Topology Optimization and Lightweight Design of Stamping Dies for Forming Automobile Panels

Ting Su
State Key Laboratory of Advanced Design and Manufacture for Vehicle Body, Hunan University, Changsha, 410082, Hunan, China
Tao He
3Jianglu Machinery & Electronics Group Co. Ltd., Xiangtan, 411100, Hunan, China
Renqi Yang
3Jianglu Machinery & Electronics Group Co. Ltd., Xiangtan, 411100, Hunan, China
Maojun Li (✉ maojunli@hnu.edu.cn)
State Key Laboratory of Advanced Design and Manufacture for Vehicle Body, Hunan University, Changsha, 410082, Hunan, China

---

Research Article

Keywords: Topology optimization, structural behavior, lightweight design, stamping

Posted Date: August 10th, 2021

DOI: https://doi.org/10.21203/rs.3.rs-791304/v1

Abstract

The accurate prediction of deformation and stress distribution on the stamping die components is critical to guarantee structure reliability and lightweight design. This work aims to propose a new method based on numerical simulation for predicting die structural behaviors and reducing total weight. The sheet metal forming simulation was firstly conducted to obtain the accurate forming contact force during stamping process. The linear static structural analysis with different load cases was then performed to investigate the deformation and stress distribution on die structure. Topology optimization was employed to realize lightweight design while ensuring structural safety. Redesign process for die structures was conducted according to both manufacturing techniques and initial optimized results to guarantee the manufacturability of new structures. 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 around 18% weight reduction was achieved.

1. Introduction

Stamping dies are widely used for manufacturing sheet metal parts in mass production. Time-consuming trial phases are required to ensure the dies to be precise and accurate, since the quality of products is imparted by the surface finish and dimensional accuracy of dies during manufacturing process. In addition, suitable safety margin is required to prevent structural failure of dies. During life cycles, the dies are subjected to various handling operations for transportation and maintenance activities (e.g. lifting, stacking, rotating, etc.). It is important to keep the safety of dies during the handling operations, since unsafe structures may lead to abrupt catastrophic consequences. Moreover, die failure or excessive deformation during stamping process can cause huge economic losses and extend delivery time of final products. Hence, it is crucial to investigate the die structural behaviors in order for safety and lightweight design. Regarding the study on die structural behaviors, the most challenging aspect is to determine the applied loads during stamping process [1, 2]. Currently, the actual forming contact force is difficult to be recorded via conventional experimental method. 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 during stamping process. An innovative approach was proposed by Pereira et al. [5] to investigate the contact situations in 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 the parameters, including bend ratio, ultimate tensile strength of the workpiece material. While the effects induced by the forming parameters (i.e. friction coefficient, blank holder force) were limited. Rafiee et al. [7] created a 2D plane strain model to obtain the contact pressure distribution on the trimming/blanking die edges. In this model, intricate blank events (i.e. elastic and plastic deformation, strain hardening and crack initiation and propagation) were taken into account. They demonstrated that the punch was more likely to be worn out compared with other areas as severe contact pressure was loaded on the punch.

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 literatures about die structural analysis suggested coupling the forming simulation and die solid finite element analysis into a combined model and subjected to the dynamic analysis using the universal finite element softwares (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 summarized the change into three phases. The finite element analysis approach was performed by Sun et al. [9] to study the failure of dies during stamping processes and the failure mechanisms were explored, 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] created a coupled finite element model (FE model), which was then submitted to the Ls-Dyna to receive deformation and stress distribution on the punch. In the model, the punch was taken as a elastic deformable body and the whole forming process was considered as a dynamic problem. They discussed the maximum values of stress and displacement during dynamic stamping process.

Although the method including die structural analysis and sheet metal forming simulation (SMF) could be accomplished in one calculation, and obtained stress-strain state of die structure and blank forming results at the same time, it also introduced several problems, including large model, sophisticated contact settings as well as long solving time. Firstly, some small structural features (e.g. fillets and drawbeads) exhibited limited influences on the die structural stress-strain state, while required to be fine meshed for sheet metal forming process, resulting in increasing total element amount and large model. Secondly, during the entire dynamic analysis process for sheet metal forming and structural analysis, it was significant to define the contact characteristics between die components. In addition, the contact conditions should be assigned to the interfaces of die/blank, which changed in each stamping steps. As all the die components were taken as elastic ones, the contact conditions were extremely intricate and led to a enormously long solving time especially for large parts on industrial scale. Recently, a number of research works indicated the considerable potential of stress distribution prediction for further lightweight design. Nie et al. [11] proposed a approach integrating stress analysis and structural optimization to realize lightweight design of an actual polymer pipe extrusion die, achieving 21.67% mass reduction without changing specified performances (productivity, static stiffness, compressive strength and assembly property). The topology optimization procedures were conducted by Xu et al. [12] to achieve weight reduction and stiffness improvement of a blankholder. The weight of original blankholder was successfully reduced by 28.1%. The model was taken as a dynamic problem and calculated in Ls-Dyna for the dynamic loading conditions during stamping process. 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 proposing and exploring a methodology to predict the stress distribution and displacement of die components, and subsequently reduce 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 the demonstration according to the described methodology, in order to validate the methodology for an industrial purpose. Finally, the main conclusions were detailed in Sect. 4.

2. Methodology

Figure 1 shows the proposed methodology for drawing dies structural analysis and topology optimization. The process in the sheet metal forming simulation initiated with a 3D part model, and eventually recorded the reaction forces between die components and sheet blank. The process steps included following steps.

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 FE-model was developed before stamping simulation.
3. Stamping forming simulation: it was conducted in the sheet metal forming software packages, in which the dies were represented as 2D rigid surfaces and the deformation was not taken into account.
4. Contact forces: after stamping simulation, the contact forces acting on each contact surface between die components and sheet blank were obtained. The contact forces were calculated by stamping process simulation on each node of the thin shell elements, they could be taken as the nodal forces, working 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 structural analysis model was through a technique called load mapping algorithm. With the implementation of load mapping algorithm, the accurate reaction forces determined from the stamping forming simulation were applied as force boundary condition 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 elements nodes. In this procedure, the die components were defined as elastic deformable bodies against the given nodal forces, so that elastic deformation experienced by the die components were calculated. The results of die structural analysis would be in good agreement with actual engineering only under the reaction of accurate forming contact forces, since the suffering loads on the die surfaces depended primarily on the contact forces. Therefore, the next work in terms of the proposed methodology was to conduct displacement and stress analysis of dies through the adoption of numerical simulation. The main objective of die structural analysis was to receive contour plot of displacements and stresses under the reaction of accurate forming contact forces, including the following steps for static structural analysis:

1. Simplification: geometry 3D CAD model needed to be clean up prior to 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 entire 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 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. In comparison with a tetrahedral mesh, hexahedral mesh was time-consuming to generate for the geometry with complex structure features, such as the stamping dies [14].
3. Boundary conditions: different loading cases were considered in this step and the accurate contact force extracted from finished sheet metal forming simulation was taken as one of the force boundary conditions to apply on the static structural analysis model for structural behaviors prediction. By the means of load mapping, the precise reaction forces obtained from the sheet metal forming simulation were transferred to die static structural analysis model, working as the force boundary conditions. In addition to the stamping forming load case, die lifting load case was also taken into account, since lifting of the die in the lifting lugs was considered as an necessary operation for die transportation and cleaning. Besides, the supporting of press was considered as the constraint condition. In general, the fixation holes on the bolster side of the drawing dies 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 (ν), and density (ρ), were required to calculate the stiffness matrix in die static structural analysis model.
5. Solver calculation: after establishment of 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.), elastic deformation experienced by the drawing die components. These ones were significant parameters to evaluate die component behavior during stamping process for die structure strength verification.

The standard sheet metal forming simulations were carried out to obtain forming loads on the die during operation, then die structural analysis was implemented to investigate the elastic behavior of 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 achieve lightweight design, the 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 a 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. The following points were typical steps for topology optimization:

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 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 die topology optimization model, design variables, constraints, together with optimization objective needed to be defined.
3. Topology optimization: the commercial topology optimization solver OptiStruct have 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 obtained 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 totally different element meshes were implemented for the sheet metal forming simulation and structural analysis, which would bring reduction of model scale, yielding a total shorter solving time consequently. Whereas in the conventional coupling model, in order to achieve accurate results, the high-densities element meshes 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 enabled the complexity of problem to be simplified and the calculation could be considered as 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 FE-model for structural analysis. While in the case of conventional method, it was considered as a dynamic problem and required the contact characteristics to be assigned to each interface between die components, thus causing the contact conditions settings become sophisticated. This methodology has provided an effective approach for dealing with the structural behaviors 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

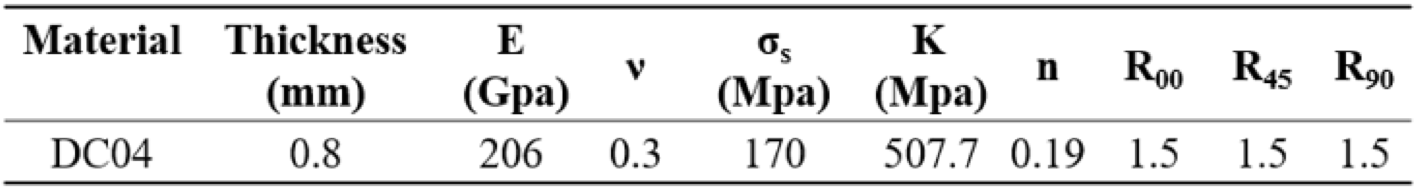
In this section, a long beam drawing die was selected as the object to illustrate the procedures presented in Fig. 1 and validate the effectiveness of proposed methodology for industrial purpose. In the case study, sheet metal stamping process was first conducted to obtain forming nodal forces, which were taken as the boundary conditions for die structural analysis model consequently. The structural behaviors of 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 were considered.

