IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 5, Issue 9, June 2025
Impact Factor: 7.67

Finite Element Analysis of Two Wheeler Rim By Using Composite Material

Dr J. Kingston Barnabas[1], Saran S[2], Dhivan M[3], Santhosh P[4], Vignesh S[5]
Department of Mechanical Engineering[1-5]
Anjalai Ammal Mahalingam Engineering College Kovilvenni, Tamilnadu, India

Abstract: *The cylindrical cam and follower mechanism is highly used in the packaging industry to obtain the sequential operation of material filling in the packaging machine. The occurrence of frictional wear between the cam and follower involves the reduction of cam life that eventually affects the quality of packing in package industry. In this work, an attempt is made to simulate the dynamic analysis with different material. Dynamic simulations are made for the cylindrical cam follower arrangement using a commercially available finite element code ANSYS. The simulation of analysis is carried out to predict the deformation of cam. The simulation of dynamic analysis is also conducted to investigate the effect of vibration on the performance of cam follower mechanism during a packing operation.*

Keywords: Rim wheel, Finite Element Analysis, ANSYS, Dynamic Analysis

DOI: 10.48175/568