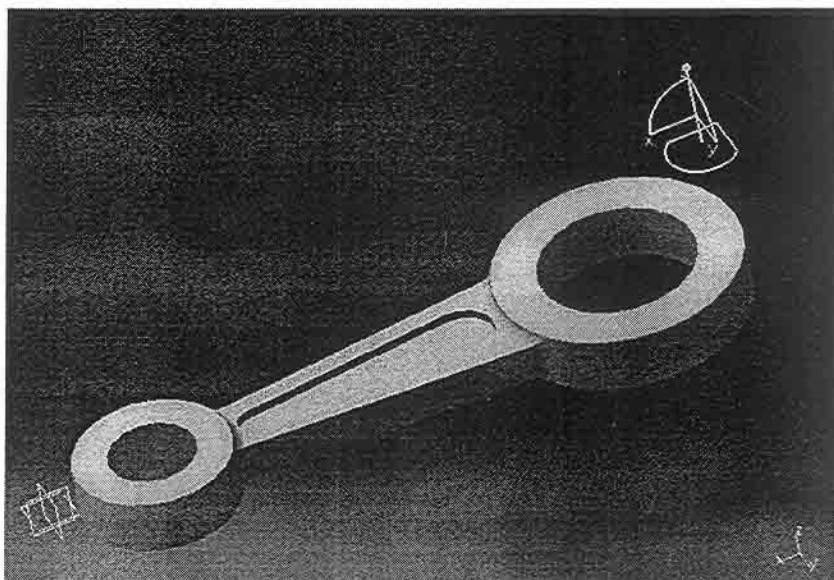


Exercise 2.

Sketching Profiles and creating a solid part using **Pad**

Use the Part Design application and the Sketching tool to create the part shown below.



Details of the part are provided in the drawing overleaf.

Hints:

1. You should start by creating a sketch of the outer profile. You may find it useful to create some construction geometry to help you to generate the sketch. Take care that there is only a single closed boundary when you exit from the sketcher.
2. When you have created a solid representing the main profile, you will need to create additional sketches to represent the pocket, and thickened end sections. You can sketch directly onto the face of a solid by selecting the face as the sketching plane when you enter the sketcher.
3. You can take advantage of symmetry in the part to simplify the modelling process

You will need to use the following functions in the sketcher to create the profiles:



Circle Tool



Trim Tool



Line Tool



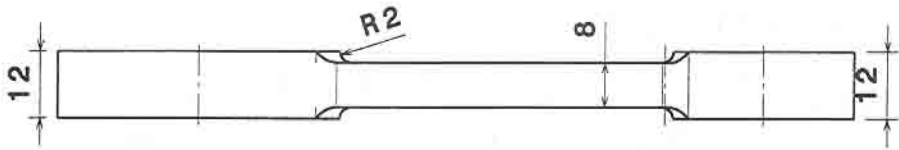
Constraint Tool



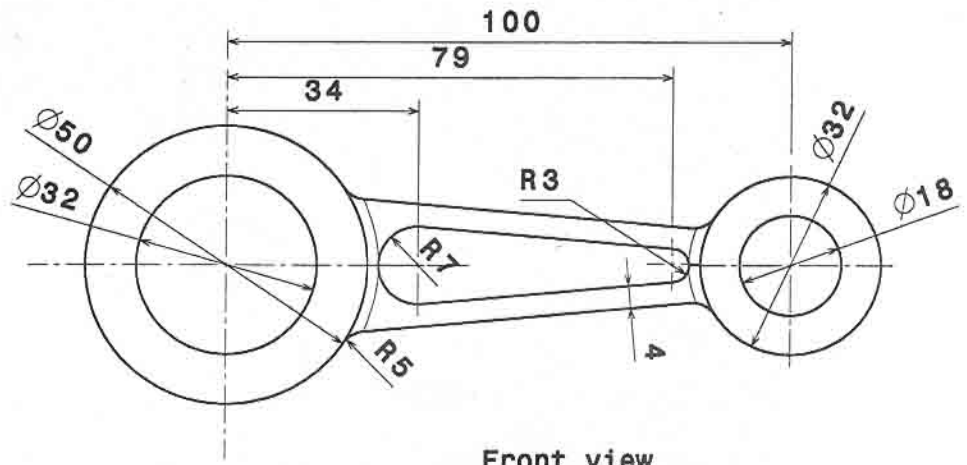
Corner Tool



Constraint in a Box



Top view
Scale: 1:1



Front view
Scale: 1:1