3.1. Sheet metal forming simulation

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

Subsequently, the study set some parameters related to the sheet metal forming simulation. The material for the blank was DC04, with the thickness of 0.8 mm, the properties for the blank is presented in Table 1. Figure 2 shows the input true stress-strain curve for the DC04 material.

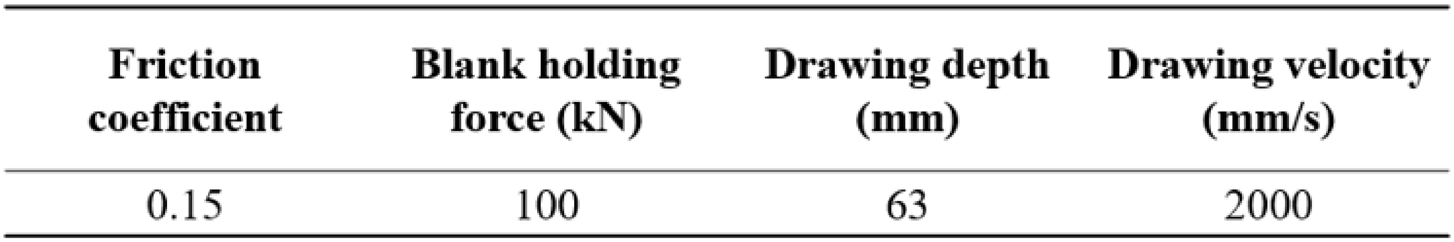
Table 1 Detail material properties used for the blank.



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	0.19	1.5	1.5	1.5

The stamping process model consisted of the same surface of 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 sheet metal forming process, the upper die was first meshed and then by offsetting technique the punch meshes and blankholder meshes were generated, as illustrated in Fig. 3. The key control parameters in sheet metal forming simulation are 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 set 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.

Table 2 Key control parameters selected for the sheet metal forming simulation.



Friction coefficient	Blank holding force (kN)	Drawing depth (mm)	Drawing velocity (mm/s)
0.15	100	63	2000

Since large strains 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 sheet metal forming simulation model was submitted to the solver Ls-Dyna, to calculate the accurate nodal forces on each elements nodes during sheet metal forming simulation.

The results of sheet metal forming simulation were 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.

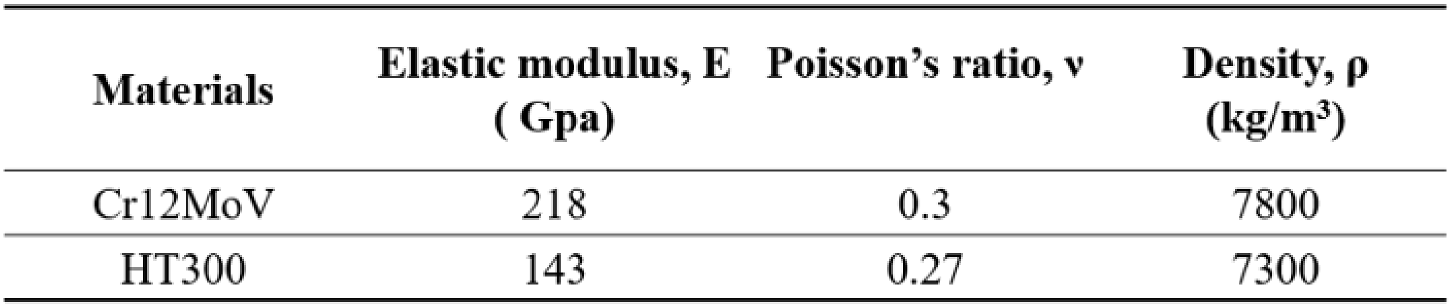
3.2. Linear static structural analysis

The sheet metal forming simulation of long beam was firstly conducted in order 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 cases were considered in the die structural analysis, in which the lifting forces during transportation were also applied to the die analysis model._

To prevent unnecessary problems, some small geometry features (e.g. fillets included in die 3D geometric models) were removed prior to discretizing the geometry models into 3D meshes. To get a perception of the die components, different views of the original die were shown in the following figures, as illustrated in Fig. 5.

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 referred as the die post, which 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 fracture.

Table 3 Material properties for the die components.



Materials	Elastic modulus, E (Gpa)	Poisson’s ratio, ν	Density, ρ (kg/m^3)
Cr12MoV	218	0.3	7800
HT300	143	0.27	7300

Different load cases were considered in this study. The first load case was the forming load case, which represented the die components under peak load condition. 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 the load case 1 is presented in Fig. 6. The obtained nodal forces were implemented on the corresponding elements 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 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 linear structural analysis 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 load case could be viewed in Figs. 7 and 8, For the punch, the maximum von mises stress reached 471 MPa, occurring at the sharp corner of the punch surface, as shown in Fig. 7. Sharp corners tended to concentrate stress, resulting in highly localized stress in these areas [17, 18]. For the die shoe, the stress eccentricity was observed in the reinforcement ribs, the von mises stress in the Y direction was obviously larger than that in the -Y direction, due to the asymmetrical shape of the target part. The von mises stress of the reinforcement ribs in Y direction reached 85 MPa, while only 26 MPa recorded in the -Y direction, as illustrated in Fig. 8.

The results of displacement are shown in Figs. 9 and 10. The deformation experienced by the punch (including the direction and magnitude of the displacements) was also taken into account. The magnitude of the deformation experienced by the punch reached 0.209 mm, as shown in Fig. 9. The punch was mainly deformed mainly in the Y and Z directions. Specifically, the relative displacement in Y direction of the punch was 0.197 mm. The deformation in X direction was shown in Fig. 10, which was negligible, as only 0.044 mm was recorded in positive direction and -0.040 mm in negative direction. Besides, it was worth noting that the deformation in X direction was symmetrical about the Y axis. This was because the shape of the target part in the X direction was almost symmetrical, and the contact forming forces in the X direction loaded on both sides of the punch were almost equal which could cancel each other out.

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 load case was also considered as a static condition. In the lifting load 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 lifting load case, since the die shoe structure suffering the greatest force during the lifting operation. The structure supported to be lifted in four lifting lugs. The established mechanical model of the lifting load case was shown in Fig. 11, where P_1, P_2 were the equivalent pressure transferred by the weight of other die components ($g=9.8\ m/s^2$).

Results for the lifting load case were presented in Figs. 12 and 13. For the original die shoe structure, the maximum tensile principal stress (i.e. P1 Stress) of 11.8 MPa was achieved in the case of lifting load, and the maximum displacement was around 0.022 mm.

3.3. Topology optimization of stamping die component

The topology optimization method exhibits great potential in terms of mass reduction [16, 19]. 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), materials properties (defined the same as those in linear structure analysis as mention before) were required prior to define 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. The rest of the volume was assigned as available design space which would be optimized, as shown in Fig.14. The non-design space with blue color in the figure remained unchanged during optimization, and the green space was assigned as design space.

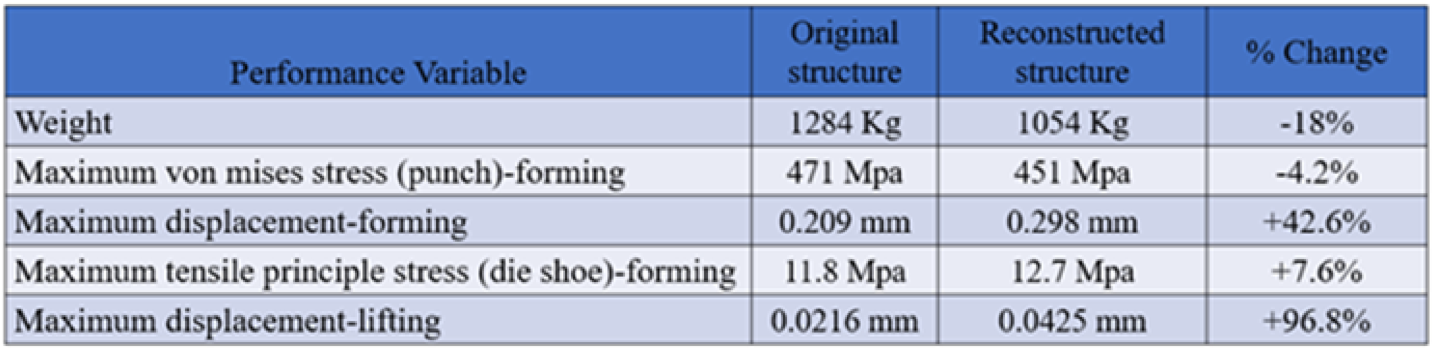
In the establishment of topology optimization problem for the long beam drawing die, the design variables, constraints, together with optimization objective were determined. Setting a volume fraction of 0.30, the volume of the optimized design area was required to be at most 30% of the initial design space volume. The objective function was defined to find the minimum compliance (i.e. the maximum stiffness) for given a certain available amount of material volume.

