Available online at www.sciencedirect.com

SciVerse ScienceDirect

Procedia CIRP 3 (2012) 501 – 506

www.elsevier.com/locate/procedia

45th CIRP Conference on Manufacturing Systems 2012

On an Empirical Investigation of the Structural Behavior of Robots

C. Doukas[a], J. Pandremenos[a], P. Stavropoulos[a], P. Foteinopoulos[a], G. Chryssolouris[a],*

[a]*Laboratory for Manufacturing Systems & Automation, Department of Mechanical Engineering & Aeronautics, University of Patras, 26500, Greece*
* Corresponding author. Tel.: +30-261-099-7262; fax: +30-261-099-7744.*E-mail address*: xrisol@lms.mech.upatras.gr

Abstract

In the paper, the structural behavior of industrial robots is investigated. The objective is the development of a model, capable of predicting the robot's accuracy, under certain arm positions and loading conditions. The Finite Element Method (FEM) is used for the model's development. An extended investigation into the total robot accuracy of the joint effect is conducted. The accuracy of the robot, under ranging loads at different positions, has been mapped and discussed.

Keywords: Robot; Stiffness

1. Introduction

In order for the manufacturing industry to increase its production flexibility, it has been proposed that the milling machines be substituted with robots [1]. However, the accuracy and repeatability (+-0.07mm) of a typical robotic arm [2] is not as high as this of an ordinary CNC milling machine with typical value of ±0.016 mm or better [3]. Since the accuracy issues affect the quality of the final product, they have to be resolved in order for the penetration of machining robots to be increased in industry.

This paper focuses on the modeling of a robotic arm, in FE environment. The purpose is to simulate its behavior under various loading scenarios and create a visual representation of the robot's performance in its working envelop, in terms of accuracy.

Developments in machining and tool design technology, especially in milling operations, reflect the requirements for flexibility in order to adapt the changes taking place in the market and in the global economic environment [1]. Makhanov et al. [4] presented a new approach to tool-path optimization of milling robots, based on a global interpolation of the required surface by a virtual surface composed of tool trajectories. Kao et al. [5] presented a robot-based computer-integrated manufacturing (CIM) automation. Abele et al. [6] described the modeling of the robot structure and the identification of its parameters focusing on the analysis of the system's stiffness and its behavior during the milling process.

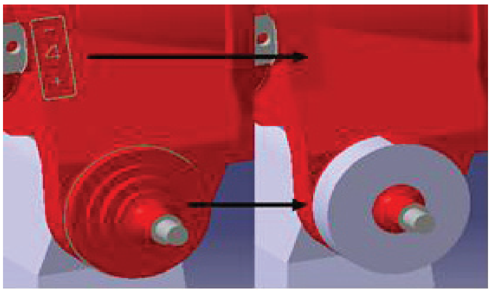

Fig. 1. Illustration of simplifications (right) performed on the robot model

2. Modeling approach

2.1. Geometry

The study was conducted using a six degrees of freedom robot of 130kg payload [2]. The CAD files were imported as STP file in a CAD environment in

doi:10.1016/j.procir.2012.07.086

order for the geometric model of the robotic arm to be created and then exported to the FE environment.

A number of geometrical simplification/modifications were required in order for the model to be correctly imported into the FE environment. These changes affect the meshing quality of the geometry as well as the computational time requirements. All changes were kept to the absolutely minimum, in order to allow correct meshing, but without affecting the accuracy of the results (Fig. 1)

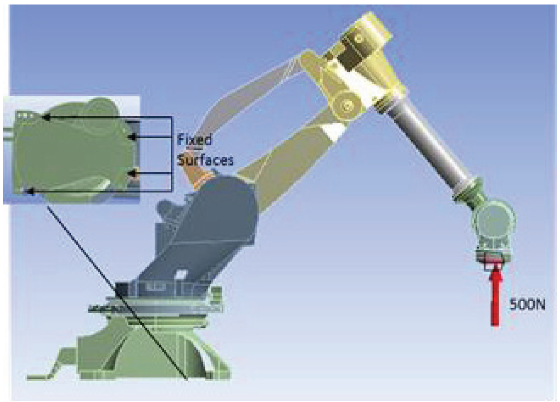

Fig. 2. Force application vector and applied constraints

2.2. FE Simulation

The geometric model was imported as a static case into FE environment in order to simulate the effect of the loads applied to the flange end-effector during a milling process.

All the parts are considered being deformable bodies and the material used is structural steel. The properties data set used is shown in Table 1 [7].

The loading scenario used in the given simulation is presented in Fig.2. The base of the arm is fixed and the appropriate load is applied to the robot end-effector.

All constraints between the assembly parts were added manually to the axes of rotation with the use of the tool of the FE environment provided.

For the original model, part connectivity was assigned to be Fixed or Revolute, which later allows the user to manipulate the static position of the arm so as for the simulation to be performed in different orientations.

Equivalent Stress (Von-Mises) and Total Deformation solutions were used for assessing the validity of the solutions with the default mesh element size. Deformation was selected as the target solution, while stress was added as a control to test the consistency of the solutions. These tests were run with the same loading condition in the initial robot Calibration Position and the modified orientation, Position (Fig. 3).

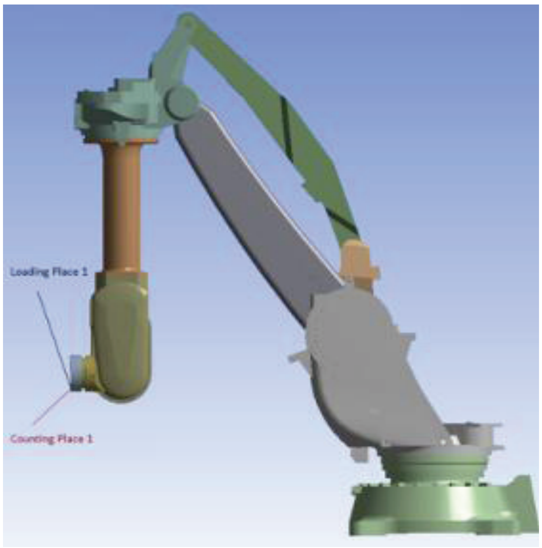

Fig. 3. Simulation position

The results of the FE simulation are shown respectively below visually depicting the areas of increased stress and deformation by a color scale. The maximum and minimum stresses and deformations have also been tagged (Fig. 4a, Fig. 4b).

Table 1. Mechanical properties of the applied material

Property	Value
Tensile Yield Strength	250 Mpa
Compressive Yield Strength	250 MPa
Tensile Ultimate Strength	460 Mpa
Young Modulus	200GPa
Poison ratio	3,00E-01
Density	7850 kgm-3
Coefficient of thermal expansion	0,000012 c-1
Ductility coefficient	0,213
Ductility Exponent	-0,106
Strength Coefficient	920 Mpa
Strength Exponent	-0,106
Cyclic Strength Coefficient	1 Gpa
Cyclic Strain Hardening Exponent	0,2

The final element size selected was 0.03m.

In order for the model to further be developed and higher accuracy level to be achieved, three models with different configurations have been tested. The initial model (Model 1) was used as the base for the other two (improved) versions. The difference of each model lies in the connection type of the mating parts.