Topology optimization and lightweight design of stamping dies for forming automobile panels

Ting Su1,3 · Tao He2 · Renqi Yang2 · Maojun Li1,3

Received: 4 August 2021 / Accepted: 3 July 2022 / Published online: 11 July 2022 © The Author(s), under exclusive licence to Springer-Verlag London Ltd., part of Springer Nature 2022

Abstract
Accurate prediction of deformation and stress distribution on stamping die components is critical to guarantee structure reliability and lightweight design. This work aims to propose a new method for predicting die structural behaviors and reducing total weight based on numerical simulation. Sheet metal forming simulation was firstly conducted to obtain the accurate forming contact force. The linear static structural analyses under different load conditions were performed to investigate the deformation and stress distribution of the die structure. Topology optimization was employed to realize the lightweight design on the premise of ensuring structural safety. According to the manufacturing techniques and initial optimization results, the die structure was redesigned to guarantee the manufacturability of the new structure. The proposed methodology has several advantages of decreasing model scale, precluding intricate contact condition settings as well as time saving. A long beam stamping die used for forming automobile panels was selected to validate the proposed methodology, and about 18% weight reduction was achieved.

Keywords Topology optimization · Structural behavior · Lightweight design · Stamping

1 Introduction
Stamping dies are widely used in the mass production of sheet metal parts. Time-consuming trial phases are required to ensure the die precision and accuracy since the product quality significantly depends on the surface finish and dimensional accuracy of dies. In addition, a sufficient safety margin is required to prevent structural failure of dies. Throughout their life cycles, the dies are required to undergo various handling operations for transportation and maintenance activities, such as lifting, stacking, and rotating. It is vital to ensure their safety during such handling operations since unsafe structures might lead to abrupt catastrophic consequences. Moreover, die failure or excessive deformation during the stamping process can cause huge economic losses and extend the delivery time of final products. Hence, it is crucial to investigate the die structural behaviors for safety and lightweight design. Regarding the study on die structural behaviors, the most challenging aspect is to determine the applied loads during the stamping process [1, 2]. Currently, the actual forming contact force is difficult to be recorded via conventional experimental methods. With the availability of finite element analysis (FE analysis) and increasing hardware instruments, the die structure deformation and introduced stress distribution under loading conditions can be predicted. The forming loads in the stamping process should be calculated accurately for die structural analysis and weight reduction [3, 4].

Previous numerical simulation works were conducted regarding the forming contact force in the stamping process. An innovative approach was proposed by Pereira et al. [5] to investigate the contact situations in the sheet metal stamping process. And then this numerical simulation method was implemented to explore the contact pressure distribution between die and workpiece within the die radius area in sheet metal forming processes [6]. They concluded that peak contact pressures were sensitive to a number of
parameters, including bending ratio and ultimate tensile strength of workpiece material. While the impacts of forming parameters (i.e., friction coefficient, blank holder force) were limited. To determine the contact pressure distribution on the trimming/blanking die edges, Rafiee et al. [7] created a 2D-plane strain model considering the intricate blank events in the stamping process, including elastic and plastic deformation, strain hardening, and crack initiation and propagation. It demonstrated that the punch was more likely to wear out than other areas as a consequence of excessive contact pressure.

The capability of assessing deformation and stress distribution on the die structure was particularly relevant for structure strength verification and forming quality assurance. Some published literature about die structural analysis suggested coupling the forming simulation and die solid finite element analysis into a combined model and then subjected to dynamic analysis using universal finite element software (e.g., Ls-Dyna, Ansys, Abaqus, etc.). Zhang et al. [8] outlined the procedures to obtain the stress and strain field distribution during U-channel forming process using Abaqus software. They studied the strain history of some representative points on the die and divided it into three phases. Sun et al. [9] employed the finite element analytic method to study die failure in the stamping process and related die failure mechanisms, including large principal stresses and large shear stresses. Effective suggestions were given to alleviate the principal stress concentration. To evaluate the stress distribution on the punch, Wang et al. [10] established a coupled finite element model (FE model), which was subsequently submitted to the Ls-Dyna to receive deformation and stress distribution on the punch. The punch was modeled as an elastic deformable body, and the entire forming process was taken as a dynamic problem. They discussed the maximum values of stress and displacement during the dynamic stamping process.

