

Finite Element Analysis and Multibody Dynamics of 6-DOF Industrial Robot

Authors : Rahul Arora, S. S. Dhami

Abstract : This paper implements the design structure of industrial robot along with the different transmission components like gear assembly and analysis of complete industrial robot. In this paper, it gives the overview on the most efficient types of modeling and different analysis results that can be obtained for an industrial robot. The investigation is executed in regards to two classifications i.e. the deformation and the stress tests. SolidWorks is utilized to design and review the 3D drawing plan while ANSYS Workbench is utilized to execute the FEA on an industrial robot and the designed component. The CAD evaluation was conducted on a disentangled model of an industrial robot. The study includes design and drafting its transmission system. In CAE study static, modal and dynamic analysis are presented. Every one of the outcomes is divided in regard with the impact of the static and dynamic analysis on the situating exactness of the robot. It gives critical data with respect to parts of the industrial robot that are inclined to harm under higher high force applications. Therefore, the mechanical structure under different operating conditions can help in optimizing the manipulator geometry and in selecting the right material for the same. The FEA analysis is conducted for four different materials on the same industrial robot and gear assembly.

Keywords : CAD, CAE, FEA, robot, static, dynamic, modal, gear assembly

Conference Title : ICMMRE 2017 : International Conference on Mechanical, Mechatronics and Robotics Engineering

Conference Location : Melbourne, Australia

Conference Dates : November 29-30, 2017