THERMO-STRUCTURAL DESIGN AND FABRICATION OF A TWO-STROKE IC ENGINE PISTON

ABSTRACT

Internal Combustion (IC) engines, produces work by the combustion of fuel and air, i.e. by converting the chemical energy of the fuel into mechanical energy of the reciprocating piston. The main components of an IC engine are the cylinder, cylinder head, piston, piston rings, piston pin, crank and crank shaft. The liquid fuel on combustion, phase changes into gaseous state, which impinges on the piston and pushes it to cause reciprocating motion. The reciprocating motion of the piston is then converted into rotary motion of the crankshaft with the help of connecting rod. IC engines are used in marine, locomotives, aircrafts, automobiles and other industrial applications.

During the combustion of fuel in an IC engine very high temperature of the order of 2100-2200°C is produced inside the combustion chamber. The pressure generated inside the combustion chamber during its cycle of operation is 50-60 bar. This high pressure and temperature exerts a severe thermo structural loads on an IC engine components namely combustion chamber, piston and valves.

The combustion chamber is designed for the pressure loads and is provided with adequate number of fins for heat transfer so as to prevent seizure of the IC engine sub-parts. Similarly the piston is designed with high strength-to-weight ratio material to withstand the thermo structural shock of the combusting fuel and to pass the mechanical energy to the crank and crankshaft.

Fins are provided on the exterior of an IC engine to optimize the performance, and the efficiency of an IC engine, by providing exterior extended cooling surfaces. These fins are designed to optimize the heat loss and the temperature inside an IC engine such that the thermal efficiency is optimal and also the weight of the engine is not increased beyond optimal level.

This study is an attempt to understand the effects of the piston geometry and material on its functional requirement using commercially available Finite Element Analysis (FEA) code ANSYS 14.0. Varying trends of these parameters are tabulated and determined and the values, which give optimized piston geometry, are
documented. Further, an effort has been made to optimize the fin profile and fin array parameters for a given heat flux using GAMBIT and FLUENT/ANSYS.

In the first phase of the project the problem statement is formulated along with literature survey and the initial drafting modeling is completed along with initial FEA studies. The same problem statement will be analyzed and studied in detailed during the second phase of the project.