

STATIC STRUCTURAL ANALYSIS OF TRUCK CHASSIS

S. Venkateswarlu¹, G. Narendra², B. Sriram Achari³, G. V. Lahari⁴, K. Umesh Chandra⁵ & S. Anwar Basha⁶

¹Professor, Department of Mechanical Engineering, GPCET, Kurnool, Andhra Pradesh, India

^{2, 3, 4, 5, 6} Research Scholar, Department of Mechanical Engineering, GPCET, Kurnool, Andhra Pradesh, India

ABSTRACT

Automobile chassis is one of the major elements of road motor vehicles. Normally automobile is divided into two parts they are Body and Chassis. Thus this chassis design and analysis form the most important stage of vehicle manufacturing. It is a basic structure of a vehicle. Chassis is the integral part of the vehicle it bears the entire load of the vehicle including its self-weight. A weight of chassis is one of the major influencing factors & it must be strong enough to resist shocks, twist, vibrations and stresses [1]. In designing of chassis frame maximum stress and maximum deflections are the important factors. Static analysis of chassis is carried out by finite element method and we obtain the results like stress & deformation. The most suitable method for analysis is the finite element method [2]. A three-dimensional chassis model is created in Creo 5.0 imposed in ANSYS 18.2. Finite element analysis truck chassis was carried out using ANSYS & its static behavior was studied taking 3 different types of materials like Steel, Aluminum alloy, Carbon steels

KEYWORDS: *Ansys, Automotive Vehicles, Creo 5.0, Finite Element Analysis, Twisting*