Results from the topology optimization are shown in Fig. 15, 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 figures indicated that the density of this material was 1.0 and the material could not be removed.

It was worth noting that the obtained initial optimized structure was a conceptual design, which could not be fabricated with current manufacturing techniques. Thus, further efforts should be made to reconstruct the structure based on the initial optimized results. In addition, when executing the die structure reconstruction, both current manufacturing techniques and design principles for die structure in industrial practice should also be taken into account. The reconstructed structure is shown in Fig. 16. As mentioned earlier in the proposed methodology, to verify the structural strength, the redesigned structure was meshed and then subjected to the same load cases. The same steps (as illustrated in Fig. 1) were conducted with the original die structure and analyses were made in both cases (i.e. forming load case and lifting load case). In order to study the efficiency of weight reduction, results (i.e. stresses, displacement as well as the weight of the new structure) would be compared with the original structure. Results of the linear structural analysis are shown in Figs. 17 to 20.

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. In comparison of the results, it was found that the reconstructed structures had a reduced mass of 18%, with 4.2% decrease of maximum von mises stress for the punch. For the reconstructed structure, the maximum tensile principal stress was 12.7 MPa, fulfilling the constraints settings upon it when lifting the structure in four supported lugs.

Table 4 Comparison between original and reconstructed die structure.



Performance Variable	Original structure	Reconstructed structure	% Change
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)-forming	11.8 Mpa	12.7 Mpa	+7.6%
Maximum displacement-lifting	0.0216 mm	0.0425 mm	+96.8%

4. Conclusions

This work proposed a new methodology for analyzing the structural performance and realizing weight reduction of drawing dies. The forming forces was determined by sheet metal forming simulation, and then linear static analysis was conducted using the obtained forming forces as boundary condition for the case study. Topology optimization and structure reconstruction were carried out to achieve die weight reduction. Check cycles were implemented to guarantee the performance of redesigned structures, and different load cases were considered. Compared with the previous methodology coupling sheet metal forming simulation and structural analysis into one FE-model, 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.

A long beam drawing die on an industrial scale was selected to illustrate the procedures of the proposed methodology. To achieve comprehensive and guarantee lightweight design, two different load cases were considered in 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 18% weight reduction was achieved with a slight difference of structural performance. It demonstrated that the proposed methodology was an effective numerical approach for predicting deflection and stress of cast die components and achieving certain weight reduction.

Declarations

Authors' Contributions

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

Funding

This work was supported by National Natural Science Foundation of China (51905163) and Open Research Fund of State Key Laboratory of High Performance Complex Manufacturing, Central South University (Kfkt2020-13).

Ethical approval

Not applicable.

Consent to participate

Not applicable.

Consent to publish

Not applicable.

Competing interests

The authors declare that they have no competing interests.

Availability of data and materials

Not applicable.

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 YD, Chen QR, Sun WJ (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 XY, 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(5):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. Rafiee 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 GQ, 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 JS, Lee KH, Lee HP (2000) Effects of geometry and ®llet radius on die stresses in stamping processes. J Mater Process Technol 104:254–264
10. Wang HY, Xie H, Liu QM, Shen YF, Wang PJ, Zhao LC (2018) Structural topology optimization of a stamping die made from high-strength steel sheet metal based on load mapping [J]. Struct Multidiscip Optim 58(2):769–784
11. Nie Y, Lin YJ, Sun W, Huang T. A methodology for optimal lightweight design of moulds and dies: a case study. Proceedings of ASME 2015 International Mechanical Engineering Congress and Exposition, November 13–19, 2015, Houston, Texas, USA
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. Sheet metal forming simulation with elastic tools in LS-DYNA. Proceedings of Numisheet, September 1–5, 2008, Interlaken, Switzerland
14. Nilsson A, Birath F. Topology Optimization of a stamping die. AIP Conference Proceedings, 2007, 908: 449–454
15. Hamasaki H, Nakazono M, Hino R, Yoshida F, Manabe H, Kondo H, Toropov VV (2014) Stiffness improvement of stamping die by means of topology optimization. Adv Mater Res 939:266–273
16. Burkarta M, Liewaldb M, Wieda J, Todzya T, Hartmanna 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. 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

19. 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

Figures

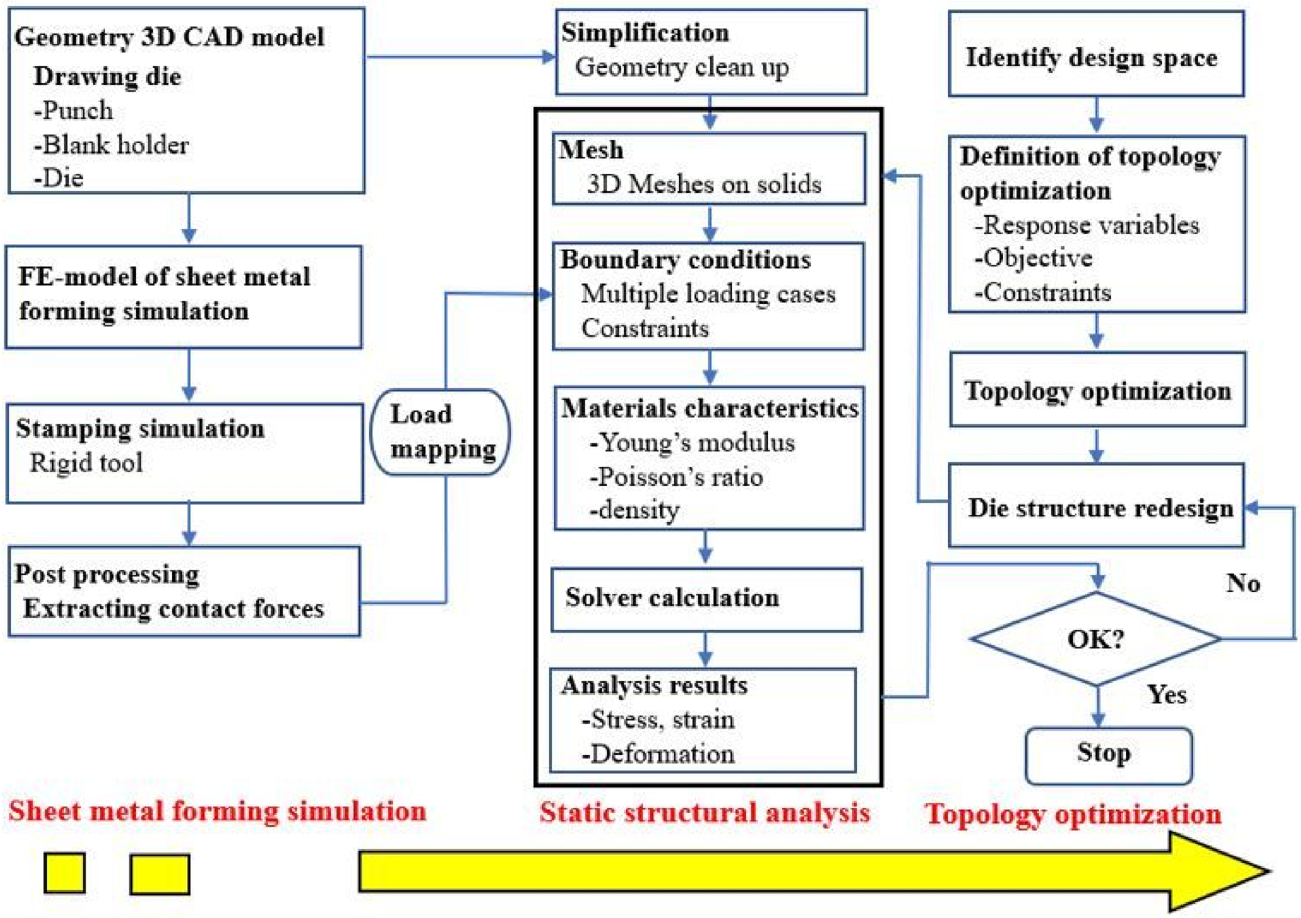


Figure 1

Proposed methodology for the simulation.

