عنوان مقاله:

Validation of use of FEM (Abaqus) for structural static analysis of loading platform

محل انتشار:

ششمین همایش بین المللی دانش و فناوری مهندسی برق، کامپیوتر و مکانیک ایران (سال: 1400)

تعداد صفحات اصل مقاله: 8

نویسندگان:

Masoud Pourghavam - *Student of Mechanical Engineering at Sadjad University of Technology, Mashhad, Iran*

Parisa Jodeir - *Student of Mechanical Engineering at Sadjad University of Technology, Mashhad, Iran*

خلاصه مقاله:

Loading Platforms are used on construction sites for shifting and transfer of materials at different levels of the building. It consists of a steel prop, fixed frame, guardrail, safety door, etc. The loading platform model was generated in SolidWorks ۲۰۲۱ and analyzed in Abaqus using finite element method (FEM). Throughout the analysis, the load has been applied at the end of the platform to generate maximum moment and stresses in the platform. Initially, the factors of safety were obtained using manual calculations for the platform. Later, the results for static analysis are obtained .using Abaqus. In static analysis, the platform's self-weight and payload are considered

کلمات کلیدی:

Loading Platform, FEA, Simulation, Analysis, Factor of Safety

لینک ثابت مقاله در پایگاه سیویلیکا: