

DESIGN OPTIMISATION AND ANALYSIS OF CONNECTING ROD OF VEHICLE ENGINE USING CAE TOOLS

SHRIPRASAD CHORGHE¹, SIDDHARTH DHOLAM², SWAPNIL KAMTHE³,
SAHILI SALVE⁴ & A. D. DHALE⁵

^{1,2,3,4}Project Members, Jai Trimurti CHS, Mumbai, Pune, Maharashtra, India

⁵Associate Professor, Jai Trimurti CHS, Mumbai, Pune, Maharashtra, India

ABSTRACT

The main aim of the project is to analyse various stresses and fatigue parameters acting on connecting rod, optimise shape and weight. Connecting rod is an intermediate & important engine component which connects piston & crankshaft. It is subjected to multiple compressive & tensile forces. Major consideration in this case is gas force. The high magnitude gas force is responsible for many kinds of failure. These failures need to be prevented & for this purpose analysis was needed to be done. In this project, connecting rod of Hero Honda splendor is chosen as a model for study whose dimensions data belongs to P. G. Charkha & S. B. Jaju's research paper. This project considers two cases, first, static load stress analysis of the connecting rod, and second, optimisation for weight. In this project analysis is done on four stroke single cylinder petrol engine connecting rod. The model was developed in SOLIDWORKS software, saved in IGES format and then imported to ANSYS workbench. Using ANSYS workbench 11 model was analysed for various stresses by applying suitable boundary conditions & using different modules of ANSYS workbench 11. The Von Misses stresses, shear stresses, elastic strain, total deformation and various fatigue parameters like life, damage, safety factor, biaxiality indication, equivalent alternating stresses, etc. are analysed. Here two materials were studied for their performance, viz., Structural Steel & Aluminum Alloy. Shape and weight optimization was done for both the materials. Aluminum being light in weight and having more yield strength became the suitable material. The results obtained from the stress analysis were used to modify the design of existing connecting rod, so that better performance i.e. reduced inertia, fatigue life and manufacturability can be obtained under varying load conditions.

KEYWORDS: ANSYS Workbench, Connecting Rod, FEA, Optimisation, Static Load, Stress Analysis