

Worksheet #8. 3D CAD part/assembly modelling.

To become familiar with the basic functionality of 3d parametric solid modelling, work through these SolidWorks online tutorials, and then model the four components detailed on pages 2 onwards.

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	

General tips on modelling:

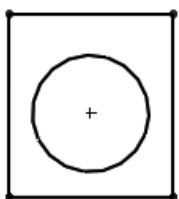
- In your imagination break the component down into a collection of simpler solid shapes.
- Choose an appropriate base feature to create first and build the model up from that.
- In the more complex models, such as components 2, 3 & 4, it is normally better to leave details such as chamfers, fillets and small holes until last.

For each model assign the correct material, right click **Material** feature in design tree, and then obtain the component mass using SolidWorks:

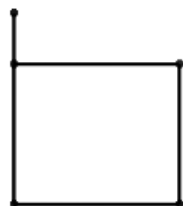
Tools > Mass Properties...

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.



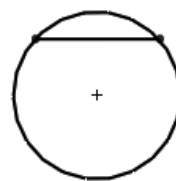
Good



Bad



Bad

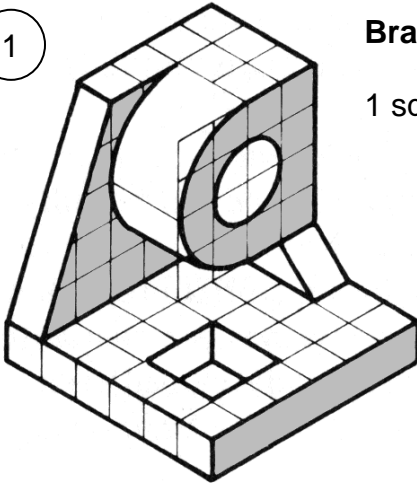


Bad

1

Bracket.

1 square=10mm



Material: ABS Plastic
Mass: (84g)

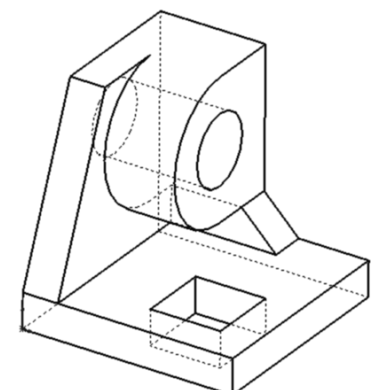
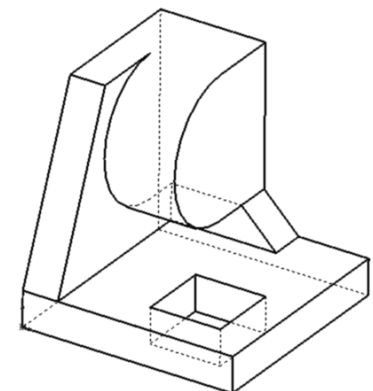
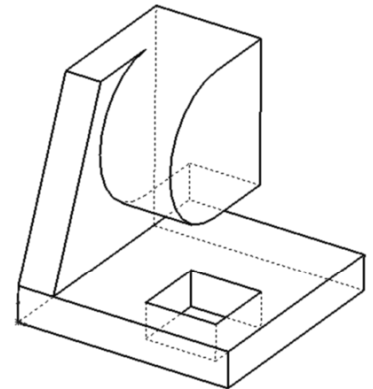
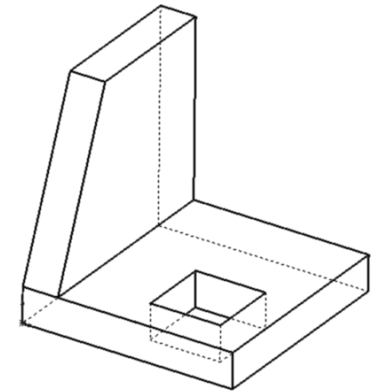
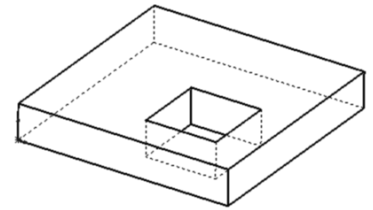
For this example we will build up the model in the stages shown on the right here.

