

### Exercise 5.

#### Analysing a part using Generative Part Structural Analysis

In this exercise you should use the **Part Design** application to create a 3D solid model of the bracket component shown below, and then use the **Generative Part Structural Analysis** application to perform a finite element analysis on the part.

Generative Part Structural Analysis provides a highly automated structural analysis environment and allows you to easily perform a structural analysis on a solid part.

Figure 5.1 shows a picture of the bracket design, and details of the part are shown in figure 5.2 overleaf.

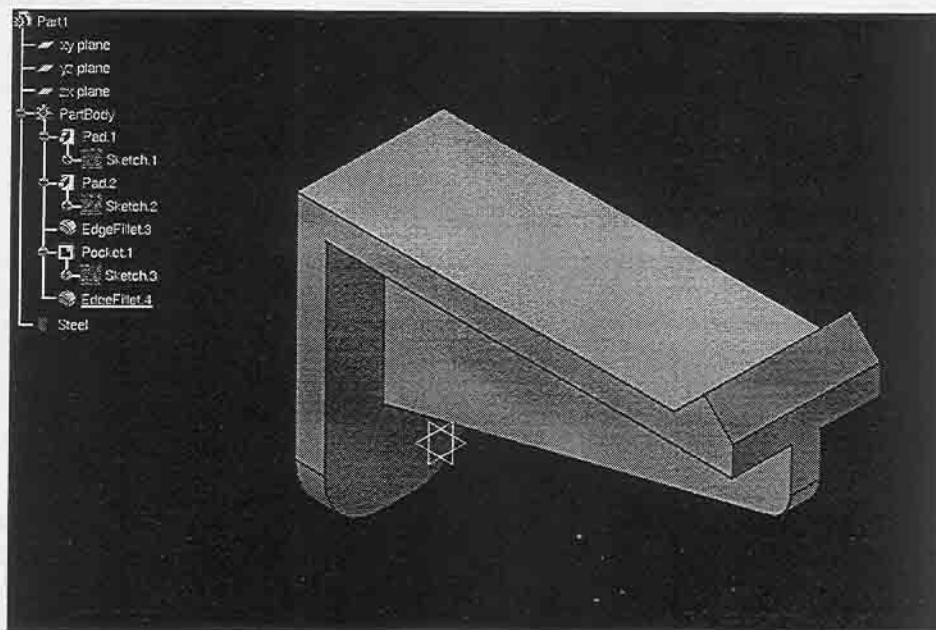


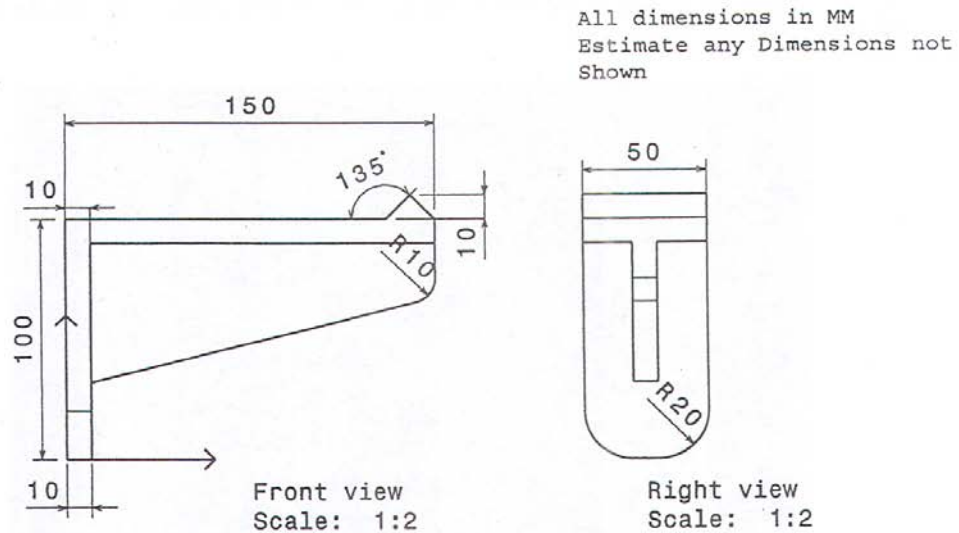
Figure 5.1 3D view of bracket component

Hints:

You will need to complete the following steps to complete the analysis:

1. Create a solid model of the part using the Part Design application

2. Assign a material to the part (e.g. Steel) using the **Assign Material** function
3. Create a new Analysis part
4. Insert your bracket part into the analysis environment
5. Specify that the rear vertical face of the part should be fixed in all degrees of freedom using the **Create Clamp** function
5. Apply a load of 100N in the  $-Z$  direction distributed across the angled front face of the part
6. Run a static analysis
7. Examine the stress distribution and deformed shape for the component



**Figure 5.2 Details of Bracket Component**