**MODAL ANALYSIS OF INTAKE MANIFOLD OF A CARBURETOR**

**ABSTRACT**

This project focused on simulation testing of automotive intake manifold design using Computer Aided Engineering software. Finite element Random Vibration analysis is conducted on intake manifolds designs for material cast aluminum alloy AlSi12 and glass-fibred polyamide PA6 GF30. The purpose of this project is to study the computational maximum stress on the model due to the effect of engine vibrations and pressure pulsation loads. The input data of vibrations and pressure pulsation loads are taken from studies. The software used is SOLIDWORK PREMIUM 2014. Based on the simulation results obtained, the maximum stress of both materials is compared to distinguish which is better in resisting the vibration loads applied.