Analysis of a 440C steel roller bearing using finite element method

Muhammad, Aisha[a, c]; Ali, Mohammed A.H.[a]; Shanono, Ibrahim Haruna[b, d]; Abdullah, Nor Rul Hasma[b]

[a] Faculty of Manufacturing Engineering, Universiti Malaysia Pahang, Malaysia
[b] Faculty of Electrical Engineering, Universiti Malaysia Pahang, Malaysia
[c] Department of Mechatronics Engineering, Bayero University, Kano, Nigeria
[d] Department of Electrical Engineering, Bayero University, Kano, Nigeria

ABSTRACT
One of the important component of working machines or conveyer belt is the roller bearing. They are applied in different sectors ranging from automotive to heavy duty machineries in high speed drills. Their performance greatly affects the machines. This study illustrate the analysis of the contact stress developed in a roller bearing using ANSYS software. The Finite Element simulating results revels maximum stress distribution value of 6.597E-09 MPa under the applied loading condition appearing on the roller balls. The total moment for the variation to time is 2.5868E-07 experienced at 7.7 s while the relative rotation in the Z-direction is at its peak at 6 s. Therefore, this study shows that FEA in ANSYS has provide a faster and cheaper method compared to the production of numerous prototypes.