Journal of Computers Vol. 35 No. 5, October 2024, pp. 237-252
doi: 10.53106/199115992024103505016

Intelligent Driving Decision-Making Strategy for New Energy Vehicles Based on Lightweight Reinforcement Learning Model

Wen-Tao Li[1,2], Zhi Zhang[1*], Xiang-Yu Yi[1], Xiao-Bo Dong[1],
Liang-Gui Zhang[1], and Li-Yun Yang[1]

[1] Department of Automobile Engineering, Hebei Institute of Mechanical and Electrical Technology,
Xingtai City 054000, Hebei Province, China

{wentao3982, zhang_zhi5278, yuyu0989, xiaobo9832,
lianggui3897, liyun987}@163.com

[2] Xingtai New Energy Vehicle Lightweight Technology Research and Development Center,
Xingtai City 054000, Hebei Province, China

Received 26 September 2024; Revised 2 October 2024; Accepted 17 October 2024

Abstract. The explosive growth of new energy vehicles requires optimization of the automotive processing process. This article focuses on the processing of the rear panel in automotive panels. Firstly, the mesh optimization of the stamping die surface of the rear panel is carried out, and the adaptive mesh method of the die surface is used to improve the calculation accuracy of finite element calculations in these local areas. Then, in order to predict the forming performance of the stamping model for automotive panels, Dynaform software was used to numerically simulate the stamping process, analyze the forming effect of the parts under certain process conditions, and provide direction for the optimization of the model. Through experimental settings with different processing parameters, different forming effects of the rear panel were obtained. Then, through orthogonal experiments, the optimal processing scheme was determined. Finally, based on the optimal processing scheme, simulation analysis was conducted to optimize the thickness of the thin plate in the automotive forming process, and further optimize the stamping model structure.

Keywords: new energy vehicles, mold optimization, finite element analysis, stamping compensation

1 Introduction

The covering parts of new energy vehicles are the core components of the car body, which have functionality and aesthetics. The shape of the car covering parts is complex and the structure is diverse, occupying the vast majority of the exterior surface of the car. In the car covering parts, most parts are obtained through thin plate stamping processing. The covering parts generally include the car roof, engine hood, doors, fenders, back doors, etc. Car coverings are attached to the surface of the car, so the surface quality of the parts is required to be high during the manufacturing process. At the same time, the structural dimensions are large, so the processing difficulty is relatively high. With the popularization of new energy vehicles and the rise of new forces in car manufacturing, competition in the automotive industry is becoming increasingly fierce. Improving the forming quality of car panels, shortening production cycles, and reducing production costs are the primary goals of the new energy vehicle manufacturing process [1].

The structure of automotive coverings is complex and the structural dimensions are large, so the forming of coverings is mainly based on thin plate stamping. In the actual stamping production practice of automotive cover parts, the factors that affect the forming quality of cover parts are the qualitative theoretical analysis of the stamping process and the experience of technical personnel. Qualified stamped parts must undergo multiple trial productions to obtain. The stamping process requires stamping molds. In the mold design process, it is necessary to consider the formation of defects such as wrinkling, cracking, rebound, and poor surface quality of automotive coverings, which require a lot of time, material resources, and design experience. Therefore, the forming quality of automotive coverings depends more on the structural design of automotive stamping molds and the quality of mold production [2].

With the development of computer technology, numerical analysis can predict wrinkling, cracking, and slight

* Corresponding Author

rebound during the stamping process of automotive panels. Numerical simulation analysis technology for stamping forming provides a scientific method for mold design, process design, and process parameter optimization, and has become the most powerful tool for stamping mold design and stamping part forming in the automotive manufacturing industry. However, the comprehensive impact of various process parameters on the forming quality of automotive panels is often complex and difficult to analyze, resulting in the designed molds often not meeting the requirements of the product. However, in the process of numerical analysis, the combination of finite element simulation and optimization techniques can be used to understand the forming conditions of automotive panels in advance, and predict various defects that may occur during the stamping process, such as thinning, wrinkling, cracking, etc., through parameter simulation before mold manufacturing, to determine the required process and technical parameters for forming [3]. The combination of the above methods can effectively shorten the mold design cycle, shorten the mold debugging time, reduce development costs, and greatly improve the forming quality of automotive panels. Based on the above methods, this article focuses on the quality of automotive panel molding, with the goal of designing and manufacturing panel molds, and studying virtual design and optimization methods for molds. The main work done in this article is as follows:

1) Firstly, the mesh optimization of the mold surface is carried out, and the adaptive mesh method of the mold surface is adopted to improve the calculation accuracy of finite element calculation in these local areas.

2) In order to predict the forming performance of automotive panel stamping models, taking the automotive rear panel as an example, Dynaform software was used to numerically simulate the stamping process, analyze the forming effect of the parts under certain process conditions, and provide direction for the optimization of the model.

3) Through simulation analysis, the optimization of thin plate thickness in the automotive forming process has been completed, and the optimization of the stamping model structure has also been achieved.

2 Related Work

Many scholars have conducted relevant research on the mold design and optimization of new energy vehicle cover parts. In terms of automotive sheet metal part forming, Huiwen Mao et al. introduced a numerical simulation and mold design of the injection molding process of automotive rear door cover parts, and used ANSYS and HyperWorks computer-aided software to analyze the characteristics of the selected materials and improve the lightweight characteristics of the rear door panel [4].

Changyong Deng and his team from Chongqing conducted research on the defects such as tensile cracking and wrinkling that occurred during the stamping process of a lightweight floor component for a new energy vehicle. They used a combination of finite element simulation and experimental design to optimize the process plan. Taking edge pressure, friction coefficient, and die pressing speed as influencing factors, two functions, maximum thinning rate and maximum thickening rate, were proposed as the objective functions for forming quality evaluation, and multi-objective optimization was carried out. Finally, the reliability of the optimization scheme was verified through actual mold testing [5].

Xin Jiao, based on the characteristics and usage requirements of automotive panels, analyzed the forming performance indicators of steel plates. The fuzzy analytic hierarchy process was used to obtain the weights of each evaluation indicator value, and the fuzzy comprehensive evaluation was used to obtain the forming performance decision parameters of the material. Then, a stamping simulation model was established using DYNAFORM to simulate and analyze the processing of the panels and obtain reasonable experimental process parameters [6].

Xianhui Luo from SAIC General Motors has difficulty in stamping and controlling the size of the battery box cover for automotive power due to its large size and high sealing requirements. Fully optimize the stamping process of the battery box cover through Autoform finite element virtual simulation and soft mold part verification. Propose the idea of "inverted stamping" to solve the problem of material dropping and deformation of the part, and optimize the mold design based on this. The formability, size control, and equipment passability of physical components are excellent, which has certain reference significance for the development of future projects [7].