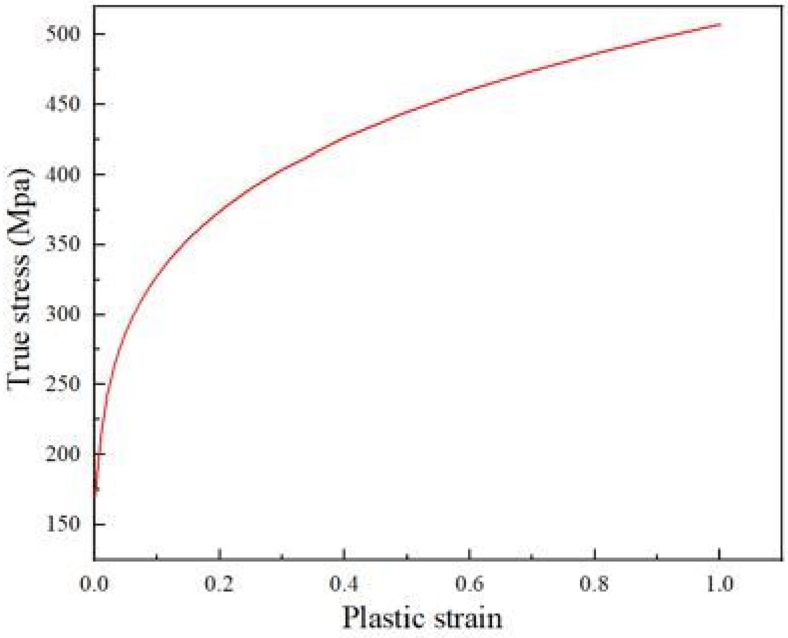


Figure 2

True stress-strain curve of the DC04 material.

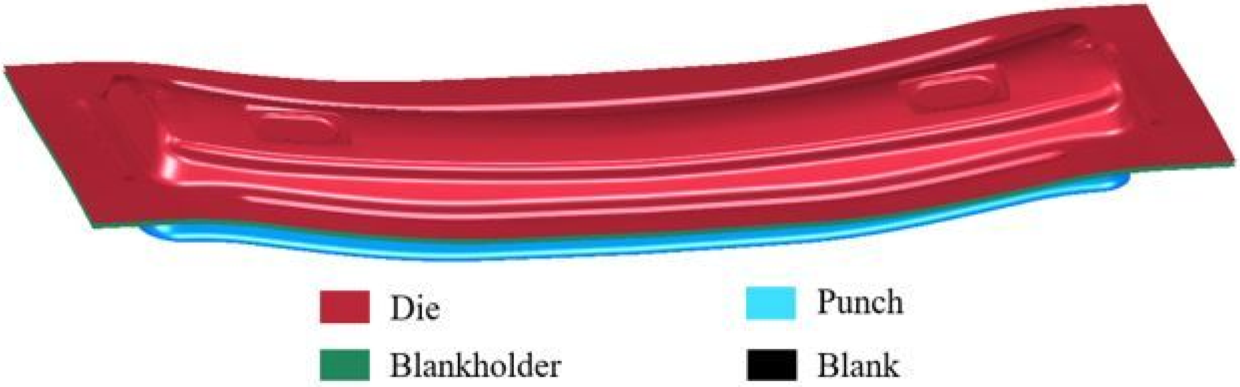


Figure 3

Selected case model for sheet metal forming simulation.

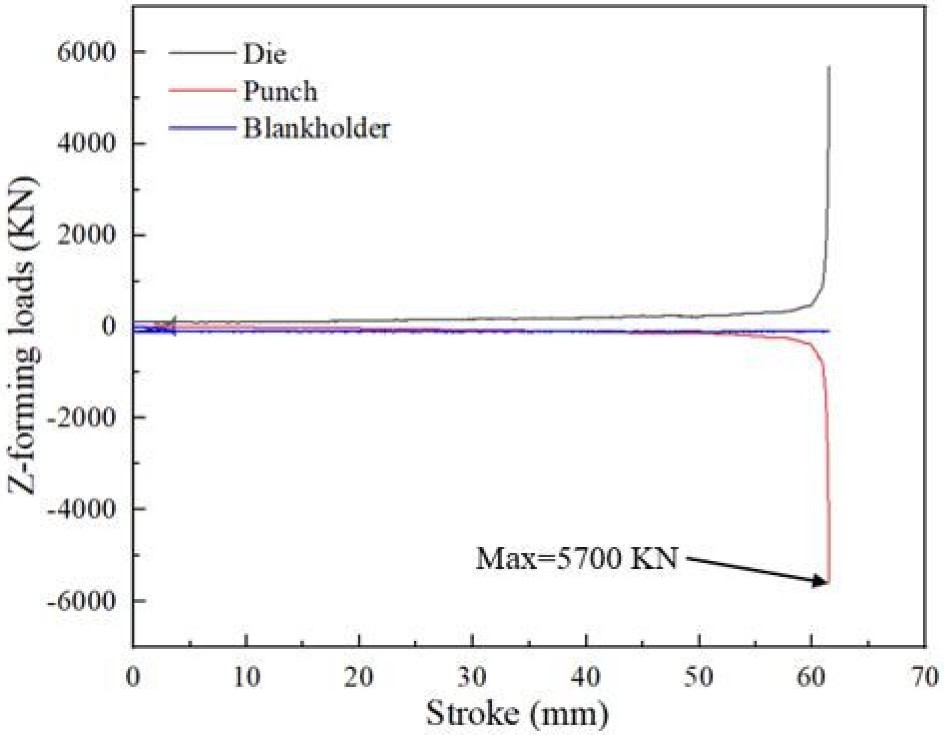
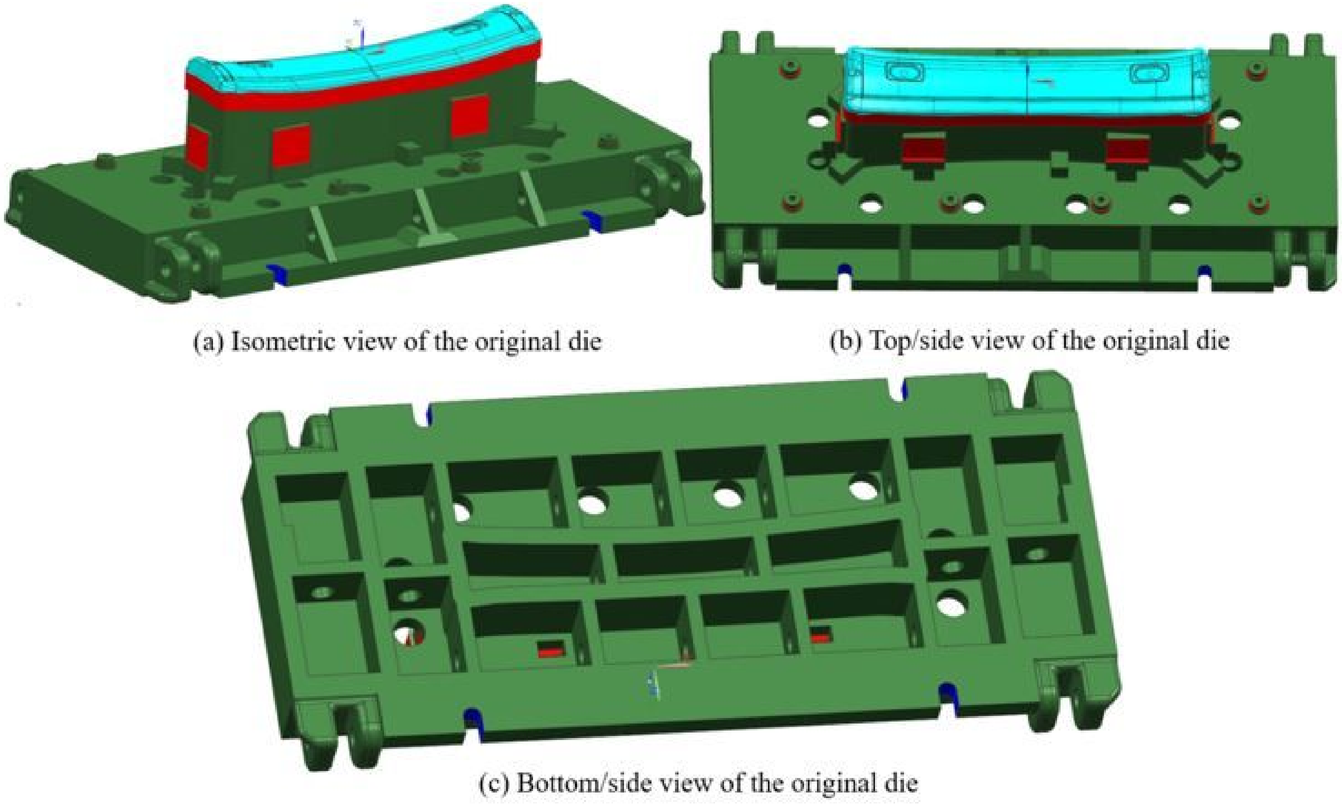


Figure 4

Load-displacement curves obtained from the SMF simulation



(a) Isometric view of the original die

(b) Top/side view of the original die

(c) Bottom/side view of the original die

Figure 5

Schematic diagram of the long beam drawing die component.

