International Journal of Control and Automation
Vol.9, No.1 (2016), pp.93-102
http://dx.doi.org/10.14257/ijca.2016.9.1.09

Thermo-Mechanical Coupled Stamping Simulation about the Forming Process of High-Strength Steel Sheet

Li Xiaoda[1, 2], Zhang Xiangkui[1], Hu Ping[1*] and Zhan Xianghui[2]

[1]School of Automotive Engineering, Dalian University of Technology, Dalian 116024, China [2]Zhuhai College of Jilin University, Zhuhai 519041, China
Email: liliabc0185@sina.com, 112754994@qq.com, pinghu56@gmail.com, zhanxh_1215@163.com

Abstract

The developing trend of Automotive Lightweight has brought about the application of High-Strength Steel sheet in the automotive body, but High-Strength Steel sheet has poor formability with cold stamping process, and the existing cold stamping process cannot fully meet the needs of high strength steel forming. The hot stamping technology of High-Strength Steel sheet is the effective method to solve the above-described problem. It can greatly shorten the tryout time of die, and can effectively solve some problems like discarding of die, unreasonable process, and poor forming quality etc. In this paper, mainly for the Thermo-Mechanical Coupling process of sheet metal, introduce the thermal field analysis of High-Strength Steel sheet, the interaction of deformation and temperature field, and the main steps of Thermo-Mechanical Coupling. And for the U Shaped part, built the Finite element model and set process parameters with software 'Dynaform'. Simulated its forming process of Thermo-Mechanical Coupling with solver 'LS-DYNA'. The changing temperature field, thickness, and thickness reduction rate were calculated. The results show that, Thermo-Mechanical Coupled Stamping Simulation is an effective method to forecast high temperature formability of High-Strength Steel Sheet.

***Keywords**: Thermo-Mechanical Coupling, Hot stamping, Hot forming, Stamping simulation, High-Strength Steel Sheet*

1. Introduction

With the rapid development of Auto industry, its energy saving, emission reduction and security have been growing concerned, and automobile lightweight has been the power promoting the development of the automotive industry. High-Strength Steel sheet is widely used in automotive industry because of its light quality, high strength, and it can reduce the weight of auto body greatly and improve the crash safety. However, compared with the ordinary steel, High-Strength Steel sheet has higher Tensile Strength and Yield Strength, lower elongation, so it has poor formability at room temperature and it is usually prone to some defects like crack and spring back[1, 2]. Hot forming process is one of the key technologies to solve forming difficulty, improve performance and quality of sheet metal forming. The process of Hot Stamping technology can be seen from figure 1[3]. The High Strength steel sheet needs to be heated to the Austenite temperature range. And then, the Austenite steel sheet will be quickly transferred to the forming die with some transfer device like manipulator and quickly formed at high temperature. In the state of press packing, by using a cooling circuit arranged in the die to ensure a certain cooling rate, the work piece can be quenched and be gotten Martensite at room temperature in the end, and the finished part has high Tensile Strength, which can reach 1500MPa.