Although the traditional dynamic method of coupling die structural analysis and sheet metal forming simulation (SMF) allowed for the acquisition of die stress-strain state and blank forming results in one calculation, it also introduced several issues, such as large model, sophisticated contact settings, as well as long solving time. Firstly, some small structural features on die-forming surfaces (e.g., fillets and drawbeads) were required to be fine meshed for sheet metal forming, leading to increased element amount and enlarged model. Secondly, it was significant to define the contact characteristics between different die components during the entire dynamic analysis. In addition, the contact conditions should be assigned to the interfaces of die/blank, which would change with each stamping step. As all the die components were taken as elastic ones, the contact conditions were extremely intricate, resulting in an enormously long solving time, especially for large parts on an industrial scale. Recently, several research works indicated the considerable potential of stress distribution prediction for further lightweight design. Nie et al. [11] proposed an approach for realizing the lightweight design of an actual polymer pipe extrusion die by integrating stress analysis and structural optimization, achieving 21.67% mass reduction without compromising specified performances (productivity, static stiffness, compressive strength, and assembly property). The topology optimization procedures were conducted by Xu et al. [12] to reduce the weight of a blankholder while improving its stiffness. The original blankholder was effectively decreased in weight by 28.1%. The model was taken as a dynamic problem and calculated in Ls-Dyna. Some researchers also proposed a method to couple different meshes including coarse 3D meshes of elastic dies and fine surface meshes [13].

The main objectives of this work are to propose and explore a methodology for predicting the stress distribution and displacement of die components and subsequently reducing the weight of related die components. This work was organized into four sections. Section 2 described the proposed methodology for structural analysis and topology optimization of drawing dies. In Sect. 3, a long beam drawing die was selected as a demonstration to validate the proposed methodology for an industrial purpose. Finally, the main conclusions were detailed in Sect. 4.

## 2 Methodology

Figure 1 presents the proposed methodology for structural analysis and topology optimization of drawing dies. The process in the sheet metal forming simulation started with a 3D part model and eventually recorded the reaction forces between die components and sheet blank. The steps in the process are listed as follows.

1. 3D CAD model of the drawing die: die, punch, and blankholder, positioned according to their operation (CATIA V5, Siemens NX, etc.).
2. FE model of sheet metal forming simulation: a finite element model was developed before the stamping simulation.
3. Stamping forming simulation: it was conducted in the sheet metal forming software packages, where the dies were represented as 2D rigid surfaces with no deformation.
4. Contact forces: the contact forces acting on each contact surface between die components and sheet blank were determined following stamping simulation. The contact forces on each node of the thin shell elements were calculated using stamping process simulation, and
the nodal forces were taken as the boundary condition in the structural analysis.

Based on the above steps, the standard sheet metal forming simulation was conducted, and the accurate contact forces during the forming process were recorded. The recommended method of transferring accurate forming contact forces from 2D rigid die surfaces to a structural analysis model was through a technique called load mapping algorithm. With the implementation of the load mapping algorithm, the accurate reaction forces determined from the stamping forming simulation were applied as force boundary conditions on the die structural analysis model. The forming contact forces were taken as nodal force components in $X$, $Y$, and $Z$ directions and implemented on the corresponding element nodes. In this procedure, the die components were defined as elastically deformable bodies against the given nodal forces, allowing the elastic deformation experienced by the die components to be calculated. Since the suffering loads on the die surfaces were primarily dependent on the contact forces, the results of die structural analysis would be in good agreement with actual engineering only if accurate forming contact forces were applied. The main objective of die structural analysis was to receive the displacements/stresses distribution under the reaction of accurate forming contact forces, which included the following procedures:

1. **Simplification:** geometry 3D CAD model needed to be cleaned up before meshing. For instance, fine-meshed fillets and drawbeads were indispensable for accurate sheet metal forming analysis, while such small features needed to be removed due to their limited influences on
the overall stress distribution of die structure. Simplification of such small structure features would eliminate the need for high mesh densities at these locations. This simplification operation was necessary to balance the quality and quantity of the mesh and improve the calculation efficiency of die structural analysis.

2. Mesh: the mesh used for die structural analysis was 3D solid element mesh, which was generated to represent the 3D CAD geometry of the drawing die components. The 3D mesh was composed of hexahedral elements or tetrahedral elements created from the die components. For geometries with complicated structure features, such as stamping dies, the hexahedral mesh was more time consuming to generate than the tetrahedral mesh [14].