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:

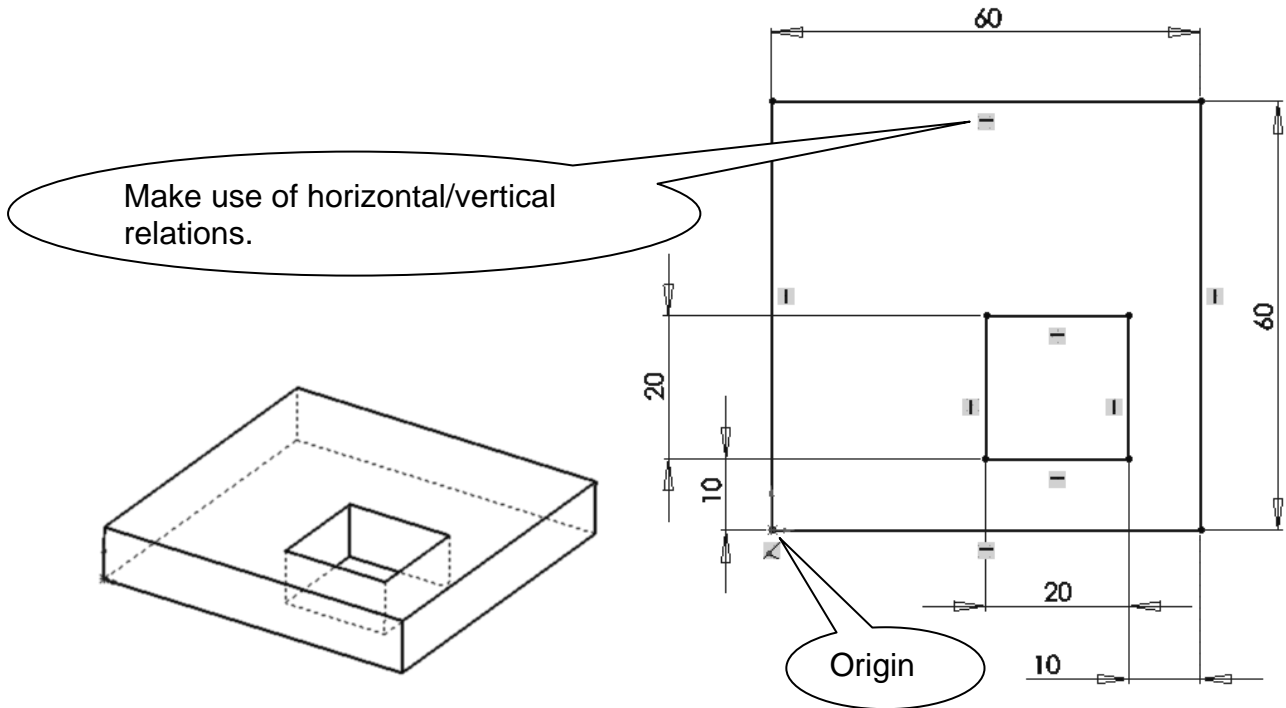
- creating an **Extruded Cut** feature, where solid material is cut (removed) from the model. This makes the hole.



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.

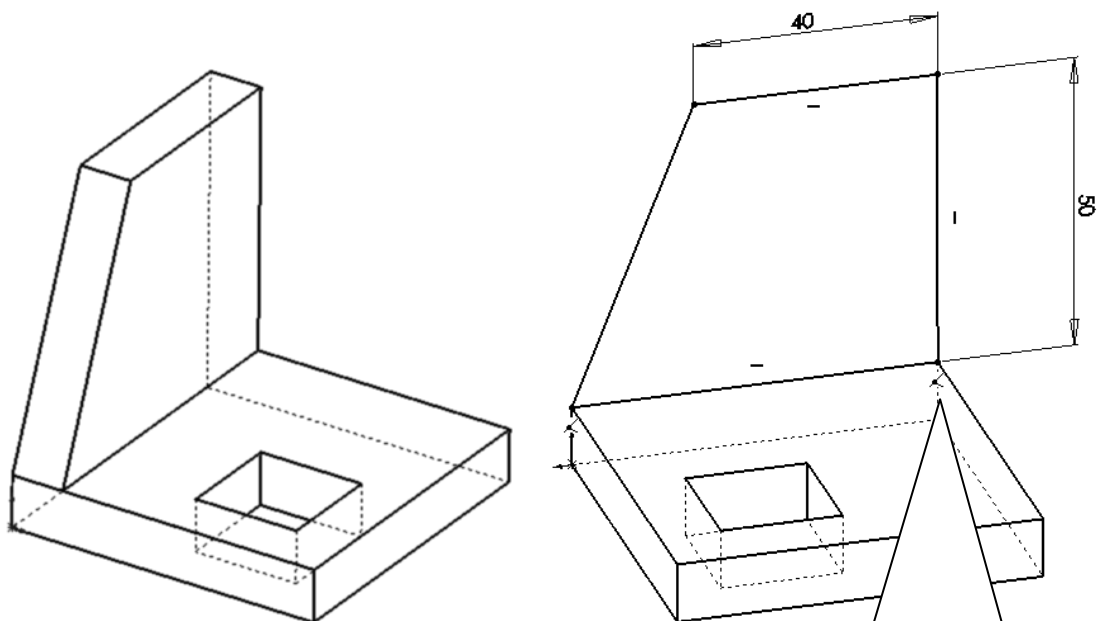


- Create the **Extruded Bore/Base** feature using this sketch.

