Hot forging of cylindrical tubes. On the factors influencing lateral resisting forces

Carlos García Garino*, Claudio Careglio†, Luc Papeleux+, Anibal Mirasso† and Jean-Philippe Ponthot+

*† Facultad de Ingeniería e Instituto para las Tecnologías de la Información y Comunicación (ITIC)
Universidad Nacional de Cuyo (UNCuyo)
Centro Universitario, 5500, Mendoza, Argentina
e-mail: carlos.garcia.garino@ingenieria.uncuyo.edu.ar, web page: http://itic.uncu.edu.ar

+ Aerospace and Mechanical Engineering Department. LTAS-Computational mechanics.
Liège Université. Allée de la Découverte 13a
4000 Liège, Belgium
Email: JP.Ponthot@uliege.be L.Papeleux@uliege.be - Web page:http://metafor.ltas.ulg.ac.be/

ABSTRACT

Hot forging process is a very well-known industrial process. In this paper the hot forging of a circular cylindrical tube is discussed. The tube is encased by a cylindrical die and it is forged due to a punch that moves downwards the tube. In order to prevent the tube slips down, the punch load have to be equilibrated by vertical frictional forces that are produced at the interface between the tube and the die.

The frictional interface forces depends on several variables: i) a gap that closes the die over the tube and defines a normal force on the tube wall. This gap is denoted by u_g; ii) the length of the frictional interface in between the tube and the die and, finally, iii) the frictional coefficient in between the die and the tube walls.

The geometry and material properties are taken from the work of Schnetzer et. al. [1], numerical simulation is conducted with ANSYS code [2] based on a Finite Element Method [3]. Large deformation plasticity model is taken into account [4] and Coulomb model is chosen in order to model the frictional contact interface in between the tube and the die. Twelve different Finite Element meshes are considered in order to know how the mesh refinement affects the obtained results.

A parametric study about the frictional forces required to equilibrate the Punch load is conducted. In order to tackle that, one hundred different values for the gap u_g are chosen, varying from 0 to 0.2 mm, with intervals of 0,002 mm. Three different values are considered for the interface length L_g: 100, 150 and 200 mm. Finally six different values are chosen for the friction coefficient μ from 0.05 to 0.3 with 0.05 intervals. Consequently 1800 different elastoplastic large deformations finite element studies have to be simulated. In practice this suggests to perform a Computational Mechanics Parameter Sweep Experiments on a distributed computing infrastructure [5]

REFERENCES

[1] T. Schnetzer, C. Careglio, D. Benvenutti, C, García Garino and A. Mirasso, A, *Estudio numérico de cargas de forjado en tubos de acero sin costura*. IX EnIDI, Mendoza, 2017 (in Spanish)

[2] O.C. Zienkiewicz and R.L. Taylor, *The finite element method*, McGraw Hill, Vol. I., 1989, Vol. II, (1991).

[3] ANSYS® (2017). *Documentation Release 17.0*, Help System, Structural Analysis Guide ANSYS, Inc

[4] C. Careglio et. al. Global and Local Mechanical Responses for Necking of Rectangular Bars Using Updated and Total Lagrangian Finite Element Formulations, Mathematical Problems in Engineering. Volume 2016, Article ID 1729638, 12 pages. http://dx.doi.org/10.1155/2016/1729638

[5] E. Pacini et. al. A bio-inspired scheduler for minimizing makespan and flowtime of computational mechanics applications on federated clouds. Journal of Intelligent & Fuzzy Systems 31 (2016) 1731–1743. DOI:10.3233/JIFS-152094. IOS Press