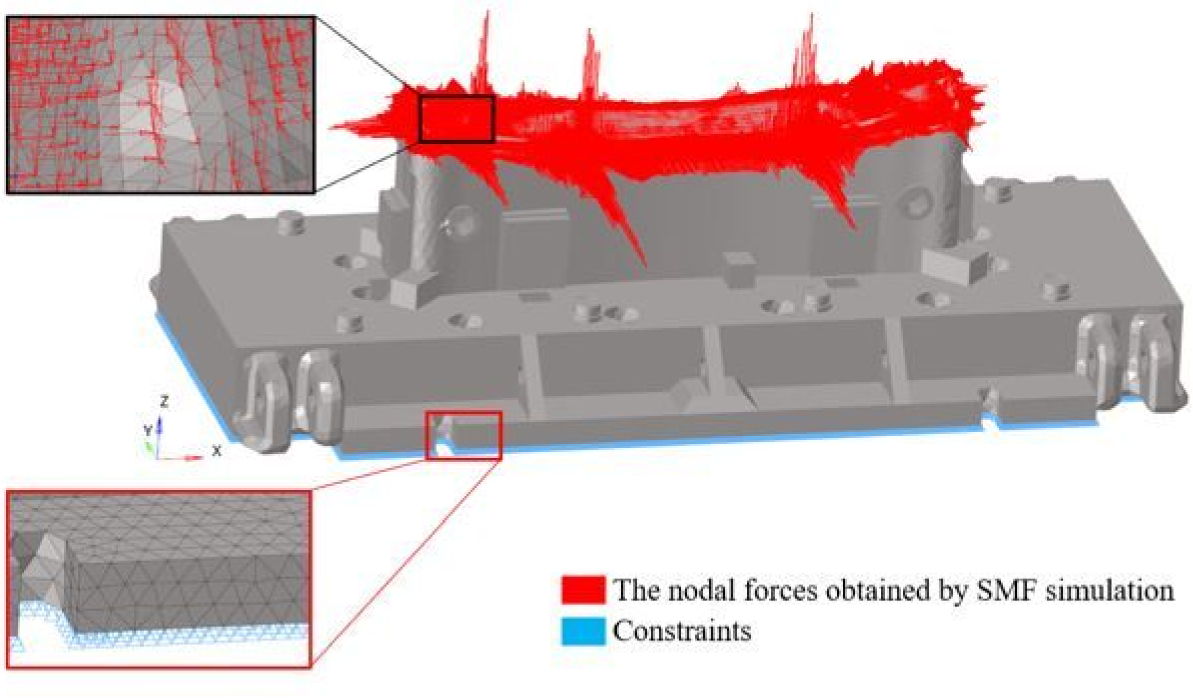


Figure 6

The established linear structural model for the forming load case.

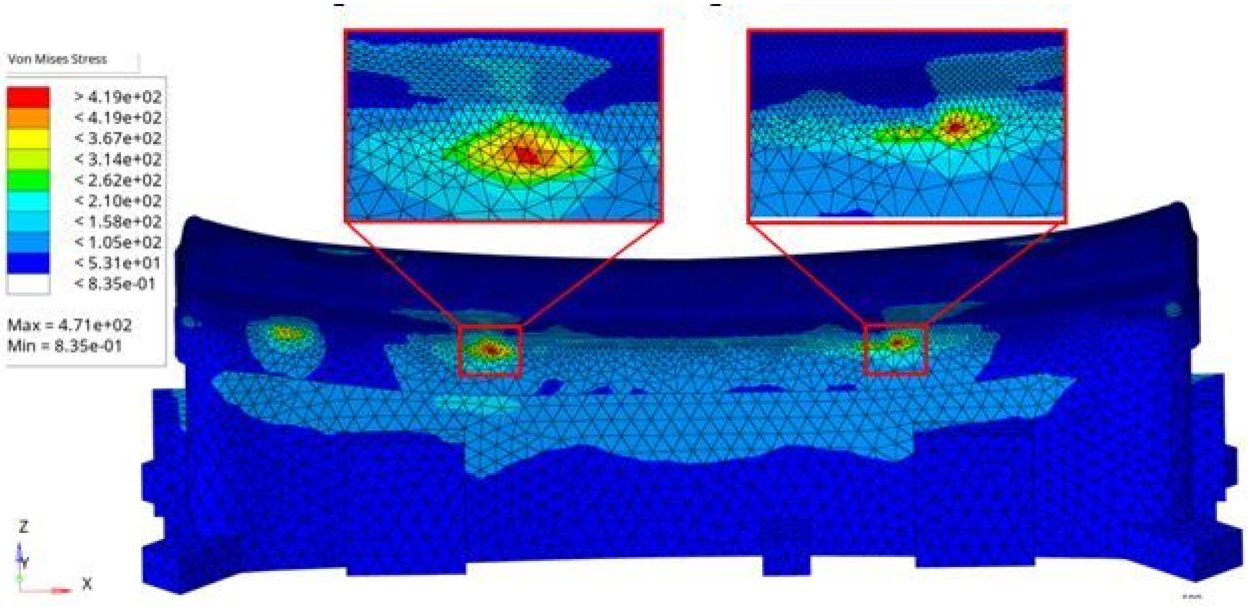


Figure 7

Von mises stress distributions of the punch.

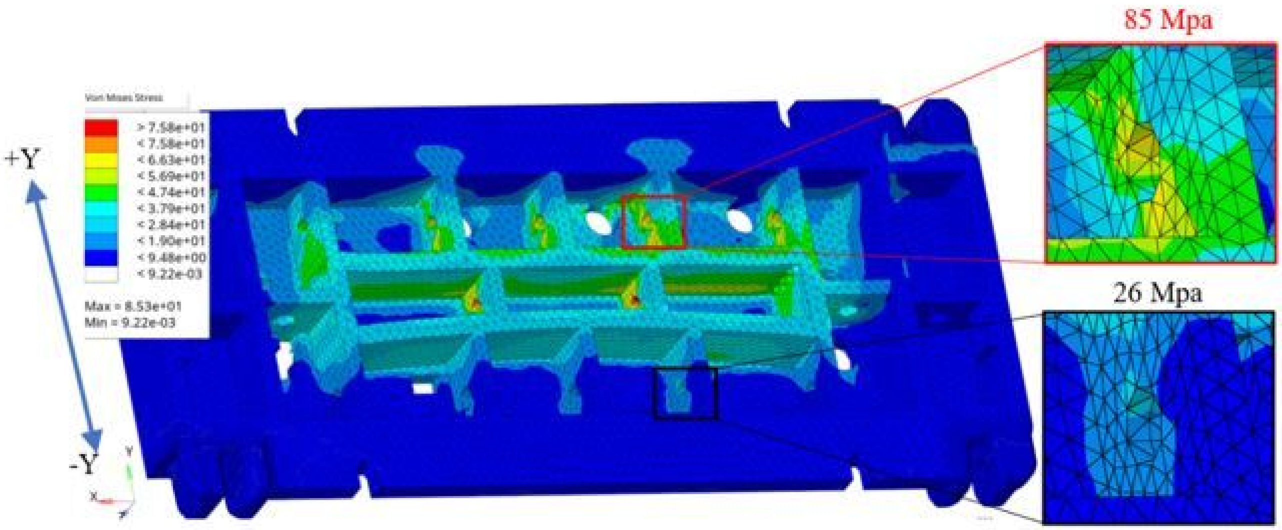


Figure 8

Von mises stress distributions of the die shoe.

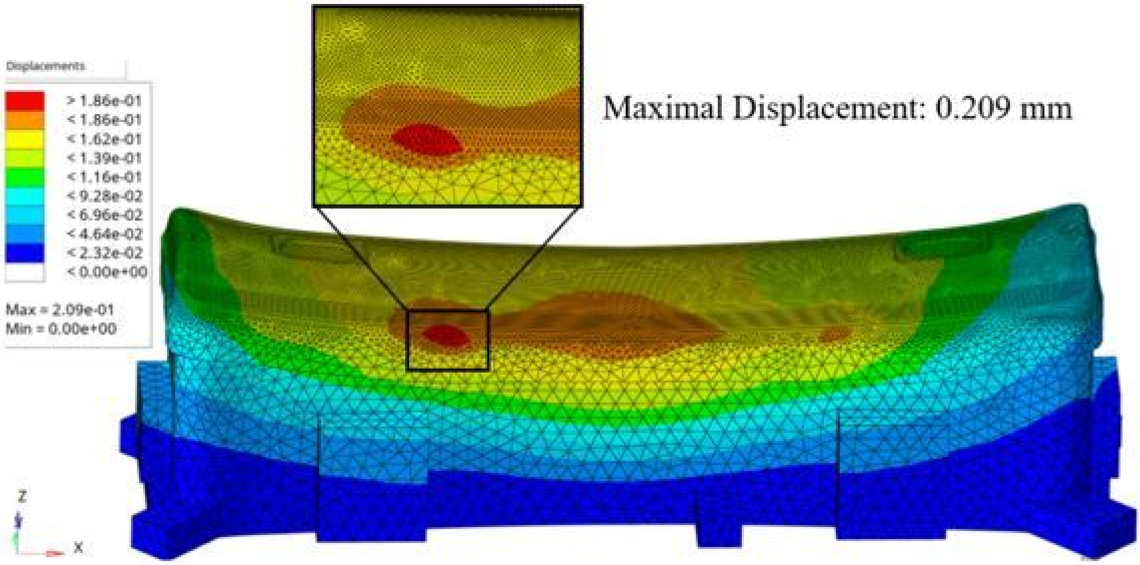


Figure 9

Displacement analysis of the punch (magnitude).