3. Boundary conditions: different loading cases were considered in this procedure. The precise forming contact forces extracted from the sheet metal forming simulation were transferred to die static structural analysis model, which served as the force boundary conditions of the stamping forming case. Apart from the stamping case, the die lifting case was taken into account, since lifting die in the lifting lugs was regarded as a required activity for die transportation and cleaning. Besides, the support of the press machine was regarded as a constraint condition. In general, the fixation holes on the bolster side of the drawing die clamped tightly, generating constraints in all displacement directions.

4. Materials characteristics: material characteristics based on the drawing of die components must be assigned to the elements. Generally, three material characteristic values, the elastic modulus ($E$), Poisson’s ratio ($\nu$), and density ($\rho$), were required to calculate the stiffness matrix in the die static structural analysis model.

5. Solver calculation: after establishing the die structural analysis model, the model should be submitted to the commercial finite element solver software for calculation, such as Nastran, Ansys, OptiStruct, etc.

6. Structural analysis results: the results were obtained after calculation, such as stress (three principal stresses, von mises stress, etc.) and elastic deformation experienced by the drawing die components. These were significant parameters to evaluate the die component behavior during the stamping process.

The standard sheet metal forming simulations were carried out to obtain forming loads on the die surface, then die structural analysis was implemented to investigate the elastic behavior of the die component. The next objective of this proposed method was to take the structural response of die components into account to realize weight reduction. As one of the most effective approaches to achieving lightweight design, topology optimization was selected to determine the optimal material distribution in the design domain. The principal objective of topology optimization was to solve a problem of topology design for minimum compliance for a linear elastic structure subjected to one or more constraints (e.g., maximum volume fraction constraint, static displacement constraint) [15]. The element densities of the given design space, which formed the design variables for the given available topology optimization model were modified according to the force distribution and elastic calculation results, creating an optimized design for the component [16]. Identifying design space would be discussed in this subsection, in which the typical steps for the topology optimization were illustrated.

1. Identify design space: non-design space and design space needed to be defined according to the boundary conditions to establish the topology optimization model. The optimum material distribution within the given design space was obtained by positioning and rearranging the structural design elements [17], while the characteristics of the non-design space would remain unchanged during the optimization calculation. To guarantee the forming of stamping parts, the die materials contacting with the blank in the stamping process were set as non-design space [8]. In addition to the part touching materials, some assembly mating structure features (e.g., outer shape of the dies) were also taken as non-design space to be maintained during the topology optimization. The rest of the space was designated as the design space for further topology optimization.

2. Definition of topology optimization: in the establishment of the die topology optimization model, design variables, and constraints, together with optimization objectives needed to be defined.

3. Topology optimization: the commercial topology optimization solver OptiStruct has been increasingly used in many industries for its comprehensive techniques. It was the recommended optimization software in the proposed methodology to generate the optimal die material distribution.

4. Die structure redesign: based on the initial topology optimization results, the 3D CAD geometric model of die components had to be redesigned since obtained topology optimization structures were conceptual. Die structures were conducted according to both the casting and manufacturing techniques to guarantee the manufacturability of the new structures [16]. Most importantly, a check cycle would be applied to guarantee the performance of the redesigned structures, as illustrated in the right column of Fig. 1. The redesigned die structures were analyzed, subjecting to the same loading case (operation case and transportation case) applied in the original structure structural analysis. The obtained results of optimized structures would be compared with
the von mises stress and deformation on the original die structures.

In conclusion, the sheet metal forming simulation was first conducted to acquire the accurate forming forces, which were transferred to the die structural analysis model and topology optimization model using a technique called load mapping algorithm. Subsequently, die topology optimization was performed to receive the optimal material distribution. Based on the initial optimized structure, the redesigned die structures that required structure strength verification were created. The proposed methodology has several obvious advantages in comparison with the strength analysis method that coupling the sheet metal forming simulation and structural analysis into one FE model. Firstly, in this proposed methodology, two different element meshes were implemented for the sheet metal forming simulation and structural analysis, which would bring a reduction of the model scale, yielding a total shorter solving time consequently. Whereas in the conventional coupling model, to achieve accurate results, the high-density element meshes were required to be created, resulting in enormously increased model scale, especially for large industrial parts. Apart from the advantage of time saving, this proposed method made the calculated results become reliable and in good agreement with engineering practice. This innovative method simplified the complexity of the problem, and the calculation could be considered linear static analysis rather than a dynamic problem since the forming contact forces derived from sheet metal forming simulation were treated as the load boundary conditions in the FE model for structural analysis. While in the conventional method, it was regarded as a dynamic problem, which required the contact characteristics to be assigned to each interface between die components, thus causing the contact conditions settings to become sophisticated. This methodology has provided an effective approach for dealing with structural behavior prediction and weight reduction for drawing dies. The detailed procedures would be evaluated in the case of a particular long beam drawing die in the next section.