In terms of modeling analysis and optimization methods for stamping molds, Haijun Gong conducted reverse modeling and die-casting mold design for the aluminum alloy signal amplifier housing of automobiles. The ATOS blue laser scanning system was used for measurement, and then Geomagic Design X software was applied to process and reverse model the point cloud data obtained from the measurement. After analyzing the die-casting process of the reverse prototype, the die-casting mold design theory and experience were combined to design the die-casting mold for the product [8].

Bingxin Liu introduced a finite element analysis method for molds, which uses a numerical model based on finite element analysis to simulate the temperature distribution inside a hot press can and verify the known experimental data. Then, the typical mold structure covered with composite materials and simplified auxiliary materials will be introduced into the numerical model. In addition, a method combining numerical modeling with genetic algorithm is proposed to optimize the design parameters of the mold substructure with the maximum standard deviation of the curing degree of composite material parts as the goal and the deformation of the mold under improved conditions as the constraint, in order to improve the synchronization of curing. The maximum standard deviation of the curing degree of composite material parts increased by 17.21% [9].

Jingjing Li used finite element software to numerically simulate the extrusion process of TC4 titanium alloy T-shaped thin-walled profiles. The single factor method and orthogonal experimental method were used to simulate and combine different structural parameters. The SDV value of the outlet standard velocity field deviation was used as a reference to study the effects of inlet fillet, die hole position, and guide groove shape on the metal forming effect after extrusion. The flow law of the metal during the extrusion process was also analyzed [10].

Shubo Xu, taking the car trunk floor cover as the research object, used the finite element analysis software Autoform to study the influence of changes in blank size, stretching ribs, and edge pressure on the forming performance of the cover. Finally, the optimal design parameters were obtained, numerical simulation was carried out, and the stretching mold of the cover was optimized. It was found that the mold nodes were subjected to large forces and prone to damage. Based on the optimization results, a new mold structure was designed [11].

The composition structure of this article is as follows: Chapter 2 mainly discusses the relevant research results of scholars, Chapter 3 completes the modeling and analysis of the mold surface through the adaptive method of the mold surface, Chapter 4 completes the optimization analysis of the stamping process, Chapter 5 is the experimental results and the analysis process of the results, and Chapter 6 is the conclusion section, summarizing the work of this article and listing further research directions.

3 Accurate Surface Analysis of Sheet Metal Mesh Model

The metal sheet material used in stamping forming is generally a flat sheet with a simple shape. Therefore, before conducting numerical simulation of stamping, it is necessary to perform finite element mesh discretization analysis on the sheet metal. During finite element discretization analysis, only uniform mesh element sizes can be set. When the initial mesh size of the billet is set too large, it cannot be fully formed at the severe deformation of the mold surface, resulting in calculation errors. The grid adaptive method can enable the structure of the grid to dynamically change with the numerical simulation process of stamping. In areas where physical quantities change dramatically, the grid structure can self adapt into a more refined grid structure to reflect the characteristics of that area, improving the computational accuracy of finite element calculations in these local areas. Therefore, this section completes the establishment of the mesh model for the stamping model of automotive panels.

3.1 Regional Division of Deep Drawing Model

The drawing die is the most demanding of all automotive cover production dies, and its main structure includes two parts: the cavity surface and the pressing surface, because their roles in the drawing process are different and their structures are also different. The cavity surface only needs to meet the gap requirements at the final moment of forming, while the pressing surface area is already compressed at the initial moment of forming. During forming, the sheet metal continuously flows through this surface into the mold cavity, so the pressing surface area needs to be considered throughout the entire stamping stroke [12]. The schematic diagram of the model structure is shown in Fig. 1.

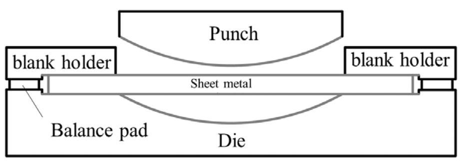

Fig. 1. Model structure diagram

This article uses the grid bias method to construct an accurate profile grid. The main principle of the grid bias method is to consider the nodes on the grid to be biased and the nodes that are offset by one sheet thickness in the direction of the sheet metal as node normal vectors, and the intersection points of the node connections and the sheet metal grid as mapping points, to achieve the mapping of the tool grid to the sheet metal grid. Then, through finite element interpolation calculation, the thickness change value of the mapping point is calculated, and finally, the thickness change value is used to reverse bias the tool grid nodes to obtain the accurate profile grid nodes.

According to the division of the mold surface matching area, using the same node connection method for concave and convex molds can only construct accurate surface grids for static and non fully dynamic matching areas. Because there is no sheet metal remaining in the fully dynamic mating area after forming, it is not feasible to construct an accurate surface mesh by offsetting the mold mesh in this area with the thickness distribution information of the formed sheet metal. Therefore, it is possible to dynamically match the sheet metal during the stamping process.

The continuous flow behavior in the region is used to calculate the optimal gap values for each position in the region, and then the gap values at each position are used as the virtual sheet thickness values for that position to construct a virtual sheet metal mesh for the fully dynamic matching region. The virtual sheet metal mesh of this region is then combined into the final sheet metal mesh after forming, to construct an overall virtual sheet metal mesh that can reflect both the forming results and the forming process of the sheet metal. In this way, all the surface areas that come into contact with the sheet metal during the stamping process (i.e., the surface matching area) must be able to find mapping elements on the sheet metal mesh for the nodes on the mold mesh.

Build the model into a grid structure as shown in Fig. 2. The line connecting nodes N_1 and N_2 on the concave and convex mold grids is taken as the normal direction of node N_1. Therefore, the intersection point O between the line and the sheet metal grid is the mapping from the tool node to the sheet metal grid. After calculating the areas of AOB, AOC, and BOC, the thickness values of nodes A, B, and C, the area comparison method is used to calculate the thickness change value t at point O through shape function interpolation. N_1 is reverse biased by t to obtain the biased node NP_1. After offsetting all nodes, the control vertices of the surface patch are correspondingly offset based on the node offset to achieve the transformation of the adjusted mesh model to the surface, thereby obtaining the mold surface with unequal gaps.