ISSN: 2005-4297 IJCA

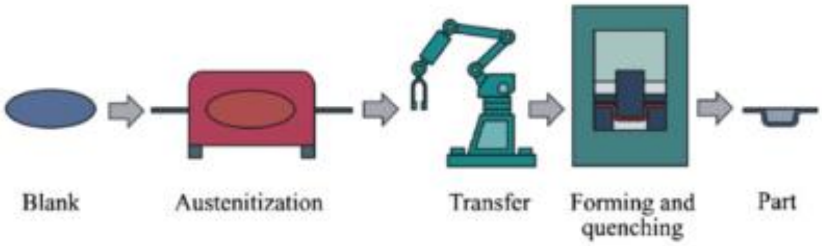


Figure 1. Basic Hot Stamping Process Chain

Due to the influence of material property, stamping blank size, heating temperature, the punching speed and lubrication condition and other factors, hot stamping of High-strength steel become extremely complex and difficult to control. Computer Simulation technology can not only save a lot of equipment costs, can also precisely control the process parameters in forming process, through some simple operation, it can change the pressing conditions, greatly improve the efficiency of research. Hot Stamping simulation of High-Strength steel is to simulate flow deformation laws of sheet metal during the stamping process, temperature distribution, stress distribution, and to analyze the impact of thermoforming process parameters on the forming process by establishing the finite element model. The simulation can be the references to help engineers optimize the process parameters and design the die next, and shorten the product development cycle and reduce costs. It is a focus of current research to study the Hot Stamping process with numerical simulation, a large number of scholars at home and abroad participate in it, Bergman and Oldenburg has simulated hot stamping process earlier[4], Karbasian introduced the development of hot forming technology, MyungKi Park from Korea simulated the hot forming process of a B-pillar and had an analysis of the flow characteristics of sheet metal[5], Ma Ning and Hu Ping also had a study on hot stamping process, hot stamping simulation and experiment, and had developed the module on the hot forming numerical simulation of High-Strength steel sheet based on the self-development software KMAS(King-Mesh analysis system)[6-8]. In this paper, it introduces the basic theory about Thermo-Mechanical Coupling of High-Strength steel sheet, including the thermal field analysis of hot stamping, the interaction of deformation and temperature field, and the main steps of Thermo-Mechanical Coupling. With Thermo-Mechanical Coupling method, a U-shaped part is for study in this article, and just study the forming process of sheet metal. The results show the temperature and the thickness distribution, which produces the basis for next hot stamping die design and manufacture.

2. Thermo-Mechanical Coupling Analysis of High Strength Steel Sheet

During the hot stamping process, there is close relationship between thermal field and deform, internal stress field. The mutual influence of two physical field ‘heat and force’ is called thermal coupling. So the essence on Thermo-Mechanical Coupling of Rigid-Viscoplastic Finite Element Analysis method is：In the same set of finite element mesh system，it needs to solve the heat conduction equation considering the influence of velocity field and plastic flow equation considering the temperature effect respectively, until getting the corresponding converged solution. A complete description of thermoplastic forming process should include mechanical deformation during forming process, mechanical energy, conversion process of internal energy converted to heat energy and heat conduction process. Having an analysis of Thermo-Mechanical Coupling needs consider both the plastic deformation of part and the heat exchange among part, die and environment. So Thermo-Mechanical Coupling analysis includes 2 main problem: Deformation analysis and Temperature field analysis.

2.1 **Temperature Field Analysis**

Thermal analysis is used to calculate the temperature distribution and other thermal physical parameters about a system or component, like acquisition or loss of heat, thermal gradient, heat flux, etc. According the principle of conservation of energy, the heat balance of solid's heat transfer process can be gotten.

(1)

In the formula, ,, are the heat flux in the direction x, y, z, is material density (kg/m3), c is heat capacity of material (), T is temperature, and t is represented as time, q is body heat.

In the heat conduction problem of solid, it usually assumed that heat flux is proportional to the temperature gradient, that is:

(2)

In the formula, is the thermal conductivity in different directions, the heat flowing from high to low temperature, the right end of the above formula is presented as negative sign. Considering the analyzed material is isotropy, therefore, So:

(3)

The above formula is the heat balance of solid's heat transfer process.

2.2 **Interaction between Deformation and Temperature Field**

The stress and strain of the thermoplastic forming have the influence on the temperature field in the form of internal heat sources and warm by rubbing. During studying the process of hot deformation, usually deformable body and die can be treated as a whole for analysis, and it assumed that the die is rigid and there is no any deformation, so there is no inner heat source in the die, the plastic deformation work in deformation body can be converted to the inner heat source, and friction energy can be the surface heat flux on the contact surface between the work piece and tooling. Therefore, the body heat and heat flux can be expressed as follows:

(4)

In the formula: is Equivalent Stress, is Equivalent Strain rate, and is Heat generation, is friction stress and relative sliding velocity between the work piece and die.

The influence of temperature field on the plastic deformation can be considered in the flow stress of metal, namely that, the flow stress is the function of equivalent strain, strain rate and temperature.

(5)

2.3 The Main Steps of the Thermal Coupling

The Flow chart of Thermo-Mechanical Coupling is shown in Figure 2[9].

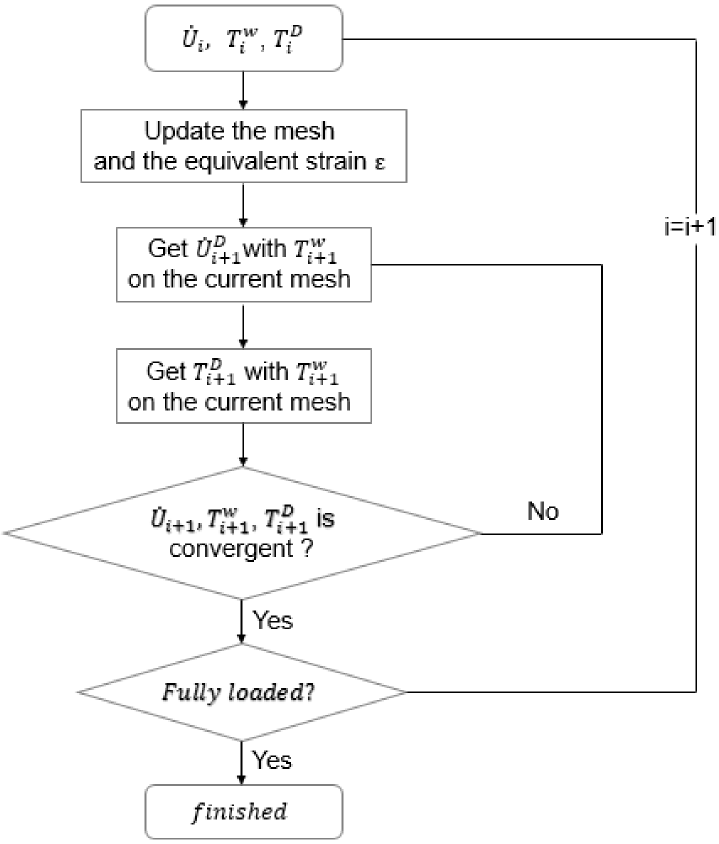


Figure 2. The Flow Chart of Thermo-Mechanical Coupling

Step1: According Node Velocity and Time Step ， update finite element mesh of work piece, and in order to give grid configuration at time. And then, calculate the equivalent strain of the new configure.

Step2: Make the velocity field and temperature field of convergence at time the initial solution of time, that is:

(6)

Here, the left index of variable represents the number of iteration loop between deformation and thermal analysis.

Step 3. Iteration loop, the left index j=0, 1, 2, until convergence.

a) According, analyze the work piece's deformation, the results converge to.
b) According and, have the work piece's thermal analysis, the results converge to.
c) According, conduct the die's thermal analysis, the results converge to.
d) Check the convergence, if there exist difference between, and,,, it is treated as not converge, the number of iterative loop j will be plus 1, and then go back the beginning of step 3. If getting the satisfied convergence result, the number of time will be plus 1, then go back step 1.

3. Building the FEM Model

Build the geometric model of the U shaped part with 3D software, the material used is Boron steel 22MnB5, and the designed thickness of part is 1.6mm, as is shown in figure3. Import the model in format IGES to 'Dynaform' which is the nonlinear finite element analysis software on sheet forming, and mesh it with quadrilateral B-T (Belytschko-Tsay) element. And the FEM model of part's blank and tools built is shown in figure4, Punch, die and binder are all rigid body and the blank is the deformation body.

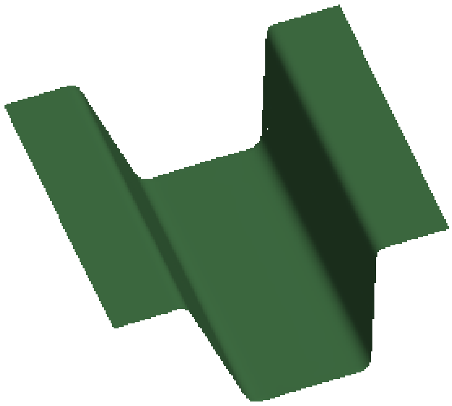


Figure 3. The Geometric Model of U Shaped Part

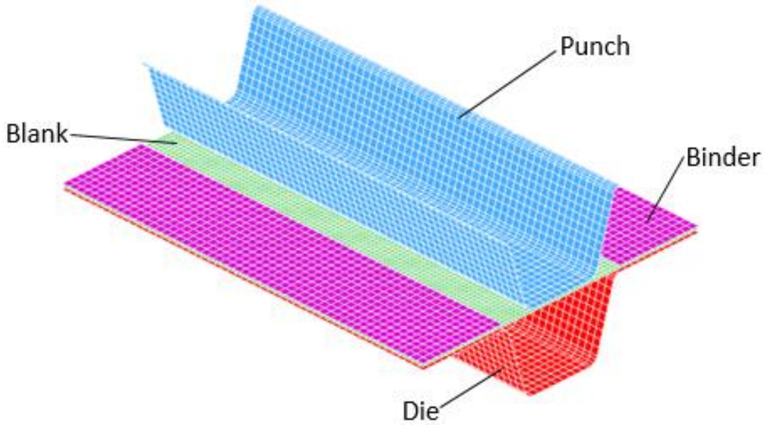


Figure 4. Assembly Structure of the U Shaped Part's Sheet and Tools

4. Thermal Parameters of Material 22mnb5

Choose thermal elastic-viscoplastic material: MAT-106. With Thermal coupling analysis, and taking into account the radiation heat transfer and convection heat transfer between tools, sheet metal and environment. The temperature of Sheet metal and tools is assumed to be isotropic and the initial temperature of the sheet is set to 800 ℃, Ambient temperature is 20 ℃. True Stress-Strain curve of 22MnB5 at different temperatures is shown in Figure 5 and Table 1 shows the Thermal parameters [10].

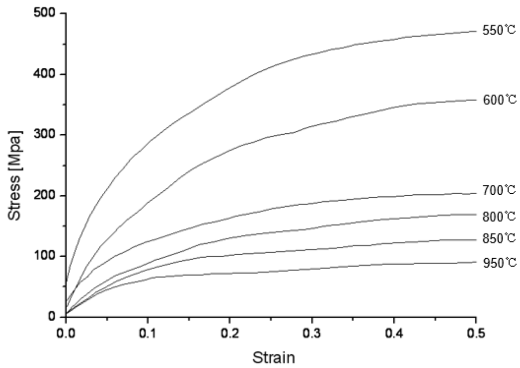


Figure 5. The Stress-Strain Curve of 22mnb5 at Different Temperature

Table 1 Thermal-Mechanical Material Properties for 22mnb5 Steel [10]

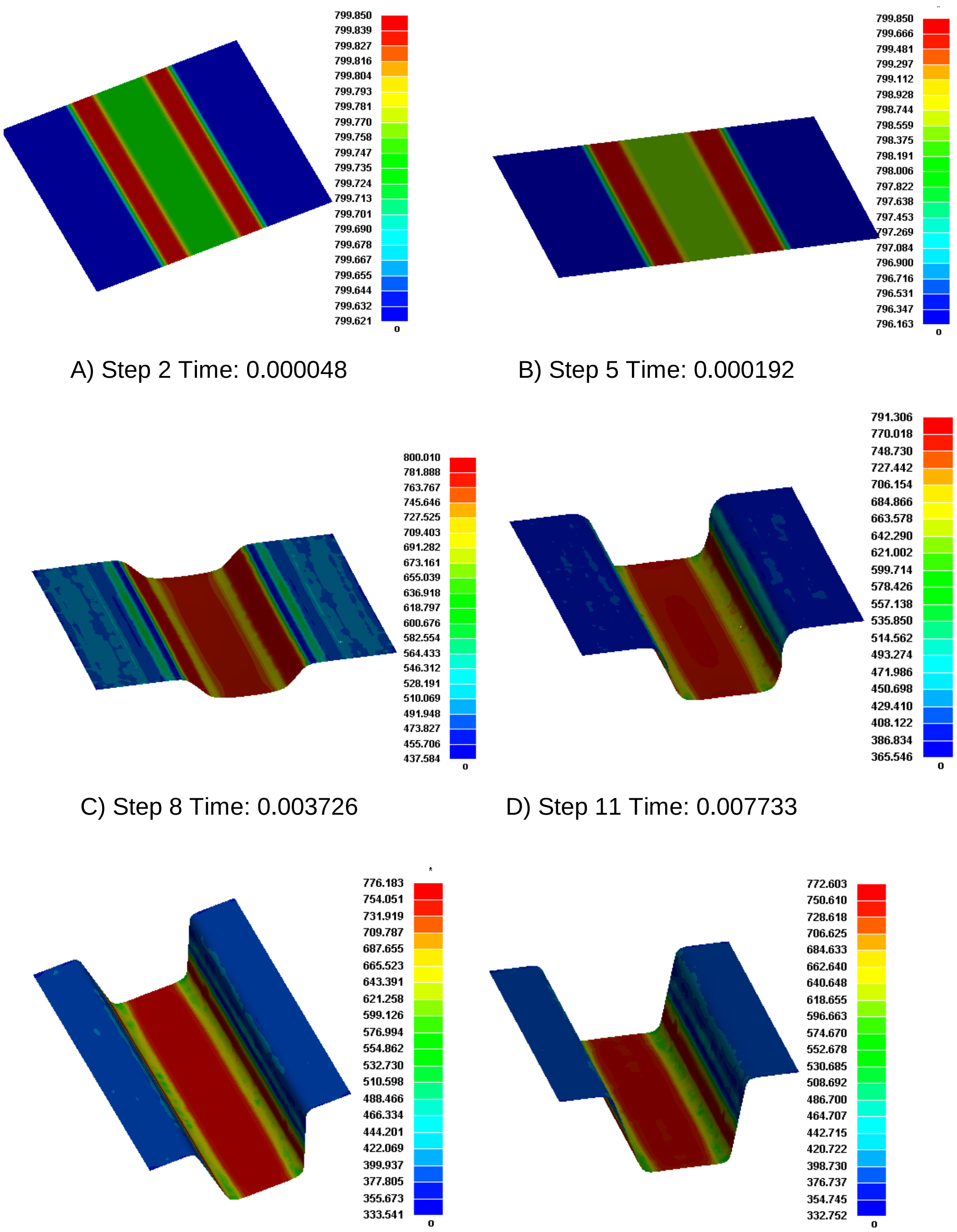
Parameters	Value										
T/℃	20	100	200	300	400	500	600	700	800	900	1000
E /MPa	212	207	199	193	166	158	150	142	134	126	118
Poisson Ratio	0.284	0.286	0.289	0.293	0.298	0.303	0.310	0.317	0.325	0.334	0.343
P	4.28	4.21	4.10	3.97	3.83	3.69	3.53	3.37	3.21	3.04	2.87
C	$6.2X10^9$	$8.4X10^5$	$1.5X10^4$	$1.4X10^3$	258	78.4	35.4	23.3	22.2	30.3	55.2
k/	30.7	31.1	30.0	27.5	21.7		23.6		25.6		27.6
C_p/	444	487	520	544	561	573	581	586	590	596	603

