Modelling and Finite Element Simulation for FDSC Multi-Point Forming Based on ABAQUS

Wang Weiwei, Jia Binbin, Sun Tingting, Di he, Zhao chen

School of Materials Science and Engineering, Harbin Institute of Technology, P. R. China

Abstract

In recent years, a new flexible sheet metal forming method, named force-displacement-separated-control multi-point forming (FDSC MPF), has been developed. By the 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. At the same time it has the flexible forming features of stamping die rapid reconstruction. But such advantages of FDSC MPF bring great problems in modeling and simulation----hundreds of punch's heads move individual and each one rotates or swings with the forming of sheet in different directions and velocities. Further more, contact condition between punch's heads and forming sheet are also changeable and unsteady.

This paper introduces the modeling and FEM simulation of FDSC multi-point forming based on ABAQUS. Python is used as programming languages to secondary develop the software and to establish a simulation platform. An additional module was developed for the automatic modelling with the functions of: (a) basic parameters input such as material performance, plate size (b) rapid assembly of multi-point forming punches and sheet automatically (c) automatic definition of the analytical step and the output of variables and (d) automatic definition of interaction conditions, loading and meshing.

The additional module developed and the model established were used to simulate the forming of different curved surfaces such as saddle surface, cylindered surface and others to verify its efficiency and accuracy. The results showed that the pre-treatment modelling for FDSC multi-point forming can be established automatically and quickly. The simulation can be accomplished within a receivable speed in ABAQUS.

Simulation results give a detailed illustration about the process and the characteristics of sheet deformation under the condition of FDSC multi-point forming. The advantages of FDSC multi-point forming in inhibition of forming defects such as wrinkling and springback were confirmed. Future more, experimental deformations were carried out and the simulating results are compared with that of experiment. Comparison shows a good consistency between the experimental data and simulation data.

Keywords: modelling, FEM, multi-point forming, ABAQUS