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 Bore/Base** feature using this sketch.

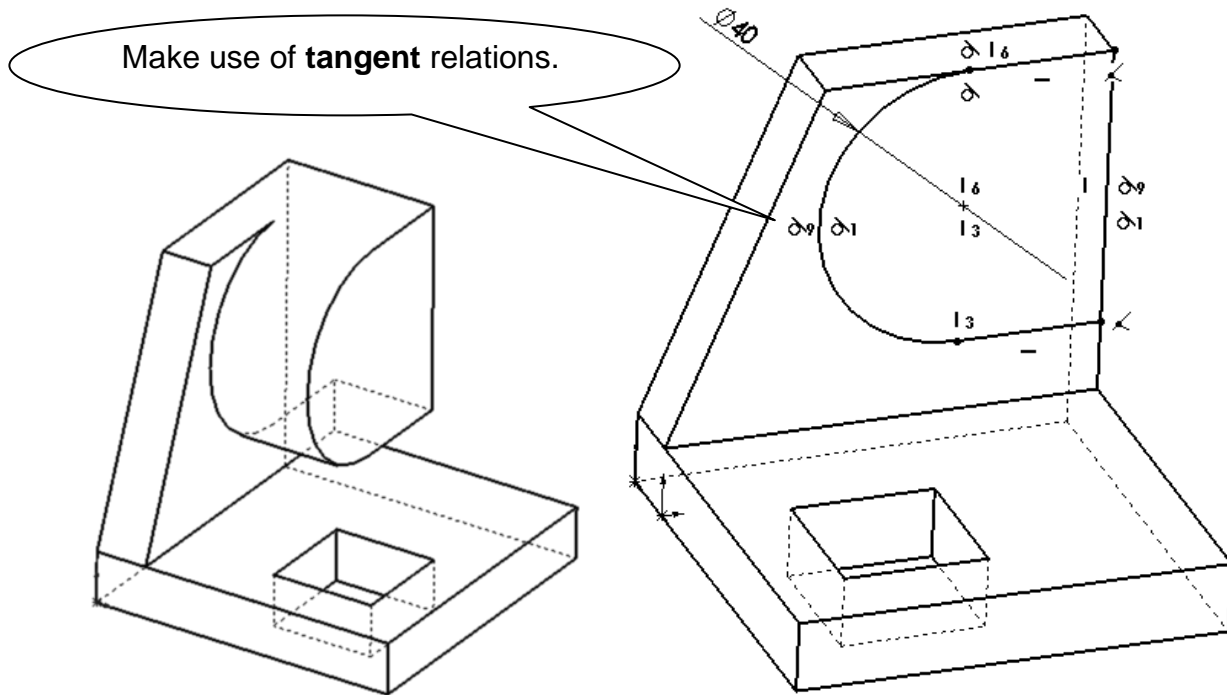


Feature 3)

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.

