MECH 4450 Term Project Report

Project 2 (Static structure)

Kong Xiangzhou 20026414

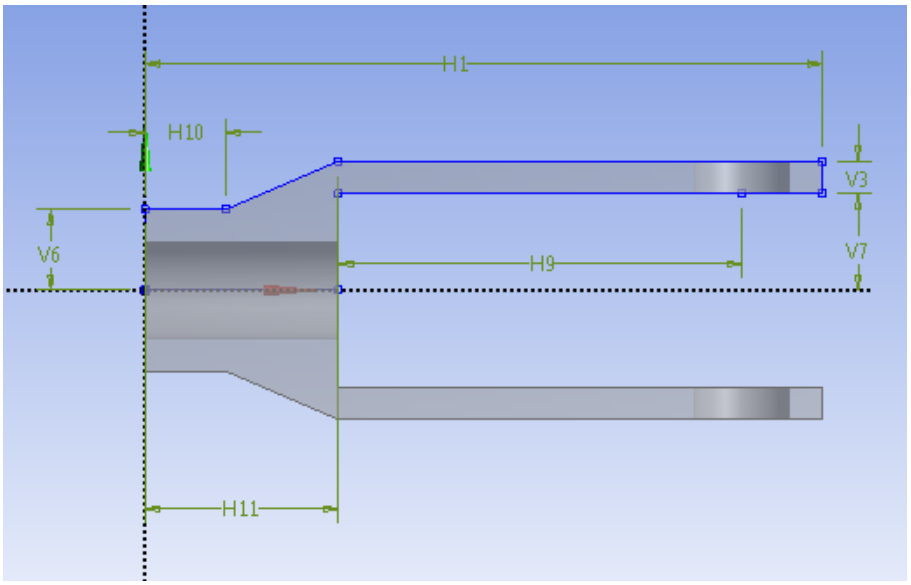
1 Introduction

TODO

2 Program modelling

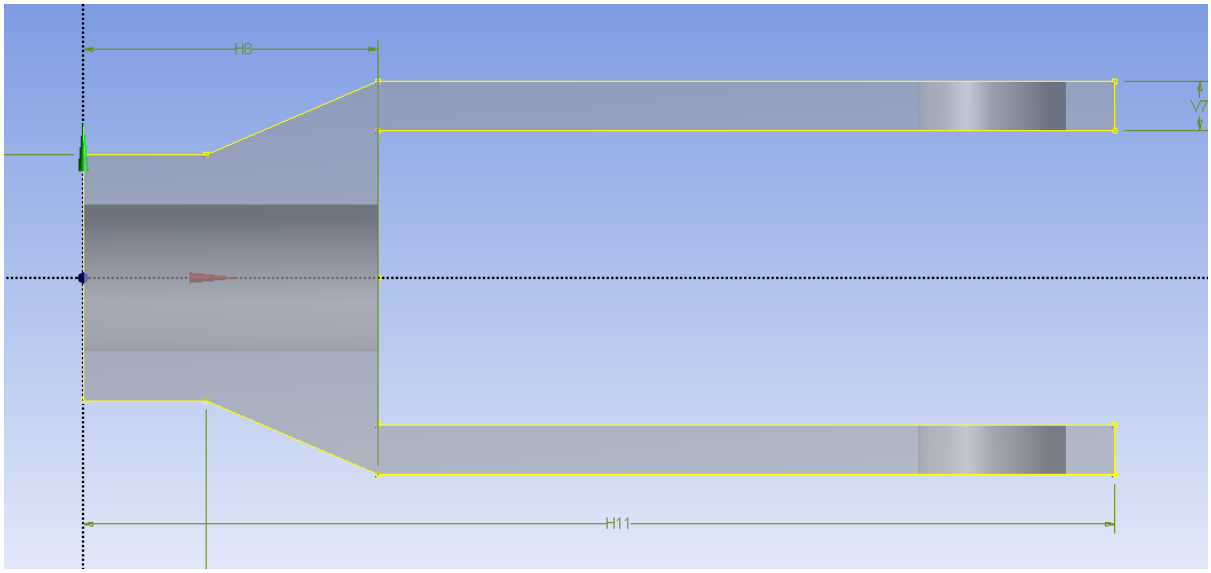
2.1 Geometry

The top view of original design is shown below:

