Static Loading Analisis in Design Tubulas Space Frame Prototype For Electric Vehicle Using Solidworks 2016

by
Indra Purwanto
Study Program of Automotive Engineering, Majoring of Engineering
The State Polytechnic of Jember

ABSTRACT

Strength and safety can be improved in determining the type of vehicle frame, choosing the type of material used, selecting profiles, safety factors, and the production process. Significant developments in computer technology have resulted in vehicle frame design problems being very easy to work on before the final prototype is made. Thus a component can be tested and evaluated before being manufactured. In this study, an analysis of the design plan of the tubular space frame was carried out using the Solidworks 2016 software with the Finite Element Analysis method to obtain the stress, deflection, and safety factor values, for the allowable stress in this study was calculated manually. This study aims to determine the value of the strength of the Tubular Space Frame and to analyze the strength of the frame, the material or materials used in this study are Carbon Steel AISI 4130, ASTM A53, and ASTM A238. The results of the analysis of the strength of the frame assuming a load of 256 N on the AISI 4130 Carbon Steel material, the maximum stress value is 29.7 Mpa, the deflection is 0.327 mm, and the safety value is 2. The ASTM A53 Carbon Steel material gets a maximum stress value of 29.8 Mpa. the deflection value is 0.315 mm, and the safety value is 4, while the ASTM A238 material gets a maximum stress value of 29.6 Mpa, the deflection occurs 0.334 mm, and the safety value is 2. The allowable stress of AISI 4130 carbon steel material is 230 Mpa, the material ASTM A53 carbon steel is 120 Mpa, and ASTM A283 is 82,5 Mpa.

Keywords: *stress, deflection, safety factor*