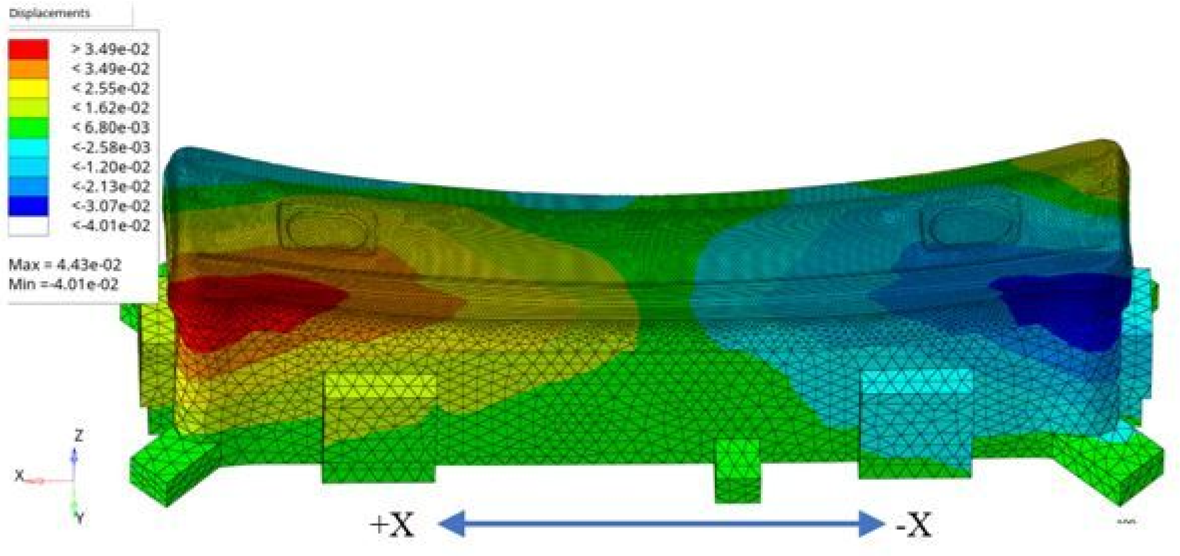


Figure 10

Displacement analysis of the punch (X-comp).

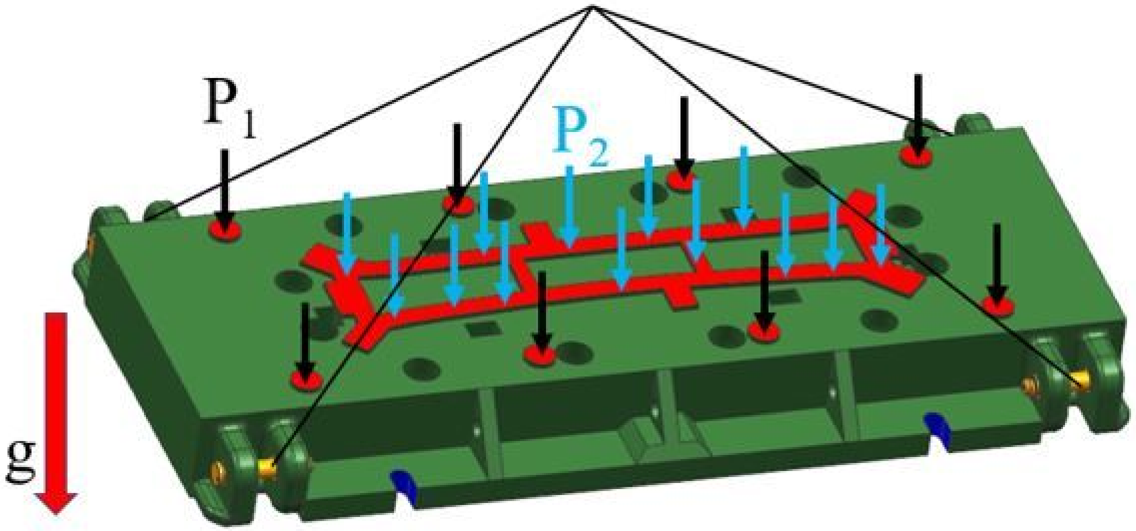
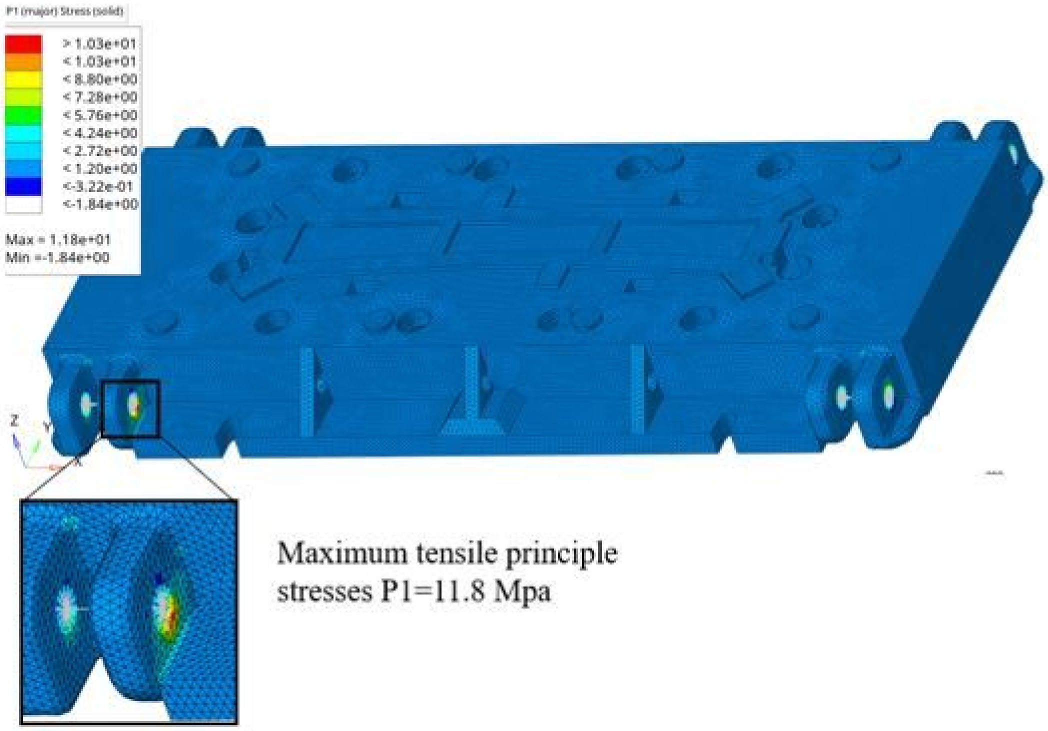


Figure 11

The mechanical model of the lifting load case.



Maximum tensile principle stresses P1=11.8 Mpa

Figure 12

P1 Stresses results of the original die shoe for lifting load case

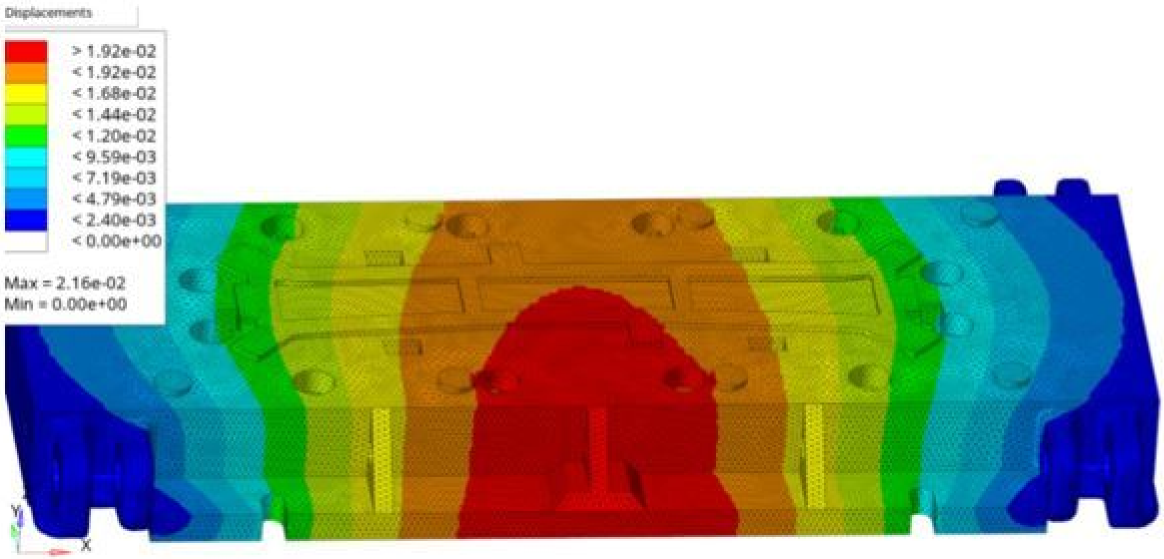


Figure 13

Displacement analysis of the original die shoe for lifting load case.

