Asociación Argentina
de Mecánica Computacional

Mecánica Computacional Vol XXXVI, págs. 1481-1481 (resumen)
José G. Etse, Bibiana M. Luccioni, Martín A. Pucheta, Mario A. Storti (Eds.)
San Miguel de Tucumán, 6-9 Noviembre 2018

THERMOMECHANICAL STUDY OF MULTI-STAGE FORGING OF SEAMLESS STEEL PIPES

Tomás F. Schnetzer[a], Claudio A. Careglio[a,b], Carlos G. García Garino[a,b], Aníbal E. Mirasso[a,b] and Jean-Philippe Ponthot[c]

[a]*Facultad de Ingeniería, Universidad Nacional de Cuyo, Centro Universitario, Parque San Martín, 5500 Mendoza, Argentina, tschnetzer@fing.uncu.edu.ar, ccareglio@uncu.edu.ar, aemirasso@uncu.edu.ar, cgarcia@itu.uncu.edu.ar*

[b]*ITIC, Universidad Nacional de Cuyo, Centro Universitario, Parque San Martín, 5500 Mendoza, Argentina*

[c]*Aerospace and Mechanical Engineering Department, University of Liège, 13a Allée de la découverte, 4000 Liège, Belgium, JP.Ponthot@ulg.ac.be*

Keywords: Hot forging, seamless steel pipe, Plasticity, finite strains.

Abstract. Metal forging is an important industrial process, with manufacturing applications in the oil, automotive and aeronautic industries, among others. Generally, the process involves finite strains, as well as interactions between the manufactured product, the punch and the dies. Depending on the geometry of the obtained piece, several forging stages are required to attain the final product. Sometimes cold forging is enough, but in general it is necessary to use a hot process. In this work, the numerical study of the multi-stage hot forging process of seamless steel pipes is presented. To achieve this, the heating of the pipe and the several forging stages are simulated, as well as the cooling process. The process is simulated considering a short duration process and disregarding inertial effects. To model the material behavior and the complexity of the process, plasticity with finite strains is considered. The interactions of the contact surfaces of the piece, dies and punch are simulated using a frictional contact interface, for which a Coloumb model is used. By means of numerical simulation the loads acting on the punch, in conjunction with the plastic strain and temperature distributions in the pipe, are determined. Finally, the conclusions are presented.