5. Boundary Conditions on Finite Element Analysis of Hot Stamping

According the actual status of stamping process, with Single Action Drawing, and Friction Coefficient is set to 0.18 adding graphite lubrication. Thermal Boundary Conditions: Radiation Coefficient is 0.8SBC, Heat Transfer Coefficient is 5W/ m2K. When parts and tools are in contact, the Thermal Boundary Conditions are closed, and at this time, Heat Transfer Coefficient is also set to 5W/ m2K, Thermal Conductivity is 40W/MK and Radiation Coefficient is 7.6SBC. There are 2 stages in hot stamping process: Blank holding and Drawing. The process parameters of 2 stages: In blank holding stage, the velocity of blank holder is set to 2000mm/s. And In drawing stage, the velocity of blank holder is set to 5000mm/s, the holding force is 5 KN.

6. The Simulation Result

During the forming process of the sheet, the temperature distribution of four time are shown in figure6. Figure 7 and Figure 8 show the thickness distribution and the reduction rate of Thickness. As can be seen from the figures, the position of Maximum thickness change after forming is the tangent between rounded corners at the bottom and the sidewall, and it thinning rate of thickness is up to 21%. The thinning rate of thickness at top rounded corners is 1.7% to 7%, and the maximum thickening of part is not more than 0.49%, there is no wrinkling. From the calculation results, under the process conditions, the size distribution of the formed parts is better, and has better formability. Figure 9 is the actual hot stamping part.