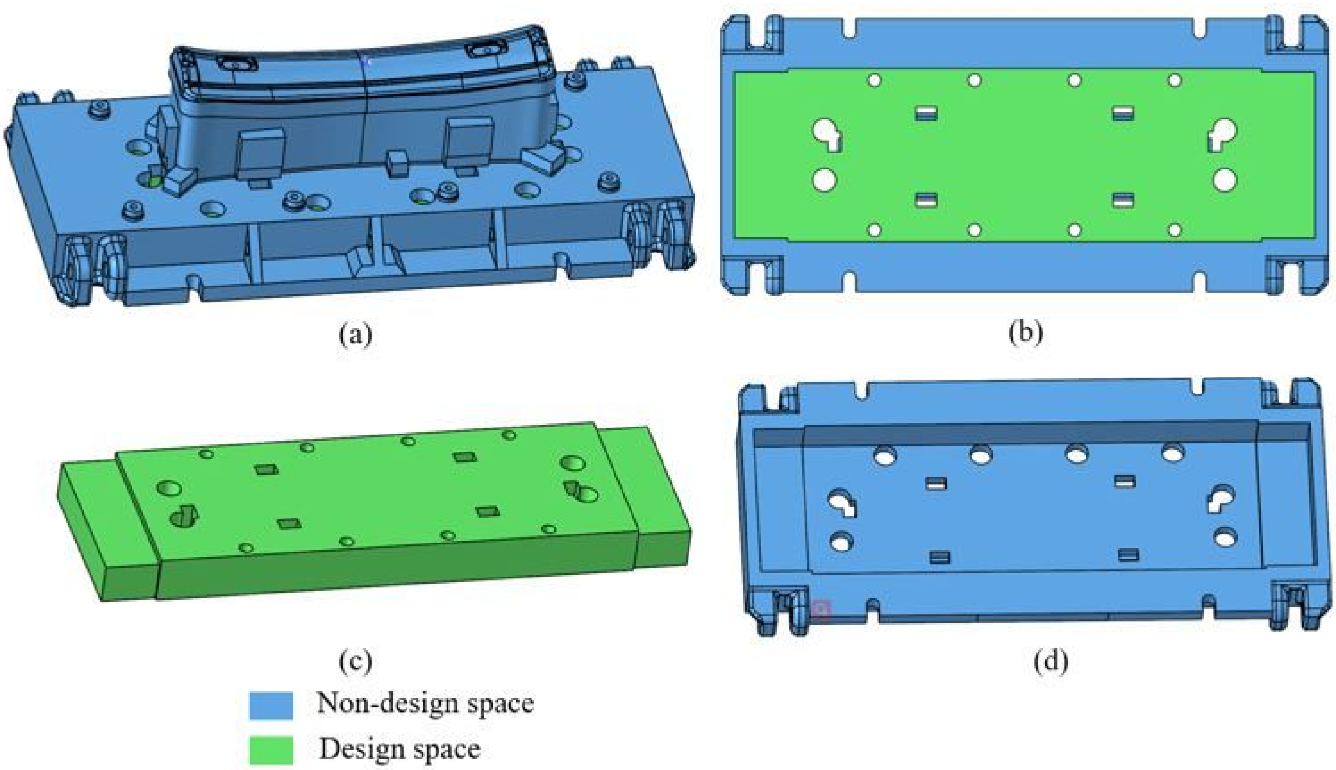


Figure 14

Definition of non-design space and design space.

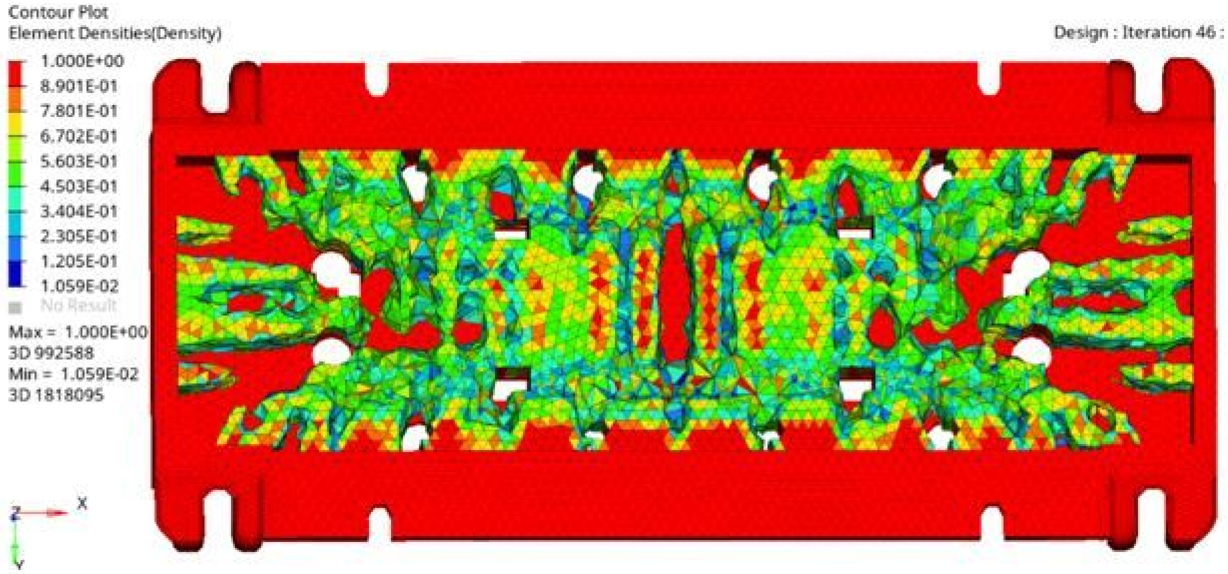


Figure 15

Topology optimization results of the die shoe under multiple load case constraints .

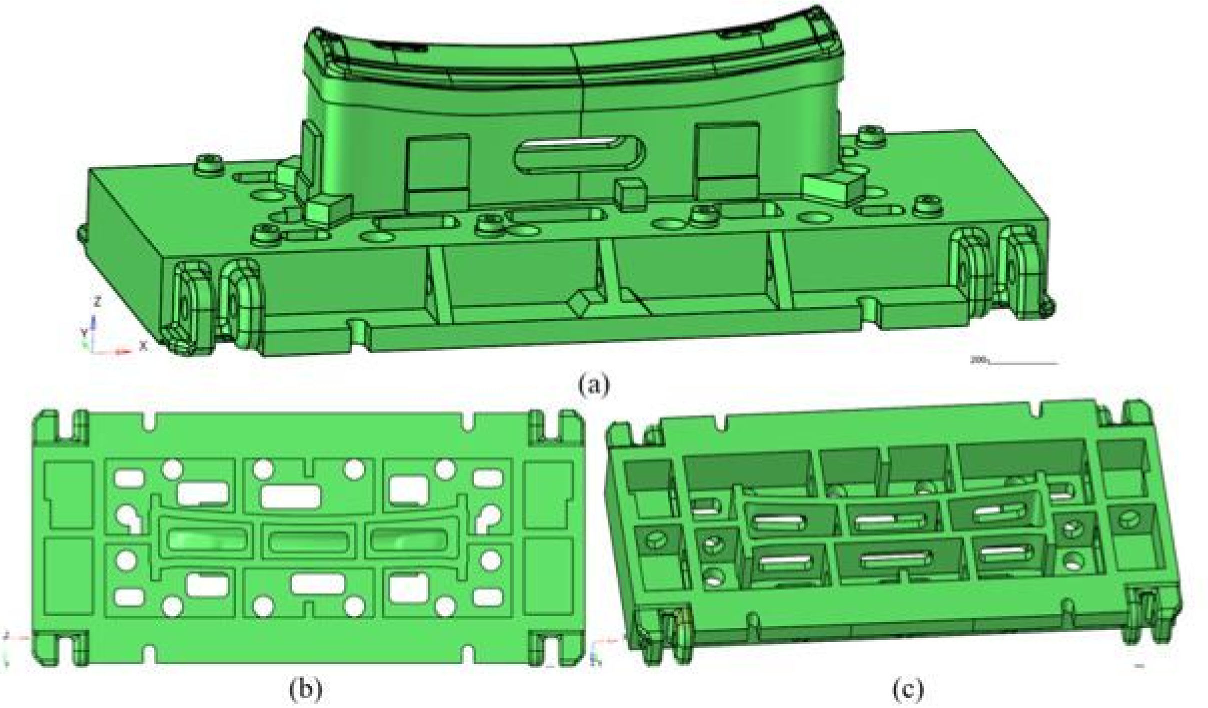


Figure 16

Model of the reconstructed structure.

