317/C

Finite element analysis of an excavator bucket to satisfy the industrial requirement

A.G.P. Lankathilaka, Nimali T. Medagedara*, and P.D.S.H. Gunawardane

Department of Mechanical Engineering, Faculty of Engineering Technology,
The Open University of Sri Lanka, Nawala, Nugegoda, Sri Lanka.

An excavator is a machine used for construction activities. This consists of several components such as house, undercarriage, boom, stick and bucket. Excavators are found in many different sizes depending on the job that is required. Also the various types of buckets are used for different types of work. The aim of this research was to design and simulate an adjustable excavator bucket for mini wheel excavators. Design was done according to the specific industrial requirement, and the parameters of the bucket, such as bucket capacity and dimensions of the bucket, were theoretically calculated and some parameters were chosen from the manual guide of the selected brand. AISI4130 Alloy Steel was used for the design. The bucket was modelled and analyzed using SolidWorks modelling software.

SolidWorks static analysis was used for the simulation, and boundary conditions are selected after hand calculations for the bucket. Different mechanisms were used for the adjusting and optimizing, and most suitable mechanismselected. Different boundary conditions such as load, fixtures, and pressure are the common such conditions of this simulation. The forces are defined such as digging force applied direction, and hinge plate holes used as the fixed points. The maximum digging force engaged from the main cylinder the same as arm and bucket cylinders, into the cutting edge of the bucket. These forces are defined by the selected excavator user manual. The displacement and stress strain failure were analysed, considering the practical application of the excavator bucket. After interpreting FEA results, stress displacement and strain are found to be within allowable stress limit. Moreover, design modification are made for reduced displacement by attaching the wear plates.

This paper discusses the stress, strain and displacement analysis of the design under various boundary conditions. The static conditions were modelled using SolidWorks software package and 346171 3D tetrahedral solid elements and 556850 nodes were used for the simulation. The Project was limited for the mini wheel excavator clean-up buckets and the BOB CAT brand, model E55W mini wheel excavator was selected for the analysis.

Key words: SolidWorks, FEA-Finite Element Analysis

tmmed@ou.ac.lk Tel: +94 718571135