Compensation of elastic die and press deformations during sheet metal forming by optimizing blank holder design

M Burkart*, P Essig, M Liewald, M Beck and M Mueller

1Mercedes-Benz AG, HPC C133, 71059 Sindelfingen, Germany
2Institute for Metal Forming Technology, University of Stuttgart, Holzgartenstr. 17, 70174 Stuttgart, Germany
E-Mail: maximilian.burkart@daimler.com

Abstract. Shortened product development processes of car body components made of sheet metal combined with the upcoming lack of experts and craftsmen’s in that field do challenge car manufacturers fundamentally. Required tryout time phases and manufacturing costs of complete die sets for car body panels are significantly influenced by their proper function in production and therefore by their highly developed manufacturing process. Looking in more detail, elastic deformation of costed two components and forming press behaviour under load impacts the contact areas between the sheet metal and the load specific spotted areas of the tool. Especially in deep drawing such contact areas between matrice and blank holder are strongly influenced by the sheet metal properties and the part flange behaviour in terms of thickening and thinning during the process. Modern advanced forming simulation models indeed increase precision of sheet metal forming simulation remarkably. For example, new friction models and structural modelling of forming die and forming press components do improve simulation accuracy, while still using rigid tool components. Considering tool stiffness and press deformation in the die design by generating specific blank or geometries via stiffness optimisation, elastic deformations and pressure distribution presumably can be homogenised. In this respect, this paper presents a numerical approach for minimising elastic deformations in drawing operations and thus reducing trial time in the early engineering face. Here a stiffness optimised blank holder is designed based on topology optimisation, taking into account press deformation characteristics such as used drawing pins underneath the blank holder and the drawing cushion box integrated in the press bolster. Afterwards, the new designs compared to conventionally designed die based on structural simulations. The main finding of this contribution is that new approach presented on the one hand leads to improved tool stiffness and reduced tryout effort and more robust production conditions on the other hand.

1. Introduction

Facing major challenges such as the lack of experts and craftsmen’s or the shortening of product development processes of car body components, manufacturing companies are forced to continuously improve their development processes, especially in the automotive industry. This particularly applies to the process chain in die manufacturing, which is accompanied by design and part development departments. Here, after the generation of sheet metal part CAD models, die engineering defines the process plan, consisting of numerous sheet metal operations such as forming, cutting, and flanging in order to produce a workpiece according to specification. The process plan is therefore crucial for robust and feasible part production. When transferring digital results into physical manufacturing, die tryout depicts an essential stage due to cost and time consuming manual spotting efforts [1]. These efforts are
based on the differences between the forming simulation results and the manufactured geometries. Elastic deformations due to process forces induced by press ram action and die during sheet metal part forming are unknown. For this reason, surface pressures within the die are homogenized by iterative spotting steps during die tryout. Homogeneous surface pressures in contact areas reduce wear in series production to a minimum and ensure required part quality and die performance [2]. However, the initial contact areas at the beginning of the drawing process change due to straining effects of the sheet during the forming process. Furthermore, draw-in is influenced by part-dependent bearing areas and draw beads in the blank holder region. In turn, inhomogeneous pressure distribution in bearing areas leads to irregular draw-in of the blank and undesired surface defects of press part due to local pressure areas. Therefore, it is crucial to assure a homogeneous pressure distribution in bearing areas between matrice and blank holder to control tribological conditions and flange draw-in during the forming operation.

2. State of the art

To reduce tryout effort, present research work focuses on compensating specific press characteristics or unknown tool stiffness. Here, the main objective in published state of the art is to extend simulation models by considering press components in order to efficiently perform manual spotting. Pilthammar added components like the blank holder, the drawing cushion box, and the cushion pins to the forming simulation model in order to predict a virtual spotting image. However, the model is restricted due to the lack of information about component characteristics such as the exact design of the press components [3, 4].

