8th International Conference on Physical and Numerical Simulation of Materials Processing (ICPNS)

14–17 October 2016

Seattle, Washington | Hosted by Purdue University

SESSION 1: MODELS AND METHODS, SALON A

Co-Chairs: Mengnie Li, Kunming University of Science and Technology; Langping Wang, Harbin Institute of Technology; Yanfeng Han, Shanghai Jiao Tong University

SUNDAY, OCTOBER 16, 2016

Design and validation of parametric modelling platform based on ABAQUS

Weiwei Wang; Binbin Jia; Nan Zheng; Chen Zhao, Harbin Institute of Technology at Weihai

ABSTRACT

Force–displacement-separated-control multi-point forming (FDSC MPF) is a newly developed technology for sheet metal forming. In this technology, displacement is controlled by the upper punches, and forming load is controlled by the lower punches, displacement–time curves and force–time curves are different when forming different curved surfaces. By this method, normal constraints can be applied on the sheet metal surface in the whole deforming process and cover the full surface. This gets an effectively control on the sheet metal forming defects especially on wrinkling. But such advantages of FDSC MPF bring great problems in modeling and simulation – hundreds of punch's heads move individually and each one rotates or swings with the forming of sheet in different directions and velocities. Furthermore, contact condition between punch's heads and forming sheet are also changeable and unsteady.

This paper introduces the modeling and FEM simulation of FDSC MPF based on ABAQUS. Python is used as programming language to secondary develop the software and to establish a simulation platform. This platform's functions are as follows (a) basic parameters input such as material performance, plate size (b) rapid and automatic assemble of MPF punches and sheet (c) automatic definition of interaction conditions, loading and meshing (d) to complete the modeling and analysis of various modes of MPF.

The platform established has been used to simulate different modes of MPF to verify its efficiency and accuracy. The results showed that the pre-treatment modeling for multi-point forming can be established automatically and quickly. The simulation can be accomplished within a receivable speed in ABAQUS.

In order to verify the function of this parametric analysis platform further, simulation of sail surface panels of FDSC MPF with different parameters are conducted and the experimental deformations are carried out. The simulating results are compared with that of experiment. Comparison showed a good consistency between the experiment and simulation.

KEYWORDS: modelling, parameterized, ABAQUS, multi-point forming