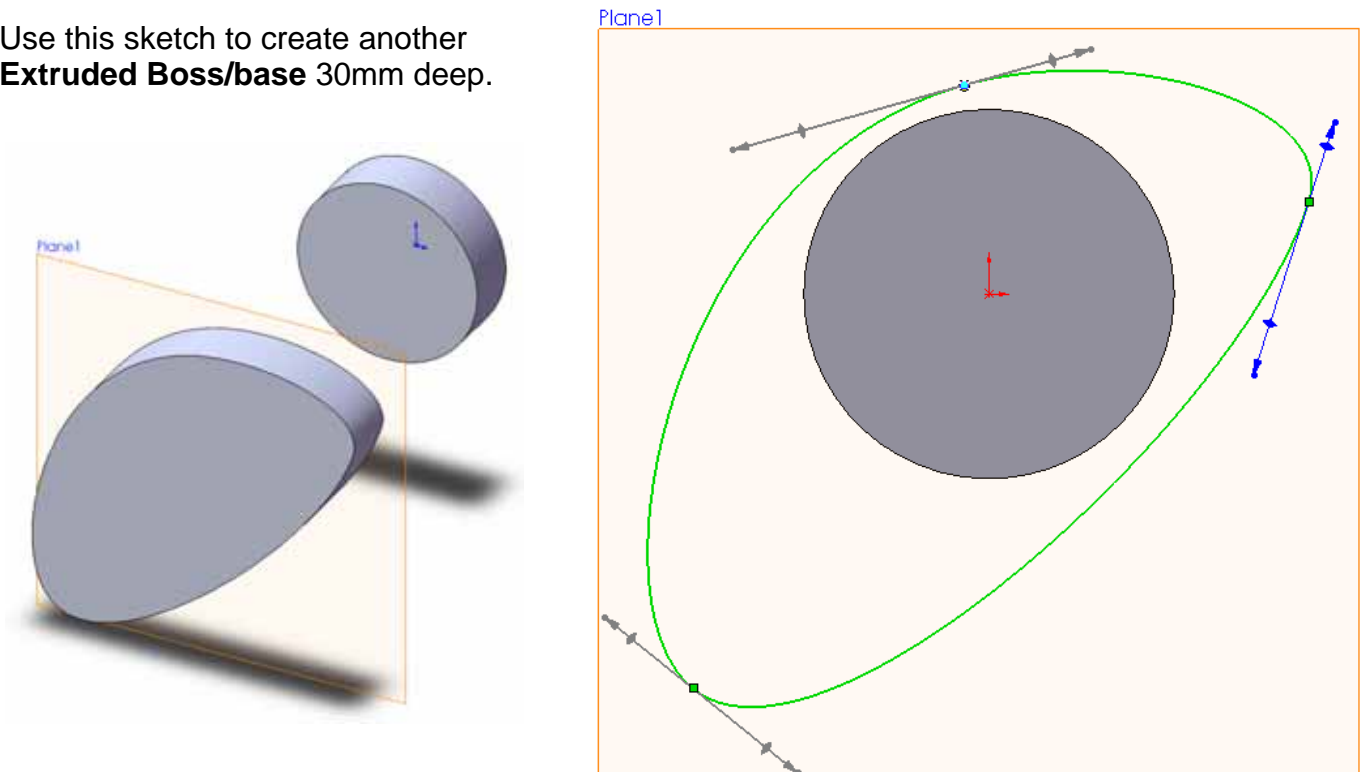
Create new **Reference Plane**:

Select **Front** plane, then **Insert > Reference Geometry > Plane**

Set distance to 200mm.

On this plane sketch a random closed spline shape like this:

Use this sketch to create another **Extruded Boss/base** 30mm deep.



Insert > Boss/Base > Loft

Select two opposite faces. Observe Preview. Experiment with dragging blue points around, effecting twist of loft feature.

5.2 Surface continuity examination.

Complete feature. Examine surface shape, curvature:

View > Display > Curvature

Examine surface reflection appearance:

View > Display > Zebra Stripes

Note what happens at join of different surfaces.

Select/right click/Edit feature on Loft.

Experiment with the **start/end constraints** parameter settings. OK the feature and re-examine curvature and zebra stripes.