Zgoll developed a coupled simulation model, which combines a forming simulation with a static FEM-simulation improving the engineering process. A so-called fingerprint tool is used to measure elastic deformations of the upper and lower press components, such as the press ram and the press table, enhancing his model. Simulation results led to new forming surfaces compensating elastic deformations in die design. The models' computing time is approximately 50% higher than that of standard sheet metal forming simulation. So far, Zgoll could not approve positive effects on cost or time by use of his novel simulation approach [5].

In the context of component stiffness optimization, different design concepts such as bionic and topology optimization approaches can be distinguished [6, 7]. Topology optimization is represented by numerical calculation methods determining an optimized component design, based on specified optimization objectives. Possible design objectives are the maximization of stiffness or the reduction of mass or volume. Topology optimizations algorithms describe structures by relocating and rearranging originating design elements [8]. Thereby, resulting component designs are restricted by predefined boundary conditions, load cases, and of course available design space. The structural topology optimization calculation considers the components' internal force/stress distribution, regarding its stiffness or mass [9].

Figure 1 represents a characteristic process for design optimization, using the core element topology optimization. The principle of this topology optimization is based on the numerical Finite Element Method (FEM), which mainly applies the Solid Isotropic Material with Penalization (SIMP) method. Here, the elements’ densities are adjusted according to the internal force distribution of the optimized structure [10]. After smoothing the component surface, the SIMP-method generates an output STL-mesh representing the optimized design. Subsequently, the proposed surface model has to be transferred to a solid body that specifies the final component design using a Computer-Aided Design (CAD) software. Therefore, it is crucial to consider the manufacturability as well as appropriate machining processes when re-designing the resulting geometry. Finally, a structural simulation and strength calculation validate the design of the optimized component [9].

Figure 1. Structural design process using topology optimization [6, 8, 9].

Selection of approach and software → Definition of design restrictions → Topology optimization → Surface smoothing & STL export → Redesign in CAD → Validation

Figure 1. Structural design process using topology optimization [6, 8, 9].
In recent years, research in the field of topology optimization of tool structures associated with sheet metal forming disclose a rapidly increasing number of publications. Thereby, research work particularly focuses on the design of forming tools as well as press system components. In the context of the optimization of sheet metal forming tools by topology optimization, the main focus is on mass reduction. On the one hand, this leads to cost savings through reduced material use and energy consumption. On the other hand, the systematic reduction of accelerated masses can influence the dynamic tool behavior and does minimize reaction forces, such as part holder forces [11]. The research work of Dongkai presents a plane tool for deep drawing. The blank holder structure of this rectangular cup tool was optimized to decrease its mass. Within the optimization approach, loads and contact areas resulting from the sheet metal forming simulation were specified as design restrictions for complying with required forming conditions. The weight of the blank holder could be reduced by 28.1 % [12]. Further publications present mass reductions of a lower die by 20 % [13]. In general, topology optimization offers excellent potential for mass reduction while maintaining the stiffness of tool components [14].

Another topology optimization approach is improving press component stiffness. By optimizing the forming press, Raž and Čechura improved the stiffness of a press crossbeam by up to 50 %. As a result, the required process precision during part production could be achieved [15]. Further research work presented similar results, for example, by reducing the weight of the press frame by 13,66 % and simultaneously increasing its stiffness [16]. However, an adaption of the press components is not sufficient for series part production, because of part-specific elastic deformation characteristics and long term invests in press machine standards.

Nevertheless, Pilthammer’s research presents how the elastic deformation behavior of the press table can be considered in die manufacturing process [17]. By simulating real table deformations with a topology-optimized simplified model, elastic table behavior could be characterized and reproduced. Generating such an inverse substitute model enabled the prediction of elastic tool deformations via virtual die-tryout for an enhanced rework of forming contact areas.

In conclusion, contact areas between blank, blank holder and matrice during sheet metal forming are not only influenced by material thickening effects, but also by elastic press and tool deformations. These deformations are mainly resulting from the superposition of blank holder and drawing cushion box deflections, due to their stiffness characteristics and process loads. Therefore, topology optimization methods can lead to improved process characteristics by adapting die design.

