**Abstract**

The purpose of the car wheel rim provide’s a firm base on which to fit the tire. Its dimensions, shape should be suitable to adequately accommodate the particular tire required for the vehicle. In this study a tire of car wheel rim belonging to the disc wheel category is considered. Design in an important industrial activity which influences the quality of the product. The wheel rim is designed by using modelling software Solidworks2016. In modelling the time spent in producing the complex 3-D models and the risk involved in design and manufacturing process can be easily minimised. So the modelling of the wheel rim is made by using SOLIDWORKS. Later this SOLIDWORKS model is imported to ANSYS for analysis work. ANSYS software is the latest used for simulating the different forces, pressure acting on the component and also for calculating and viewing the results. A solver mode in ANSYS software calculates the stresses, deflections, bending moments and their relations without manual interventions, reduces the time compared with the method of mathematical calculations by a human. ANSYS static analysis work is carried out by considered two different materials their relative performances have been observed respectively. In addition to this rim is subjected to vibration analysis (modal analysis), a part of dynamic analysis is carried out its performance is
observed. In this paper by observing the results of both static and modal analysis obtained forged steel is suggested as best material.