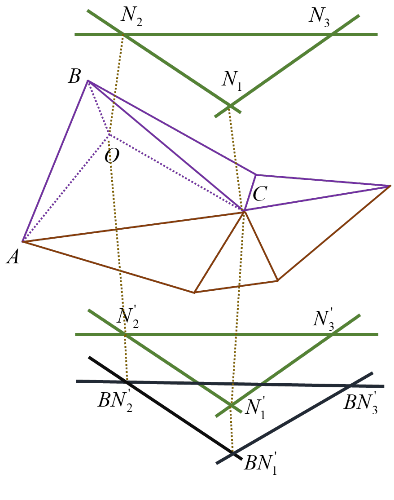

Fig. 2. Schematic diagram of model grid bias

3.2 Finite Element Model of Grid Model

When establishing a grid model, the use of grid adaptive methods can predict the predicted amount of changes in the sheet metal during the processing. The grid adaptive method allows the structure of the grid to dynamically change during the numerical simulation process of stamping. The quasi-static deformation process of sheet metal forming is treated as a dynamic process, therefore the solution process is expressed as:

$$Ma(t)+Dv(t)+Ks(t)=L(t) \tag{1}$$

In the formula, M is the mass matrix, D is the damping matrix, K is the stiffness matrix, L is the node load vector, and s, v, and a correspond to the displacement vector, velocity vector, and acceleration vector of the node at system time t. The dynamic explicit algorithm uses the central difference scheme as the time integration scheme to discretize time. If there is a time increment Δt at time t, and the displacement can be used to express velocity $v(t)$ and acceleration $a(t)$ using the center difference method [13], then the expression method for velocity and acceleration is:

$$v(t)=\frac{(s_{t+\Delta t}-s_{t-\Delta t})}{2\Delta t} \tag{2}$$

$$a(t)=\frac{(s_{t+\Delta t}-2s_t+s_{t-\Delta t})}{\Delta t^2} \tag{3}$$

Therefore, the finite element solution method for obtaining the mesh of the mold surface is:

$$\left(\frac{M}{\Delta t^2}+\frac{D}{2\Delta t}\right)s_{t+\Delta t}=L(t)-\left(K-\frac{2M}{\Delta t^2}\right)s_t-\left(\frac{M}{\Delta t^2}+\frac{D}{2\Delta t}\right)s_{t-\Delta t} \tag{4}$$

In order to ensure the smooth flow of stamping thin plates, the drawing die for automobile covering parts generally does not repair the concave die fillet. However, after dividing the concave die surface into grids, the grid becomes a whole. Without changing the topology of the entire concave die grid, the concave die grid is divided into static area grid, dynamic area grid, and fillet area grid. Based on the virtual sheet metal grid, the mesh model of the research and assembly area is biased. After the bias is completed, the mesh divided in the previous step is stitched to construct the accurate surface grid of the entire mold [14]. The specific optimization process is shown in Fig. 3.

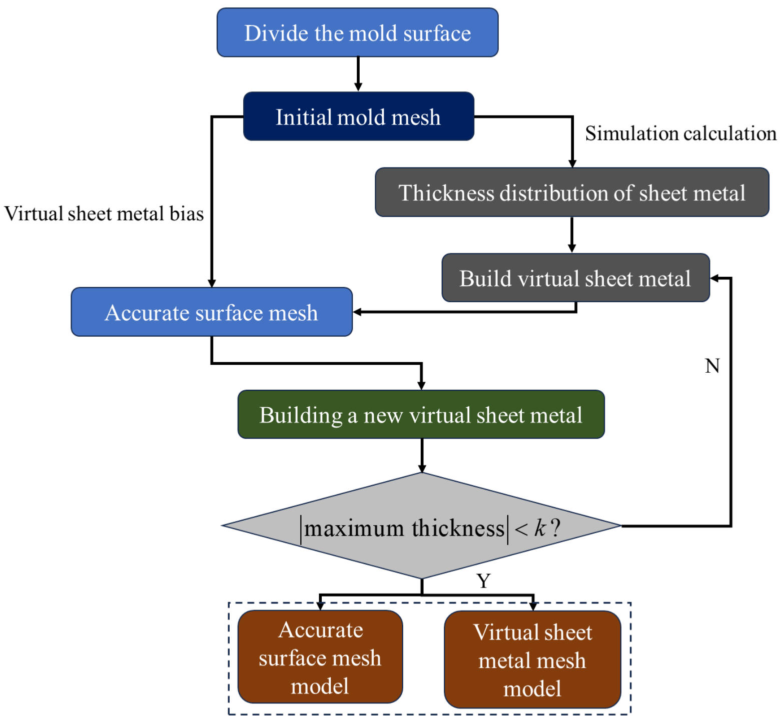

Fig. 3. Grid optimization process

When iteratively constructing an accurate surface mesh model, in order to minimize the number of iterative calculations while ensuring mesh accuracy, it is necessary to set a threshold for iterative calculations, which is based on using the precise surface mesh constructed in the previous and subsequent iterations for stamping simulation.

After the above process, the construction of the mesh model of the mold surface has been completed, laying a theoretical foundation for further finite element analysis.

4 Simulation Analysis of Deep Drawing Process for Automotive Panels

In order to predict the formability of automotive panels, this article takes the automotive rear panel as an example. Firstly, Dynaform software is used to numerically simulate the stamping process, analyze the forming effect of the part under certain process conditions, and study its forming limit, thickness variation, and stress-strain situation to reduce the risk of defects such as wrinkling and tearing during the forming process of the rear panel [15]. This article uses Dynaform software as a tool to simulate and analyze the stamping process of automotive rear panels. The analysis process is shown in Fig. 4, which mainly includes steps such as 3D modeling, finite element modeling, solution calculation, result analysis and optimization.

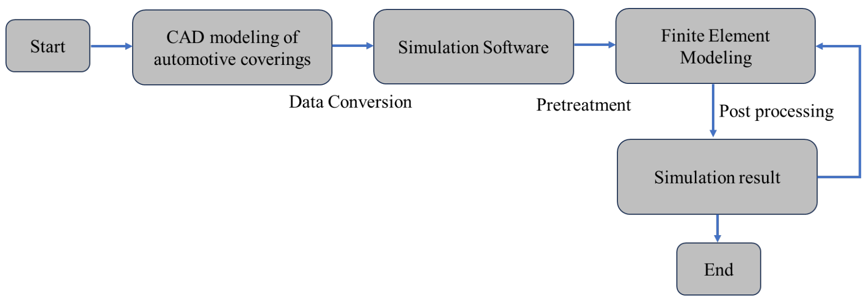

Fig. 4. Simulation analysis flowchart

The specific steps of Dynaform software in sheet metal forming process are as follows:

1) Use SolidWorks to create surface models of convex molds, concave molds, edge rings, and sheet metal according to requirements, then save them in a file format that Dynaform can recognize, and import the model data into Dynaform software.

2) Use the mesh generation tool of Dynaform software to perform mesh generation on convex molds, concave molds, edge rings, and sheet metal, check for mesh defects (including element normal vectors, mesh boundaries, overlapping nodes, negative angles, elements, etc.), and make modifications.

3) Define the properties of convex molds, concave molds, edging rings, and sheet metal, and set corresponding process parameters including contact type, edging force, friction coefficient, stamping speed, mold clearance, etc.

4) Adjust the relative positions of the convex mold, concave mold, edge pressing ring, and sheet metal, check the direction and trajectory of the movement of the convex mold and concave mold, and ensure that the mold action is correct and error free.

5) Input the established process parameters and run the LS-DYNA solver for solving.

6) After entering the post-processing unit, open the solution result file, which can be displayed in the form of cloud maps, contour lines, animations, etc. You can observe the deformation process of the sheet metal to grasp the variation law of the process parameters in the sheet metal forming process.

7) Analyze the simulation results to determine if they meet the requirements and identify any issues. If the results are not satisfactory, redefine the shape and motion curve of the tool, modify the relevant process parameters of the blank, punch, die, and edge ring, and re simulate and analyze until satisfactory results are obtained.

Create a 3D model of the car's rear panel. Due to the general modeling function of Dynaform software, this article uses the 3D CAD software SolidWorks to create a geometric model of the rear panel of a certain car model, as shown in Fig. 5, with dimensions of approximately 1480mm × 640mm × 280mm. The material of the rear panel is DC05 steel, with a thickness requirement of 0.8mm. The mechanical properties of the panel material are shown in Table 1.

Table 1. Performance parameters of DC05 steel

Parameter	Tensile strength	Hardening index (n)	Poisson's ratio (v)	Elastic modulus	Yield strength
Numerical value	350 R_m/MPa	0.226	0.3	210 E/GPa	120-180 R_{el}/MPa

During the forming process of the parts, process auxiliary surfaces need to be added. Similarly, SolidWorks software is used for design to obtain a 3D model of the car rear panel stamping forming, as shown in Fig. 5. Then, the parts are saved in .igs format. The forming process of parts is stamping → trimming → flanging, and the quality of stamping directly determines the final product quality. Therefore, this article will focus on the research of stamping forming process.

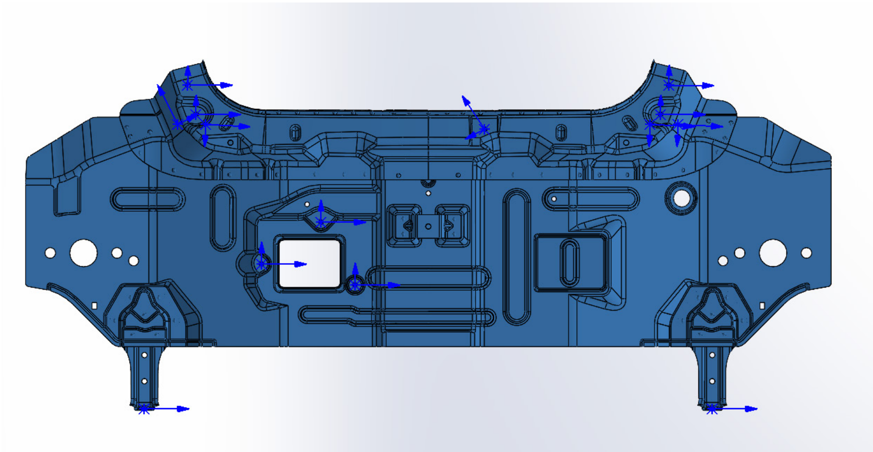

Fig. 5. Car rear panel model

4.1 Finite Element Modeling

In the pre-processing module, the concave mold was meshed with a scale of 39787 elements, and the mesh was checked and repaired using software mesh inspection and repair tools [16]. Finally, the finite element model of the car rear panel was established as shown in Fig. 6.

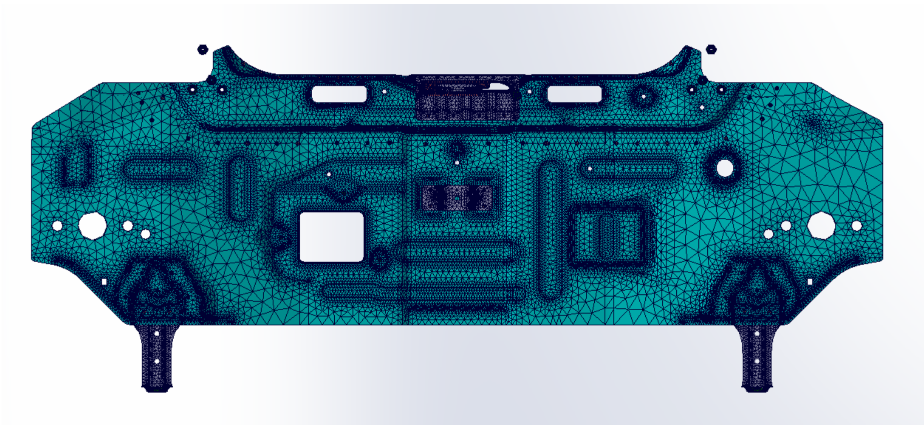

Fig. 6. Finite element mesh modeling

Using the BSE module of the software for stamping negative angle inspection, as shown in Fig. 7, the stamping negative angle inspection results of the parts were obtained based on the selected stamping direction, and no stamping negative angle was found. The rear panel of the car is in the safe forming area.

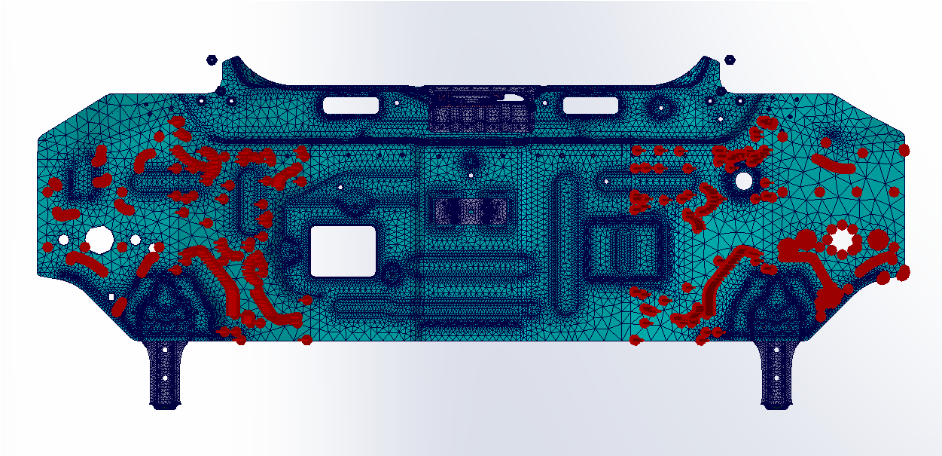

Fig. 7. Negative pressure inspection of rear panel

After grid division, the necessary data for the offset analysis of the concave die, including the edge ring and convex die, were obtained. Then, the BSE module was used to reverse calculate the shape of the billet, followed by grid division and gravity loading analysis. The required blank model for the analysis is shown in Fig. 8.

