Preliminary Numerical Analysis of a Platform Structure

Asrul Khairi Sohaimi[1,a], Anwar P.P. Abdul Majeed[2,b] and Jamaluddin Mahmud[1,c,*]

[1]Faculty of Mechanical Engineering, Universiti Teknologi MARA 40450 Shah Alam, Selangor, Malaysia

[2]Faculty of Manufacturing Engineering, Universiti Malaysia Pahang

26600 Pekan, Pahang, Malaysia

[a]askrule8@gmail.com, [b]anwarmajeed@imamslab.com, [c]jm@salam.uitm.edu.my

Abstract

Finite element analysis (FEA) has become an increasingly important tool in evaluating structural performance of platforms. This includes the use of this tool to perform strength, stability checks, optimisation of structural design upon subjected to design loads as well as to perform failure investigation of platforms upon subjected to special loading conditions such as impacts. Traditionally, small and medium enterprises utilises laboratory (physical) testing and heuristic experience to fabricate and test the platforms, nonetheless this approach is deemed too time consuming and cost demanding. Therefore, this study aims at reconstructing the actual design based on engineering drawings as well as performing stress analysis by means of a commercial FEA software package, ANSYS v14.0 by investigating the effects of two distinct loading impacts on a mild steel platform. The maximum stress and maximum displacement values obtained via the FEA simulation was then compared with values obtained via theoretical computation. The results obtained via the FEA simulation was found to be in good agreement with the exact solution. The present study is non-trivial as it contributes towards the knowledge in the design and optimisation of
complex steel structures.

Keywords: Finite Element Analysis, Stress Analysis, Engineering Design, Platform Structure

DOI: https://doi.org/10.4028/www.scientific.net/AMM.680.280