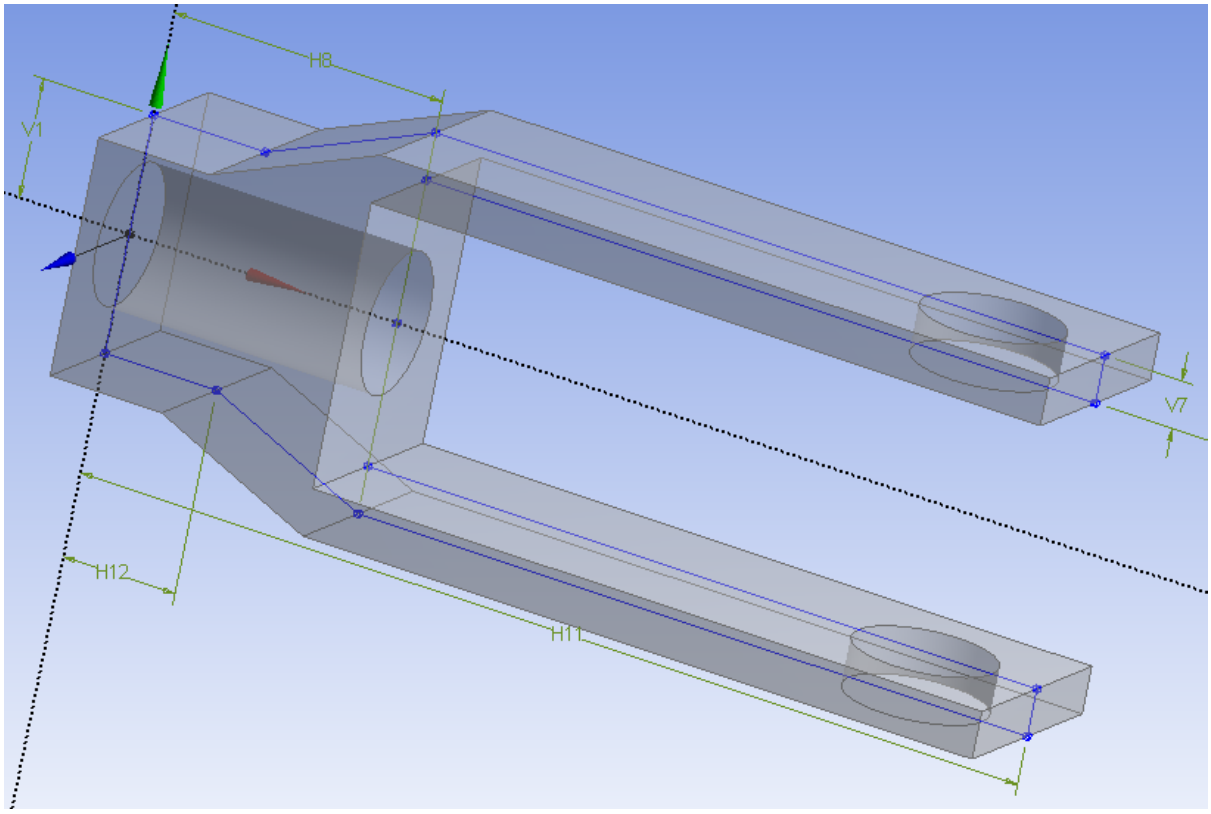


Where $H1 = 42cm$, $H10 = 5cm$, $H11 = 12cm$, $H9 = 25cm$, $V3 = 2cm$, $V6 = 5cm$, $V7 = 6cm$, diameters of all holes are $6cm$.

