

عنوان مقاله:

Dynamic Modeling of a Parallel Flexible Manipulator based on the Abaqus Interface with MSC.ADAMS

محل انتشار:

کنفرانس بین المللی یافته های نوین پژوهشی در مهندسی صنایع و مهندسی مکانیک (سال: 1394)

تعداد صفحات اصل مقاله: 15

نویسنده:

Amin Valizadeh - Department of Mechanical Engineering Ferdowsi University of Mashhad, Mashhad, Iran

خلاصه مقاله:

Flexible manipulator's distinct dominance over rigid ones has drawn scientists' attention increasingly. This type of manipulators is widely used in an extensive range of applications starting from simple pick and place operations to micro surgeries and space manipulators. The deep perception of the dynamic model leads to accurate knowledge of manipulator's behavior. In the classic approach developed methods are required to solve the complex nonlinear equations which are more tough in closed chain mechanisms. In addition, the equations are much more complicated in flexible structures. A new method of dynamic modeling and simulation of a 2-DOF parallel manipulator with flexible links based on the interface between a finite element modeling software (Abaqus) and a multibody dynamics simulation software (MSC.ADAMS) is proposed in this paper. The CAD modeling and translating the kinematic constraints are presented in detail. Finally the simulation results of the structural model and mathematical model are compared by implementing a full state feedback controller in Matlab/Simulink block diagram. The results demonstrate the method for parallel manipulators with flexible structures is beneficial and practical in industrial applications as well as mathematical nonlinear equations.

کلمات کلیدی:

Planar Manipulator, Flexible links, Nonlinear Equations, Finite Element Modeling, closed chain mechanisms

لینک ثابت مقاله در پایگاه سیویلیکا:

<https://civilica.com/doc/430011>

