

CEST02\_219

## ANALYSIS AND OPTIMIZATION OF CONNECTING ROD

<sup>1</sup>Altayeb Mahfouth\*, <sup>2</sup>Muhand Murchid

<sup>1,2</sup>Dept of Mechanical Engineering, Sabratha University  
, Altayeb\_M85@yahoo.com, miral.muhand@gmail.com

### ABSTRACT

In this study, a Finite Element Analysis and design optimization study were carried out on a connecting rod using Cad software Solidworks. First, the connecting rod was modeled in Solidworks. Then, two finite element models for tensile and compressive loads were analyzed. The sensitivity analysis was conducted; a convergence has been achieved with 1.5 mm uniform element length. The peak stresses mostly occurred in the transition area between pin end, crank end, and shank region. The obtained results from the stress analysis were used in the optimization study. The optimization study was carried out to investigate weight reduction opportunities by modifying some connecting rod dimensions with constraining allowable stresses and factor of safety. The percentage weight reduction obtained was 11.88% by optimization with a minimum factor of safety having a value of 2.2 and maximum Von Mises stress having a value of 127 MPa. The maximum stress occurred in the static structural analysis are less than the yield strength of the material.

**Keywords:** Connecting Rod, Solidworks, Modeling, Finite Element Analysis, Optimization