3 Case study

In this section, a long-beam drawing die was selected as the object to illustrate the procedures presented in Fig. 1 and validate the effectiveness of the proposed methodology for industrial purposes. This drawing die was used to manufacture an automotive inner liner part. In the case study, the sheet metal stamping process was first conducted to obtain forming nodal forces, which were taken as the boundary conditions for die structural analysis model consequently. The structural behaviors of the long-beam drawing die subjected to two different load cases were revealed in the die structural analysis, in which the deformation experienced by the die components, the von mises stress and the weight of the die components was studied.

3.1 Sheet metal forming simulation

The accurate contract forming loads acting on the die should be analyzed to perform die structural analysis and topology optimization. The accurate contract forming loads in this proposed methodology was obtained through a sheet metal forming simulation. The starting point was the 3D CAD geometric design of drawing die components including die, punch, and blank holder.

Subsequently, the study set some parameters related to the sheet metal forming simulation. The material for the blank was DC04, with a thickness of 0.8 mm, the properties of the blank was presented in Table 1. Figure 2 presents the input true stress–strain curve for the DC04 material. The blank meshed with the full integration Belytschko–Tsay shell elements with 7 through-thickness integration points. Material type 37 was selected for the blank. Material type 37 is a

| Table 1: Detail material properties used for the blank |
|----------------|---------|------|-------|----------|-------|-------|-------|
| **Material**  | **Thickness** (mm) | **E** (GPa) | **ν** | **σ_s** (MPa) | **K** (MPa) | **n** | **r_00** | **r_45** | **r_90** |
| DC04          | 0.8     | 206  | 0.3   | 170      | 507.7   | 0.21  | 1.87   | 1.3    | 2.14    |

![Fig. 2 True stress–strain curve of the DC04 material](image)
material model, which is fully iterative and applicable only for shell elements. Hill’48 yield function was adopted to describe the initial anisotropy of the blank. Hill’48 yield function can accurately reflect the anisotropic behavior of materials with a relatively high thickness anisotropy coefficient (such as steel materials). The Hill’48 yield function can be written as follows [18].

\[
F(\sigma_{22} - \sigma_{33})^2 - G(\sigma_{33} - \sigma_{11})^2 + H(\sigma_{11} - \sigma_{22})^2 + 2L \sigma_{23}^2 + 2M \sigma_{31}^2 + 2N \sigma_{12}^2 - 1 = 0
\]  

(1)

where \( F, G, H, L, M, N \) are anisotropic parameters relating to the yield stress. \( \sigma_{11}, \sigma_{22}, \sigma_{33} \) are tensile yield stresses, while \( \sigma_{12}, \sigma_{23}, \sigma_{31} \) are shear yield stresses. The stamping process is typically thought as a plane stress state. The above model can be simplified into 2D as follows.

\[
F(\sigma) = (\sigma_1^2 + \sigma_2^2 - \frac{2r}{r+1} \sigma_1 \sigma_2)^{1/2}
\]  

(2)

where \( \sigma_1 \) and \( \sigma_2 \) are principal stresses, and \( r \) is the mean value of anisotropic coefficients along three directions (i.e., rolling direction, diagonal direction, and transversal direction), which is given as follows.

\[
r = \frac{r_{00} + 2r_{45} + r_{90}}{4}
\]  

(3)

The stamping process model consisted of the same surface of the target part (a long beam) as the original drawing die. The tools were assumed to be perfectly rigid, and therefore only the surfaces in contact with the blank were required in the model. In the sheet metal forming process, the upper die meshed first, and then the punch and blank holder meshes were generated by the offsetting technique, as illustrated in Fig. 3. The key control parameters in the sheet metal forming simulation were presented in Table 2. The upper die velocity was set as 2000 mm/s, while the low punch was stationary in the sheet metal forming simulation. The tool velocity implemented in the stamping process was not the physical velocity in real production, but the virtual speed increased from the physical it. A suitable velocity setting for sheet metal forming simulation could speed up the stamping process and reduce the solving time of simulation while receiving a satisfying approximation of the forming process.