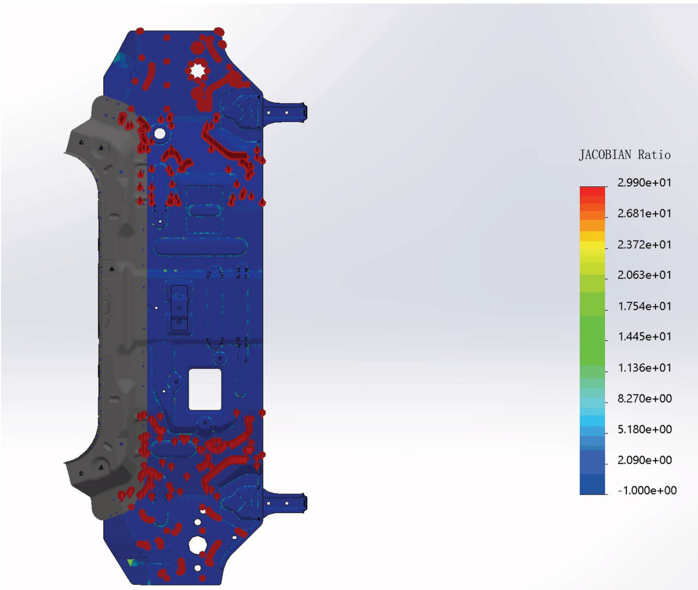

Fig. 8. Model of thin plate after force loading

In Dynaform software, the AutoSetup module is used to analyze and set up the thin plate stamping process, define the material parameters of the sheet metal, define forming tools such as concave dies, convex dies, and edge rings in the tool settings module, and define the interface of the forming tools. Set the friction coefficient between tools, define the clamping and stamping processes, analyze and set the corresponding mold speed and edge pressure. When setting up the software, the control parameters for solving are all default parameters. After completing all settings, submit the solution to obtain the basic parameters of the punch and die of the stamping die.

4.2 Variable Pressure Forming Process

Variable edge force refers to the loading form of edge force that varies with the stroke of the edge ring during the stamping process. This loading method can effectively control the flow law of materials during the stamping process, thereby controlling the occurrence of defects such as tearing or wrinkling of parts. In the case where the forming of the rear panel of the automobile cannot meet the requirements under the constant edge pressure loading method, this section will use the variable edge pressure loading method to simulate and analyze the stamping process of the rear panel of the automobile [17]. Taking the incremental variable edge force as an example for research, Fig. 9 shows the interface defining variable edge force in Dynaform.

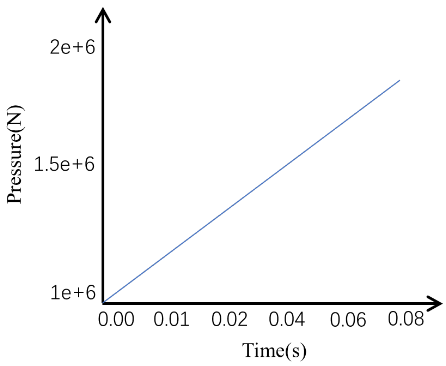

Fig. 9. Model of thin plate after force loading

In the post-processing module, reading the d3plot file obtained from simulation analysis can obtain the strain map of the car rear panel, as shown in Fig. 10.

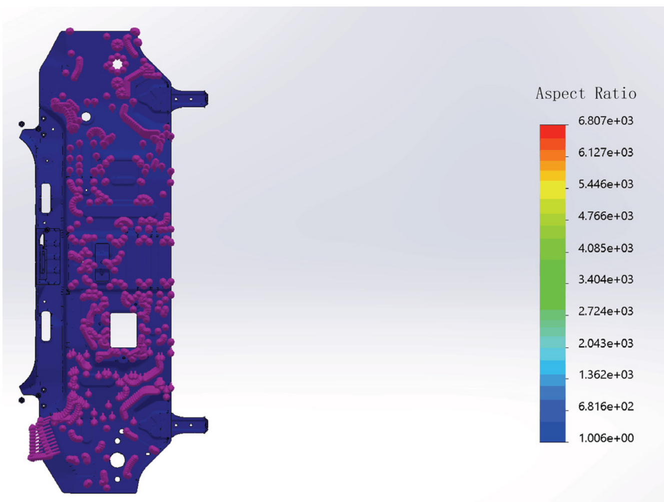

Fig. 10. Main strain diagram of rear panel

From the above figure, it can be seen that the forming quality of the car rear panel is good, with no defects such as tearing or wrinkling. The minimum thickness of the car engine hood is 0.537mm, and the maximum thickness is 0.792mm, which means the maximum thinning rate is 23.28% and the maximum thickening rate is 13.10%. The maximum principal strain of the car rear panel is 0.245, which is a relatively small strain value. The use of variable edge force loading method can basically meet the forming requirements, but it still needs to be optimized.

4.3 Optimization of Processing Technology Parameters

In the process of stamping forming, the forming quality of parts is not only affected by the edge pressure, but also by multiple other process parameters, such as sheet thickness, friction coefficient, mold clearance, stamping speed, etc. In order to achieve better forming results, numerical simulation analysis methods will continue to be used to study the influence of different influencing factors on the forming effect, and orthogonal experimental design will be used to optimize the above influencing factors and obtain the best forming process plan.

Select four forms of variable edge force, namely (a) increasing type, (b) decreasing type, (c) V type, and (d) Λ type, and conduct numerical simulation analysis on different loading methods using numerical simulation analysis. At the same time, control other process parameters as friction coefficient 0.132, stamping speed 6000mm/s, and mold clearance 0.79mm. The forming results of the rear panel under different loading pressures are shown in Fig. 11.

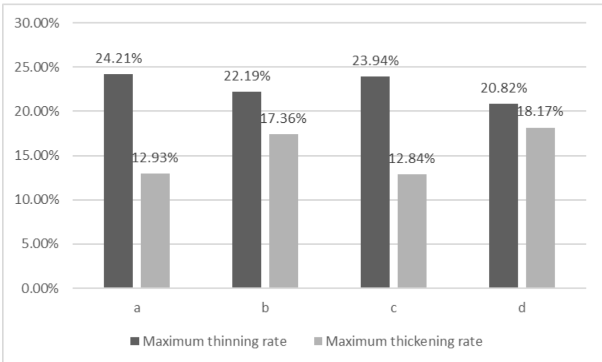

Fig. 11. The forming effect of different edge pressing forces

The forming effect is better under the variable edge force of loading method a and loading method c, and the results are relatively close; Under two types of variable edge forces, loading method b and loading method d, wrinkling appeared on the surface of the part. Further analysis reveals that under the two types of variable edge force loading methods, b and d, the edge pressure in the later stage of automotive engine hood stamping is relatively small, resulting in an increased risk of wrinkling and a reduced risk of part tearing. The difference in the forming effect of the car engine cover plate between loading modes a and c is relatively small, indicating that the edge pressure in the early stage of part stamping has a small impact on the final forming effect of the part. Through the analysis of the influence of variable edge force on the formability of automobile engine hood, it is concluded that, without considering the influence of other process parameters, the incremental variable edge force loading method has the best effect among the four selected variable edge force loading methods in this paper.