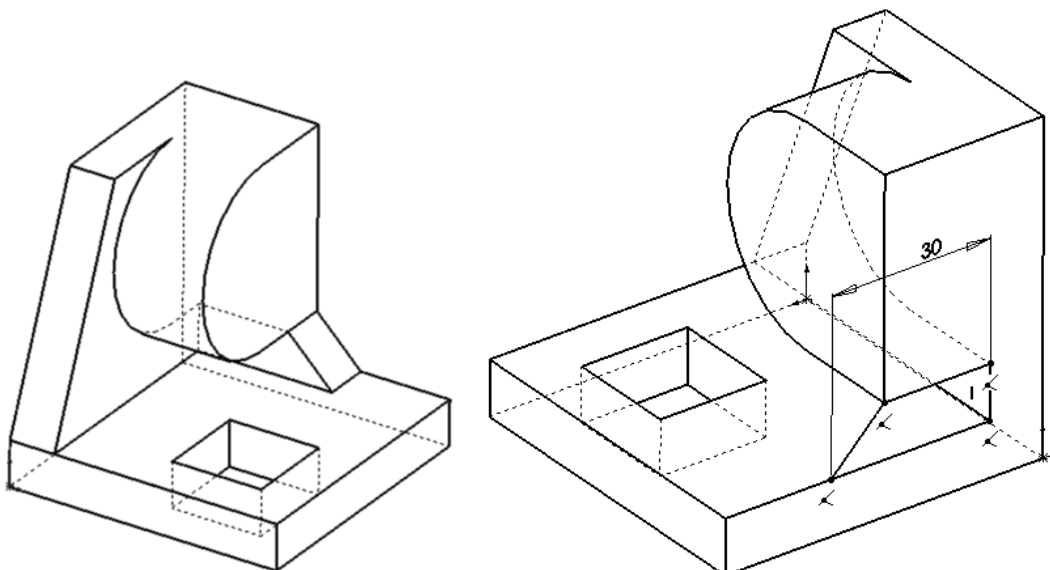


- Create the **Extruded Boss/Base** feature using this sketch.

Feature 4)

Making the small web connecting the base to the lug.

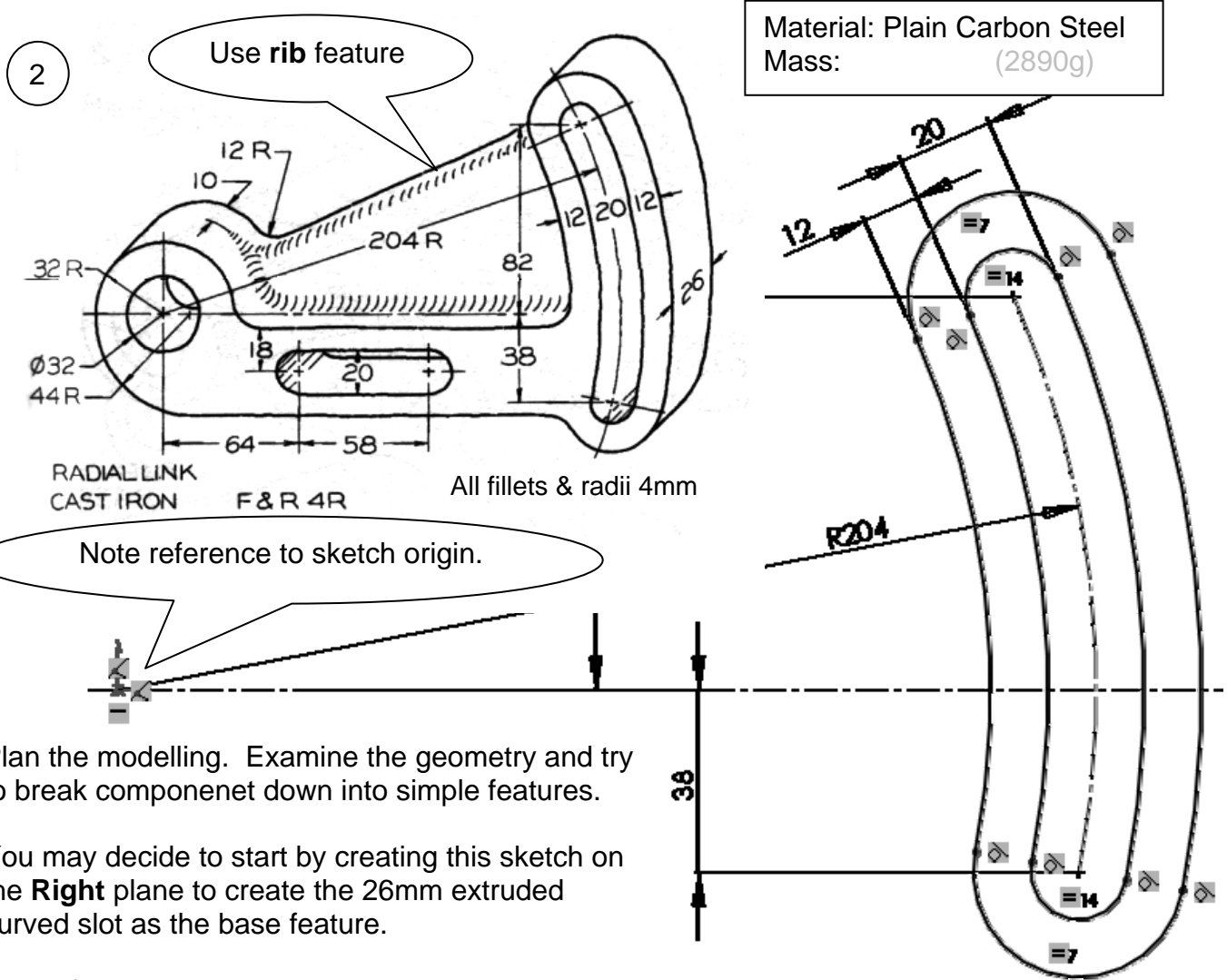
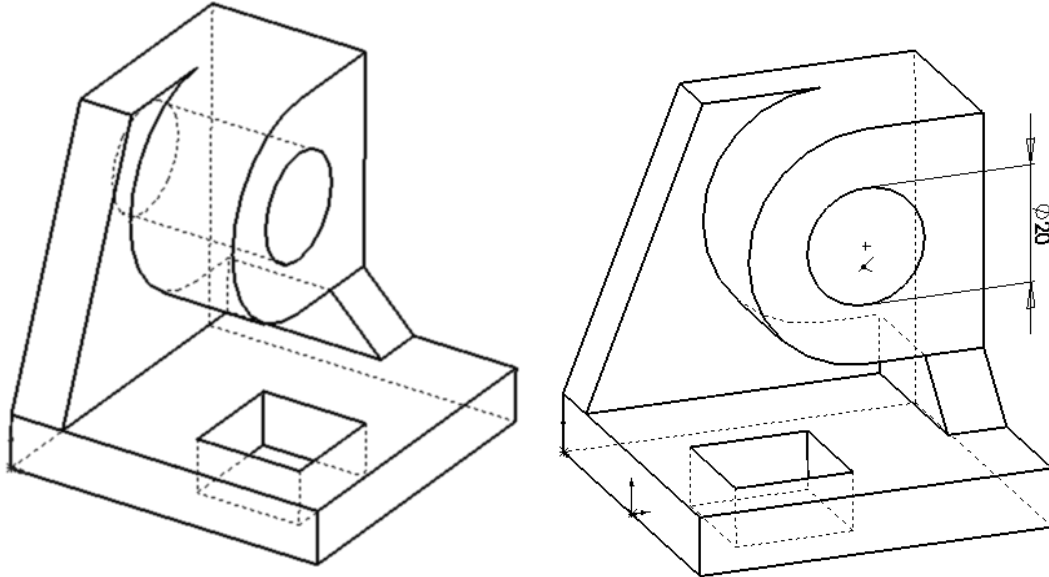
- Select the back face of the lug for sketching.
- Create the sketch shown here. Make good use of relations as you sketch.
- Create the **Extruded Boss/Base** feature using this sketch.