Since large strains were involved in the process, nonlinear analysis was performed [15]. It was a dynamic problem in which the contact loads on the blank varied in each forming time step. The established the sheet metal forming simulation model was submitted to the solver Ls-Dyna, to calculate the accurate nodal forces on each element’s nodes during the sheet metal forming simulation. The results of the sheet metal forming simulation are detailed in Fig. 4, which presented the load–displacement curves from the generated RCFORC file during SMF simulation. The abscissa represented the stamping stroke, while the ordinate indicated the corresponding resultant forces in \( X, Y, \) and \( Z \) directions of all contact nodes between the die components and blank. It was noteworthy that the resultant forces escalated rapidly and peaked at the end of press stroke (i.e., simulation end time). The obtained peak forming force in the \( Z \) direction was 5700 kN (as illustrated in Fig. 4), which was implemented as boundary conditions in the next procedure for structural analysis.

### 3.2 Linear static structural analysis

The sheet metal forming simulation of the long beam was firstly conducted to evaluate the accurate contact forces at the end of stamping. As the drawing dies generally operated in the linear range of materials, linear static structural analysis was performed in this subsection to investigate the structural behaviors of die components. Besides the attained contact loads, another loading case was considered in the die structural analysis, in which the lifting forces during transportation were also applied to the die analysis model.

![Fig. 3 Selected case model for sheet metal forming simulation](image-url)
To prevent unnecessary problems, some small geometry features (e.g., fillets included in die 3D geometric models) were removed before discretizing the geometry models into 3D meshes. Due to the complexity of drawing die structure, 10-node 3D tetrahedral elements offering more precision with six additional nodes were chosen to discretize the die components. In this work, the punch and die shoe were modeled with approximately 430,000 tetrahedral elements. The punch, also known as die post, was constructed of Cr12MoV cast steel, and the die shoe was fabricated by HT300 cast iron. In linear static structural analysis, three material properties (i.e., elastic modulus, Poisson’s ratio, and density) were required to predict die structural behaviors under loading. The material properties for the punch and die shoe are presented in Table 3. Cast steel, which was a ductile material, exhibited similar performance when subjected to tension and compression conditions. While for the brittle material, cast iron, behaved much stronger in compression than in tension, with very little yielding occurred and the failure mode was predominated by fracture.

Different load cases were considered in this study. The first load case was the forming case, which represented the die component under peak load conditions. The force boundary was attained from the sheet metal forming simulation, where the contact pressure and drawbead forces on the die were calculated. Such contact pressure vectors could be represented by X, Y, and Z nodal contact force components, which were directly transferred from rigid shell meshes to the deformable solid elements established for linear structural analysis, using an efficient load mapping algorithm. The die shoe was clamped or bolted to the bolster of the press, thus the model for this load case was constrained in the X, Y, and Z directions at the bottom surface nodes. The established linear structural model for load case 1 is presented in Fig. 5. The obtained nodal forces were implemented on the corresponding element nodes in X, Y, and Z directions. The length of the arrows reflected the magnitude of the load, and the direction of the arrows indicated the direction of the load.

The analysis models were calculated in the linear solver Optistruct. To explore the structural behaviors of this long beam drawing die, the results of the linear structural analysis

![Fig. 4](image-url) Load–displacement curves obtained from the SMF simulation

**Table 3** Material properties for the die components

| Materials | Elastic modulus, $E$ (GPa) | Poisson’s ratio, $\nu$ | Density, $\rho$ (kg/m$^3$) |
|-----------|-----------------------------|------------------------|-----------------------------|
| Cr12MoV   | 218                         | 0.3                    | 7800                        |
| HT300     | 143                         | 0.27                   | 7300                        |

![Fig. 5](image-url) The established linear structural model for the forming case

The nodal forces obtained by SMF simulation

Constraints

© Springer
were analyzed. In this study, the die shoe was constructed of HT300 cast iron, a brittle material that exhibited stronger performance in compression than tension. Thus, the die shoe was most susceptible to failure due to tensile stresses. Hence, the maximum tensile principal stress (i.e., P1 stress) was applied as the main indicator to evaluate the behaviors of the die shoe, although all three principal stresses (P1, P2, P3 stress) were calculated by the solver. Meanwhile, the die post made of Cr12MoV cast steel was judged by the von mises stress.