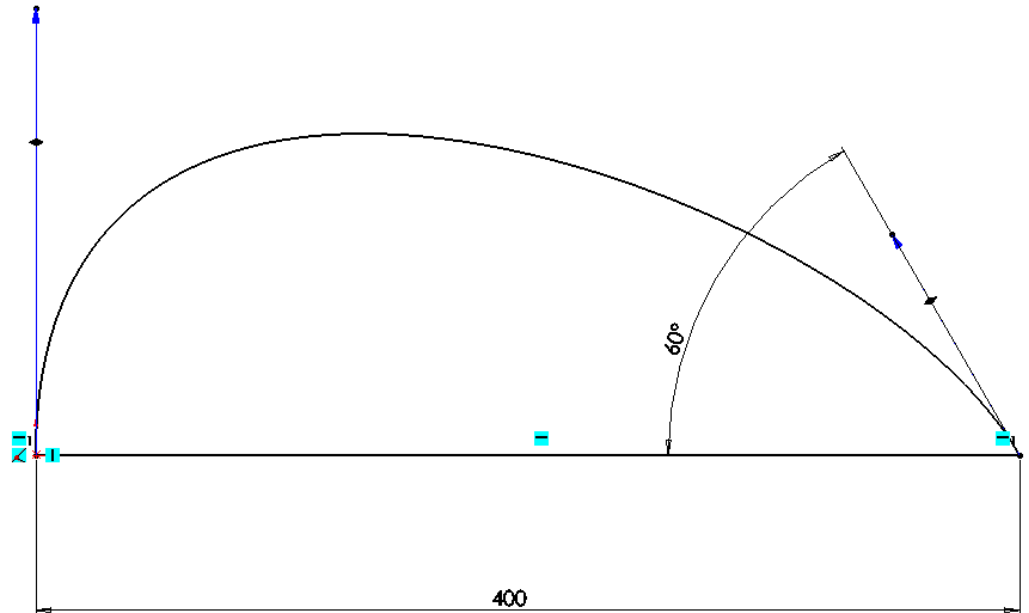
5.3 Freeform modelling.

Practise using the loft feature to model a motorcycle fuel tank, following these steps:

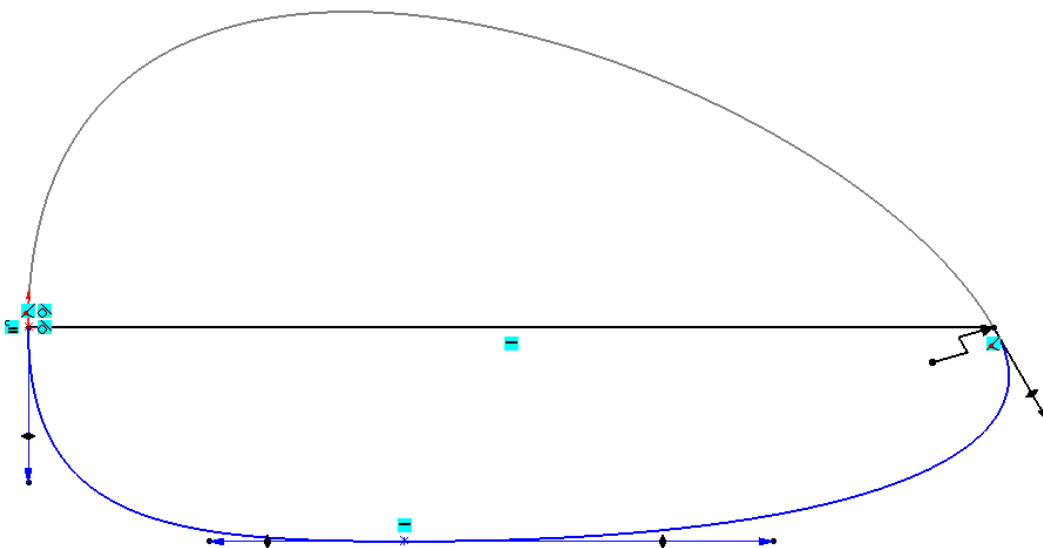
Tank is symmetrical, so model half.

Create three separate sketches to use as the profiles for the loft.

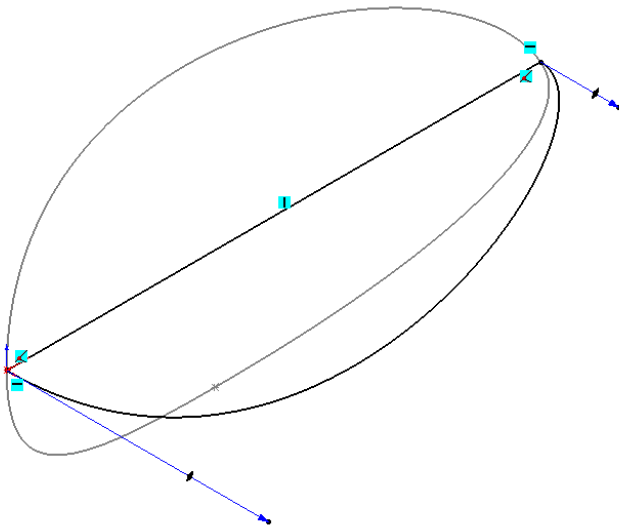
Sketch 1, on the **Right** plane: Add the appropriate **Smart dimensions & Relations** to the spline and horizontal line.



