**STRUCTURAL STATIC ANALYSIS OF CYLINDER HEAD**

**ABSTRACT**

The main aim of this work is to predict the design performance based on the stress/strain and behavior of cylinder head under various operating conditions. The effects of engine operating conditions such as combustion gas maximum internal pressure, components on the stress and thermal stress behavior of the cylinder head have been analyzed. The analysis was carried out using a finite element analysis (FEA) software package, which is use to simulate and predict the Von-Mises stress and strain pattern and thermal distribution of the cylinder head structure during the combustion process in the engine and the geometry modeling was carried out using a popular computer-aided engineering tool, **Solid works**. The result can be used to determine the quality of the design as well as identify areas which require further improvement. In this investigation, structural analyses of the cylinder head highlight several areas of interest. The maximum stress is found not exceeding the material strength of cylinder head, and thus the basic design criteria, namely no yielding and no structural failure under firing load case, can be satisfied. In addition, the effect of thermal stress/strain provides a good indication on structural integrity and reliability of the cylinder head, which can be improved in the early stages of design. This steady-state finite element method (FEM) stress analysis can play a very effective role in the rapid prototyping of the cylinder head.