It is resembled as below:



The 3D model built is then as below, where the height of the component is assumed to be $10cm$:

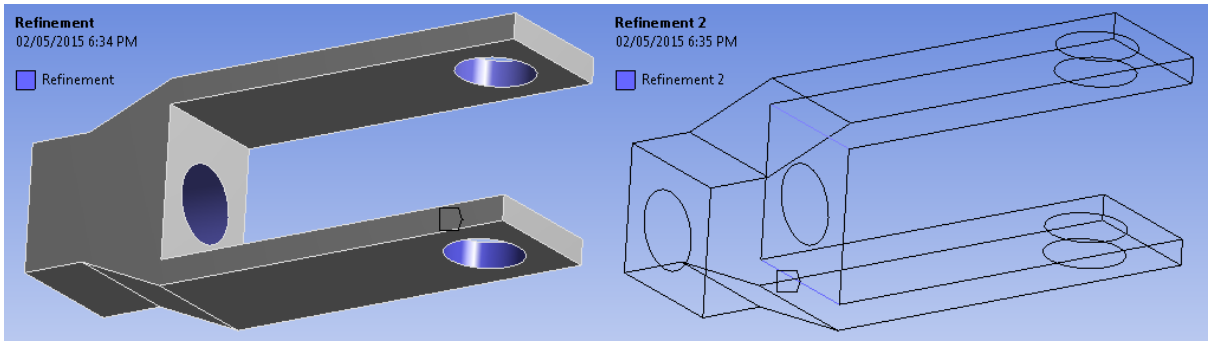


The boundary conditions are the loads, where symmetric properties on both axis can be assumed.