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.



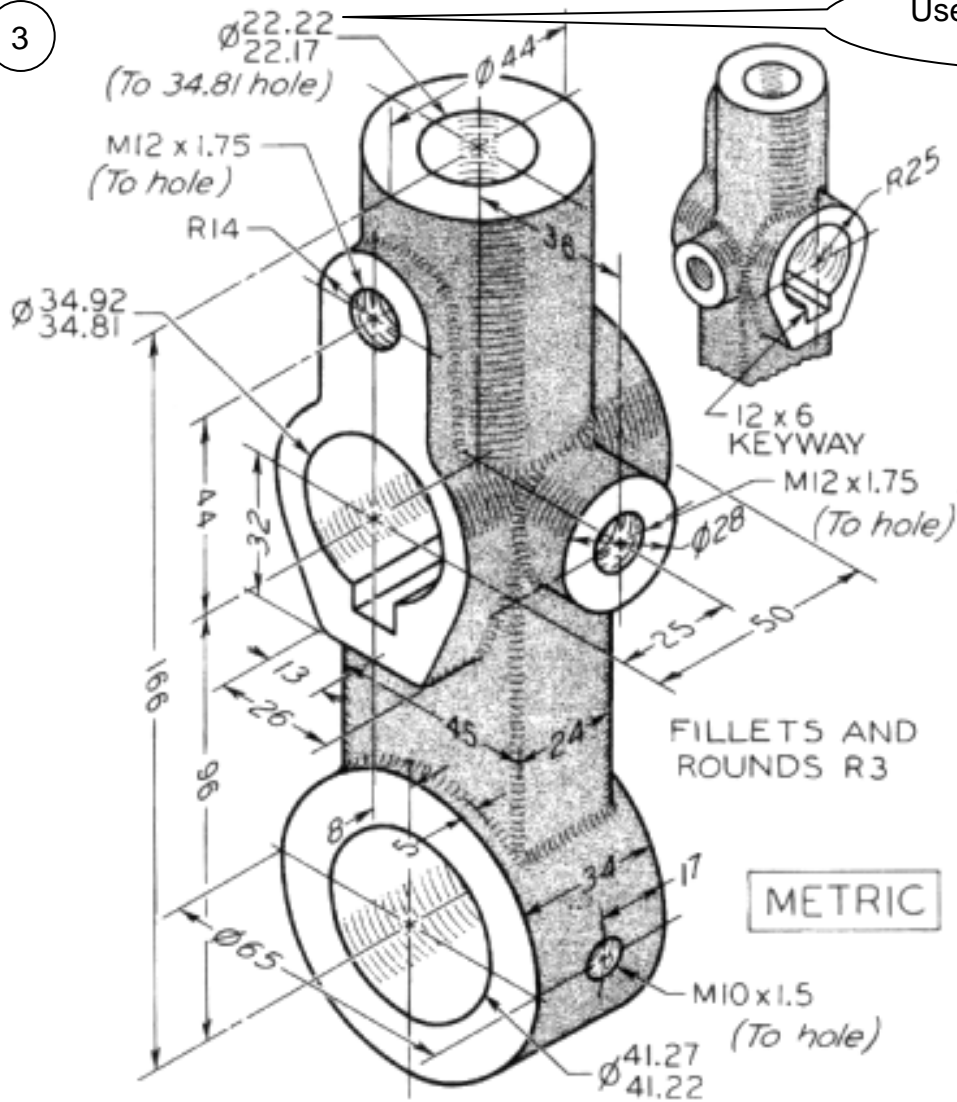
Plan the modelling. Examine the geometry and try to break component down into simple features.