Results and analyses of the stresses in forming case can be viewed in Figs. 6 and 7. For the punch, the maximum von mises stress reached 471 MPa, occurring at the sharp corner of the punch surface, as shown in Fig. 6. Sharp corners tended to concentrate stress, resulting in highly localized stress in these areas [17, 19]. For the die shoe, the stress eccentricity was observed in the reinforcement ribs, the von mises stress in the Y direction was significantly greater than that in the −Y direction due to the asymmetrical shape of the target part. The von mises stress of the reinforcement ribs in the Y direction reached 85 MPa, while only 26 MPa was recorded in the −Y direction, as illustrated in Fig. 7.

The second load case was a transportation case, which was an operation for transportation and cleaning. In production, transportation of stamping dies was set as a four-point lifting operation and lifting lugs were applied to attach all die components to facilitate transport. Since the dies were generally lifted with a very small and negligible acceleration, the lifting case was also considered a static condition. In the lifting case, the nodes in the lifting lugs were
constrained in all degrees of freedom except for the X-axis rotation. Only the die shoe instead of the entire beam drawing die was selected to conduct structural analysis for the lifting case since the die shoe structure suffered the greatest force during the lifting operation. The structure was supported to be lifted in four lifting lugs. The established mechanical model of the lifting case is shown in Fig. 10, where $P_1$ and $P_2$ were the equivalent pressure transferred by the weight of other die components ($g = 9.8 \text{ m/s}^2$).

Results for the lifting case are presented in Figs. 11 and 12. For the original die shoe structure, the maximum tensile principal stress (i.e., $P_1$ stress) of 11.8 MPa was achieved in the lifting case, and the maximum displacement was around 0.022 mm.

### 3.3 Structural optimization of stamping die component

The topology optimization method exhibits great potential in terms of mass reduction [16, 20]. In this subsection, the topology optimization process of the long beam drawing die component was conducted, and then the structures were redesigned based on the initial topology optimization results. Most importantly, the linear structural analysis of the redesigned structure was performed for strength verification by imposing the same level of forming load and a lifting load. To analyze the results of redesigned die structure, the displacements and von mises stress in the structure were investigated and compared with ones from the original structure.

The boundary conditions (derived from SMF simulation) and material properties (defined the same as those in linear structure analysis as mentioned before) were required before defining the design space, which was set as design variables of the die topology optimization model. The outer materials contacting with the blank for forming and assembling with other components were determined as non-design space, which would be excluded from density manipulation for optimization solver. Since one of the objectives of die structure optimization was to obtain the optimal distribution and size of reinforcement ribs, the bottom region of the die shoe was filled and assigned as the design space for topology optimization calculation. Figure 13 shows the FE model for topology optimization.

In the establishment of the topology optimization problem for the long beam drawing die, the design variables, constraints, together with the optimization objective were determined. Setting a volume fraction of 0.30, the volume of the optimized design area was required to be at most 30% of the initial design space volume. The objective function was defined to find the minimum compliance (i.e., the maximum stiffness) under a certain available amount of material volume. Results from the topology optimization are shown in Fig. 14, which met the objective function and constraints to minimize the compliance and use a volume fraction of 0.30. The outer red frame in the figure indicated that the material density was 1.0, and the material could not be removed.

Since the obtained topology optimization result in Fig. 14 was only a conceptual design with optimal material distribution, further efforts should be undertaken to reconstruct the structure with reference to the material distribution results of the topology calculation. Additionally, the general die...
design principles and current manufacturing techniques for die structure in industrial practice should also be taken into account when reconstructing die structures. The reconstructed structure is shown in Fig. 15. For the reconstructed die shoe, compared with the original die structure created using the traditional method, the location of reinforcement ribs was modified according to the topology optimization results, while the main reinforcement ribs were still arranged along the major force transmission position of punch based on the general die design principles, to resist the impact of forming force. Besides, the thickness of reinforcement ribs was reduced, and the number and dimension of lightening holes were increased. Furthermore, to achieve a relatively larger weight reduction, two lightening holes were introduced on the sidewall of the punch according to the structural characteristic of the automobile stamping die.