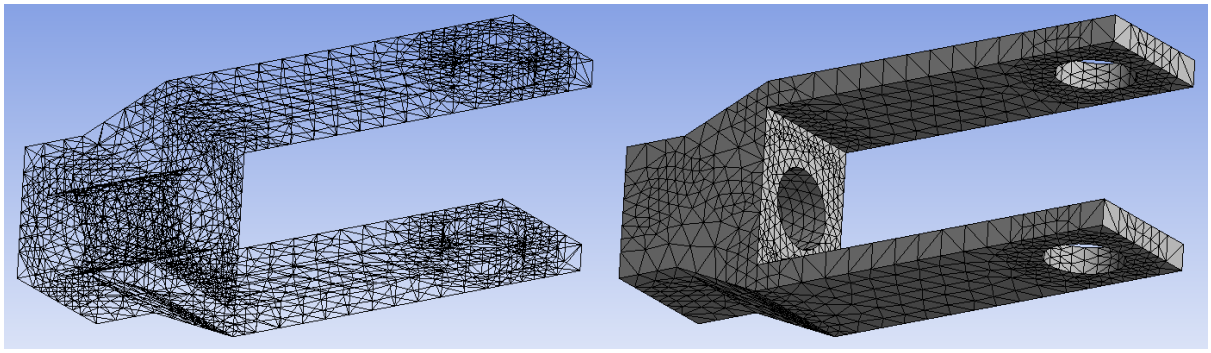
3 FEM analysis

3.1 Mesh setup

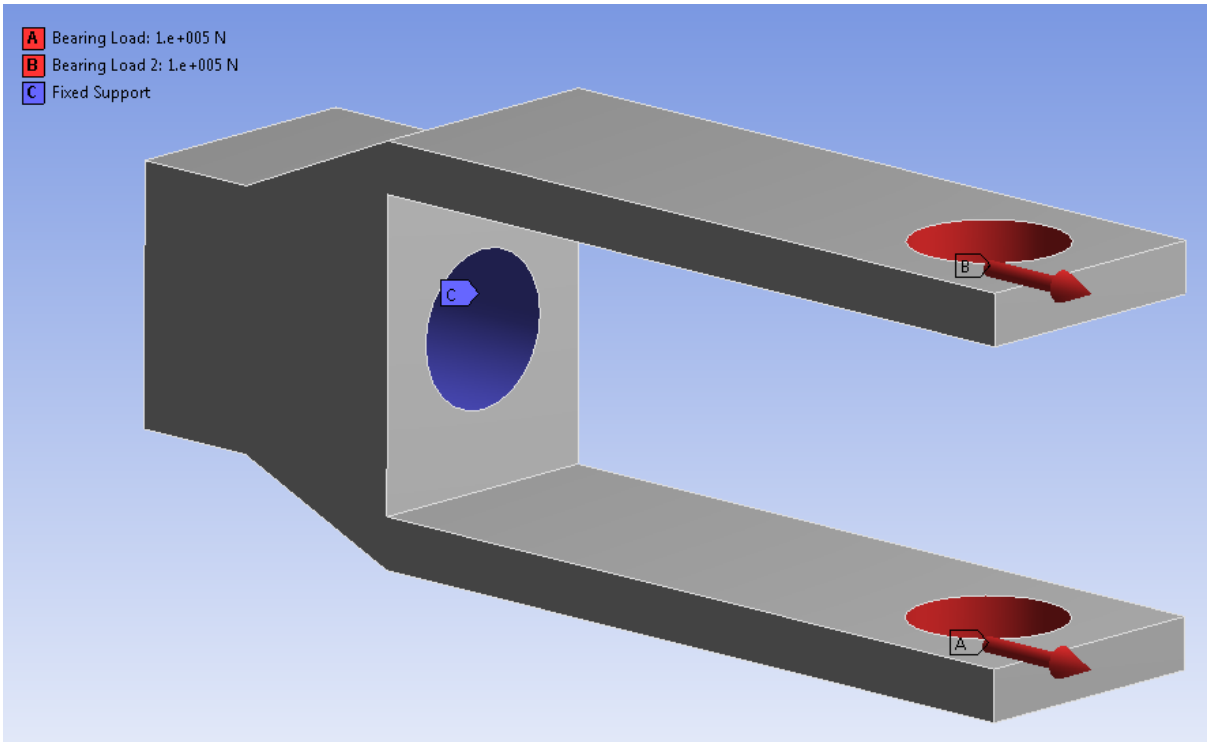
For the mesh, two refinements are added as below, where the first one (*Refinement*) is for the cylindrical surface of loading, and the second one (*Refinement 2*) is for the sharp edges of 90 degree where stress concentration might occur.



The overall mesh with a size of $2cm$ is shown below:

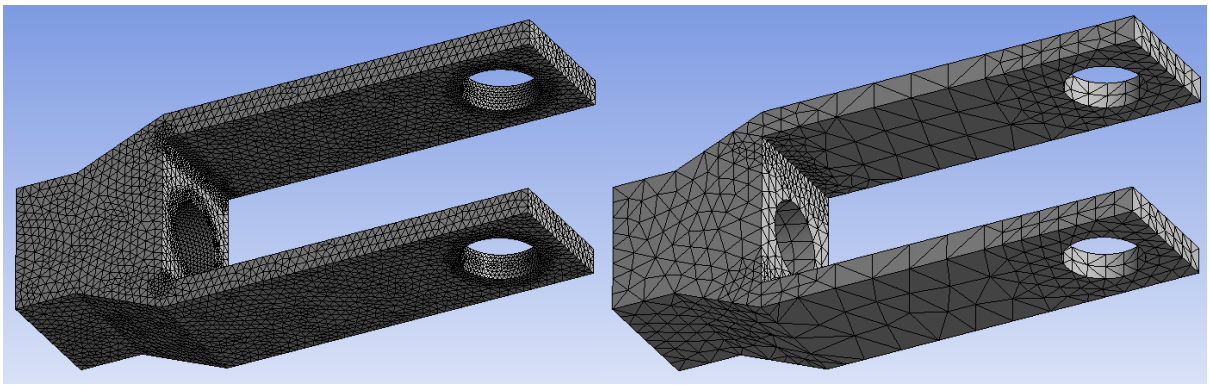


3.2 Boundary conditions setup

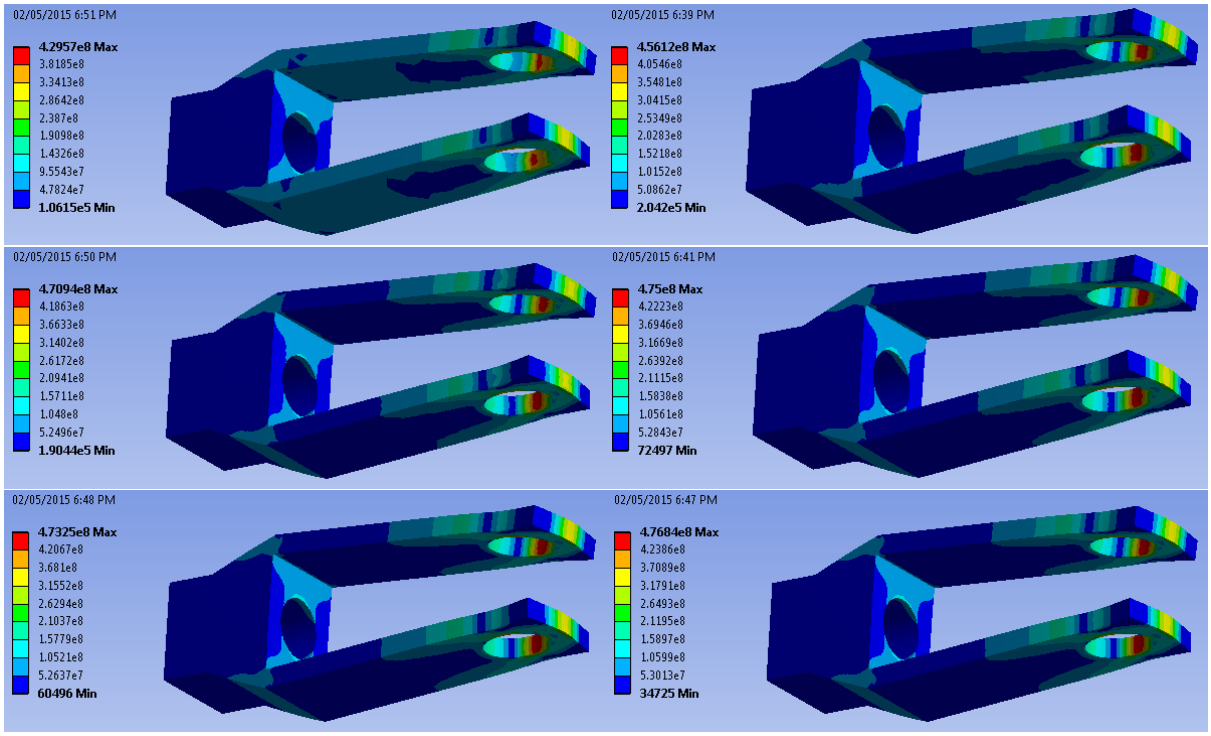


3.3 Convergence study

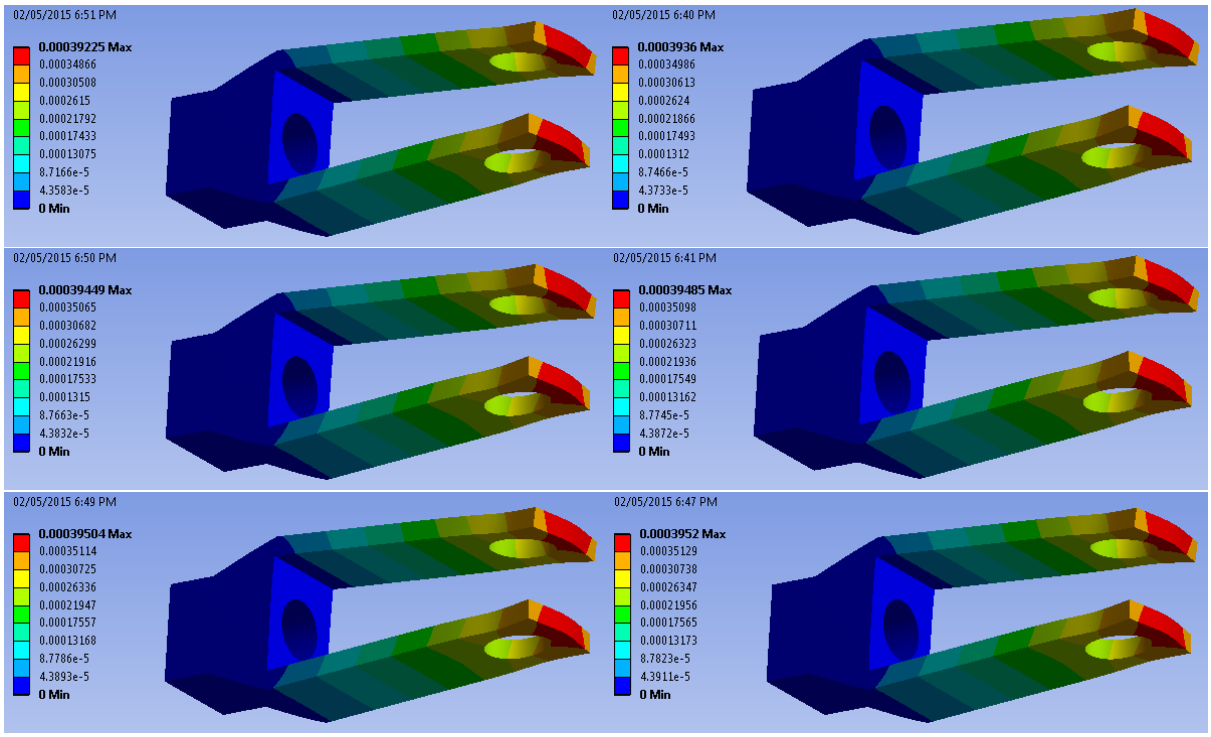
For convergence study, mesh sizes of $3cm$, $2cm$, $1.5cm$m $1cm$, $0.8cm$ and $0.65cm$ are used. The mesh of minimum ($0.65cm$) and maximum ($3cm$) mesh size are shown below:



