

DESIGN AND STRUCTURAL ANALYSIS OF DISC BRAKE IN AUTOMOBILES

MAHMOODHASANDAKHIL¹, A. K. RAI², P. RAVINDER REDDY³ & AHMEDABDULHUSSEIN JABBAR⁴

^{1,2,4}Department of Mechanical Engineering, SHIATS -DU, Allahabad, Uttar Pradesh, India

³Professor & Head, Department of Mechanical Engineering, Chaitanya Bharathi Institute of Technology,
Hyderabad, Andhra Pradesh, India

ABSTRACT

This paper presents optimized design of performance of disc brake using finite element analysis is to evaluate the performance under severe braking conditions. Cast iron and stainlesssteel are used as disc brake materials. ANSYS 12.0 is a dedicated Finite element package used for determining the temperature distribution, variation of stresses and deformation across the disc brake. It has been made to investigate the effect of the temperature distribution with the deformed shape and stress distribution of disc brake rotor design by using different braking conditions. From the results of the above data, the service life and long term stability is ensured. A steady static structure analysis has been carried out to investigate the temperature variation across the disc using the axis symmetric finite elements. Further structural analysis is also carried out by Coupled Field Analysis. An attempt is also made to suggest a best combination of material, flange width and wall thickness used for disc brake rotor, which yields a low temperature variation across the rotor, less deformation and minimum Von-mises stress possible.

KEYWORDS: Disc Brake, Structural Analysis, Finite Element Method, Ansys