A) Step 2 Time: 0.000048 B) Step 5 Time: 0.000192

C) Step 8 Time: 0.003726 D) Step 11 Time: 0.007733

E) Step 14 Time: 0.010137 F) Step 17 Time: 0.011240

Figure 6. The Temperature Distribution of the Sheet at Different Time

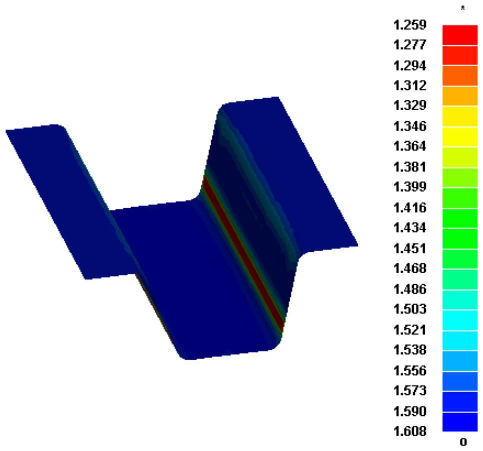


Figure 7. The Thickness Distribution of the Part after Hot Forming

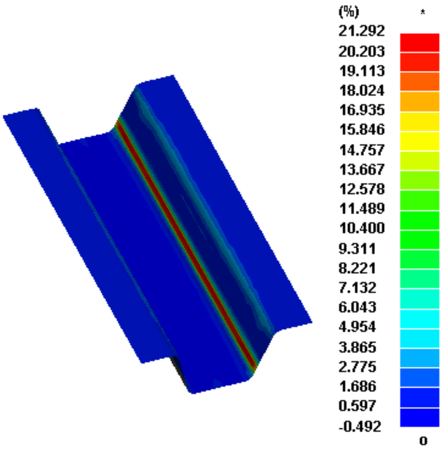


Figure 8. The Thinning Rate of the Part after Hot Forming

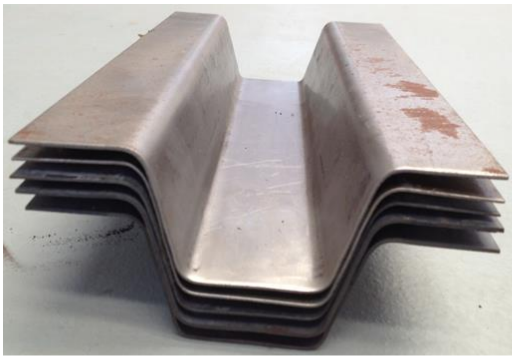


Figure 9. The Actual Hot Forming Part

7. Conclusion

Hot stamping process of High-Strength steel is a new deformation process which is different from traditional cold forming of sheet metal. The research focus of this paper is the stamping process analysis of part's formability. During the forming process, sheet metal deformation is always accompanied by the temperature changes of sheet and tools, it is a problem of thermal coupling. Sheet at high temperature has a small deformation resistance, and meanwhile, it generates heat transfer between the sheet and tools, which can lower temperature itself. And it will produce heat when steel deformation happening, furthermore, change the original distribution of temperature field. As the temperature changing and the work piece deformation, the nature of the mechanical and thermal will change greatly. Therefore, it must consider the interactions of deformation and temperature fields to do the hot forming FEM simulation. The paper also takes the U shaped part for example to have a Thermo-Mechanical Coupling simulation for its forming process.

Acknowledgement

The authors acknowledge financial supports from National Natural Science Foundation of China (No. 11472071) and the 'Three Levels' backbone teacher training program of Zhuhai College of Jilin University.

References

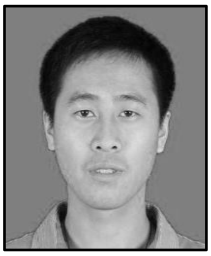
[1] H.U. Ping and M.A. Ning, "Development of Hot Forming Technology For Ultra High Strength Steel And Its Mechanical Problem", Advances on Mechanics, vol. 41, no. 3, **(2011)**, pp. 310-334.
[2] H.Z. Li, X. Wu and G.Y. Li, "Prediction of Forming Limit Diagrams for 22MnB5 in Hot Stamping Process", Journal of Materials Engineering and Performance, **(2013)**
[3] H. Karbasian, and A.E. Tekkaya, "A review on hot stamping", Journal of Materials Processing Technology, **(2010)**.
[4] G. Bergman and M. Oldenburg, "A finite element model for thermomechanical analysis of sheet metal forming", International Journal for Numerical Methods in Engineering, vol. 59, no.9, **(2004)**, pp. 1167-86.
[5] M.K. Park, H.S. Son and T.H. Kim, "Formability, flow and heat transfer simulation of hot press forming b-pillar part and tools", AIP Conference Proceedings, NUMIFORM2010, **(2010)**
[6] M.A. Ning, "Study of hot forming for high strength steel:numerical simulation-static explicit algorithm", Chinese Journal of Computational Mechanics, vol. 28, no. 3, **(2011),** pp. 371-376.
[7] N. Ma, "Constitutive Theory and Experiment Analysis Of Hot Forming For High Strength Steel", Chinese Journal of Theoretical and Applied Mechanics, vol. 43, no. 2 **(2011)**, pp. 346-354.
[8] N, Ma, "Coupled Models of Hot Forming of Ultra Strength Steel Sheets. Steel Research International", vol. 81, no, 9, **(2010)**, p. 900-903.
[9] Q. Zhou, "Study on the hot stamping process of high strength steel sheet", Shanghai: Tongji University, **(2007)**.
[10] A.B. Shapiro. "Using LS-DYNA for hot stamping. 7th European LS-DYNA Conference", Salzburg, **(2009)**.

Author

She took her Ph.D. at the School of Automotive Engineering, Dalian University of Technology

She is Associate Professor, Department of Mechanical and Automotive Engineering, Zhuhai College of Jilin University

The main research directions: Lightweight design of automobile body, Numerical simulation on stamping process of sheet metal



Zhang Xiangkui (1973-), School of Automotive Engineering, Dalian University of Technology, Dalian 116024, China [2]Zhuhai College of Jilin University, Zhuhai 519041, China112754994@qq.com