

## SolidWorks 2011-2012 – Coursework

### *Exercise 4.*

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 SimulationXpress application to perform a finite element analysis on the part.

SolidWorks Simulation provides options for analyzing contacting components as part of stress analysis. You can assign component or local contact conditions to faces that are touching or have gaps between them.

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

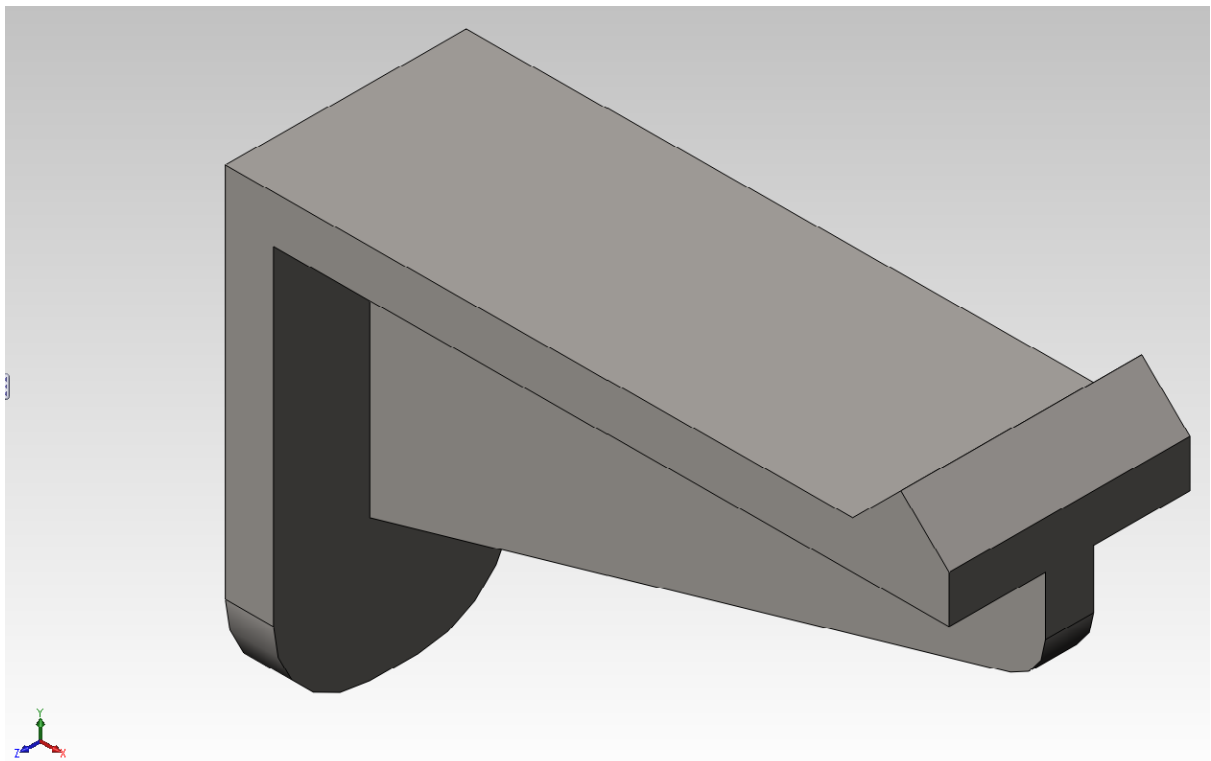


Figure 4.1 3D view of Bracket component

### Hint

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 Material function
3. Tool → SimulationXpress
4. In Option select results save location

5. Follow the wizard instruction
6. Select the rear vertical face of the bracket to be fixed
7. Apply a load of 100N in the -Y direction distributed across the angled front face of the part
8. Click Run
9. Examine the stress distribution and deformed shape for the component

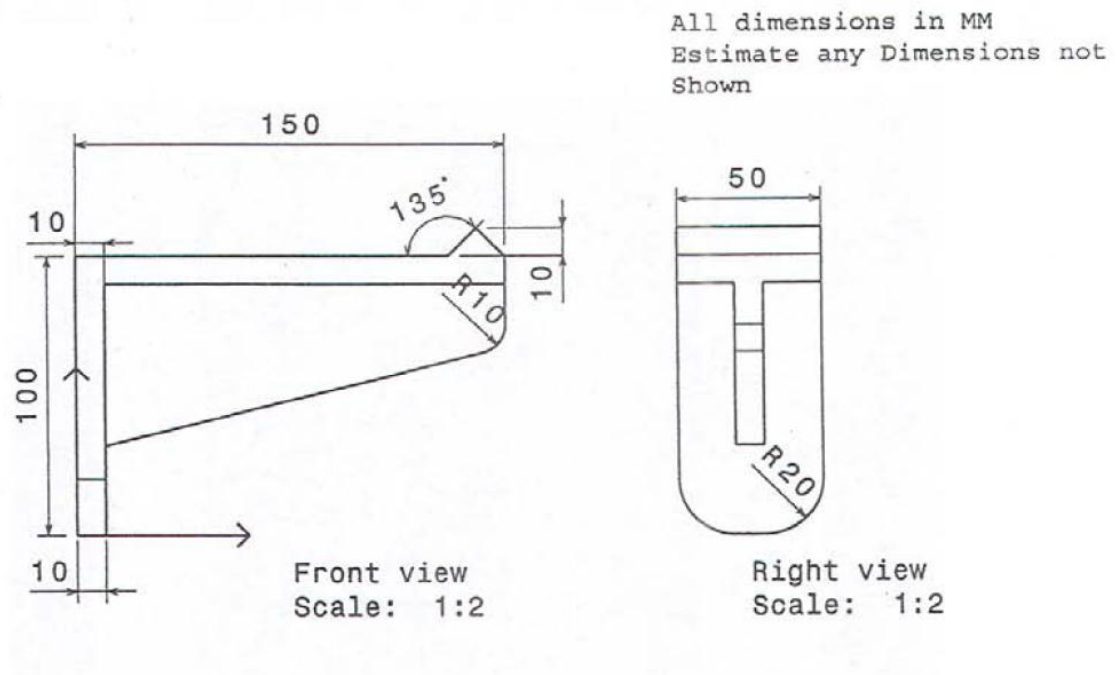


Figure 5.2 Details of Bracket Component