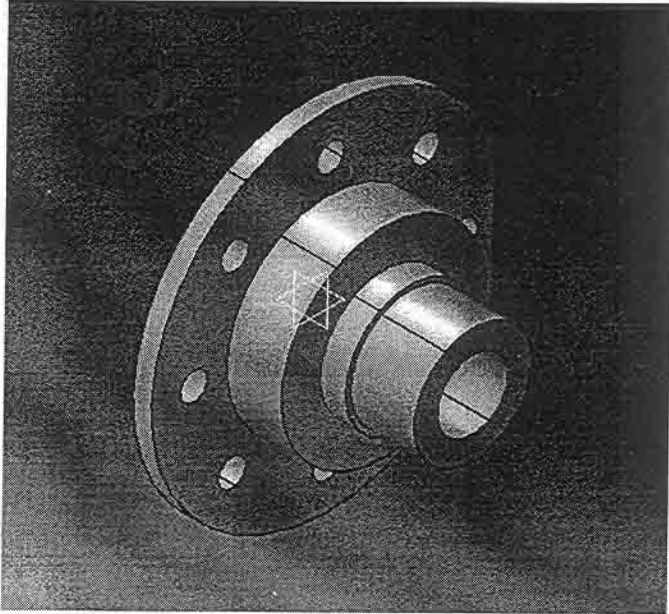# CATIA V5 – Coursework

Exercise 1.

Sketching profiles and creating a solid part using **Shaft**

Use the **Part Design** application to create a solid model of the part shown below. The part should be constructed by revolving a sketch of its cross sectional geometry.

Hints:
Create a sketch of the cross section of the bearing. You will need to create an axis entity to represent the axis of revolution to be used when the solid part is created
The bolt holes can be generated as a circular array of hole entities

You will find the following functions useful:

Sketching

Profile Tool

Axis Tool

Circle Tool

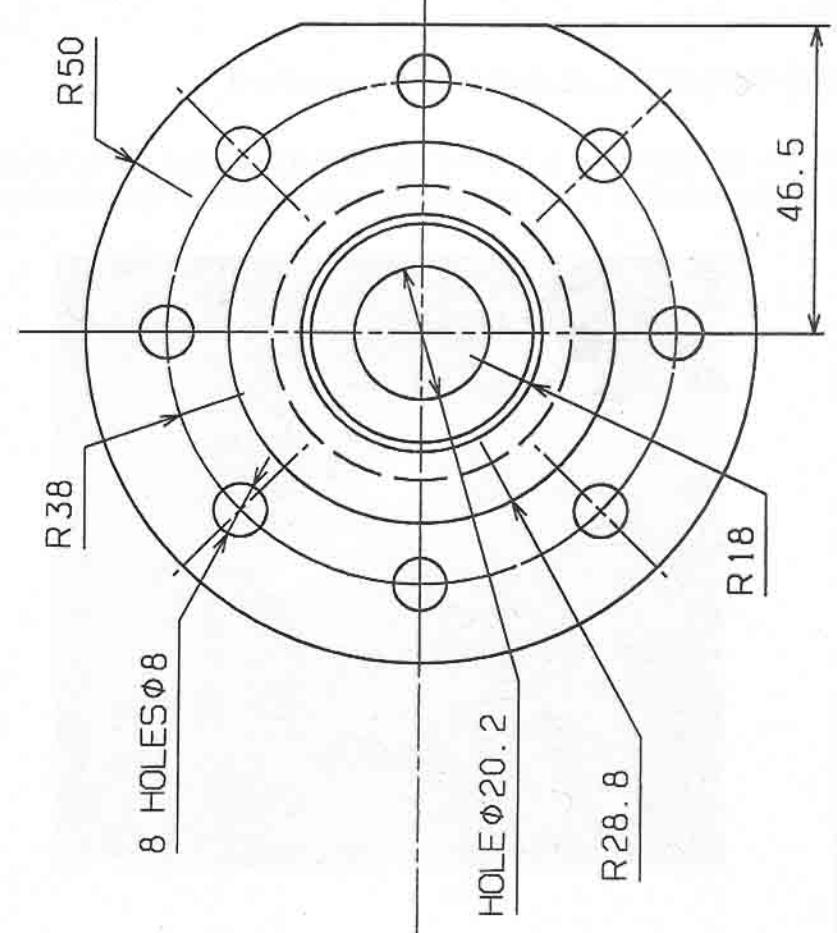Rotate Tool

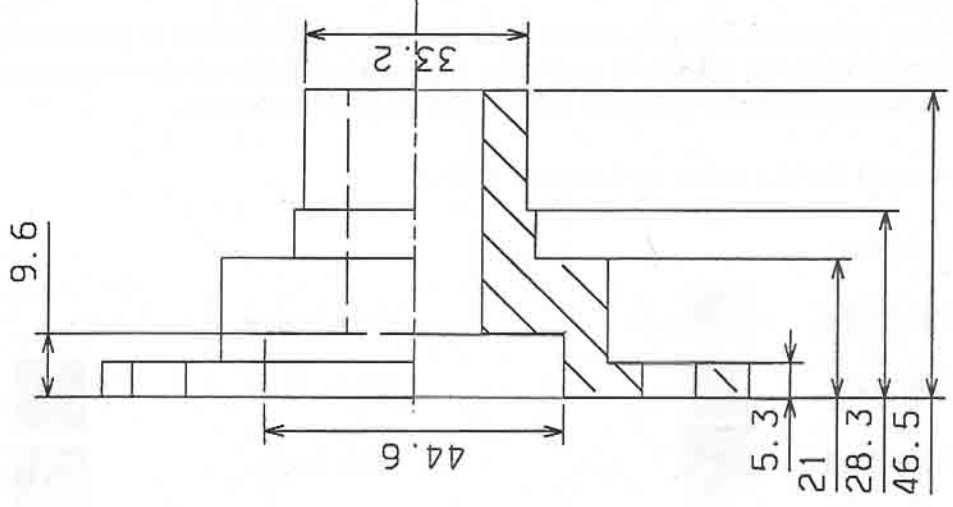Constraint Tool

Part Design

Shaft

Hole Tool

Circular Pattern Tool

R50

R38

8 HOLES ⌀8

HOLE ⌀20.2

R28.8

R18

46.5

33.2

9.6

44.6

5.3

21

28.3

46.5

SHEET SIZE A4	ITEM	PART No.	DESCRIPTION		No. OFF	MATERIAL	SPEC.	REMARKS
			DRAWN					
			CHECKED	APPROVED	ISSUED	APPROVED		

TITLE :- Bearing Exercise

SCALE 1:1

FINISH

DRAWING No.