Sketch 2, on the **Right** plane:



Sketch 3, on the **Top** plane:

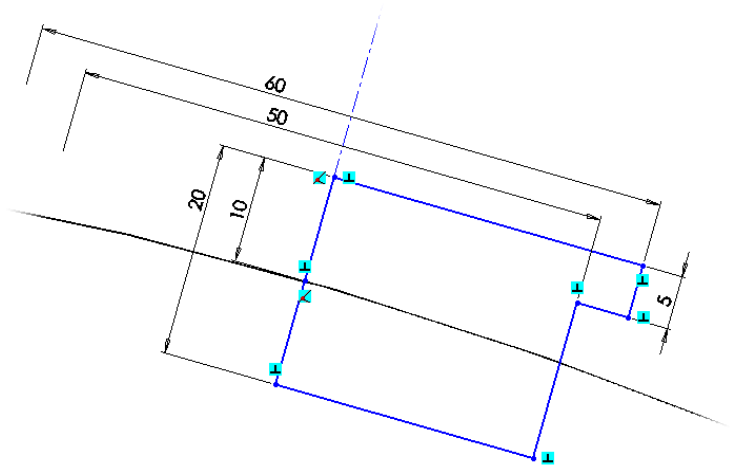
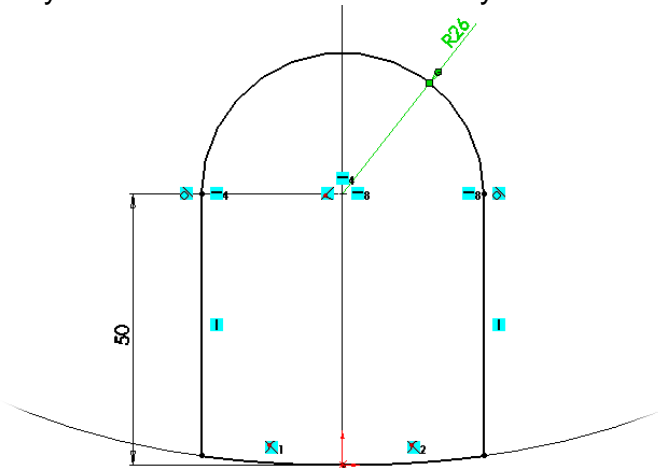


Create the **Loft** feature, experiment with the **Start/End Constraints** settings, etc.

Mirror the tank half.

Add an **Extruded Cut** for the frame cut out.

You may like to use the **Tools > Sketch Tools > Intersection curve**. If this leaves fragments, you may need to extend the sketch beyond the tank profile.



Add a **Revolved Boss/Base** for the fuel filler.

Add any extra finishing **Rounds**, etc.

Add a **Shell** feature to hollow out the tank.

5.4 Sweeps.

Sweeps are similar, but use one profile and a path. See the **Help > SolidWorks Tutorials** for more.

6 2D drawings, photo-rendered images & data sharing.

6.1 2D drawings of 3d model.

Help > SolidWorks tutorials - Lesson 3 – Drawings
Advanced Drawings

6.2 Photo-rendered 2d image of 3d model.

Help > SolidWorks tutorials - PhotoWorks

Tools > Add-ins - PhotoWorks

6.3 Data sharing.

To save SolidWorks file in a different format, to open in other cad software.

File > Save as...

If specific format is not listed, try using a neutral format:

6.3.1 Neutral formats.

File formats that are not software specific:

For most accurate 3d model, use one of these if possible:

STEP	*.stp	Preferred, as it usually captures the models features.
PARASOLID	*.x_t	Good for cad software built on parasolid kernel, e.g. Pro/Engineer.
IGES	*.igs	Good for most 3d cad software.

For faceted approximations to the 3d model geometry:

STL	*.stl	Usual format for rapid prototyping.
VRML	*.wrl	Web presentation.
DXF	*.dxf	Limited for 3d, but works. Useful for AutoCAD users and 2d work.

6.3.2 E-drawings – SolidWorks free viewer program.

Download from <http://www.solidworks.com>

Save as a E-Drawing part/assembly/drawing send your models to people without SolidWorks

Further work:

Help > SolidWorks Tutorials

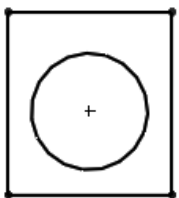
Online tutorial title:	Completed:
1) 30 minute lesson.	
2) Lesson 1 – Parts	
3) Lesson 2 – Assemblies	
4) Assembly mates	
5) Fillets	
6) Lofts	
7) Pattern Features	
8) Revolves & Sweeps	
Any of the remaining tutorials, as required....	

General tips on modelling:

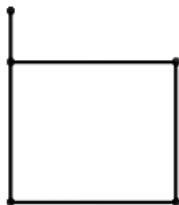
- In your imagination break the component down into a collection of simpler solid shapes.
- Choose an appropriate base feature to create first and build the model up from that.
- In the more complex models it is normally better to leave details such as chamfers, fillets and small holes until last.

REMEMBER:

- Each sketch you create should end up being **Fully Defined**. This means that its location, with respect to the model origin or other geometry, and sizes are all specified. Blue lines in sketches are Under Defined.
- Every sketch for a feature must consist of lines that form continuous loops or perimeters. These loops or perimeters must not intersect themselves. No lines should lie over other lines or be duplicated. Lines must join at their ends.



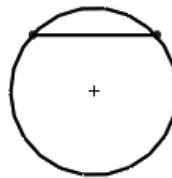
Good



Bad



Bad



Bad