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.

3

Use lower limit values



Material: Brass
Mass: (2001g)

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

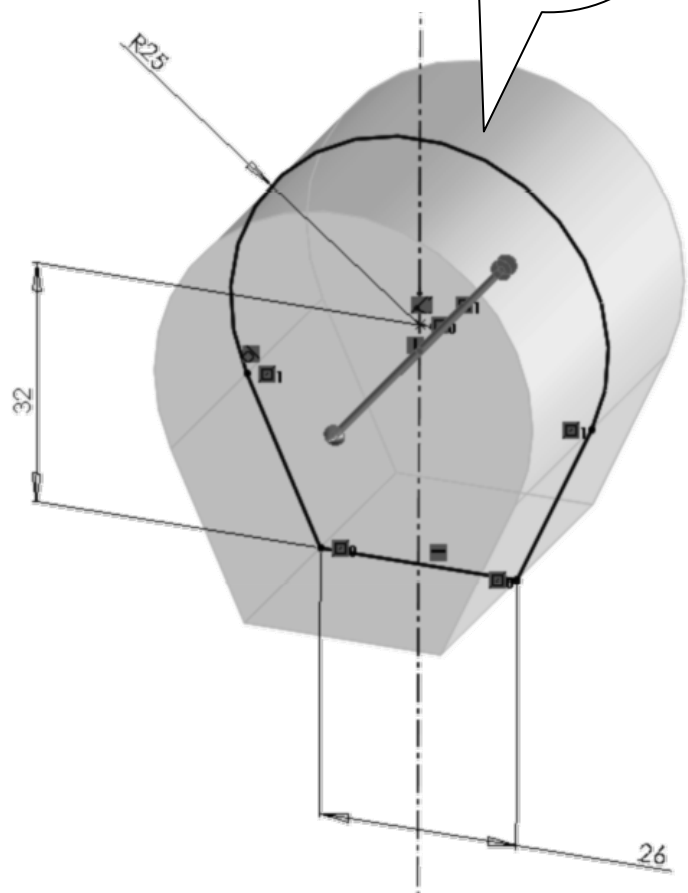
Examine the component geometry.