Against this background, research work reported about in this paper has concentrated on the topology optimization of a blank holder in order to increase its stiffness regarding the drawing cushion box and cushion pins behavior. The main objective was to ensure simulated blank holder contact areas by compensating inhomogeneous drawing cushion box pressure distribution. The presented holistic optimization process for a new blank holder design was evaluated by a nominal forming simulation and a structural FEM simulation of the nominal blank holder, regarding its elastic deformation. As shown in figure 2, the total tryout effort could be reduced by improved sheet metal forming simulations or die designs. By implementing both simulation approaches, benefits could be combined in a holistic process chain improvement without increasing engineering effort significantly.

![Figure 2. Improvement in die manufacturing process by decreasing tryout effort by the implementation of new methodologies considering sheet metal forming simulation and die design.](image)

In the following section, the topology optimization process of an outer hood panel blank holder is presented. The required boundary conditions for topology optimization are specified regarding the blank
holders’ geometry and optimization process. Furthermore, different load cases are defined considering two stages of contact areas during forming process. Different contact areas result from homogeneous pressure distribution and from the consideration of straining effects. The resulting optimized blank holder geometries are finally compared to the nominal blank holder geometry.

3. A general blank holder design optimization approach

The following section illustrates the proposed optimization methodology for blank holder structure. The method combines sheet metal forming simulation results with topology optimization using an approach including press characteristics, such as the drawing cushion box and cushion pins. The blank holder investigated and improved in this work is designed for drawing an outer hood panel. The nominal blank holder consists of EN-JS 2070 material and was already manufactured for series production. Due to intensive manual spotting work required, it became apparent that the stiffness of the blank holder should be improved. Results gained by structural FEM simulation in figure 3 show nominal elastic deformations resulting from the forming load for the hood panel. Inhomogeneous deformation is caused by the critical rear area of hood due to the extremely reduced blank holder stiffness. Therefore, topology optimization offers potential for a re-designed blank holder to generate a homogeneous pressure distribution, which leads to homogeneous deformations. Thereby, Siemens NX 12.0 software system was used for the blank holder design and Altair Inspire 2018.3 software for topology optimization targeting performed structural stiffness in this rear area.

![Blank holder design](image)

**Figure 3.** Structural FEM analysis of (a) CAD model of nominal blank holder and drawing cushion box components under load, resulting in (b) elastic deformation in z-direction.

The flow chart in figure 4 describes the method of the blank holder stiffness optimization performed. The main stages are listed on the left-hand side of the figure. The right-hand side illustrates the iteration process using the SIMP-based topology optimization method considering multiple load cases:

- **Step 1:** As a reference for comparison and validation, a nominal geometry is defined.
- **Step 2:** Contact areas with corresponding pressure values are derived for different load cases, based on a nominal forming simulation.
- **Step 3:** SIMP-based topology optimization is used to generate an optimized component design.
- **Step 4:** Design restrictions and design space are determined due to the die specification standards and given boundary conditions as well as press specifications. These specifications ensure a final design that meets series production conditions.
- **Step 5:** After smoothing the component surface, the SIMP-method generates an output STL-mesh of coordinate surface representing the optimized design.
- **Step 6:** Using CAD-software, the proposed surface model is transferred to a solid body that specifies the final component design. Considering the manufacturability while re-designing the geometry is crucial.
Figure 4. Die design optimization approach applied on blank holder structure using topology optimization and variation of pressure distribution.

4. Blank holder topology optimization regarding stiffness

Following sections describe the implementation of the topology optimization of the blank holder based on given boundary conditions. Furthermore, drawing cushion forces and two different load cases were considered in the optimization process. Finally, the proposed optimization results are described and compared, regarding their masses. The final stiffness evaluation will be carried out in section 5.