The size of the mold gap can affect the flow state of the sheet metal between the convex and concave molds, thereby affecting the forming quality of the parts. A too small mold gap may lead to severe thinning and even tearing of the parts, while a too large mold gap may cause wrinkling of the parts. In order to study the influence of mold clearance on forming, four mold clearance values of (a) 0.635mm, (b) 0.750mm, (c) 0.845mm, and (d) 0.905mm were set for stamping simulation analysis. At the same time, other process parameters were controlled as increasing variable edge force, friction coefficient of 0.132, and stamping speed of 6000mm/s. The molding results under different gaps are shown in Fig. 12.

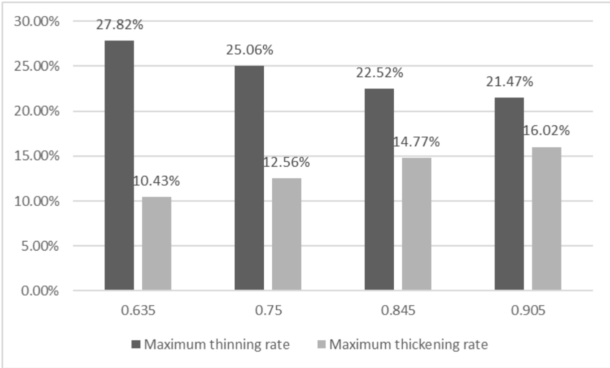

Fig. 12. Molding effect under different mold clearances

When the mold gap is 0.635mm, the maximum thinning rate reaches 27.82% and the maximum thickening rate is 10.43%, indicating severe thinning. When the mold gap is 0.750mm, the maximum thinning rate reaches 25.06% and the maximum thickening rate is 12.56%. The thinning problem is improved, but the thickening increases. When the mold gap is 0.845mm, the maximum thinning rate reaches 22.52% and the maximum thickening rate is 14.77%. The thinning is further improved, and the thickening continues to increase. When the mold gap is 0.905mm, the maximum thinning rate reaches 21.47% and the maximum thickening rate is 16.02%. While the thinning improves, the thickening continues to increase. To further understand the variation of part thickness with mold clearance, the curves of maximum thinning rate and maximum thickening rate with mold clearance are presented. Through numerical simulation analysis of the forming of the car rear panel under different mold clearances, it can be concluded that:

(1) When the gap between the car engine hood and the mold is 0.635mm, the final part becomes severely thinner, with a maximum thinning rate of 27.82%; When the mold gap is 0.905mm, the maximum thickening rate of the part is 15.03%, and the part will wrinkle.

(2) From Fig. 12, it can be seen that the maximum thickening rate of the part increases with the increase of the mold gap, while the maximum thinning rate decreases with the increase of the mold gap.

4.4 Orthogonal Experiment

Due to the different physical meanings or orders of magnitude of each parameter, it is necessary to perform de indexing and quantization processing, set the loading method a as the reference data group, calculate the closeness between each data group and the reference data group, and calculate the closeness using the absolute value of the interpolation between the two.

$$\Delta_i = \left| x_0(k) - x_i(k) \right| \left\{ k \in [1, m]; i \in [1, n] \right\} \quad \textbf{(5)}$$

The maximum range is expressed as:

$$\Delta(\max) = \max_i \max_k \left\{ \Delta_i(k) \right\} \quad \textbf{(6)}$$

The minimum range is expressed as:

$$\Delta(\min) = \min_i \min_k \left\{ \Delta_i(k) \right\} \quad \textbf{(7)}$$

The formula for calculating the degree of association of the reference data group a is as follows:

$$\rho(a) = \frac{\Delta(\min) + \lambda\Delta(\max)}{\Delta_{ai}(k) + \lambda\Delta(\max)} \quad \textbf{(8)}$$

In the formula, λ is the resolution coefficient. The larger the value, the higher the similarity between parameters. The smaller the value, the lower the similarity between parameters. This article sets λ as 0.55.
The correlation order of each group of parameters is represented as:

$$U_i = \left[\rho(1_i) + \rho(2_i)\right] / 2 \tag{9}$$

The correlation between the parameters of each group and the corresponding parameters of reference group a is expressed as:

$$g(i) = \frac{1}{N} \sum_{k=1}^{N} U_i(k) \tag{10}$$

Therefore, by obtaining the correlation between various parameters and sorting the correlation degree of each parameter, the parameter with the highest correlation degree among the four sets of data can be obtained.

Selecting two variables, variable edge and mold gap, as orthogonal experimental factors, the evaluation indicators are maximum thinning rate and maximum thickening rate. Based on the analysis in the previous section, the influence of various factors on the forming results is known, and the experimental design is shown in Table 2.

Table 2. Parameter settings for orthogonal experiments

Experiment number	Tensile strength	Hardening index (n)	Poisson's ratio (v)	Elastic modulus
	Blank holder force (kN)	Ram speed (*mm/ s*)	Frictional coefficient	Die gap (mm)
1	Incremental type	5000	0.125	0.750
2	Incremental type	6000	0.132	0.845
3	Decreasing type	5000	0.125	0.750
4	Decreasing type	6000	0.132	0.845
5	V-type	5000	0.125	0.750
6	V-type	6000	0.132	0.845
7	Λ type	5000	0.125	0.750
8	Λ type	6000	0.132	0.845

The results of range analysis can also be used to study the individual impact patterns of each influencing factor in orthogonal experiments, and to determine the optimal levels of each factor in orthogonal experiments. For the stamping forming of the automotive rear panel studied in this article, the changing patterns of various influencing factors under interactive effects are basically consistent with the effects of single factors.

Therefore, for the stamping forming of the car rear panel, the thickening of the parts is mainly in the edge pressing part, which will be cut off in subsequent processes, so it does not affect the final quality of the parts. The thinning of the parts is an important quality indicator of the product, and an excessive thinning rate will lead to insufficient strength of the car rear panel. For the maximum thinning rate, the optimal combination is A2B2C1D3, which is a decreasing variable edge force, stamping speed of 6000mm/s, friction coefficient of 0.132, and mold clearance of 0.845mm.