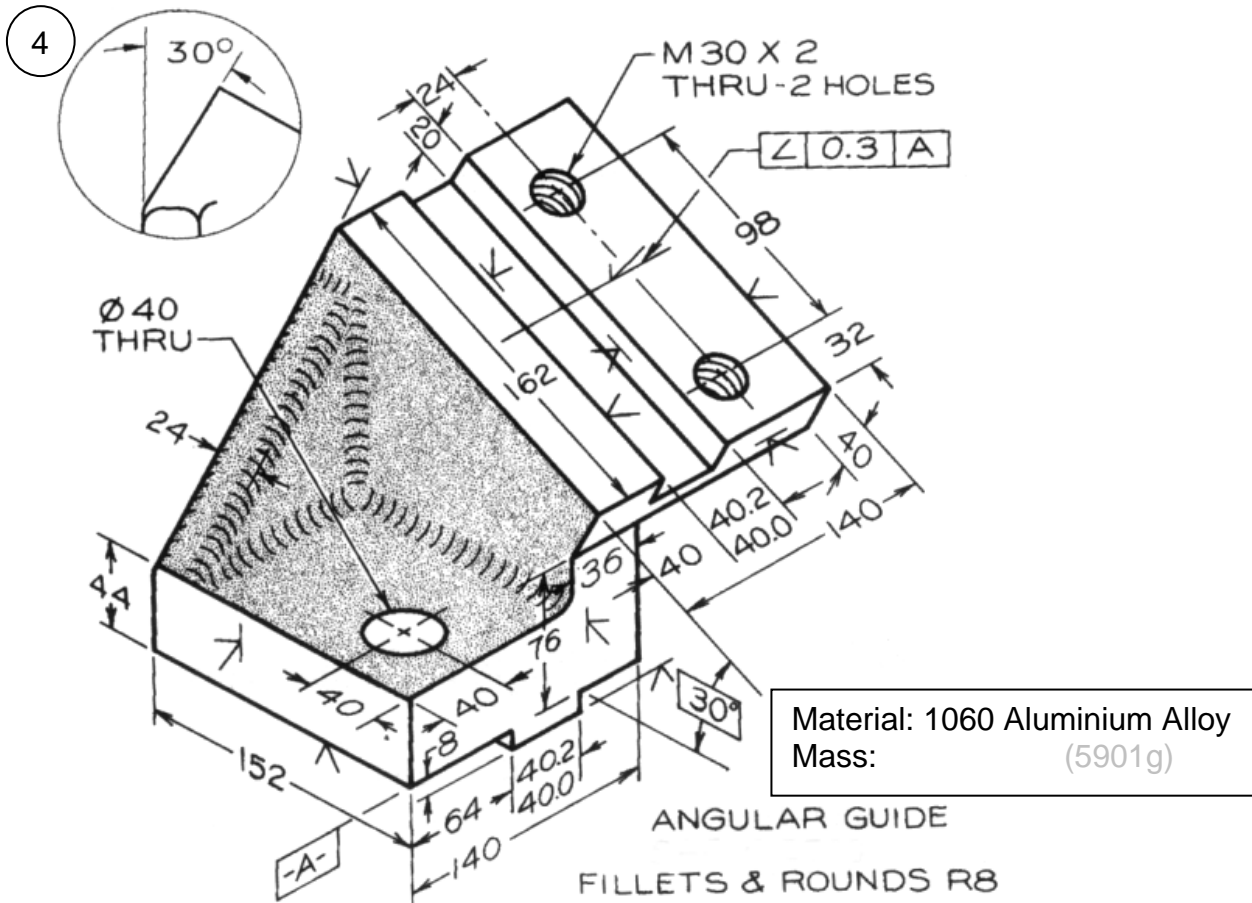
You may make the following observations:

- 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 to achieve the same end result of course. You could start by modelling the central lug as the base feature, sketching on the **Right** plane.

