Analysis of the Lower Control Arm in a Front Suspension System Using Finite Element Analysis and Analysis of Variance (ANOVA) Method(Book Chapter)

Ridzwan, A.N., Kadir, A.R.A., Azid, I.A., Hamid, M.N.A., Razak, T.A.A., Ahmad, M.N.

Abstract:
In this paper, the existing lower control arm structural analysis and performance were investigated and analyzed by the implementation of reversed engineering method used in designing the three-dimensional model referring to the actual part. The initial design is modeled by using SolidWorks 2016 software and validated by using modal analysis method in Abaqus CAE. The natural frequencies can be compared with the load impact testing experiment to the actual part physically to determine the percentage deviation. After the model was validated, Abaqus CAE software used to analyze the structural strength-ability in a static condition. The result will be the main reference in creating the new design concept of the ASHIMORI lower control arm as the Abaqus CAE will be used for the finite element analysis process. The target safety factor of the all-new design must be lower than the actual concept which is defined between the different design and material. The main factor will be determined by using Analysis of Variances (ANOVA) method and the final design will be compared with the actual part focusing on the maximum stress value, performance and structural analysis in a static state. © 2022, The Author(s), under exclusive license to Springer Nature Switzerland AG.