

عنوان مقاله:

Thin-Walled Tube Bending Process Simulation using ABAQUS Software and an Investigative Comparison between Simulation, Experimental and Stress Analysis Results

محل انتشار:

یازدهمین کنفرانس ملی مهندسی ساخت و تولید ایران (سال: 1389)

تعداد صفحات اصل مقاله: 8

نویسندگان:

M Salem - *Master Student of Mechanical Engineering, Isfahan University of Technology*

M Farzin - *Associate Professor, Department of Mechanical Engineering, Isfahan University of Technology*

M Kadkhodaei[✉] - *Assistant Professor, Department of Mechanical Engineering, Isfahan University of Technology*

خلاصه مقاله:

In this paper, the bending process of thin-walled tubes by rotary draw method is initially simulated in ABAQUS software. Then the necessary experiments are carried out, and the simulation results associated to the quality of the product in terms of changes in the wall thickness and ovality of the cross section as well as the required moment for tube bending are compared with the empirical findings. The comparison demonstrates the high accuracy of the simulation results. The required moment is also calculated by the stress analysis method, and an integrated discussion is made over the errors of simulation, experiment and analytical stress analysis.

کلمات کلیدی:

Rotary draw bending- Stress analysis-ABAQUS

لینک ثابت مقاله در پایگاه سیویلیکا:

<https://civilica.com/doc/98118>