5 Simulation Experiment Results and Analysis

Robust optimization of automotive coverings is generally achieved during the production of parts by optimizing the process parameters of the forming process to ensure the quality and stability of the parts. The optimization process needs to consider various factors, such as the physical properties of materials, forming processes, key influencing factors, mold design, etc. The surface rebound compensation technology is used in the production process of automotive coverings to accurately calculate the material's elasticity and resilience, predict the deformation of automotive coverings, and achieve high-precision compensation of the mold surface. This can reduce the number of trial molds in the production process, improve production efficiency, and ensure the accuracy and consistency of automotive coverings.

The rebound amount of the rear panel parts after stretching and trimming is controlled to be around 1.27mm.

However, in order to ensure that the edge tolerance and free surface tolerance of the parts meet the requirements, and to ensure welding fit with surrounding parts during the assembly of the vehicle body parts, according to the enterprise's requirements, the maximum rebound amount of the rear panel needs to be controlled within ± 1mm. Rebound is caused by the influence of stress on the sheet metal during the stamping process. When the external force disappears, the release of stress inside the material will cause it to return to its original shape, resulting in the phenomenon of rebound. The complexity of different covering parts varies, the stamping process is not consistent, and the stress state during deformation is also different. Therefore, it is necessary to analyze the stress state of the parts before formulating a rebound strategy. AutoForm finite element software can analyze the stress-strain state of materials under stress. Several points are uniformly selected on the rear panel, and the main stress distribution cloud map is simulated and analyzed as shown in Fig. 13.

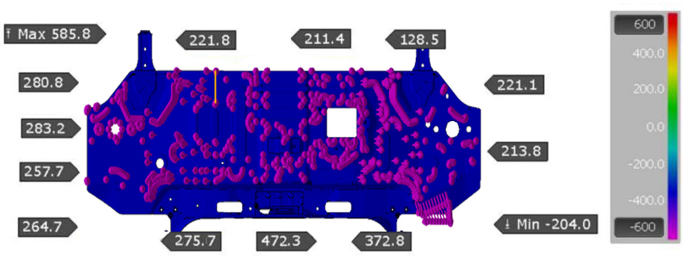

Fig. 13. Principal stress cloud map

Select all parameters of the deep drawing process as the parameters for rebound compensation analysis, and use the wall thickness of the surrounding plate stamping as the deviation standard deviation as the evaluation index. The calculation formula is:

$$\begin{cases} T_i = \sum_{i=1}^{n} x_i \\ t_i = \dfrac{T_i}{r} \\ R = \max\left(T_1, T_2, T_3, T_4\right) - \min\left(T_1, T_2, T_3, T_4\right) \end{cases} \tag{11}$$

T_i is the corresponding experimental indicator, t_i is the average value of the experimental indicator, r is the number of times each level appears on any column, and R is the difference between the maximum and minimum values of the experimental indicator at each level under various process parameters. The optimal level of each process parameter is determined by T_1. The smaller the T_1, the smaller the standard deviation of the wall thickness deviation of the formed part at this level of process parameters, that is, the closer the actual wall thickness of the formed part is to the theoretical wall thickness.

The value of wall thickness rebound is the difference between the actual wall thickness and the theoretical wall thickness, and the calculation formula is as follows:

$$t_{\text{huitan}} = t - t' \tag{12}$$

$$t_{\text{huitan}} = kt + b \tag{13}$$

t_{huitan} is the value of wall thickness rebound, t is the value of actual wall thickness, and t' is the value of theoretical wall thickness. Then, based on the relationship between theoretical wall thickness and wall thickness rebound values (13), a series of experimental data fitting and quality inspection were carried out using curve fitting tools. The relationship between theoretical wall thickness and wall thickness rebound was used to compensate for the rebound of wall thickness, resulting in higher accuracy of the wall thickness dimensions of the formed parts.

Based on the above analysis, the final stage after the parts are completely trimmed is selected for rebound compensation. Using the geometric node displacement compensation method, reverse compensation is carried out by calculating the magnitude and direction of rebound, continuously iterating new compensation surfaces, and ultimately achieving the required part accuracy. The specific flowchart of geometric node displacement compensation is shown in Fig. 14.

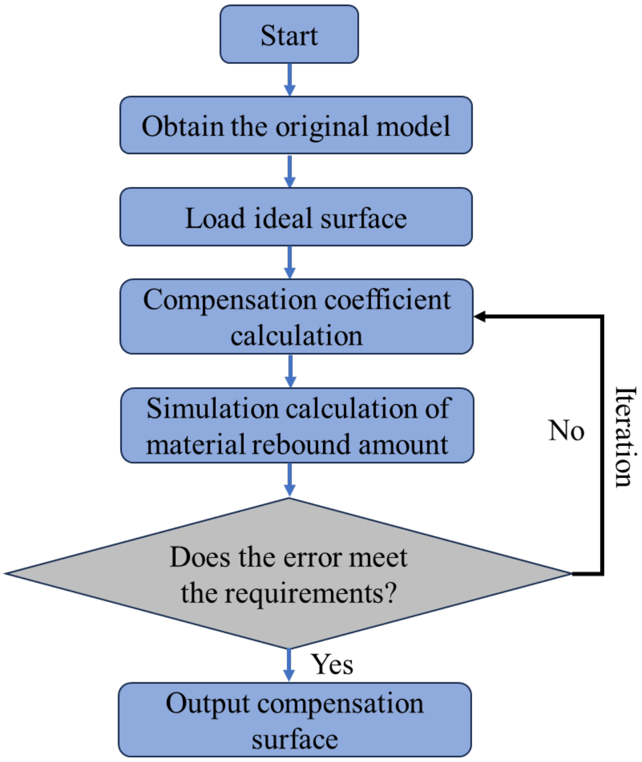

Fig. 14. Surface compensation process

Through rebound compensation simulation, the optimal surface for reducing the springback of the part during forming is constructed, and the maximum springback of the part meets the target. However, due to changes in the mold surface, the forming quality under the robust optimization solution will also change, and it is necessary to conduct another simulation evaluation of the stamping quality under this surface. The fully stretched area of the part accounts for 90.21%. Prove that the overall drawing quality of the parts after rebound compensation is good, with low rebound and no risk of wrinkling or cracking. Therefore, under the fifth rebound compensation square die surface, when the key process parameters are combined as concave die fillet radius of 11.62mm, friction coefficient between sheet metal and mold of 0.132, edge pressure of 867KN, and drawing bead resistance coefficient of 0.241, the maximum thinning rate of the part is 0.211, the maximum rebound amount is 0.81mm, and the overall drawing quality is high, with quality robustness reaching 6sigma level, which can be used as the final result to guide trial production.