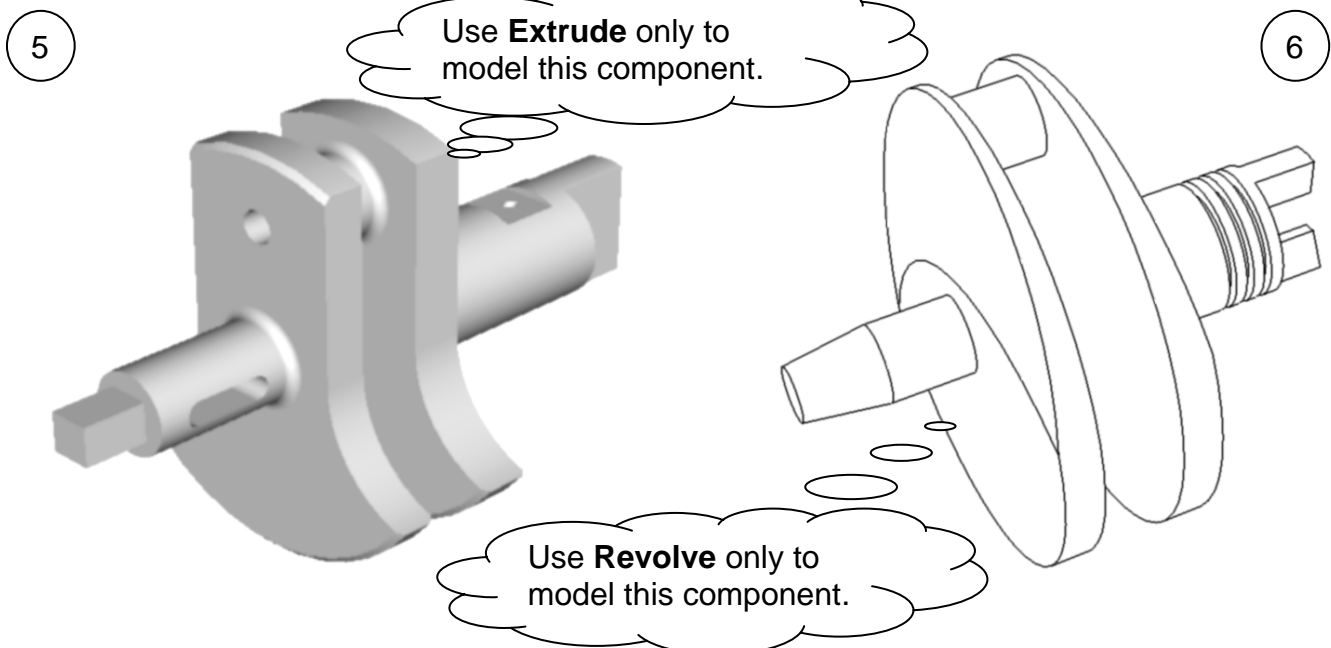




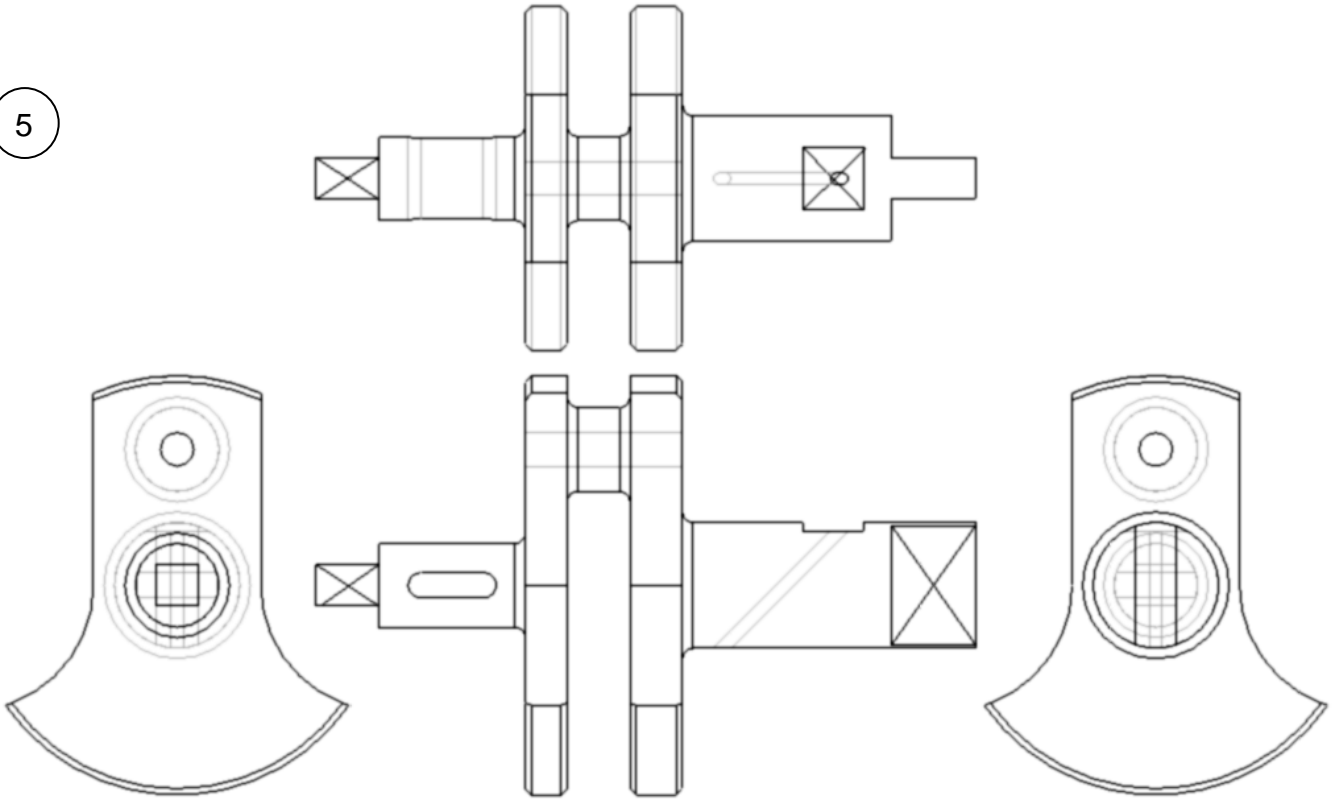
Examine the component geometry.

- 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.
- It is normally better to leave details such as chamfers, fillets and small holes until last.

Now, model these two crankshaft examples, using your own dimensions.



5



6

