Hot stamping process simulation based on reliable heat boundary condition

He Huang*, Qing Wang
Automobile Engineering Department, Hunan Financial & Industrial Vocational-Technical College, Hengyang, 421002, Hunan, P. R. China
*Corresponding author e-mail: 14158592@qq.com

Abstract. The finite element analysis is an essential precondition for the process design of hot stamping processes. The quality and the significance of the simulation results are strongly dependent on the heat boundary condition used in FE models. In this work, the heat boundary condition in hot stamping simulation was determined by analyzing the characteristics of heat transfer in the interface between the blank and the tools. The numerical simulation to the hot stamping process was made by using LS-Dyna software. It is found that the temperature changes of the hot stamped part are inhomogeneous. This results in the inhomogeneous mechanical properties.

1. Introduction
At recent years, the hot stamping has become more and more common technology in automotive industries to manufacture lightweight vehicle components with ultra high strength and weak springback [1-2]. However, the hot stamping process compared with room temperature forming is very complex. Plastic deformation, strong temperature drop and phase transformation of quenchable steel arise simultaneously in hot stamping process so that the appropriate process parameters before manufacturing cannot be easily determined in general [3-4]. Finite element (FE) simulation is the powerful tool for an analysis to thermo-mechanical coupling process and helps the manufacturer in understanding the effect of the various parameters in the hot stamping process better [5]. In this paper, the characteristics of heat transfer in the interface between the blank and the tools was firstly analyzed to effectively determine the heat boundary condition of hot stamping simulation. Then, FE simulation of hot stamping processes for sample part geometry was made by using LS-Dyna software to analyze and evaluate the processing technology of hot stamping.

2. FE Simulation

2.1. FE model
The FE model adopted for simulation was shown as Fig.1. The part before hot stamping was a rectangular blank, with 240 mm in length, 160 mm in width and 2 mm in thickness. After forming, a U-shape part, a representative shape of the safe components in auto body, was obtained and the stamping depth was 60 mm. FE simulations were carried out for the entire hot stamping process. The FE code LS-Dyna was used for running the simulations. The FE model used corresponds to a quarter of the actual geometry due to symmetry, as shown in Fig.1.
2.2. Heat boundary condition

For modeling the hot stamping process accurately, a reliable heat transfer boundary condition data is required. The hot sheet metal stamping process can be divided into three steps in the simulation: approach, forming and quenching phases. This causes the blank to be cooled heterogeneously during the forming. All modes of heat transfer occur between the different components existing in a hot stamping process notably: blank, die and punch. During the approach phase, the blank is cooled not only by radiation with ambient air, the punch and the die and convection with ambient air but also by the conduction with the die. The effective heat flux in this phase can be expressed as:

\[ q' = q_{\text{conv}} + q_{\text{rad}} + q_{\text{cond}} \]  

Where \( q' \) (W·m⁻²) is the effective heat flux in approach phase, \( q_{\text{conv}} \) (W·m⁻²) is the heat flux by the convection heat transfer, \( q_{\text{rad}} \) (W·m⁻²) is the heat flux by the radiation heat transfer, \( q_{\text{cond}} \) (W·m⁻²) is the heat flux by the heat conduction. \( q_{\text{conv}}, q_{\text{rad}} \) and \( q_{\text{cond}} \) can be respectively written as:

\[ q_{\text{conv}} = h_{\text{conv}} (T_{\text{blank}} - T_\infty) \]  

\[ q_{\text{rad}} = \varepsilon \sigma (T_{\text{blank}}^4 - T_\infty^4) + \varepsilon \sigma (T_{\text{blank}}^4 - T_{\text{die}}^4) + \varepsilon \sigma (T_{\text{blank}}^4 - T_{\text{punch}}^4) \]  

\[ q_{\text{cond}} = h_{\text{cond}} (T_{\text{blank}} - T_{\text{die}}) \]

Where \( h_{\text{conv}} \) (W·m⁻²K⁻¹) is the convective heat transfer coefficient, \( h_{\text{cond}} \) (W·m⁻²K⁻¹) is the conductive heat transfer coefficient, \( \sigma \) is the Stefan–Boltzmann constant, \( \varepsilon \) is the emissivity of the surface of blank, \( T_{\text{blank}}, T_\infty, T_{\text{die}} \) and \( T_{\text{punch}} \) (K) are the temperatures of the blank.

In the forming phase, a dynamic contact occurs at the blank/tools interface. The heat transfer in the interface can be characterized by a radiation and conduction. Thus, the boundary condition in this step is given as:

\[ q'' = q_{\text{rad}} + q_{\text{cond}} \]  

Where \( q'' \) (W·m⁻²) is the effective heat flux in forming phase, \( q_{\text{conv}} \) and \( q_{\text{rad}} \) can be calculated according to the equation (3) and (4).

The quenching phase occurs just after the forming step. It can be characterized by a static contact between the blank and the tools. Because the contact of interface between the blank and the tools is perfect in this phase, the heat transfer is due solely to conduction:

\[ q''' = h_{\text{cond}} (T_{\text{blank}} - T_{\text{die}}) + h_{\text{cond}} (T_{\text{blank}} - T_{\text{punch}}) \]
Where $q'''$ (W·m⁻²) is the effective heat flux in quenching phase.

3. Results and discussions

Fig. 2 shows the forming state and the temperature changes at different time during hot stamping. It can be seen that the whole forming time is about 2.9 s. The temperature starts to drop quickly as soon as the blank contacted the tools. Then, the heat from the formed blank dissipates quickly into the tools through the contact interface after the full contact is reached in Fig. 3b and Fig. 3c. In addition, through the observation of temperature distribution after hot forming from Fig. 3d, it can be clearly found that the temperature changes are inhomogeneous. There is a higher cooling rate in the sidewall of the blank compared with the other locations. In the ultimate, this affect the changes of micro-hardness and strength.

Figure 2. The forming state at different time of hot stamping simulation
4. Conclusion

Base on the appropriate material model and the boundary conditions of heat transfer and friction coefficient, the FE simulation of hot stamping process have been made to analysis the history and distribution of both deformation and temperature. It was found that the mechanical properties of hot stamped part showed an inhomogeneous phenomenon within the sample geometry. The sidewall of the part has a higher tensile strength compared with the other locations. The results compared with the experiments have good agreement. These indicate that numerical simulation is an effective tool for predicting the evolutionary process of hot stamping and optimizing the process parameters of forming.

References

[1] M. Ghazani, B. Eghbali, Characterization of the hot deformation microstructure of AISI 321 austenitic stainless steel, Mat. Sci. Eng. A 730 (2018) 380–390.
[2] A. Michael, N. Bas, L. Peter, Constitutive behaviour under hot stamping conditions, J. Mater. Process. Tech. 228 (2016) 34–42.
[3] H. Atef, J. Timo, K. Ali, Effect of silicon on the hot deformation behavior of microalloyed TWIP-type stainless steels, Mater. Design. 154 (2018) 117–129.
[4] L. Nan, Y.S. Chao, G. Ning, Experimental investigation of boron steel at hot stamping conditions, J. Mater. Process. Tech. 228 (2016) 2–10.
[5] K. Mori, P.F. Bariani, B.A. Behrens, Hot stamping of ultra-high strength steel parts, CIRP Ann-Manuf. Techn. 66 (2017) 755–777.