ABSTRACT

The essential of a car wheel rim is to provide a firm base on which to fit the tyre. Its dimensions, shape should be suitable to adequately accommodate the particular tyre required for the vehicle. In this project a tyre of car wheel rim belonging to the disc wheel category is considered. Design is an important industrial activity which influences the quality of the product. The wheel rim is modelled by using modelling software SOLIDWORKS 2016. By using this software, the time spent in producing the complex 3-D models and the risk involved in the design and manufacturing process can be easily minimised. So the modelling of the wheel rim is made by using SOLIDWORKS. Later this wheel rim modal is imported to ANSYS WORKBENCH 16 for analysis work. ANSYS WORKBENCH 16 is the latest software used for simulating the different forces, pressure acting on the component and also calculating and viewing the results. By using ANSYS WORKBENCH 16 software reduces the time compared with the method of mathematical calculations by a human. ANSYS WORKBENCH 16 static structural analysis work is carried out by considering four different aluminium alloys namely 6061, 6066, 7049, and 7075 their relative performances have been observed respectively. In addition to wheel rim is subjected to modal analysis, part of dynamic analysis is carried out and its performance is observed. In this analysis by observing the results of both static and dynamic analysis obtained, the best material is suggested for making an effective rim structure.