4.1. Boundary conditions regarding stiffness optimization (according to step 1, 3 & 4)

The optimization of the blank holder geometry was based on numerous design restrictions. Implemented boundary conditions and specifications for optimizing the blank holder originated from the process planning and the die design department. The process planning initially defined the active tool surfaces as well as the blank holder force required for ensuring a robust series part production. The design department engineer provided spatial boundary conditions, for example, the maximum tool space or guiding positions within the tool depending on the target press. The resulting design space for optimization was transferred to Inspire software, and functional areas were defined. These areas are represented by contact areas or cushion pin positions.

Based on given explicit design objectives, the design space subsequently was adapted via topology optimization. The first objective was to increase blank holder stiffness, which is significant to reduce elastic deformation during part production. Furthermore, this ensures homogeneous pressure distribution in the active contact areas during forming. A further objective of the optimization process was a mass reduction of 30 %, taking into account the restriction of minimum wall thickness required for the casting process.

4.2. Elastic characteristics of cushion box and cushion pins (according to step 2 & 3)

Since the stiffness behavior of a press generally is unknown, it is necessary to consider the elastic deformations of the press components, especially its lower components such as drawing cushion box and cushion pins. On the one hand, the approach presented in this paper assumes that the deflections of
the die emerge as specific to the sheet metal part. On the other hand, cushion pins and the drawing cushion box are also considered, as these have a significant influence on the size and location of merging contact areas. For the consideration of the elastic deformation, static FEM simulations of the drawing cushion box, including the cushion pins, were carried out based on the cushion pin pattern used for this hood. The force applied onto the structure in following topology optimization was implemented considering the drawing cushion box deformation, which results in individual forces for each cushion pin. Figure 5 shows the resulting elastic deformations based on a static structural simulation. The local deformation reveals how press structure is influencing pressure distribution during forming.

![Figure 5. Structural FEM for drawing cushion box and cushion pins resulting in local elastic deformation in z-direction regarding part specific drawing cushion load.](image)

4.3. Simulated load cases (according to step 2)

For the topology optimization of the blank holder geometry, two different load cases, represented by different contact areas, were investigated and implemented in the optimization software in order to generate a sufficient tool structure. During the deep drawing of sheet metal parts, the active contact areas are mainly influenced by the elastic deformations of tool and the press components as well as the thickening or thinning of the sheet metal due to the straining effect by deep drawing [18]. Based on these input parameters, resulting local surface load conditions onto the blank holder structure are calculated and corresponding contact surfaces are derived as boundary constraints via sheet metal forming simulation. According to these specific constraints, die structure analysis determines the adaption of design space for die structure topology optimization. The load cases investigated differ in pressure values and areas, leading to an extended investigation of the optimization process, represented in figure 6.

- **Load case 1.** Assuming a homogenous pressure distribution calculated by conventional sheet metal forming simulation specifies the first load case. In this case, a numerical factor in the simulation model, called *tool stiffness*, compensates emerging straining effects, resulting in the homogenous pressure distribution shown in figure 6 (a).
- **Load case 2.** In contrast to load case 1, load case 2, as shown in figure 6 (b), differs mainly in the compensation of the straining effects. By changing the *tool stiffness* value in simulation, pressure distribution is shifting locally. Areas with higher thinning rates do not remain in contact, and therefore, pressure values decrease. As a result, contact pressures of remaining areas increase.
Figure 6. Simulated pressure distribution for the two defined load cases represented by (a) load case 1 homogenized spotting and (b) load case 2 by variation of tool stiffness value based on thinning/thickening effects.

The load cases shown in figure 7 were implemented in Inspire software, using the defined boundary conditions. From these load cases the design space required for topology optimization was derived. Figure 7 (a) shows the nominal model used for evaluation. Figure 7 (b) represents the simulation model for load case 1 and 2, whereby only the constraint surfaces were varied. The applied forces were obtained from static FEM simulation performed in section 4.2.

Figure 7. Implemented geometries used for Inspire with (a) nominal geometry as a structural comparison and (b) extended design space for load case 1 and 2 for topology optimization.