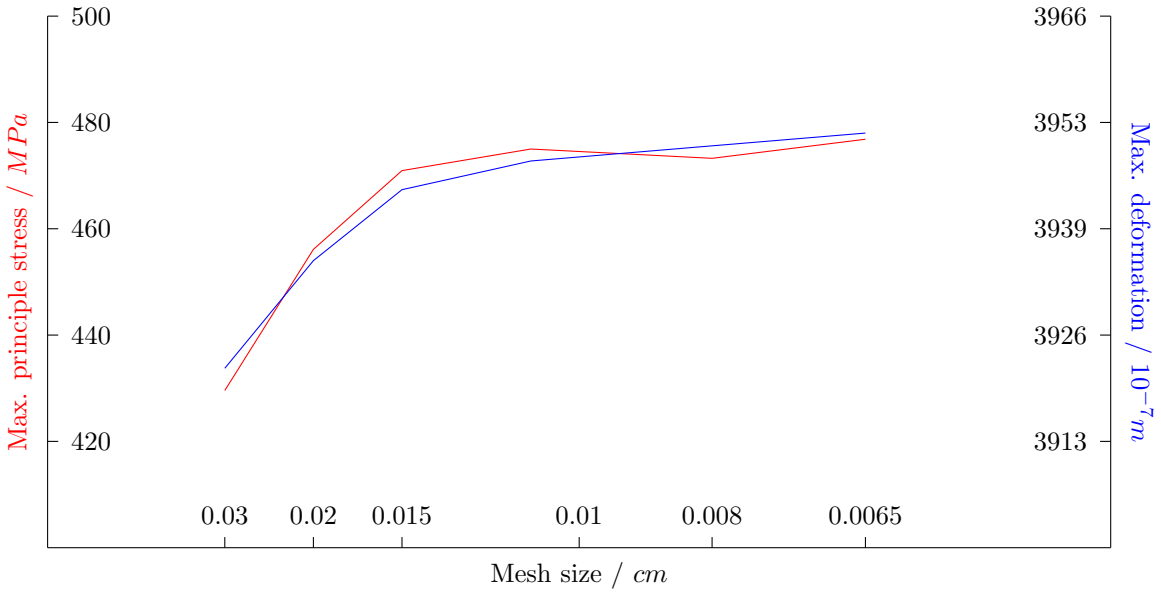
The results for principle stresses are below, listed in size-decreasing order.



The results for deformations are below, listed in size-decreasing order.



The change of both results with mesh sizes can be plotted below (x-axis in reciprocal scale):


500
480
460
440
420
3966
3953
3939
3926
3913
Max. principle stress / MPa
Max. deformation / $10^{-7}m$
0.03
0.02
0.015
0.01
0.008
0.0065
Mesh size / cm