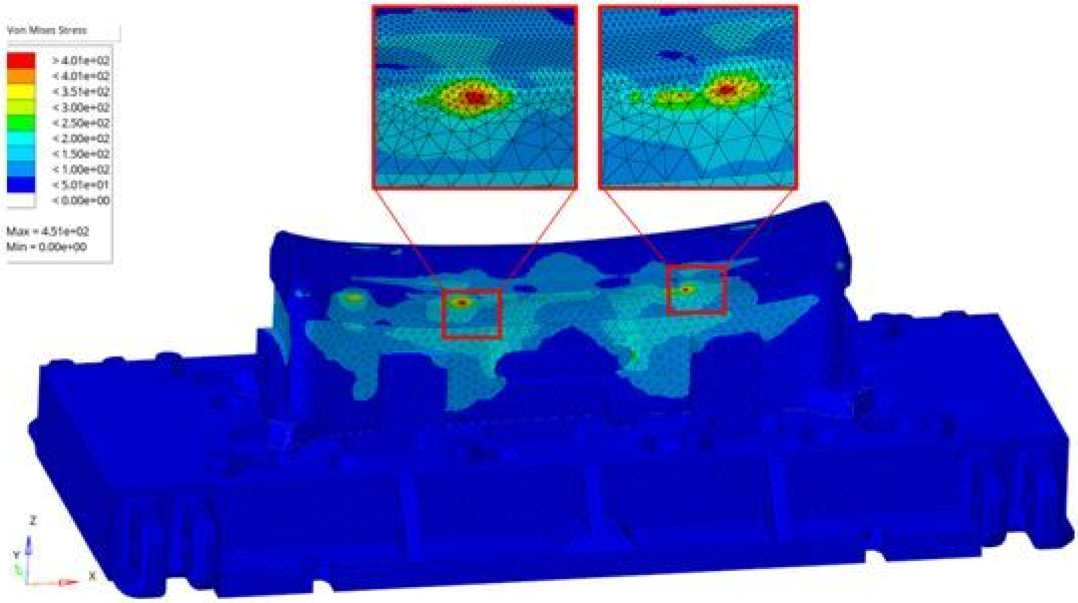


Figure 17

Von Mises Stress distributions of the reconstructed punch.

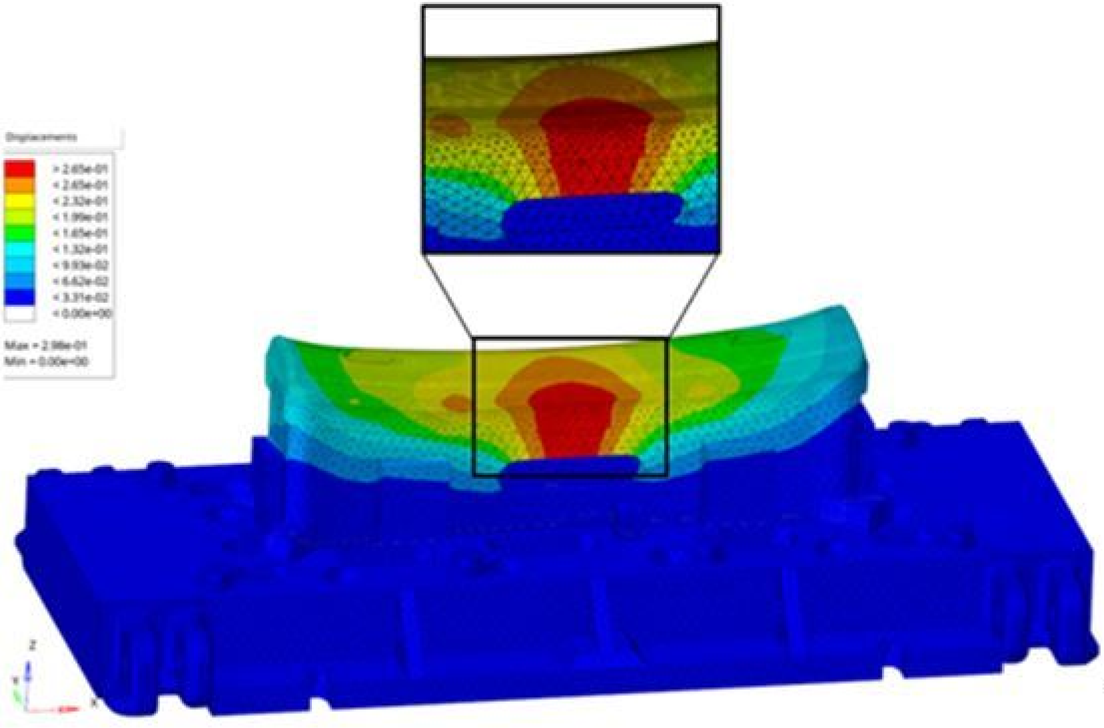


Figure 18

Displacement distribution of the redesigned structure (magnitude).

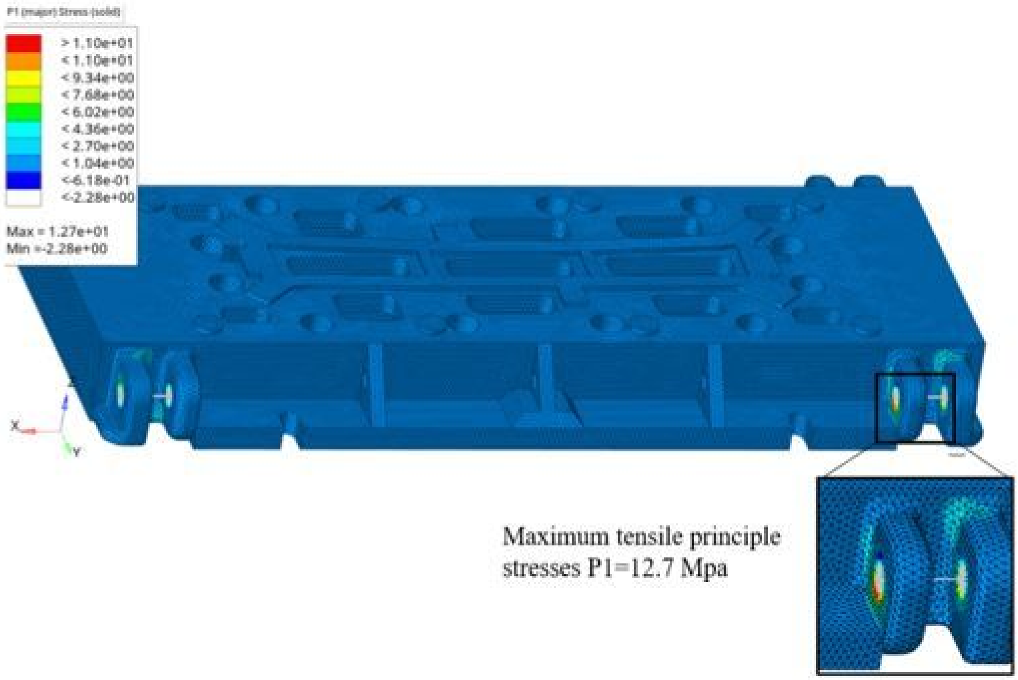


Figure 19

P1 Stresses results of the reconstructed die shoe for lifting load case.

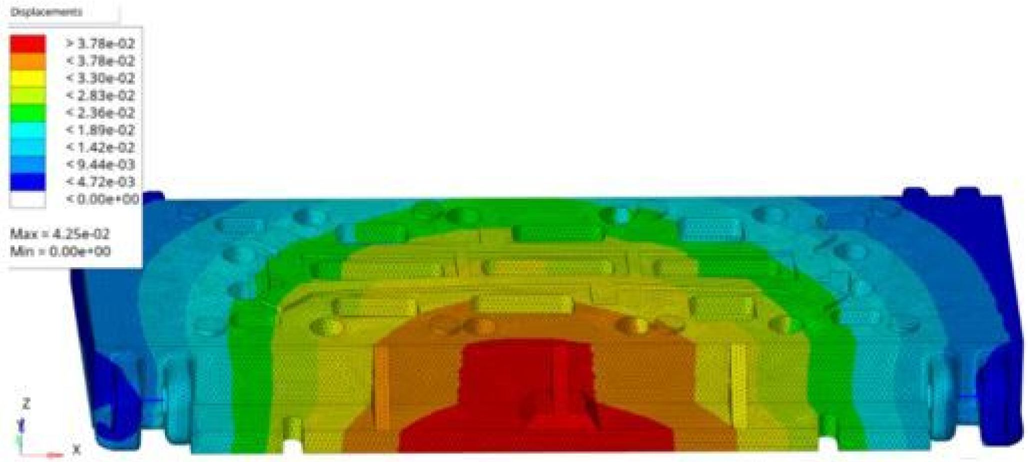


Figure 20

Displacement analysis of the reconstructed die shoe for lifting load case