As mentioned in the proposed methodology, to verify the structural strength of the reconstructed structure, the same processes (as illustrated in Fig. 1) were conducted with the new structure, and analyses were carried out in both cases (i.e., forming case and lifting case). The linear static analysis results of the reconstructed structure are shown in Figs. 16, 17, 18 and 19. The maximum von mises stress on

Fig. 13 FE model for topology optimization of die shoe with a isometric view and b bottom view, c isometric view, and d top view of the non-design space

Fig. 14 Topology optimization results of the die shoe under multiple constraints

Fig. 15 Model of the reconstructed structure including a isometric view and b bottom view

Fig. 16 Von mises stress distribution on the reconstructed punch
the reconstructed punch decreased to 451 MPa after optimization. The von mises stress was uniformly distributed on the reinforcement ribs, as shown in Fig. 17, indicating that the stress eccentricity on the reinforcement ribs of the reconstructed die shoe was effectively alleviated.

To evaluate the effectiveness of this methodology, a table was summarized as presented in Table 4, where the numerical values of displacements, stresses, and weight in the original and reconstructed die were illustrated. It was found that the reconstructed structures had a reduced mass of 18%, with a 4.2% decrease in maximum von mises stress for the punch. For the reconstructed structure, the maximum tensile principal stress was 12.4 MPa, fulfilling the constraints settings upon it when lifting the structure in four supported lugs. Besides, for the optimized die structures with decreased weight, the displacement enhancement of 42.6% and 94.4% were recorded for the forming and lifting cases, respectively (Fig. 20). This could be attributed to the stiffness reduction of the die structures. The displacement increment of die elastic deformation in stamping forming could compromise the geometry accuracy of stamped parts, while it was negligible in the lifting case. Since die structures generally experience stiffness degradation and displacement increment after weight reduction, additional efforts should be applied to achieve geometric compensation and correction of the die forming surface by considering surface deformation data of optimized die structures, consequently acquiring the appropriate die surface geometry.

![Fig. 17 Von mises stress distribution on reinforcement ribs of reconstructed die shoe](image1)

![Fig. 18 Displacement distribution of the redesigned structure (magnitude)](image2)

![Fig. 19 Tensile principal stress result of the reconstructed die shoe for the lifting case](image3)

![Fig. 20 Displacement analysis of the reconstructed die shoe for the lifting case](image4)

### Table 4 Comparison between original and reconstructed die structure

| Variables                     | Original structure | Reconstructed structure | Differences |
|-------------------------------|--------------------|-------------------------|-------------|
| Weight                        | 1284 kg            | 1054 kg                 | −18%        |
| Maximum von mises stress (punch)-forming | 471 MPa            | 451 MPa                 | −4.2%       |
| Maximum displacement-forming  | 0.209 mm           | 0.298 mm                | +42.6%      |
| Maximum tensile principle stress (die shoe)-lifting | 11.8 MPa          | 12.4 MPa                | +5.1%       |
| Maximum displacement-lifting  | 0.0216 mm          | 0.0420 mm               | +94.4%      |
4 Conclusions

This work proposed a new methodology for analyzing the structural performance and realizing the weight reduction of drawing dies. Compared with the previous methodology, the proposed methodology presented several obvious advantages including good consistency with engineering practice, reduced model scale, prevention of sophisticated contact conditions setting, and consequently time saving.

1. A long beam drawing die on an industrial scale was selected to illustrate the procedures of the proposed methodology. To achieve a comprehensive and lightweight design, two different load cases were considered in both linear static structural analysis and optimization procedures, including the forming case and transportation case. The reconstructed models were analyzed and compared with the original die structure. It was found that an 18% weight reduction was achieved with a slight difference in structural performance.

2. The proposed method in this paper can be used to predict the stress distribution and verify the structural strength of stamping die components. It can also be utilized to optimize the die structure that was designed at an early stage, so as to reasonably improve the die structure, consequently ensuring the die structures meet the requirements of strength and lightweight design at the same time. Moreover, the attempt of this article could put forward new ideas for improving the current die structure design method.

