Structural analysis of brake disc using dynamic simulation

ABSTRACT

This paper deals with the natural frequency simulation test of disc brake functionality using computer aided engineering software. The finite element analysis technique is applied to predict the failure region on the brake disc and to identify the critical locations of disc brake. The disc brake rotor implemented on the front axle of Perodua Myvi 1.3 L model with grey cast iron materials which commonly used in industry was studied. The disc brake rotor surface thicknesses on both sides were reduced into 3 steps, 0.5 mm, 1.0 mm and 1.5 mm by using CATIA V5 software. Modal analysis was applied for each type of disc brake rotor model including normal brake disc surface to investigate the natural frequency for each type of brake disc rotor model. The results of natural frequency from each type of the disc brake rotor surface thickness were recorded. 24 types of mode shapes were obtained from the simulation and compared with 4 types of surface thickness. The same mode shape shows that the highest reduction mass becomes the lowest value of frequency and the highest mass becomes the higher value of the frequency. It is due to the effect of Inertia Force. However, at the last mode shape on highest reduction of mass shows, the frequency was maximum. It is predicted that the instabilities are due to the repetitions of force applied to the rotor area.

Keyword: CATIA; ANSYS; Inertia force; Natural frequency; Mode shape