6 Conclusion

This article takes the rear panel of automobile cover parts as the research object, analyzes its forming mechanism, and based on this, selects the loading method of variable edge force when the constant edge pressure cannot meet the forming requirements of the automobile. Based on orthogonal experiments, the process parameters are optimized, and finally, stamping forming experiments are conducted. The conclusion and innovation of this article are as follows:

1) Firstly, a mesh model of the stamping die for the automotive rear panel was established, and the mesh adaptive algorithm was used to optimize and analyze the mesh model of the die, obtaining the basic parameters of the die.

2) The stamping process of automobile rear panel was studied using numerical simulation method, and the simulation results were analyzed based on the forming limit, thickness variation, and maximum principal strain. The results showed that in the case where a constant blank holder force could not obtain qualified results, a variable blank holder force forming method was proposed to achieve better forming effects.

3) The influence of process parameters such as variable edge force, friction coefficient, stamping speed, and mold clearance on forming performance during the forming process was studied through simulation analysis, and corresponding influencing laws were obtained. For the formation of the automotive rear panel studied in this article, the most reasonable combination of processing parameters was obtained. The maximum thinning rate of the part increases with the increase of friction coefficient, while the maximum thickening rate decreases. The impact of stamping speed on the thickness of the part is relatively small. The maximum thinning rate of the part decreases with the increase of mold clearance, while the maximum thickening rate increases.

4) Based on orthogonal experiments, the stamping process parameters of the car rear panel were analyzed, and the optimized variables were variable pressure edge force, friction coefficient, stamping speed, and mold clearance. The optimization objective was the maximum thinning rate, and the optimal process parameters were obtained. Then, numerical simulation analysis was used again to obtain the best forming effect.

5) Finally, based on the optimal processing technology, the feasibility of the results was verified through simulation experiments. The experimental results were consistent with the simulation results in terms of the forming state and thickness changes of the parts, which also verified the accuracy of the simulation analysis.

The further research directions of this article are summarized as follows:

1) All research based on stamping simulation in this article requires that the sheet metal mesh and tool mesh be divided into triangular meshes during stamping simulation mesh division, or into quadrilateral meshes during simulation. After the simulation calculation is completed, the quadrilateral mesh is split into triangular meshes for subsequent research. Further research is needed on precise surface machining methods based on quadrilateral meshes or the coexistence of triangular meshes and quadrilateral meshes.

2) The research on the forming theory of automotive panels is relatively shallow. Based on the characteristics of automotive panels, the mechanism of their forming process can be studied more deeply, laying a more solid foundation for simulation analysis.

3) This article did not study the springback during the forming process of automotive panels, which is also an important factor affecting automotive panels. Further research can be conducted in this direction.

References

[1] Q.-H. Lin, Q.-S. Chen, Y.-P. Wang, Lightweight Design and Stamping Quality Control of Automobile Panels Made of Aluminum Instead of Steel, Journal of Taiyuan University (Natural Science Edition) 41(2)(2023) 39-44.

[2] S.-M. Li, C.-Q. Su, W. Gao, Influence of cavity pressure on forming quality of automobile outer covering parts during stamping process, Die & Mould Industry 50(3)(2024) 33-39.

[3] A.-P. Tan, Y. Duan, X.-X. Tao, J. Luo, Y.-B. Li, Numerical Simulation Analysis and Mold Design of Movable Cover Plate Based on Moldflow and UG, Journal of Netshape Forming Engineering 15(9)(2023) 65-73.

[4] H.-W. Mao, Y.-M. Wang, D.-Y. Yang, Study of injection molding process simulation and mold design of automotive back door panel, Journal of Mechanical Science and Technology 36(5)(2022) 2331-2344.

[5] C.-Y. Deng, S.-J. Dong, W. Tan, Design and optimization on stamping parameters for lightweight floor parts of a new energy vehicle, Forging & Stamping Technology 43(10)(2018) 68-71+75.

[6] X. Jiao, X. Li, Processing Parameters Optimization of the Automobile Panels Based on Fuzzy Analysis, Machinery Design & Manufacture (2)(2023) 112-116.

[7] X.-H. Luo, H.-Q. Cai, J. Guo, Optimization of Stamping Process and Die Structure Design for the Upper Cover of Automobile Battery Box, Die & Mould Manufacture 23(11)(2023) 1-3+6.

[8] H.-J. Gong, T. Zhou, H. Li, J. Peng, Reverse Modeling and Die Casting Die Design of Aluminum Alloy Parts Based on Geomagic, Foundry 69(6)(2020) 596-601.

[9] B.-X. Liu, X.-L. Dang, M. Liu, L. He, Design of Composite Hot Pressing Die Based on Finite Element Method, Chemical Engineering & Machinery 50(6)(2023) 915-919.

[10] J.-J. Li, J.-B. Zhao, Z.-Q. Zhang, F.-L. Meng, C.-G. Bai, Design and optimization on extrusion die for TC4 titanium alloy T-shaped thin-walled profile, Forging & Stamping Technology 48(11)(2023) 159-166.

[11] S.-B. Xu, X.-M. Xue, G.-C. Ren, C.-N. Jing, Finite element analysis of the forming of car trunk floor cover, Die and Mould Technology (6)(2021) 54-60.
[12] W.-C. Du, Application of Sigma Robustness Analysis Software in Stamping Die Tryout, Die & Mould Manufacture 21(5)(2021) 21-24.
[13] H.-J. Yan, B. Xing, S.-J. Zhang, S.-B. Ma, W. Wang, Z.-G. Yan, Z.-S. Liu, Study on springback analysis and die compensation of front anti-collision beam based on Dynaform, Journal of Plasticity Engineering 30(8)(2023) 35-41.
[14] Y. Bai, Q.-Q. Guo, M. Jia, Y.-B. Han, H. Li, X. Zhang, Strong and weak blank holder control technology on deep drawing for stainless steel hyperbolic special-shaped skin, Forging & Stamping Technology 48(12)(2023) 111-120.
[15] J. You, H.-Y. Gong, S.-B. Liu, X.-P. Jia, P.-Q. Xu, Z.-H. Liao, Stamping and optimization on 6016 aluminum alloy radiator shell based on Dynaform and response surface method, Forging & Stamping Technology 47(3)(2022) 54-58.
[16] S.-X. Yan, T.-P. Li, C.-L. Li, W. Liu, J. Wang, Two Dimensional Finite Element Mesh Generation Algorithm for Waveguide Discontinuity Analysis, Journal of China Academy of Electronics and Information Technology 18(4)(2023) 309-313.
[17] Y. Wang, Variable blank holder force stamping technology for automotive shield part based on FEM, Forging & Stamping Technology 47(8)(2022) 41-46.