Author contribution T Su: methodology, data collection, and writing-original draft; T He: data collection and writing-reviewing; R Yang: methodology and data collection; M Li: conceptualization, investigation, funding acquisition, writing-original draft, and editing.

Funding This work was supported by the Open Research Fund of the State Key Laboratory of High-Performance Complex Manufacturing (Kfkt2020-13).

Availability of data and materials Not applicable.

Declarations

Ethics approval Not applicable.

Consent to participate Not applicable.

Consent for publication Not applicable.

Competing interests The authors declare no competing interests.

References

1. Xie Y, Yue Y, Tang W, Feng M, Guo Y, Wang D, Zhu Y (2020) Topology optimization of blank holders in nonisothermal stamping of magnesium alloys based on discrete loads. Int J Adv Manuf Technol 106:671–681
2. Sun Y, Chen Q, Sun W (2015) Numerical simulation of extrusion process and die structure optimization for a complex magnesium doorframe. Int J Adv Manuf Technol 80(1–4):495–506
3. Li G, Long X, Zhou M (2019) A new design method based on feature reusing of the non-standard cam structure for automotive panels stamping dies. J Intell Manuf 30(3):2085–2100
4. Li X, Qian L, Sun C, Ning W (2021) The effect of loading mode on isothermal local loading forming of magnesium alloy rib-web component. Int J Adv Manuf Technol 114:2485–2497
5. Pereira MP, Yan W, Rolfe BF (2010) Sliding distance, contact pressure and wear in sheet metal stamping. Wear 268(11–12):1275–1284
6. Pereira MP, Yan W, Rolfe BF (2008) Contact pressure evolution and its relation to wear in sheet metal forming. Wear 265(11–12):1687–1699
7. Raffe R, Rolfe BF, Yan WY (2008) Simulation of contact pressure in sheet metal trimming/blanking process. Adv Mater Res 41–42:463–468
8. Zhang L, Zhao G, Li HP (2010) Research on stress-strain field and fatigue lifetime of the hot stamping die. Appl Mech Mater 34–35:1630–1634
9. Sun J, Lee K, Lee H (2000) Effects of geometry and fillet radius on die stresses in stamping processes. J Mater Process Technol 104:254–264
10. Wang H, Xie H, Liu Q, Shen Y, Wang P, Zhao L (2018) Structural topology optimization of a stamping die made from high-strength steel sheet metal based on load mapping. Struct Multi-discip Optim 58(2):769–784
11. Nie Y, Lin Y, Sun W, Huang T (2015) A methodology for optimal lightweight design of moulds and dies: a case study. In: Proceedings of ASME 2015 International Mechanical Engineering Congress and Exposition. Houston, Texas, USA, pp 13–19
12. Xu D, Chen J, Tang Y, Cao J (2012) Topology optimization of die weight reduction for high-strength sheet metal stamping. Int J Mech Sci 59(1):73–82
13. Haufe A, Roll K, Bogon P (2008) Sheet metal forming simulation with elastic tools in LS-DYNA. In: Proceedings of the 7th international conference and workshop on numerical simulation of 3D sheet metal forming processes. Interlaken, Switzerland, pp 743–748
14. Nilsson A, Birath F (2007) Topology optimization of a stamping die. In: Proceedings of the 9th international conference on numerical methods in industrial forming processes, pp 449–454
15. Hamasaki H, Nakazono M, Hino R, Yoshida F, Manabe H, Kondo H, Torigop V (2014) Stiffness improvement of stamping die by means of topology optimization. Adv Mater Res 939:266–273
16. Burkart M, Liewald M, Wied J, Todry T, Hartmann M, Müller M (2020) Optimization of a part holder design considering dynamic loads during return stroke of tool and ram. Procedia Manuf 47:861–866
17. Hasebe N, Iida J (1983) Intensity of corner and stress concentration factor. J Eng Mech 109(1):346–356
18. Hill R (1948) A theory of the yielding and plastic flow of anisotropic metals. Proc R Soc Lond Ser A 193:281–297
19. Yu S, Long M, Chen H, Chen D, Liu T, Duan H, Cao J (2018) Stress and friction distribution around slab corner in continuous casting mold with different corner structures. Metall Mater Trans B 49(3):866–876
20. Koşar F, Yaman K (2020) Structural design of an auto panel die component based on the contact pressure using topology optimization. Adv Mater Res 1158:165–176

Publisher’s Note Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.