4.4. Optimization results (according to step 5 & 6)

Based on the load cases, the optimized solutions were calculated by the SIMP optimization method as shown in figure 4. Compared to the nominal blank holder, the results shows significant differences in design. The calculated reaction force distribution is dependent on assumed load cases and the optimization objectives and is represented by the new structural design. Figure 8 shows the optimization result for load case 1 and 2 in different views. Regarding the optimization results depicted in figure 8 (a) and (b), a minor difference between the optimizations becomes apparent. The optimization results are basically characterized by a material accumulation between the cushion pins and the contact areas, as well as, the remaining design features of the blank holder.

Due to the decreased contact area of load case 2, the local bearing forces are higher, which leads to additional structural support. According to the optimization method, the connection directed to the guiding column in the corner areas is reduced to a minimum and therefore is only relevant for blank holder positioning.
Figure 8. Proposed topology optimization results for (a) load case 1 and (b) load case 2 for blank holder shown in figure 3.

The two resulting design proposals show a changed structural geometry with different masses compared to the nominal geometry. In figure 9 the different masses are compared. The new design based on load case 1 shows a reduction of 44% in mass. Load case 2 led to a mass reduction of 43%, caused by the additional stiffness support. Regarding manufacturability and die design standardization requirements an empirical redesign factor of 1.3 was used, adapting the optimized geometries mass. The mass could be significantly decreased compared to the mass of the nominal blank holder. In addition to stiffness optimization, this leads to material and energy efficiency in series part production, due to reduced mass of blank holder. Especially concerning servo press technology, mass reduction enables process potentials. The resulting stiffness of the optimized blank holder geometries compared to the nominal design is evaluated in the following section.

Figure 9. Mass comparison of proposed design for both load cases including the redesigning factor, in relation to the nominal design mass.

5. Stiffness evaluation
The main objectives of the improved blank holder structure are to ensure an increased stiffness and a homogenous pressure distribution during the forming process. For evaluating these objectives the elastic deformations resulting from implemented load cases were used. Based on boundary conditions, load cases and applied forces, the Inspire software calculates displacement via FEM. Figure 10 shows the nominal displacement compared to the optimized designs regarding their specific load behavior.
Due to the inhomogeneous force application resulting from the press characteristics, an inhomogeneous displacement of the blank holder results for each individual load case. The deformation behavior of the nominal geometry does not correspond to the force distribution of the cushion pins as shown in figure 5. The stiffness optimized geometries compensate for the cushion pin force differences by means of specific material accumulation. Thus it becomes apparent that in both load case 1 and 2 the elastic blank holder deformation in relevant regions corresponds to the force distribution resulting from the drawing cushion box and cushion pins deformation. Nevertheless, the presented deformations in both load cases differ according to the local structure.

The middle areas of the blank holder are characterized by increased deformations, as indicated by the red areas in figure 10 (b) and (c). In order to verify whether the deflection of these areas has a significant effect on the required stiffness, an additional evaluation criterion was applied. The deformation behavior of the bearing area can be evaluated by a load simulation of blank holder structure simulation by Siemens NX software, in which the drawing cushion box and cushion pins, as shown in figure 3, are positioned beneath the blank holder structure. The forming force is introduced into the tool via the simulated contact areas. Therefore the contact areas deformation in negative z-direction. The simulation result, shown in figure 11, indicates that the critical rear area displacement is improved by the optimization. Due to this, the global blank holder deformation is homogenized, enabling much more homogeneous contact areas and pressure distribution.
6. Conclusion
This paper shows that the stiffness of blank holder structure can be topology optimized regarding applied process characteristics, not only reducing mass/volume. Since the deformation of the blank holder during the sheet metal forming process is significantly influenced by its structure and the drawing cushion box as well as cushion pins. Drawing cushion box and pins deflections were simulated and considered by means of static FEM. Presented investigations indicate that, on the one hand, resulting reaction forces and, on the other hand, local displacement of the drawing cushion box must be taken into account for the part specific die design due to their influences on emerging blank holder contact areas. By topology optimization of the blank holder, its stiffness is adjusted to the inhomogeneous force application and the desired contact area. Homogeneous or straining of sheet metal part flange based contact areas can be used as target values. In addition, the mass of the blank holder is reduced by the optimization approach, as well as relevant guiding areas are highlighted and can be prioritized during the design process. By redesigning the optimization proposal and manufacturing by sand casting, a force distribution optimized blank holder structure could be enabled. This is especially crucial for the reduction of time and costs during die tryout. Further research will focus on validating and evaluating the presented optimization and simulation approach. Drawing cushion box deformation measurement as well as the digitalization of spotting images can be used to improve the boundary conditions for blank holder topology optimization. Finally, a holistic implementation by the manufacturing and analysis of an optimized series die can validate more the thoroughly presented design process.

7. References
[1] Birkert A R, Haage S and Straub M 2013 Umformtechnische Herstellung komplexer Karosserieteile: Auslegung von Ziehanlagen (Berlin, Heidelberg: Springer Berlin Heidelberg)
[2] Lange K (ed) 1990 Blechbearbeitung (Umformtechnik Handbuch für Industrie und Wissenschaft /
hrsg. von Kurt Lange ; Bd. 3) 2nd edn (Berlin: Springer)
[3] Pilthammar J, Sigvant M and Kao-Walter S 2016 J. Phys.: Conf. Ser. 734 32036
[4] Pilthammar J, Sigvant M and Kao-Walter S 2018 Int J Solids Struct 151 76–90
[5] Zgoll F, Kuruva S, Götze T and Volk W 2019 IOP Conf. Ser.: Mater. Sci. Eng. 651 012021
[6] Lippert R B 2016 Gestaltung von Additive Manufacturing Bauteilen 3D-Druck beleuchtet ed R Lachmayer et al (Berlin, Heidelberg: Springer Berlin Heidelberg) pp 45–55
[7] Mattheck C 2006 Verborgene Gestaltgesetze der Natur: Optimalformen ohne Computer 1st edn (Karlsruhe: Forschungszentrum Karlsruhe)
[8] Schumacher A 2013 Optimierung mechanischer Strukturen: Grundlagen und industrielle Anwendungen 2nd edn (Berlin, Heidelberg: Springer)
[9] Klahn C, Meboldt M, Fontana F, Leutenecker-Twelsiek B and Jansen J (eds) 2018 Entwicklung und Konstruktion für die Additive Fertigung: Grundlagen und Methoden für den Einsatz in industriellen Endkundenprodukten 1st edn (Würzburg: Vogel Business Media)
[10] Goelke M 2018 Practical Aspects of Structural Optimization: A Study Guide 3rd edn (Academic Program Altair University)
[11] Burkart M, Liewald M, Wied J, Todzy T, Hartmann M and Müller M 2020 Procedia Manufacturing 47 861–6
[12] Xu D, Chen J, Tang Y and Cao J 2012 International Journal of Mechanical Sciences 59 73–82
[13] Kruschke G 2016 Enorme Kräfte von bis zu 2.500 Tonnen: Schneller und präziser fertigen https://www.produktion.de/wirtschaft/audi-entwickelt-neue-werkzeuggeneration-311.html?page=2 (accessed 7 Nov 2019)
[14] Eracheri J, Venkatesan R and Kumar N 2018 15th International LS-DYNA® Users Conference 1–6
[15] Ráž K and Čechura M 2019 IOP Conf. Ser.: Mater. Sci. Eng. 651 012015
[16] Zhao X, Liu Y, Hua L and Mao H 2016 Struct Multidisc Optim 54 375–89
[17] Pilthammar J, Sigvant M, Hansson M, Pålsson E and Rutgersson W 2017 J. Phys.: Conf. Ser. 896 12068
[18] Fann K-J, Liewald M, Riedmüller K, Randonjic R and Karadogan C 2019 Proc. of the 2nd APSTP